

# **Thermo-Calc Documentation Set**

**Thermo-Calc Version 2024a**



Copyright 2024 Thermo-Calc Software AB. All rights reserved.

Information in this document is subject to change without notice. The software or database described in this document is furnished under a license agreement or nondisclosure agreement. The software or database may be used or copied only in accordance with the terms of those agreements. You can [read more on our website](#).

Thermo-Calc Software AB

Råsundavägen 18, SE-169 67 Solna, Sweden

+46 8 545 959 30

<https://thermocalc.com/>

# Contents

---

## **Thermo-Calc Documentation Set . 1**

### **Installation Guide ..... 1**

#### ***Installing Thermo-Calc* ..... 2**

Key to Using this Guide ..... 3

Installation Terminology ..... 5

About the Thermo-Calc  
Components Being Installed ..... 7

What is Installed with a Full  
Standalone or Full Network  
Client Option? ..... 7

Thermo-Calc ..... 7

Databases ..... 7

Software Development Kits  
(SDKs) ..... 8

TC-Python ..... 8

TC-Toolbox for MATLAB® ... 8

What Does the Custom Option  
Allow Me to Do During an  
Installation? ..... 8

System Requirements ..... 9

General ..... 9

Network Client Installations:  
Sentinel RMS License Software . 9

Platforms Tested and  
SDK Availability ..... 10

Platforms Tested ..... 10

SDK Availability by Platform .... 10

#### ***License and Installation Types* ..... 11**

---

License and Installation Types ..... 12

Sentinel RMS License  
Management Software: Use  
Current Version ..... 12

About License Keys and Files ..... 13

Standalone Installations (SUNLL) .... 14

Multi-User Network Installations  
(NWL) ..... 15

Consolidated Network  
Installations ..... 15

Distributed Network  
Installations ..... 16

License Installations on a  
Server Network ..... 16

#### ***Preparing for Installation* ..... 18**

First Steps of Installation ..... 19

Installation Default Directories and  
the License File ..... 20

License File Installation  
Location for All Operating  
Systems ..... 20

Installation Default  
Directory: Windows ..... 21

Installation Default  
Directory: macOS ..... 22

Installation Default Directory: Linux 23

Installation Default  
Directory: Databases, Database  
Checker, and TDB Editor ..... 24

Database Checker ..... 24

TDB Editor ..... 25

Installation Default Directory: Materials .....	26	Install the Software and Start Using Thermo-Calc .....	40
Installation Default Directory: Process Metallurgy Materials .....	28	Add New Components to an Existing Installation .....	41
Installation Default Directory: Custom Property Models	29	<b>Network Client Installations</b> .....	<b>42</b>
Subnet Information for Network Licenses .....	30	Full Network Client Installations ....	43
macOS Installation Notes .....	31	Install the Sentinel RMS Management License Software .....	43
Linux Installation Notes .....	32	Complete Steps 1 to 3 of a Full Standalone Installation .....	43
openSUSE Administration (sudo) Users .....	32	Choose the Installation Type and Enter the Computer Server Details .....	43
Opening Thermo-Calc in Linux .	32	Windows: About Installing the License Manager .....	45
<b>Standalone Installations</b> .....	<b>33</b>	Supported Windows Versions ..	45
Full Standalone Installations .....	34	Windows: Installing the Sentinel RMS License Manager	46
Request your License File from Thermo-Calc Software .....	34	Uninstall Any Older Versions of Sentinel RMS License Manager .....	46
Download your Thermo-Calc Installation Software .....	34	Download and Install Thermo-Calc .....	47
Run the Applicable Installation File for your Operating System .	34	Install the Sentinel RMS License Manager Software ....	47
Select the Installation Type and Upload the License File .....	35	Choose the Installation Directory .....	47
Install the Software and Start Using Thermo-Calc .....	36	Choose the Setup Type and Install the Software .....	48
Custom Standalone Installations ....	38	Install the Thermo-Calc Network License Using WIAdmin .....	48
Select the Installation Mode and the Components to Include or Exclude .....	38		
Set the Installation Directory and Choose the License File ....	39		

Install Additional Tool Options .....	49	Windows Installations .....	55
Linux: About Installing the License Server .....	50	How to Install a Thin Client on Windows .....	55
Preparing for Installation .....	50	How to Install a Thin Client on Linux .....	56
Uninstall Older Versions of the License Server .....	50	Custom Network Client Installations .....	58
Requires Root Access or sudoers List .....	50	Install the Licensing Software .....	58
Linux 32-bit Installations on a 64-bit OS .....	50	Complete Steps 1 to 4 of a Full Standalone Installation .....	58
Thermo-Calc License File .....	51	Select the Installation Mode and the Components to Include or Exclude .....	58
Differences Between 32- and 64-bit Scripts .....	51	Set the Installation Directory and the File Path to the License .....	59
openSUSE Administration (sudo) Users .....	51	Install the Software and Start Using Thermo-Calc .....	60
Supported Linux Operating Systems and Versions .....	52	Unattended Network Client Installations .....	61
Linux: Installing the Sentinel RMS License Server .....	52	Create an Option File in a Text Editor to Enter the Options .....	61
Stop Older Versions of the Sentinel RMS License Server .....	53	Go to the Directory Where the Installation File is Saved .....	62
Write a Script to Start the License Server .....	53	<b><i>Request License Files</i></b> .....	<b>63</b>
Confirm the HOSTNAME variable and open the UDP port .....	54	Request a License File from Thermo-Calc Software .....	64
Start the License Server .....	54	Getting License Information Using Thermo-Calc .....	65
Install Additional Tool Options .....	54	Requesting the License File for Windows or Linux .....	66
Thin Client Installations .....	55	Requesting the License File for macOS .....	68
Installation Requirements .....	55		

---

**Quick Install Guides ..... 70****Windows® Standalone Quick Install Guide ..... 71**

This guide is applicable to: ..... 71

Other Installations ..... 71

Request a License File ..... 71

Install the Software ..... 73

Download and Run your Thermo-Calc Installation Software ..... 73

Select the Installation Mode ..... 73

**macOS® Standalone Quick Install Guide ..... 75**

This guide is applicable to: ..... 75

Other Installations ..... 75

Request a License File ..... 75

Install the Software ..... 76

Download and Install XQuartz76

Download and Run your Thermo-Calc Installation Software ..... 77

Select the Installation Mode ..... 77

**TC-Python Installation Guide ..... 79**

About the TC-Python Documentation ..... 80

Which TC-Python Installation Should I Choose? ..... 82

Installing TC-Python with a Bundled Python-Interpreter ..... 83

Limitations ..... 83

Install an IDE (Integrated Development Environment) ..... 84

Configure PyCharm to Use the Bundled Python-Interpreter ..... 84

Run a TC-Python Example ..... 85

Open the TC-Python Project in PyCharm ..... 85

Installing TC-Python Into a Python-Interpreter of Your Choice ..... 87

Install a Python Distribution ..... 87

Install Anaconda ..... 87

Install Thermo-Calc and the TC-Python SDK ..... 87

Install TC-Python ..... 88

Install an IDE (Integrated Development Environment) ..... 89

Open the IDE and Run a TC-Python Example ..... 90

Open the TC-Python Project in PyCharm ..... 90

Installation Default Directory: TC-Python ..... 92

macOS: Setting Environment Variables ..... 93

Fixing Potential Issues with the Environment ..... 94

Optional Packages ..... 95

Updating to a Newer Version ..... 96

TC-Python Diagnostic Script ..... 97

**TC-Toolbox for MATLAB® Installation Guide ..... 98**

---

About TC-Toolbox for MATLAB®	99
TC-Toolbox Documentation	100
API Reference Documentation	100
Installation Instructions	100
Thermo-Calc Help (Limited)	101
Installation Default Directory: TC-Toolbox and the Examples	102
Installing TC-Toolbox	103
Automatic Installation of TC-Toolbox	103
Manual Installation of TC-Toolbox	103
Multiple Windows Users on Same Machine	103
Installer Cannot Find the MATLAB® Installation Directory	104
Installing TC-Toolbox with the Custom Option	105
Request your License File from Thermo-Calc Software	105
Download your Thermo-Calc Installation Software	105
Install MATLAB®	105
Run the Thermo-Calc Installation File as an Administrator	105
Select the Custom Installation Type	106
Set the Thermo-Calc	107

---

Installation Directory and Choose the Setup Type	
Install and Start Using Thermo-Calc as an Add-on in MATLAB®	108
Test the Installation	108
Testing a TC-Toolbox Installation	109
TC-Toolbox Installation Troubleshooting	110
Diagnostic Script	110
MATLAB® Installation Directory Not Found	110
Determining the Licensing Software Version (Network Installs)	111
Uninstalling TC-Toolbox for MATLAB®	112
<b>Troubleshooting</b>	<b>113</b>
License Errors	114
Determining the Current License File Directory Path	116
From Thermo-Calc	116
Specifying a New Server Name or IP Number	118
Firewalls	119
Client/Server Mismatch	120
Compare computer names on the client computer and server	120
Try these options to correct the issue	121

macOS Installations: Running the License Initialization Script .....	122	Delete Old License Features from the License Manager ..	133
Linux Installations: Running the License Initialization Script .....	123	Shut Down a License Manager .....	133
<b>Sentinel RMS License 9.7 Tools .....</b>	<b>125</b>	Setting WlmAdmin Preferences .....	134
GetHostID: Get HostID/Mac Address .....	126	Sldemo: Show Basic API Functions	136
Windows .....	126	Wechoid: Get a Computer Locking Code .....	140
Linux 32-bit .....	126	<b>Updating or Uninstalling the Software .....</b>	<b>142</b>
Linux 64-bit .....	126	Updating the Software for Interim Releases .....	143
Ismon: Monitor License Manager Transactions .....	127	Installing an Updated Version of the Software .....	143
Options .....	127	Check for a Software Update ..	143
Ispool: Maintain Redundant License Manager Pool .....	128	Follow the Installation Instructions for the Applicable OS .....	144
Options .....	128	Option to Save Backup User Created Files .....	144
Ispool - L Options .....	128	Uninstalling Thermo-Calc .....	149
Isrvdown: Shut Down the License Manager .....	130	<b>The Thermo-Calc Software User Guide .....</b>	<b>1</b>
Iswhere: Display a List of License Managers .....	131	<b>About the Thermo-Calc Software .....</b>	<b>2</b>
Options .....	131	The Thermo-Calc Software .....	3
WlmAdmin Administrative Tool ...	132	About the Thermo-Calc Add-on Modules .....	4
WlmAdmin Functions .....	132	Additive Manufacturing (AM) Module .....	4
View License Manager Information .....	133	Diffusion Module (DICTRA) .....	4
Add or Remove a License Manager from the Defined Server List .....	133		
Add a License Code to the License Manager from a File	133		

Precipitation Module (TC-PRISMA) .....	5	Typographical Conventions .....	38
Process Metallurgy Module .....	6	<b>Thermo-Calc User Guide: Graphical Mode .....</b>	<b>2</b>
Property Model Libraries for Steel and Nickel .....	7	<b><i>Working in Graphical Mode .....</i></b>	<b>3</b>
Graphical Mode vs Console Mode ..	8	The Graphical Mode Desktop Layout .....	4
Graphical Mode Activities vs Console Mode Modules .....	10	Menu, Toolbar, and Keyboard Shortcuts .....	6
Graphical Mode vs Console Mode: Calculation Types .....	13	Organizing the Windows on the Desktop .....	10
About the File Types and Formats ..	18	<b><i>Projects and Activities .....</i></b>	<b>13</b>
Displaying Thermo-Calc License Information .....	19	About the Activity Nodes .....	14
<b><i>About the Documentation and Examples .....</i></b>	<b>22</b>	Activity Successor and Predecessor Nodes .....	15
Examples, Manuals, and Materials Folders .....	23	Graphical Mode Activity Node Workflow .....	18
About the Manuals (Documentation) Folder .....	28	Creating Activities and Successors ..	20
Thermo-Calc Installation Guides .....	29	Cloning Activities and Trees .....	23
Graphical Mode User Guide ..	29	Moving or Grouping Nodes and Using the Grid .....	25
Console Mode User Guide ....	29	Node or Window Status Markers ...	29
Thermo-Calc General Reference .....	30	My Project and Activity Menus .....	31
About the Examples .....	34	My Project Node .....	31
Examples by Product .....	34	Activity Node General .....	31
Help Resources .....	37	Saving Project Files .....	35
'Online' Help: No Internet Required .....	37	Including Calculated Results with Project Files .....	36
Console Mode Help .....	37	Saving a Project File with Results .....	36
		Opening Project Files .....	38

---

Opening Project Files in the Same Version of Thermo-Calc ...	38
Opening Recent Project Files ...	40
About Project Files and Plot Themes .....	40
Opening Project Files Including Results and with Different GES Versions .....	41
Opening Example Project Files in Thermo-Calc .....	42
Appending Project Files .....	44
Converting Project Files to a Newer Version of Thermo-Calc .....	45
About Backup Files .....	45
Converting Old Project Files - No Results Saved .....	46
Converting Old Project Files - Original with Saved Results .....	46
Opening Old Project Files in the Original Version of Thermo-Calc .....	48
The Calculation Types in Graphical Mode .....	49
About the Uncertainty Calculations	53
Working with Batch Calculations ...	59
Data File Rules .....	61
General Rules .....	61
Parameter, Parameter Name, and Parameter Value Rules .....	61
Data File Example and Column Definitions .....	64

---

Skipping Composition Validation .....	66
Skipping Parameter Validation .....	66
Getting Started Links and Templates .....	68
Creating a Project with the Quick Start Wizard .....	73
Creating a Project from a Template .....	77
Steel Library TTT Diagram Template .....	79
Steel Library CCT Diagram Template .....	80
Process Metallurgy Template ...	82
<b><i>Defining and Using the Activity Nodes</i></b> .....	<b>84</b>
System Definer .....	85
Define the System .....	86
Configuration Window Settings	86
How To Load or Save a Material .....	92
About the Material Files .....	97
Austenitic Stainless Steel .....	99
Duplex Stainless Steel .....	99
High Speed Steel .....	99
High Temperature Steel .....	99
HSLA Steel .....	99
Low Carbon Steel .....	99
Maraging Steel .....	100

Martensitic Stainless Steel ..	100	Ternary Calculator .....	126
Tool Steel .....	100	Configuration Window	
Training Steel .....	100	Settings .....	126
Creating a Database Package ..	101	Scheil-Gulliver Solidification	
Clearing a Database Package		Calculations .....	129
Message .....	102	Classic Scheil Simulation .....	129
Equilibrium Calculator .....	103	Scheil Simulation with Back	
Define the Equilibrium		Diffusion in Primary Phase .....	130
Calculator .....	103	Scheil with Solute Trapping .....	131
Conditions Tab Settings .....	104	Other Options: Fast Diffusers	
About Advanced Mode .....	107	and Scheil Simulations .....	132
Calculating and Plotting		Homogenization, Segregation	
Functions .....	109	Profiles, and the Diffusion	
Options Tab Settings .....	110	Module .....	133
Single Equilibrium and Grid ..	111	Summary .....	133
One Axis .....	113	Scheil in Graphical Mode vs	
Phase Diagram .....	113	Console Mode .....	134
Calculating with a Fixed Phase ..	114	Scheil Calculator .....	135
Material to Material Calculator ..	116	Define the Scheil Calculator ..	137
Define the Material to		Configuration Window	
Material Calculator .....	117	Settings .....	138
Material to Material		Advanced Options .....	143
Calculator Conditions Tab		Scheil Calculator Available	
Settings .....	117	Plot Variables .....	146
How To Load or Save a		Scheil Calculations Frequently	
Material .....	120	Asked Questions .....	148
Binary Calculator .....	123	Property Model Calculator .....	153
Configuration Window		Property Model Calculator	
Settings .....	123	Workflow .....	153
		About the Property Models ..	155
		General Models .....	155

Calculation Types .....	158	General Rules .....	183
Examples and Video		Diffusion Calculator Thermal	
Tutorials .....	158	Profile .....	185
Define the Property Model		Isothermal Calculations .....	185
Calculator .....	160	Non-isothermal Calculations	186
Configuration Window		Including Probe Data from	
Settings .....	161	an AM Calculator .....	188
Diffusion Calculator .....	166	Guidelines for Importing Data	
Available Graphical Mode		Files to Thermal Profiles .....	189
Features .....	167	Data File Requirements .....	189
Using the Templates .....	167	General Rules .....	189
Diffusion Templates .....	168	Options Tab Settings .....	190
Diffusion Calculator .....	169	Simulation Conditions .....	191
Define the Diffusion		Timestep Control .....	192
Calculator .....	169	Classic and Homogenization	
Conditions Tab Settings .....	170	Model Settings .....	193
General .....	172	Classic Model Specific .....	193
Region Settings .....	173	Homogenization Model	
Boundary Conditions .....	174	Specific .....	194
Composition Profiles .....	176	About the Homogenization	
Composition Profiles: Given		Functions .....	195
by Function .....	178	Pause, Resume, and Cancel	
Composition Profiles: Table		Diffusion Calculations .....	196
Input (i.e. Data File Import) .	179	How to Pause and	
Composition Profiles: Scheil		Resume a Job .....	196
Segregation Profiles .....	180	How to Cancel a Job .....	198
Guidelines for Importing Data		Diffusion Calculator Plot	
Files to Composition Profiles ..	181	Renderer .....	199
Data File Requirements .....	181	Simple Mode .....	199
General Rules .....	182	Advanced Mode .....	202

Precipitation Calculator .....	204	Accessing the 'Pre-existing Size Distribution' Settings .....	227
Available Options .....	205	Defining the Preexisting Size Distribution .....	228
Using the Template .....	205	Calculation Type Settings ....	228
Precipitation Calculator ...	207	Numerical Parameters on the Options Tab .....	230
Additive Manufacturing (AM) Module .....	207	Pause, Resume, and Cancel Precipitation Calculations .....	232
Define the Precipitation Calculator .....	207	To Pause and Resume a Job	232
Matrix Phase Settings .....	208	To Cancel a Job .....	234
Composition Settings .....	209	Precipitation Calculator Plot Renderer .....	234
Matrix Phase Settings .....	209	Plot Settings .....	234
Grain Growth Settings .....	212	Separate Multimodal PSD	234
Selecting the Disordered Phase as a Matrix Phase .....	213	3D → 2D .....	235
Precipitate Phase Settings ...	214	Yield Strength .....	235
Precipitation Calculator Thermal Profile .....	220	Non-Isothermal Simulations	239
Isothermal Calculations ...	220	Plot Renderer .....	242
Non-isothermal Calculations .....	221	Plot Toolbar .....	242
Including Probe Data from an AM Calculator .....	223	Configuration Settings .....	243
Guidelines for Importing Data Files to Thermal Profiles .....	224	Settings .....	244
Data File Requirements ...	224	Table Renderer .....	249
General Rules .....	225	Configuration Settings .....	252
Particle Size Distribution (PSD) .....	226	Experimental File Reader .....	257
		<b>General Property Models .....</b>	<b>259</b>
		About the Property Models .....	260
		General Models .....	260

Calculation Types .....	263	Liquid Fraction Parameters ..	279
Examples and Video Tutorials	264	Kou CSC Model .....	282
Scheil Advanced		Easton CSC Model .....	282
Options: Property Models .....	265	References .....	283
About the Coarsening Property		Crack Susceptibility	
Model .....	268	Coefficient Property Model	
Coarsening Property Model		Settings .....	284
Settings .....	270	Configuration Settings .....	284
Configuration Settings .....	270	Scheil Parameters .....	284
Plot Renderer Settings .....	271	CSC Model Selection .....	286
About the Columnar to Equiaxed		Additional Scheil	
Transition (CET) Property Model ..	272	Calculation Type Settings ..	286
Determine the Dendrite Tip		Thermal Mode and Liquid	
Radius and Tip Undercooling ..	272	Fraction Parameters	
Apply the Undercooling to the		(Clyne and Davies) .....	288
CET Model .....	273	Solid Fraction for	
References .....	273	Coherency and Solid	
Columnar to Equiaxed		Fraction for Coalescence	
Transition (CET) Property		(Easton) .....	288
Model Settings .....	274	Plot Renderer Settings .....	289
System Definer		Driving Force Property Model	
Configuration Settings .....	274	Settings .....	290
Property Model Calculator		Configuration Settings .....	290
Configuration Settings .....	274	Plot Renderer Settings .....	290
Plot Renderer Settings .....	276	About the Equilibrium with	
About the Crack Susceptibility		Freeze-in Temperature Property	
Coefficient Property Model .....	277	Model .....	292
Input Parameters, Scheil		Equilibrium with Freeze-in	
Settings, and CSC Model		Temperature Property Model	
Selection .....	277	Settings .....	293
Clyne and Davis CSC Model		Configuration Settings .....	293
Theory and Background .....	278		

---

Plot Renderer Settings .....	296
Equilibrium Property Model Settings .....	297
Configuration Settings .....	297
Plot Renderer Settings .....	298
About the Interfacial Energy Property Model .....	300
Interfacial Energy Property Model Settings .....	301
Configuration Settings .....	301
Plot Renderer Settings .....	302
Liquid and Solidus Temperature Property Model Settings .....	303
Configuration Settings .....	303
Plot Renderer Settings .....	304
Phase Transition Property Model Settings .....	306
Configuration Settings .....	306
Plot Renderer Settings .....	307
Scheil Property Model Settings ....	309
Configuration Settings .....	309
Plot Renderer Settings .....	312
About the Spinodal Property Model .....	314
Spinodal Property Model Settings .....	315
Configuration Settings .....	315
Plot Renderer Settings .....	316
About the T0 Temperature	317

---

Property Model .....	
Vary Temperature or Conditions .....	317
Gibbs Energy Addition .....	318
T0 Temperature Property Model Settings .....	319
Configuration Settings .....	319
Plot Renderer Settings .....	321
About the Yield Strength Property Model .....	322
Intrinsic Strength .....	322
Constant Addition of Strength	324
Grain Boundary Strengthening	324
Solid Solution Strengthening ..	324
Precipitation Strengthening ...	325
Mean Precipitate Radius and Particle Size Distribution (PSD)	328
Yield Strength Property Model: Simplified Mode Settings .....	330
General Configuration Settings .....	330
Simplified Mode Settings ....	331
Additional Settings .....	332
Yield Strength Property Model: Advanced Mode Settings .....	332
General Configuration Settings .....	333
Advanced Mode Settings ....	333

Additional Precipitation Parameters (Advanced Mode) .....	337	By Plot Type .....	364
References .....	339	By Calculation Type .....	366
Additional Settings .....	339	Plot Type: Cross Plot and Statistical .....	369
Yield Strength Property Model: Calculation Type and Grid Definitions .....	339	Statistical (with a Histogram or Probability Axis Type) .....	369
Calculation Type Settings .....	340	Cross Plot (Parity Plot) .....	371
Grid Definition Settings .....	340	Plot Type: Parallel Coordinates .....	374
Additional Settings .....	341	Tips to Interpret the Plot Data .....	375
Yield Strength Property Model: Plot and Table Settings .....	341	Visualization Window Plot Options .....	375
<b><i>Scheduler and Event Log Windows 344</i></b>		Select and Deselect Lines .....	376
Using the Scheduler .....	345	Save As .....	378
Event Log Window .....	347	Copy .....	378
<b><i>Working with Plots and Tables 348</i></b>		Plot Renderer Configuration Window Options .....	379
Performing Projects, Trees and Activities .....	349	Filter Quantities .....	379
Saving and Printing the Output (Graphical Mode) .....	350	Plot Type: Heat Map, Contour, and 3D .....	380
Saving Diagrams .....	350	Heat Map .....	380
Saving (Exporting) Tables .....	351	Contour .....	382
Printing Output .....	352	3D .....	383
Plotting Options .....	353	Plot Type: Flexible Mode and Grouped Mode .....	385
Common Quantities Available for Plots and Tables .....	358	Flexible Mode .....	385
Plot Types .....	362	Grouped Mode .....	385
By Calculator .....	363	Switching Between Modes .....	385
		Plot Type: CCT Mode .....	387
		Plot Type: TTT Mode .....	389

Plot Type: Simple Mode and Advanced Mode .....	391	Plot Labels .....	407
Simple Mode .....	391	Plot Labels: Entering a Mixture of Plain Text and Equations .....	408
Advanced Mode .....	391	Plot Labels: Rotation Angle and Show Anchor .....	410
Switching Between Modes .....	391	Moving Plot Labels .....	411
Zooming In and Out .....	393	About Legend Styles .....	412
Resetting a Plot to the Default View .....	393	Legend and Label Styles .....	413
Plotting Several Calculations in One Diagram .....	394	Legend Styles .....	413
Plot Renderer: Convert a Plot to a Table and Export the Data .....	396	Label Styles .....	413
Converting a Plot to a Table on the Plot Renderer .....	396	<b>Additive Manufacturing (AM) Module .....</b>	<b>416</b>
Switching Between a Plot and a Table in the Visualizations Window .....	396	About the Additive Manufacturing (AM) Module .....	417
Saving the Table (Exporting the Data) .....	396	Overview of the Content .....	418
Merging Plots into One Diagram .....	398	Databases .....	419
Adding Axes to a Diagram .....	400	Available Options .....	420
Rotating 3D Plots .....	402	Using the Templates .....	420
<b>Plot and Table Formatting .....</b>	<b>403</b>	Material Properties and Libraries .....	423
Setting Background Colours .....	404	Create or Import Your Own Material Library .....	423
Changing the Plot Properties .....	405	Using Data with other Add-on Modules .....	424
Edit Individual Plot Lines .....	405	Additive Manufacturing (AM) Module and TC-Python .....	424
Color Switching Plots Between Light and Dark Mode .....	405	Additive Manufacturing (AM) Module Help Resources .....	425
Additional Plot Color Changes are Locked to the Original Mode .....	406	'Online' Help: No Internet Required .....	425

<ul style="list-style-type: none"> <li>Additive Manufacturing (AM) <ul style="list-style-type: none"> <li>Module Specific Information ..425</li> </ul> </li> <li>Network License Restrictions ..... 426</li> <li>General Theory and Background ..427 <ul style="list-style-type: none"> <li>Additive Manufacturing <ul style="list-style-type: none"> <li>Simulation Types .....428 <ul style="list-style-type: none"> <li>Steady-state .....428</li> <li>Transient .....429</li> <li>Transient with Heat Source <ul style="list-style-type: none"> <li>from Steady-state ..... 429</li> </ul> </li> </ul> </li> </ul> </li> <li>Additive Manufacturing <ul style="list-style-type: none"> <li>Module Theory .....430 <ul style="list-style-type: none"> <li>Energy Equation: The <ul style="list-style-type: none"> <li>Enthalpy Formulation ..... 430</li> </ul> </li> <li>The Equivalent Heat <ul style="list-style-type: none"> <li>Capacity Method Using <ul style="list-style-type: none"> <li>Thermo-Calc .....432</li> </ul> </li> <li>The Enthalpy Method Using <ul style="list-style-type: none"> <li>Thermo-Calc .....434</li> </ul> </li> </ul> </li> <li>Steady-state Formulation ... 435</li> <li>Fluid Flow .....435</li> <li>Modeling Heat Source ..... 436 <ul style="list-style-type: none"> <li>Gaussian Heat Source .....436 <ul style="list-style-type: none"> <li>Double Ellipsoidal <ul style="list-style-type: none"> <li>(Goldak) Heat Source .....437</li> </ul> </li> <li>Conical Heat Source .....438</li> </ul> </li> </ul> </li> <li>Numerical Boundary <ul style="list-style-type: none"> <li>Conditions .....438 <ul style="list-style-type: none"> <li>Boundary Conditions for <ul style="list-style-type: none"> <li>Energy Equation ..... 439</li> </ul> </li> </ul> </li> </ul> </li> </ul> </li> </ul> </li> </ul> </li> </ul>	<ul style="list-style-type: none"> <li>Boundary Conditions for <ul style="list-style-type: none"> <li>Navier-Stokes Equations ..440</li> </ul> </li> <li>Thermophysical Properties <ul style="list-style-type: none"> <li>of Powder Material .....440</li> </ul> </li> <li>References .....441</li> <li>About Heat Source Calibration441 <ul style="list-style-type: none"> <li>Heat Source Calibration <ul style="list-style-type: none"> <li>with Experiment Data ..... 443</li> </ul> </li> <li>Examples Related to Heat <ul style="list-style-type: none"> <li>Source Calibration .....443</li> </ul> </li> <li>Reference .....443</li> </ul> </li> <li>About the Keyhole Model ..... 444 <ul style="list-style-type: none"> <li>References .....446</li> </ul> </li> <li>Additive Manufacturing Workflow447 <ul style="list-style-type: none"> <li>Setting Up the Additive <ul style="list-style-type: none"> <li>Manufacturing Simulation ..... 448</li> </ul> </li> <li>General Overview ..... 448</li> <li>Workflow 1: When You <ul style="list-style-type: none"> <li>Need to Obtain the Material <ul style="list-style-type: none"> <li>Properties Data for the <ul style="list-style-type: none"> <li>Calculation ..... 449</li> </ul> </li> </ul> </li> </ul> </li> <li>Workflow 2: When You <ul style="list-style-type: none"> <li>Already Have the Material <ul style="list-style-type: none"> <li>Properties Data Available - <ul style="list-style-type: none"> <li>Material Library ..... 449</li> </ul> </li> </ul> </li> <li>System Definer .....449</li> <li>Scheil Calculator (Workflow <ul style="list-style-type: none"> <li>1) .....450</li> </ul> </li> <li>Materials Libraries <ul style="list-style-type: none"> <li>(Workflow 1 or 2) .....450</li> </ul> </li> <li>AM Calculator ..... 450</li> </ul> </li> </ul> </li> </ul>
---	--

Visualizations Window .....	451	AM Calculator: Working with Probe Data .....	474
Additive Manufacturing Templates .....	452	Workflow to Include Probe Data from Diffusion or Precipitation Simulations ....	474
Working with the Activity Nodes for an AM Simulation .....	454	Probes .....	474
AM Calculator .....	455	Condense Time- Temperature Data .....	475
AM Calculator Conditions Settings .....	456	AM Plot Types .....	475
Calculation Type (Steady- state) .....	457	AM Calculator Plot Renderer Successors .....	476
Global Settings .....	458	Table View (2D) .....	476
Geometry .....	460	3D Plot .....	477
Heat Source .....	461	3D Plot Display Options ...	477
Keyhole Model Settings ...	463	3D Slice Plot .....	477
Scanning Strategy .....	464	Plot Over Line (2D) .....	477
Top Boundary Conditions ...	465	Probe (2D) .....	478
Probe Positions (Transient Simulations Only) .....	466	Heat Source Calibration .....	478
Using Probe Data in Diffusion and Precipitation Calculations	466	AM Calculator Plot Renderer .	478
Heat Source Calibration Experiment File Requirements	467	Plot Settings .....	479
Experiment File Requirements .....	467	Working with AM Visualizations .....	484
Working with the Experiment Data Table .....	469	How the Project Nodes are Connected to the Configuration and Visualizations Tabs .....	484
AM Calculator Materials Properties Settings .....	470	Configuration Window ....	485
AM Calculator Options Settings .....	472	Visualizations Window ....	486
		Visualizing Heat Source Calibrations .....	488

Visualizing Heat Source Calibrations .....	489	Process Metallurgy Calculator .....	505
AM Calculator Experiment Data .....	489	Network License Restrictions .....	506
Plot Renderer Configuration Window .....	492	Setting Up a Process Metallurgy Simulation .....	507
Adding the Heat Source Calibration Plots .....	492	Equilibrium Simulations .....	508
Navigating the Heat Source Calibration Table .....	493	Process Simulations .....	509
Working with the Heat Source Calibration Parameters .....	494	Calculation Types .....	509
Visualizations Window .....	495	Working with the Process Metallurgy Module .....	510
Heat Source Calibration Parameters Plot .....	495	Working with the Process Schedule .....	519
Melt Pool Dimensions Plot .....	496	About Dynamic Time Stepping .....	521
3D Plot and Plot Over Line	497	About Pressure-Dependent Calculations .....	523
AM Visualizations Window Plot Toolbar .....	498	Working with the Material Manager .....	524
Rotate, Zoom, and Pan 3D Plots: AM Calculations .....	500	Process Metallurgy Slag Properties .....	526
<b><i>The Process Metallurgy Module</i></b> .....	<b>501</b>	References .....	528
About the Process Metallurgy Module .....	502	Adiabatic Calculations: Material Enthalpy .....	529
Available Options with the Process Metallurgy Module .....	504	Process Metallurgy Calculator .....	530
Using the Template .....	504	Conditions Tab .....	531
Process Metallurgy Template .....	505	Options Tab .....	531
		Plots and Tables .....	532
		Defining the Equilibrium Simulation .....	533
		Equilibrium Simulation: Conditions Tab .....	538

Defining the Process Simulation ...	544	Plot Renderer Settings .....	574
Process Simulation: Conditions Tab .....	548	About the Equilibrium with Freeze-in Temperature Nickel Property Model .....	575
Process Simulation: Process Model Settings .....	550	References .....	576
Edit Process Model .....	550	Equilibrium with Freeze-in Temperature Nickel Property Model Settings .....	577
Process Simulation: Materials Tab	554	Configuration Settings .....	577
Process Simulation: Process Schedule Tab .....	556	Plot Renderer Settings .....	581
Options Tab .....	558	About the Solvus for Ordered Phase Property Model .....	583
Material Manager .....	558	Solvus for Ordered Phase Property Model Settings .....	583
Process Material Equilibrium and Grid .....	559	Configuration Settings .....	584
Timestep Control .....	561	Advanced Settings .....	585
<b><i>Nickel Model Library Property Models</i></b> .....	<b>562</b>	Plot Renderer Settings .....	586
About the Nickel Model Library Property Models .....	563	About the Strain-Age Cracking Property Model .....	587
About the Antiphase Boundary Energy Property Model .....	565	Single Temperature Mode .....	588
References .....	565	Temperature Interval Mode .....	588
Antiphase Boundary Energy Property Model Settings .....	567	Strain-Age Cracking Property Model Settings .....	588
Configuration Settings .....	567	Configuration Settings .....	589
Plot Renderer Settings .....	569	Plot Renderer Settings .....	591
About the Coarsening Nickel Property Model .....	571	<b><i>Steel Model Library Property Models</i></b> .....	<b>592</b>
Coarsening Nickel Property Model Settings .....	572	About the Steel Model Library Property Models .....	593
Configuration Settings .....	572	Recommended Composition Ranges for Steel Models .....	596

---

References .....	598
About the Bainite Property Model	599
References .....	600
Bainite Property Model Settings .....	601
Configuration Settings .....	601
Plot Renderer Settings .....	602
About the CCT Diagram Property Model .....	603
CCT Diagram Property Model Settings .....	603
Configuration Settings .....	604
Calculation Setting .....	605
Custom Calculation Setting .....	607
Plot Renderer Settings .....	610
About the Critical Transformation Temperatures Property Model .....	612
Critical Transformation Temperatures Property Model Settings .....	613
Configuration Settings .....	613
Plot Renderer Settings .....	615
About the Ferrite Property Model	616
Growth Models .....	616
Nucleation Rate and Interfacial Mobility .....	616
Reference .....	617
Ferrite Property Model Settings .....	617

---

Included Elements and Important Notes .....	617
Configuration Settings .....	618
Custom Parameters .....	620
Plot Renderer Settings .....	621
Reference .....	621
About the Martensite Fractions Property Model .....	622
Reference .....	622
Martensite Fractions Property Model Settings .....	623
Configuration Settings .....	623
Plot Renderer Settings .....	624
About the Martensite Temperatures Property Model .....	625
References .....	625
Martensite Temperatures Property Model Settings .....	626
Configuration Settings .....	626
Plot Renderer Settings .....	627
About the Martensitic Steel Strength Property Model .....	628
Tempering .....	629
Note about Hardness Data .....	632
References .....	632
Martensitic Steel Strength Property Model Settings .....	633
System Definer Configuration Settings .....	633

Property Model Calculator Configuration Settings .....	633	T_05: Stable and the Metastable Fe-C Phase Diagrams .....	14
Plot Renderer Settings .....	635	T_06: Serially Coupled Equilibrium Calculators .....	16
About the Pearlite Property Model .....	637	Table Renderer Results .....	16
References .....	638	T_07: User-Defined Functions .....	19
Pearlite Property Model Settings .....	639	T_08: Scheil and Equilibrium Solidification .....	22
Configuration Settings .....	639	To Save the Table (via Table Renderer) .....	24
Plot Renderer Settings .....	641	T_09: Carbide Driving Force Heat Map .....	26
About the TTT Diagram Property Model .....	642	T_10: Scheil Solidification with Back Diffusion .....	28
TTT Diagram Property Model Settings .....	642	T_11: Surface Tension in Cu-Zr .....	30
Configuration Settings .....	643	T_12: Viscosity in Cr-Ni .....	33
Calculation Setting .....	644	T_13: Scheil Solidification with Solute Trapping .....	36
Custom Calculation Setting .....	646	T_14: Fe-Cr-Ni Transition Comparison Using the Material to Material Calculator .....	38
Plot Renderer Settings .....	648	T_15: Molar Volume and Thermal Expansion Coefficients for L12-type Al3Sc and Al3Zr Compounds .....	41
<b>Graphical Mode Examples Guide .....</b>	<b>1</b>	T_16: Electrical Resistivity and Thermal Conductivity for Pure Copper .....	44
<b>Thermo-Calc Examples Collection .....</b>	<b>1</b>	T_17: Al2O3-MgO Phase Diagram .....	48
Graphical Mode Examples List by Product .....	2	T_18: Scheil with Delta Ferrite to Austenite Transition .....	50
T_01: Calculating a Single-Point Equilibrium .....	3	Reference .....	51
T_02: Stepping in Temperature in the Fe-C System .....	5		
T_03: Fe-C Phase Diagrams .....	8		
T_04: Fe-Cr-C Ternary Phase Diagram at 1000 K .....	11		

<b><i>Thermo-Calc General Property Models Examples Collection</i></b> .....	<b>52</b>
PM_G_01: Phase Transition .....	54
PM_G_02: Coarsening and Interfacial Energy .....	58
PM_G_03: Driving Force and Interfacial Energy .....	64
PM_G_04: Yield Strength .....	67
References .....	71
PM_G_05: Yield Strength NiAlCr .....	72
References .....	73
PM_G_06: Yield Strength HEA .....	74
Reference .....	75
PM_G_07: Hot Crack Susceptibility .....	76
References .....	77
PM_G_08: Spinodal .....	78
PM_G_09: T-Zero Temperature .....	80
PM_G_10: Freeze-in Thermal Conductivity .....	82
Reference .....	83
PM_G_11: Freeze-in Electrical Resistivity .....	84
Reference .....	85
PM_G_12: Solidus and Liquidus Batch Calculation .....	86
PM_G_13: Ti Alloy Design for Additive Manufacturing .....	89
Reference .....	91
PM_G_14: Ti_Fe_T-Zero_ .....	92

Martensite .....	
Reference .....	93
PM_G_15: Columnar to Equiaxed Transition (CET) of a NiAlCr Alloy .....	94
Reference .....	96
PM_G_16: Columnar to Equiaxed Transition of a CMSX-4 Alloy .....	97
References .....	100
PM_G_17: Columnar to Equiaxed Transition of an IN718 Alloy .....	101
Reference .....	104
<b><i>Steel Model Library Examples Collection</i></b> .....	<b>105</b>
PM_Fe_01: Fe-Cr-C Martensite with Intercritical Annealing .....	106
Reference .....	108
PM_Fe_02: Fe-Mn Martensite Morphologies .....	109
PM_Fe_03: Fe-C-Mn Pearlite .....	111
Reference .....	114
PM_Fe_04: Critical Temperatures .....	115
PM_Fe_05: Fe-C-Mn-Si-Ni-Cr-Mo Bainite .....	118
Reference .....	119
PM_Fe_06: TTT Diagram for Fe-C-Mn-Si-Cr-V .....	120
Reference .....	122
PM_Fe_07: Hardenability Design of Steel .....	123

PM_Fe_08: CCT Diagram for Fe-C-Mn-Si-Cr-V .....	125
Reference .....	127
PM_Fe_09: Fe-C-Ni Ferrite .....	128
References .....	131
PM_Fe_10: Martensitic Steel Strength .....	132
Reference .....	134
<b><i>Nickel Model Library Examples Collection</i></b> .....	<b>135</b>
PM_Ni_01: Lattice Parameter of $\gamma/\gamma'$ .....	136
Reference .....	137
PM_Ni_02: Antiphase Boundary Energy of $\gamma'$ .....	138
References .....	139
PM_Ni_03: Critical Temperatures of Alloy 718 .....	140
Reference .....	141
PM_Ni_04: Strain Age Cracking (SAC) .....	142
Reference .....	144
<b><i>Process Metallurgy Module Examples Collection</i></b> .....	<b>145</b>
PMET_01: Basic Oxygen Furnace (BOF) .....	146
PMET_02: Desulphurization in a Ladle Furnace (LF) .....	148
PMET_03: Argon Oxygen Decarburization (AOD) .....	150
Background Description .....	150

Slag Basicity .....	152
S Content in Steel .....	153
Cr Content in Slag .....	154
Amount of Slag .....	155
Liquid Slag Fraction .....	156
PMET_04: Basic Oxygen Furnace (BOF) Kinetics .....	157
PMET_05: Lab Scale Ladle Furnace (LF) Kinetics .....	158
About the Plot Results .....	159
Setting Up the LF Process Simulation .....	159
Results and Experimental Analysis .....	160
Reference .....	163
PMET_06: Ladle Furnace (LF) Kinetics .....	164
Background Overview .....	165
PMET_07: Vacuum Oxygen Decarburization Kinetics .....	167
Reference .....	167
PMET_08: Steel Deoxidation on Tapping .....	168
About Deoxidation or “Killing” of Steel .....	168
<b><i>Diffusion Module (DICTRA) Examples Collection</i></b> .....	<b>170</b>
D_01: Homogenization of a Binary Fe-Ni Alloy .....	171
Visualizations .....	171

D_02: Ferrite(bcc)/Austenite(fcc) Transformation in a Binary Fe-C Alloy .....	174
Visualizations .....	174
D_03: Evolution of an Fe-Cr-Ni Diffusion Couple .....	178
Visualizations .....	178
References .....	180
D_04: Fe-C Moving Boundary: Austenite to Ferrite .....	181
Visualizations .....	181
D_05: $\gamma/\alpha/\gamma$ Diffusion Couple of Fe-Ni-Cr alloys .....	186
Visualizations .....	187
References .....	194
D_06: Diffusion Through a Tube Wall .....	195
Visualizations .....	195
D_07: Multiphase Carburization of an Alloy .....	200
Visualizations .....	200
Reference .....	204
D_08: Microsegregation During Solidification .....	205
Visualizations .....	205
D_09: Ni Post Weld Heat Treatment .....	208
Reference .....	212
D_10: Iron (Fe) Homogenization in Scheil .....	213

Visualizations .....	213
Reference .....	220

***Precipitation Module (TC-PRISMA) Examples Collection .....*** **221**

P_01: Isothermal Precipitation of Al <sub>3</sub> Sc .....	223
Example Settings .....	223
Visualizations .....	224
References .....	227
Tutorial-P_01-Isothermal .....	228
P_02: Stable and Metastable Carbides - Isothermal .....	237
Example Settings .....	237
Visualizations .....	238
P_03: Stable and Metastable Carbides - TTT Diagram .....	241
Example Settings .....	241
Visualizations .....	242
P_04: Precipitation of Iron Carbon Cementite .....	245
Example Settings .....	245
Visualizations .....	246
Reference .....	247
P_05: Precipitation of $\gamma'$ in Ni Superalloys - Isothermal .....	248
Example Settings .....	248
Visualizations .....	249
Reference .....	252

P_06: Precipitation of $\gamma'$ in Ni Superalloys - Non-isothermal .....	253
Example Settings .....	253
Visualizations .....	255
Reference .....	260
P_07: Continuous Cooling Transformation (CCT) Diagram of Ni-Al-Cr $\gamma$ - $\gamma'$ .....	261
Example Settings .....	261
Visualizations .....	262
Tutorial-P_07-CCT .....	265
P_08: Precipitation of Cu-Ti CU4TI with Assumptions of Sphere and Needle Morphologies .....	270
Example Settings .....	270
Visualizations .....	272
References .....	277
P_09: Precipitation of Al-Sc AL3SC with Assumption of Sphere and Cuboid Morphologies .....	278
Example Settings .....	278
Visualizations .....	280
References .....	284
Tutorial-P_09-Al-Sc-Sphere_ Cuboid .....	285
P_10: Initial Particle Size Distribution of Fe-Cr-C .....	294
Example Settings .....	294
Visualizations .....	296
P_11: Interfacial Energy Function	299

Example Settings .....	299
Visualizations .....	300
Reference .....	302
P_12: Comparing Growth Rate Models for an Al-Zr System .....	303
Example Settings .....	304
Visualizations .....	304
Reference .....	306
P_13: Paraequilibrium Precipitation of Cementite Fe-C- Cr .....	307
Example Settings .....	307
Visualizations .....	308
Reference .....	310
P_14: Grain Growth and the Zener Pinning Effect .....	311
Example Settings .....	311
Visualizations .....	312
Results Discussion .....	313
References .....	315
P_15: Smooth Transition from Paraequilibrium to Ortho- equilibrium .....	316
Example Settings .....	317
Visualizations .....	318
Reference .....	320
P_16 Isothermal Coarsening and a 3D to 2D Stereological Conversion	321
Example Settings .....	322

Visualizations .....	323	AM_05: Using AM Calculator Probe Data with the Diffusion Module (DICTRA) .....	364
Reference .....	328	Background .....	364
<b><i>Additive Manufacturing (AM)</i></b>		Configuration and Calculation Set Up .....	365
<b><i>Module Examples Collection</i></b> .....	<b>329</b>	Diffusion Calculations .....	370
AM_01: Transient Simulation of a Single Track .....	330	Visualizations .....	373
Material Properties .....	330	Reference .....	374
Reference .....	331	Other Resources .....	374
Model Configuration .....	331	AM_06a: Calibrating a Heat Source for a 316L Steel .....	375
Visualizations .....	333	Material Properties .....	376
Other Resources .....	337	Configuration and Calculation Set Up .....	376
AM_02: Transient and Steady- state Simulations of a Single Track	338	AM Calculator Configuration Settings .....	376
Material Properties .....	338	Visualizations .....	376
Reference .....	339	Calibrated Functions .....	377
Model Configuration .....	339	Reference .....	380
Visualizations .....	341	AM_06b: Using the Calibrated Heat Source for a 316L Steel .....	381
Other Resources .....	344	Material Properties .....	381
AM_03: Steady-state Simulations	345	Configuration and Calculation Set Up .....	381
Results Discussion .....	347	Visualizations .....	382
Visualizations .....	352	Gaussian Heat Source and Keyhole Model .....	382
Other Resources .....	352	Double Ellipsoidal Heat Source .....	384
AM_04: Scheil Transient Steady- state .....	353		
Visualizations .....	353		
Steady-state .....	354		
Transient Single Track .....	357		
Transient Multilayer .....	359		

Reference .....	385
<b>Thermo-Calc User Guide: Console Mode</b> .....	<b>1</b>
<b><i>Working in Console Mode</i></b> .....	<b>2</b>
The Console Mode Layout .....	3
Console Window and Tabs .....	6
Edit the Console Tab Properties .....	6
Open and Close Console Tabs ... ..	6
View, Copy, or Save a Command History .....	7
Console Mode Results Window .....	9
Event Log Window .....	10
Keyboard Shortcuts to Navigate Between Tabs .....	11
<b><i>Modules and Commands</i></b> .....	<b>13</b>
Console Mode Workflow .....	14
Defining a System in Console Mode .....	16
Console Mode Modules .....	18
Moving Between Modules and Submodules .....	23
Entering Command Names or Abbreviations .....	24
Using the Command Line Prompt .. ..	26
Main Menu Commands .....	27
Wild Card Characters .....	28
<b><i>File Types</i></b> .....	<b>30</b>
Working with the Console Mode File Types .....	31
Opening Console Mode Macro Files .....	32

Opening a Macro from the Main Menu .....	32
Opening a Macro Using a Command .....	33
Drag and Drop a Macro File into the Console Window .....	33
Log Files .....	36
Macro Files .....	37
Workspace Files .....	40
Links to Workspace Commands .....	41
Editing the Default Console Directory .....	43
Experimental Data Files (*.POP, *.DOP) .....	46
<b><i>Calculations</i></b> .....	<b>49</b>
About the Gibbs Energy System (GES) Module .....	50
Calculation Engine: GES5 vs GES6 .....	50
Equilibrium Calculations .....	52
The Equilibrium Calculation Module (POLY3) .....	53
Setting Conditions for Equilibrium Calculations .....	54
Calculating an Equilibrium .....	57
Calculating an Equilibrium with a Fixed Phase .....	58
Calculating an Equilibrium with Suspended or Dormant Phases .....	59
Property Diagrams .....	60
About Property Diagrams .....	61

Calculating and Plotting a Property Diagram .....	62	POTENTIAL_DIAGRAM .....	86
Calculating a Property Diagram One Phase at a Time .....	64	Calculating a Potential Diagram	88
Calculating Several Properties in the Same Diagram .....	65	Calculating a Potential Diagram with Different Pressure .....	90
Phase Diagrams .....	66	Scheil Simulations .....	91
About Phase Diagrams .....	67	About the Scheil-Gulliver Solidification Simulations .....	92
Calculating and Plotting a Phase Diagram .....	68	Classic Scheil Simulation .....	92
About Binary Phase Diagrams ..	69	Scheil Simulation with Back Diffusion in Primary Phase ....	93
BINARY_DIAGRAM .....	70	Scheil with Solute Trapping ...	93
About Ternary Phase Diagrams	70	Other Options: Fast Diffusers and Scheil Simulations .....	95
TERNARY_DIAGRAM .....	72	Homogenization, Segregation Profiles, and the Diffusion Module .....	96
About Quasi-Binary and Quasi- Ternary Phase Diagrams .....	72	Summary .....	96
Calculating a Quasi-Binary Phase Diagram .....	74	Scheil in Graphical Mode vs Console Mode .....	97
Calculating a Quasi-Ternary Phase Diagram .....	75	Examples of Scheil- Gulliver Calculations .....	98
T0 Temperature Simulations .....	76	Simulating a Scheil Solidification Process .....	100
About T0 Temperature Simulations .....	77	State Variables for Scheil .....	102
Making a T0 Temperature Simulation .....	78	Aqueous Solutions and Pourbaix Diagrams .....	105
Paraequilibrium .....	80	About Aqueous Solutions and the Pourbaix Diagram .....	106
About Paraequilibrium .....	81	Calculating a Pourbaix Diagram .....	109
Calculating a Paraequilibrium ..	82	Plotting More Pourbaix .....	112
Potential Diagrams .....	85		
About Potential Diagrams .....	86		

Diagrams .....	Commands to Save and Print Diagrams .....	131
Stepping Calculations in an Aqueous Solution .....	Commands to Save Tables .....	132
POURBAIX_DIAGRAM Module	Saving Diagrams .....	132
Getting started with the Pourbaix module .....	Printing Diagrams .....	133
What is a Pourbaix diagram? .....	Modifying Diagrams .....	134
What database should I use? .....	Loading Saved Diagrams .....	136
Choosing a custom aqueous solution database .....	Working with the Console Mode Output .....	137
Copy and Rename the POLY3 file .....	Pause, Resume or Stop Output to the Console Window .....	137
Enforce a Pause .....	Controlling Console Output .....	137
Pourbaix Main Options .....	<b>Commands by Module .....</b>	<b>138</b>
Tabulation Reactions .....	System Utilities (SYS) Commands .....	139
About the Tabulation Reaction Module .....	ABOUT .....	140
Tabulating a Reaction .....	BACK .....	141
Tabulating a Substance or Solution Phase at Fixed Composition .....	CLOSE_FILE .....	142
<b>Results .....</b>	DISPLAY_LICENSE_INFO .....	143
Console Mode Results .....	EXIT .....	144
Using the POST Module .....	GOTO_MODULE .....	145
Plotting Diagrams .....	HELP (? and ??) .....	146
Saving and Printing the Output in Console Mode .....	HELP .....	147
	HP_CALCULATOR .....	148
	INFORMATION .....	149
	MACRO_FILE_OPEN .....	150
	OPEN_FILE .....	153
	SET_COMMAND_UNITS .....	154

SET_ECHO .....	155	AMEND_STORED_EQUILIBRIA	185
SET_GES_VERSION .....	156	CHANGE_STATUS .....	187
SET_INTERACTIVE .....	157	COMPUTE_EQUILIBRIUM .....	192
SET_INTERACTIVE_MODE .....	158	COMPUTE_TRANSITION .....	194
SET_LOG_FILE .....	159	CREATE_NEW_EQUILIBRIUM	196
SET_PLOT_ENVIRONMENT .....	160	DEFINE_COMPONENTS .....	197
SET_TC_OPTIONS .....	161	DEFINE_DIAGRAM .....	198
STOP_ON_ERROR .....	162	DEFINE_MATERIAL .....	201
Database (TDB) Module		DELETE_INITIAL_	
Commands .....	163	EQUILIBRIUM .....	205
AMEND_SELECTION .....	164	DELETE_SYMBOL .....	206
APPEND_DATABASE .....	165	ENTER_SYMBOL .....	207
DATABASE_INFORMATION .....	166	EQUILIBRIUM_CALCUL .....	210
DEFINE_ELEMENTS .....	167	Settings for the	
DEFINE_SPECIES .....	168	Minimization of an	
DEFINE_SYSTEM .....	169	Equilibria .....	210
GET_DATA .....	170	EVALUATE_FUNCTIONS .....	212
LIST_DATABASE .....	171	GLOBAL_MINIMIZATION .....	213
LIST_SYSTEM .....	172	KEEP_COMP_SET_NUMBERS	215
REJECT .....	173	LIST_AXIS_VARIABLE .....	216
RESTORE .....	174	LIST_CONDITIONS .....	217
SET_AUTO_APPEND_		LIST_EQUILIBRIUM .....	218
DATABASE .....	175	LIST_INITIAL_EQUILIBRIA .....	219
SWITCH_DATABASE .....	176	LIST_PHASE_ADDITION .....	220
POLY Module Commands .....	177	LIST_STATUS .....	221
ADD_INITIAL_EQUILIBRIUM	181	LIST_SYMBOLS .....	222
ADVANCED_OPTIONS .....	184	LOAD_INITIAL_EQUILIBRIUM	223

MAKE_COMPONENT_	
ENTERED .....	224
MAKE_COMPONENT_	
SUSPENDED .....	225
MAJOR_CONSTITUENTS .....	226
MAP .....	227
NEW_COMPOSITION_SET .....	229
OUTPUT_FILE_FOR_SHOW .....	231
PARAEQUILIBRIUM .....	232
PHASE_ADDITION .....	235
POST .....	236
PRESENT_PHASE .....	237
READ_WORKSPACES .....	238
REINITIATE_MODULE .....	239
SAVE_WORKSPACES .....	240
Important Information	
About Map and Step	
Commands .....	240
SELECT_EQUILIBRIUM .....	242
SET_ALL_START_VALUES .....	243
SET_AXIS_VARIABLE .....	245
SET_CONDITION .....	247
SET_INPUT_AMOUNTS .....	250
SET_INTERACTIVE .....	251
SET_NUMERICAL_LIMITS .....	252
SET_REFERENCE_STATE .....	254
SET_START_CONSTITUTION .....	256
SET_START_VALUE .....	257

SHOW_FOR_T= .....	258
SHOW_VALUE .....	260
STABILITY_CHECK .....	261
STEP_AND_MAP .....	262
STEP_WITH_OPTIONS .....	264
TABULATE .....	277
TOGGLE_ALTERNATE_MODE .....	278
T-ZERO_TEMPERATURE .....	279
Gibbs Energy System	
(GES) Commands .....	280
Solution Models in the GIBBS	
(GES) Module .....	283
About the ENTER_PHASE	
Command .....	291
ADD_COMMENT .....	295
AMEND_ELEMENT_DATA .....	296
AMEND_PARAMETER .....	298
AMEND_PHASE_DESCRIPTION .....	301
AMEND_SYMBOL .....	305
CHANGE_STATUS .....	308
ENTER_ELEMENT .....	309
ENTER_FUNCTION .....	310
ENTER_PARAMETER .....	312
ENTER_PHASE .....	314
ENTER_SPECIES .....	316
ENTER_SYMBOL .....	317
LIST_CONSTITUTION .....	321
LIST_DATA .....	322

Command Prompts .....	322	LIQUID_PHASE_NAME .....	344
Output Differences: GES5 vs GES6 .....	323	SAVE_FILE_NAME .....	345
LIST_PARAMETER .....	325	SET_NUMERICAL_LIMITS .....	346
LIST_PHASE_DATA .....	327	SKIP_STEP_CALCULATION .....	347
Output Differences: GES5 vs GES6 .....	327	SOLUTE_TRAPPING .....	347
LIST_STATUS .....	328	START_WIZARD .....	348
LIST_SYMBOL .....	331	Back Diffusion and Solute Trapping Settings .....	350
READ_GES_WORKSPACE .....	332	TEMPERATURE_STEP .....	352
READ_WORKSPACES .....	333	TERMINATION_CRITERIA .....	353
REINITIATE .....	334	TEST_INTERVAL_FOR_GLOBAL .....	353
SAVE_GES_WORKSPACE .....	335	USE_BACK_DIFFUSION .....	354
SAVE_WORKSPACES .....	336	PARROT Module Commands .....	355
Important Information About Map and Step Commands .....	336	About the ENTER_ PARAMETER Command .....	358
SCHEIL Module Commands .....	338	AMEND_PARAMETER .....	362
CALCULATE_BELOW_SOLIDUS .....	339	AMEND_SYMBOL .....	365
CALCULATE_EVAPORATION_ PROPERTIES .....	339	COMPILE_EXPERIMENTS .....	368
CALCULATE_FROM_START_ TEMPERATURE .....	340	CONTINUE_OPTIMIZATION .....	369
CALCULATE_FROM_GAS .....	340	CREATE_NEW_STORE_FILE .....	370
DELTA_FERRITE_AUSTENITE_ TRANSITION .....	341	EDIT_EXPERIMENTS .....	371
EVALUATE_SEGREGATION_ PROFILE .....	342	ENTER_PARAMETER .....	372
GLOBAL_MINIMIZATION .....	343	LIST_ALL_VARIABLES .....	374
		LIST_CONDITIONS .....	375
		LIST_PARAMETER .....	376
		LIST_PHASE_DATA .....	378
		LIST_RESULT .....	379

LIST_STORE_FILE .....	384	EXPERIMENT .....	419
LIST_SYMBOL_IN_GES .....	385	EXPORT .....	421
OPTIMIZE_VARIABLES .....	386	FLUSH_BUFFER .....	422
READ_PARROT_WORKSPACES	387	GRAPHICS_PLOT .....	423
RECOVER_VARIABLES .....	388	IMPORT .....	424
REINITIATE .....	389	LABEL_DATA .....	425
RESCALE_VARIABLES .....	390	LIST_ALL_EQUILIBRIA .....	426
SAVE_PARROT_WORKSPACES	391	LIST_CONDITIONS .....	427
SET_ALTERNATE_MODE .....	392	LIST_EQUILIBRIUM .....	428
SET_EXTERNAL_PARAMETER	393	LIST_STATUS .....	429
SET_FIX_VARIABLE .....	394	LIST_SYMBOLS .....	430
SET_OPTIMIZING_CONDITION	395	MAKE_POP_FILE .....	431
SET_OPTIMIZING_VARIABLE ..	397	READ_WORKSPACES .....	432
SET_OUTPUT_LEVELS .....	398	REINITIATE_MODULE .....	433
SET_SCALED_VARIABLE .....	399	RESTORE_ALL_WEIGHTS .....	434
SET_STORE_FILE .....	400	SAVE_WORKSPACES .....	435
EDIT_EXPERIMENTS (ED_EXP)		SELECT_EQUILIBRIUM .....	436
Submodule Commands .....	401	SET_ALL_START_VALUES .....	437
ADVANCED_OPTIONS .....	404	SET_ALTERNATE_CONDITION	439
CHANGE_STATUS .....	405	SET_CONDITION .....	442
COMMENT .....	410	SET_NUMERICAL_LIMITS .....	445
COMPUTE_ALL_EQUILIBRIA ...	411	SET_REFERENCE_STATE .....	447
CREATE_NEW_EQUILIBRIUM	412	SET_START_CONSTITUTION ...	449
DEFINE_COMPONENTS .....	413	SET_START_VALUE .....	450
DELETE_SYMBOL .....	414	SET_WEIGHT .....	451
ENTER_SYMBOL .....	415	SHOW_VALUE .....	453
EVALUATE_FUNCTIONS .....	418		

STORE_ALL_WEIGHTS .....	454	APPEND_EXPERIMENTAL_	
TABLE_HEAD, TABLE_VALUES		DATA .....	501
and TABLE_END .....	455	CHANGE_LEGEND .....	502
TRANSFER_START_VALUES ....	457	DUMP_DIAGRAM .....	503
Tabulation Reaction		ENTER_SYMBOL .....	504
(TAB) Commands .....	458	FIND_LINE .....	507
ENTER_FUNCTION .....	459	LIST_DATA_TABLE .....	508
ENTER_REACTION .....	461	LIST_PLOT_SETTINGS .....	509
LIST_SUBSTANCES .....	463	LIST_SYMBOLS .....	510
SET_ENERGY_UNIT .....	466	MAKE_EXPERIMENTAL_	
SET_PLOT_FORMAT .....	467	DATAFILE .....	511
SWITCH_DATABASE .....	468	MODIFY_LABEL_TEXT .....	512
TABULATE .....	470	PLOT_DIAGRAM .....	513
TABULATE_DERIVATIVES .....	471	PRINT_DIAGRAM .....	514
TABULATE_REACTION .....	476	QUICK_EXPERIMENTAL_PLOT	515
TABULATE_SUBSTANCE .....	479	REINITIATE_PLOT_SETTINGS ..	516
REACTOR (REACTOR_SIMULATOR)		RESTORE_PHASE_IN_PLOT ....	517
Commands .....	484	SELECT_PLOT .....	518
CREATE_DIVIDERS .....	485	SET_AXIS_LENGTH .....	519
CREATE_PIPES .....	486	SET_AXIS_PLOT_STATUS .....	520
CREATE_STAGE_BOXES .....	487	SET_AXIS_TEXT_STATUS .....	521
EXECUTE_POLY3_COMMAND	489	SET_AXIS_TYPE .....	522
LIST_RECORDS .....	490	SET_COLOR .....	523
READ_WORKSPACE .....	491	SET_CORNER_TEXT .....	525
SAVE_WORKSPACES .....	492	SET_DIAGRAM_AXIS .....	526
START_SIMULATION .....	493	SET_DIAGRAM_TYPE .....	528
POST Module Commands .....	495	SET_FONT .....	529
ADD_LABEL_TEXT .....	498		

SET_INTERACTIVE_MODE .....	530	Multicomponent Phase Diagram	
SET_LABEL_CURVE_OPTION ..	531	Examples .....	565
SET_PLOT_FORMAT .....	532	Examples Calculating	
SET_PLOT_OPTIONS .....	533	Thermodynamic and Related	
SET_PLOT_SIZE .....	535	Quantities .....	567
SET_PREFIX_SCALING .....	536	Single Equilibrium Calculation	
SET_RASTER_STATUS .....	537	Examples .....	571
SET_REFERENCE_STATE .....	538	Potential Diagram Example .....	573
SET_SCALING_STATUS .....	540	Examples of Scheil-	
SET_TIC_TYPE .....	541	Gulliver Calculations .....	574
SET_TIELINE_STATUS .....	542	Example Calculating Chemical	
SET_TITLE .....	543	Ordering .....	576
SET_TRUE_MANUAL_SCALING	544	Examples of Gibbs Energy System	
SUSPEND_PHASE_IN_PLOT ....	545	(GES) .....	577
TABULATE .....	546	Notes Related to the	
<b>Technical Notes .....</b>	<b>547</b>	Example .....	579
Technical Note: Pipe or Redirect		Example Calculating the T0	
Output Data from Console Mode	548	Temperature .....	582
Windows .....	548	Examples Calculating	
Linux .....	548	Paraequilibrium .....	583
macOS .....	548	Example Using Functions and	
<b>Console Mode Examples .....</b>	<b>549</b>	Variables .....	584
Thermo-Calc Console Mode		Reference .....	584
Examples .....	550	Example Using PARROT and EDIT_	
Binary Phase Diagram Examples ..	552	EXPERIMENT .....	585
Property Diagram Examples .....	556	Pourbaix Diagram Examples .....	587
Ternary Phase Diagram Examples	561	The PAQ2 Database .....	587
		Additional Examples in this	
		Series .....	587
		Step 1: Single-Point	
		Calculations for H2O-NaCl	
		system .....	588

Step 2: Single-Point Calculations for Fe-X (X = Cr-Ni-Co) system .....	588
Step 3: Single-Point Calculations for Fe-Cr-Ni-Co + H <sub>2</sub> O-NaCl system .....	589
Step 4: Pourbaix Diagram Mapping for Fe-Cr-Ni-Co + H <sub>2</sub> O-NaCl system - define the variables .....	589
Pourbaix Examples TCEX40A, 40B, 40C, and 40E .....	590
POURBAIX_DIAGRAM Commands for Options 1 to 4 .....	594
Option 1: Start a Completely New Pourbaix Diagram Calculation .....	594
Chemical system (elements) .....	595
Initial bulk composition .....	595
Option 2: Open an Existing File and Plot Other Property Diagrams .....	600
Option 3: Open an Existing File and Make Another Pourbaix Calculation .....	602
Option 4: Open an Existing File and Make another Stepping Calculation .....	604
Example Using Tabulate Reaction (TAB) .....	607
Example Using REACTOR .....	608
Examples with Thermophysical Properties .....	609

Application Examples .....	612
<b>Data Optimization User Guide .....</b>	<b>1</b>
<b>Introduction .....</b>	<b>2</b>
Introduction to Data Optimization ..	3
Data Optimization .....	4
The Least-Squares Method .....	6
The CALPHAD Approach .....	7
References .....	7
<b>Working with Data Optimization in Thermo-Calc .....</b>	<b>8</b>
Data Optimization in Thermo-Calc ..	9
Data Optimization Workflow .....	10
The PARROT Module .....	12
The EDIT_EXPERIMENTS Module ...	13
Data Optimization in Other Modules .....	14
Optimization File Types .....	15
Optimization Workspaces .....	16
Method for Optimization and Simulation .....	17
<b>Creating the POP File .....</b>	<b>18</b>
Experimental Information .....	19
POP File Syntax .....	22
Entering Experimental Information	24
Creating a New Equilibrium .....	25
Setting Equilibrium Conditions and Uncertainty .....	26
Setting Reference States .....	27

Entering the Experimental Value and its Uncertainty .....	29	Optimizing the System in PARROT ..	50
Entering Many Equilibria as a Table .....	30	Changes that Require POP File Recompilation .....	51
Setting Site Fraction Start Values .....	32	Optimization Workflow .....	52
Commenting About an Equilibrium .....	33	Ensuring Computation of all Equilibria .....	53
Simultaneous Use of Binary and Ternary Experiments .....	34	Setting Weights .....	55
Using Stability Conditions .....	35	Optimizing and Evaluating Results ..	56
Grouping Equilibria with Labels .....	36	The Critical Set of Experiments ..	57
Dividing the POP File into Several Blocks .....	37	Continuing the Optimization and Resetting Variables .....	58
<b><i>The Setup File</i> .....</b>	<b>38</b>	Updating the Set of Optimizing Variables .....	60
Creating the Setup File .....	39	Reducing the Number of Optimizing Variables .....	62
Defining the System .....	40	Plotting Intermediate Results .....	63
Creating Additional Composition Sets of a Phase .....	40	Superimposing Experimental Data from EXP Files .....	64
Entering the Optimizing Variables ..	41	<b><i>Finalizing the Optimization</i> .....</b>	<b>65</b>
Entering Optimizing Variables in Parameter Specifications .....	42	Finishing the Optimization .....	66
Entering Interaction Parameters .....	44	Rounding off Optimizing Variable Values .....	67
Initializing Optimizing Variables in PARROT .....	45	Updating the Database File .....	68
Creating the Workspace File .....	46	Creating a Database File .....	69
Compiling the POP File .....	47	Updating the Setup File and the POP file .....	70
Returning Control to the Console ..	48	<b><i>Alternate Mode</i> .....</b>	<b>71</b>
<b><i>Optimizing in PARROT</i> .....</b>	<b>49</b>	About Alternate Mode .....	72
		Optimizing in Alternate Mode .....	73

Preparing the POP File for Alternate Mode .....	74	Gas .....	95
Examples of SET_ALTERNATE_CONDITION .....	75	Compound Energy Formalism .....	96
Entering Other Experiments in the POP with Alternate Mode ..	76	Substitutional Liquid .....	97
<b>Troubleshooting .....</b>	<b>77</b>	Ionic Two-Sublattice Liquid Model ..	98
Useful Guidelines .....	78	References .....	98
Excluding and Including the Correct Equilibria .....	80	Models for Ordered Phases .....	99
Conflicting Data .....	81	4SL Model for FCC, HCP, and BCC ..	100
<b>Experiments and Experimental Data</b>	<b>82</b>	2SL Model for FCC, HCP, and BCC ..	102
Phase Diagram Data .....	83	Magnetic Ordering .....	103
Data for Individual Compounds .....	84	Molar Volumes and High Pressure ..	104
Single Phase Mixing Enthalpies or Partial Enthalpies .....	86	Excess Models .....	105
Enthalpies of Formation and Enthalpies of Reactions .....	87	General Einstein Model .....	106
Chemical Potentials via EMF Measurements .....	88	<b>Database Manager User Guide .....</b>	<b>1</b>
Driving Force for Metastable Phases .....	89	<b>Introduction to the Database Manager Guide .....</b>	<b>2</b>
Pressure-Temperature-Volume EOS Data .....	90	Thermodynamic and Kinetic/Mobility Databases .....	3
Crystal Structure, Lattice Parameters, and Site-Occupancy ...	91	Initialization of the TDB Module .....	4
Magnetism and Other Atomistic Properties .....	92	<b>The Database Definition File and Keywords .....</b>	<b>6</b>
Data About Systems of Different Orders .....	93	Database Definition File Syntax .....	9
<b>Thermodynamic Models .....</b>	<b>94</b>	ELEMENT .....	11
		SPECIES .....	12
		PHASE .....	14
		Examples of Recommended Suffixes .....	14
		GIBBS Phase-type Codes .....	15
		Additional Clarification .....	16

CONSTITUENT .....	20	DIFFUSION .....	64
ADD_CONSTITUENT .....	22	ZEROVOLUME_SPECIES .....	65
COMPOUND_PHASE .....	23	<b>Database Definition File Examples .. 66</b>	
ALLOTROPIC_PHASE .....	24	Example 1: A Steel Database .....	67
TEMPERATURE_LIMITS .....	25	Example 2: A Custom Database for the Sb-Sn Binary System .....	69
DEFINE_SYSTEM_DEFAULT .....	26	Example 3: A Public Database for the Fe-Cr-C Ternary System .....	73
DEFAULT_COMMAND .....	27	<b>Database Checker .....</b>	<b>87</b>
DATABASE_INFORMATION .....	28	About the Database Checker .....	88
TYPE_DEFINITION .....	29	Custom Databases: GES5 and GES6 Calculation Engine Version .....	89
FTP_FILE .....	33	Using the Database Checker .....	90
FUNCTION .....	34	Example of Working with the Database Checker .....	90
PARAMETER .....	38	<b>TDB Editor .....</b>	<b>92</b>
GIBBS Parameter Name .....	39	About the TDB Editor .....	93
Parameters for the Two Sublattice Ionic Liquid Model ...	45	Installing VSCode and the TDB Editor .....	94
OPTIONS .....	48	System Requirements .....	94
TABLE .....	49	How to Install the TDB Editor Extension .....	94
ASSESSED_SYSTEMS .....	50	READ ME Section Included with the Extension .....	95
REFERENCE_FILE .....	54	Using the TDB Editor .....	97
LIST_OF_REFERENCE .....	56	How to Create a TDB File .....	97
ADD_REFERENCE .....	57	How to Open a TDB File .....	97
CASE and ENDCASE .....	59	Example of Working with the TDB Editor .....	97
VERSION_DATA .....	60		
<b>Working with the Mobility Databases .....</b>	<b>61</b>		
Extensions to Database Definition File Syntax .....	62		
PARAMETER .....	63		

Customize the Color .....	98	INCLUDE .....	19
<b>DATAPLOT User Guide .....</b>	<b>1</b>	LINETYPE .....	19
<b><i>Introduction to the DATAPLOT User</i></b>		Interactive Plotting .....	20
<b><i>Guide</i> .....</b>	<b>2</b>	Formatting DIGLIB Symbols in	
Using this Guide .....	3	LaTeX Documents .....	21
About the DATAPLOT Graphical		LaTeX Formatting Codes .....	23
Language .....	4	LTEXT vs LaTeX Commands for	
DATAPLOT Examples .....	4	Labels .....	23
Plotting Experimental Data Files .....	5	<b><i>PROLOGUE Commands</i> .....</b>	<b>24</b>
Graphical Mode .....	5	PROLOG .....	25
Console Mode .....	6	DIAGRAM_TYPE .....	26
<b><i>Using DATAPLOT Graphical</i></b>		TIC_TYPE .....	27
<b><i>Language</i> .....</b>	<b>7</b>	TITLE .....	28
DATAPLOT File Structure .....	8	XLENGTH .....	29
DATAPLOT Language Syntax .....	9	XSCALE .....	30
Coordinate Systems .....	10	XTEXT .....	31
Graphical Operation Codes .....	11	XTYPE .....	32
Tables or Blocks .....	13	YLENGTH .....	33
Drawing a Polygon .....	14	YSCALE .....	34
Drawing an Analytical Function .....	15	YTEXT .....	35
Painting of an Enclosed Area .....	16	YTYPE .....	36
Writing a Text .....	17	<b><i>DATASET Commands</i> .....</b>	<b>37</b>
Plotting a Symbol .....	18	ATTRIBUTE .....	38
Other Commands .....	19	BLOCK .....	39
General .....	19	BLOCKEND .....	40
ATTRIBUTE .....	19	CHARSIZE .....	41
CLIP .....	19	CLIP .....	42
COLOR .....	19		

COLOR .....	43	Macro Text .....	64
DATASET .....	45	Example 4 – Draw Curves Defined by Functions .....	67
DATAPPOINT .....	46	Output Example .....	67
DRAWLINE .....	47	Macro Text .....	67
FONT .....	48	Example 5 – Use Included Files for Predefined Symbols .....	69
Font Numbers .....	48	Output Example .....	69
FUNCTION .....	49	Macro Text .....	70
GLOBALSIZE .....	50	Example 6 – Triangular Diagrams for Ternary Systems .....	72
INCLUDE .....	51	Output Example .....	72
LINETYPE .....	52	Macro Text .....	72
PAINT .....	53	Example 7 – Color Codes .....	78
PCFUNCTION .....	54	Output Example .....	78
SYMBOLSIZE .....	55	Macro Text .....	78
STRING .....	56	Example 8 - DIGLIB Symbols .....	83
TEXT .....	58	Output Example .....	83
		Macro Text .....	83
<b>Examples of DATAPLOT Files and Outputs .....</b>	<b>59</b>	Example 9 - DIGLIB Fonts and Lines	91
Example 1 – Draw Lines and Symbols .....	60	Output Example .....	91
Output Example .....	60	Macro Text .....	91
Macro Text .....	60	<b>Pourbaix Diagrams .....</b>	<b>1</b>
Example 2 – Draw Polygons and Symbols .....	62	<b>Introduction to Pourbaix Diagrams ..</b>	<b>2</b>
Output Example .....	62	About Pourbaix Diagrams in Thermo-Calc .....	3
Macro Text .....	62	GES5 and GES6 Calculation Engine Version .....	3
Example 3 – Using Strings and Various Line Types .....	64		
Output Example .....	64		

References .....	4
Pourbaix Diagrams .....	5
Working with Pourbaix Diagrams ....	7
Key Concepts .....	9
Effective Interaction Ratio .....	9
Solubility .....	9
Basic Settings and Definitions .....	11
<b><i>Thermodynamic Data Requirements</i></b>	<b>13</b>
Required Thermodynamic Data .....	14
Including or Excluding the Gas Phase .....	15
<b><i>Examples of Pourbaix Diagrams for Fe</i></b> .....	<b>19</b>
Pourbaix Diagrams with Gas Phase Excluded .....	20
Pourbaix Diagrams with Gas Phase Included .....	21
<b><i>Other Examples of Pourbaix Diagrams</i></b> .....	<b>22</b>
Variations of Pourbaix Diagrams ....	23
Pourbaix Diagrams for Complex Alloys .....	26
<b><i>Pourbaix Diagram References</i></b> .....	<b>28</b>
<b>Thermo-Calc: General Reference</b> .	<b>1</b>
<b><i>Parameters, Functions, and Variables</i></b> .....	<b>2</b>
Specifying Parameters .....	3
About Operators and Functions .....	5
Available Operators for Writing	5

Functions .....	
Thermodynamic Variables .....	6
Common Thermodynamic Variables .....	6
How to Read the Tables .....	7
Intensive Variables .....	8
Extensive Variables .....	9
Special Quantities .....	13
The u-Fraction Variable .....	15
Suffixes .....	16
Normalizing Suffixes .....	16
Derived Variables and Partial Derivatives .....	23
Defining New Derived Variables and Functions .....	24
Derived Variables for Aqueous Solutions .....	25
Defining New Derived Variables for Aqueous Solutions .....	28
Thermodynamic Variables and Wildcard Characters .....	29
Units of State Variables and Derived Variables .....	30
User-Specified Units .....	31
Examples of User-Defined Units ....	32
For Intensive Variables of a Defined System .....	32
For Extensive Variables of a Defined System .....	33

For Intensive Variables of a System Component .....	34	Mode Examples .....	
For Extensive Variables of a System Component .....	34	References .....	67
For Intensive Variables of a Species in a Phase .....	36	Surface Tension Model for Oxide Slag .....	69
For Extensive Variables of a Species in a Phase .....	37	Viscosity of the Metallic Liquids Model .....	70
State Variables for Scheil .....	41	Viscosity of the Ionic Liquids Model .....	72
State Variables in Console Mode DICTRA .....	43	Molar Volume Model .....	73
Intensive Properties .....	43	<b>Database Files .....</b>	<b>74</b>
Extensive Properties .....	43	About the Database Files .....	75
<b>Thermophysical Properties Data and Variables .....</b>	<b>48</b>	Custom Databases: GES5 and GES6 Calculation Engine Version .....	76
About Thermophysical Properties Data Parameters .....	49	Database Installation Folder .....	77
Thermophysical Properties Variables .....	53	About Cached Database Files .....	78
Electrical Resistivity Model .....	55	Editing the Database Initialization File or Unencrypted Database File .....	79
Electrical Conductivity for Ionic Liquid Model .....	59	Database Checker .....	80
Thermal Conductivity Model .....	61	<b>Global Settings .....</b>	<b>81</b>
Surface Tension of Metallic Liquid Alloys Model .....	65	Global General Settings .....	82
Modified Guggenheim Model Description .....	65	Graphical Mode: Default Units .....	87
Redlich-Kister-Muggianu Model Description .....	66	Graphical Mode: Activities .....	89
Plot Variables (Both Models) .....	67	Global Settings: System Definer .....	90
Graphical Mode and Console .....	67	Global Settings: Equilibrium Calculator .....	92
		Global Settings: Diffusion Calculator .....	93
		Global Settings: Precipitation Calculator .....	94

---

Global Settings: Process Metallurgy	95
Global Settings: Scheil Calculator	96
Global Settings: Tables	97
Global Settings: Graphical and Console Mode Plotting	98
Global vs Local Plot Settings	99
Local Plot Property Settings	99
Global Plot Settings: Themes	101
About Plot Themes	101
Working with Plot Themes	102
How to Create a Custom Plot Theme	105
How to Import a Custom Plot Theme	105
How to Export a Plot Theme	106
How to Delete a Custom Plot Theme	107
How to Rename a Custom Plot Theme	107
Predefined Theme Settings	108
Global Plot Settings: Plot Area Size	111
Global Plot Settings: Design the Look and Feel	113
Global Settings: Console Mode Default Appearance	119

# **Installation Guide**

**Windows<sup>®</sup>, macOS<sup>®</sup>, and Linux Operating Systems**

**Thermo-Calc Version 2024a**

*Including the TC-Python and TC-Toolbox for MATLAB<sup>®</sup> Installation Guides*



---

# Installing Thermo-Calc

---

**In this section:**

---

Key to Using this Guide .....	3
Installation Terminology .....	5
About the Thermo-Calc Components Being Installed .....	7
System Requirements .....	9
Platforms Tested and SDK Availability .....	10

## Key to Using this Guide



This guide is available in PDF format: (1) On the [website](#) or (2) when in Thermo-Calc go to **Help** → **Manuals Folder** → **Installation**.

This installation guide is for all Thermo-Calc installations running on Windows, macOS, or Linux operating systems. It is also applicable to previously installed versions of Thermo-Calc (the new version is installed next to the old version).

This guide includes information about:

- The types of licenses and installations available.
- Installing the Thermo-Calc software and its components on standalone computers, servers, and client computers.
- Installing the license management software for network installations.
- Retrieving and installing license files.
- Details about the specific instructions needed to install two of the SDKs, *TC-Python* and *TC-Toolbox for MATLAB*®.
- Troubleshooting to solve common installation problems.
- Updating and uninstalling the software.

### Icons in this Guide

Name	Description
Information	 Provides important information and indicates that more detail is located in the linked or named topic.
Note	 The information can be of use to you. It is recommended that you read the text or follow the link.
Operating system	The operating system is indicated when the information is specific to a <i>Windows</i> , <i>macOS</i> ,

Name	Description
	or <i>Linux</i> operating system.
 License file	The Thermo-Calc license file, which contains the license keys and is installed on either a standalone computer or a network server.
	The Thermo-Calc software and its components (including databases and SDKs) to be installed.

## Typographical Conventions

Convention	Definition
Forward arrow →	The forward arrow symbol → instructs you to select a series of menu items in a specific order. For example, <b>File → Save Project</b> is equivalent to: From the <b>File</b> menu, select <b>Save Project</b> .
<b>Boldface font</b>	A <b>boldface</b> font indicates that the given word(s) are shown that way in on a toolbar button or as a menu selection. For example, if you are told to select a menu item in a particular order, such as <b>File → Save Project</b> , or to click <b>Save</b> .
<i>Italic font</i>	An <i>italic</i> font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or elsewhere in the guide.
<code>code</code> and <b>code bold</b>	A <code>code</code> font indicates you are to make a keyboard entry. It also shows a programming code or code examples. The <code>code bold</code> font highlights the entry.

## Installation Terminology

Term	Definition
client	A computer without a license file of its own but connected through the network to a computer with the license information. The client receives a license token from the computer where the license management software is installed.
consolidated network	A type of network installation where the Thermo-Calc software and a Thermo-Calc licensing software are installed on a single server computer. This allows all computers in the network to access both the software and a license on the server via a client installation.
distributed network	A type of network installation where the Thermo-Calc software is installed on one server computer in a network and the Thermo-Calc licensing software is installed on a different computer. After a thin client installation, all computers in the network can access both the software and the network license key via the server.
host ID	The MAC address on the computer that a license is locked to.
license file	The file (called <i>lserverc</i> ) that contains all the purchased license keys. It is issued by Thermo-Calc Software, usually by email.
license key	The information contained in the license file that represents the products purchased and which is then authorised for installation.
license management software (see Sentinel RMS License Software)	The Thermo-Calc license management software. When working in Windows it is called <i>License Manager</i> (full name is Sentinel RMS License Manager by SafeNet) and on Linux it is called <i>License Server</i> (full name is Sentinel License Sever by Safenet).
license server	The use of the term <i>license server</i> (lower case letters) is reserved for use for scripts and option files (for example, in an <a href="#">Unattended Network Client Installations</a> ). For the purposes of the step-by-step instructions it is not used to refer to the type of network installation that runs the license management program. See <i>Sentinel RMS Software</i> below.
Network License (NWL)	The license type used with the licensing software. The NWL can only be used on one specific computer that hosts the licensing software. This restricts license usage to a certain subnet or subnets.  The software may be used on the number of sites you have paid for and by the number of persons you have paid for on any number of network connected computers
Sentinel RMS License Software	The Thermo-Calc license management software that supplies the client computers in your network with license tokens when it runs the Thermo-Calc software. This is available for both Windows (called the License Manager) and Linux (called the License Server, but not to be confused with <i>license server</i> in lower case letters

---

<i>Term</i>	<i>Definition</i>
	reserved for scripts).
Single-User-Node Locked-License (SUNLL)	A unique license file for one designated computer at one designated site. It cannot be copied or moved outside the site. If the software is installed on a portable computer, it can be temporarily used off-site. The license can be used without network access (to the License Server).
standalone computer	A computer which uses a SUNLL license type.
TC-Toolbox	The short name for the TC-Toolbox for MATLAB® SDK.
thin client	An installation where the software is installed on a file server but is run from a client computer. The licensing software may be installed on the file server (consolidated network installation) or on a different computer (distributed network installation).
XQuartz	An open-source X Windows System, which is required to run Thermo-Calc on a Mac.

---

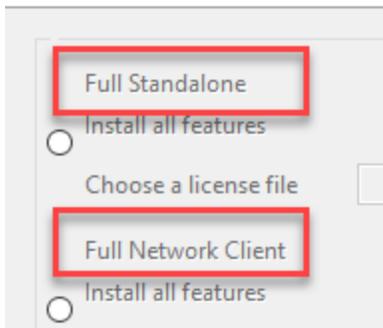
## About the Thermo-Calc Components Being Installed

---

### What is Installed with a Full Standalone or Full Network Client Option?

At the start of the installation process, the following components are installed when you choose either the **Full Standalone** or **Full Network Client** options. If you want to exclude a component (perhaps one of the SDKs), then you would use the **Custom** option.

#### Select Installation Mode



A full installation means that all listed feature components are installed but the availability of the individual components is based on the license you have. This is also applicable to the databases and software development kits (SDKs).

#### Thermo-Calc

The Thermo-Calc software includes several add-on features and capabilities that are packaged with the software and included with a full standalone or network client installation. For example, there are the specialized add-on products Diffusion Module (DICTRA), Precipitation Module (TC-PRISMA), and Process Metallurgy Module as well as the Steel Model Library. For most of these specialized features, demonstration (*demo*) versions and/or examples are available for all users but advanced features are only available with a license.

#### Databases

All the latest databases are installed. As with the software features, availability of a specific database in the software itself is based on license. However, several demonstration (*demo*) databases are included and available to all users.

---

## Software Development Kits (SDKs)

When the **Full Standalone** or **Full Network Client** option is chosen, it also installs the software development kits (SDKs) (e.g. TC-Python, TQ-Interface, TC-Toolbox for MATLAB®, etc.).

### TC-PYTHON

The TC-Python program is installed when the **Full Standalone** or **Full Network Client** option is chosen, but there are additional steps required to complete the installation as described in the *TC-Python Installation Guide*.

### TC-TOOLBOX FOR MATLAB®

If a MATLAB® installation on your computer is found during this installation process, the TC-Toolbox for MATLAB® (available only for Windows) is automatically installed, otherwise you may need to do a manual installation. For details, see the *TC-Toolbox for MATLAB Installation Guide*®.

## What Does the Custom Option Allow Me to Do During an Installation?

During the installation process, there is some flexibility to choose what to include or exclude with the installation by choosing the **Custom** option. This is primarily to the level of being able to choose the SDKs or to exclude all databases or Thermo-Calc itself. That is, you cannot choose specific Thermo-Calc features or databases as these are controlled by the license and not the installation procedure.

---

## System Requirements

---



[About the Thermo-Calc Components Being Installed](#)

### General

A full list of [system requirements](#) and program availability is found on the Thermo-Calc website. You can also see the [roadmap for platform support](#) to use for planning purposes.

### Network Client Installations: Sentinel RMS License Software



Network client installations using the Sentinel RMS license management software require the use of the most recent version, 9.7.

For details of the system requirements to install Sentinel RMS, see [Windows: About Installing the License Manager](#) and [Linux: About Installing the License Server](#).



For macOS, the Sentinel RMS License software is installed on a Windows or Linux computer server where the Thermo-Calc license file must also be located.

## Platforms Tested and SDK Availability

A full list of [system requirements](#) and program availability is found on the Thermo-Calc website. You can also see the [roadmap for platform support](#) to use for planning purposes.

### Platforms Tested

Thermo-Calc and its components are tested on these platforms for Thermo-Calc 2024a:

<i>Windows</i>	<i>macOS</i>	<i>Linux<sup>1, 2</sup></i>
<ul style="list-style-type: none"> <li>Windows 10 64-bit</li> <li>Windows 11</li> </ul>	<ul style="list-style-type: none"> <li>Sonoma 14.x</li> <li>Ventura 13.x</li> <li>Monterey 12.x</li> </ul>	<ul style="list-style-type: none"> <li>Ubuntu 22.04 (Jammy Jellyfish)</li> <li>Ubuntu 20.04 (Focal Fossa)</li> <li>CentOS 7.9</li> <li>OpenSUSE 15.4 and 15.5 (Leap)</li> </ul>
<p><sup>1</sup> Thermo-Calc Software cannot guarantee full functionality if the software is installed on other Linux distributions.</p> <p><sup>2</sup> If you are installing on CentOS 7, it is recommended you install gtk2 before you run the installation script.</p>		

### SDK Availability by Platform

The software development kits (SDKs) are available as follows for the indicated platforms:

<i>SDK</i>	<i>Windows</i>	<i>macOS</i>	<i>Linux</i>
TC-Python	x	x	x
TQ-Interface	x	not available	x
TC-Toolbox for MATLAB®	x	not available	not available
TC-API	x	not available	x

---

# License and Installation Types

---

**In this section:**

---

License and Installation Types .....	12
About License Keys and Files .....	13
Standalone Installations (SUNLL) .....	14
Multi-User Network Installations (NWL) .....	15

## License and Installation Types

Installation procedures are based on the license type, the installation type, and the operating system.



Visit our website for more information about [Licensing Options](#).



For macOS, the Sentinel RMS License software is installed on a Windows or Linux computer server where the Thermo-Calc license file must also be located.

<i>License type</i>	<i>Installation Type</i>	<i>Windows</i>	<i>macOS</i>	<i>Linux</i>
SUNLL	Standalone	Yes	Yes	Yes
NWL	Consolidated network	Yes	No	Yes
	Distributed network	Yes	No	Yes
	License installations on a server network	Yes	Yes	Yes

### Sentinel RMS License Management Software: Use Current Version

The most recent version (9.7) of the Sentinel RMS License Manager (Windows) and License Server (Linux) is included with the network license Thermo-Calc software download.

However, due to security issues reported by the vendor, it is important that previous network license installations of the Sentinel RMS are upgraded to this version number.

For more information for your operating system, see [Windows: About Installing the License Manager](#) and [Linux: About Installing the License Server](#)

---

## About License Keys and Files

---

To run Thermo-Calc and its components you need a valid license key for each product. The license keys are included in the license file emailed to you by Thermo-Calc Software.

You can copy the license file to the applicable computer(s) before or after installing the software—the license file itself is not required during the installation process. However, it is required to make calculations in Thermo-Calc.



[Request a License File from Thermo-Calc Software](#)

## Standalone Installations (SUNLL)

---

The standalone installation is used with the Single-User Node-Locked License (SUNLL). The Thermo-Calc software  and a Thermo-Calc license file  are installed on a standalone computer. You can do a full (all operating systems) or a custom installation (Windows and Linux only).



 To get started, go to [First Steps of Installation](#) and then [Standalone Installations](#).

## Multi-User Network Installations (NWL)

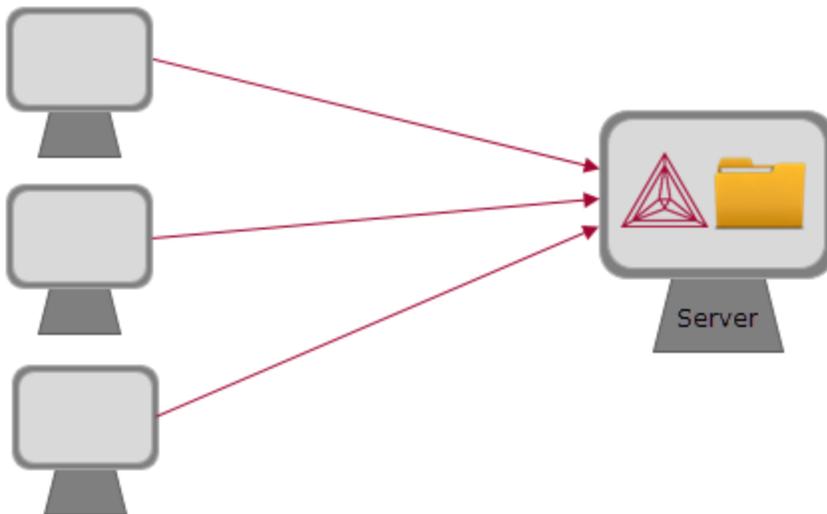
There are different types of multi-user network installations: consolidated, distributed, and where the licensing software is installed on a computer network. There is also a subtype of installation called thin client, which is a step done with the consolidated and distributed network installations.



**Thin client installations:** A thin client installation is where the Thermo-Calc software is installed on a computer server but is run from a client computer. The licensing software can be installed on the computer server (a consolidated network) or on a different computer (a distributed network). This type of installation is available for Windows and Linux operating systems. The client computer must have the Thermo-Calc software installed. This type of installation mainly sets environmental variables and paths on the client computer.

### Consolidated Network Installations

The consolidated network installation is used with a Network License (NWL) and it is available for Windows and Linux operating systems.

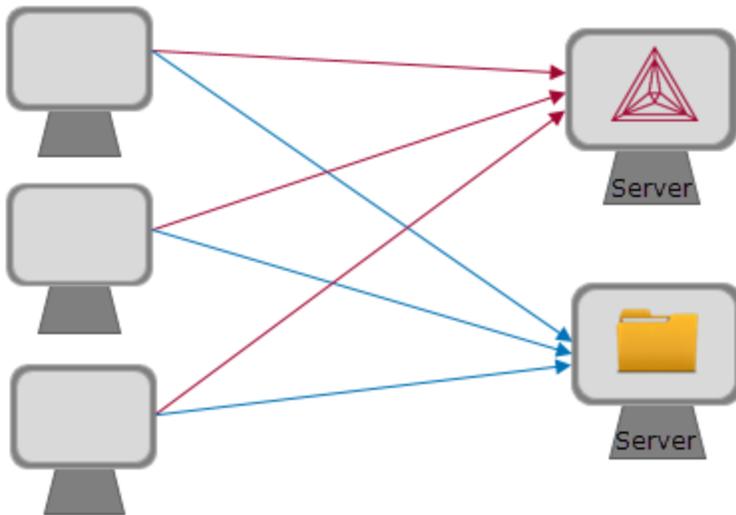


The Thermo-Calc software  and the license management program are installed on the same server where the license file  is also copied. A thin client installation allows all computers (the clients) in the server network to access both the software and the licensing on the same computer server.

 To get started, go to [First Steps of Installation](#) and then [Network Client Installations](#).

## Distributed Network Installations

The distributed network installation is used with a Network License (NWL) and is available for Windows and Linux operating systems.

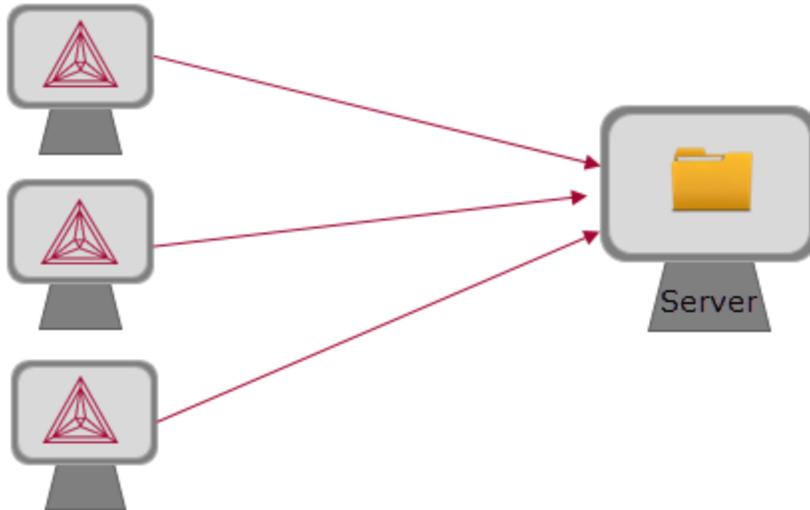


The Thermo-Calc software  is installed on one computer server in a network and the licensing software is installed on a different computer server where the license file  is also copied. A thin client installation allows all computers (the clients) in the server network to access both the software and the licensing on the two computer servers.

 To get started, go to [First Steps of Installation](#) and then [Network Client Installations](#).

## License Installations on a Server Network

When a license file is installed on a server network, it is used with a Network License (NWL). It is available for all platforms. However, for macOS the server containing the licensing software must be set up on Windows or Linux.



The licensing software is installed on a computer server where the license file  is also copied. The Thermo-Calc software  is installed on every computer in the network that needs it.



To get started, go to [First Steps of Installation](#) and then [Network Client Installations](#).

---

# Preparing for Installation

---

**In this section:**

---

First Steps of Installation .....	19
Installation Default Directories and the License File .....	20
Installation Default Directory: Windows .....	21
Installation Default Directory: macOS .....	22
Installation Default Directory: Linux .....	23
Installation Default Directory: Databases, Database Checker, and TDB Editor .....	24
Installation Default Directory: Materials .....	26
Installation Default Directory: Process Metallurgy Materials .....	28
Installation Default Directory: Custom Property Models .....	29
Subnet Information for Network Licenses .....	30
macOS Installation Notes .....	31
Linux Installation Notes .....	32

## First Steps of Installation

---

This section reviews some of the things you need to do before installing the Thermo-Calc software.

- Learn about the different default directories where the software and license files are installed.
- For network installations, collect subnet information.
- Make sure the person installing the software has administrator rights for Windows and macOS.
- For macOS, install XQuartz and make sure the security preferences are set correctly.
- For a Linux Cent OS 7 installation, also install gtk2 before you run the installation script.

```
yum install gtk2
```



**Should I get my license file before or after software installation?** The Thermo-Calc license file itself is not required during the installation process. However, it is required to make calculations in Thermo-Calc. The license file has details about your computer(s) that in some cases cannot be determined accurately until *after* installation (especially for a network installation). For most users it is recommended you request the file before you start installing.

---

## Installation Default Directories and the License File

---

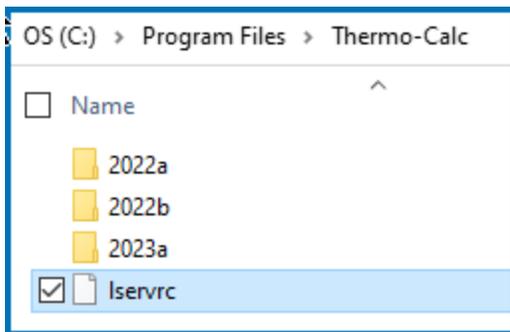
The default directories where the software and its components are installed, including where the license file is stored, vary by platform and user type.

If you want to select specific components to install or if you want to install the software in a non-default installation directory, then do a **custom** installation (Windows and Linux only). In the table, *<user>* stands for the user name and *<version>* for the version of Thermo-Calc, for example 2024a.

### License File Installation Location for All Operating Systems

The Thermo-Calc license file is installed outside of the installation program folder. If you have more than one version of Thermo-Calc installed then there are additional folders by version number with only one license file.

For macOS the license file is put in `/Users/Shared/Thermo-Calc`. The example below is for a Standalone Windows installation.



See the following for more information:

- [Installation Default Directory: Windows](#)
- [Installation Default Directory: macOS](#)
- [Installation Default Directory: Linux](#)
- [Installation Default Directory: Databases, Database Checker, and TDB Editor](#)
- [Installation Default Directory: Materials](#)
- [Installation Default Directory: Process Metallurgy Materials](#)
- [Installation Default Directory: Custom Property Models](#)
- [Installation Default Directory: TC-Python](#)
- [Installation Default Directory: TC-Toolbox and the Examples](#)

## Installation Default Directory: Windows



Administrative rights are required to install and uninstall Thermo-Calc.

Windows User Type	Default Directory
Administrator	<p>The Thermo-Calc program is installed here:</p> <pre>Program Files\Thermo-Calc\<i>&lt;version&gt;</i></pre> <p>The default folders where documents, materials files, examples, and other folders that sometimes require additional licenses are installed under <b>Public Documents</b> for ALL USERS.</p> <p>For a local user these files are copied to the user's <b>Documents</b> folder where this local copy is associated to the user login. The <b>Public Documents</b> folder always contains the original set of contents as per the installation.</p> <pre>C:\Users\Public\Public Documents\Thermo-Calc\<i>&lt;version&gt;</i></pre> <p>After installing Thermo-Calc, and when you first open the program, the set of folders is copied from the source Public Documents location into a user Documents folder:</p> <pre>C:\Users\<i>&lt;user&gt;</i>\Documents\Thermo-Calc\<i>&lt;version&gt;</i></pre> <div data-bbox="461 989 1386 1125" style="border: 1px solid red; padding: 5px;"> <p> It is recommended that you work with the files that are copied to the <i>user</i> folder.</p> </div> <p> In some cases when an interim software update is installed there is also a backup folder created in this folder. For details, see <a href="#">Updating the Software for Interim Releases</a>.</p>
Server network	<p>License software installations on a server network (the License Manager software)</p> <pre>C:\Program Files (x86)\Common Files\SafeNet Sentinel\Sentinel RMS License Manager\WinNT\</pre>

## Installation Default Directory: macOS

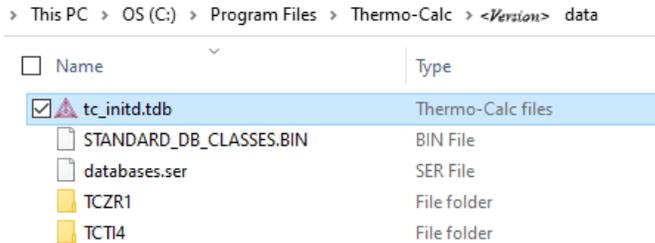
<i>macOS User Type</i>	<i>Default Directory</i>
Administrator (user name and password required)	<p>The Thermo-Calc program is installed in the <b>Applications</b> folder:</p> <p><code>Thermo-Calc-&lt;version&gt;.app</code> and <code>uninstall-Thermo-Calc-&lt;version&gt;.app</code></p> <p>The documents, materials, examples, and other folders that sometimes require additional licenses are installed here:</p> <p><code>/Users/Shared/Thermo-Calc/&lt;version&gt;</code></p> <p> In some cases when an interim software update is installed there is also a backup folder created in this folder. For details, see <a href="#">Updating the Software for Interim Releases</a>.</p> <p>To go to this folder, in Finder, from the <b>Go</b> main menu select <b>Go to folder</b>. Enter the above file path and click <b>Go</b>.</p>

## Installation Default Directory: Linux

Linux User Type	Default Directory
Non root user	<p>The Thermo-Calc program including the documents, materials, examples, and other folders that sometimes require additional licenses are installed here:</p> <pre data-bbox="662 457 1029 478">/home/&lt;user&gt;/Thermo-Calc/&lt;version&gt;</pre>
Root user	<p>The Thermo-Calc program including the documents, materials, examples, and other folders that sometimes require additional licenses are installed here:</p> <pre data-bbox="662 594 943 615">/opt/Thermo-Calc/&lt;version&gt;</pre> <p data-bbox="643 663 683 705"> In some cases when an interim software update is installed there is also a backup folder created in this folder. For details, see <a href="#">Updating the Software for Interim Releases</a>.</p>
License software installations on a server network (the License Server software)	<p>If you are doing a network installation then these files are available as a tar-file from the Thermo-Calc website. Contact <a href="#">Support</a> for more information.</p> <p>In the downloaded tar-file, there is a <b>restart_lserv</b> script file, which can be used as a template for creating a script that starts up the License Server.</p>

## Installation Default Directory: Databases, Database Checker, and TDB Editor

The standard Thermo-Calc databases and the database initialization file `tc_initd.tdb` are located in a subfolder to the Thermo-Calc installation. For example, in a Windows Standalone installation, you can find these in a folder called **data** where the file path is `C:\Program Files\Thermo-Calc\<version>\data`.



### Database Checker

The Thermo-Calc software package includes a program to check that the syntax of unencrypted database files is correct. The program applies the syntax rules set out in the *Database Manager User Guide* and reports errors and issues warnings. This program is intended for advanced users who develop and manage databases.



Thermo-Calc accepts some deviations from these syntax rules. Although warnings can be ignored, errors must be corrected. Reported errors must be corrected in the TDB file before the database can be loaded by the application. Sometimes the warnings can also reveal the need for additional corrections as shown in the example below.

For example, a warning is generated if a phase name is detected that is also an abbreviation of another phase name in the database. Although allowed, it is not recommended as this can (in some cases) lead to unexpected results because abbreviations are also used when entering commands in the Console and included in macro files. The following shows why in this case an abbreviation is not recommended.

Imagine that a macro file uses the abbreviation `ABC` to refer to a phase in a database that is named `ABCDE`. As long as there is no other phase in the database with the same exact name or initial abbreviation, it works. But if a new phase is added to the database and is actually named `ABC` then the macro can in error refer to the wrong phase. The macro still runs but

now refers to the phase `ABC` instead of `ABCDE` because `ABC` is an exact match, *which takes precedence over abbreviations*. The Database Checker displays a warning because this situation can only occur when one phase name is an abbreviation of another.

With respect to using this tool, the executable `DatabaseChecker` file is found in the Thermo-Calc home directory.

The program can also be launched by selecting **Tools** →  **Database Checker** from the main menu in Thermo-Calc.

## TDB Editor

The TDB Editor is an editing tool for those who develop their own custom databases for use with Thermo-Calc or other applications. The tool speeds up the database editing process by providing immediate feedback through syntax coloring, syntax checking, and by allowing formatting and easy navigation to items of interest in the TDB file.

During installation, the `tdb-language-support.vsix` file is installed in the Thermo-Calc home directory.



In order to access the most-up-to-date version of the VSCode plugin, it must be updated every time a *new* version of Thermo-Calc is installed.



For information about installing the TDB Editor, which is separate from the main installation and can be done later, see the *Database Manager's User Guide* contained in the *Thermo-Calc Documentation Set* or press F1 in Thermo-Calc and search the help.

## Installation Default Directory: Materials

When you are working with the System Definer or Material to Material Calculator you can open the default **Materials** folder included with your installation.

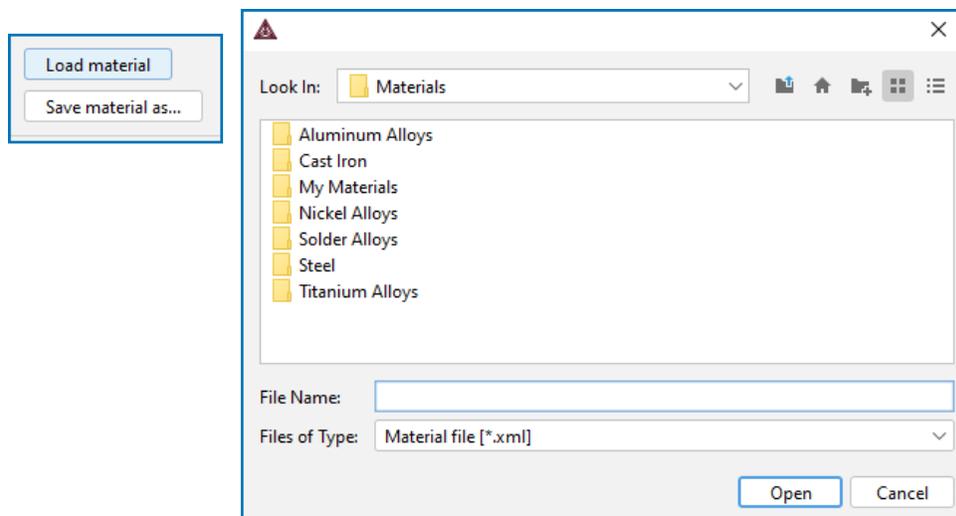
At the bottom of the window under **Material**, click **Load material** to navigate to a material file.



There is a separate folder for materials saved using the Process Metallurgy Calculator. See [Installation Default Directory: Process Metallurgy Materials](#).



For the Additive Manufacturing (AM) Module, the Material Properties library is managed by the software.



The default directory where these material files are stored is based on operating system:

<i>Operating System</i>	<i>Materials Installation Directory</i>
Windows	C:\Users\ <user&gt;\documents\thermo-calc\2024a\materials< td=""> </user&gt;\documents\thermo-calc\2024a\materials<>
Linux	On Linux all user files are installed where the program is installed. If installed as root the default is:  /opt/Thermo-Calc/2024a/Materials

<b>Operating System</b>	<b>Materials Installation Directory</b>
macOS	/Users/Shared/Thermo-Calc/2024a/Materials

## Installation Default Directory: Process Metallurgy Materials

---

With the Process Metallurgy Calculator you can save materials to work with again. This is the default folder where these are stored based on your operating system:

<i>Operating System</i>	<i>Process Metallurgy Materials Installation Directory</i>
Windows	<code>C:\Users\<user&gt;\documents\thermo-calc\2024a\processmetallurgymaterials< code=""></user&gt;\documents\thermo-calc\2024a\processmetallurgymaterials<></code>
Linux	On Linux all user files are installed where the program is installed. If installed as root the default is: <code>/opt/Thermo-Calc/2024a/ProcessMetallurgyMaterials</code>
macOS	<code>/Users/Shared/Thermo-Calc/2024a/ProcessMetallurgyMaterials</code>

## Installation Default Directory: Custom Property Models



All Property Models (whether these are predefined or custom models) must be located in a subdirectory of the Property Model directory. You can go to the **Options** window then on the **General** tab the **Model directory** field displays the default directory.

To open the **Options** window:

- Windows: Select **Tools** → **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press on the keyboard.

Predefined models included with the Property Model Calculator are installed in the locations below based on your operating system. If you create your own models using TC-Python, this is the default directory where these are saved.

<i>Operating System</i>	<i>Property Model Installation Directory</i>
Windows	<code>C:\Program Files\Thermo-Calc\2024a\PropertyModels</code>
Linux	<p>On Linux all user files are installed where the program is installed. If installed as root the default is:</p> <p><code>/opt/Thermo-Calc/2024a/PropertyModels</code></p> <p>or:</p> <p><code>/home/&lt;username&gt;/Thermo-Calc/2024a/PropertyModels</code></p>
macOS	<code>/Applications/Thermo-Calc-2024a.app/Contents/Resources/PropertyModels</code>

---

## Subnet Information for Network Licenses

---

This is related to [Full Network Client Installations](#).

For network licenses, the subnet information is also required. Ask your network administrator for the *network address for the subnet*. Additional requirements are included elsewhere for network installations.

- Subnets should be in the format `aaa.bbb.ccc***` (for example: `192.168.29.0/24` or `192.168.29.*`)
- When installing on Sentinel RMS License Server version 9.7, a maximum of seven (7) subnets are permitted.

An alternative way to get this information is to run the command:

- *Windows:* `ipconfig /all`
- *macOS/Linux:* `ip a`

## macOS Installation Notes

---

### ***How to Restart After an Installation***

In some types of installations (e.g. a network installation), you may have to restart your Mac to use Thermo-Calc for the first time.

### ***How to Install XQuartz***

The XQuartz open-source X Windows system is required to run Thermo-Calc on macOS but it is not required to install Thermo-Calc. Before installing Thermo-Calc, you may also need to ensure that the computer security permissions are set.

1. In a web browser go to <https://www.macupdate.com> or to download it directly from XQuartz, <https://www.xquartz.org/>.
2. In the **Search Mac Apps** field, enter *XQuartz*.
3. Click the **XQuartz** search result. For example, based on the recent version, the name is XQuartz-2.7.11.dmg.
4. Click **Download**.
5. Double-click the **XQuartz.pkg** file. Follow the instructions.
6. Click **OK** when the message about logging out displays and click **Close** when the installation is successful.

### ***How to Enable the Security Settings***

1. From the Apple main menu, select **System Preferences**.
2. Click **Security and Privacy**.
3. Click **General**. At the bottom of the **Security & Privacy** window, click the lock icon to make changes. Enter the administrator password as required.
4. If you have already tried to launch the Thermo-Calc installer, you can click **Open Anyway** to proceed. Otherwise, under **Allow apps downloaded from:** click **Anywhere** and then **Allow from Anywhere**.
5. Continue with the applicable installation type (standalone or network).



Remember to reset your security after installing Thermo-Calc.

---

## Linux Installation Notes

---

The following are details you need to know regarding your Linux installation as well as how to open Thermo-Calc.

### openSUSE Administration (sudo) Users

Note the following if you are a sudo user (administrator) in openSUSE installing Thermo-Calc using this command:

```
sudo ./Thermo-Calc-linux-<full-version-number>.run
```

When this command is used, you will do a text-based installation instead of having a graphical interface to guide you. The information is the same, just without the user interface.



For example, the *<full-version-number>* is not just 2024a, it is what is found in the **Help** → **About** window in Thermo-Calc and it changes every release. You can find the [current full version number listed on our website](#).

### Opening Thermo-Calc in Linux

To open Thermo-Calc double-click the **Thermo-Calc-2024a** application or from the Thermo-Calc installation directory enter `$ ./Thermo-Calc-2024a.sh` in the terminal.

---

## Standalone Installations

---

You can either do a full or a custom standalone installation. The license type is called a Single-User Node-Locked License (SUNLL).

**In this section:**

---

Full Standalone Installations .....	34
Custom Standalone Installations .....	38
Add New Components to an Existing Installation .....	41

## Full Standalone Installations

---

The following describes a full standalone installation. These steps assume you have completed the section [First Steps of Installation](#).

### Request your License File from Thermo-Calc Software



**Updates to Thermo-Calc:** Customers who subscribe to the annual maintenance plan can skip this step if you have received your license file in an email from Thermo-Calc Software AB. Save the license file to your computer to use during software installation. The license file name must be the same as the existing file name to ensure there are no errors during the update.

Follow the instructions for your operating system in the section [Request a License File from Thermo-Calc Software](#).

### Download your Thermo-Calc Installation Software

- *Web downloads:* For web links sent in an email from Thermo-Calc Software, go to the URL and log in with the information provided to download the software.
- *DVD installations:* Insert the DVD and follow the instructions.

### Run the Applicable Installation File for your Operating System

1. Navigate to the folder for your operating system and click the link to download the software package, then go to the folder where you downloaded it.



Administrative rights are required for Windows and macOS installations.

#### **Windows**

Double-click the **.exe** file to launch the **Setup** program.

#### **macOS**

Double-click the downloaded file then follow the instructions.



For macOS and when you first run the installer, you may see a message that *Thermo-Calc-osx-<version> can't be opened because Apple cannot check it for malicious software*. If this happens, select **Show in Finder**. Then, in the opened **Finder** window right-click the downloaded file `Thermo-Calc-osx-<version>` and select **Open**.

## Linux

To start the Setup Wizard in a Terminal window, enter these commands in the directory where the installation file is saved:

```
> chmod +x linux-installation-x64-2024a.run
> ./linux-installation-x64-2024a.run
```



If you are an openSUSE root user, see [Linux Installation Notes](#) for some information about your installation.

2. On the **Setup** window click **Next**.
3. On the **License Agreement** window click the button to accept the license agreement then click **Next**.

## Select the Installation Type and Upload the License File



### [About the Thermo-Calc Components Being Installed](#)

1. In the **Select Installation Mode** window click **Full Standalone: Install all features**.



If you want to exclude a component (e.g. an SDK), then use the **Custom** option. See [Custom Standalone Installations](#). For TC-Toolbox for MATLAB®, also see [Installing TC-Toolbox with the Custom Option](#).

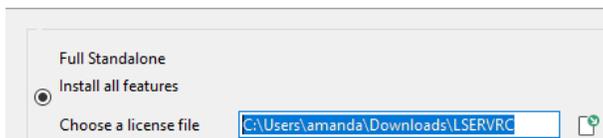


TC-Python is installed when the **Full Standalone** or **Full Network Client** option is chosen, but there are additional steps required as described separately for this SDK.

- For a **Full Standalone** installation, in the **Choose a license file** field either enter the file path or click the file button  to navigate to where you saved the license file emailed to you from Thermo-Calc Software. The license file is called **lservrc**.

 Setup

Select Installation Mode



To prevent errors, leave this field blank if you did not get a license file yet.

TC-Python is installed for all users even without a license. The **Install TC-Python** window is a reminder that additional installation procedures are required after Thermo-Calc is installed.

## Install the Software and Start Using Thermo-Calc

- In the **Ready to Install** window click **Next**.
  -  See the separate instructions for TC-Toolbox for MATLAB®.
- Click **Finish** and start using Thermo-Calc. For Linux users, log out and log back in to finish the installation.



macOS installations and if you have previous Thermo-Calc installations: When you launch Thermo-Calc with the new or updated version, the splash screen hides the **Import User Settings** dialog box. In order to proceed you need to click the splash screen and then click **Yes** or **No** in the dialog box.



If you did not get your license file see [Request a License File from Thermo-Calc Software](#).

---

## Custom Standalone Installations

---

### [About the Thermo-Calc Components Being Installed](#)

Use the **Custom** option to have more control over what component features to include or exclude, for example, to exclude an SDK.

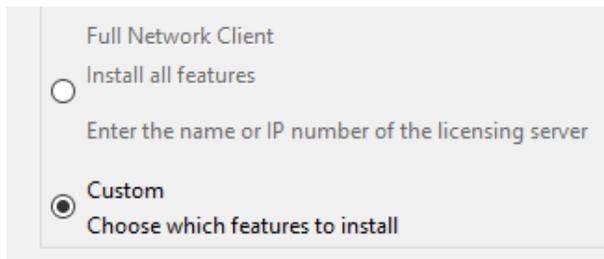
This topic describes how to do a custom installation on a standalone computer for either a Windows or Linux operating system. These steps assume you have completed these sections:

- [First Steps of Installation](#)
- Steps 1 to 3 of a [Full Standalone Installations](#) where you request your license file, download the software and then have started to set up the program up to the **Select Installation Mode** window.

## Select the Installation Mode and the Components to Include or Exclude

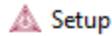
### [About the Thermo-Calc Components Being Installed](#)

1. In the **Select installation mode** window click **Custom** (the option appears greyed out, but it is available to select). Click **Next**.

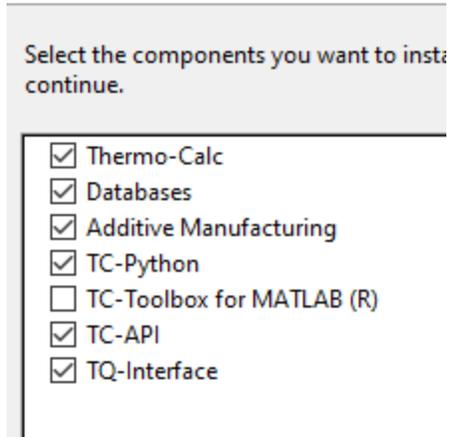


2. On the **Specify Configuration** window the **Create Shortcut on Desktop** checkbox is selected by default. Click **Next**.

3. In the **Select Components** window, click to select or deselect the checkboxes as needed. Click **Next**.



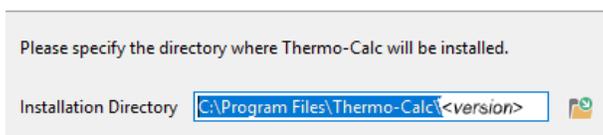
### Select Components



## Set the Installation Directory and Choose the License File

1. In the **Installation Directory** window, either accept the default path or enter a path to where you want to install the software (see [Installation Default Directories and the License File](#)). Click the file button to **Browse for Folder** and navigate to another directory.

### Installation Directory



To prevent errors, leave this field blank if you do not have a license file yet.

2. In the **Choose Setup Type** window, click **Standalone**.

**Choose Setup Type**

Select one of the following setup types.  
See installation guide for detailed description of the choices.

Standalone  
 Choose a license file  

Network Client  
 Enter name or IP number of the licensing server

3. In the **Choose a license file** field enter the path to your license file (called **lserverc**) or click the file button  to navigate to a location on the computer and open the file.
4. TC-Python is installed for all users even if you do not have a license. The **Install TC-Python** window is a reminder that additional installation procedures are required after Thermo-Calc is installed. See the *TC-Python Installation Guide* for details.

## Install the Software and Start Using Thermo-Calc

1. In the **Ready to Install** window click **Next**.
  -  See the separate instructions for TC-Toolbox for MATLAB®.
2. Click **Finish** and start using Thermo-Calc. For Linux users, log out and log back in to finish the installation.



macOS installations and if you have previous Thermo-Calc installations: When you launch Thermo-Calc with the new or updated version, the splash screen hides the **Import User Settings** dialog box. In order to proceed you need to click the splash screen and then click **Yes** or **No** in the dialog box.



If you did not get your license file see [Request a License File from Thermo-Calc Software](#).

---

## Add New Components to an Existing Installation

---

If you want to add new components to an existing Thermo-Calc installation (for example, you purchased a license for TC-Toolbox for MATLAB®), run the installation file again as in Step 3 of [Full Standalone Installations](#).

1. For all OS you need to have administrative level rights. For example in Windows, when you double-click the executable Thermo-Calc installation **.exe** file to launch the **Setup** program.



Linux OS: If Thermo-Calc was not installed by root the first time, then administrative rights are not required.

2. Click **Next**. Click to accept the license agreement.
3. Select the additional component(s) to install.
4. Click **Next** to start the installation, which adds the new component(s).



Use the **Uninstall** program to *remove* components. See [Uninstalling Thermo-Calc](#) or [Uninstalling TC-Toolbox for MATLAB®](#).

---

## Network Client Installations

---

This installation type is available for all operating systems, however for macOS the License Server must be run on a Windows or Linux computer. It is applicable to the consolidated and distributed network types of installations as well as for the licensing installation on a server network.



### [About the Thermo-Calc Components Being Installed](#)

#### **In this section:**

---

Full Network Client Installations .....	43
Windows: About Installing the License Manager .....	45
Linux: About Installing the License Server .....	50
Thin Client Installations .....	55
Custom Network Client Installations .....	58
Unattended Network Client Installations .....	61

---

## Full Network Client Installations

---

The following outlines how to do a full network client installation.

### Install the Sentinel RMS Management License Software

Follow the instructions for your operating system:

#### Windows

- [Windows: About Installing the License Manager](#)
- [Windows: Installing the Sentinel RMS License Manager](#)

#### Linux

- [Linux: About Installing the License Server](#)
- [Linux: Installing the Sentinel RMS License Server](#)

### Complete Steps 1 to 3 of a Full Standalone Installation

These steps assume you have completed the instructions for [Full Standalone Installations](#) to download and run the software up to the **Select Installation Mode** window. It is up to you whether you get the computer information for the license files before or after installing the software.

### Choose the Installation Type and Enter the Computer Server Details



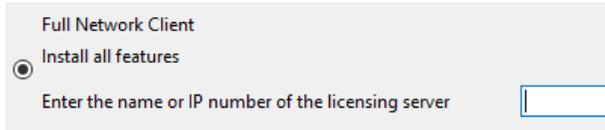
#### [About the Thermo-Calc Components Being Installed](#)

1. In the **Select Installation Mode** window click **Full Network Client** to **Install all features** (the option appears greyed out, but it is available to select).



If you want to exclude a component (perhaps one of the SDKs), then use the **Custom** option and see [Custom Network Client Installations](#).

2. In the field, enter the name or IP number of the computer where the licensing software is installed. Click **Next**.



Full Network Client

Install all features

Enter the name or IP number of the licensing server

3. TC-Python is installed for all users even if you do not have a license. The **Install TC-Python** window is a reminder that additional installation procedures are required after Thermo-Calc is installed. See the *TC-Python Installation Guide* for details.
4. In the **Ready to Install** window click **Next**.
5. When the **Completing the Thermo-Calc Setup Wizard** window opens, click **Finish**. For *Linux* users, log out and log back in to start using Thermo-Calc.



macOS installations and if you have previous Thermo-Calc installations: When you launch Thermo-Calc with the new or updated version, the splash screen hides the **Import User Settings** dialog box. In order to proceed you need to click the splash screen and then click **Yes** or **No** in the dialog box.



If you did not get your Thermo-Calc license file see [Request a License File from Thermo-Calc Software](#).

## Windows: About Installing the License Manager



If you have previously installed an older version of Sentinel RMS License Manager, you must first uninstall the old version then **reboot** the server before starting with a new installation.

The Sentinel RMS License Manager service supplies the client computers in your network with license tokens when it runs Thermo-Calc software. The Sentinel RMS License Manager software is installed on a computer that acts as a license server in your network. This is related to [Full Network Client Installations](#) and administrative rights are required for all steps.



Remember to collect subnet information before beginning. See [Subnet Information for Network Licenses](#).



A network license can only be used on one specific computer that hosts the License Manager. The network license determines which client computers receive license tokens using the IP addresses and IP ranges.

### Supported Windows Versions

These Windows operating systems are supported for the installation of Sentinel RMS License Manager. When ready, go to [Windows: Installing the Sentinel RMS License Manager](#).



For macOS, the Sentinel RMS License software is installed on a Windows or Linux computer server where the Thermo-Calc license file must also be located.

## Windows 32-bit Operating Systems

- Windows 7
- Windows 8.1
- Windows 10 v1909
- Windows Server 2008

## Windows 64-bit Operating Systems

- Windows 7
- Windows 8.1
- Windows 10 v1909
- Windows Server 2008
- Windows Server 2008 R2
- Windows Server 2012
- Windows Server 2012 R2
- Windows Server 2016
- Windows Server 2019

## Windows: Installing the Sentinel RMS License Manager



First review [Windows: About Installing the License Manager](#). Then if this is a new installation of the Sentinel RMS License Manager, start with Step 2.

### *Uninstall Any Older Versions of Sentinel RMS License Manager*

1. With Administrator rights, log onto the server that hosts the Sentinel RMS License Manager.
2. Save a copy of the license file LSERVRC found in the default directory—the license file can be saved as a text file: `C:\Program Files (x86)\Common Files\SafeNetSentinel\Sentinel RMS License Manager\WinNT\.`



After saving the existing license file and uninstalling the old version of Sentinel RMS, ensure you delete the old license file (LSERVRC) in the above folder. This license is not required anymore.

3. Reboot the computer hosting the Sentinel RMS License Manager.
4. Continue with the installation and use the preselected default settings.

### *Download and Install Thermo-Calc*

This step is not required to complete the server installation, however some administrators may want to do it at this point.

1. Download Thermo-Calc.
  - *Web downloads:* For web links sent in an email from Thermo-Calc Software, go to the URL and log in with the information provided to download the software.
  - *DVD installations:* Insert the DVD and follow the instructions.
2. Install Thermo-Calc. For testing purposes, and in order to confirm the installation is working correctly, install the Thermo-Calc software on the same computer as the Sentinel RMS License Manager. If required after testing, Thermo-Calc can always be uninstalled.

### *Install the Sentinel RMS License Manager Software*

1. From the download, extract the files `license_server_win_9.7.zip` and `LS_tools_win.zip`.
2. Right-click `setup.exe` and select **Run as Administrator** from the menu.

### *Choose the Installation Directory*

1. Click to accept the license agreement.
2. Enter your **User Name** and **Organization** and select to install the License Manager for **All users**.
3. In the **Destination Folder** window click **Next** to accept the suggested destination directory or **Change** to select another directory.

## Choose the Setup Type and Install the Software

1. In the **Setup Type** window, select **Complete**. Click **Next**.
2. Select the **Unblock the system firewall for the Sentinel RMS License Manager** checkbox.



Selecting the checkbox is required to run Thermo-Calc.

3. Click **Next** and then **Install**.
4. Click **Finish** to exit the InstallShield Wizard.

## Install the Thermo-Calc Network License Using WIAdmin



Only the WIAdmin tool can be used to update the network license used by the License Manager.

1. Save the source file containing the license code. The source file can be kept as a .txt file
2. Go to the previously saved file `LS_tools_win.zip`.
3. Right-click the file `WlmAdmin.exe` and select **Run as Administrator** from the menu.
4. If the license server is not found under subnet servers, go to **Edit → Defined Server List**.
5. Enter the host name or IP address of the license server.
6. Click **Add** and then **OK**.
7. Click **View → Refresh All** to update the list.
8. In the left main window right-click the name of a License Manager.
9. From the menu select **Add Feature → From a File → To Server and its File**.
10. To add the license from a file, select **From a File**. This permanently adds the license code to the License Manager license file as well as to the License Manager memory.
11. In the windows that open, browse to the source file that contains the license code and click **Open**.

## *Install Additional Tool Options*



[Sentinel RMS License 9.7 Tools](#)

## Linux: About Installing the License Server

---

The Sentinel RMS License Server service supplies the client computers in your network with license tokens when it runs Thermo-Calc software. The Sentinel RMS License Server software is installed on a computer that acts as a license server in your network. This is related to [Full Network Client Installations](#).

### Preparing for Installation

Read and be aware of the following before you begin the section [Linux: Installing the Sentinel RMS License Server](#).



Remember to collect subnet information before beginning. See [Subnet Information for Network Licenses](#).

### *Uninstall Older Versions of the License Server*

- Older versions of the License Server must be stopped by with `lsrvd` before running the new installation. This is Step 1 in [Linux: Installing the Sentinel RMS License Server](#).
- If you have a license server: Upgrade your license server if the new one is a higher version than the present one. Clients work with newer license servers (managers) but not necessarily with ones older than the client.

### *Requires Root Access or sudoers List*

Root access to your computer is required or you need to be on the `sudoers` list.

### *Linux 32-bit Installations on a 64-bit OS*



To run a license server 32-bit on a Linux 64-bit OS, 32-bit library support is needed.

## Thermo-Calc License File



The name of the **license file** must be in lower case letters (**lserverc**) because Linux commands and operations are case-sensitive.

- You need a valid **lserverc** network license file supplied by Thermo-Calc Software ([Requesting the License File for Windows or Linux](#)) as well as the actual License Server—a file (program) called **lserver**.
- The Thermo-Calc license file needs to be readable and writable by the license server (license manager).

## Differences Between 32- and 64-bit Scripts

Sentinel RMS License Server version 9.7 supports both 32- and 64-bit. However, when writing scripts be aware of script naming and do not mix the scripts.

- All 32-bit scripts are named `<script_name>`, for example `restart_lserv`
- All 64-bit scripts are named `<script_name>64`, for example, `restart_lserv64`



It is not possible to use Linux 32-bit scripts/tools to manage license server 64-bit and vice versa, you cannot use Linux 64-bit scripts/tools to manage license server 32-bit

## openSUSE Administration (sudo) Users

Note the following if you are a sudo user (administrator) in openSUSE installing Thermo-Calc using this command:

```
sudo ./Thermo-Calc-linux-<full-version-number>.run
```

When this command is used, you will do a text-based installation instead of having a graphical interface to guide you. The information is the same, just without the user interface.



For example, the *<full-version-number>* is not just 2024a, it is what is found in the **Help** → **About** window in Thermo-Calc and it changes every release. You can find the [current full version number listed on our website](#).

## Supported Linux Operating Systems and Versions



For macOS, the Sentinel RMS License software is installed on a Windows or Linux computer server where the Thermo-Calc license file must also be located.

Both Linux 32-bit and 64-bit operating systems are supported for the installation of Sentinel RMS License Server.

- Red Hat Enterprise Linux (RHEL) 7.x
- Red Hat Enterprise Linux (RHEL) 8.x
- SUSE Linux Enterprise Server (SLES)12.x
- SUSE Linux Enterprise Server (SLES) 15.x
- Debian Linux 9.x
- Debian Linux 10.x
- Ubuntu Linux 16.x (LTS)
- Ubuntu Linux 18.x (LTS)
- Ubuntu Linux 20.x (LTS)
- OpenSUSE Leap 42.x
- OpenSUSE Leap 15.x

## Linux: Installing the Sentinel RMS License Server



First review [Linux: About Installing the License Server](#). Then if this is a new installation of the Sentinel RMS License Server, start with Step 2.

Sentinel RMS License Server version 9.7 supports both 32- and 64-bit. However, when writing scripts be aware of script naming and do not mix the scripts.

- All 32-bit scripts are named `<script_name>`, for example `restart_lserv`
- All 64-bit scripts are named `<script_name>64`, for example, `restart_lserv64`



It is not possible to use Linux 32-bit scripts/tools to manage license server 64-bit and vice versa, you cannot use Linux 64-bit scripts/tools to manage license server 32-bit

### *Stop Older Versions of the Sentinel RMS License Server*

1. To stop an old version of the License Server already running, go to the directory where the old file is found, and enter the commands:

- 32-bit license server

```
>chmod +x <lsrvdown>
>./lsrvdown <host_name>
```

- 64-bit license server

```
>chmod +x <lsrvdown64>
>./lsrvdown64 <host_name>
```

2. Edit existing scripts that start the older version of the License Server so that they start the newer version instead.

For more information about the **lsrvdown** utility, go to <https://docs.sentinel.thalesgroup.com/softwareandservices/rms/>

### *Write a Script to Start the License Server*

The following is an excerpt from the template script for starting the License Server found in the `restart_lserv` file.

Replace the paths for `LICENSE_SERVER_DIR`, `LICENSE_FILE_DIR` and `LOG_DIR` to match your system. Save it as a script:

```
# set the directory where the executables are located
LICENSE_SERVER_DIR=/usr/local/Thermo-Calc/2024a/Licensing-files
# set the directory of the license file lservrc
LICENSE_FILE_DIR=/usr/local/Thermo-Calc/2024a/Licensing-files
# set the directory where you wish to save the log-files
LOG_DIR=/tmp
```

```
# stop the running server
# (omit this line if calling this script at startup)
$LICENSE_SERVER_DIR/lsvrdown $HOSTNAME
# start the license server
$LICENSE_SERVER_DIR/lserv -s $LICENSE_FILE_DIR/lservrc -l $LOG_DIR/lserv_use.log
-f $LOG_DIR/lserv_err.log
```

### *Confirm the HOSTNAME variable and open the UDP port*

The hostname should always correspond to the DNS record verify this by doing the following.

1. Before starting the License Server make sure that the value of the `$HOSTNAME` variable is correct. This value can be shown by typing the `> nslookup $HOSTNAME` command in the terminal window:

The value should be the same IP address that is shown when you use the `ifconfig` command.

2. Open the UDP port 5093 in your firewall. In case you have installed SELinux you might also need to disable it by setting `SELINUX=permissive` in the `/etc/selinux/config` file.

### *Start the License Server*

1. To start the License Server, run the script created as root. Enter these commands in the console:

```
> chmod +x <name_of_script_file>
> ./<name_of_script_file>
```

2. If you want the License Server to start when you boot the system, put a copy of the start script under `/etc/rc.d/init.d` and start the script from the file `/etc/rc.d/rc.local`.

### *Install Additional Tool Options*



[Sentinel RMS License 9.7 Tools](#)

## Thin Client Installations

---

This topic describes how you install a thin Thermo-Calc client to run Thermo-Calc from a file server. It is applicable to consolidated and distributed network installations for either a Windows or Linux operating system.

### Installation Requirements

- You need access to the directory where Thermo-Calc is installed on the file server.
- You need the IP number or name of the computer where the license file is installed.
- For *Linux* you may have to disable your firewall. If SELinux is installed, try disabling it by setting `SELINUX=permissive` in `/etc/selinux/config`.

### Windows Installations

The following details may also be of use to those installing in a Windows environment.

- Requests license server name or address and sets LSHOST as a user environment variable accordingly.
- Sets `TC<version>_HOME` (where `<version>` is the software version, e.g. `TC22A_HOME`) as a user environment variable to point to the location from which the thin client installer is running.



The default folders where documents, materials files, examples, and other folders that sometimes require additional licenses are installed under **Public Documents** for ALL USERS. See [Installation Default Directory: Windows](#).

### *How to Install a Thin Client on Windows*

1. Map the file server share as a network drive so that it gets an assigned drive letter (e.g. **E:**).
2. Open the `\Thermo-Calc\<version>` directory on the file server, where *version* is the current version of the software.

3. Open the directory **Thin-client**. Double-click the `Thermo-Calc-client-windows-<version>.exe` file.
4. Click to accept the terms of the license agreement. Click **Next**.
5. Enter the name or IP number of where the license file and the License Manager software is installed. Click **Next**.
6. By default the **Create Shortcuts on Desktop** checkbox is selected. Click **Next**.



The shortcuts to the program are added to the user's Windows roaming profile as well as placed on the user's Desktop (not the system's).

7. In the **Ready to Install** window, click **Next**.
8. Click **Finish**. To start Thermo-Calc find the program on the server and start it from there.

## How to Install a Thin Client on Linux

1. Add the following to the `smb.conf` file, where `<user>` is your user name:

```
[public]
path=/home/<user>/Thermo_Calc/2024a
public=yes
writable=yes
comment=smb share
guest OK=yes
force user=<user>
```

2. From the client, mount the directory on the server computer that contains the Thermo-Calc server installation (by default, this directory is called **/Thermo-Calc/2024a**). How the directory is mounted depends on the Linux distribution being used as well as how the Thermo-Calc directory on the server computer is shared. Use the Linux command `mount`. For example:

```
> mkdir public
> sudo mount -t cifs //ip_number_server_computer/public ./public -o
user=<user>
```

3. Open the mounted directory on your computer, and run `client-install-linux-x64.run` or `client-install-linux-x86.run` in the **/Thin-client** directory.
4. If you do this from the terminal window, use the following to start the installation (assuming that the file server is a 64-bit computer):  

```
> ./client-install-linux-x64.run
```
5. Click **Forward** and click to select the license agreement. Click **Forward**.
6. Enter the name or IP number of the License Server. Click **Forward**.
7. If you are installing as a non-root user, by default, the **Create Shortcuts on Desktop** checkbox is selected. Click **Forward**.
8. In the **Ready to Install** window click Forward.
9. Click **Finish**. Log out and log back in to start using Thermo-Calc.

---

## Custom Network Client Installations

---

This installation type is available for all operating systems, however for macOS the License Server must be run on a Windows or Linux computer. The requirements are the same as for a full client installation.

### Install the Licensing Software

Follow the instructions for your operating system:

- [Windows: About Installing the License Manager](#), or
- [Linux: About Installing the License Server](#)

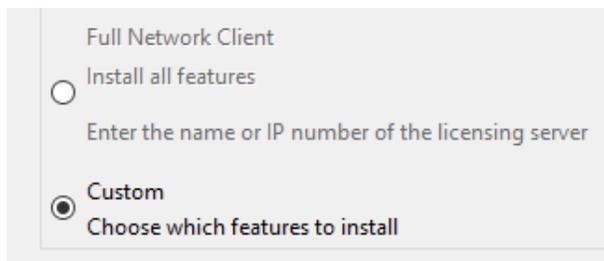
### Complete Steps 1 to 4 of a Full Standalone Installation

Follow the instructions for [Full Standalone Installations](#) to download and run the software. It is up to you whether you get the computer information for the license files before or after installing the software.

### Select the Installation Mode and the Components to Include or Exclude

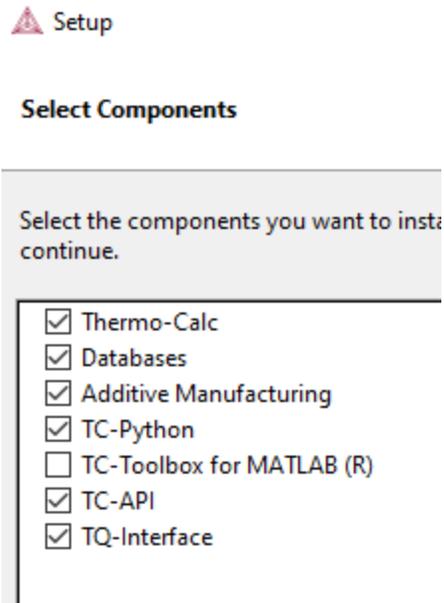
#### [About the Thermo-Calc Components Being Installed](#)

1. In the **Select installation mode** window click **Custom** (the option appears greyed out, but it is available to select). Click **Next**.



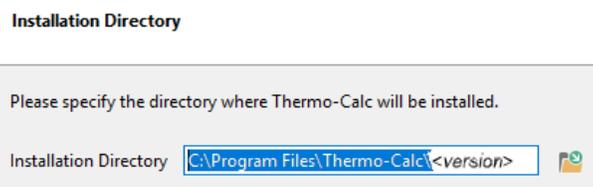
2. On the **Specify Configuration** window the **Create Shortcut on Desktop** checkbox is selected by default. Click **Next**.

3. In the **Select Components** window, click to select or deselect the checkboxes as needed. Click **Next**.



## Set the Installation Directory and the File Path to the License

1. In the **Installation Directory** window, either accept the default path or enter a path to where you want to install the software (see [Installation Default Directories and the License File](#)). Click the file button  to **Browse for Folder** and navigate to another directory. Click **Next**.



To prevent errors, leave this field blank if you do not have a license file yet.

- In the **Choose Setup Type** window, click **Network Client** and enter the name or IP number of the computer where the licensing software is installed. Click **Next**.

#### Choose Setup Type

Select one of the following setup types.  
See installation guide for detailed description of the choices.

Standalone  
 Choose a license file

Network Client  
 Enter name or IP number of the licensing server



If you have selected to install **TC-Python** there is a reminder window that additional installation procedures are required after Thermo-Calc is installed. See the *TC-Python Installation Guide*.

## Install the Software and Start Using Thermo-Calc

- In the **Ready to Install** window click **Next**.
  -  See the separate instructions for TC-Toolbox for MATLAB®.
- Click **Finish** and start using Thermo-Calc. For Linux users, log out and log back in to finish the installation.



macOS installations and if you have previous Thermo-Calc installations: When you launch Thermo-Calc with the new or updated version, the splash screen hides the **Import User Settings** dialog box. In order to proceed you need to click the splash screen and then click **Yes** or **No** in the dialog box.



If you did not get your license file see [Request a License File from Thermo-Calc Software](#).

## Unattended Network Client Installations

---



**Terminology note:** The use of the term *license server* (lower case letters) is applicable to the option files as described in this section. For the purposes of the step-by-step instructions it is not used to refer to the type of network installation that runs the licensing software.

Sometimes it is convenient to install Thermo-Calc network clients without providing user input after the installation is initiated (an unattended network client installation). This is available for Windows and Linux operating systems.



Make sure you know the name or IP number of the computer where the licensing software and the license file are to be installed.

### Create an Option File in a Text Editor to Enter the Options

The following is an example of an option file for a custom network client installation, where *<name/IP number>* is the name or IP number of the computer where the license file is installed:

```
installation_mode=custom
setup_type=client
enable-components=thermo,databases,tq,tcapi,toolbox,tcpython
icons=1
license_server=<name/IP number>
```

Below is a shorter example, specifying that a full Thermo-Calc network client should be installed:

```
installation_mode=full_client
license_server=<name/IP number>
```

---

## Go to the Directory Where the Installation File is Saved



Thermo-Calc can only be installed in unattended mode from the command line prompt.

Start the installation in unattended mode using the option file created with this command:

- For *Windows*: `<installationfile>.exe --mode unattended --optionfile <path\optionfile>`
- For *Linux*: `<installationfile>.run --mode unattended --optionfile <path/optionfile>`

To see other options when you start the Thermo-Calc installation program from the command line prompt, run this command:

- For *Windows*: `<installationfile>.exe --help`
- For *Linux*: `<installationfile>.run --help`

---

# Request License Files

---

**In this section:**

---

Request a License File from Thermo-Calc Software .....	64
Getting License Information Using Thermo-Calc .....	65
Requesting the License File for Windows or Linux .....	66
Requesting the License File for macOS .....	68

---

## Request a License File from Thermo-Calc Software

---

The following explains how to retrieve the necessary diagnostic and network adaptor information about your computer and then send an email to Thermo-Calc Software to get your license file(s) containing the license key(s).

To set up the correct licensing:

1. Make a note of your company or organization's *site number*. You can find the site number on an invoice or any correspondence from Thermo-Calc Software.
2. Retrieve your computer diagnostic and network adapter information.
3. Email Thermo-Calc Software with this information.
4. A Thermo-Calc Software representative emails you the license file. Save the license file **lservrc** (there is no file suffix) to a location where you can easily access it later.



[Thermo-Calc Support Center on our website](#)

---

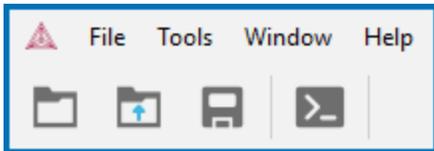
## Getting License Information Using Thermo-Calc

---

If you have already installed Thermo-Calc, you can view the license information in the GUI. Select **Help** →  **Show License Info**.

In Console Mode you can run the SYS module command DISPLAY\_LICENSE\_INFO.

1. Open Thermo-Calc.
2. Click **Switch to Console Mode**.



3. At the `SYS:` prompt, enter `DISPLAY_LICENSE_INFO`. You can output the information to a `FILE` or to the `SCREEN`.

```
SYS:DISPLAY_LICENSE_INFO
OUTPUT TO SCREEN OR FILE /SCREEN/:SCREEN|
```

4. For both options, the information is displayed on screen. Copy the section containing the **Ethernet** and **IP address**. You can save it to a text file or paste it directly into an email. If you choose `FILE` as the output, then you can attach the file to an email.

## Requesting the License File for Windows or Linux



**Upgrades to a new version of Thermo-Calc:** Skip this section if you are upgrading to a new version of Thermo-Calc and (and you have a maintenance plan). Your license is sent to you in an email from Thermo-Calc Software AB. Save it to your computer to use during software installation.

1. To access the computer information you need to provide to get a license, start Thermo-Calc and select **Help → Show License Info**.

```
License Information

License diagnostics program
  build: 26030 created: Tue May 03 09:30:22 2022

Log generated at: 13-05-2022 14:34:42
Safenet environment variables
LSHOST: NO-NEI
LSERVRC: C:\Program Files\Thermo-Calc\lservrc

Local (client) info
hostname:      TC-LI-05
license version: 8.5.1.0017
```

2. Copy the information from the **License Information** window. A lot of information may be shown below the initial screen, so make sure you scroll down to the end when you select and copy the window's contents. (Alternatively, you can copy the same information from the console by using the SYS module command DISPLAY\_LICENSE\_INFO.)
3. Send an email to [support@thermocalc.com](mailto:support@thermocalc.com). Make sure to include the following in the subject line:

**License diagnostic information from <Name>** where Name is replaced with your **name**, the **organization name** and the **Site Number**. *You can find the five-digit site number on an invoice or any correspondence from Thermo-Calc Software.*

```
Designated Site 12345
XYZ Company
Dpt. of Material Science
900 Main Street
Stockholm 11190
SWEDEN
```



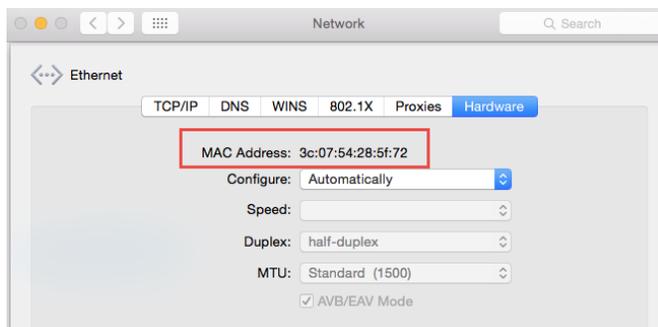
4. A representative from Thermo-Calc Software replies to the email and attaches your license file(s).
5. Save the license file(s) to your computer or network.

## Requesting the License File for macOS



**Upgrades to a new version of Thermo-Calc:** Skip this section if you are upgrading to a new version of Thermo-Calc and (and you have a maintenance plan). Your license is sent to you in an email from Thermo-Calc Software AB. Save it to your computer to use during software installation.

1. From the Apple main menu, select **System Preferences**.
2. Click **Network**.
3. In the left column select **Ethernet** or **Built-in Ethernet** (do not select a WiFi connection as a local static MAC address is required).
4. Click **Advanced** → **Hardware**. The **Network** window shows you the **MAC Address**. For example, the MAC address (the host ID) might be **3c:07:54:28:5f:72**.



5. Send an email to [support@thermocalc.com](mailto:support@thermocalc.com) and copy and paste the MAC address into the body of the email. Make sure to include the following in the subject line:

**License diagnostic information from <Name>** where Name is replaced with your **name**, the **organisation name** and the **Site Number**. *You can find the five-digit site number on an invoice or any correspondence from Thermo-Calc Software.*

Designated Site 12345  
XYZ Company  
Dpt. of Material Science  
900 Main Street  
Stockholm 11190  
SWEDEN

To: support@thermocalc.com

Cc:

License diagnostic information from Jane Smith, XYZ Company, Site 12345

Add attachments  tclog.log (7.3 KB) 

6. A representative from Thermo-Calc Software replies to the email address provided and attaches your license file or files.
7. Save your license file to your computer or network.

---

## Quick Install Guides

---

**In this section:**

---

Windows® Standalone Quick Install Guide .....	71
macOS® Standalone Quick Install Guide .....	75

## Windows® Standalone Quick Install Guide

---

This quick guide helps you do a full standalone installation. A *standalone* installation is used with the Single-User Node-Locked License (SUNLL), where the software and the license file are together on one computer.

*This guide is applicable to:*

- Windows operating systems
- Full Standalone installation (SUNLL)
- Upgrading to a new standalone version of Thermo-Calc (maintenance plan only)

### *Other Installations*



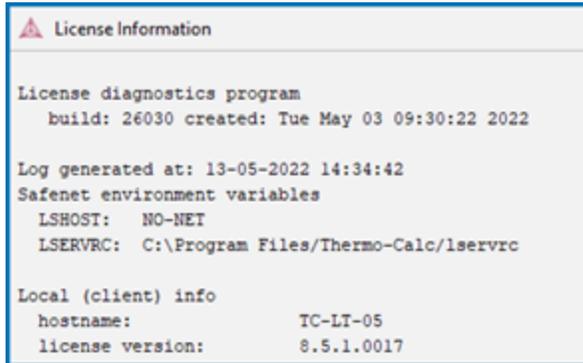
For instructions about other operating systems, network installations, or installing an SDK (e.g. TC-Python or TC-Toolbox for MATLAB®) search the *Thermo-Calc Installation Guide*, which is also [available on our website](#). You can also review the [Licensing Options](#) included on our website.

## Request a License File



**Upgrades to a new version of Thermo-Calc:** Skip this section if you are upgrading to a new version of Thermo-Calc and (and you have a maintenance plan). Your license is sent to you in an email from Thermo-Calc Software AB. Save it to your computer to use during software installation.

1. To access the computer information you need to provide to get a license, start Thermo-Calc and select **Help → Show License Info**.



```
License Information

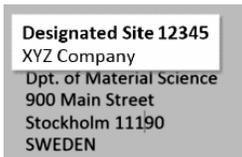
License diagnostics program
  build: 26030 created: Tue May 03 09:30:22 2022

Log generated at: 13-05-2022 14:34:42
Safenet environment variables
  LSHOST: NO-NET
  LSERVR: C:\Program Files\Thermo-Calc\lservrc

Local (client) info
  hostname: TC-LI-05
  license version: 8.5.1.0017
```

2. Copy the information from the **License Information** window. A lot of information may be shown below the initial screen, so make sure you scroll down to the end when you select and copy the window's contents. (Alternatively, you can copy the same information from the console by using the SYS module command DISPLAY\_LICENSE\_INFO.)
3. Send an email to [support@thermocalc.com](mailto:support@thermocalc.com). Make sure to include the following in the subject line:

**License diagnostic information from <Name>** where Name is replaced with your **name**, the **organization name** and the **Site Number**. *You can find the five-digit site number on an invoice or any correspondence from Thermo-Calc Software.*



```
Designated Site 12345
XYZ Company
Dpt. of Material Science
900 Main Street
Stockholm 11190
SWEDEN
```



To: support@thermocalc.com

Cc:

License diagnostic information from Jane Smith, XYZ Company, Site 12345

Add attachments  tcllog.log (7.3 KB)

4. A representative from Thermo-Calc Software replies to the email and attaches your license file(s).
5. Save the license file(s) to your computer or network.

## Install the Software

### Download and Run your Thermo-Calc Installation Software



Administrative rights are required to complete this installation.

1. For web links sent in an email from Thermo-Calc Software, go to the URL and log in with the information provided. Then navigate to the Windows folder and click the link to download the software.
2. Go to the folder where you downloaded the software. Double-click the **.exe** file to launch the **Setup** program.
3. On the **Setup-Thermo-Calc** window click **Next**.
4. On the **License Agreement** window click to accept the license agreement. Click **Next**.

### Select the Installation Mode

1. In the **Select Installation Mode** window click **Full Standalone: Install all features**.

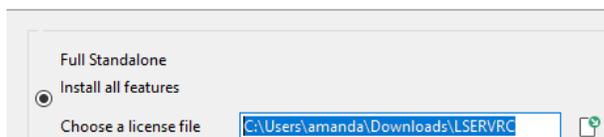


TC-Python is installed when the **Full Standalone** or **Full Network Client** option is chosen, but there are additional steps required as described separately for this SDK.

2. For a **Full Standalone** installation, in the **Choose a license file** field either enter the file path or click the file button  to navigate to where you saved the license file emailed to you from Thermo-Calc Software.

 Setup

Select Installation Mode





The license file is called **lservrc**. Click **Next**. To prevent errors, leave this field blank if you did not get a license file yet.



TC-Python is installed for all users even if you do not have a license. The **Install TC-Python** window is a reminder that additional installation procedures are required after Thermo-Calc is installed. See the *TC-Python Installation Guide*.

3. In the **Ready to Install** window click **Next**. Click **Finish** and start using Thermo-Calc.

## macOS® Standalone Quick Install Guide

This quick guide helps you do a full standalone installation. A *standalone* installation is used with the Single-User Node-Locked License (SUNLL), where the software and the license file are together on one computer.

*This guide is applicable to:*

- macOS
- Full Standalone installation (SUNLL)
- Upgrading to a new standalone version of Thermo-Calc (maintenance plan only)

### Other Installations



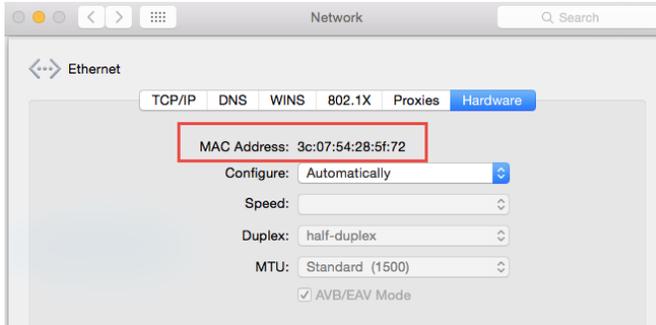
For instructions about other operating systems, network installations, or installing an SDK (e.g. TC-Python or TC-Toolbox for MATLAB®) search the *Thermo-Calc Installation Guide*, which is also [available on our website](#). You can also review the [Licensing Options](#) included on our website.

## Request a License File



**Upgrades to a new version of Thermo-Calc:** Skip this section if you are upgrading to a new version of Thermo-Calc and (and you have a maintenance plan). Your license is sent to you in an email from Thermo-Calc Software AB. Save it to your computer to use during software installation.

1. From the Apple main menu, select **System Preferences**.
2. Click **Network**.
3. In the left column select **Ethernet** or **Built-in Ethernet** (do not select a WiFi connection as a local static MAC address is required).
4. Click **Advanced** → **Hardware**. The **Network** window shows you the **MAC Address**. For example, the MAC address (the host ID) might be **3c:07:54:28:5f:72**.



5. Send an email to [support@thermocalc.com](mailto:support@thermocalc.com) and copy and paste the MAC address into the body of the email. Make sure to include the following in the subject line:

**License diagnostic information from <Name>** where Name is replaced with your **name**, the **organisation name** and the **Site Number**. *You can find the five-digit site number on an invoice or any correspondence from Thermo-Calc Software.*

Designated Site 12345  
XYZ Company  
Dpt. of Material Science  
900 Main Street  
Stockholm 11190  
SWEDEN



6. A representative from Thermo-Calc Software replies to the email address provided and attaches your license file or files.
7. Save your license file to your computer or network.

## Install the Software

### Download and Install XQuartz

The XQuartz open-source X Windows system is required to run Thermo-Calc on macOS X.

1. In a web browser go to <https://www.macupdate.com> or to download it directly from XQuartz, <https://www.xquartz.org/>.
2. In the **Search Mac Apps** field, enter *XQuartz*.
3. Click the **XQuartz** search result. For example, based on the recent version, the name is XQuartz-2.7.11.dmg.

4. Click **Download**.
5. Double-click the **XQuartz.pkg** file. Follow the instructions.
6. Click **OK** when the message about logging out displays and click **Close** when the installation is successful.

### *Download and Run your Thermo-Calc Installation Software*



Administrative rights are required to complete this installation.

1. For web links sent in an email from Thermo-Calc Software, go to the URL and log in with the information provided. Then navigate to the Mac folder and click the link to download the software.
2. Double-click the downloaded file to launch the installation.



For macOS and when you first run the installer, you may see a message that *Thermo-Calc-osx-<version> can't be opened because Apple cannot check it for malicious software*. If this happens, select **Show in Finder**. Then, in the opened **Finder** window right-click the downloaded file `Thermo-Calc-osx-<version>` and select **Open**.

3. On the **Setup-Thermo-Calc** window click **Next**.
4. On the **License Agreement window** click to accept the license agreement. Click **Next**.

### *Select the Installation Mode*

1. In the **Select Installation Mode** window click **Full Standalone: Install all features**.

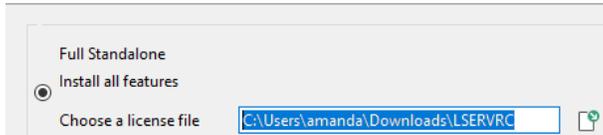


TC-Python is installed when the **Full Standalone** or **Full Network Client** option is chosen, but there are additional steps required as described separately for this SDK.

- For a **Full Standalone** installation, in the **Choose a license file** field either enter the file path or click the file button  to navigate to where you saved the license file emailed to you from Thermo-Calc Software.

 Setup

Select Installation Mode



The license file is called **lservrc**. Click **Next**. To prevent errors, leave this field blank if you did not get a license file yet.



TC-Python is installed for all users even if you do not have a license. The **Install TC-Python** window is a reminder that additional installation procedures are required after Thermo-Calc is installed. See the *TC-Python Installation Guide*.

- In the **Ready to Install** window click **Next**. Click **Finish** and start using Thermo-Calc.



macOS installations and if you have previous Thermo-Calc installations: When you launch Thermo-Calc with the new or updated version, the splash screen hides the **Import User Settings** dialog box. In order to proceed you need to click the splash screen and then click **Yes** or **No** in the dialog box.

---

# TC-Python Installation Guide

---

**In this section:**

---

About the TC-Python Documentation .....	80
Which TC-Python Installation Should I Choose? .....	82
Installing TC-Python with a Bundled Python-interpreter .....	83
Installing TC-Python Into a Python-interpreter of Your Choice .....	87
Installation Default Directory: TC-Python .....	92
macOS: Setting Environment Variables .....	93
Fixing Potential Issues with the Environment .....	94
Optional Packages .....	95
Updating to a Newer Version .....	96
TC-Python Diagnostic Script .....	97

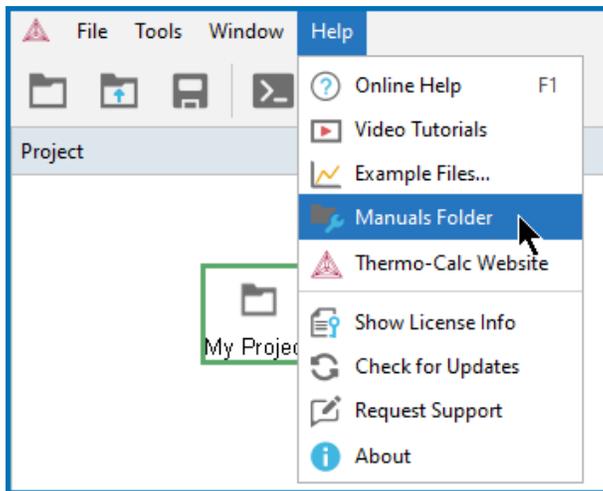
## About the TC-Python Documentation



The TC-Python help is [available on the Thermo-Calc website](#) where you can also access [additional resources](#). You can alternatively access the locally installed documentation from the **Help** menu in Thermo-Calc.

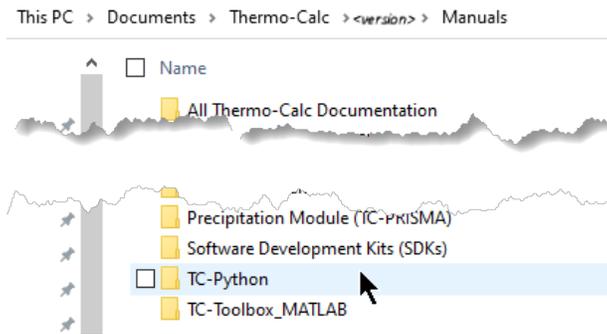
### Opening the Locally Installed Documentation

1. In Thermo-Calc from the menu, select **Help** → **Manuals Folder**.

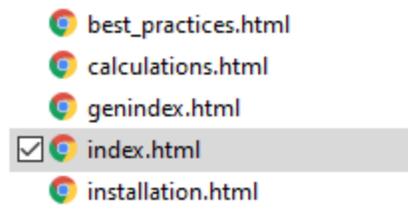


You can also navigate to the **Manuals** folder outside of Thermo-Calc and based on your operating system installation location.

2. In the window that opens, double-click to open the **TC-Python** folder.



3. Double-click the **Index** HTML file (or any of the HTML files).



4. The API reference documentation opens as a searchable HTML file in your local browser.

---

## Which TC-Python Installation Should I Choose?

---



A license is required to run TC-Python. Also remember to review the [System Requirements](#) page on our website.

There are these options to install TC-Python.



See the [Videos about TC-Python](#) page for additional learning resources.

### ***Install with the Bundled Python-interpreter***

This is a straightforward installation where TC-Python is preinstalled together with some popular Python packages.

This is the recommended option for new users to TC-Python who may only want to run the examples. The available preinstalled packages (e.g. numpy, scipy, scikit-learn, pyvista, and matplotlib) can also be sufficient for many advanced users.



[Installing TC-Python with a Bundled Python-interpreter.](#)



[Optional Packages](#)

### ***Install with a Python-interpreter of Your Choice***

This installation requires additional steps and is the recommended option for more advanced use and has full flexibility.



[Installing TC-Python Into a Python-interpreter of Your Choice.](#)

## Installing TC-Python with a Bundled Python-interpreter

---

TC-Python can be installed using a Python-interpreter that is bundled to Thermo-Calc. This interpreter has TC-Python preinstalled together with some popular Python packages. This is the recommended option for new users to TC-Python who may only want to run the examples. The available preinstalled packages (e.g. numpy, scipy, scikit-learn, pyvista, and matplotlib) can also be sufficient for many advanced users.



The bundled interpreter package is also used to run the Property Models in Thermo-Calc. Do not make changes to interpreter packages or this can stop Thermo-Calc from working correctly. To fix issues caused by changes to an interpreter, completely uninstall Thermo-Calc and start over, either with the bundled interpreter package or using interpreters of your choice.



If you are planning to do more advanced calculations that require other external packages then you may want to follow these instructions instead: [Installing TC-Python Into a Python-interpreter of Your Choice](#)

### Limitations



The following examples cannot be run with the bundled Python-interpreter as additional packages are required: `pyex_M_01_Input_from_file.py` (pandas) and `pyex_M_02_Output_to_file.py` (pandas, lxml, h5py).

The bundled Python 3.9.18 interpreter contains the following major packages.



If you have recommendations for additional packages to include (see below), send an [email to Thermo-Calc Support](#).

<i>Package</i>	<i>Version</i>
matplotlib	3.8
numpy	1.26.0
scikit-learn	1.3.1
scipy	1.11.3
pyvista	0.42.2
TC-Python	2024a

## Install an IDE (Integrated Development Environment)

Any editor can be used to write the Python code, but an IDE is recommended, e.g. PyCharm. These instructions are based on the use of PyCharm.

Use of an IDE gives you access to code completion, which helps when you use the API as it gives you the available methods on the objects you are working with.

1. Navigate to the PyCharm website: <https://www.jetbrains.com/pycharm/download>.
2. Click to choose your OS and then click **Download**. You can use the **Community** version of Pycharm.
3. Follow the instructions. It is recommended you keep all the defaults.



For Mac installations, you also need to set some environment variables as described below in [macOS: Setting Environment Variables](#).

## Configure PyCharm to Use the Bundled Python- interpreter

Open PyCharm and configure the interpreter:

1. Go the menu **File** → **Settings**.
2. Navigate in the tree to **Project: YourProjectName** and choose **Project Interpreter**.
3. Click on the settings symbol close to the **Project Interpreter** dropdown menu and choose **Add**.
4. Select **System Interpreter** and add the bundled Thermo-Calc Python 3 interpreter, which is located in a folder based on the operating system:

<i>Operating system</i>	<i>Path to the bundled Python-interpreter</i>
Windows	C:\Program Files\Thermo-Calc\2024a\python\python.exe
Linux	/home/UserName/Thermo-Calc/2024a/python/bin/python3
macOS	/Applications/Thermo-Calc-2024a.app/Contents/Resources/python/bin/python3

5. Select the interpreter and confirm.

## Run a TC-Python Example

It is recommended that you open one or more of the included examples to both check that the installation has worked and to start familiarizing yourself with the code.

### *Open the TC-Python Project in PyCharm*



When you first open the TC-Python project and examples, it can take a few moments for the Pycharm IDE to index before some of the options are available.

1. Open PyCharm and then choose **File** → **Open**. The first time you open the project you will need to navigate to the path of the TC-Python installation based on your operating system.

<i>Operating system</i>	<i>Path to the TC-Python folder</i>
Windows	C:\Users\UserName\Documents\Thermo-Calc\2024a\SDK\TC-Python
Linux	/home/UserName/Thermo-Calc/2024a/SDK/TC-Python
macOS	/Users/Shared/Thermo-Calc/2024a/SDK/TC-Python

2. Click the **Examples** folder and then click **OK**.
3. From any subfolder:
  - Double-click to open an example file to examine the code.
  - Right-click an example and choose **Run**.



If you are not following the recommended approach and create a *new* project (**File** → **New Project...**), you need to consider that by default the options to choose the interpreter are hidden within the **Create Project** window. For this, click **Project Interpreter: New Virtual Environment** and in most cases choose your *System Interpreter* containing the Python bundled to Thermo-Calc instead of the default *New Virtual Environment*.

## Installing TC-Python Into a Python-interpreter of Your Choice

---

TC-Python can be installed using a Python-interpreter of your choice. This is the recommended option for any more advanced usage and provides full flexibility.



If you are new to TC-Python and want to mainly run the examples, then it is recommended you install the bundled Python-interpreter. The available preinstalled packages (e.g. numpy, scipy, scikit-learn, pyvista, and matplotlib) can also be sufficient for many advanced users. See [Installing TC-Python with a Bundled Python-interpreter](#).

### Install a Python Distribution



If you already have a Python distribution installation, version 3.8 or newer, skip this step.

These instructions are based on using the Anaconda platform for the Python distribution. Install version 3.8 or newer to be able to work with TC-Python, although it is recommended that you use the most recent version.

#### INSTALL ANACONDA

1. Navigate to the Anaconda website: <https://www.anaconda.com/download/>.
2. Click to choose your OS (operating system) and then click **Download**. Follow the instructions. It is recommended you keep all the defaults.

### Install Thermo-Calc and the TC-Python SDK

1. Install Thermo-Calc.
2. When the installation is complete, open the TC-Python folder that includes the \*.whl file needed for the next step. There is also an **Examples** folder with Python files you can use in the IDE to understand and work with TC-Python.

## Install TC-Python

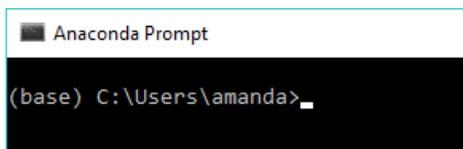
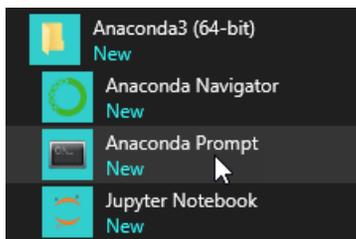
### [Updating to a Newer Version](#)

On Windows, it is recommended you use the Anaconda distribution prompt (i.e. Anaconda), especially if you have other Python installations.



Do not use Virtual Environments unless you have a good reason to do so. Also see [Fixing Potential Issues with the Environment](#).

1. Open the command line. For example, in Anaconda on a Windows OS, go to **Start** → **Anaconda** → **Anaconda Prompt**.



2. At the command line, enter the following. Make sure there are no spaces at the end of the string or in the folder name or it will not run:

```
pip install <path to the TC-Python folder>/TC_Python-<version>-py3-none-any.whl
```



On Linux, and depending on the interpreter, *pip3* is typically used.

Operating system	Path to the TC-Python folder
Windows	C:\Users\UserName\Documents\Thermo-Calc\2024a\SDK\TC-Python
Linux	/home/UserName/Thermo-Calc/2024a/SDK/TC-Python
macOS	/Users/Shared/Thermo-Calc/2024a/SDK/TC-Python

- Press <Enter>. When the process is completed, there is a confirmation that TC-Python is installed.



If your computer is located behind a proxy-server, the default pip-command will fail with a network connection error. In that case you need to install the dependency py4j in a special configuration:

```
pip install -proxy user:password@proxy_ip:port py4j jproperties
```



Go to the TC-Python Help **Troubleshooting** section for detailed information. This can be found [on our website](#) or from the installed documentation where you can search for "pip install fails" with "Failed to establish a new network connection" or similar.

## Install an IDE (Integrated Development Environment)

Any editor can be used to write the Python code, but an IDE is recommended, e.g. PyCharm. These instructions are based on the use of PyCharm.

Use of an IDE gives you access to code completion, which helps when you use the API as it gives you the available methods on the objects you are working with.

- Navigate to the PyCharm website: <https://www.jetbrains.com/pycharm/download>.
- Click to choose your OS and then click **Download**. You can use the **Community** version of Pycharm.
- Follow the instructions. It is recommended you keep all the defaults.



For Mac installations, you also need to set some environment variables as described below in [macOS: Setting Environment Variables](#).

## Open the IDE and Run a TC-Python Example

After you complete all the software installations and set up the pip install path, you are ready to open the IDE to start working with TC-Python.

It is recommended that you open one or more of the included examples to both check that the installation has worked and to start familiarizing yourself with the code.

### *Open the TC-Python Project in PyCharm*



When you first open the TC-Python project and examples, it can take a few moments for the Pycharm IDE to index before some of the options are available.

1. Open PyCharm and then choose **File** → **Open**. The first time you open the project you will need to navigate to the path of the TC-Python installation based on your operating system.

<i>Operating system</i>	<i>Path to the TC-Python folder</i>
Windows	C:\Users\UserName\Documents\Thermo-Calc\2024a\SDK\TC-Python
Linux	/home/UserName/Thermo-Calc/2024a/SDK/TC-Python
macOS	/Users/Shared/Thermo-Calc/2024a/SDK/TC-Python

2. Click the **Examples** folder and then click **OK**.

3. From any subfolder:

- Double-click to open an example file to examine the code.
- Right-click an example and choose **Run**.



Also see [Optional Packages](#) if you are using the Additive Manufacturing Module for simulations.

## Installation Default Directory: TC-Python

---

These are the default paths to the TC-Python folder based on your operating system.

<i>Operating system</i>	<i>Path to the TC-Python folder</i>
Windows	<code>C:\Users\UserName\Documents\Thermo-Calc\2024a\SDK\TC-Python</code>
Linux	<code>/home/UserName/Thermo-Calc/2024a/SDK/TC-Python</code>
macOS	<code>/Users/Shared/Thermo-Calc/2024a/SDK/TC-Python</code>

For a bundled Python-interpreter installation, these are the paths to this location based on your operating system.

<i>Operating system</i>	<i>Path to the bundled Python-interpreter</i>
Windows	<code>C:\Program Files\Thermo-Calc\2024a\python\python.exe</code>
Linux	<code>/home/UserName/Thermo-Calc/2024a/python/bin/python3</code>
macOS	<code>/Applications/Thermo-Calc-2024a.app/Contents/Resources/python/bin/python3</code>

## macOS: Setting Environment Variables

---

In order to use TC-Python on Mac you need to set some environment variables.

```
TC24A_HOME=/Applications/Thermo-Calc-2024a.app/Contents/Resources
```

If you use a license server:

```
LSHOST=<name-of-the-license-server>
```

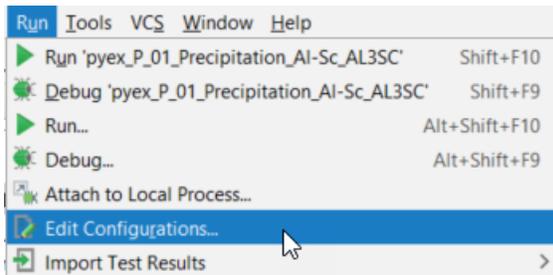
If you have a node-locked license:

```
LSHOST= NO-NET
```

```
LSERVRC=/Users/Shared/Thermo-Calc/lserverc
```

In PyCharm, you can add environment variables in the configurations.

Select **Run** → **Edit Configurations** to open the **Run/Debug Configurations** window. Enter the environment variable(s) by clicking the button to the right of the **Environment Variables** text field. Now the environment variable(s) will be set for each new configuration by default. The same way for configuring the environment variables can be used on other operating systems as if necessary.



## Fixing Potential Issues with the Environment

---

In most cases you should run TC-Python within your **global** Python 3 interpreter and not use Virtual Environments unless you have a good reason to do so.

If there are problems with the interpreter settings, for example, a common problem when you first use TC-Python is the message *No module named tc\_python*, which you can resolve in the Settings window as follows:

1. Go the menu **File** → **Settings**.
2. Navigate in the tree to *<Project.YourProjectName>* and choose **Project Interpreter**.
3. Click on the settings symbol close to the **Project Interpreter** drop-down menu and choose **Add**.
4. Now choose **System Interpreter** and add your existing Python 3 interpreter.
5. Select your added interpreter and confirm.



If you are not following the recommended approach and create a *new* project (**File** → **New Project...**), you need to consider that by default the options to choose the interpreter are hidden within the **Create Project** window. For this, click **Project Interpreter: New Virtual Environment** and in most cases choose your *System Interpreter* containing the Python bundled to Thermo-Calc instead of the default *New Virtual Environment*.



If you really need to use a Virtual Environment, it is recommended you review the hints provided in the **Best Practices** section in the full TC-Python HTML help or PDF.

---

## Optional Packages

---

Plotting results of Additive Manufacturing simulations requires the package *pyvista*. It is automatically installed if using the option *[am-plotting]* during the installation:

```
pip install <path to the TC-Python folder>/TC_Python-<version>-py3-none-any.  
,!whl[am-plotting]
```



The bundled interpreter contains *pyvista* by default.

## Updating to a Newer Version

---



If you are using the bundled Python-interpreter automatically installed with Thermo-Calc, these steps are not required.



These steps are required if you have previously followed the instructions in [Installing TC-Python Into a Python-Interpreter of Your Choice](#).

When updating to a newer version of Thermo-Calc, you always need to also install the latest version of TC-Python. It is not sufficient to run the installer of Thermo-Calc (except for bundled versions of the Python interpreter).

```
pip install <path to the TC-Python folder>/TC_Python-<version>-py3-  
none-any.whl
```

In case of problems you may wish to uninstall the previous version of TC-Python in advance:

```
pip uninstall TC-Python  
  
pip install <path to the TC-Python folder>/TC_Python-<version>-py3-  
none-any.whl
```

However, this should not normally be required.



On Linux typically *pip3* is used.

You can check the currently installed version of TC-Python by running:

```
pip show TC-Python
```

---

## TC-Python Diagnostic Script

---

A diagnostic script is available to help you if you have problems running TC-Python.



It is important to run the diagnostic script EXACTLY the same way as you run your TC-Python script (in the same IDE, same project, same Python environment, same Jupyter notebook, etc).

The script automatically prints useful suggestions that, in many cases, will help you to fix the problem yourself. If you are unable to solve the problem, contact us for support. You can find the script in these locations:

- Miscellaneous example (diagnostics.py)
- In the **Troubleshooting** section of the *TC-Python API Reference Documentation*. Also refer to the **Best Practices** in the same guide to see if there are any other solutions or tips.
- On our [website](#).

---

# TC-Toolbox for MATLAB® Installation Guide

---

**In this section:**

---

About TC-Toolbox for MATLAB® .....	99
TC-Toolbox Documentation .....	100
Installation Default Directory: TC-Toolbox and the Examples .....	102
Installing TC-Toolbox .....	103
Installing TC-Toolbox with the Custom Option .....	105
Testing a TC-Toolbox Installation .....	109
TC-Toolbox Installation Troubleshooting .....	110
Uninstalling TC-Toolbox for MATLAB® .....	112

## About TC-Toolbox for MATLAB®

---

TC-Toolbox for MATLAB® (TC-Toolbox) provides an interface to the commonly used MATLAB® software for scientific and engineering computing. This software development kit (SDK) is ideal for fast realization of ideas and visualization of results during research and development activities. The TC-Toolbox API includes calculation types available in Thermo-Calc such as single point equilibrium, property (step) and phase (map) diagrams, Scheil solidification simulations, as well as all the general Thermo-Calc Property Models (e.g. yield strength, spinodal, driving force, etc.).

With the addition of licenses for the applicable software and databases, you can further maximize your integration with MATLAB® to include the capabilities of the Steel Model Library Property Models (e.g. bainite, pearlite, etc.), as well as access the range of functionality included with the Diffusion Module (DICTRA) and the Precipitation Module (TC-PRISMA). Generally, all the functionality available in Graphical Mode or via the TC-Python SDK is available.

TC-Toolbox for MATLAB® is available on Windows platforms.



A license is required to run TC-Toolbox for MATLAB® and MATLAB® must also be installed on your computer prior to the TC-Toolbox installation. Also remember to review the [System Requirements](#) page on our website.

## TC-Toolbox Documentation

---

There are various locations and formats of the documentation you need related to this SDK.



The *TC-Toolbox for MATLAB® API Reference Documentation* is available from these general locations (1) within the MATLAB® environment—both as help and as part of the tab completion (2) directly on the [Thermo-Calc website](#), and (3) locally in the Thermo-Calc installation (**Help>Manuals Folder**).

### API Reference Documentation

The *TC-Toolbox for MATLAB® API Reference Documentation* is available either as interactive help (in HTML format or when in MATLAB® using tab completion or accessed via different menus or key combinations) or as a standalone PDF.

There are these ways to access the content:

- Directly in MATLAB® via the Help menu then under **Supplemental Software** click **TC-Toolbox**. Help is also available when writing code and using the tab completion feature and/or pressing different key combinations as described in the *Best Practices* section of the help.
- From the website: [HTML Help](#) or [PDF](#)
- Locally in a Thermo-Calc installation:
  - HTML: **Help > Manuals Folder > TC-Toolbox\_MATLAB**, then click any HTML file.
  - PDF: **Help > Manuals Folder > Software Development Kits (SDKs)**, then double-click to open the *TC-Toolbox for MATLAB® API Programmer Guide*. This is a copy of the HTML API reference documentation.

### Installation Instructions

The installation instructions are only included in the Thermo-Calc help (press F1), and in the PDF version of the main installation guide found, for example:

- On our website as part of the full [Thermo-Calc Installation Guide](#).
- Once Thermo-Calc is installed:
  - Locally in a Thermo-Calc installation: **Help > Manuals Folder > Installation Guides**.
  - Press F1 in Thermo-Calc to search the help.

### *Thermo-Calc Help (Limited)*

For TC-Toolbox, there is limited content included in the Thermo-Calc help (press F1 when in Thermo-Calc) because it is replaced by the *TC-Toolbox for MATLAB® API Reference Documentation*. The exception is installation instructions to describe how to install the SDK.

## Installation Default Directory: TC-Toolbox and the Examples

The software component and the examples are installed as described below.

This is the default path to the installation folder for the TC-Toolbox component on Windows operating systems. Be aware that the initial *installation files and examples* are placed in **Public\Documents**, then after opening Thermo-Calc for the first time, these are copied to the **Documents** folder for each user. The *documentation* is installed elsewhere and as described in [TC-Toolbox Documentation](#).

There are included examples with all TC-Toolbox installations. These are the same as the examples included with TC-Python although with a different suffix (*matex*) and prefix (*\*.m*) in the file name. You can read summaries about the examples as described for TC-Python in the online help.

### Path to the TC-Toolbox folder

Both the installation files AND the examples are installed in the following folders.

The default Windows folders where the TC-Toolbox examples are installed starts with the original installation placing these under **Public Documents** for ALL USERS.

For a local user, and after Thermo-Calc is opened for the first time, these files are copied to the user's **Documents** folder where this local copy is associated to the user login. The **Public Documents** folder always contains the original set of contents as per the installation.

```
C:\Users\Public\Public Documents\Thermo-Calc\<version>\SDK\TC-Toolbox-MATLAB
```

After installing Thermo-Calc, and when you first open the program, the set of folders containing the examples is copied from the source Public\Documents location into a user Documents folder:

```
C:\Users\<user>\Documents\Thermo-Calc\<version>\SDK\TC-Toolbox-MATLAB\Examples
```



It is recommended that you work with the files that are copied to the *user* folder.

## Installing TC-Toolbox

---

There are automatic or manual installation instructions for TC-Toolbox for MATLAB®.

### Automatic Installation of TC-Toolbox

When the following conditions are met, Thermo-Calc automatically installs the TC-Toolbox for MATLAB® component on your computer.

1. MATLAB® is already installed.
2. There is only one Windows user on the computer where TC-Toolbox is being installed.



Administrator privileges are needed when you start the Thermo-Calc installer.

Then follow the regular installation instructions for Thermo-Calc, choosing whether you use a Standalone or Network installation. If required also see [Testing a TC-Toolbox Installation](#).



A reminder regarding the license. Although you do not need to have a license file during the installation process (it can be added after the fact), you do need to have purchased a license for TC-Toolbox for MATLAB® to run the program within MATLAB®.

### Manual Installation of TC-Toolbox

The installation is not automatic if:

- There are multiple Windows users on the same machine, or
- The installer cannot find the directory path to the MATLAB installation.

#### *Multiple Windows Users on Same Machine*

If there are multiple windows users on the same machine, then the following manual steps are done at the end of the automatic installation.

1. A message at the end of the Thermo-Calc installation process displays with instructions.
2. An Explorer window automatically opens to the `C:\Users\Public\Documents\Thermo-Calc\2024a\SDK\TC-Toolbox-MATLAB` folder.
3. Double-click the `InstallTCToolboxMATLAB.cmd` file to finalize the process. This briefly launches MATLAB® and installs TC-Toolbox. If required see [Testing a TC-Toolbox Installation](#).

### *Installer Cannot Find the MATLAB® Installation Directory*

If the Thermo-Calc installer cannot find the MATLAB® installation directory,

1. Start the version of MATLAB® that you want to install TC-Toolbox in.
2. Open and run the MATLAB® script included with your installation of Thermo-Calc.

For example on Windows, `C:\Users\`



You can also follow the instructions for [Installing TC-Toolbox with the Custom Option](#).

---

## Installing TC-Toolbox with the Custom Option

---



MATLAB® must be installed before you install TC-Toolbox.

As described in [Installing TC-Toolbox](#), if MATLAB® is already installed, then Thermo-Calc automatically installs the files you need either using the *Full Standalone* or *Full Network Client* installation where you **install all features**. Otherwise you need to use the **Custom** installation.

The following generally describes the **Custom** option for either a Standalone or Network installation. Use the **Custom** option if MATLAB® is installed but the Thermo-Calc installer is not finding the directory path so you need to enter the path manually.

Some steps are described in more detail in other parts of this guide.

### Request your License File from Thermo-Calc Software



A reminder regarding the license. Although you do not need to have a license file during the installation process (it can be added after the fact), you do need to have purchased a license for TC-Toolbox for MATLAB® to run the program within MATLAB®.

### Download your Thermo-Calc Installation Software

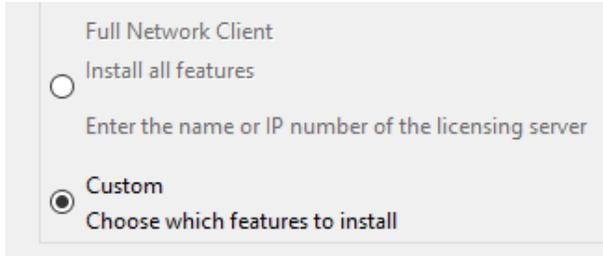
### Install MATLAB®

### Run the Thermo-Calc Installation File as an Administrator

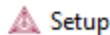
1. Locate the Thermo-Calc installation file on your computer. Right-click the **.exe** file and choose **Run as administrator** to launch the **Setup** program.
2. Accept the **License Agreement** and click **Next**.

## Select the Custom Installation Type

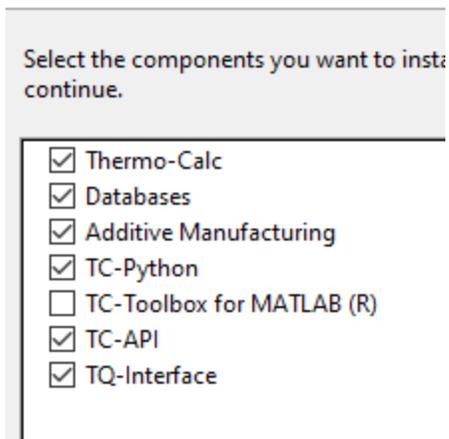
1. In the **Select Installation Mode** window click **Custom** (the option appears greyed out, but it is available to select). Click **Next**.



2. On the **Specify Configuration** window the **Create Shortcut on Desktop** checkbox is selected by default. Click **Next**.
3. In the **Select Components** window, click to select the **TC-Toolbox for MATLAB** checkbox.



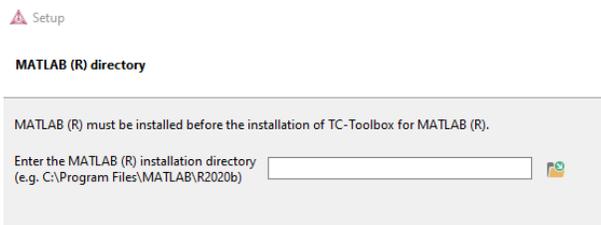
### Select Components





Remember, you must have MATLAB® already installed and may need to know the file path to the MATLAB® folder. If the **TC-Toolbox for MATLAB** checkbox is NOT selected, this confirms that the Thermo-Calc installer did NOT locate the MATLAB® installation (or it is not installed yet).

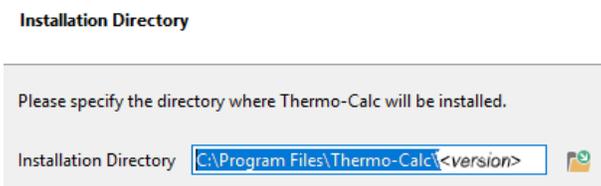
4. Click **Next**.
5. If you selected the **TC-Toolbox for MATLAB** checkbox, the **MATLAB directory** window opens.



- If MATLAB® is already installed but the Thermo-Calc installer could not find the path, then enter the file path to the MATLAB® installation directory or click the file button  to **Browse for Folder** and navigate to the directory. Click **Next**.
- If MATLAB® is NOT installed, click **Cancel** to exit the Thermo-Calc installer program. Once MATLAB® is installed you can return to this process.

## Set the Thermo-Calc Installation Directory and Choose the Setup Type

1. In the **Installation Directory** window, either accept the default path or enter a path to where you want to install the Thermo-Calc software. Click the file button  to **Browse for Folder** and navigate to another directory.



2. In the **Choose Setup Type** window, click **Standalone** or **Network Client**.

**Choose Setup Type**

Select one of the following setup types.  
See installation guide for detailed description of the choices.

Standalone  
 Choose a license file  

Network Client  
 Enter name or IP number of the licensing server

- For a Standalone installation: In the **Choose a license file** field enter the path to your license file (called **lserverc**).
- For a Network installation: Enter the details as required into the **Enter the name or IP number of the licensing server** field. Click **Next**.



If the **TC-Python** checkbox is also selected to be installed (which it is by default), the **Install TC-Python** window is a reminder that additional installation procedures are required after Thermo-Calc is installed.

## Install and Start Using Thermo-Calc as an Add-on in MATLAB®

1. In the **Ready to Install** window click **Next**.



If MATLAB® is not installed (and you selected the TC-Toolbox checkbox) a message displays to confirm that TC-Toolbox will not be installed until you complete the MATLAB® installation. If MATLAB is installed, click **Back** to confirm that the installation path is correct, otherwise **Cancel** this Thermo-Calc installation, install MATLAB and start again.

2. Click **Finish**.

## Test the Installation



[Testing a TC-Toolbox Installation](#)

---

## Testing a TC-Toolbox Installation

---

Generally you can check the installation by first looking in the **Home** tab to see if the program is in the menu. Then proceed to running the diagnostic script or another example.

To check the installation, in MATLAB®:

1. From the **Add-Ons** menu, select **Manage Add-Ons**.

If TC-Toolbox is in the list it is likely it installed correctly but it is recommended you run the diagnostic script or another example to confirm.

2. Run the diagnostics script from the folder `C:\Users\ or open and run any other example to test.`



The TC-Toolbox diagnostic script is also available from our [website](#).

## TC-Toolbox Installation Troubleshooting

---

Below are some suggestions to help troubleshoot a TC-Toolbox for MATLAB® installation.

### Diagnostic Script

A diagnostic script is available to help you if you have problems running TC-Toolbox.



It is important to run this script EXACTLY the same way as you run your MATLAB® script.

The script automatically prints useful suggestions that, in many cases, will help you to fix the problem yourself. If you are unable to solve the problem, contact us for support.

The script is based on the version of Thermo-Calc installed and named `tc-toolbox-diagnostic-script-2024a.m`. You can find the script in these locations:

- As an example (`diagnostics.m`) located in this folder  
`C:\Users\\Documents\Thermo-Calc\2024a\SDK\TC-Toolbox-MATLAB\Examples\Miscellaneous`
- In the **Troubleshooting** section of the *TC-Toolbox for MATLAB® API Reference Documentation*. Also refer to the **Best Practices** in the same guide to see if there are any other solutions or tips.
- On our [website](#).

### MATLAB® Installation Directory Not Found

For Windows installations of the TC-Toolbox for MATLAB® component, you may get an error message or window asking for the MATLAB® installation directory path.



The most common reason for this is that MATLAB® is not installed. If this is the case, exit the Thermo-Calc installation, install MATLAB® and then run the Thermo-Calc installation program again.

If MATLAB® is installed and Thermo-Calc just did not find the file path during the installation setup process, then enter the path in the field when prompted.

---

## Determining the Licensing Software Version (Network Installs)

Sometimes you may need to know the version of License Manager (Windows) or License Server (Linux) software being used. This can be useful while troubleshooting network installations.

- On *Windows* open a command prompt and go to the directory where `lservnt.exe` is found. For example:

```
>C:\Program Files (x86)\Common Files\SafeNet Sentinel\Sentinel RMS  
License Manager\WinNT
```

Type the command `>lservnt.exe -help` and the version is displayed.

- On *Linux* open a Terminal window and go to the directory where the `lserv` script is saved. For example:

```
>cd opt/Thermo-Calc/<version>/Licensing-files
```

Type the command `>./lserv -help` and the version is displayed.

---

## Uninstalling TC-Toolbox for MATLAB®

---

Uninstallation requires that the user is logged in with administrator rights and Thermo-Calc is installed for this same user. Then the uninstallation is automatically completed as part of the uninstallation of Thermo-Calc. Otherwise a manual step is required.

To uninstall TC-Toolbox (when not automatically uninstalled):

1. Start MATLAB®.
2. In the **Home** tab from the **Add-Ons** menu, select **Manage Add-Ons**.
3. Right-click **TC-Toolbox** and choose **Uninstall**.

---

# Troubleshooting

---



There are diagnostic scripts available for TC-Python and TC-Toolbox for MATLAB® (these are included as examples in the Miscellaneous folder). These scripts are also available on our [website](#).

## In this section:

---

License Errors .....	114
Determining the Current License File Directory Path .....	116
Specifying a New Server Name or IP Number .....	118
Firewalls .....	119
Client/Server Mismatch .....	120
macOS Installations: Running the License Initialization Script .....	122
Linux Installations: Running the License Initialization Script .....	123

## License Errors

<i>Issue</i>	<i>Platform</i>	<i>Installation type</i>	<i>Action or link to more information</i>
How do I add or update a standalone (SUNLL) license on a Windows computer where Thermo-Calc is already installed?	Windows	Standalone	<a href="#">Determining the Current License File Directory Path</a>
How do I add or update a standalone (SUNLL) license on a Linux computer where Thermo-Calc is already installed?	Linux	Standalone	<a href="#">Determining the Current License File Directory Path</a>
How do I add or update a network license on Windows?	Windows	Network	<a href="#">Determining the Current License File Directory Path</a>
How do I add or update a network license on Linux?	Linux	Network	<a href="#">Determining the Current License File Directory Path</a>
No license file found. Copy a valid license file to the installation directory after completing the installation.	All	All	You get this message if the wrong path or no license file is found at a path entered during the installation process. See <a href="#">Installation Default Directories and the License File</a> and <a href="#">Determining the Current License File Directory Path</a>
TC-Toolbox for MATLAB®	Windows	All	<a href="#">TC-Toolbox Installation Troubleshooting</a>
Warning Environment variable "LSHOST" is not set. Software will not function properly.	All	Network	<a href="#">Specifying a New Server Name or IP Number</a>
Problem running post-install step. Installation may not complete correctly. Unable to run post-installation script, run the license initialization script as root after the installation.	Linux and Mac	Standalone	<a href="#">macOS Installations: Running the License Initialization Script</a> or <a href="#">Linux Installations: Running the License Initialization Script</a>

---

<i>Issue</i>	<i>Platform</i>	<i>Installation type</i>	<i>Action or link to more information</i>
Firewall warnings related to license management software (License Manager and License Server)	Windows and Linux	Network	<a href="#">Firewalls</a>
Client / Server naming mismatches	Windows and Linux	Network	<a href="#">Client/Server Mismatch</a>
InstallScript engine missing	Windows	Network	You are not notified if the InstallScript scripting engine is not installed. Install the ISScript.msi file and then run the downloaded file from Thermo-Calc again as described in <a href="#">Windows: About Installing the License Manager</a>

---

## Determining the Current License File Directory Path

---

This topic explains how to find out where your license file is installed, either by default or if a different directory was chosen for previously installed versions. For new installations, where you may not have had a license file when you installed Thermo-Calc, you can also use these instructions to determine the correct file path.

### *From Thermo-Calc*

If you have already installed Thermo-Calc, you can view the path to the license file in the GUI. Select **Help** → **Show License Info**.

In Console Mode you can run the SYS module command `DISPLAY_LICENSE_INFO`.

### **Windows Standalone Installations**

*How do I add or update a standalone (SUNLL) license on a Windows computer where Thermo-Calc is already installed?*

1. Unpack the .zip file that contains your standalone license and save the LSERVRC file on your computer. Find out where to place the new license file by opening a Command Prompt Window (cmd.exe) and type in the command line:

```
>set LSERVRC
```

2. Press `Enter`. The output from the command is the path to the license file (including the name of the license file). For example: if the output is

```
LSERVRC=C:\Users\\Documents\Thermo-Calc\lservrc
```

then the new license file should be placed in `C:\Program Files\Thermo-Calc\2024a\`

### **Linux Standalone Installations**

*How do I add or update a standalone (SUNLL) license on a Linux computer where Thermo-Calc is already installed?*

Find out where to place the new license file by opening a Terminal window and type in the command line:

```
echo $LSERVRC
```

## **Windows Network Installations**

*How do I add or update a network license on Windows?*

Only the WIAdmin tool can be used to update the network license used by the License Manager. See [Install the Thermo-Calc Network License Using WIAdmin](#).

## **Linux Network Installations**

*How do I add or update a network license on Linux?*

Replace your current license file with the new license file. The license file is located in the same directory as the license server.

---

## Specifying a New Server Name or IP Number

---

The Thermo-Calc Setup wizard does not warn you if the name or IP number entered to the server (with the license management software) is incorrect.

If the name or IP number is not set at all, or if it is set to an empty string, then when you start Thermo-Calc the following message displays: *Warning Environment variable "LSHOST" is not set. Software will not function properly.*

### Windows

This section explains how to specify or change the name or IP number of the server where the License Manager is installed and for a consolidated network client installation.

To install a license for an existing Thermo-Calc program, follow these steps:

1. Close all running Thermo-Calc programs.
2. Open a command line window.
3. Specify the name or IP number of the licensing software server. Type:

```
set LSHOST=<name/IP number>
```

To display the current name or IP number, type:

```
set LSHOST
```

### macOS

Permanently set the environmental variable LSHOST to specify a new licensing server name or IP number. Refer to the file `/Applications/Thermo-Calc-2024a.app/Contents/Info.plist`.

In this file, you can manually set the `TC24A_HOME` variable, which specifies the home directory of the Thermo-Calc program. See the `Info.plist` file for an example.

### Linux

You can set or change the name or IP number of the license server at a later time by setting the environmental variable "LSHOST" to this name or number.

---

## Firewalls

---

### **Windows**

For a Windows License Manager installation you may have to change the system firewall settings to allow the License Manager to communicate with the computers running the client applications. Go to the Windows Firewall settings window to see if there are any problems. If prompted, select Unblock the system firewall for the Sentinel RMS License Manager and click Next.

### **Linux**

For a Linux License Server installation you need to open the UDP port 5093 in your firewall. In case you have installed SELinux you might also need to disable it by setting SELINUX=permissive in the `/etc/selinux/config` file.

---

## Client/Server Mismatch

---

For some network installations there can be a naming mismatch that causes errors. The server (where the license management software is installed) gets information from its DNS servers that an IP address (aaa.bbb.ccc) belongs to a computer (Computer A). But the client computer identifies itself as having a different IP address (aaa.bbb.ccc.eee) for the same computer (Computer A).

Even if the IP address of the client computer belongs to a subnet range listed in the license file, there is a problem with accessing the license file because the computer name does not match the information.

You may need your organisation's network administrator to assist with this issue.

### **How to Check**

#### *Compare computer names on the client computer and server*

1. On the client computer, start a command prompt **cmd.exe** and type `ipconfig /all`. Press `Enter`. A variety of data is displayed on screen. Write down the following information for later comparison:
  - Hostname
  - DNS Suffix
  - Current IPv 4 Address
2. In the command window, type `nslookup <client_computer_name>`, where *client\_computer\_name* is the actual name of the client. Press `Enter`.

This shows what information is saved on the DNS Servers used by the client computer. This information should match what is shown in the output from the first step.

3. On the server that holds the license management software, open a command prompt and run the command line: `nslookup client_computer_name`. The output lists the information for this server.

Does it match the information given by the command `ipconfig /all` on the client computer? If not try the following.

### *Try these options to correct the issue*

1. If the client computer has a dynamic (DHCP) IP address, either change it to a static one or make a reservation on the DHCP Server so that the computer always get the same IP address.
2. Update the records on the DNS Servers used by both the client computer and the server computer.

You can also configure the computer names and IP addresses in the local Hostfile of the client and server:

1. On the client computer, make a backup copy of the file C:\Windows\System32\drivers\etc\hosts.
2. Open the original in a simple text editor.
3. Add a new line which contains the IP and name of the server, for example:  
192.168.29.27 license\_server.thermocalc.se.
4. Save the changes.
5. On the server, make a backup copy of the file /etc/hosts.
6. Open the original in a simple text editor.
7. Add a new line which contains the IP and name of the client, for example:  
192.168.29.47 Computer\_A.thermocalc.se.
8. Save the changes.
9. In the host file, there is an example of how the operating system (Windows or Linux) wants the line in the hosts file to look like.

## macOS Installations: Running the License Initialization Script

---

As part of the Standalone installation of Thermo-Calc, the Setup runs a license initialization script. If this part of the installation process did not complete properly for a standalone installation, you must manually run the license initialization script. You need the administrator username and password for these steps.

### How to Run the Script

For this script to successfully initialize the license, the license file must be called **lservrc** and be located in the folder:

```
/Users/Shared/Thermo-Calc
```



#### [Installation Default Directory: macOS](#)

1. Open a terminal window. The **Terminal** application is in the */Applications/Utilities* folder.
2. At the prompt, enter this command:

```
$ sudo /Applications/Thermo-Calc-  
2024a.app/Contents/Resources/license-initialization-macOS-x86_  
64.sh
```

If you are not already logged in as an administrator, you probably cannot run the `sudo` command. If this is the case, then you can use the `su` command in the terminal window to login as an administrator. Contact your local system administrator for help.

3. Enter the administrator password and press `Enter`.
4. When you restart your computer and run the license initialization script, you are ready to start using Thermo-Calc.



If you did not install a license file during this procedure, you can still start Thermo-Calc although you will get the **No license file found** message.

# Linux Installations: Running the License Initialization Script

As part of a Standalone installation of Thermo-Calc, the Setup runs a license initialization script. If you get a warning message during the standalone installation Unable to run post-installation script, run the license initialization script as root after the installation, then you need to manually run the script as root.

## *openSUSE Administration (sudo) Users*

Note the following if you are a sudo user (administrator) in openSUSE installing Thermo-Calc using this command:

```
sudo ./Thermo-Calc-linux-<full-version-number>.run
```

When this command is used, you will do a text-based installation instead of having a graphical interface to guide you. The information is the same, just without the user interface.



For example, the *<full-version-number>* is not just 2024a, it is what is found in the **Help → About** window in Thermo-Calc and it changes every release. You can find the [current full version number listed on our website](#).

## *How to Run the Script*



For this script to successfully initialize the license, the license file must be called **lservrc** and be located in the directory where you have installed Thermo-Calc.



### Installation Default Directory: Linux

1. In the console, go to the Thermo-Calc installation directory.
  - The license file is put in `/home/<user>/Thermo-Calc` if you are installing as a non-root user, where `<user>` is your user name.
  - If you are installing as root, then this directory is `opt/Thermo-Calc/`.

2. Enter this command:

```
$ sudo sh license-initialization-linux.sh
```

If you are not able to execute the `sudo` command because you are not on the sudoers list, you must log in as a root user with the command `su`, and then use `sh` to run the license initialization script. For example, on a 64-bit computer, enter the following:

```
$ sh license-initialization-linux.sh
```

3. Log out of Thermo-Calc and then log back in to save these changes.

---

## Sentinel RMS License 9.7 Tools

---

These tools relate to Network Client installations of the Sentinel RMS License software.



[Windows: Installing the Sentinel RMS License Manager](#) or [Linux: Installing the Sentinel RMS License Server](#)

### In this section:

---

GetHostID: Get HostID/Mac Address .....	126
Ismon: Monitor License Manager Transactions .....	127
Ispool: Maintain Redundant License Manager Pool .....	128
Isrvdown: Shut Down the License Manager .....	130
Iswhere: Display a List of License Managers .....	131
WlmAdmin Administrative Tool .....	132
Sldemo: Show Basic API Functions .....	136
Wechoid: Get a Computer Locking Code .....	140

## GetHostID: Get HostID/Mac Address

GetHostID is a tool that retrieves the HostID/Mac address from the computer hosting the license server and saves it to a text file.



For Linux the output file is called `gethostid.log`. For Windows it is called `TC_Hostid.txt`.

- This is related to the [Full Network Client Installations](#) using the Sentinel RMS License Manager (Windows) or Server (Linux), version 9.7.
- The tool is applicable to Windows and Linux (32-bit and 64-bit) operating systems. See [Windows: Installing the Sentinel RMS License Manager](#) or [Linux: Installing the Sentinel RMS License Server](#).

### Windows

1. Extract `getHostID_win.zip`.
2. Right-click `getHostID_win.bat` and select **Run as Administrator** from the menu.

The output is saved in a file: `TC_Hostid.txt` located in the same folder as `getHostID_win.bat`.

### Linux 32-bit

1. Extract `gethostid.tar`.
2. Run the command: `> ./gethostid`.

The output is saved in a file: `gethostid.log` located in the same directory as `gethostid`.

### Linux 64-bit

1. Extract `gethostid64.tar`.
2. Run the command: `> ./gethostid64`.

The output is saved in a file: `gethostid.log` located in the same directory as `gethostid64`.

---

## Ismon: Monitor License Manager Transactions

---

- This is related to the [Full Network Client Installations](#) using the Sentinel RMS License Manager (Windows) or Server (Linux), version 9.7.
- The tool is applicable to Windows and Linux (32-bit and 64-bit) operating systems. See [Windows: Installing the Sentinel RMS License Manager](#) or [Linux: Installing the Sentinel RMS License Server](#).

### Platforms

Windows, Linux 32-bit, Linux 64-bit

The tool retrieves information about all features currently licensed by the Sentinel RMS License Manager, licenses, and clients.

On UNIX computers, you can use "Poll-interval-in-seconds" to specify that Ismon should keep monitoring and reporting usage activity instead of displaying information once and stopping. If specified, Ismon waits for that many seconds between re-polls of the License Manager.

Ismon monitors all licensed applications supported by a License Manager. If the License Manager supports licensed applications from multiple vendors, then all licenses for all vendors are displayed.

- On Windows, use `lsmon [Server-host]`
- On Linux 32-bit, use `lsmon [Poll-interval_in_seconds] [Server-host]`
- On Linux 64-bit, use `lsmon64 [Poll-interval_in_seconds] [Server-host]`

### OPTIONS

- *Poll-interval\_in\_seconds*: Enable periodic update at indicated rate. This option is supported on UNIX only).
- *Server-host*: The name of the computer on which the License Manager is running. If Server-host is omitted, Ismon will attempt to talk to the Sentinel RMS License Manager on the computer indicated in the LSHOST environment variable or in the LSHOST (or lshost) file. If the variable or the file does not exist, then it will attempt to contact a License Manager using the broadcast mechanism. If Ismon fails to find a Sentinel RMS License Manager, it will display an error message and exit.

## Ispool: Maintain Redundant License Manager Pool

- This is related to the [Full Network Client Installations](#) using the Sentinel RMS License Manager (Windows) or Server (Linux), version 9.7.
- The tool is applicable to Windows and Linux (32-bit and 64-bit) operating systems. See [Windows: Installing the Sentinel RMS License Manager](#) or [Linux: Installing the Sentinel RMS License Server](#).

Ispool is a redundant License Manager administration utility (command-line) for Windows and UNIX.

### OPTIONS

- **h** Displays a list of Ispool usage options.
- **a** *<license manager>* Adds the License Manager to the redundant License Manager pool. Specify the hostname or IP address of the License Manager system. Modifies the redundant license file.. On Windows, dynamic addition of License Manager in an existing redundant pool is not supported.
- **d** *<license manager>* Deletes the License Manager from the redundant License Manager pool. Specify the hostname or IP address of the License Manager system. Modifies the redundant license file. On Windows, dynamic deletion of License Manager in an existing redundant pool is not supported.
- **l** Displays the host name of the leader redundant License Manager.
- **p** Displays the list of License Managers in the redundant License Manager pool.
- **g** *<feature> <version>* Displays the license token distribution criteria for the specified feature/version. If the version is not specified, it must be replaced with empty quotation marks: " ".
- **L** *<event> <OFF/ON>* Disables/enables logging for an event. The events you select will be logged into the License Manager trace log file (only if error logging is enabled).

### LSPOOL - L OPTIONS

Events that can be logged using the -L option are given below. The following events are temporarily logged (except 1) and need to be set each time the License Manager is restarted.

- 0 Logs all events.
- 1 Logs when the License Manager is up. On by default. 2 Logs election of the leader License Manager.
- 3 Logs the two-way redundant License Manager heartbeat:> Leader License Manager periodically logs the update status received from all its followers  
> Follower License Manager periodically logs the update about the contact established with the current leader License Manager
- 5 Logs the usage information event, shared by the leader, regarding the number of tokens issued. 7 Logs the license token distribution information synchronization event.
- 9 Logs when the License Manager is down.
- 11 Logs addition of a license into the file or memory. Also logs deletion of a license from the memory.

---

## Isrvcdown: Shut Down the License Manager

---

- This is related to the [Full Network Client Installations](#) using the Sentinel RMS License Manager (Windows) or Server (Linux), version 9.7.
- The tool is applicable to Windows and Linux (32-bit and 64-bit) operating systems. See [Windows: Installing the Sentinel RMS License Manager](#) or [Linux: Installing the Sentinel RMS License Server](#).

Isrvcdown utility is used to shut down the named License Manager.

### **Platforms**

Windows, Linux 32-bit, Linux 64-bit

- On Windows and Linux 32-bit use `lsrvcdown host-name`
- On Linux 64-bit use `lsrvcdown64 host-name`

Isrvcdown can be run from the operating system command prompt by an administrative user on a client or License Manager computer.

- On UNIX, only the user logged in as “root” can use Isrvcdown.
- On Windows, any user with administrative privileges may use Isrvcdown unless the License Manager was started by a user logged in as Administrator, in which case only Administrator can shut the License Manager down. The computer on which you are running Isrvcdown must be in the same network domain as the License Manager computer.



The same user must install the Sentinel RMS License Manager and also run this utility.

---

## Iswhere: Display a List of License Managers

---

- This is related to the [Full Network Client Installations](#) using the Sentinel RMS License Manager (Windows) or Server (Linux), version 9.7.
- The tool is applicable to Windows and Linux (32-bit and 64-bit) operating systems. See [Windows: Installing the Sentinel RMS License Manager](#) or [Linux: Installing the Sentinel RMS License Server](#).

### Platforms

Windows, Linux 32-bit, Linux 64-bit

Iswhere command displays the network names of the computers running the License Manager. By default, Iswhere displays for each License Manager the address of the computer on which that License Manager is running as well as its host name.

Iswhere can be run on Windows and UNIX. Iswhere should be run from the operating system command prompt by an administrator or application user on a standalone, client, or License Manager computer.

Iswhere should be run in the following format:

- Windows and Linux 32-bit: `lswhere [option]`
- Linux 64-bit: `lswhere64 [option]`

### OPTIONS

- `d` Displays details on the License Managers found on the network.
- `r` Displays only the IP addresses of the License Managers found on the network (resolves the host names). Iswhere can also be run without specifying any command line options.

---

## WlmAdmin Administrative Tool

---

- This is related to the [Full Network Client Installations](#) using the Sentinel RMS License Manager for Windows, version 9.7.
- The tool is applicable to Windows operating systems. See [Windows: Installing the Sentinel RMS License Manager](#).

WlmAdmin performs a variety of administrative functions for the license management activities. It displays elaborate information that includes the License Manager information, feature information, and license and client information.

### **Platform**

Windows



WlmAdmin can administer a remote Linux license server.

### **WlmAdmin Functions**

- Provides information on the License Managers.
- Details on licenses and clients using them in the network. For example, the administrator can see which users have acquired that license and who are in a queue waiting for the license.

If you are using redundant License Managers, you can use WlmAdmin as an informational tool to:

- Add or delete a redundant License Manager from the redundant License Manager file.
- Display a list of all License Managers in the redundant License Manager pool and display the name of the License Manager leader.
- Display the distribution criteria for specific licenses (only high assurance model is allowed, where the reigning leader server keeps all the license tokens).

---

## View License Manager Information

- Click the name of a License Manager in the left-hand pane of the main window. In the right-hand pane of the display, you see the IP address of the computer containing the License Manager and its host name.

## Add or Remove a License Manager from the Defined Server List

- To add a License Manager: Right-click the name of a License Manager in the list of Subnet Servers. From the menu select **Add to Defined List**.
- To remove a License Manager: Right-click the name of a License Manager in the list of Defined Servers. From the menu select **Remove from Defined List**.

## Add a License Code to the License Manager from a File

1. Right-click the name of a License Manager in the left-hand pane of the main window.
2. From the menu select **Add Feature > From a File > To Server and its File**.
3. To add the license from a file, select **From a File**. This permanently adds the license code to the License Manager license file as well as to the License Manager memory.
4. In the new windows, browse to the source file that contains the license code and click **Open**.
5. Select the license code file.

## Delete Old License Features from the License Manager

1. Right-click the file `wlmAdmin.exe` and select **Run as Administrator**.  
If the license server is not found under subnet servers, go to **Edit > Defined Server List**.
2. Enter the host name or IP address of the license server.
3. Click **Add** and **OK**.
4. Click **View > Refresh All** to update the list.
5. Right-click the name of a License Manager in the left-hand pane of the main window.
6. From the menu select **Remove all Features**

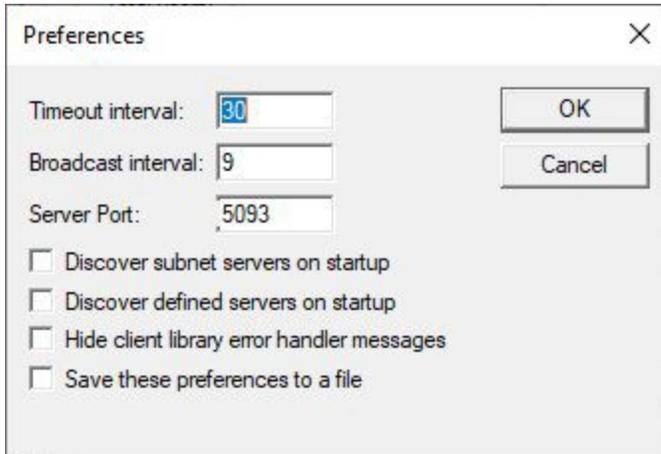
## Shut Down a License Manager

- Right-click the name of a License Manager in the left-hand pane of the main window. From the menu select **Shutdown Server**.

## Setting WlmAdmin Preferences

WlmAdmin allows you to choose what to display when it starts.

From the **Edit** menu, select the **Preferences** command to view WlmAdmin Preferences:



- **Timeout Interval:** The number of seconds after which WlmAdmin will stop trying to contact a License Manager. If you have a large network or a slow one, you might want to increase this number if you find WlmAdmin has difficulty retrieving information from License Managers.
- **Broadcast Interval:** Specify the number of seconds between broadcasts to the network to discover License Managers.
- **Server Port:** To set any custom port (other than the default port 5093) on which the RMS License Manager is running.



Set this port number before you add that License Manager in the list of defined servers.

- **Discover subnet servers on startup:** To automatically discover and display the License Managers on your subnet.

- **Discover defined servers on startup:** To always show the License Managers you have defined.
- **Show client library error handler messages:** To pass license management error messages from Sentinel RMS to WlmAdmin.
- **Save these preferences to a file:** To save your choices.

---

## Sldemo: Show Basic API Functions

---

- This is related to the [Full Network Client Installations](#) using the Sentinel RMS License Manager for Windows, version 9.7.
- The tool is applicable to Windows operating systems. See [Windows: Installing the Sentinel RMS License Manager](#).

sldemo is a Windows-based graphical sample program to show basic API functions. Evaluate it using the steps described here to understand the basic API calls and test the connectivity between client and license server

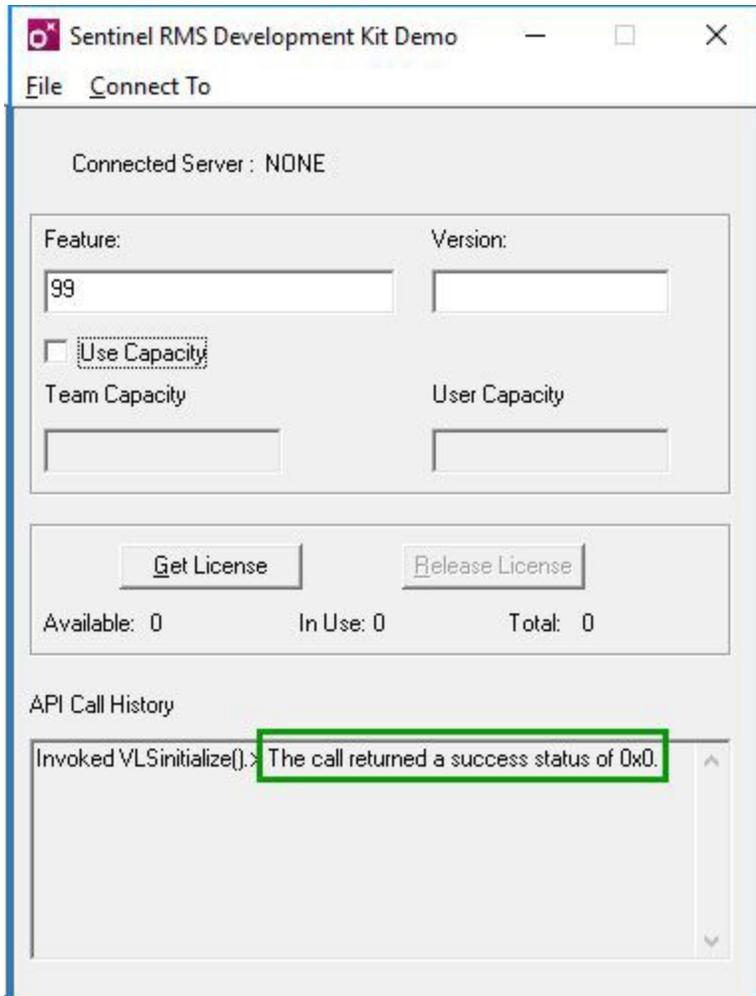


sldemo can be used to test the connectivity to a remote Linux license server

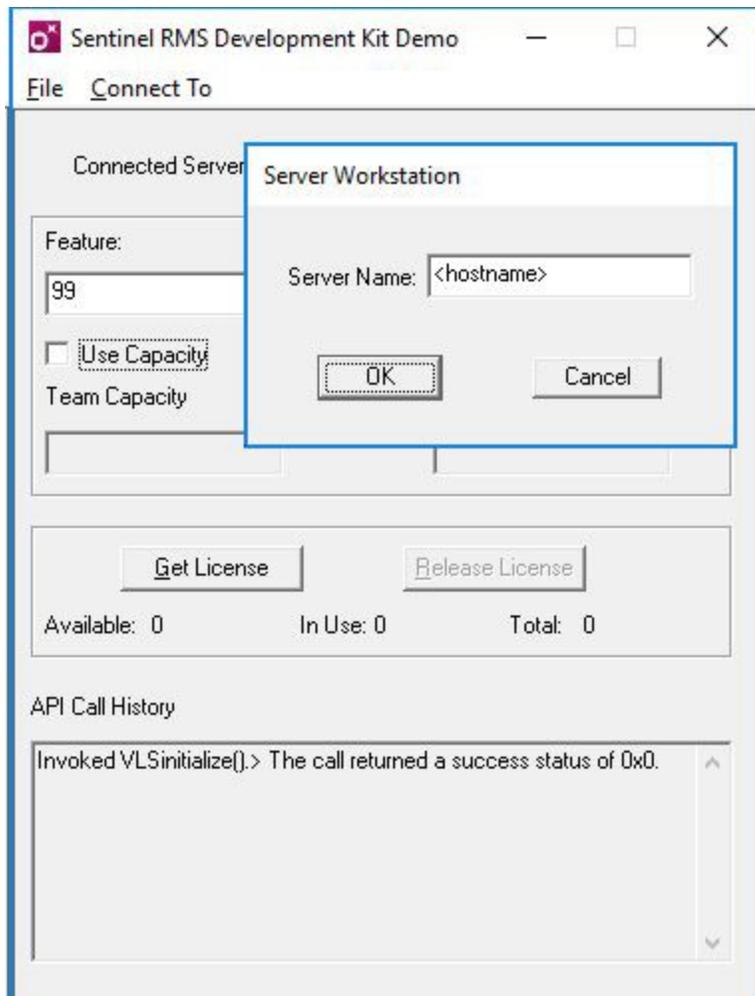
### **Platform**

Windows

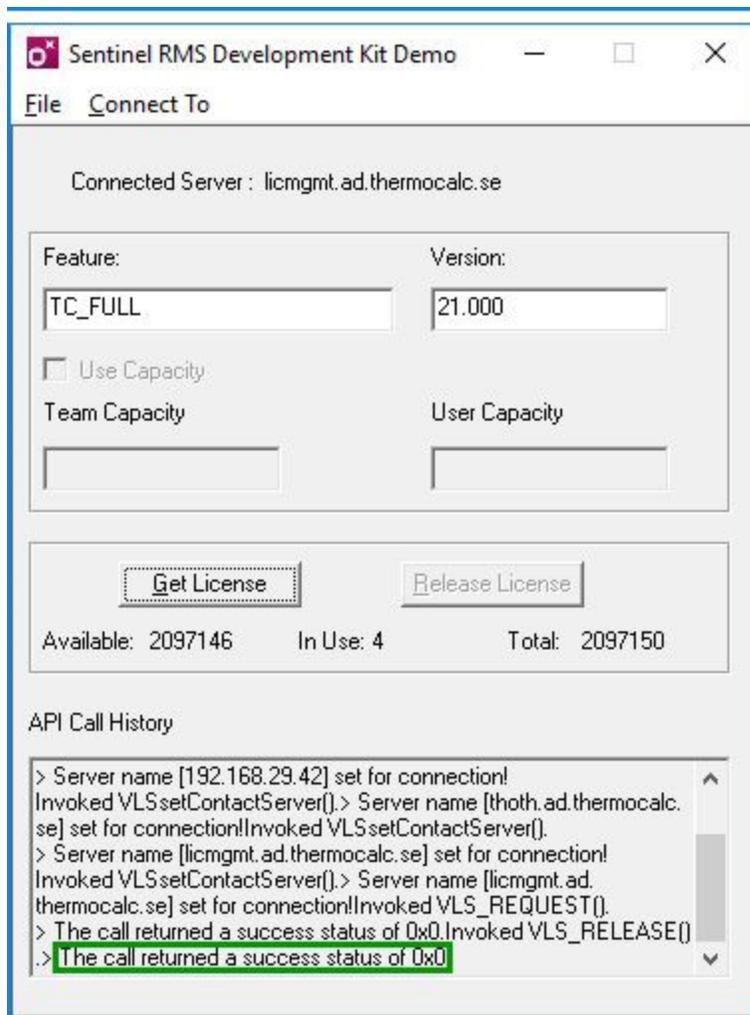
Right-click on the executable and select **Run as administrator**. sldemo calls the [VLSinitialize API](#) when the application is invoked successfully.



Set the contact server (License Manager) under the **Connect To > Server** menu option. Enter a `hostname` , which is the hostname or IP address of the License Server.



Click **OK** and enter in the field one of the license features available in the network license file. Where *Feature* is the name of the feature ex. **TC\_FULL** and *Version* is the license version in the NWL (Network) license. Click **Get License**. The message `The call returned a success status of 0x0` means that the connection client <> license server works.



To save the log, click **File > Save log**.

---

## Wechoid: Get a Computer Locking Code

---

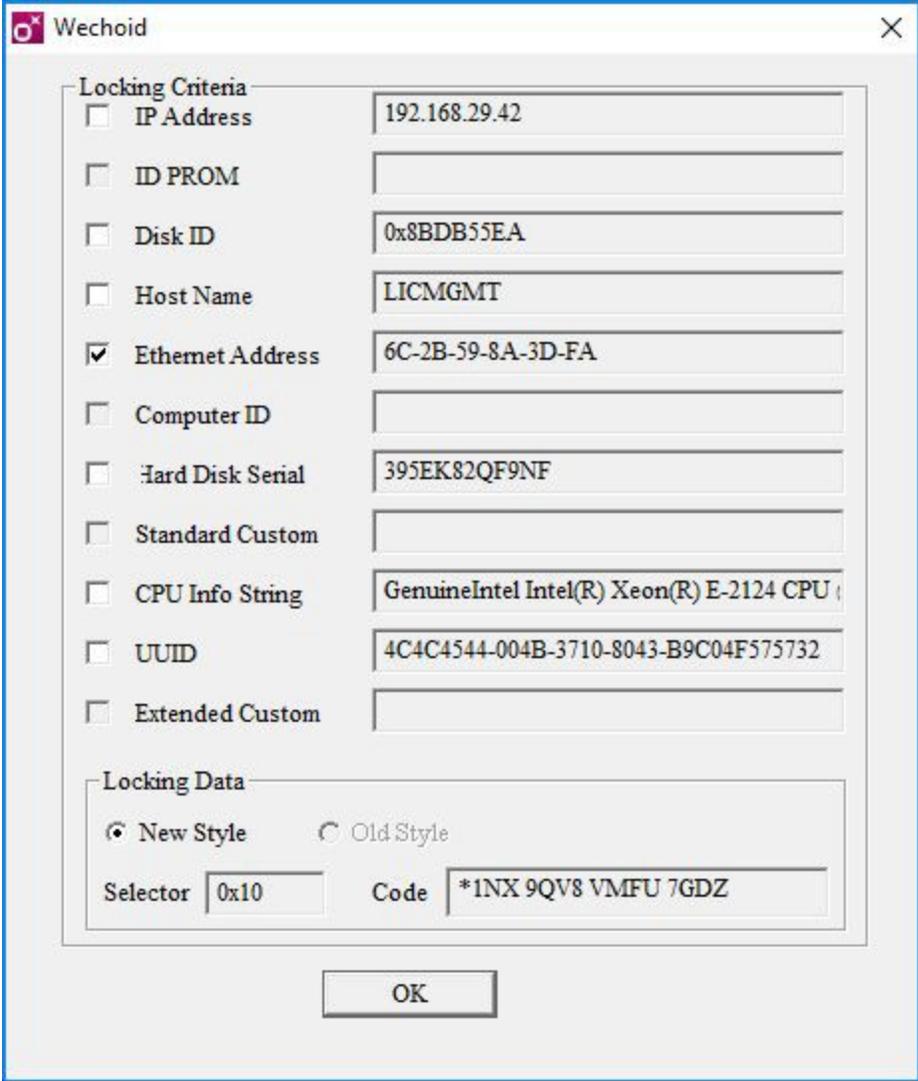
- This is related to the [Full Network Client Installations](#) using the Sentinel RMS License Manager for Windows, version 9.7.
- The tool is applicable to Windows operating systems. See [Windows: Installing the Sentinel RMS License Manager](#).

Before your software vendor creates a license code with any computer restrictions, you must get the locking code for your computer with the Wechoid program. Your software vendor will tell you which computers you must run Wechoid on. The Wechoid utility outputs either one or two hexadecimal locking codes depending upon how your software vendor has configured it. You will send all locking information to your software vendor. Your software vendor will use this information, which is specific to your computers, to generate new license codes.

### ***Platform***

Windows

When launched, the utility shows the available lock criteria (a checkmark is added next to each) and the values. Click to clear (uncheck) all checkboxes except for **Ethernet Address**.



---

# Updating or Uninstalling the Software

---

 Also see [Uninstalling TC-Toolbox for MATLAB®](#)

**In this section:**

---

Updating the Software for Interim Releases .....	143
Uninstalling Thermo-Calc .....	149

## Updating the Software for Interim Releases

---

The full version of Thermo-Calc software is released twice a year. The software releases are named after the year of the release followed by an “a” for the first release of the year or a “b” for the second release of the year. For example, this release is 2024a.

Occasionally after a full release, an interim release version, an update, is required to fix a serious bug or to address an external issue that occurred after the full release. Software licenses come with a free 12 month Maintenance & Support Subscription, starting from the time of purchase, and this includes these software updates.



Go to our website to confirm the [current distributed software version](#) and determine if there are updates, or to learn more about [Maintenance and Support](#) options.

### Installing an Updated Version of the Software

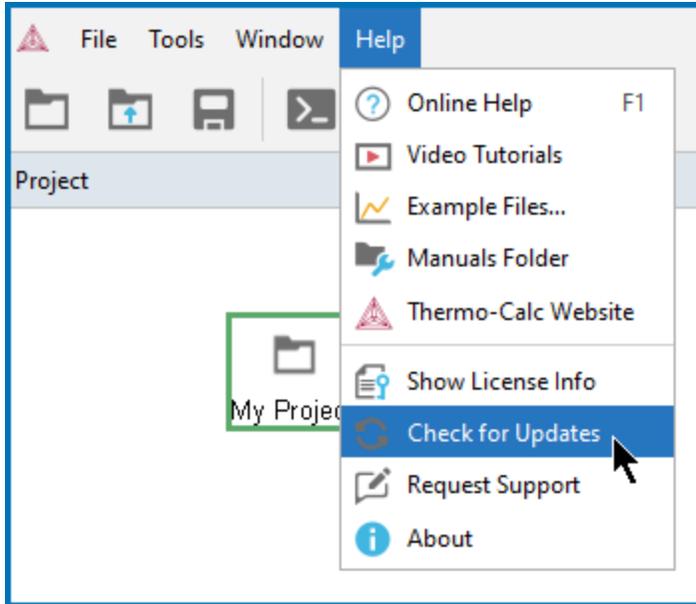
#### Check for a Software Update

You can set your software to automatically check for updates upon start up.

- From within the software on the **Options** window, **General** tab, set **Check update interval** to **On start up**.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.
- If you do not have Automatic Updates on you can go to **Help** →  **Check for Updates** .



## Follow the Installation Instructions for the Applicable OS

After the software has checked for updates and determined there is one, the applicable installation file for your operating system is downloaded. Then follow the instructions provided in the Installation Guide for the specific operating system and license type.

**i** For an update to the software, no additional license files are required.

## Option to Save Backup User Created Files

**i** There are significant operating system differences regarding what backup files are saved if you are installing an update on Windows versus macOS or Linux.

During the installation process, several default folders are added to your system that include documentation, materials, and example files. This is the same for all operating systems. During an update, Thermo-Calc runs a check of these default folders to determine if there are changes or new user files that are saved in these folders. For example, you might have saved a completely new project \*.tcu file into the Examples folder, or after running one of the included examples such as *T\_06\_Serial\_Equilibrium\_Calculators.tcu*, the default file is changed and now includes results.

On Windows operating systems, Thermo-Calc gives you the opportunity after the installation to specifically save a backup of these user created or modified files to your Documents folder when you open Thermo-Calc for the first time after the update.

However, for macOS and Linux operating systems, the behavior is different due to limitations enabling the identification of changes to some files.

Operating System	When does backup occur?	What files are copied?
Windows	After installation, when you open the software for the first time	Documents, Materials, and/or Examples files are copied when you choose <b>Yes</b> at the prompt (see the Windows section for details).
macOS	During installation	There is an option during the installation process to save a backup copy of the Examples folder even if no changes are made to files (see the macOS or Linux sections for details).
Linux		<div style="border: 2px solid red; padding: 10px;">  <p>Materials and Documents folders are not automatically copied even if there are file changes or additions. However, you can always manually back up these folders (see the macOS or Linux sections for details).</p> </div>

## Windows

First complete the installation update. Then when you open Thermo-Calc for the first time, and if Thermo-Calc identifies that there are changed files, you are prompted with a **Software Update - Action Required** window that identifies the folders where the applicable files are changed or added.



These file types are not kept, i.e. these are excluded: dll, exe, lib, mltbx, pdf, whl, html, and htm.

Click **Yes** to save a backup of these changed files or **No** to continue opening Thermo-Calc as normal where the default set of folders are copied from the source `Public Documents` location into a user `Documents` folder.

If you choose to save a backup of the files, the new backup folder is added to this same location, for example:

```
C:\Users\<user>\Documents\Thermo-Calc\2024a_backup
```



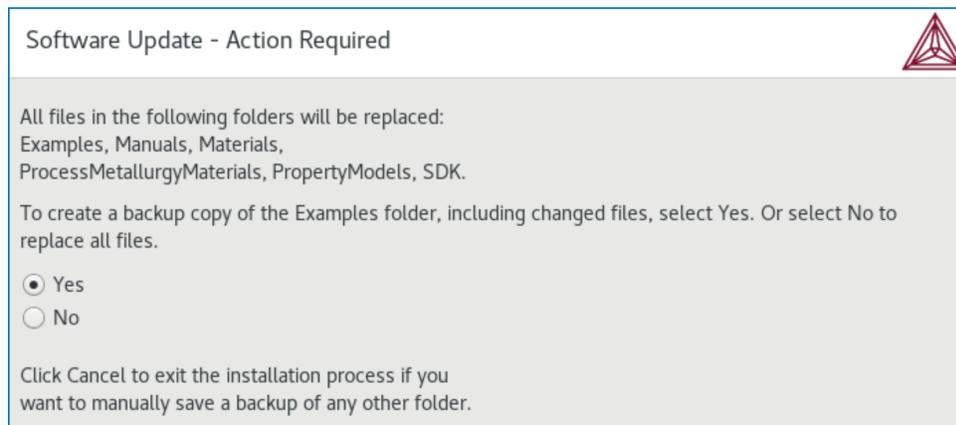
[Installation Default Directory: Windows](#)

## macOS

During the installation process you are prompted with a **Software Update - Action Required** window.



This window always displays whether there are changes or additions to the folders or not.



The Materials and Documentation folders do not automatically have a backup saved. If you think there are user files you want to keep it is recommended that you click **Cancel** at this prompt to exit the installer, then manually copy the folders to another location. Then start the installation update process again.

To create a backup copy of the `Examples` folder, including changed files (i.e. all files are copied, not just the changed or new ones). Options at this prompt are to click:

- **Yes** to create the backup.
- **No** to replace all files.
- **Cancel** to exit the installation process, i.e. to manually save a backup of any other folder.

The default backup folder containing a full copy of all the contents of the `Examples` folder is saved to this location, for example:

```
/Users/Shared/Thermo-Calc/2024a/Examples_backup
```



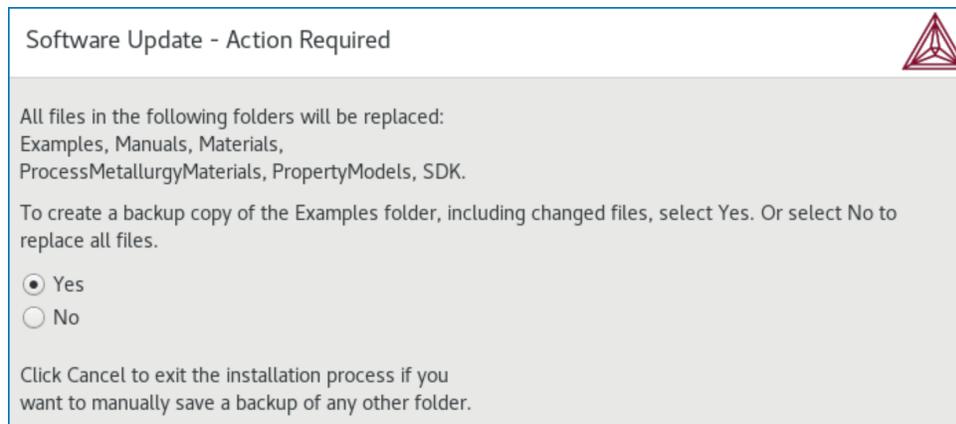
[Installation Default Directory: macOS](#)

## Linux

During the installation process you are prompted with a **Software Update - Action Required** window.



This window always displays whether there are changes or additions to the folders or not.



The Materials and Documentation folders do not automatically have a backup saved. If you think there are user files you want to keep it is recommended that you click **Cancel** at this prompt to exit the installer, then manually copy the folders to another location. Then start the installation update process again.

---

To create a backup copy of the `Examples` folder, including changed files (i.e. all files are copied, not just the changed or new ones). Options at this prompt are to click:

- **Yes** to create the backup.
- **No** to replace all files.
- **Cancel** to exit the installation process, i.e. to manually save a backup of any other folder.

The default backup folder containing a full copy of all the contents of the `Examples` folder for Linux depends if you are a root super user requiring administrative privileges or a common user.

For example, for a common user:

```
/home/<user>/Thermo-Calc/2024a/Examples_backup
```

For example, for a root user:

```
/opt/Thermo-Calc/2024a/Examples_backup
```



[Installation Default Directory: Linux](#)

---

# Uninstalling Thermo-Calc

---



## [Uninstalling TC-Toolbox for MATLAB®](#)

### Windows 10

1. From the **Control Panel** Desktop app, go to **Programs → Programs and Features → Uninstall or change a program**.
2. In the list of programs, locate the version of Thermo-Calc you want to uninstall. Click to highlight that row.
3. Click **Uninstall/Change**.
4. In the **Setup** window, **Uninstallation Mode** dialog box, choose to uninstall the **Entire application** or **Individual components**. Follow the instructions to finish.

### macOS

Deleting the program from the Applications folder does not remove the program from your operating system. Run the **Thermo-Calc-*<version>*-uninstaller** program located in the Applications folder.

1. Go to the **Applications** folder and double click the file that starts with **Thermo-Calc-*<version>*-uninstaller**. Enter the administrator's user name and password.
2. Click **Yes** to choose **Entire application** or select individual components. Otherwise click **No** to quit.
3. The **Uninstalling Thermo-Calc** window shows the progress. Click **OK** when prompted.

### Linux

1. To start the uninstallation procedure, in the Thermo-Calc installation directory use the command

```
./Thermo-Calc-2024a-uninstaller.run
```

For *Thin Client* installations, use the command

```
./Thermo-Calc-client-2024a-uninstaller.run
```

2. To remove the whole application, including all its components, select **Entire application** and click **Forward**, or to remove specific components, select **Individual components** and click **Forward**.



If you intend to reinstall Thermo-Calc 2024a it is important to (1) Restart the computer before you reinstall the software and (2) remove the environmental variable TC24A\_HOME.

# **The Thermo-Calc Software User Guide**

**Thermo-Calc Version 2024a**



---

# About the Thermo-Calc Software

---

**In this section:**

---

The Thermo-Calc Software .....	3
About the Thermo-Calc Add-on Modules .....	4
Graphical Mode vs Console Mode .....	8
Graphical Mode Activities vs Console Mode Modules .....	10
Graphical Mode vs Console Mode: Calculation Types .....	13
About the File Types and Formats .....	18
Displaying Thermo-Calc License Information .....	19

## The Thermo-Calc Software

---

Thermo-Calc Software develops computational tools used to predict and understand materials properties, allowing you to generate computational materials data without costly, time-consuming experiments or estimations based on the limited data available.

Thermo-Calc can be used to fill the gaps in material property data and make predictions of material behavior throughout the materials life cycle. With Thermo-Calc, you can make better decisions about your products and improve your material processing conditions with accurate, reliable materials data.

The Thermo-Calc software (also referred to as the *Thermo-Calc program* or *the software* to distinguish it from the company name) is used to perform thermodynamic calculations. It can calculate complex homogeneous and heterogeneous phase equilibria, and then plot the results as property diagrams and phase diagrams.

The software fully supports stoichiometric and non-ideal solution models and databases. These models and databases can be used to make calculations on a large variety of materials such as steels, alloys, slags, salts, ceramics, solders, polymers, subcritical aqueous solutions, supercritical electrolyte solutions, non-ideal gases, and hydrothermal fluids or organic substances. The calculations take into account a wide range of temperature, pressure, and compositions conditions.



There are several resources available on our website to help you learn how to use Thermo-Calc and other Add-on Modules. Go to the [Getting Started Guides](#) page and choose one of the guides that provides new users an introduction to setting up simple calculations in Graphical Mode. The information there is also in this help documentation.

---

## About the Thermo-Calc Add-on Modules

---

Thermo-Calc Software offers several Add-on Modules that allow users to extend the functionality of the software in your area of specialization. All Add-on Modules are built into the Thermo-Calc platform, so there is no need for additional installation. This also creates a unified workflow and allows data to move seamlessly between the modules.



Go to the [Add-On Modules page](#) on our website for more information about license requirements, which is also mentioned throughout the documentation.

### Additive Manufacturing (AM) Module

The Additive Manufacturing (AM) Module (also referred to as the AM Module) is an Add-on Module to Thermo-Calc and it is available in Graphical Mode as the AM Calculator. The aim of the Additive Manufacturing Module is to better understand the laser powder bed fusion (LPBF) process by predicting the temperature distribution and melt pool geometry as a function of process parameters and using material properties from the Thermo-Calc thermodynamic and properties databases. Special focus is to have a unified treatment of alloy-dependent material properties and process parameters when solving the multiphysics problem of a moving heat source that melts and solidifies metal powder. The multiphysics simulation involves thermal conduction, fluid flow, evaporation-, radiation-, and convective-heat loss.

The Module can simulate the transition from conduction- to keyhole-mode. With experimental melt pool data you can calibrate the heat source. The calibrated heat source can be used to predict additional process conditions and/or to construct process maps (through the TC-Python Software Development Kit).

Furthermore, once you have the temperature distribution, for instance as a function of time and space, you can also use this information for other Thermo-Calc Add-on-Module simulations such as with the Diffusion Module (DICTRA) or the Precipitation Module (TC-PRISMA), or for input to other external finite element programs.

### Diffusion Module (DICTRA)

The Diffusion Module (DICTRA) is an Add-on Module to Thermo-Calc. It is used for simulation of diffusion controlled transformations in multicomponent systems. The simulation calculations are both time- and space-dependent. The Diffusion Module (DICTRA) is available in both Graphical Mode (as the *Diffusion Calculator*) and Console Mode (as the *DICTRA module*).

The Diffusion Module (DICTRA), which is often just referred to as DICTRA, is ideally suited to solve diffusion simulations that include a moving boundary (*Stefan problems*). The multicomponent diffusion equations in the various regions of a material are solved under the assumption that thermodynamic equilibrium holds locally at all phase interfaces. Simulations are one-dimensional and three different geometries can be performed: planar, cylindrical, and spherical.

Examples of cases that have been simulated using the Diffusion Module (DICTRA) include:

- Microsegregation during solidification
- Homogenization of alloys
- Growth/dissolution of carbides, nitrides and intermetallic phases
- Coarsening of precipitate phases
- Interdiffusion in compounds, e.g. coating systems
- Austenite to ferrite transformations in steel
- Carburization, nitriding and carbonitriding of high-temperature alloys and steels
- Post weld heat treatment
- Sintering of cemented-carbides

## Precipitation Module (TC-PRISMA)

The Precipitation Module, or TC-PRISMA, is an Add-on Module to Thermo-Calc and it is available in Graphical Mode as the *Precipitation Calculator*.

The Module treats concurrent nucleation, growth/dissolution and coarsening under arbitrary heat treatment conditions in multi-component and multi-phase systems using Langer-Schwartz theory and the Kampmann-Wagner numerical approach. It is a general computational tool for simulating kinetics of diffusion controlled multi-particle precipitation processes in multicomponent and multiphase alloy systems.

You can use the Precipitation Module for:

- Concurrent nucleation, growth/dissolution and coarsening of precipitates
- Normal grain growth and Zener pinning
- Temporal evolution of particle size distribution
- Average particle radius and number density
- Volume fraction and composition of precipitate
- Nucleation rate and coarsening rate

- Time-Temperature-Precipitation (TTP) diagrams
- Continuous-Cooling-Transformation (CCT) diagrams
- Estimation of multi-component interfacial energy
- Estimation of yield stress using the Yield Strength Property Model

## Process Metallurgy Module

The Process Metallurgy Module is an Add-on Module to Thermo-Calc and it is available in Graphical Mode as the *Process Metallurgy Calculator*. The Add-on Module is designed to model reactions that occur in metallurgical processes. Although primarily used in steelmaking and steel refining processes—such as basic oxygen furnaces, electric arc furnaces, ladle furnace metallurgy, and so forth—applications are not limited to steelmaking.

There are two main branches of calculations possible: **Equilibrium** and **Process simulation**. Equilibrium calculations do not consider any kinetics, the process simulation includes reaction kinetics.

In general, for both types, the main difference compared to using a standard Equilibrium Calculator is that it is easy to handle the different materials present in a metallurgical process. The materials used in the process can be pre-defined, saved and used for the equilibrium calculations. Metallic materials can be defined in element weight percent, oxide materials in weight percent of oxide components, the gas phase in volume percent of gas components and its amount can be defined in normal cubic meters, and so forth. In principle this can be done in a standard Equilibrium Calculator. However, it is much easier when using the Process Metallurgy Calculator as this is designed for this specific purpose.

When using the **Equilibrium** branch of calculations, both isothermal and adiabatic calculations are possible. Adiabatic calculations assume no heat and mass exchange with the environment during the equilibrium reaction, meaning that the temperature changes as a result of exothermal or endothermal reactions taking place.

For a **Process simulation** branch of calculation, the reaction kinetics of the process are considered. This is done by dividing the system into zones. Typically one would have one steel zone containing liquid metal and a slag zone containing liquid oxide (slag). The kinetic model assumes that only a fraction of the steel zone reacts with a fraction of the slag zone per time step. This reacting fraction of the whole system is termed the reaction zone (in literature it has become known as Effective Equilibrium Reaction Zone or EERZ).

You can use the Process Metallurgy Module to calculate the following:

- 
- The equilibrium between custom-defined steels, slags, and gasses.
  - The equilibrium between other metallic and non-metallic phases.
  - Desulfurization, dephosphorization, and decarburization.
  - Any partition coefficient, for example the partitioning of sulfur between the liquid steel and slag phase.
  - Slag characteristics, such as slag basicity or sulfur capacity.
  - The fraction of liquid and solid in the slag.
  - The temperature change in an adiabatic process.
  - Kinetics of the reaction between phases. Typically this is the reaction between a liquid metal and slag phase, but it could also be the reaction between a solid oxide and slag (simulation of refractory wear) or between a solid metal and liquid metal (simulation of dissolution of alloy), etc.

## **Property Model Libraries for Steel and Nickel**

There are specialized Property Models available for those working with steel and nickel materials. Both libraries are available with the Property Model Calculator and the applicable database licenses.

The Steel Model Library includes the following models for those working in the steel industry: Bainite, CCT Diagram, Critical Transformation Temperatures, Ferrite, Martensite Fractions, Martensite Temperatures, Martensitic Steel Strength, Pearlite, and TTT Diagram.

The Nickel Model Library includes the following models specially designed for those working with nickel alloys: Antiphase Boundary Energy, Coarsening, Equilibrium with Freeze-in Temperature, Solvus for Ordered Phase, and Strain-Age Cracking.

## Graphical Mode vs Console Mode

---

There are two interfaces available in Thermo-Calc: *Graphical Mode* with a graphical user interface (GUI) and *Console Mode*, which uses a command line interface. Thermo-Calc and the Diffusion Module (DICTRA) are available in both modes, and all other Add-on Modules are only available in Graphical Mode.

If you are a new user, Graphical Mode is a good way to learn how to work with Thermo-Calc. Depending on what you want to simulate, in some cases Console Mode can have more functionality but until you are comfortable with the concepts, it is recommended you start in Graphical Mode, and for all new Add-on Modules, these are only available in Graphical Mode.



In Graphical Mode calculations are set up, carried out, and visualized as part of a *project*. The steps in the project are performed with *activities*. There are templates and a Wizard available to guide you through the process of defining the project. See [Getting Started Links and Templates](#).

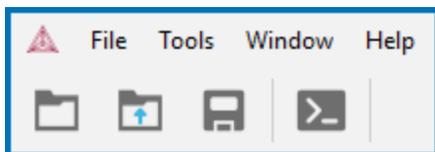


In Console Mode you work with *modules*, which are managed using commands typed at a prompt. Some modules, called *response-driven modules*, ask you a series of questions that typically take you through the process of defining your system, setting calculation conditions, performing calculations and plotting the results.

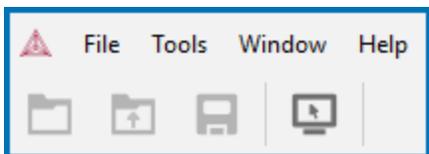
### How to Switch Between Modes

The first time you open Thermo-Calc, it defaults to Graphical Mode. For any future instance it defaults to the last mode used.

Along the top of the GUI is the Graphical Mode Toolbar where you switch between modes.



In Console Mode, except for the **Switch to Graphical Mode** button, the toolbar is not used.



See [Menu, Toolbar, and Keyboard Shortcuts](#) for information about the **Tools**, **Window**, and **Help** menus, which are available in both modes.

## File Extensions

The file extensions are different based on the mode you are in. In Console Mode the extensions are also based on whether you are working with Thermo-Calc or the Diffusion Module (DICTRA).



In Graphical Mode, the Thermo-Calc project files have the extension \*.t<sub>cu</sub>. This includes the examples for the Add-on Modules.



In Console Mode, Thermo-Calc macro files have a \*.TCM extension and the Diffusion Module (DICTRA) file extension is \*.DCM. In addition to macro files in Console Mode there are also *Log* files and *Workspace* files, which also differ.

## Modes Run Simultaneously

The two modes can be run simultaneously, but there is no communication between them. What you do in Graphical Mode does not affect the state of the Console Mode session and vice versa. One exception is the plot settings.



[Global Settings: Graphical and Console Mode Plotting](#)

## Data Optimization and Thermodynamic / Kinetic Assessments



Although many calculations can be done in either mode, data optimization and thermodynamic or kinetic assessments are only available in Console Mode.

## Graphical Mode Activities vs Console Mode Modules

Although many calculations can be done in either Graphical Mode (GM) or Console Mode (CM), only data optimization and thermodynamic or kinetic assessments are available in Console Mode. Conversely, certain Calculators are only available in Graphical Mode as indicated in the table.



[About the Activity Nodes](#) (Graphical Mode) and [Console Mode Modules](#)

### Graphical Mode (GM) versus Console Mode (CM)

GM Activity	CM module (Short Name)	Primary Functions
AM Calculator	–	Run additive manufacturing (AM) simulations using the Add-on Additive Manufacturing (AM) Module. In addition, probe data generated from an AM calculation can be used with the Diffusion Calculator and/or Precipitation Calculator. Additional licenses are required for full access to this Add-on Module and/or the extended use of the probe data.
Binary Calculator	BIN	Calculate binary phase diagrams. Access to specific databases designed for BIN, such as TCBIN, is required.
System Definer	DATA	Select appropriate databases, define chemical systems and retrieve the data required for calculations.
Diffusion Calculator	DICTRA	Run simulations of diffusion controlled transformations in Thermo-Calc using the Diffusion Module (DICTRA). An additional license is required for full access to this Add-on Module.
–	DIC_PARROT	This is to do data optimization for the Diffusion Module (DICTRA) and Precipitation Module (TC-PRISMA). It allows you to assess experimental data and use such data to optimize calculations. An additional license is required for full access to this Add-on Module.
–	ED_EXP	Part of the PARROT submodule, use it to edit experimental data points to make reliable optimizations.
This is the calculation engine	GIBBS or GES	Handles thermodynamic models and quantities. Generally you do not need to use this, but you can get information about models, and list, add or amend the thermodynamic parameters in use.
–	PARROT	Used for data optimization to assess experimental data and use the data to optimize calculations.

<i>GM Activity</i>	<i>CM module (Short Name)</i>	<i>Primary Functions</i>
Precipitation Calculator	–	Set the conditions for, and perform, a precipitation calculation using the Precipitation Module (TC-PRISMA). An additional license is required for full access to this Add-on Module.
Property Model Calculator	–	Predict and optimize material properties based on their chemical composition and temperature. There are preconfigured <b>General Models</b> available for all users and <b>Steel Models</b> and <b>Nickel Models</b> available with additional licenses.
Equilibrium Calculator	POLY	Equilibrium calculation command module. Specify conditions and make calculations.
Material to Material Calculator	–	To calculate materials transitioning from one into the other.
Plot Renderer	POST	This is a submodule to POLY and is for post-processing and plotting of calculation results. Modify property and phase diagrams, add labels, change the diagram colors and so forth.
Ternary Calculator	POTENTIAL	Calculate and plot gas potential diagrams in a ternary system. The potential of two gas species are used as the diagram axes.
–	POURBAIX	Calculate and plot pH-Eh diagrams (Pourbaix diagrams) and property diagrams for heterogeneous interaction systems involving aqueous solutions.
Process Metallurgy Calculator	–	Use this to model with the Process Metallurgy Module and for reactions that occur in metallurgical processes. Although primarily used in steelmaking and steel refining processes (such as basic oxygen furnaces, electric arc furnaces, ladle furnace metallurgy and so forth), applications are not limited to steelmaking. An additional license is required for full access to this Add-on Module.
–	REACTOR	Simulate chemical reaction processes in several feed-forward steady-state stages or in several dynamic stages.
Scheil Calculator	SCHEIL	Simulate and plot Scheil-Gulliver solidification processes with no diffusion in the solid phases and with/without consideration of back diffusion of interstitial components (such as C, N, O, S).
–	SYS	Interact with the operating system and change environmental settings. Create and execute macro files.
–	TAB	Tabulate properties of chemical reactions and substances (stoichiometric or solution phases).

---

<i>GM Activity</i>	<i>CM module (Short Name)</i>	<i>Primary Functions</i>
Ternary Calculator	TERN	Calculate ternary phase diagrams.

## Graphical Mode vs Console Mode: Calculation Types

When you are using Thermo-Calc and the Add-on Modules, you generally first set up a system, select the databases to use, and then define various parameters related to the simulation you want to conduct. Once this is set up you decide what *Calculation Type* to perform, choose a *Plot type* (or a table for single equilibrium, for example) to use and then eventually get a result.

How Thermo-Calc handles the calculation behind the scenes is the same whether you use Graphical Mode (GM) or Console Mode (CM). However, there are differences with the workflow and for historical reasons some calculation types are named differently based on whether you are in one mode or another.



[The Calculation Types in Graphical Mode](#) or [Calculations](#)



For the Diffusion Calculator in Graphical Mode there are no calculation types you need to choose when setting up the simulation.

In addition, the plot types (diagrams) you choose to output the information are also different based on whether you are in Graphical or Console Mode. These plot types are often based on the type of calculation being done, where in some cases you can only do it in one mode and not the other. For example, precipitation calculations are only available with the Precipitation Calculator, GM only, but you can do diffusion calculations in both modes.

### Comparison of Calculation Types in Graphical Mode vs Console Mode

The table lists most of the calculation types that you will want to perform. The first column is primarily based on the calculation types you can select when working in GM and when using the various calculators as indicated in the second column. The third column is the module or command in CM that you would use to perform the same, or similar, type of calculation.

Calculation Type	Graphical (GM)	Console (CM)	Definition
Equilibrium calculations	Equilibrium Calculator	POLY3 module	An equilibrium defines the amount(s) and composition(s) of one or several phases that have the minimum Gibbs energy at a given temperature and pressure. A prerequisite for a successful

Calculation Type	Graphical (GM)	Console (CM)	Definition
			equilibrium calculation is that the number of degrees of freedom according to Gibb's phase rule must be zero. This means that the right number of independent conditions must be set in order to successfully be able to calculate an equilibrium.
Single equilibrium	Property Model Calculator ( <a href="#">Equilibrium model</a> ), Equilibrium Calculator, and Material to Material Calculator	POLY3 module	To calculate a single equilibrium. The results from this calculation are displayed as follows: <ul style="list-style-type: none"> <li>Graphical Mode: In the Visualizations window for a Table Renderer or in the Event Log window (for the Property Model Calculator)</li> <li>Console Mode: With the LIST_EQUILIBRIUM command.</li> </ul>
Single (no axes)	Property Model Calculator and Process Metallurgy Calculator	POLY3 module	To calculate a single point. The results from this calculation are displayed as follows: <ul style="list-style-type: none"> <li>Graphical Mode: In the Visualizations window for a Table Renderer or in the Event Log window.</li> <li>Console Mode: With the LIST_EQUILIBRIUM command.</li> </ul>
One Axis	Property Model Calculator, Equilibrium Calculator, Material to Material Calculator, and Process Metallurgy Calculator	POLY3 module	To vary a quantity on the X-axis.
Property diagrams	One Axis: available with Equilibrium Calculator, Material to Material Calculator, and Property Model Calculator	POLY3 module, STEP command	A property diagram is calculated by stepping one of the conditions that were defined for the equilibrium calculation and plotting some other property against this variable. For example stepping through temperature and plotting equilibrium

<i>Calculation Type</i>	<i>Graphical (GM)</i>	<i>Console (CM)</i>	<i>Definition</i>
			composition of a certain phase, or stepping through composition and plotting equilibrium phase fractions, and so forth.
Phase diagrams	Equilibrium Calculator, Binary Calculator, Material to Material Calculator, and Ternary Calculator	POLY3 module, MAP command, BINARY module, and TERNARY module  The Material to Material Calculator is not available in Console Mode	A phase diagram is mapped within a specific space constructed by two (or more) independent mapping variables. These are mostly temperature and composition, but can be many other, such as temperature and pressure, activity of oxygen and activity of sulfur, and so forth.
Scheil calculations	Scheil Calculator	Scheil module	Simulate a non-equilibrium transformation or partial-equilibrium transformation, also called a Scheil-Gulliver solidification.
Gibbs energy curves	Binary Calculator	BINARY module	To calculate at a constant temperature over the whole composition range.
Activity curves	Binary Calculator	BINARY module	To calculate at a constant temperature over the whole composition range.
Phase fractions	Binary Calculator	BINARY module	To calculate as a function of temperature at a constant composition.
Isothermal section	Ternary Calculator	TERNARY module	To calculate the ternary system for the whole composition range at a constant temperature.
Monovariant lines	Ternary Calculator	TERNARY module	To calculate the variation of the liquid compositions with temperature.
Liquidus projection	Ternary Calculator	TERNARY module	The projection liquid surface and the monovariant lines are calculated over the given temperature range.
Potential diagrams	not available	Potential Diagram module and POLY3	A potential diagram uses the

<i>Calculation Type</i>	<i>Graphical (GM)</i>	<i>Console (CM)</i>	<i>Definition</i>
		module	activities of two major species in the gaseous mixture phase as mapping variables. For example, use this for a metal-oxide/sulphide-gas interaction system to calculate a diagram based on a specific temperature and pressure .
T0 temperature simulations	This is available as the T-Zero Temperature Property Model using the Property Model Calculator	POLY3 module	The T0-temperature is the temperature at which two phases have the same Gibbs energy for a certain composition for a certain alloy composition.
Paraequilibrium	not available	POLY3 module	A paraequilibrium is a partial equilibrium where one interstitial component, such as carbon (C) and nitrogen (N), in a multicomponent alloy can diffuse much faster than the other components (the substitutional elements, including the matrix element and alloying elements).
Grid	Property Model Calculator, Equilibrium Calculator, and Process Metallurgy Calculator	not available	With the Equilibrium Calculator, it computes the equilibrium of each grid point in a two-dimensional grid.  With the Property Model Calculator, it evaluates two axis variables of the selected quantities in the specified range and number of steps.
Min/Max	Property Model Calculator	not available	Evaluates the Property Model(s) for all variations of the selected quantities at the given limits.
Uncertainty	Property Model Calculator and Process Metallurgy Calculator	not available	Evaluates the Property Model(s) where the values of the quantities are sampled from Gaussian distributions.
Aqueous solutions, Pourbaix diagram	Equilibrium Calculator	Pourbaix Diagram module, POLY module	A Pourbaix diagram is a kind of phase diagram that shows the stability boundaries for a metal-aqueous interaction system. The phase boundaries are shown as a function of pH (acidity) and Eh

<i>Calculation Type</i>	<i>Graphical (GM)</i>	<i>Console (CM)</i>	<i>Definition</i>
			(standard hydrogen electronic potential).
Tabulation reaction	not available	TAB module	To tabulate thermodynamic functions of any type of substance, stoichiometric phase or solution phase, or reaction.
Diffusion simulation	Diffusion Calculator	DICTRA monitor	Detailed simulation of diffusion controlled reactions in multicomponent systems.
Isothermal	See below	POLY3, DICTRA monitor	Isothermal calculations in general can be done in all Console Mode modules, for example an Isothermal diffusion in DICTRA monitor and an isothermal calculation in POLY3 using the STEP command.
Isothermal and non-isothermal	Precipitation Calculator	See above	You can calculate nucleation, growth/dissolution and coarsening of particles size classes under these conditions: isothermic (precipitation simulation at constant temperature) or non-isothermic (precipitation simulation with user-defined temperature profile).
TTT and CCT	Precipitation Calculator	not available	The TTT (Time-Temperature-Transformation) and CCT (Continuous Cooling Transformation) diagram can be simulated for one or more particles.
Heat source calibration	AM Calculator	not available	This is available for the Additive Manufacturing (AM) Module for Steady-state simulations, import parameters for a range of experimental melt pool data then store the calibrated heat source for further use.

---

## About the File Types and Formats

---

The file formats (and the file extensions / suffixes) are based on whether you are working in Graphical or Console Mode, and in Console Mode, whether you are working with Thermo-Calc and/or the Diffusion Module (DICTRA).

### **Graphical Mode**

In Graphical Mode you work with one file type, a *project* file, which has a \*.t<sub>cu</sub> file extension. This is applicable to Thermo-Calc and all the Add-on Modules and Property Model Libraries.

### **Console Mode**

In Thermo-Calc Console Mode there are file *types*—**macro** (including log files) and **workspace**—and *extensions* —\*.TCM, \*.POLY3, \*.GES5, \*.PAR, \*.POP, and \*.EXP.



A reminder that Linux and macOS are case sensitive; when applicable ensure you enter file extensions with capital letters.



Workspace files are not currently available with GES6.



[Working with the Console Mode File Types](#)

---

## Displaying Thermo-Calc License Information

---

You can start (and install) the software without a valid Thermo-Calc license but you cannot do any calculations. To show information about the available and installed licenses, from the main menu select **Help** →  **Show License Info**.

This is an example of part of a **License Information** window for a network installation. It only shows the fields you would expect to see as the details are unique for each installation.

### License Information

---

```
License diagnostics program
  build: 19853 created: Mon Apr 08 14:24:19 2019
```

```
Log generated at: 24-04-2019 10:59:13
```

```
Safenet environment variables
```

```
LSHOST:
LSERVRC:
```

```
Local (client) info
```

```
hostname:
license version:
ethernet:    #1
ethernet:    #2
ip address:  #1
ip address:  #2
dns name:    #1
dns name:    #2
```

```
short locking code:
short lock name:
short lock IP-address:
short lock ethernet:
```

### **Network Licenses**

For a *network* license, under each license type with instances checked out, there is a list of who checked out each license.



In Graphical Mode, an instance is checked out when an Equilibrium Calculator activity is created, and checked back in when that Equilibrium Calculator is removed. For the Add-on Diffusion (DICTRA), Precipitation (TC-PRISMA), Process Metallurgy, and/or Additive Manufacturing Modules this also applies to the Diffusion Calculator, Precipitation Calculator, AM Calculator, and Process Metallurgy Calculator, respectively.



In Console Mode, an instance is checked out when you enter the POLY module and is checked back in when you exit the POLY module.

- The **Features** column is a list of the specific software, databases and API licenses purchases.
- The **Version** is an internal version number for the feature.
- The **number of licenses** column is how many instances of each license type available to be checked out simultaneously.
- The **in use** column shows how many license instances are checked out.

### ***Console Mode License Restrictions***

- If you start Thermo-Calc without a valid Thermo-Calc license you cannot leave the SYS module.
- If all the licenses are checked out for a network client installation of Thermo-Calc, you may not be able to enter the POLY or DICTRA modules until a license is available.
- To enter the DICTRA and the DIC\_PARROT modules, you need a valid Diffusion Module (DICTRA) license key.
- If you enter and use either the DIC\_PARROT module or the PARROT module, then you cannot enter the other module in the same session. You must close down Thermo-Calc and launch the program again to enter the other data optimization module.

### ***Network Computer License Restriction***

If you have a network computer installation of Thermo-Calc, then you may not be able to do any calculations even if you have access to a valid network license file. This is because others who are part of your network installation may have checked out all allowed instances of the software.

If you are running a network installation of Thermo-Calc, you can see how many instances of the client licenses are currently checked out and how many of them are left. You can also see which client computers have checked out the licenses.

---

## About the Documentation and Examples

---

**In this section:**

---

Examples, Manuals, and Materials Folders .....	23
About the Manuals (Documentation) Folder .....	28
About the Examples .....	34
Help Resources .....	37
Typographical Conventions .....	38

## Examples, Manuals, and Materials Folders

There are several ways to access the examples, manuals (containing PDF documentation), and materials data folders.

<i>Item</i>	<i>Main Menu</i>	<i>My Project Configuration window</i>	<i>System Definer or Material to Material Calculator</i>	<i>From the Windows Start Menu</i>	<i>On your Computer</i>	<i>Website</i>
Manuals (PDF documentation, DATAPLOT EXP files, TC-Python and TC-Toolbox for MATLAB® HTML help)	x	-	-	x	x	x
Software Development Kits (SDKs) (TC-Python, TC-Toolbox for MATLAB®, TQ-Interface)*	x	-	-	-	x	x
Examples (project and macro files)*	x	x	-	x	x	x
Materials**	-	-	x	x	x	x
Property Models	-	-	-	-	x	-
Process Metallurgy Materials***	-	-	-	-	x	-

\* The SDK folder is installed during installation and available when you have a license for the applicable SDK. There are also differences based on the API as to where examples are installed.

\*\* The Material Properties libraries for the Additive Manufacturing (AM) Module are handled by the software.

\*\*\* The Process Metallurgy Module user-defined material compositions are saved from the Process Metallurgy Calculator and managed from the **Options** tab.

---

## ***From the Main Menu***

Once your Thermo-Calc software is installed you can open the Examples and PDF documentation (*Manuals*) from the main menu:

- To open the Manuals folder select **Help → Manuals Folder**. Note for Linux openSUSE users: This link is not functional. To access the folder, navigate to the default location based on your installation type
- To open the Examples based on whether you are in Console Mode or Graphical Mode, select **File → Examples Files** or **Help → Examples Files**.

## ***My Project Configuration Window***

When in Graphical Mode, click the **My Project** node. In the **Configuration** window under **Getting Started**, click **Example Files**.

## ***Installed Materials Folder***

When you are working with the System Definer or Material to Material Calculator you can open the default **Materials** folder included with your installation.

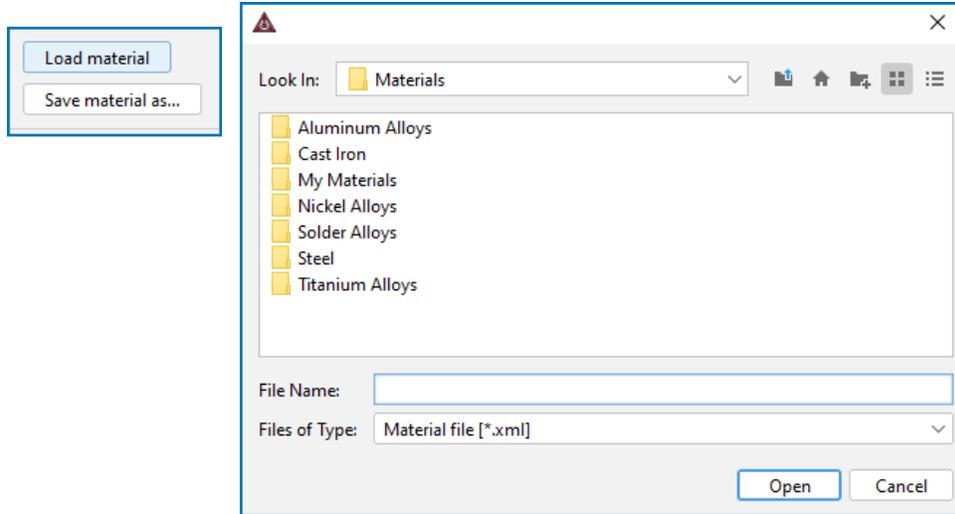
At the bottom of the window under **Material**, click **Load material** to navigate to a material file.



There is a separate folder for materials saved using the Process Metallurgy Calculator. See [Installation Default Directory: Process Metallurgy Materials](#).



For the Additive Manufacturing (AM) Module, the Material Properties library is managed by the software.



The default directory where these material files are stored is based on operating system:

<i>Operating System</i>	<i>Materials Installation Directory</i>
Windows	C:\Users\ <user&gt;\documents\thermo-calc\2024a\materials< td=""> </user&gt;\documents\thermo-calc\2024a\materials<>
Linux	On Linux all user files are installed where the program is installed. If installed as root the default is: /opt/Thermo-Calc/2024a/Materials
macOS	/Users/Shared/Thermo-Calc/2024a/Materials

### ***Start Menu (Windows Only)***

On Windows, once Thermo-Calc is installed, you can locate the Examples and Materials folders, plus all the Manuals. Go to the Start menu and in the search field enter **Thermo-Calc**. A list of the installed versions is provided or click the **Apps** or other tabs to browse search results.

### ***On Your Computer (All Operating Systems)***

The PDFs, example files and materials xml files are included with your installation in a folder location based on the operating system.

In the table, <user> stands for the username and <version> for the version of Thermo-Calc, for example 2024a.

## Windows File Paths

<i>User Type</i>	<i>Default Directory</i>
Administrator	<p><b>Software</b> Program Files\Thermo-Calc\&lt;version&gt;</p> <p><b>Examples, Manuals, Materials, Property Models, Process Metallurgy Materials and SDK folders:</b> Users\Documents\Thermo- Calc\&lt;version&gt;</p>

## macOS File Paths

<i>User Type</i>	<i>Default Directory</i>
Administrator (user name and password required)	<p><b>Examples, Manuals, Materials, Property Models, Process Metallurgy Materials and SDK folders:</b> /Users/Shared/Thermo-Calc/&lt;version&gt;</p> <p>To go to this folder, in Finder, from the <b>Go</b> main menu select <b>Go to folder</b>. Enter the above file path and click <b>Go</b>.</p>

## Linux File Paths

<i>User Type</i>	<i>Default Directory</i>
Non root user	home/<user>/Thermo-Calc/<version>
Root user	usr/local/Thermo-Calc/<version>

---

## Website



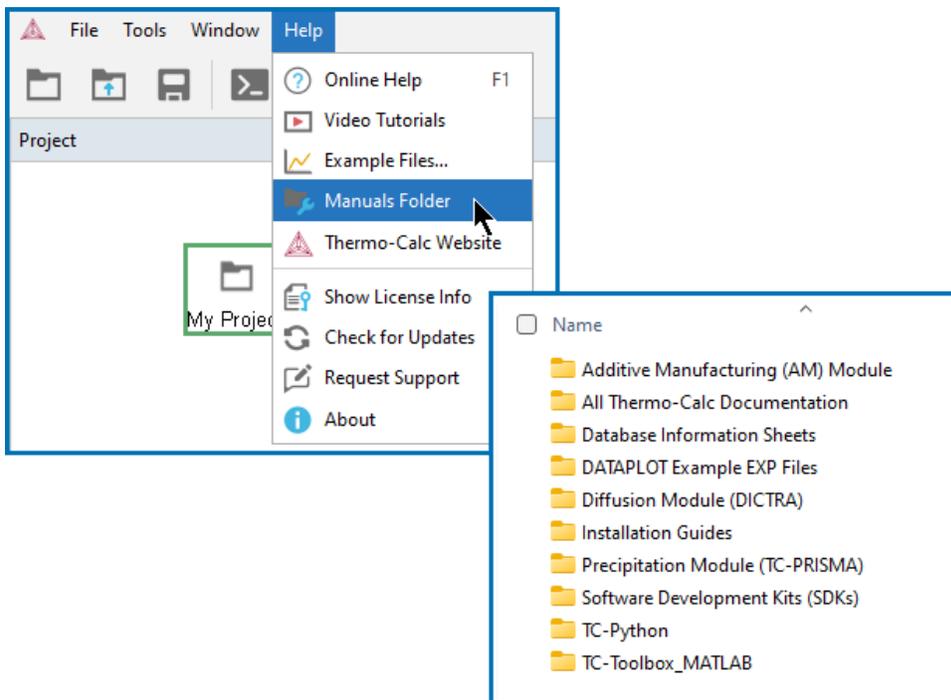
Many resources are available from our website. On the [Support → Documentation](#) page you can download the current and archived PDF documentation and examples, or navigate to the TC-Python and TC-Toolbox for MATLAB® help and diagnostics tools. For current technical information and examples collections for the databases, you can access these by database starting on the [Products → Databases](#) page.

## About the Manuals (Documentation) Folder

There is documentation available in PDF format from the **Help** → **Manuals** folder as part of your installation. Except for two of the SDKs, this is the same information as is contained in the Thermo-Calc online help, just in a different format. The contents of each folder are described below.



It is recommended you search the help (press F1 in Thermo-Calc) if you cannot find what you are looking for in a particular folder or PDF.



A *Documentation Set* contains a variety of user guides packaged together in one PDF. For the Process Metallurgy Module and specialized Property Models (Steel Model Library and Nickel Model Library), all the content is contained in the *Thermo-Calc Documentation Set* or online help.

## Additive Manufacturing (AM) Module

The *Additive Manufacturing Module User Guide* is a summary PDF for the functionality of the Additive Manufacturing (AM) Module and its AM Calculator. This is a highly specialized product and it is interconnected to Thermo-Calc features such as the Scheil Calculator, Diffusion Calculator, and Precipitation Calculator. It requires additional licenses for some functionality. The user guide is not comprehensive. Use the *Thermo-Calc Documentation Set* or online help for all other content.

## All Thermo-Calc Documentation

The *Thermo-Calc Documentation Set* is one large PDF that includes the information needed to work with the Thermo-Calc software. By having all content in one PDF it enables you to search the content just like you would when in the online help (with some exceptions). For example, this PDF does not include specific database or other specialized content that can require additional licenses.

The Thermo-Calc Documentation Set includes the following.

### *Thermo-Calc Installation Guides*

Includes all installation instructions for all platforms and license types.

### *Graphical Mode User Guide*

Includes details about all features and calculators included with Graphical Mode. For example, it includes details about calculators available with add-on products such as the AM Calculator, Diffusion Calculator, Precipitation Calculator, Process Metallurgy Calculator, and all the Steel Model Library and Nickel Model Library Property Model content. For the Add-on Diffusion Module (DICTRA) and Precipitation Module (TC-PRISMA), additional information is included in the separate documentation.

### *Console Mode User Guide*

Includes details about the functionality and features available with the Console Mode version of Thermo-Calc.

For example, the following is only in Console Mode and the associated documentation is included in the respective documentation.

- The PARROT module optimization content is described in the *Data Optimization User Guide*.
- If you develop your own custom databases, this is described in the *Database Manager User Guide*. There is some other generally related documentation that is found under General Reference.
- If you use the old DATAPLOT functionality for plotting, this is included in the *DATAPLOT User Guide*.
- The POURBAIX module is only in Console Mode and there is the *Pourbaix Diagrams Educational Material* guide included.
- A separate *Thermo-Calc Console Mode Example Macros* PDF is available in the folder but NOT included in the Documentation Set due to its file size and page length.

### *Thermo-Calc General Reference*

This is content that spans some of the general topics applicable to multiple applications within Thermo-Calc, i.e. it might be for both Console Mode and Graphical Mode, or needed for a specific database calculation in Thermo-Calc (i.e. model information, variables, and functions for thermophysical properties).

- **Parameters, Functions, and Variables:** Information applicable to both Console Mode and Graphical Mode with respect to understanding the format these are to be written in for Thermo-Calc to use the parameter, function, or variable. However, many of the sections are specific to Console Mode where the explicit use of these terms is more often used.
- **Thermophysical Properties Data:** This includes the model descriptions and data parameters you would need in combination with a database that has the relevant thermophysical property.
- **Database Files and Database Checker:** Information about the database initialization file, differences between Thermo-Calc encrypted and user-defined unencrypted databases, about cached database files, and more. Some content is more specific to those developing their own databases, other information is intended for advanced users, for example, the Database Checker information is a supplement to the *Database Manager User Guide*.
- **Global Settings:** In the **Options** window you can globally set general settings for Graphical Mode as well as default settings for any new activities for both Graphical Mode and Console Mode.

---

## Database Information Sheets (Technical Information)

The *Technical Information* content available as a PDF is also included in the online help system and contains details such as the elements, systems, phases, and properties available with a specific database. It includes the revision history for the database.



Starting with Thermo-Calc 2020b, databases are transitioning from the *Extended Info* PDF, which can include examples, to individual *Examples Collections*. The *Examples Collections* are only available on the website or in the help.

All *Technical Info* and *Examples Collection* PDFs for the **current version** of the database are available for download from the Thermo-Calc [website](#).

## DATAPLOT Example EXP Files

The examples are available as supplementary information as described in the *DATAPLOT User Guide*, which you can find in either the online help, *Thermo-Calc Documentation Set* or *Diffusion Module (DICTRA) Documentation Set*.

## Diffusion Module (DICTRA)

The *Diffusion Module Documentation Set* is one large PDF that includes the information needed to work with the Add-on Diffusion Module (DICTRA) and Thermo-Calc software in Graphical Mode and Console Mode. By having all content in one PDF it enables you to search the content just like you would when in the online help (with some exceptions). For example, this PDF does not include specific database or other specialized content that can require additional licenses.

The Documentation Set includes:

- All the content available in the *Thermo-Calc Documentation Set* and described above.
- Diffusion Module (DICTRA) *Quick Start Guide - Graphical Mode* and *Quick Start Guide -Console Mode*.
- *Diffusion Module (DICTRA) Command Reference* (for Console Mode only)

Also available separately in this folder:

- *Diffusion Module Quick Guide - Console Mode*
- *Diffusion Module Quick Guide - Graphical Mode*
- The *Diffusion Module (DICTRA) Console Mode Example Macros* PDF is available in the folder but NOT included in the Documentation Set due to its file size and page length.

## Installation Guides

Includes the *Quick Install* guides for Standalone (SUNLL) Windows and macOS systems, instructions for TC-Python and TC-Toolbox for MATLAB®, with the full installation instructions for all platforms and license types.



For convenience, the *TC-Python Installation Guide* is also included as a separate PDF in this folder.

## Precipitation Module (TC-PRISMA)

There is a User Guide in this folder, which includes examples, Precipitation Calculator settings, references, and details about the theoretical models and numerical methods. Use the *Thermo-Calc Documentation Set* or online help for all other content.

## Software Development Kits (SDKs)



Separate TC-Python and TC-Toolbox for MATLAB® documentation is stored in its own folder and each is available from our website. This folder has the PDF version of this content.

- **TC-Python API Programmer Guide:** This guide is the PDF format of the HTML help described below.
- **TQ-Interface Programmer Guide**
- **TC Toolbox for MATLAB® Programmer Guide:** This guide is the PDF format of the HTML help described below.

## TC-Python

This folder contains the TC-Python HTML files. Double-click `Index.html` file to open it. This help is also available on our [website](#). TC-Python includes details about working with custom property models (the Property Model Framework).

## TC Toolbox\_MATLAB

This folder contains the TC Toolbox for MATLAB® HTML files. Double-click `Index.html` file to open it. This help is also available via our [website](#).

## About the Examples

---

To learn more about how to use the software you can open and run the example projects (Graphical Mode) and macro files (Console Mode).

### **Graphical Mode Examples**

Graphical Mode examples are available for Thermo-Calc and the Add-on Modules. There are also examples for the Property Models, including the Steel Model Library and Nickel Model Library. These are in the format of project files (\*.tcu).



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

### **Console Mode Examples**

Console Mode examples are available for Thermo-Calc and the Diffusion Module (DICTRA). The *macro* files are in the format of \*.TCM and \*.DCM, respectively.



There are other file types when you are working in Console Mode. Search the Help for *About the File Formats* or *About the DICTRA File Formats*.

## Examples by Product

### Thermo-Calc Examples

A description of the Thermo-Calc examples, and how to access them, is based on whether you are using Graphical Mode or Console Mode.



[Graphical Mode Examples List by Product](#) or [Thermo-Calc Console Mode Examples](#)

---

## Additive Manufacturing (AM) Module Examples

A description of examples using the AM Calculator, which is only available in Graphical Mode, is included in the Thermo-Calc documentation.



[Additive Manufacturing \(AM\) Module Examples Collection](#)

## Diffusion Module (DICTRA) Examples

A description of Diffusion Module (DICTRA) examples, which are available for both Graphical Mode and Console Mode, is included in both the Thermo-Calc and Diffusion Module (DICTRA) documentation.



For Graphical Mode, see [Diffusion Module \(DICTRA\) Examples Collection](#)



For Console Mode, search the help or go to the Diffusion Module (DICTRA) documentation set.



All the macro files and the outputs are also combined into one file that you can search and read. This is included with your installation as a PDF (go to **Help** → **Manuals folder** → **Diffusion Module (DICTRA)** and open *Diffusion Module Console Mode Example Macros*).

## Precipitation Module (TC-PRISMA)

A description of examples using the Precipitation Calculator, which is only available in Graphical Mode, is included in both the Thermo-Calc and Precipitation Module (TC-PRISMA) documentation.



[Precipitation Module \(TC-PRISMA\) Examples Collection](#)

## Process Metallurgy Module Examples

A description of examples using the Process Metallurgy Calculator, which is only available in Graphical Mode, is included in the Thermo-Calc documentation.



[Process Metallurgy Module Examples Collection](#)

---

## Property Models, including Steel and Nickel Model Library Examples

A description of examples using the Property Model Calculator, which is only available in Graphical Mode, is included in the Thermo-Calc documentation. These examples include General, Steel, and Nickel Models.



[Thermo-Calc General Property Models Examples Collection](#)



[Steel Model Library Examples Collection](#)



[Nickel Model Library Examples Collection](#)

## TC-Python Examples

There are examples for TC-Python available after installing the SDK. A brief description of the TC-Python SDK examples, and how to access these, is available in the Help.

## TQ-Interface Examples

There are examples for the TQ-Interface available. You can also find information by searching the online help.

## TC-Toolbox for MATLAB® Examples

There are included examples with all TC-Toolbox installations. These are the same as the examples included with TC-Python although with a different suffix (*matex*) and prefix (*\*.m*) in the file name. You can read summaries about the examples as described for TC-Python in the online help.



See [Installation Default Directory: TC-Toolbox and the Examples](#) to learn where these are located.

## Help Resources

---

### 'Online' Help: No Internet Required

There are several ways to access online help in a browser. The content opens in a browser but uses local content so you do not need an Internet connection except for links to external websites.

First open Thermo-Calc, then choose one of these options:

- Press <F1> on the keyboard.
- Select **Help** →  **Online Help**.
- On the **My Project Configuration** window, click  **Online Help**.
- At the bottom of any **Configuration** window, click  **Help**.



There are several resources available on our website to help you learn how to use Thermo-Calc and other Add-on Modules. Go to the [Getting Started Guides](#) page and choose one of the guides that provides new users an introduction to setting up simple calculations in Graphical Mode. The information there is also in this help documentation.

### Console Mode Help



Console Mode is for Thermo-Calc and the Diffusion Module (DICTRA).

In Console Mode at the command line prompt, you can access help in these ways:

- For a list of all the available commands in the current module, at the prompt type a question mark (?) and press <Enter>.
- For a description of a specific command, type `Help` followed by the name of the command. You can only get online help about a command related to the current module you are in.
- For general system information type `Information`. Specify the subject or type ? and the available subjects are listed. This subject list is specific to the current module.

## Typographical Conventions

The following typographical conventions are used throughout the documentation, both online and in the PDF documents.

### Text

Convention	Definition
Forward arrow symbol →	The forward arrow symbol → instructs you to select a series of menu items in a specific order. For example, <b>Tools → Options</b> is equivalent to: From the <b>Tools</b> menu, select <b>Options</b> .
<b>Boldface font</b>	A <b>boldface</b> font indicates that the given word(s) are shown that way on a toolbar button or as a menu selection. For example, if you are told to select a menu item in a particular order, such as <b>File → Save Project</b> , or to click <b>Save</b> .
<i>Italic font</i>	An <i>italic</i> font in the body of the text indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or elsewhere in the guide.
COMMAND	For features in Thermo-Calc that use Console Mode (i.e. the command line), this font and all capital letters indicates that this is a Console Mode COMMAND. Examples of how you can use a command are written with code font. For example:  Use DEFINE_ELEMENTS followed by a list of the elements that you want in your system. (To list the elements that are available in your current database, use LIST_DATABASE and choose Elements).
<a href="#">HELP</a>	Text in <a href="#">blue and underline</a> and a page number is a link to another topic in the current or referenced guide. Command names are often also topics. Clicking the link takes you to more detail about a particular command or subject in the PDF or documentation set.
<Enter>	Text with <angle brackets> indicates a keyboard entry. Usually to press <Enter> (or Return) or to use a series of keys such as <Ctrl + S>.
code and code bold	A code font shows a programming code or code example. The code bold font highlights the entry. It is also used for file names or paths to help distinguish it from other text. e.g. <ul style="list-style-type: none"> <li>For Windows users, the documents, materials, examples and other folders that sometimes require additional licenses are installed in C:\Users\Public\Public Documents\Thermo-Calc\&lt;version&gt;."</li> </ul>
click vs press	In general, you <i>click</i> with the mouse to perform an action on the screen (e.g. click <b>Save</b> ) and you <i>press</i> keys on a keyboard to enter a set of commands (e.g. press Ctrl+S).

Convention	Definition
	 <p>When working in Console Mode, you can use keyboard shortcuts. Sometimes a window opens where you have the option to <b>Save</b>, <b>Cancel</b> or <b>Open</b> a file, for example. In these cases the instructions might say <i>click</i> Save, whereas you would need to <i>press</i> the applicable keys to perform the action.</p>

## Icons

Convention	Definition
Important	 <p>Provides important information. It is recommended that you read the text or follow the link.</p>
License Required	 <p>This indicates that additional licenses are required for the feature, database, or example.</p>
Note	 <p>The information can be of use to you. It is recommended that you read the text or follow the link to more information.</p>
Time	 <p>Indicates that something you are going to do will take some time, usually related to running an example.</p>
Tip	 <p>This is general information that can be of use but is not required knowledge.</p>
Read More	 <p>Read more at the link, which may be to our website or another source of general information.</p>

Convention	Definition
See More	 Go to the link or guide to see more general information about the topic being discussed.
Training	 Indicates there may be training opportunities available, usually related to a specific feature.
Link to more information	 Click the link to more information in the content.
Examples	 Go to the example collection to learn more.
Console Mode	 This note relates specifically to something in Console Mode.
Video	 Indicates there is a video tutorial on our website, usually in relation to an example.
Graphical Mode	 This note relates specifically to something in Graphical Mode.
Diffusion Module (DICTRA)	 This indicates that the information relates to the Add-on Diffusion Module (DICTRA).
Precipitation Module (TC-PRISMA)	 This indicates that the information relates to the Add-on Precipitation Module (TC-PRISMA).
Process Metallurgy Module	 This indicates that the information relates to the Add-on Process Metallurgy Module.
Property Models	 This indicates that the information relates to the Property Models. This includes the Steel Model Library and Nickel Model Library as well as the general Models included with all installations.
Additive Manufacturing (AM) Module	 This indicates that the information relates to the Add-on Additive Manufacturing (AM) Module.



# Thermo-Calc User Guide: Graphical Mode

Thermo-Calc Version 2024a

---

# Working in Graphical Mode

---

**In this section:**

---

The Graphical Mode Desktop Layout .....	4
Menu, Toolbar, and Keyboard Shortcuts .....	6
Organizing the Windows on the Desktop .....	10

## The Graphical Mode Desktop Layout

---

Open Thermo-Calc to view the default graphical user interface (GUI) layout for Graphical Mode.

The windows are:

1. **Project:** Create, make changes, and navigate between the activities that make up a project. Click the **My Project** node to access the Quick Start wizard and a variety of templates to create projects for all the calculator types.
2. **Configuration:** Shows the settings that can be configured for the currently selected activity.
3. **Visualizations:** Shows the results of a calculation, either plotted as a diagram or displayed in table format. For some types of calculations, there are also previews of settings shown in this window. In this example, there is only a Plot Renderer *tab* in the Visualizations window.



Previews in the **Visualizations** window are applicable to some of the Add-on Modules, i.e. for the Diffusion Calculator, Precipitation Calculator, and AM Calculator. The information is detailed in the relevant sections of the documentation.

4. **Scheduler:** Displays information about jobs, such as calculations, that are being performed or are scheduled to be performed. You can cancel scheduled jobs and if a job has failed, then you can view information about the error.
5. **Event Log:** By default this window is closed but it displays during calculation processes. You can always open it to view progress of calculations and to troubleshoot.

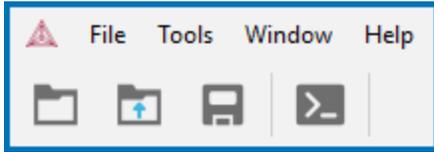
In this example, the **Visualizations** window also has a **Plot Renderer** tab. You can also have a **Table Renderer** tab and when working with calculations that have specialized settings, these tabs are also included and match the names of the nodes in the

## Project window.

The screenshot displays the Thermo-Calc software interface with several key components:

- Project Window (1):** Shows a hierarchical tree of the project structure, including Phase Transition, System Definer 1, Property Model Calculator 1, Equilibrium Calculator 1, and Plot Renderers 1 and 2.
- Configuration Window (2):** Configures the Equilibrium Calculator 1. It includes:
  - Conditions:** Composition unit (Mass: percent).
  - Condition Definitions:** Temperature (Celsius, 1000.0), Pressure (Pascal, 100000.0), System size (Mole, 1.0), and various composition values (Fe: 80.52, Ni: 1.4, C: 0.08, Mn: 1.0, Cr: 17.0).
  - Calculation Type:** Single equilibrium, One axis (selected), Grid, and Phase diagram.
  - Axis Definitions:** Quantity (Temperature), Min (400.0), Max (2000.0), Step division (50.0), Type (Linear - min no.).
  - Event Log:** A log of system events and calculations, such as "GobblerTask: Gobblid: 14:27:22,946 [pool-46-thread-1] INFO CalculationEngine: Testing POLY result by global minimization procedure".
- Visualizations (3):** A phase diagram plot showing the "Amount of all phases [mol]" versus "Temperature [°C]". The plot includes curves for BCC\_A2, FCC\_A1, LIQUID, M3C2, SKGMA, and BCC\_A2F2.
- Scheduler (4):** Shows the execution schedule for the project components.
- Plot Renderer (5):** A small icon representing the plot rendering process.

## Menu, Toolbar, and Keyboard Shortcuts



The **File** menu and toolbar are reserved for Graphical Mode but in Console Mode the **Tools**, **Window**, and **Help** menus have the same options.



For Mac users, the **Tools** menu has only one item. The global **Options** window is opened by selecting **Thermo-Calc <version> → Preferences** or press  $\langle \text{⌘} \rangle$  on the keyboard.



In Console Mode you use commands to access the other options, for example see [Main Menu Commands](#).

### Graphical Mode Menu and Toolbar Options

Button or Icon	Action	Description	Action (s)
	New Project	Create a new project. Also see <a href="#">Creating a Project from a Template</a> .	<ul style="list-style-type: none"> <li>Click the <b>New</b> button  on the toolbar</li> <li>Select <b>File → New Project</b></li> <li>Windows: Press <math>\langle \text{Ctrl+N} \rangle</math></li> <li>Mac: Press <math>\langle \text{⌘} + \text{N} \rangle</math></li> </ul>
	Open Project	Open an existing project. See <a href="#">Opening Project Files</a> .	<ul style="list-style-type: none"> <li>Click the <b>Open</b> button  on the toolbar</li> <li>Select <b>File → Open Project</b></li> <li>Windows: Press <math>\langle \text{Ctrl+O} \rangle</math>.</li> <li>Mac: Press <math>\langle \text{⌘} + \text{O} \rangle</math></li> </ul>

Button or Icon	Action	Description	Action (s)
--	Open Recent Project	Open a recent project. See <a href="#">Opening Project Files</a> .	Select <b>File</b> → <b>Open Recent Project</b> and choose from the list of projects or click <b>Clear Recent Projects</b> to reset the list. Up to the last 10 projects are listed and numbered in the order at which last opened.
	Save	Save a project. See <a href="#">Saving Project Files</a> .	<ul style="list-style-type: none"> <li>• Click the <b>Save</b> button  on the toolbar</li> <li>• Select <b>File</b> → <b>Save Project</b></li> <li>• Windows: Press &lt;Ctrl+S&gt;.</li> <li>• Mac: Press &lt;⌘ + S&gt;</li> </ul>
	Save Project As	Save an existing project with a new name. See <a href="#">Saving Project Files</a> .	<ul style="list-style-type: none"> <li>• Select <b>File</b> → <b>Save Project As</b></li> <li>• Windows: Press &lt;Ctrl+Shift+S&gt;</li> <li>• Mac: Press &lt;⌘ + S&gt;</li> </ul>
	Switch to Console Mode	This button is available when working in Graphical Mode. Click to switch to Console Mode and the command prompt.	Click the button to exit Graphical Mode and open Console Mode.
	Switch to Graphical Mode	This button is available when working in Console Mode. Click to switch to Graphical Mode and work with the GUI version.	Click the button to exit Console Mode and open Graphical Mode.
	Append Project	Combine two projects into one project file. See <a href="#">Appending Project Files</a> .	<ul style="list-style-type: none"> <li>• Select <b>File</b> → <b>Append Project</b></li> <li>• Windows: Press &lt;Ctrl+Shift+O&gt;</li> <li>• Mac: &lt;⌘ + O&gt;</li> </ul>
	Exit	Exit the program.	<ul style="list-style-type: none"> <li>• In the upper right-corner of the program, click the X.</li> <li>• Windows: Select <b>File</b> → <b>Exit</b> or press &lt;Ctrl+Q&gt; on the keyboard.</li> </ul>

Button or Icon	Action	Description	Action (s)
			<ul style="list-style-type: none"> <li>Mac: <b>Thermo-Calc</b> &lt;version&gt; → <b>Quit</b> or press &lt;⌘+Q&gt; on the keyboard.</li> </ul>
	Options	Change the global defaults for a variety of settings. See <a href="#">Global Settings</a> .	<p>To open the <b>Options</b> window:</p> <ul style="list-style-type: none"> <li>Windows: Select <b>Tools</b> →  <b>Options</b>.</li> <li>Mac: Select <b>Thermo-Calc</b> &lt;version&gt; → <b>Preferences</b> or <b>Settings</b> or press &lt;⌘&gt; on the keyboard.</li> </ul>
	Database Checker	Open the <b>Database Checker</b> , a program to check that the syntax of Thermo-Calc database files is correct. This is for advanced users who develop and manage databases. See <a href="#">About the Database Checker</a> .	Select <b>Tools</b> → <b>Database Checker</b>
--	Window menu	Highlight a specific window on the GUI	From the <b>Window</b> menu, select an option to refocus on that specific window.
	Help	<p>Open the help system where you can browse or search all the Thermo-Calc documentation.</p> <div style="border: 2px solid blue; padding: 10px; width: fit-content; margin: 10px auto;">  <p>This does not require an Internet connection. The content is stored locally.</p> </div>	<p>First open Thermo-Calc, then choose one of these options:</p> <ul style="list-style-type: none"> <li>Press &lt;F1&gt; on the keyboard.</li> <li>Select <b>Help</b> →  <b>Online Help</b>.</li> <li>On the <b>My Project Configuration</b> window, click  <b>Online Help</b>.</li> <li>At the bottom of any <b>Configuration</b> window, click  <b>Help</b>.</li> </ul>
	Video Tutorials	Open a link to the Thermo-Calc website where a variety of video tutorials are available.	<ul style="list-style-type: none"> <li>Select <b>Help</b> → <b>Video Tutorials</b></li> <li>On the <b>My Project Configuration</b> window, click  <b>Video Tutorials</b>.</li> </ul>

Button or Icon	Action	Description	Action (s)
	Examples Files	Open the examples folder based on whether you are in Console Mode or Graphical Mode. Also see <a href="#">Opening Example Project Files in Thermo-Calc</a> .	<ul style="list-style-type: none"> <li>• Select <b>File</b> → <b>Examples Files</b>.</li> <li>• Select <b>Help</b> → <b>Examples Files</b>.</li> <li>• On the <b>My Project Configuration</b> window, click  <b>Example Calculation</b>.</li> </ul>
	Manuals Folder	Open the folder on your computer that has the PDF documentation and guides. Also see <a href="#">About the Manuals (Documentation) Folder</a> .	Select <b>Help</b> → <b>Manuals Folder</b>
	Show License Info	Open the License Information window. Also see <a href="#">Displaying Thermo-Calc License Information</a> and the <i>Thermo-Calc Installation Guide</i>	Select <b>Help</b> → <b>Show License Info</b>
	Check for update	Check for updates to the software. Also see <a href="#">Updating the Software for Interim Releases</a> .	Select <b>Help</b> → <b>Check for Updates</b>
	Request Support	Open a link to the <a href="#">Support</a> on the Thermo-Calc website. There are instructions about how to submit your question.	Select <b>Help</b> → <b>Request Support</b>
	About	Includes the version number of the installed software and information about third party software licenses.	Select <b>Help</b> → <b>About</b>
	Notes	Open a <b>Notes - My Project</b> window to add notes to your project file.	At the bottom center of the <b>Configuration</b> window, click the Notes (  ) button. Once a note is added, hover your mouse over the button and it is then called <b>Read and edit notes</b> .

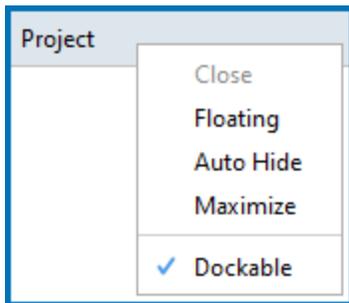
## Organizing the Windows on the Desktop

---

The windows on the Thermo-Calc desktop can be fixed or free floating. By default, most windows are fixed and open.

Using a combination of menu (right-click the window bar) and buttons (top right corner of a window), you can organize the windows as needed for your workflow.

For example, right-click the top of a window to select from the options to float, auto hide, maximize, or dock it. You can also left-click on the top bar of a window to float (i.e. detach) and move a window or to dock it again using the guides on the screen.



### ***Reset the Desktop Layout***

At any time to return to the standard desktop layout, from the main menu, select **Window** → **Reset Windows**.

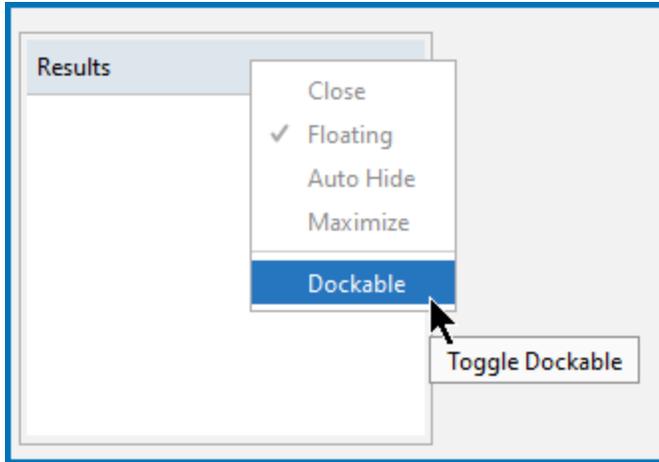
### ***Make a Window Dockable or "Permanently" Floating***

By default windows are dockable, i.e. you can move the windows around using the outline guides and adjust the layout location for each window on one monitor. When a window is NOT dockable the window can be moved around it is just does not become part of the layout, it stays floating all the time (i.e. it is "permanently" floating).



When **Dockable** is turned off (i.e. not selected) the other options are not available and there are no buttons at the top right corner of the window. However, you can reset the desktop layout to return it to the default location.

Right-click the top of a window to select **Dockable** to turn this option on and off.



### ***Auto Hide (Minimize) a Window***

When a window is fixed it can also be set to auto hide. Auto-hiding means that the window is minimized and placed as a tab either along the top or bottom or side of Thermo-Calc. A free-floating window is shown on top of the other windows and can be moved outside the frame of Thermo-Calc. when you select another window.

- Right-click the top of a window to select **Auto Hide** to turn this on and off.
- Click the pin  and unpin  buttons to **Toggle auto-hide** on and off. The buttons are on the top-right of a window.
- Click the minimize  button to **Hide active auto-hide window**. The window automatically minimizes if another window is selected. This button is available on the top right of a window that is already set to auto-hide.



The **Event Log** window is set by default to auto-hide.

### ***Maximize or Restore a Window***

- Double-click the top bar of a window (i.e. Project, Configuration, Visualizations, etc.) to maximize the window pane across the whole monitor. This hides the other windows. To restore the configuration (and the hidden windows) double-click the top bar of the window a second time.



This can be useful for small screens, or if you are working on a large screen and want to compare two configurations side-by-side.

- Right-click the top of a window to select **Maximize**. To restore the window to its previous state and location, right-click the bar of the window again and select **Restore**.

### ***Float or Fix a Window***

There are several ways to float and fix a window back to its most recent location in the layout.

- Right-click the top of a window to select **Floating** and turn this on and off.
- Left-click on the top bar of a window to float (i.e. detach) and move a window or to dock it again using the guides on the screen.
- Click the **Toggle floating**  button to float the window.
- Click the **Toggle floating**  button to turn off floating and dock the window automatically to its last location.

---

## Projects and Activities

---

### In this section:

---

About the Activity Nodes .....	14
Graphical Mode Activity Node Workflow .....	18
Creating Activities and Successors .....	20
Cloning Activities and Trees .....	23
Moving or Grouping Nodes and Using the Grid .....	25
Node or Window Status Markers .....	29
My Project and Activity Menus .....	31
Saving Project Files .....	35
Opening Project Files .....	38
Opening Example Project Files in Thermo-Calc .....	42
Appending Project Files .....	44
Converting Project Files to a Newer Version of Thermo-Calc .....	45
The Calculation Types in Graphical Mode .....	49
About the Uncertainty Calculations .....	53
Working with Batch Calculations .....	59
Getting Started Links and Templates .....	68

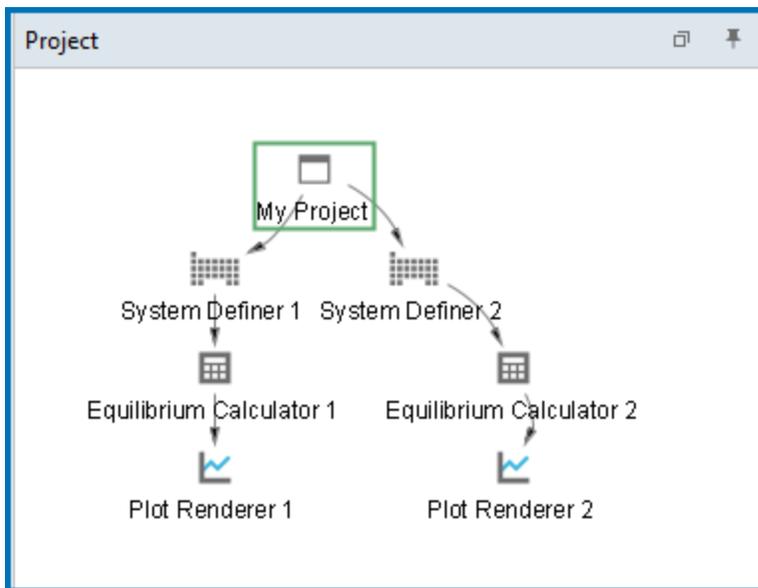
## About the Activity Nodes

### [Graphical Mode Activities vs Console Mode Modules](#)

In a *project*, a set of linked activities is called a *tree*. A result calculated within a tree is fed as input into the next *activity* in the tree. Consequently, if you have an Equilibrium Calculator with another Equilibrium Calculator as its *successor*, the successor takes the calculation results of the *predecessor* as the starting values for its calculation. When working with activities in the Project window these are also referred to as *nodes* (or *activity nodes*) in the tree structure.

The example below shows a  *My Project* node with two trees. In each tree, settings and calculation results are propagated downward until the calculation and the visualization of the results are completed in the *Plot Renderer* nodes.

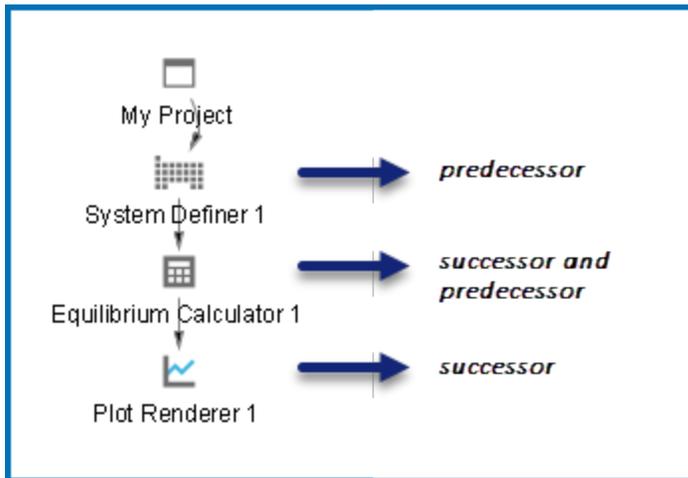
The system definitions, settings, and calculation results of the two trees are independent of each other in this example.



There are several resources available on our website to help you learn how to use Thermo-Calc and other Add-on Modules. Go to the [Getting Started Guides](#) page and choose one of the guides that provides new users an introduction to setting up simple calculations in Graphical Mode. The information there is also in this help documentation.

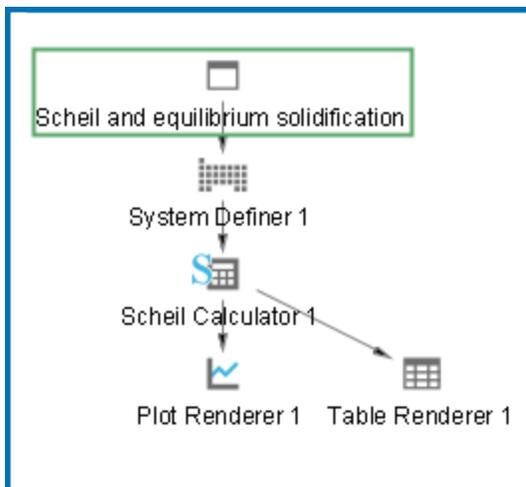
## Activity Successor and Predecessor Nodes

An activity node located below another activity node in a tree is referred to as that activity's *successor*. An activity located above another activity is called that activity's *predecessor*. A predecessor is performed before the predecessor's successors and its result is fed forward to any successor activities.



For example, to calculate and display a phase diagram, create a branch with three linked activities: A **System Definer** activity linked to an **Equilibrium Calculator** activity, which in turn is connected to a **Plot Renderer** activity.

You can determine the available successors by right-clicking a node in the **Project** window to browse what is available from the menu and follow the arrows connecting the nodes. The example below shows that **Scheil Calculator 1** is a predecessor to the successors, **Plot Renderer 1** and **Table Renderer 1**.



[Creating Activities and Successors](#) and [Cloning Activities and Trees](#)

## Activity Node Descriptions

The table below is a brief description of the activity types you work with in the **Project** window. When working with activities these are referred to as *nodes* in this documentation.

These are the project activity nodes available in Graphical Mode. Some of these require additional licenses for full feature availability. Listed in alphabetical order.

Activity Type	Description
<a href="#">AM Calculator</a>	Using the Additive Manufacturing (AM) Module this activity is used in conjunction with either the Scheil Calculator or a Material Library to define the necessary settings to better understand the laser powder bed fusion (LPBF) process by predicting the temperature distribution and melt pool geometry as a function of process parameters. There is also connectivity with the Diffusion Calculator and Precipitation Calculator to include probe data with a Thermal Profile calculation. In DEMO mode you can view the settings and experiment with some things, but no calculations can be run.
<a href="#">Binary Calculator</a>	Use it for some calculations involving two components only. It is like a combination of System Definer and Equilibrium Calculator activities with adaptations to simplify the configuration of calculations on binary systems.  <div style="border: 2px solid blue; padding: 10px; margin-top: 10px;">  To perform this activity, you need a database designed for the Binary Calculator, such as the TCBIN database. </div>
<a href="#">Diffusion Calculator</a>	Using the Diffusion Module (DICTRA), this activity is where you set the geometry, add one or more regions, enter grids and phases in the regions, and then enter compositions of phases. You can also use this in DEMO mode with up to three components. There is also connectivity to the Scheil Calculator and to other Add-on Modules such as the Additive Manufacturing (AM) Module.
<a href="#">Equilibrium Calculator</a>	Set thermodynamic conditions and define axis variables when a series of equilibrium calculations are to be performed in one or more dimensions.
<a href="#">Experimental File Reader</a>	Read experimental data files (*.EXP). This type of file contains information specifying a plotted diagram, written in the DATAPLOT graphical language.
<a href="#">Material to Material Calculator</a>	This activity calculates materials transitioning from one into the other.

Activity Type	Description
<a href="#">Plot Renderer</a>	Determine the layout of non-text based output.
<a href="#">Precipitation Calculator</a>	Set the conditions for, and perform, a precipitation calculation using the Precipitation Module (TC-PRISMA). You can also use this in DEMO mode with up to three components. There is also specialized connectivity to the Yield Strength Property Model and to other Add-on Modules such as the Additive Manufacturing (AM) Module.
<a href="#">Process Metallurgy Calculator</a>	This calculator allows for efficient setup of advanced calculations involving slag, metal and gas. Define the composition of a steel and slag system using material groups: one each for steel, slag and gas.
<a href="#">Property Model Calculator</a>	Choose one or more <b>General Models</b> to predict and optimize material properties. You can evaluate models simultaneously over a range of compositions and cross plot the results. You can then conduct an uncertainty analysis and plot the results as either a histogram or as a probability plot. With additional licenses, there are <b>Steel Models</b> and <b>Nickel Models</b> available.
<a href="#">Scheil Calculator</a>	Perform Scheil-Gulliver calculations (also known as Scheil calculations). A default Scheil calculation is used to estimate the solidification range of an alloy assuming that i) the liquid phase is homogeneous at all times and ii) the diffusivity is zero in the solid. However, it is possible to disregard the second assumption for selected components. The Scheil Calculator is also used with other specialized Add-on Modules such as the Diffusion Module (DICTRA) and Additive Manufacturing (AM) Module.
<a href="#">System Definer</a>	Define a certain thermodynamic system and read it from file into memory.
<a href="#">Table Renderer</a>	Use this activity for text-based output.
<a href="#">Ternary Calculator</a>	<p>Use it for some calculations involving three components. It is like a combination of System Definer and Equilibrium Calculator activities with adaptations to simplify the configuration of calculations on ternary systems.</p> <div data-bbox="451 1262 1386 1398" style="border: 2px solid blue; padding: 10px;">  To get a reliable result you need a database that fully assesses binary and ternary systems. </div>

## Graphical Mode Activity Node Workflow

---

You can set up a tree in the **Project** window (starting with the  **My Project** node) and then perform all the activities at once, or create and perform one activity at a time.



Additional licenses are required for some of the listed features.



You can also use the templates to add all the necessary activity nodes based on your simulation needs. See [Creating a Project from a Template](#).

The typical workflow is to:

1. *Define a System:* Add a [System Definer](#) activity (in the **Project** window) where you select a database and the elements to have as system components (in the **Configuration** window).
  - To the  **My Project** node, you can alternatively add and define one of these activity nodes: [AM Calculator](#), [Binary Calculator](#), [Ternary Calculator](#), [Process Metallurgy Calculator](#), or [Experimental File Reader](#).
2. *Set up and run a calculation:* Add an [Equilibrium Calculator](#) activity (a successor to the **System Definer**), where calculation conditions are set (temperature, pressure, etc.). This is where axis variables are set to create a property or phase diagram. These settings are also made in the **Configuration** window. You can also add a [Property Model Calculator](#), [Material to Material Calculator](#), [Precipitation Calculator](#), [Diffusion Calculator](#), or [Scheil Calculator](#) to the System Definer. The [AM Calculator](#) is also available and can either be added directly to the  **My Project** node or as a successor to the Scheil Calculator. It depends how you work with this Add-on Module.
3. *Preview some settings:* For some specialized calculations (i.e. with the Add-on Modules), you can preview the settings in the **Visualizations** window. For example, for the Diffusion Calculator (Composition Profiles and Thermal Profiles), for the Precipitation Calculator (Thermal Profiles), and with the AM Calculator a variety of settings are previewed during the set up.

- 
4. *Visualize the results:* Add either a [Plot Renderer](#) or a [Table Renderer](#) activity in the **Project** window. When calculated, this creates a plot or table in the **Visualizations** window. The results are shown in a **Plot Renderer** or a **Table Renderer** tab.



You need to **Perform** a Plot Renderer or a Table Renderer activity to visualize the calculation results from an Equilibrium Calculator.



[Getting Started Links and Templates](#) and [About the Activity Nodes](#)

## Creating Activities and Successors

You can create a new activity in these ways:

- In the **Project** window, right-click an activity and choose **Create New Activity** (📁 **My Project** node only) or **Create New Successor** (all other nodes).
- Select the activity you want to create from the submenu.
- At the bottom of the **Configuration** window, click **Create New Activity** or **Create New Successor** and select the activity to create.
- Use the available templates or the **Quick Start** wizard: Click the 📁 **My Project** node and select templates from the **Configuration** window to quickly add a set of activity nodes to the tree.



The activity nodes listed in the table are in alphabetical order not in order of selection from the various menus. Additional licenses are also required to access full functionality for any Add-on Modules.

<i>Activity</i>	<i>Possible Predecessors</i>	<i>Possible Successors</i>
📁 My Project	None; this is the first node of the tree structure. You can add activities using the Quick Start wizard or templates.   <a href="#">Getting Started Links and Templates</a>	AM Calculator, System Definer, Binary Calculator, Ternary Calculator, Process Metallurgy Calculator, or Experimental File Reader. Also choose templates from this level.
<a href="#">AM Calculator</a>	My Project, Scheil Calculator	Plot Renderer, Diffusion Calculator, Precipitation Calculator

Activity	Possible Predecessors	Possible Successors
		<div style="border: 2px solid blue; padding: 10px;">  <p>When a Diffusion Calculator or Precipitation Calculator is added, a full set of nodes, including a System Definer and Plot Renderer, are added as a template to the top My Project node. The Diffusion Calculator or Precipitation Calculator is linked to the AM Calculator.</p> </div>
<a href="#">Binary Calculator</a>	My Project	Plot Renderer
<a href="#">Diffusion Calculator</a>	System Definer, AM Calculator	Plot Renderer, Table Renderer
<a href="#">Equilibrium Calculator</a>	System Definer, Equilibrium Calculator	Equilibrium Calculator, Plot Renderer, Table Renderer
<a href="#">Experimental File Reader</a>	My Project	Plot Renderer
<a href="#">Material to Material Calculator</a>	System Definer	Plot Renderer, Table Renderer
<a href="#">Plot Renderer</a>	Equilibrium Calculator, Material to Material Calculator, Property Model Calculator, Binary Calculator, Ternary Calculator, Scheil Calculator, Process Metallurgy Calculator, Diffusion Calculator, Precipitation Calculator, AM Calculator, Experimental File Reader	None
<a href="#">Precipitation Calculator</a>	System Definer, AM Calculator	Plot Renderer, Table Renderer
<a href="#">Process Metallurgy Calculator</a>	My Project	Plot Renderer, Table Renderer
<a href="#">Property Model Calculator</a>	System Definer	Plot Renderer, Table Renderer
<a href="#">Scheil Calculator</a>	System Definer	Plot Renderer, Table Renderer, AM Calculator, Diffusion Calculator

---

<i>Activity</i>	<i>Possible Predecessors</i>	<i>Possible Successors</i>
<a href="#">System Definer</a>	My Project, System Definer	Property Model Calculator, Equilibrium Calculator, Precipitation Calculator, Scheil Calculator
<a href="#">Table Renderer</a>	Equilibrium Calculator, Material to Material Calculator, Property Model Calculator, Binary Calculator, Scheil Calculator, Process Metallurgy Calculator, Diffusion Calculator, Precipitation Calculator	None
<a href="#">Ternary Calculator</a>	My Project	Plot Renderer

## Cloning Activities and Trees

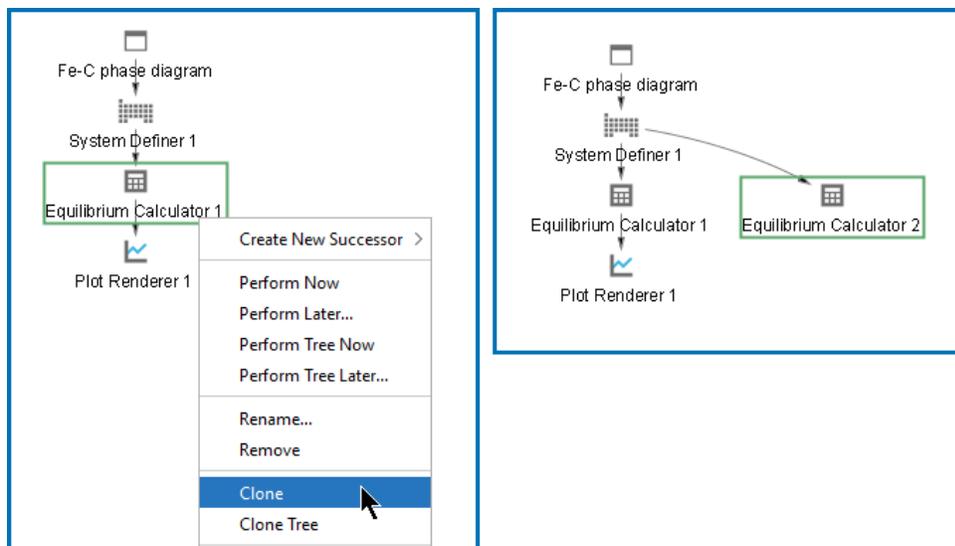
As described in [Creating Activities and Successors](#), you can clone a single activity or the selected activity and all the activities that come after it (successors). The predecessor of the selected activity also is the predecessor of the clone.

Cloned activities are configured exactly like the activities that they were cloned from, and any results of calculations or plots are also cloned. To clone a selected activity, right-click the activity and select **Clone**.

For example, if you clone the Equilibrium Calculator in this project the result is an **Equilibrium Calculator 2** created as a successor to **System Definer 1**.

The following applies to clones.

- The new Equilibrium Calculator has the same settings for conditions, functions and options as its Equilibrium Calculator 1 clone.
- If the Equilibrium Calculator is already performed, then the calculation results are cloned.
- If the activity is a Plot Renderer or a Table Renderer, then the plot or table is cloned.
- If you want to clone all the successor activities that follow a selected activity, then right-click and select **Clone Tree** instead.



*Cloning an Equilibrium Calculator activity node.*

---

## **About Equilibrium Calculator Clones and Successors**

If you create an Equilibrium Calculator as a successor to another Equilibrium Calculator, then the successor inherits all the settings for conditions, functions and options from its predecessor. In this respect, it is similar to a clone.



The Equilibrium Calculator successor is different from a clone in that it does not inherit any calculated results and it is a successor to the Equilibrium Calculator instead of a successor to its predecessor.

The calculated result from an Equilibrium Calculator that is the predecessor to another Equilibrium Calculator is the starting value for the latter's calculation. The clone of an Equilibrium Calculator on the other hand, does not receive any data as input from the Equilibrium Calculator that it is cloned from.

---

## Moving or Grouping Nodes and Using the Grid

---

In the **Project** window, you can work with the activity nodes in these ways.

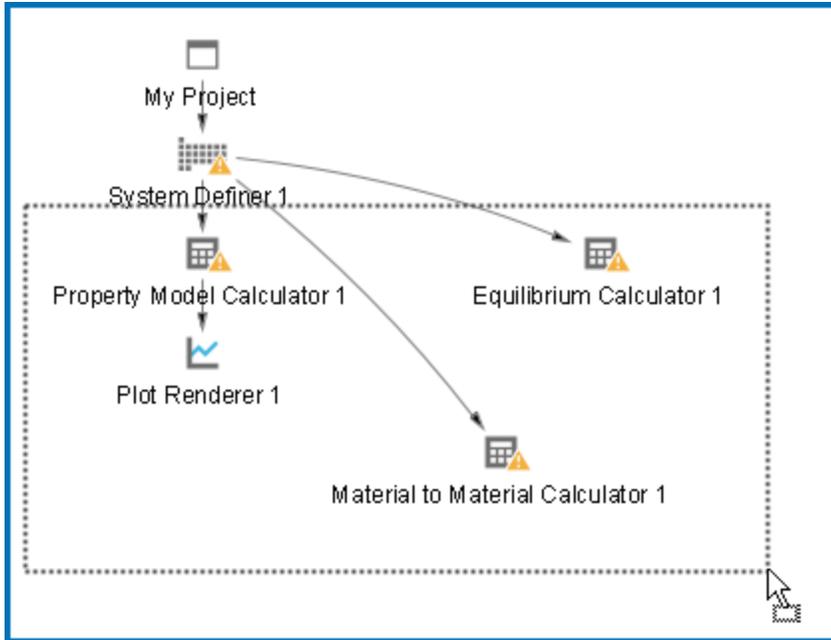
- Select and move individual nodes
- Select several nodes at a time
- Select and then group activity nodes
- Show the grid and "snap" to it
- Apply Auto Layout

### ***Select and Move Individual Nodes***

- Click an activity node to *select* it.
- To move activities, click and hold the activity node and move the cursor.

### ***Select Several Nodes at a Time***

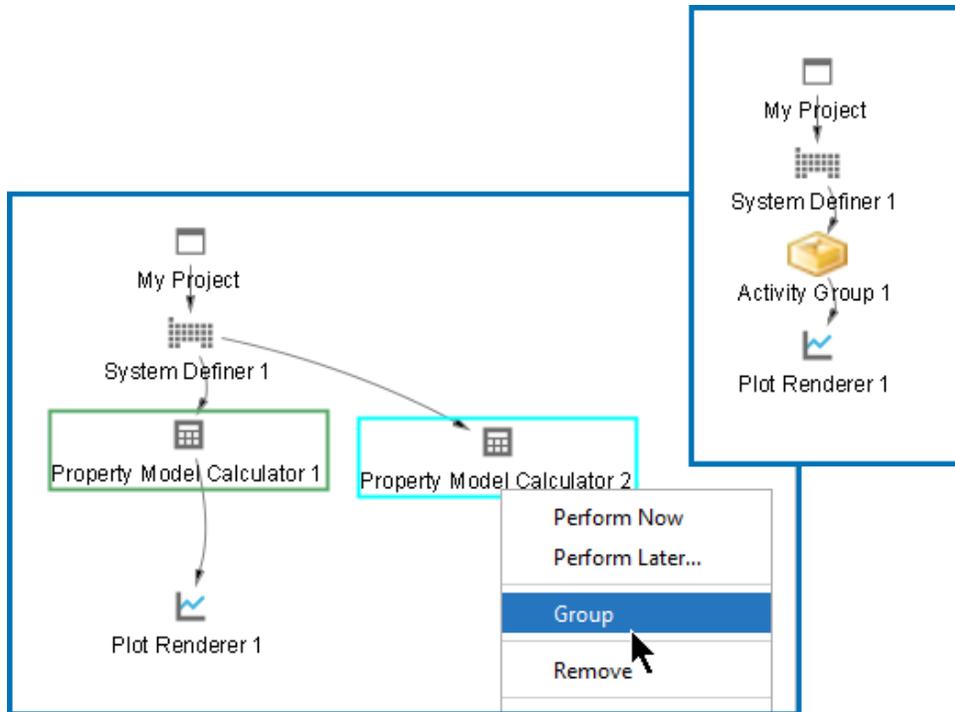
- To select several activity nodes, hold down <Ctrl> while clicking on each node you want to select.
- Click and hold the mouse button and draw a square around the activity nodes to select. Both the name and the icon that represents an activity must be within the square.



### **Group Activity Nodes**

You can group activity nodes together on the **Project** window. This also creates separate tabs on the **Configuration** window so you can more easily edit the settings for each one. It is important to have logical groupings of nodes because the calculation relies on this connectivity.

In the **Project** window, control + click to select each node, then right-click and select **Group**. An **Activity Group** node is created (you can rename it) and on the **Configuration** window there are two tabs created, named for each of the group nodes. For example, Property Model Calculator 1 and Property Model Calculator 2. The order of the tabs is based on the order of when you select each node.



### ***Apply Auto Layout***

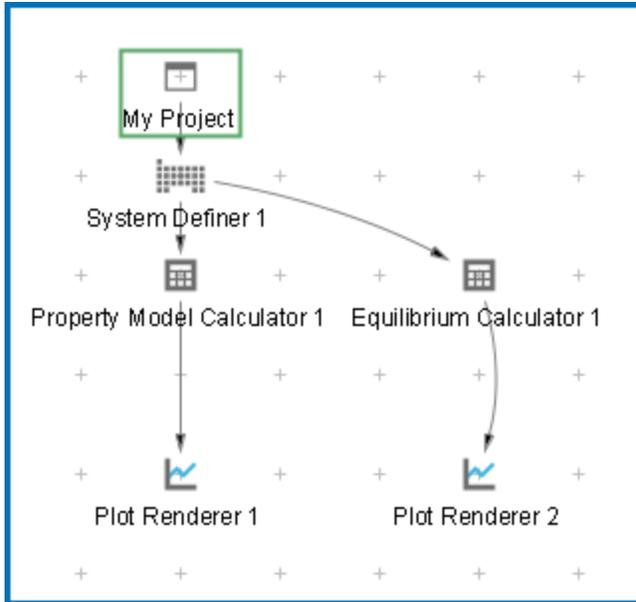
In the **Project** window, right-click anywhere and choose **Apply Auto Layout** from the menu. The nodes and the tree default to the center of the Project window.

### ***Show Grid and Snap to Grid***

To show a grid of light grey cross-hairs overlaid on a project, click **Show Grid** in the toolbar. Click the button again to turn the grid off. Or right-click in the **Project** window and select **Show Grid**.

Click **Snap to Grid** in the toolbar or right-click in the **Project** window and select **Snap to Grid** to automatically position each activity in the Project window at a cross-hairs point in the grid.

The Project window with the grid turned on and activity nodes moved from the default location. These nodes are snapped to the grid.



## Node or Window Status Markers

Status markers indicate whether an activity is ready to be performed, is being performed, or has been performed. The markers are overlaid on the activity nodes in the **Project** window. The status markers are displayed on a **System Definer** node.

 [About the Activity Nodes](#)

### Project Window Status Markers

Status	Description
	No status marker means the activity is ready to be performed (i.e. it has not been performed).
	A yellow circle with an exclamation mark means there is an error or the activity needs to be run again. For example, the necessary configurations for the activity may not be set or other prerequisites are not met.
	A green circle with a right-facing arrow means the activity is being performed.
	A pink circle with a clock face means the activity is currently scheduled to be performed or in the process of being performed.
	For the Diffusion Calculator and Precipitation Calculator, you can pause a calculation. A blue circle with a the pause symbol indicates this state.
	A green circle with a check mark means the activity has finished its calculation, i.e. it is performed.

### Scheduler Window Status Markers

Status	Description
	No status marker means the activity (the job) is scheduled to be performed (the job has not started).

Status	Description
	A yellow circle with an exclamation mark means there an error happened during the activity job.
	A green circle with a right-facing arrow means the activity (the job) is being performed.
	A green circle with a check mark means the activity (the job) performed successfully.

### **Visualizations Window Status Marker**

If you have plotted a diagram and then modify something in the tree that the Plot Renderer is associated with, then a notification —an exclamation mark enclosed in a yellow triangle— is shown in the **Visualizations** window for the applicable tab. The tab names match the node names in the **Project** window (i.e. if a node is renamed, the tab is renamed to match).



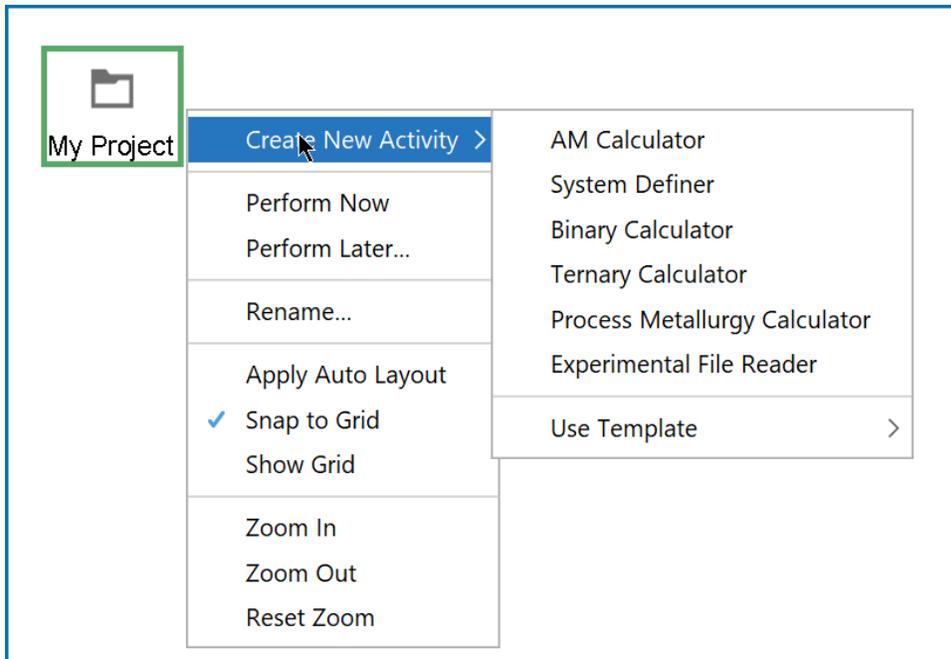
## My Project and Activity Menu

When working in the Project window you can right-click nodes to open menus with the available actions. For example, click the  **My Project** node to open the menu. Most nodes in the tree have a common menu with differences due to placement in the tree and the activity type.

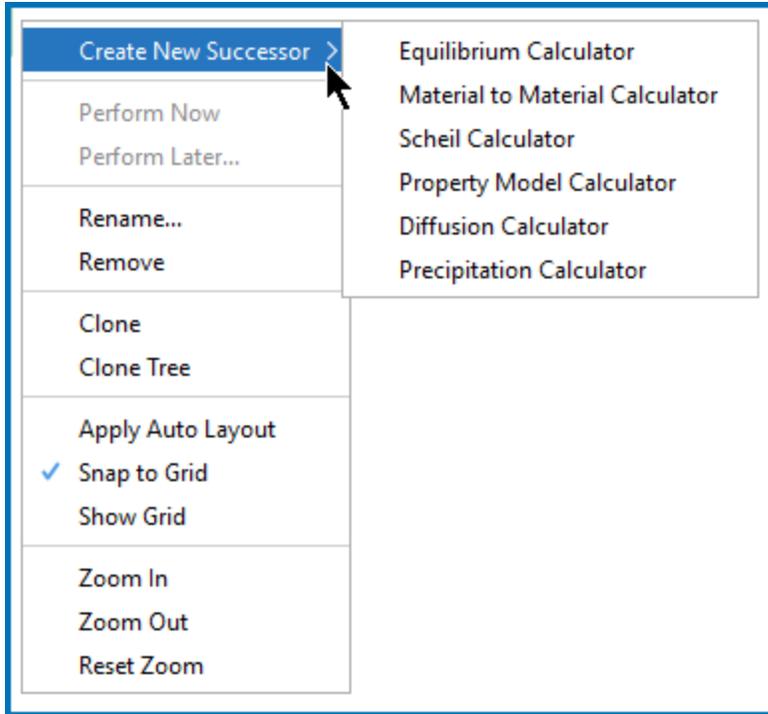
 [About the Activity Nodes](#)

### Menus

#### MY PROJECT NODE



#### ACTIVITY NODE GENERAL



## Menu Options for Activity Nodes

Option	Description	Action (s)
Create New Activity	Add AM Calculator, System Definer, Binary Calculator, Ternary Calculator, Process Metallurgy Calculator, and Experimental File Reader nodes.	Right-click <b>My Project</b> and from the <b>Create New Activity</b> menu choose an option.   <a href="#">Creating Activities and Successors</a>
Create New Activity → Use Template	Choose from the available templates to use the Wizard or add predefined nodes.	Right-click <b>My Project</b> and from the <b>Create New Activity → Use Templates</b> menu choose an option.  The templates are also available on the <b>Configuration</b> window, which you can access by clicking the <b>My Project</b> node, or from the <b>Window</b> menu.

Option	Description	Action (s)
		 <a href="#">Creating a Project with the Quick Start Wizard</a> or <a href="#">Creating a Project from a Template</a>
Create New Successor	Add a successor to the selected node.	Right-click the <b>System Definer</b> , any <b>Calculator</b> node or the <b>Experimental File Reader</b> . Then from the <b>Create New Successor</b> menu choose an option.   <a href="#">Creating Activities and Successors</a>
Add a Predecessor	Add a predecessor to the selected node.	Right-click a node to add a predecessor to it (when available).
Perform Now or Perform Later	Perform a calculation or create a plot or table immediately, or schedule it for a time in the future.	Right-click any node.   <a href="#">Using the Scheduler</a>
Perform Tree Now or Perform Tree Later	Perform a calculation for the whole project immediately, or schedule it for a time in the future.	Right-click the System Definer or any Calculator node.   <a href="#">Using the Scheduler</a>
Rename	Change the name of the node.	Right-click any node.
Remove	Delete the selected node from the tree.	Right-click any node except My Project.
Clone	Duplicate the selected node and add it to the same tree.	Right-click any node except My Project.   <a href="#">Cloning Activities and Trees</a>

<i>Option</i>	<i>Description</i>	<i>Action (s)</i>
Clone Tree	Duplicate a tree and add it to My Project.	Right-click any node except My Project.
Apply Auto Layout	Apply an automatic layout to the nodes in the tree.	Right-click any node.
Snap to Grid	Snap a node to align it with the grid.	Right-click any node.  <a href="#">Moving or Grouping Nodes and Using the Grid</a>
Show Grid	Turn the grid on or off in the Project window.	Right-click any node.
Zoom In, Zoom Out, Reset Zoom	Zoom in and out of the Project window or reset the zoom to the default.	Right-click any node.

## Saving Project Files

---

In Graphical Mode, Thermo-Calc uses a *project file* format with a .tcu suffix.



If you are working in Console Mode there are several file formats used for both Thermo-Calc and the Diffusion Module (DICTRA). See [Working with the Console Mode File Types](#).

A project file always includes project activities and the configurations. You can also include calculated results, although there are factors and limitations that need to be considered as described below and also detailed in [Opening Project Files](#) and [Converting Project Files to a Newer Version of Thermo-Calc](#).

In general, to save your project file, you can choose between the following methods:

- On the toolbar click the **Save**  button.
- Press CTRL + S (Windows) or ⌘ + S (macOS).
- From the menu, select **File** →  **Save Project**.
- When you want to give an existing project a new name, OR when saving with results, select **File** →  **Save Project As** . See [Including Calculated Results with Project Files](#) and [Saving a Project File with Results](#).



If you are working in a file that includes results, any method to save the project file continues to include the calculated results. However, if this is the first time results are being included, there are additional important factors to consider as described next.

## Including Calculated Results with Project Files



Starting with Thermo-Calc version 2022b, saved calculated results are only accessible in the same version of Thermo-Calc in which it is originally created. For example, if you have a project file with saved results from version 2020a, the results are only immediately available in 2020a. You can regenerate results in a different version of the software but that can require adjustments to the configuration if new functionality or changes impact the particular project set up, especially if there is a wide gap between release versions.

When including calculated results in the project file there are these factors to consider: Future compatibility, file size, and plot design and themes.

- **Future compatibility:** If you want to use this project file over time, in future versions of Thermo-Calc, it will require recalculation after opening. It also depends on the difference in versions; the project can require reconfiguration of the settings and other fine tuning. However, if you are going to work in the same version of the software for a short term project, for example, saving the file with results eliminates the need for re-calculating results when the file is re-opened. This is particularly advantageous when a simulation takes several hours to complete such as when working with the Add-on Diffusion (DICTRA), Precipitation (TC-PRISMA), Process Metallurgy, and/or Additive Manufacturing Modules.
- **File size:** When results are included, the size of the file increases. This is especially true when working with the Add-on Diffusion (DICTRA), Precipitation (TC-PRISMA), Process Metallurgy, and/or Additive Manufacturing Modules.
- **Plot design and themes:** When you save your results, the plot design may have local or global designs or themes applied, which are also saved as part of the calculated results. There are considerations related to this when you open the project file again, especially in future versions of the software. See [Opening Project Files](#) for more information.

## Saving a Project File with Results

1. First, determine if saving results is efficient for your particular project based on future compatibility, file size, and plot design and theme factors.

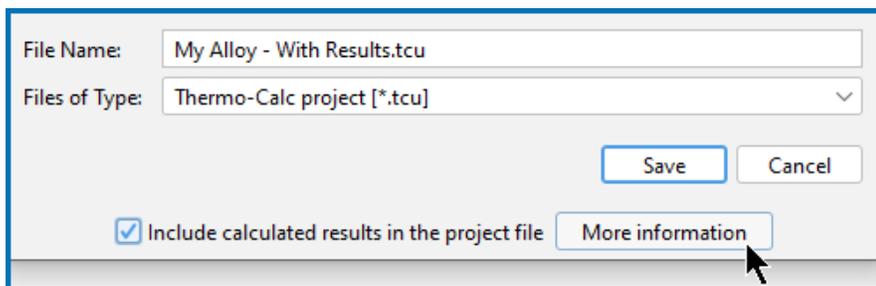
To save calculated results with the project file, choose an option:

- From the menu, chose **File** →  **Save Project As** or,
  - Press CTRL + Shift + S (Windows) or ⌘ + S (macOS).
2. On the **Save** window, select the **Include calculated results in the project file** checkbox.



The checkbox is not available (it is greyed out) if no results have yet been calculated. In this case, go back to your project, click **Perform Tree** (or any other action to generate results) and then save the project file as above.

3. Click **More information** to learn about the activity nodes with calculated results that are included / saved in the project file.



4. Enter a meaningful **File Name**. It can be useful to include the version of the software in the file name, especially if the file is converted in a future version of the software.
5. Navigate to the location where you want to save your project file and then click **Save**.



[Converting Project Files to a Newer Version of Thermo-Calc](#)

## Opening Project Files

---

Opening a project file in the same software version in which it was saved, with or without results, is straight forward and there are several methods to open a file.



See the [Saving Project Files](#) topic for useful details about saving a project file to include calculated results.

- For standard instructions to open a project file with or without results, see [Opening Project Files in the Same Version of Thermo-Calc](#). There are checkbox controls for plot themes and opening with or without results that are also described.
- To open a recent project file, see [Opening Recent Project Files](#).



Starting with Thermo-Calc version 2022b, all older project files including results generated and saved in Thermo-Calc versions 2015a to 2022a, can only be opened with results in the original software version. You can still open a project file without results and then re-perform the calculation. However, this can require a review and update to the settings and configurations, especially if there is a wide gap between versions.



For old project files being opened in a different software version, see the topic [Converting Project Files to a Newer Version of Thermo-Calc](#), where you can also learn about regenerating the results and the automatic creation of backup files in these instances.

## Opening Project Files in the Same Version of Thermo-Calc

### *To Open a Project File from Thermo-Calc*

1. To open a project file with or without results in the same version of the software in which it was created, choose a method below:

- On the toolbar, click the **Open**  button.
  - From the main menu, select **File** →  **Open Project**.
  - Press <Ctrl+O> (Windows) or <⌘ + O> (macOS).
2. On the **Open File** window review the checkboxes that control global plot themes and results:
- By default the **Apply plot theme from Tools/Options** checkbox is selected and depends on whether the global setting checkbox is also selected on the **Options** window, which applies a predefined or custom theme globally to the project file. Click to deselect the checkbox to keep the original plot design in your project file. For more information about the settings, see [About Project Files and Plot Themes](#) for additional details.



For macOS users, **Tools/Options** refers to the **Options** window, which is accessed differently. Select **Thermo-Calc <version> → Preferences** or press <⌘> on the keyboard.

- By default the **Open project with results if present** checkbox is selected. Click to deselect the checkbox if you want to exclude results for a file saved in the same version of the software.



If you are opening an older project file with results (in which case the file is converted), you are prompted by a secondary window that includes related version information and options to continue or not. **Even if the checkbox is selected, old project files still cannot be opened with results.** See [Converting Project Files to a Newer Version of Thermo-Calc](#).

3. When you are ready, navigate to the file on your computer and click **Open**.

### ***To Open a Project File from a File Location on the Computer***

Navigate to the project file on your computer then choose between these methods:

- in an open instance of Thermo-Calc, drag and drop it into the **Project** window. This replaces any other project already open.
- double-click to open it. This launches a new instance of Thermo-Calc.



The checkbox controls **Open project with results if present** and **Apply plot theme from Tools/Options** are not available when dragging and dropping or double-clicking to open.



For macOS users, **Tools/Options** is the same as the **Preferences** menu that accesses the **Options** window.



For old projects with results, and if you are dragging and dropping or double-clicking to open, you are also prompted to decide what to do next. See [Converting Project Files to a Newer Version of Thermo-Calc](#).

## Opening Recent Project Files

Select **File** → **Open Recent Project** and choose from the list of projects. Up to the last 10 projects are listed and numbered in the order in which they were last opened. You can click **Clear Recent Projects** to reset the list.

## About Project Files and Plot Themes

When opening project files, a predefined or custom plot theme can be automatically applied to all plots. There are two checkboxes that control this, one is a global setting and one is a local setting applied to a specific file as it is opened.



If you double-click or drag and drop to open a project file it automatically uses the global settings in Thermo-Calc (which may or may not be to apply a theme). This means you may inadvertently apply a theme to your plots without realizing it.

On the **Open File** window, the **Apply plot theme from Tools/Options** checkbox is selected by default and applies a theme locally to the specific project file being opened. Click to clear the checkbox to allow the specific file to keep existing plot settings.

If this checkbox is selected or cleared it only applies to the specific project file being opened, i.e. the global settings **Apply the selected theme to opened file [.tcu]** checkbox on the **Tools** → **Options** window does not change.



For macOS users, **Tools/Options** is the same as the **Preferences** menu that accesses the **Options** window.



See [Working with Plot Themes](#) for the details about how the checkbox interacts with its counterpart.



If you have a project file with calculated results and local settings applied to plots i.e. applied from the **Visualizations** window to a specific plot via the **Plot Properties** settings window, then click to clear the local setting as you open the file using **Apply plot theme from Tools/Options** checkbox. Otherwise the default global settings are applied to all plots and you would need to individually change these settings back to your original design. Alternatively, experiment with applying a predefined theme, e.g. **Publication**, as this could be sufficient for your final output.

## Opening Project Files Including Results and with Different GES Versions

When projects with calculated results created with a different Gibbs Energy System (GES) version than selected from the **Options** window, the results are discarded as these are not 100% compatible.



GES6 (**Version 6** is the default, but some may be using GES5 (**Version 5**). See [Global General Settings](#) for more information.

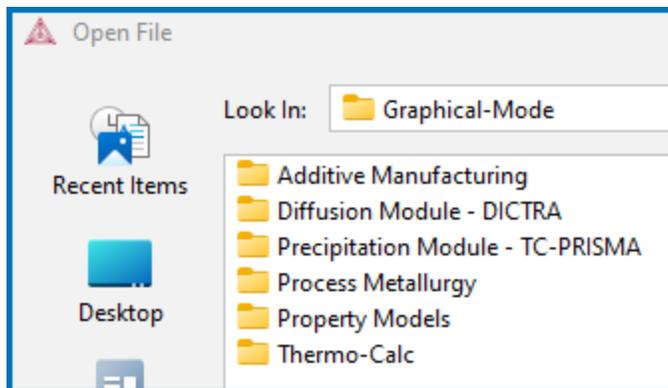
## Opening Example Project Files in Thermo-Calc



Additional licenses may be required to run calculations with the examples. The individual example descriptions detail this information.

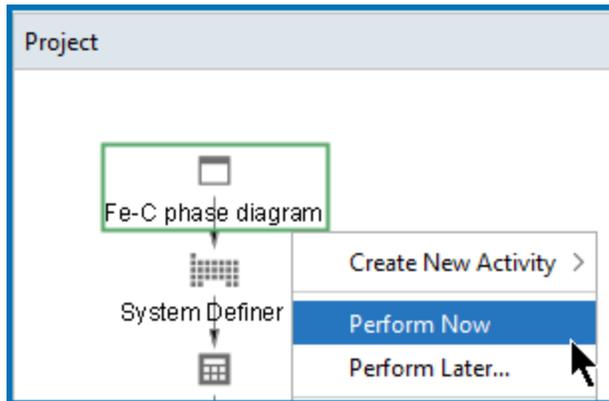
To open an example project file:

1. Open Thermo-Calc.
2. Confirm you are in Graphical Mode. If not, click the **Switch to Graphical Mode**  button.
3. There are these ways to open the folders and files:
  - From the main menu select **File** →  **Examples Files**
  - From the main menu select **Help** →  **Examples Files**.
  - Click the **My Project** node and on the **Configuration** window, click the **Example Calculation**  button.
4. The examples are divided into folders. Navigate to the folder and project file you want and double-click to open.



5. The Graphical Mode examples are stored without calculated results so you will want to run the example calculation and generate a plot or table. To do this, click the first node in the tree at the top of the **Project** window. Then right-click and select **Perform Now**. Some examples require additional licenses in order for the calculation

to run.

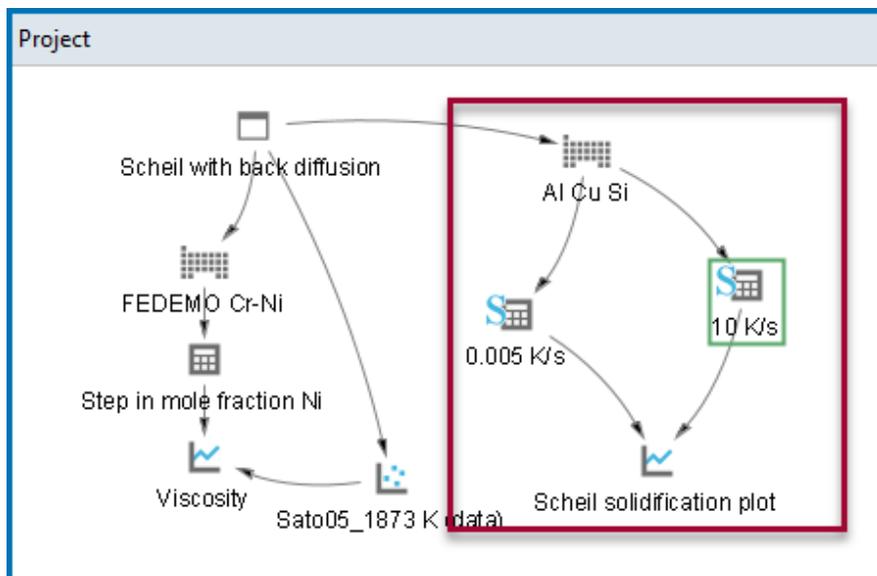


## Appending Project Files

You can only have one project file open at a time in any one instance of Thermo-Calc. However, you can *append* the nodes from additional project files to the topmost **My Project** node in the **Project** window.

To append a project file into the Project tree:

- Chose an option:
  - From the main menu, select **File** → **Append Project** .
  - Press <Ctrl+Shift+O> (Windows) or <⌘+O> (macOS).
- Then in the **Append file** window, navigate to the project file and click **Open**.
- The nodes are appended to the top node. Drag the nodes to separate and arrange within the Project window.



[Saving and Printing the Output \(Graphical Mode\) and Saving Tabulated Data for the Table Renderer](#)

# Converting Project Files to a Newer Version of Thermo-Calc

This topic discusses the various considerations related to opening old project files, with or without results, in a different version of Thermo-Calc, i.e. converting a project file. In both types of conversion (with or without results), a backup file is automatically created and can be useful during the review or regeneration of results for a converted file.

 [Saving Project Files](#) and [Opening Project Files](#)

## About Backup Files

When you first open a project file in a newer version of Thermo-Calc, a backup file is automatically created. The backup file has (backup) added to the end of the file name, e.g. My Alloy (backup).tcu. This is applicable whether the file has included results or not.

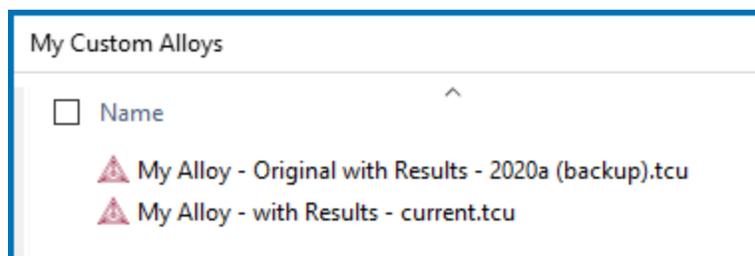


If a file including the name (backup) is also converted, then another (backup) is added to the file name, e.g. My Alloy (backup)(backup).tcu

As soon as the original project file containing results is converted, it overwrites any previously saved results. This is where the backup file can be useful.



Since the version of the software used for the original project file is not always known, you can use the information provided in the **Old project file with results** window to rename the backup file with key information once the file is converted. See [Converting Old Project Files - Original with Saved Results](#) for information about this window.



An example of the backup file created (in the same folder) when an old project file is opened in a newer version of Thermo-Calc, i.e. converted. It is recommended that users rename the file in a meaningful way in order to include a software version number and to indicate if it contains saved results.

## Converting Old Project Files - No Results Saved

When opening an old project file without results, the procedure is the same as described in [Opening Project Files](#). The main difference is that a backup file is automatically created. There can also be potential differences in setup configuration that need to be addressed.

After opening the file in the new version of Thermo-Calc, also open the **Event Log** to see if there are any messages. Try also to perform the simulation and check the results against the original or expected output, then continue as normal.

## Converting Old Project Files - Original with Saved Results

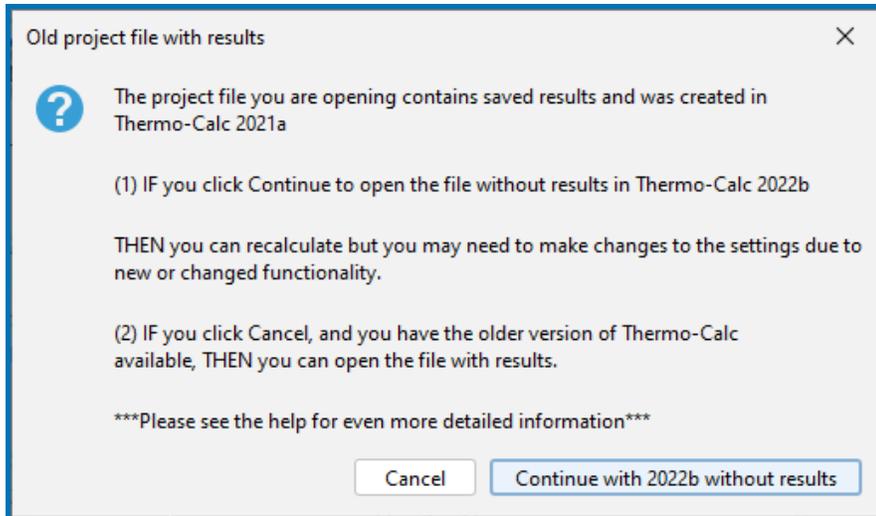


Starting with Thermo-Calc version 2022b, all older project files including results generated and saved in Thermo-Calc versions 2015a to 2022a, can only be opened with results in the original software version. You can still open a project file without results and then re-perform the calculation. However, this can require a review and update to the settings and configurations, especially if there is a wide gap between versions.

### ***To Open and Convert a Project File with Results***

1. Follow the instructions as described in [Opening Project Files](#).
2. At the point of opening the old project file including results, the **Old project file with results** window displays. This also occurs when the **Open project with results if present** checkbox is selected because no results can be included.

The window provides information about the original Thermo-Calc version in which the results were calculated as well as the version of the software in which you are opening the project file.



3. Click **Continue** to open and convert the file to the version of the software indicated. A backup file is automatically created, which includes the old results. Click **Cancel** to exit without converting the file.
  - a. If you click **Continue**, then it is recommended that you first review the setup and then regenerate the results. Review the **Event Log** to see if there are any messages.
  - b. If you click **Cancel**, or want to use the backup file to compare results between versions, then open the old or backup file in the original version of the software.

### ***Regenerating the Results for a Converted File***

As part of the project file conversion, all previous results are overwritten and the file is opened without results. However, the project structure and settings are maintained.

When the versions of the software are not far apart, then re-performing a simulation is likely to have no or few errors recorded in the Event Log.

However, to regenerate and compare results in a newer version of Thermo-Calc also depends on the original project set up and whether new or improved functionality in Thermo-Calc impacts the recalculation. Another consideration includes the availability of a newer version of a database than was used in the original calculation. In other words, there are various factors that can change the new or regenerated results compared to the original.

## Opening Old Project Files in the Original Version of Thermo-Calc

There are various reasons for wanting to open an old project file in its original version of the software. For example, if you are:

- Regenerating the results in a newer version of the software and there are errors in the **Event Log** or unexplained differences.
- Reviewing or comparing the original configuration settings to update the settings.
- Comparing results from the older version of the software to the converted version of the results.



Different versions of Thermo-Calc can be installed at the same time on your computer and have project files opened alongside each other to facilitate such comparisons.



If you are having problems with opening an old project file, or are unable to access an installed version of the software that you know your organization has a license for, complete this [Request Support](#) form on our website to get help. Be sure to include all relevant details such as your organization number.

## The Calculation Types in Graphical Mode

---



A variety of [Plot Types](#) are available based on the Calculator and, in most cases, *Calculation Type* selected. Also see [Graphical Mode vs Console Mode: Calculation Types](#).

### **AM Calculator**

The **AM Calculator** is available with the Additive Manufacturing (AM) Module. It is used to predict the temperature distribution and melt pool geometry as a function of process parameters. There are these simulation types: **Steady-state**, **Transient**, and **Transient with heat source from Steady-state**. Then for **Steady-state** simulations, you can also choose to run a **Heat source calibration** calculation type instead of only a **Single point**.

### **Binary Calculator**

The following calculation types are available for a **Binary Calculator** activity node:

- **Phase diagram:** To calculate in a defined multicomponent heterogeneous system, from one or more initial equilibria. A phase diagram is usually mapped within a specific space that is constructed by two (or more) defined independent mapping axis-variables.
- **Gibbs energy curves:** To calculate at a constant temperature over the whole composition range.
- **Activity curves:** To calculate at a constant temperature over the whole composition range.
- **Phase fractions:** To calculate as a function of temperature at a constant composition.

### **Diffusion Calculator**

Diffusion calculations are done with the **Diffusion Calculator** to have detailed simulation of diffusion controlled reactions in multicomponent systems. There are no calculation types you need to choose when setting up your simulation.

### **Equilibrium Calculator**

The following calculation types are available for an **Equilibrium Calculator** activity node:

- **Single equilibrium** (no axes): To calculate a single point. The results from this calculation are displayed in the **Event Log**.
- **One Axis**: To vary a quantity on the X-axis.
- **Grid**: Computes the equilibrium of each grid point in a two-dimensional grid.
- **Phase diagram**: To calculate in a defined multicomponent heterogeneous system, from one or more initial equilibria. A phase diagram is usually mapped within a specific space that is constructed by two (or more) defined independent mapping axis-variables.

## ***Material to Material Calculator***

The following calculation types are available for a **Material to Material Calculator** activity node:

- **Single equilibrium** (no axes): To calculate a single point. The results from this calculation are displayed in the **Visualizations** window if a Table Renderer is used.
- **One Axis**: To vary a quantity on the X-axis; the fraction of one of the materials or temperature.
- **Phase diagram**: To calculate a section in a multicomponent system between two materials, transitioning from one into the other. The phase diagram is mapped within a specific space that is constructed by the two mapping axis-variables; the fraction of one of the materials and temperature.



See [Plot Types](#) for Plot Renderer choices related to the **One Axis Calculation Type**.

## ***Precipitation Calculator***

The following calculation types are available for a **Precipitation Calculator** activity node:

- **Isothermal**: Precipitation simulation at constant temperature.
- **Non-isothermal**: Precipitation simulation with user-defined temperature profile.
- **TTT Diagram**: Time-Temperature-Transformation diagram calculation.
- **CCT Diagram**: Continuous Cooling Transformation diagram calculation.

---

## Process Metallurgy Calculator

The following calculation types are available for a **Process Metallurgy Calculator** activity node when doing an **Equilibrium** simulation.

- **Single**: To calculate a single point. The results from this calculation can be displayed in the **Visualizations** window using a **Table Renderer**.
- **One Axis**: To vary a quantity on the X-axis.
- **Grid**: Evaluates two axis variables of the selected quantities in the specified range and number of steps.
- **Uncertainty**: Evaluates calculations where the values of the quantities are sampled from Gaussian distributions. The **Mean** field is as defined under *Condition Definitions* for the respective quantity. The result is visualized as a histogram or normal probability plot by adding a Plot Render activity.

## Property Model Calculator

The following calculation types are available for a **Property Model Calculator** activity node:

- **Single** to calculate a single point. Any model can use this calculation type and no additional settings are required. The results from this calculation are displayed in the **Event Log** or if a **Table Renderer** is used, in the **Visualizations** window.
- **One Axis**: To vary a quantity on the X-axis.
- **Grid**: Evaluates two axis variables of the selected quantities in the specified range and number of steps.
- **Min/Max**: Evaluates the Property Model(s) for all variations of the selected quantities at the given limits. The **Mean** field is as defined under *Condition Definitions* for the respective quantity. The total minimum and maximum of the model(s) results are shown in the **Event** log.
- **Uncertainty**: Evaluates the Property Model(s) where the values of the quantities are sampled from Gaussian distributions. The **Mean** field is as defined under *Condition Definitions* for the respective quantity. The result is visualized as a histogram or normal probability plot by adding a Plot Renderer activity.
- **Batch**: Use this to allow for high throughput calculations and read data from a file, such as a spreadsheet, rather than, for example, having a fixed grid or random values. Results from these batch calculations can then be compared to experimental values (as long as this data is included), using a **Cross plot** on the Plot Renderer.

## ***Scheil Calculator***

When using a **Scheil Calculator** activity node there are no calculation types you need to choose when setting up your simulation.

## ***Ternary Calculator***

The following calculation types are available for a **Ternary Calculator** activity node:

- **Isothermal section:** To calculate the ternary system for the whole composition range.
- **Monovariant lines:** To calculate the variation of the liquid compositions with temperature.
- **Liquidus projection:** The projection liquid surface and the monovariant lines are calculated over the given temperature range.

## About the Uncertainty Calculations

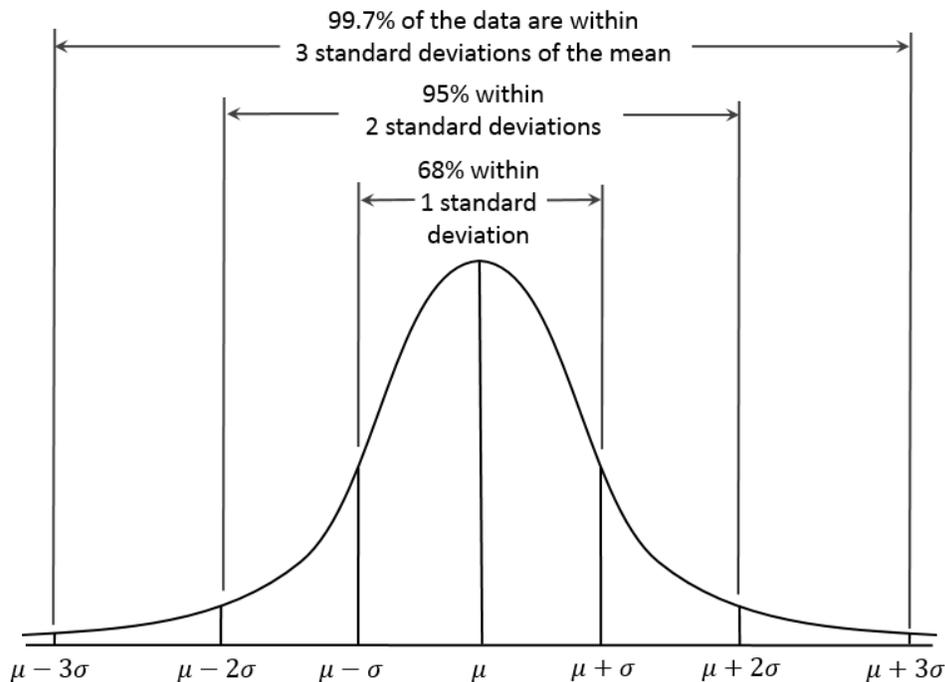
The **Uncertainty** calculation in Thermo-Calc uses the values (the mean) entered for quantities under the *Condition Definitions* section as the starting point to estimate, within a certain probability, the true variation of the quantity. Uncertainty calculations are available with the Property Model Calculator and the Process Metallurgy Calculator.

An example of a Gaussian distribution is shown next. The example uses the [Phase Transition Property Model Settings](#) and an **Uncertainty Calculation Type** to describe how the sampling of data is performed. The example [PM\\_G\\_01: Phase Transition](#) predicts the transition temperature to the unwanted brittle Sigma phase and shows how the temperature is influenced by changes to a steel alloy's composition.

### ***Gaussian Distribution, the 3-Sigma Rule***

- The *mean* is at the peak of the curve (labeled  $\mu$ ).
- The *standard deviation* ( $\sigma$ ) is a plus and minus range,  $\pm 1\sigma$ ,  $\pm 2\sigma$  or  $\pm 3\sigma$ , of the variance from the mean.

Image by Dan Kernler: [https://en.wikipedia.org/wiki/Normal\\_distribution#/media/File:Empirical\\_Rule.PNG](https://en.wikipedia.org/wiki/Normal_distribution#/media/File:Empirical_Rule.PNG).



As long as the number of events (the sample size) is very large, then the Gaussian distribution function can be used to describe physical events. A Gaussian distribution is assumed to describe how each quantity varies in an uncertainty calculation, as shown next.

## Condition Definitions and Sampling Settings

In Thermo-Calc, the mean of the quantity's distribution is defined in the *Condition Definitions* section. The **Mean** is then used in the *Sampling of Data from Gaussian Distributions* section as the starting point for how much you want to deviate from this value.

The mean defined under *Condition Definitions* is used during the sampling.

The image shows two overlapping software panels. The left panel, titled 'Condition Definitions', has a 'Composition unit' dropdown set to 'Mass percent'. It lists several quantities with their mean values: Temperature (700.0), Composition Fe (80.52), Composition Ni (1.4), Composition C (0.08), Composition Mn (1.0), and Composition Cr (17.0). The values 700.0, 1.4, 0.08, 1.0, and 17.0 are highlighted with a red box. The right panel, titled 'Sampling of Data from Gaussian Distributions', has a 'Quantity' column with checkboxes for Temperature, Mass percent Ni, Mass percent C, Mass percent Mn, and Mass percent Cr. The 'Mean' column contains the values 700.0, 1.4, 0.08, 1.0, and 17.0, which are also highlighted with a red box. The 'Δ Min/Max' column contains values 0.0, 0.4, 0.02, 0.3, and 2.0. Below this is the 'Sampling Parameters' section with 'Truncate distribution at' set to 3 standard deviations and 'Total number of samples' set to 40.

The values entered in the  $\Delta$  **Min/Max** fields are the resulting width of the distribution corresponding to the 95% confidence interval (approximately  $\mu \pm 2\sigma$ ) shown in *Gaussian distribution, the 3-sigma rule*. This confidence interval means that approximately 95% of the sampled values fall within these limits, as shown next.

## Sampling Parameter Settings

The default value for truncating the distribution is set at **3 standard deviations** in the *Sampling Parameters* section. This is also where you set the **Total number of samples**. The default is 200. In order to maintain a Gaussian distribution for the varied quantities, this number should not be too low.

---

*Sampling Parameters*

Truncate distribution at  standard deviations

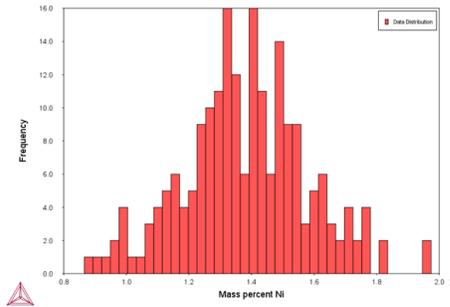
Total number of samples

You can experiment by varying the conditions and the number of samples to see the distribution results smooth out as is to be expected based on the Central Limit Theorem, as shown in the histogram plot. You can also plot the results as a normal probability.

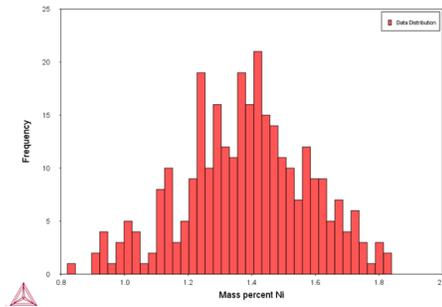
### ***Histogram Plot***

This series of histogram plots shows the uncertainty calculations sampling data from Gaussian distributions. On the **Plot Renderer** the variable is set to measure the **Composition** of **Ni**. It compares the Ni composition where the total sample size is increased. In the example, the initial sample size uses the default, 200, and then the sample size is increased to 300, 400, 600, 800, and 1000, showing how the plot follows the Gauss distribution.

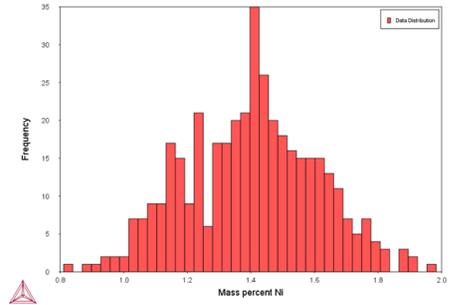
Total samples = 200



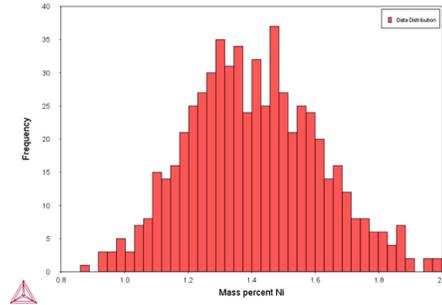
Total samples = 300



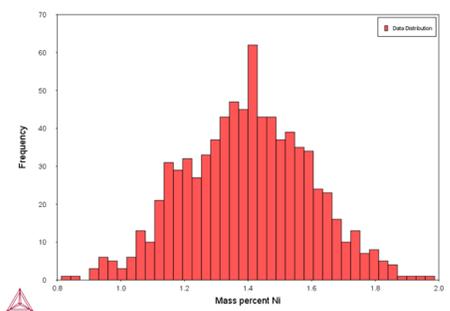
Total samples = 400



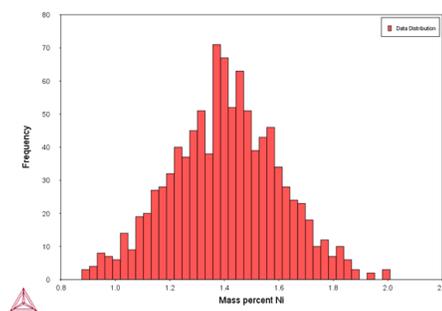
Total samples = 600



Total samples = 800

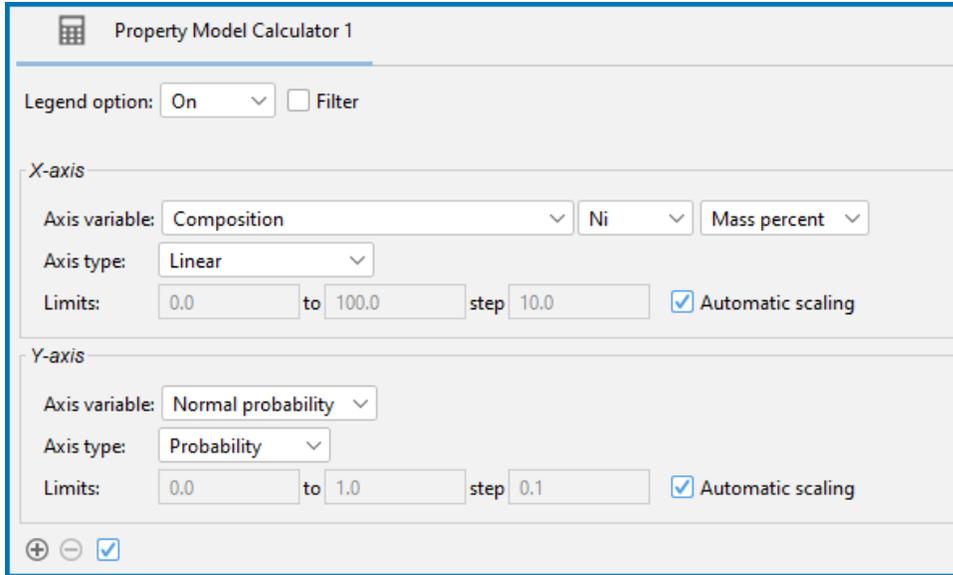


Total samples = 1000



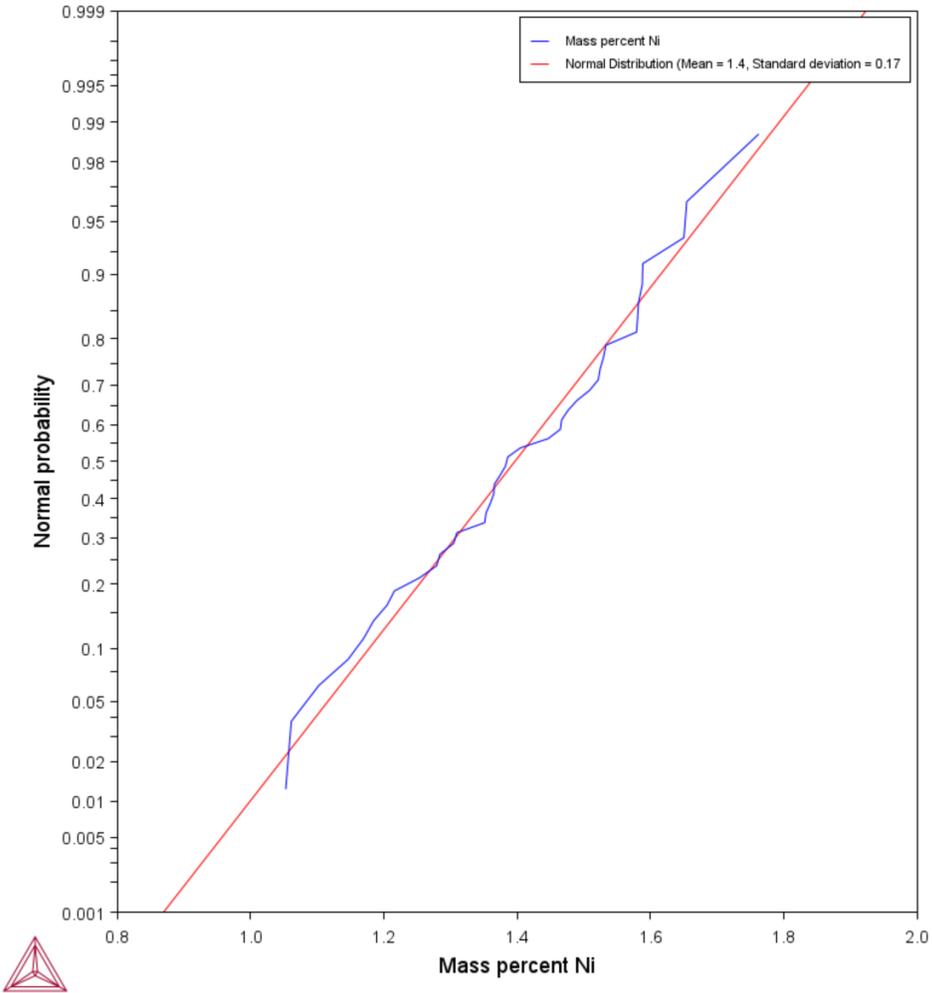
## Normal Probability Plot

You can also plot the results as a **Normal probability**. Continuing with this example, for the Property Model Calculator the X Axis variable is plotted as **Composition** and the Y Axis variable is plotted as the **Normal probability**.



The screenshot shows the 'Property Model Calculator 1' window. At the top, there is a legend option set to 'On' with a 'Filter' checkbox. Below this, the 'X-axis' settings are: 'Axis variable' set to 'Composition', 'Ni', and 'Mass percent'; 'Axis type' set to 'Linear'; and 'Limits' set to '0.0' to '100.0' with a 'step' of '10.0' and 'Automatic scaling' checked. The 'Y-axis' settings are: 'Axis variable' set to 'Normal probability'; 'Axis type' set to 'Probability'; and 'Limits' set to '0.0' to '1.0' with a 'step' of '0.1' and 'Automatic scaling' checked. At the bottom left, there are three icons: a plus sign, a minus sign, and a checked checkbox.

This is a normal probability plot example of the uncertainty calculations sampling data from Gaussian distributions. It compares the relaxed condition where the total sample size is 200. The sampled data (blue line) closely follows the ideal normal distribution (red line). Deviations are naturally larger at the tails ( $\approx 1\%Ni$  and  $\approx 2\%Ni$ ) since the number of samples are fewer there compared to the center of the distribution.



## Working with Batch Calculations

When defining a **Batch** calculation for a Property Model Calculator, you choose to load (import) a configuration data file that contains predefined data. The columns that can be included in the data file are:

- Element compositions and the associated unit
- Temperature and the temperature unit
- Experimental data and a key word
- Parameters (as long as these are both available and selected for a Property Model)
- An identifier (ID) for each row of data



Only these types of data are read / used in calculations. It is important that the data file contents are correctly entered in order for the calculation to perform smoothly. The software runs a validation of the data file prior to allowing you to perform a calculation. Warnings related to this validation display on the Property Model Calculator. You can also skip this data file validation for composition or parameter inconsistencies. See below for details.

The *Batch Control* settings, which are further described in this section, are found on the Property Model Calculator **Configuration** window. See [Configuration Window Settings](#).

### Batch Controls



For the settings in this section, it is recommended you read the [Data File Rules](#), [Data File Requirements](#), and [Skip Validation of Data File](#) sections prior to importing the data file.

In the **Data file** field, either enter a file path or click the load data file button () to navigate to a data file such as an Excel spreadsheet (with \*.xls or \*.xlsx extensions) or a \*.csv file. You can also use other programs (e.g. Google Sheets) to enter data and export to a CSV format that can be read by the software.

Click **Open** to import the data. In the **Data file** field you can see the file path to the selected file. The next time you click the load data file button () the program remembers the last location from where the file is opened.

Choose the delimiter—**Comma (,)**, **Colon (:)**, **Equals sign (=)**, **Semicolon (;)**, or **Tab**—that matches the delimiter in the data file and click the **Reload** () button.

The data is imported and displays on the Property Model Calculator:

- The data are used for the applicable fields that must match what already exists as a Property Model parameter (e.g. critical radius, grain size, etc.) as well as the composition, temperature, and units that you want to run the calculation for. Also, if the dependent element composition is not included in the data file, the batch calculation then sets the dependent element to be the same as defined in the user interface.
- The calculation runs once for every row of data included in the file.



When the data file is imported into Thermo-Calc, there is also a **Row** column autogenerated by Thermo-Calc.

Select the **Set zero amount to value (in mole-/mass-fraction)** to enter a small value for the mole/mass-fraction in cases where a Property Model has problems calculating zero amounts of elements.

Select the **Skip validation of data file** checkbox to allow a calculation to perform using the data file even when certain criteria are not met.

When selected, the validation of the data file is skipped:

- If the **Composition** (elements) do not match the system composition entered on the GUI in the System Definer or Property Model Calculator.
- If one or more **Parameters** are not correctly entered, not selected for the specific Property Model, or are unavailable with the selected Property Model, then these are ignored in the calculation.

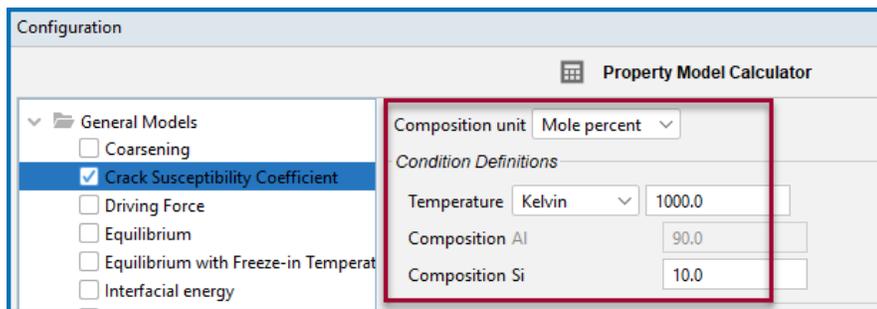


If there are any data file validation error(s), a tooltip on the **Perform** button lists and describes these.

## Data File Rules

### GENERAL RULES

- No abbreviations are allowed.
- All header text must be in English, i.e. no translations into other languages will be read correctly by the data file.
- For the settings on the Property Model Calculator (**Composition unit**, **Composition**, **Temperature**, and **Temperature unit**) when there is nothing specified in the data file, the default is taken from the GUI and used in the calculation. Also, if the dependent element composition is not included in the data file, the batch calculation then sets the dependent element to be the same as defined in the user interface.



### PARAMETER, PARAMETER NAME, AND PARAMETER VALUE RULES



Only numeric parameters available on the **Configuration** tab for the selected Property Model can be included in the calculation.

The column entries for the Parameter + *<parameter name>* plus the row entries for the *<parameter value>* must adhere to these rules. The validation checks for inconsistencies, which could be the result of any of the following criteria not being met.

- When including **Parameters** in the data file, these must be existing Property Model parameters and the applicable Property Model must be selected on the GUI.
- The *<parameter name>* is case sensitive and must be entered exactly as you see on the GUI, no abbreviations are allowed.
- The *<parameter value>* must be numeric or it is not included in the calculation.

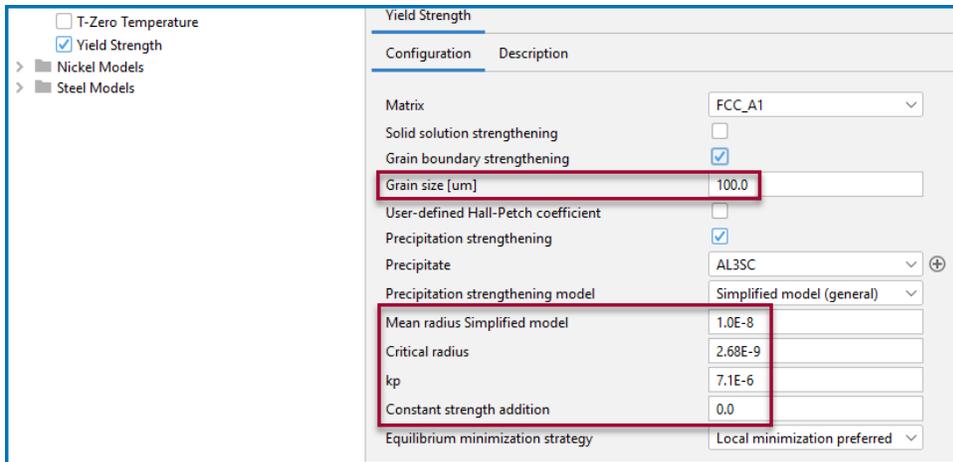


Figure 1: An example of the Yield Strength Property Model numeric parameters available for the Batch calculations. These are available on the Property Model Calculator Configuration tab after selecting Yield Strength under General Models. The parameter name must be entered in the data file exactly as you see on the GUI. No parameters from other calculators are accepted.

## Data File Requirements



Only the columns described here are read / used in the calculation. It is important to enter the information exactly as described. It is recommended that the data file is cleaned up before importing / loading it and performing calculations. All header text must also be written in English, i.e. no translations into other languages will be read correctly by the data file.



In the table below, `code font` indicates something entered into the data file (or it is an example) and any *<text in italic with angle brackets>* is a generic name for an entry.

Column Heading Description	Column Heading Requirements and Options	Row Cell Requirements and Options
<b>Composition</b> Each element of the	Enter an <i>&lt;element short name&gt;</i> . It is not case sensitive. For example: Fe.	Valid format and entries of the <i>&lt;composition value&gt;</i> :

Column Heading Description	Column Heading Requirements and Options	Row Cell Requirements and Options
composition is in a separate heading column using the <element short name> e.g. Fe, C, Mn, Si, etc.	It is not case sensitive, so alternatives in this example are fe or FE.	<ul style="list-style-type: none"> <li>• A number for the composition e.g. 0.1, 8.7, 1.00E-05, etc.</li> <li>• Text entry Bal to define the balanced element.</li> <li>• Empty cells are read as zero (0) amounts.</li> </ul>
<b>Composition unit</b>	<p>Enter the text <code>Composition unit</code>. No abbreviations allowed but it is not case sensitive.</p> <p>For example, it is acceptable to enter <code>composition unit</code> <b>OR</b> <code>Composition Unit</code>, <b>OR</b> <code>COMPOSITION UNIT</code>, etc.</p>	<p>Valid format and entries of the &lt;composition unit&gt;:</p> <ul style="list-style-type: none"> <li>• <code>mass_pct</code></li> <li>• <code>mass_frac</code></li> <li>• <code>mole_pct</code></li> <li>• <code>mole_frac</code></li> </ul>
<b>Temperature</b>	<p>Enter the text <code>Temperature</code>. No abbreviations allowed but it is not case sensitive.</p> <p>For example, it is acceptable to enter <code>temperature</code> <b>OR</b> <code>Temperature</code>, <b>OR</b> <code>TEMPERATURE</code>.</p>	<p>Valid format and entries –</p> <ul style="list-style-type: none"> <li>• Any number e.g. 273.15</li> </ul>
<b>Temperature unit</b>	<p>Enter the text <code>Temperature unit</code>. No abbreviations allowed but it is not case sensitive.</p> <p>For example, it is acceptable to enter <code>temperature unit</code> <b>OR</b> <code>Temperature Unit</code>, <b>OR</b> <code>TEMPERATURE UNIT</code>, etc.</p>	<p>Valid format and entries –</p> <ul style="list-style-type: none"> <li>• C</li> <li>• F</li> <li>• K</li> </ul>
<b>Parameters with numerical values only.</b> The heading column is defined with <code>Param</code> in front of a Property Model <parameter name>	<p><code>Param + &lt;parameter name&gt;</code></p> <p>For example: <code>Param Critical radius</code></p> <p>The &lt;parameter name&gt; has requirements–</p> <ul style="list-style-type: none"> <li>• It is case sensitive based on the available and selected Property Model parameter</li> <li>• It must be an existing Property Model parameter and be included with any selected Property Model</li> <li>• No abbreviations</li> </ul>	<p>Valid format and entries –</p> <p>The &lt;parameter value&gt; must be numeric or it is not included in the calculation.</p> <p>An example of an accepted Property Model parameter is for the <b>Yield Strength</b> model where <i>Grain size</i> and <i>Critical radius</i> are accepted (these have fields for numerical entries) but you cannot use <i>Matrix</i> or <i>Precipitate</i> as these are text choices from a drop down list.</p>

Column Heading Description	Column Heading Requirements and Options	Row Cell Requirements and Options
<b>Experimental value.</b> The heading column is defined with <code>Exp</code> followed by any <code>&lt;keyword&gt;</code> .	<code>Exp</code> + <code>&lt;keyword&gt;</code> where the <code>&lt;keyword&gt;</code> is defined by the user.  For example: <code>Exp Ms</code> . The keyword is not case sensitive so alternatives are <code>MS</code> or <code>ms</code> .	Valid format and entries – <ul style="list-style-type: none"> <li>Each <code>&lt;experimental value&gt;</code> must be numeric</li> </ul>
<b>Identifier</b> (i.e. an alloy name). The heading column is defined as <code>Id</code>	Enter the text <code>Id</code> .  It is not case sensitive so alternatives are <code>id</code> or <code>ID</code> .	Valid format and entries – <ul style="list-style-type: none"> <li>An <code>&lt;identifier&gt;</code> can be any text or number defined by the user.</li> </ul>

## DATA FILE EXAMPLE AND COLUMN DEFINITIONS

	A	B	C	D	E	F	G	H	I
1	Id	Fe	C	Cr	Exp Ms C	Exp Ms	Composition unit	Temperature unit	Param Critical radius
2	2	Bal	0.000928	0.0149	529.85	803	mole_frac	K	2.88E-09
3	3	Bal	0.000928	0.0449	489.85	763	mole_frac	K	3.88E-09
4	4	Bal	0.000324	0.0422	499.85	773	mole_frac	K	4.88E-09

This is an example of data entered into an Excel spreadsheet. When this is imported into Thermo-Calc, an additional "Row" column is automatically added and is a way for you to locate any errors in the data file that are identified as needing correction or adjustment to allow the program to read the file.

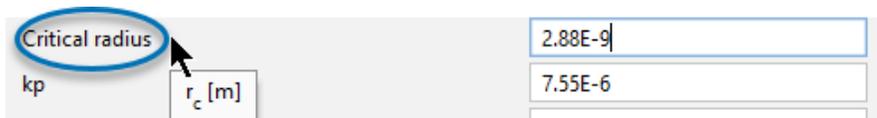
- A1 to A4 is the **Identifier** column and rows. The heading text is `Id` and then in the entries in rows A2 to A4 can be text or numbers, for example it might be an alloy name or another user-defined `<identifier>`.



When the data file is imported into Thermo-Calc, there is also a **Row** column autogenerated by Thermo-Calc.

- B1 to B4 is the **Composition** column and rows for the element `Fe`. In this example, `Bal` indicates that the balance of the composition will be calculated for this column after the `<composition values>` for `C` and `Cr` are entered.
- C1 to C4 and D1 to D4 are the **Composition** columns each for `C` and `Cr` with the specific `<composition values>`, e.g. 0.000928 for `C` and 0.0149 for `Cr` in row 2.

- E1 to E4 and F1 to F4 are the **Experimental Value** columns and rows. The header starts with `Exp` and then user-defined `<keywords>` are added, e.g. `Ms C` and `Ms`. Then specific `<experimental values>` are entered, e.g. `529.85` for `Ms C` and `803` for `Ms` in row 2.
- G1 and H1 are the headings for the **Composition unit** and **Temperature unit** columns. No abbreviations allowed but it is NOT case sensitive. Then the rows G2 to G4 and H2 to H4 include an accepted text value e.g. `mole_frac` for the composition unit and `K` for the temperature unit.
- I1 is the **Parameter** heading where you need to enter `Param` followed by the parameter name `Critical radius`, which has numeric values entered in rows I2 to I4. The `<parameter name>` matches that on the GUI and is an available and selected Property Model parameter.



### Skip Validation of Data File

Select the **Skip validation of data file** checkbox to allow a calculation to perform using the data file even when certain criteria are not met.

When selected, the validation of the data file is skipped:

- If the **Composition** (elements) do not match the system composition entered on the GUI in the System Definer or Property Model Calculator.
- If one or more **Parameters** are not correctly entered, not selected for the specific Property Model, or are unavailable with the selected Property Model, then these are ignored in the calculation.



If there are any data file validation error(s), a tooltip on the **Perform** button lists and describes these.

## Skipping Composition Validation

If the **Composition** (elements) do not match the system composition entered on the GUI in the System Definer or Property Model Calculator then the default system composition is used for any missing elements after you select the **Skip validation of data file** checkbox.

This can occur when there are more elements included in the data file but not entered on the GUI, and then these are excluded from the calculation. For example, you might have too many or too few elements included in the data file compared to the composition entered on the System Definer, which is then further viewed under *Condition Definitions* on the Property Model Calculator **Configuration** window.



When configuring the System Definer, it is recommended you select the composition to match the composition in the data file. However, if this is not possible you can use this checkbox to override any differences of the composition included in the data file and use the GUI default.

## Skipping Parameter Validation

If one or more **Parameters** are not correctly entered, not selected for the specific Property Model, or are unavailable with the selected Property Model, then these are ignored in the calculation after you select the **Skip validation of data file** checkbox.

For example, if a data file is uploaded and you are unable to perform the calculation, it means that one of the following criteria are not met for the `Param` columns and related to the `<parameter name>`.

- The parameter name is misspelled (does not match) the one on the GUI
- The parameter is not available with the selected Property Model.
- The parameter is not available with any Property Model.



It is best practice to use a data file that only includes available Property Model parameters. However, if this is not possible then use this checkbox to override the validation. Alternatives to skipping this validation, include removing the prefix `Param` from the data file (or delete this column), and confirming that the parameter you want to calculate is available and selected for the Property Model.

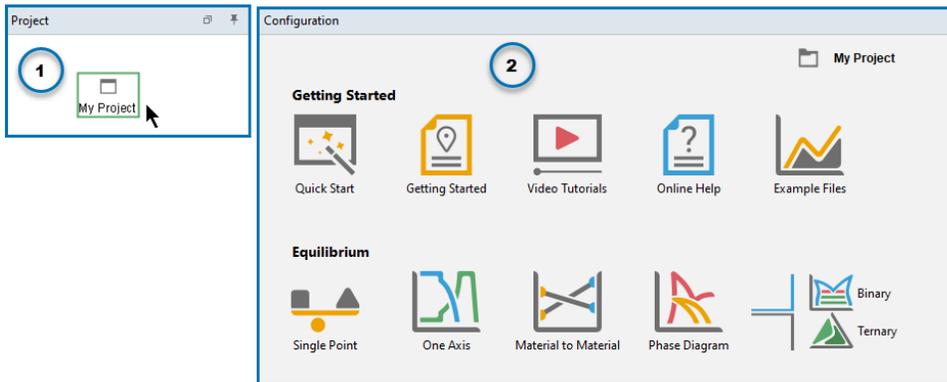
## ***Plotting Experimental Data Against Simulation Results***

You can plot experimental values loaded to the Property Model Calculator using the **Cross plot** *Plot type* available on the Plot Renderer. There are also additional options for this *Calculation Type*. See [Plot Types](#) for a list and also [Plot Type: Cross Plot and Statistical](#).

## Getting Started Links and Templates

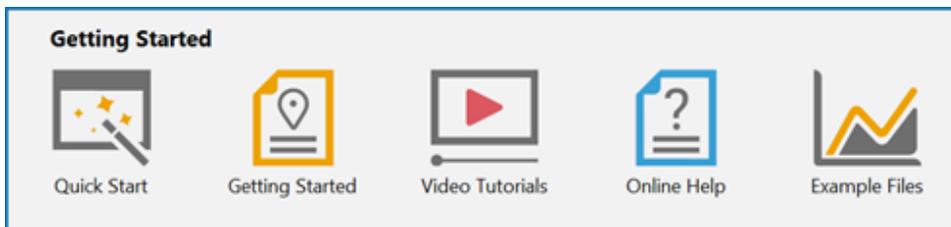
When you first open Thermo-Calc, the **My Project Configuration** window has a variety of links and templates available to get you started. From here, click any template button to add the relevant nodes to a Project tree. You can also right-click the **My Project** node and select a template from the **Create New Activity → Use Template** list.

1. In the **Project** window, click the **My Project** node.
2. The templates (plus additional links under **Getting Started**) are available to click in the **My Project Configuration** window.



### **Getting Started Links Including the Quick Start Wizard**

Even before you understand how to work with Thermo-Calc projects or use the available templates, there are some links of interest available under **Getting Started**.



- Click [Getting Started](#) to go to the *Getting Started Guides* on the website.
- Click [Video Tutorials](#) to go to the Video Tutorials on the website.
- Click **Online Help** to open the help locally in a browser (or press F1).
- Click **Example Files** to open the Graphical Mode folder containing the installed example files.

In addition to the links above, click **Quick Start** to open the template wizard, which helps you set up a project and run calculations in Graphical Mode. You can set up, calculate and visualize any of the following:

- Single point equilibrium
- One axis equilibrium
- Phase diagram
- Scheil solidification simulation

 [Creating a Project with the Quick Start Wizard](#)

## Equilibrium Templates



Under **Equilibrium**, click a template to create a project and add the applicable nodes to the **Project** window.

- **Single Point:** To calculate a single point equilibrium. The project consists of a System Definer, Equilibrium Calculator, and a Table Renderer.
- **One Axis:** To calculate a one axis diagram. The project consists of a System Definer, Equilibrium Calculator, and a Plot Renderer.
- **Material to Material:** To examine how materials transition into one another. The project consists of a System Definer, Material to Material Calculator, and a Plot Renderer.
- **Phase Diagram:** To calculate a phase diagram. The project consists of a System Definer, Equilibrium Calculator, and a Plot Renderer.

- **Binary:** To perform a binary calculation. The project consists of a Binary Calculator and a Plot Renderer.
- **Ternary:** To perform a ternary calculation. The project consists of a System Definer, Ternary Calculator, and a Plot Renderer.

## Non-Equilibrium Templates



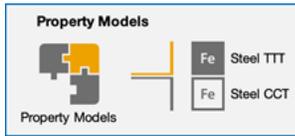
Under **Non-Equilibrium**, click a template to create a project and add the applicable nodes to the **Project** window.

- **Scheil Solidification:** To perform a Scheil calculation. The project consists of a System Definer, Scheil Calculator, and a Plot Renderer.
- **Homogenization:** To perform a Scheil solidification calculation followed by kinetic homogenization simulation of segregated composition profile. The project consists of a System Definer, a Scheil Calculator, a Diffusion Calculator, and two Plot Renderers.
- **Diffusion:** To calculate general diffusion simulations. The project consists of a System Definer, Diffusion Calculator, and a Plot Renderer.
- **Precipitation:** To simulate the evolution of particle size distributions. The project consists of a System Definer, Precipitation Calculator and a Plot Renderer.



To run full calculations with the **Homogenization**, **Diffusion** and **Precipitation** templates require additional licenses although you can run these in demo mode with up to three (3) components. See [Available Graphical Mode Features](#) and [Available Options](#).

## Property Models



Under **Property Models**, click a template to create a project and add the applicable nodes to the **Project** window.

- **Property Models:** To choose from a variety of Property Models available on a Property Model Calculator. The project consists of a System Definer, Property Model Calculator, Plot Renderer, and a Table Renderer.
- **Steel TTT:** To use the **TTT Diagram** Property Model as part of the Steel Model Library and add all the activities for a Time-Temperature-Transformation (TTT) diagram calculation. The project consists of a System Definer, Property Model Calculator, and a Plot Renderer.
- **Steel CCT:** To use the **CCT Diagram** Property Model as part of the Steel Model Library and add all the activities for a Continuous-Cooling-Transformation (CCT) diagram calculation. The project consists of a System Definer, Property Model Calculator, and a Plot Renderer. Also see [Steel Library TTT Diagram Template](#) and [Steel Library CCT Diagram Template](#).



To run full calculations with the **Steel TTT** and **Steel CCT** templates (as part of the Steel Model Library) requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases.

## Applications



Under **Applications**, click a template to create a project and add the applicable nodes to the **Project** window.

- **Process Metallurgy:** To allow for calculation and efficient setup of advanced calculations involving slag, metal, and gas. Also see [Process Metallurgy Template](#).



To run full calculations with the **Process Metallurgy** template requires both a valid Maintenance and Support Subscription (M&SS) and a license for the TCS Metal Oxide Solutions Database (TCOX8 or newer).

- **Additive Manufacturing and With Material Library:** These templates are to work with additive manufacturing. The Additive Manufacturing (AM) Module is used to better understand the laser powder bed fusion (LPBF) process by predicting the temperature distribution and melt pool geometry as a function of process parameters. Also see [Additive Manufacturing Templates](#).



To run full calculations with the **Additive Manufacturing and With Material Library** templates require a license for the Additive Manufacturing (AM) Module.

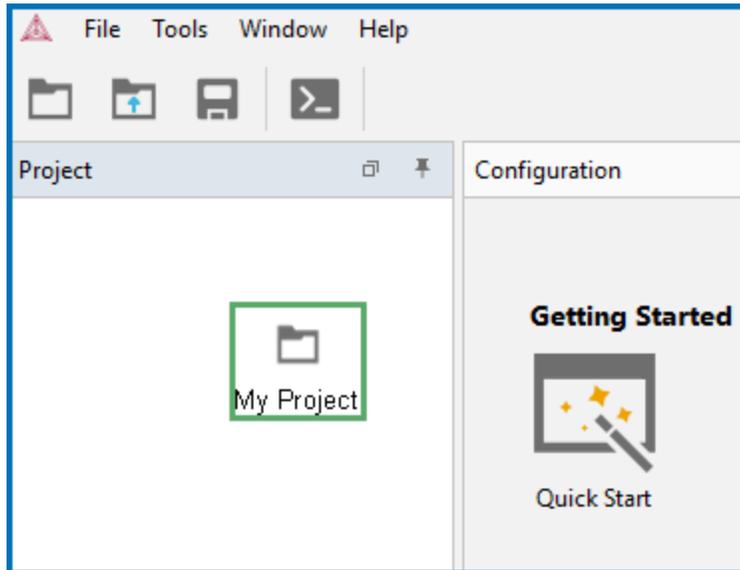


When you also have licenses either for the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA), for transient simulations you can additionally include probe data from the AM Calculator in the set up of the thermal profile for diffusion and precipitation calculations, respectively.

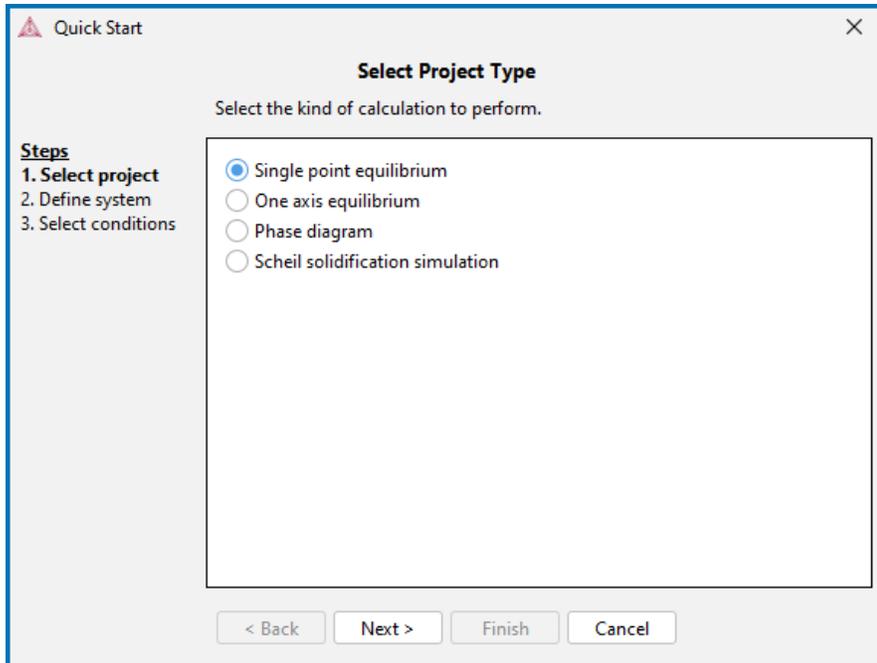
## Creating a Project with the Quick Start Wizard

To set up, perform and visualize a calculation using the Quick Start wizard:

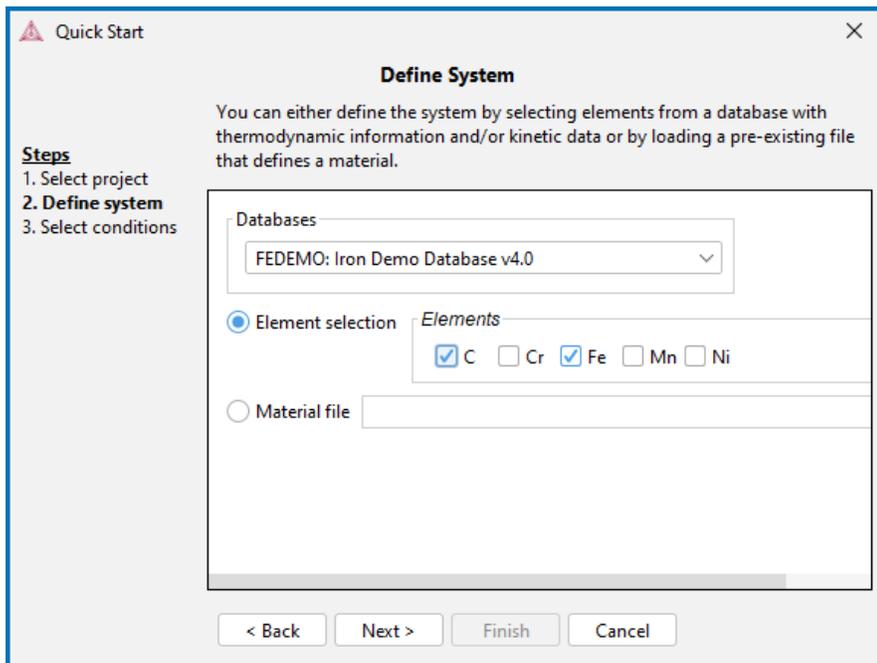
1. In the **Configuration** window click **Quick Start**. If you cannot see the **Quick Start** button, first click  **My Project** in the **Project** window.



2. In the **Select Project Type** window click to select the type of calculation to perform. **Single point equilibrium**, **One axis equilibrium**, **Phase diagram**, or **Scheil solidification simulation** (a Classic Scheil simulation). Click **Next**.

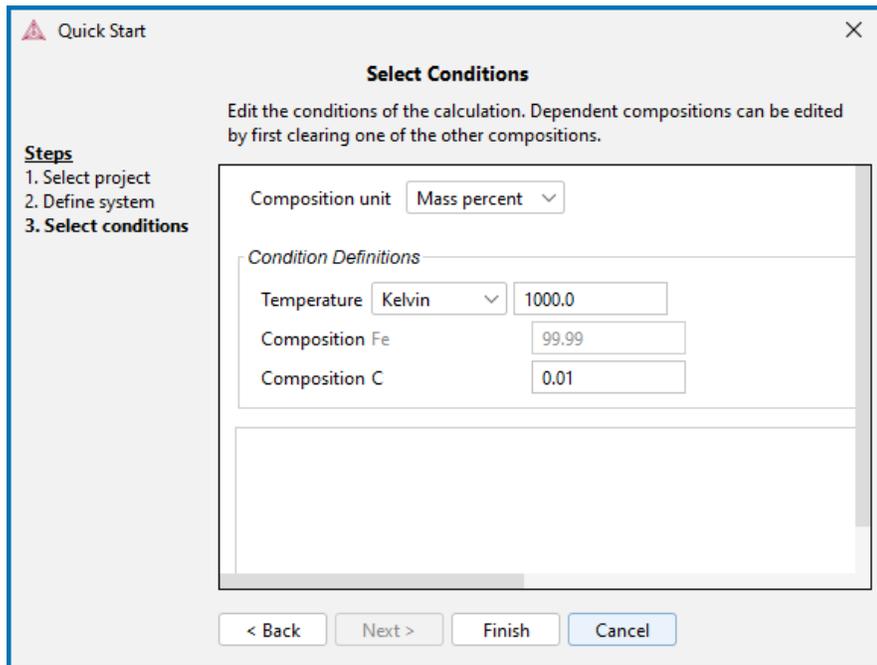


3. In the **Define Systems** window, choose an available **Database**.



- Click to select **Elements** checkboxes to include as system components. To add elements from an existing material file, click the **Material file** button to enter the file path to a material file or then click the **Select Material File**  button (to the right of the field) to navigate to a file on the computer.
- Click **Next**. The **Select Conditions** window has different options depending on the project type and the elements selected.

In this example figure, using a **Single Point Equilibrium**, choose the **Composition unit** from the list and under *Condition Definitions* specify the values of the state variables in the system.



For **One axis equilibrium** or **Phase diagram** projects, specify the **Axis Definitions** of the stepping or mapping operation. Choose minimum and maximum values for the variable(s) (select **Linear – min no. of steps**, **Linear – max step size**, or **Logarithmic – scale factors**) and whether it is **Normal** or **Separate phases**.

For a **Scheil solidification simulation** project, and where applicable, select **Fast diffuser**.



After using this Quick Start Wizard to do a Classic Scheil calculation (or with fast diffuser components), a Scheil Calculator is added to the Project window. At this point you can review [Scheil-Gulliver Solidification Calculations](#) for details about all the options using the Scheil Calculator, i.e. back diffusion and solute trapping.

- Click **Finish**. Click to expand the **Event Log** window to see information about the progress of the calculation.

Event Log		
14:02:00,441	INFO	ScheduledJob: The activity System Definer 1 executed in 5927 ms
14:02:00,445	INFO	GeneralEquilibriumCalculator: Performing general equilibrium calculation
14:02:00,613	INFO	AbstractQuantity: condition T the value 1000.0
14:02:00,613	INFO	AbstractQuantity: condition P the value 100000.0
14:02:00,614	INFO	AbstractQuantity: condition N the value 1.0
14:02:00,614	INFO	AbstractQuantity: condition W(C) the value 1.0E-4
14:02:02,076	INFO	ProcessExecutorAllocator: The subprocess completed normally
14:02:02,564	INFO	ScheduledJob: The activity Equilibrium Calculator 1 executed in 2122 ms
14:02:02,649	INFO	ScheduledJob: The activity Table Renderer 1 executed in 85 ms

The diagram is plotted in a **Plot Renderer** tab in the **Visualizations** window. If you have calculated a single-point equilibrium, then a **Table Renderer** tab shows information about the equilibrium.

Visualizations				
Table Renderer 1				
<b>System</b>				
Moles	1.00000			
Mass	55.82663	[g]		
Temperature	1000.00000	[K]		
Total Gibbs Energy	-42264.29819	[J]		
Enthalpy	24732.95322	[J]		
Volume	7.31246E-6	[m <sup>3</sup> ]		
<i>Component</i>	<i>Mole Fraction</i>	<i>Mass Fraction</i>	<i>Activity</i>	<i>Potential</i>
C	0.00046	0.00010	0.14013	-16339.51682
Fe	0.99954	0.99990	0.00619	-42276.35352
<b>Stable Phases</b>				
	<i>Moles</i>	<i>Mass</i>	<i>Volume Fraction</i>	
<b>BCC_A2#1</b>	1.00000	55.82663	1.00000	Composition ▾
<b>Composition</b>				
<i>Component</i>	<i>Mole Fraction</i>	<i>Mass Fraction</i>		
Fe	0.99954	0.99990		
C	0.00046	0.00010		

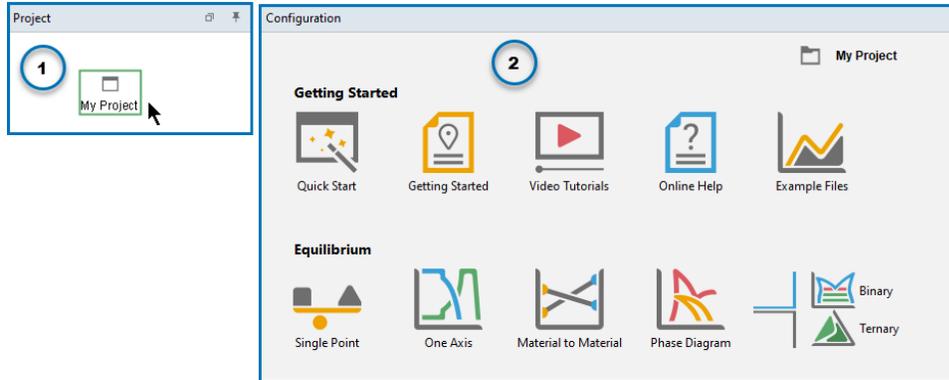
- To save the project, click **Save**  on the main toolbar. To save a diagram or table, right-click the diagram or table and select **Save As**.

## Creating a Project from a Template

Use a *Template* to create all the activities for a certain type of calculation.

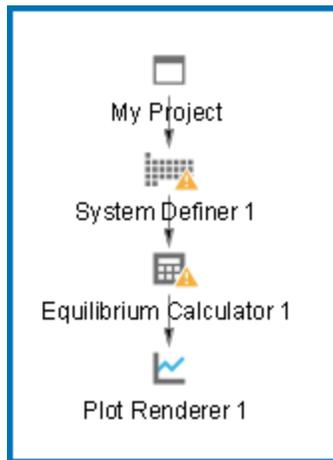
### [Getting Started Links and Templates](#)

- In the **Project** window, click the  **My Project** node.
- The templates (plus additional links under **Getting Started**) are available to click in the  **My Project Configuration** window.

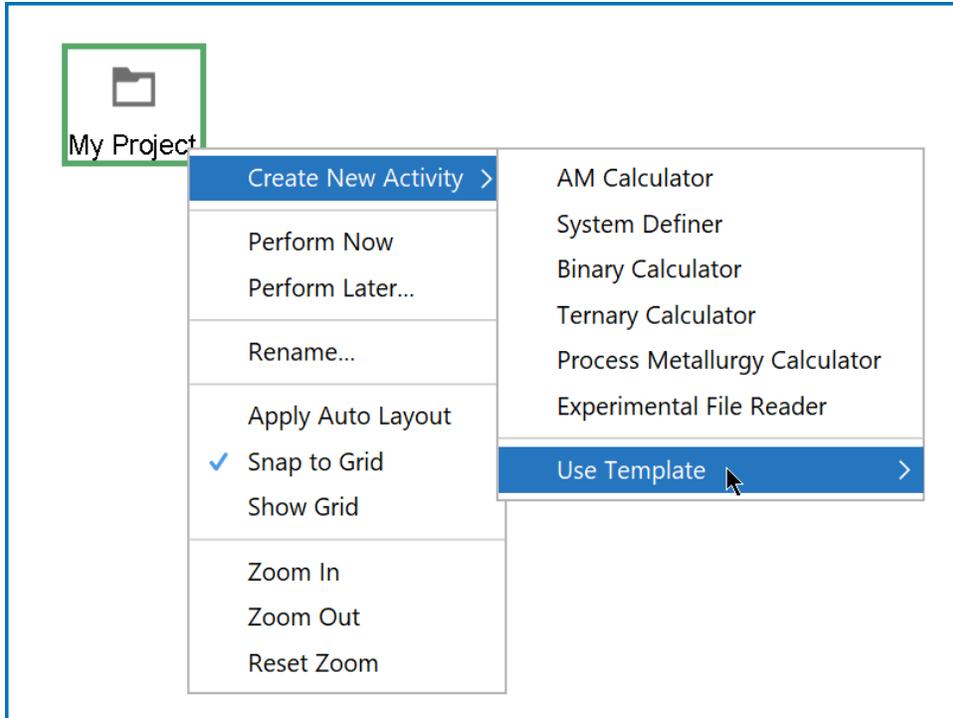


When you click a template, an activity tree is added to the **Project** window. You can then configure and perform the activities or the project.

For example, if under **Equilibrium** you click **One Axis**, a tree is added to the **My Project** node with **System Definer**, **Equilibrium Calculator**, and **Plot Renderer** nodes.



You can also add templates by right-clicking the **My Project** node and choosing an option from the **Create New Activity** → **Use Template** menu.



## Steel Library TTT Diagram Template



The use of this Steel Model Library template requires a valid maintenance license plus licenses for both the TCFE (version 9 and higher) and MOBFE (version 4 and higher) databases.

Use a **Steel TTT** template to add all the activities for a Time-Temperature-Transformation (TTT) diagram calculation. The **TTT Diagram** Property Model is automatically selected with all settings by default on the Property Model Configuration window. This makes it easy to add the project to the tree and examine the progress of transformation as a function of time at a fixed temperature.

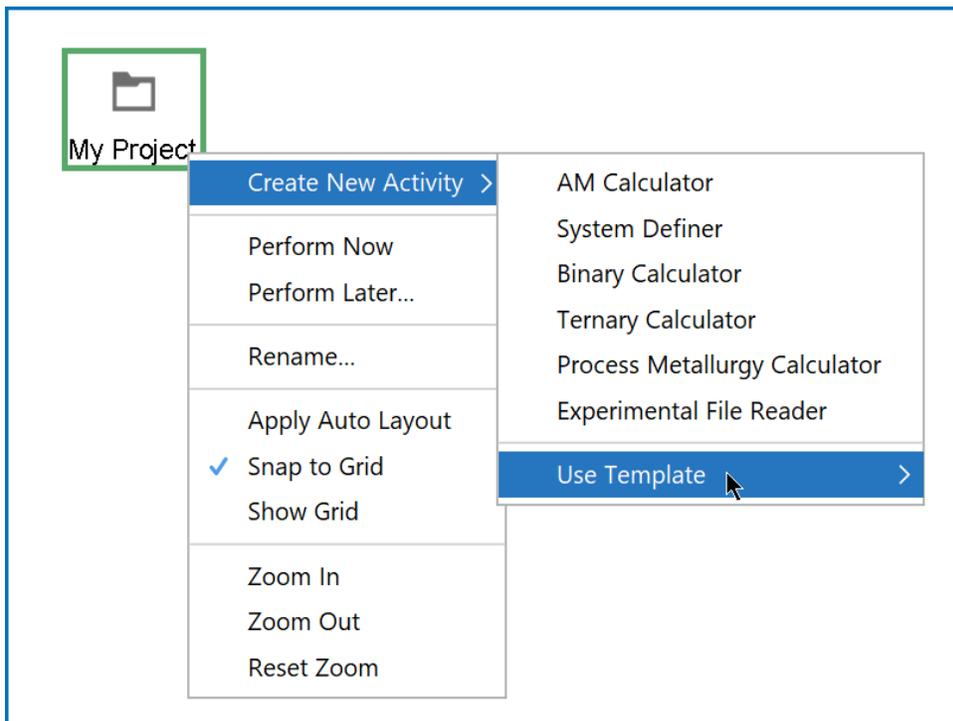
### ***How to Add a Steel Library TTT Template to a Project***

1. In the **Project** window click the  **My Project** node to display the templates in the **Configuration** window.
2. Under **Property Models**, click the top icon **Steel TTT**.



A **System Definer**, *Property Model Calculator* (renamed to **Steel Models**) and a *Plot Renderer* (renamed to **TTT**) are added to the **Project** window. Also see [Property Model Calculator](#) and [Plot Type: TTT Mode](#).

You can also add templates by right-clicking the  **My Project** node and choosing an option from the **Create New Activity** → **Use Template** menu.



## Steel Library CCT Diagram Template



The use of this Steel Model Library template requires a valid maintenance license plus licenses for both the TCFE (version 9 and higher) and MOBFE (version 4 and higher) databases.

Use a **Steel CCT** template to add all the activities for a Continuous-Cooling-Transformation (CCT) diagram calculation. The **CCT Diagram** Property Model is automatically selected on the

Property Model Calculator with all settings by default. This makes it easy to add the project to the tree and examine the progress of transformation as a function of changing temperature.

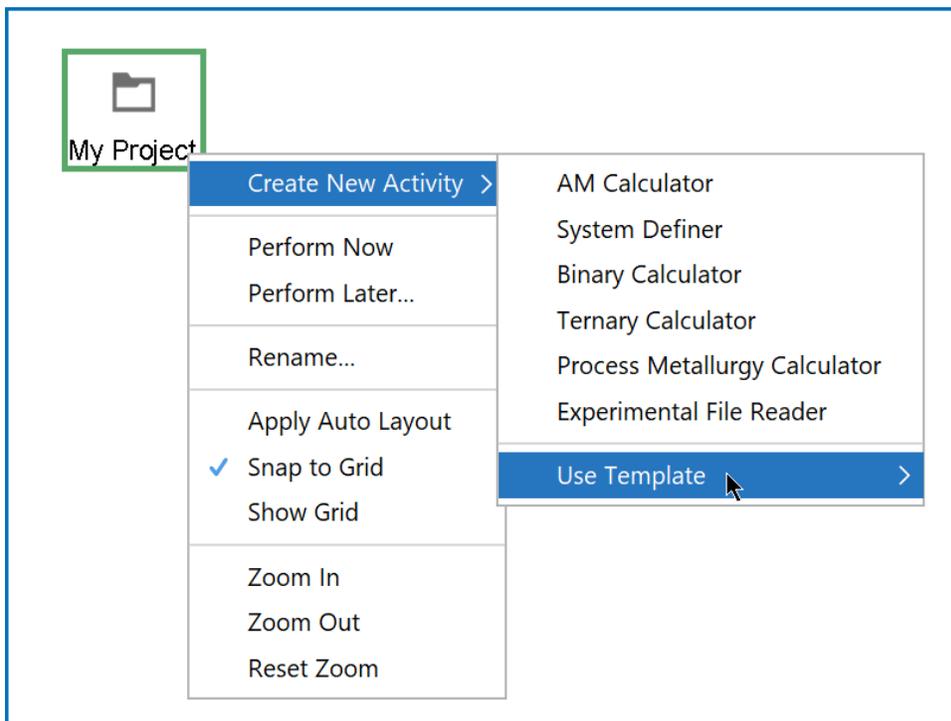
### How to Add a Steel Library CCT Template to a Project

1. In the **Project** window click the  **My Project** node to display the templates in the **Configuration** window.
2. Under **Property Models**, click the bottom icon **Steel CCT**.



A **System Definer**, *Property Model Calculator* (renamed to **Steel Models**) and a *Plot Renderer* (renamed to **CCT**) are added to the **Project** window. Also see [Property Model Calculator](#) and [Plot Type: CCT Mode](#).

You can also add templates by right-clicking the  **My Project** node and choosing an option from the **Create New Activity** → **Use Template** menu.



## Process Metallurgy Template

Use a *Process Metallurgy* template to add the Process Metallurgy Calculator and a Plot Renderer to the **Project** window. The Process Metallurgy Calculator has several preconfigured settings making it easier to setup the different groups and materials needed to define a system for slag, metal and gas used in steelmaking.

 [Process Metallurgy Calculator](#)

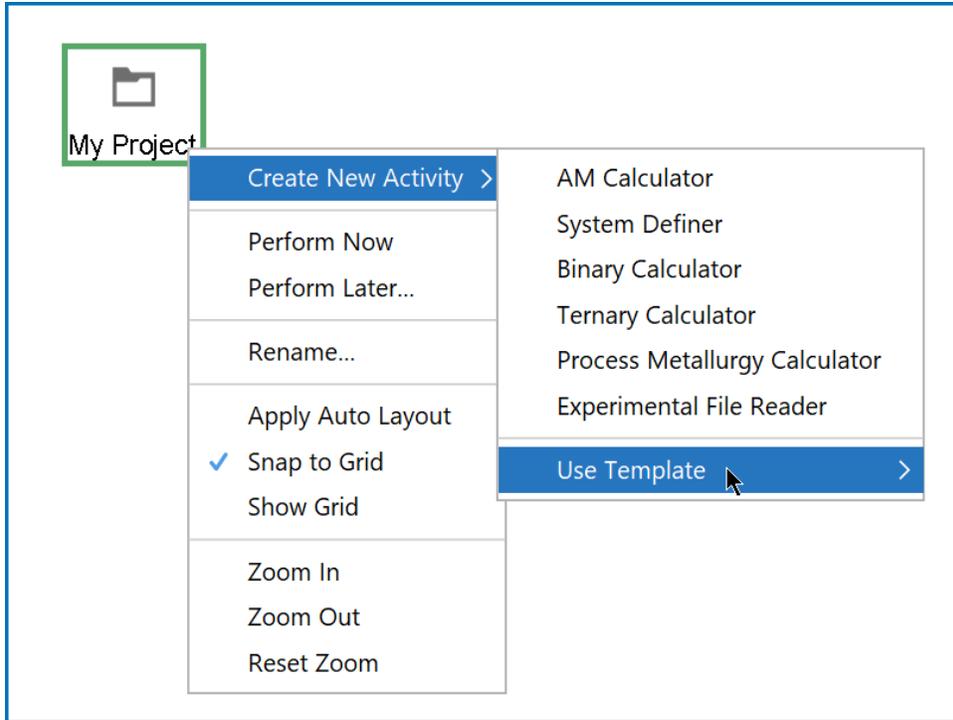
### ***How to Add a Process Metallurgy Template to a Project***

1. In the **Project** window click the  **My Project** node to display the templates in the **Configuration** window.
2. Under **Applications**, click **Process Metallurgy**.



A **System Definer**, **Process Metallurgy Calculator**, and a **Plot Renderer** are added to the **Project** window.

You can also add templates by right-clicking the  **My Project** node and choosing an option from the **Create New Activity** → **Use Template** menu.



# Defining and Using the Activity Nodes

---



For information about the AM Calculator, see [Additive Manufacturing \(AM\) Module](#)



For information about the Process Metallurgy Calculator, see [The Process Metallurgy Module](#).

## In this section:

---

System Definer .....	85
Equilibrium Calculator .....	103
Material to Material Calculator .....	116
Binary Calculator .....	123
Ternary Calculator .....	126
Scheil-Gulliver Solidification Calculations .....	129
Property Model Calculator .....	153
Diffusion Calculator .....	166
Precipitation Calculator .....	204
Grain Growth Settings .....	212
General Rules .....	225
Accessing the 'Pre-existing Size Distribution' Settings .....	227
Defining the Preexisting Size Distribution .....	228
Plot Renderer .....	242
Table Renderer .....	249
Experimental File Reader .....	257

## System Definer

---

In a **System Definer** activity, you select the database to use to retrieve thermodynamic data and define which elements the system has as components. You can also select the species to include as well as change the reference temperature and pressure for your components.

You can add the System Definer directly (right-click  **My Project** and select it from the **Create New Activity** menu) or use a variety of the available templates to automatically add the node.



Many examples use the System Definer (see [Graphical Mode Examples List by Product](#)). For details about the templates, see [Getting Started Links and Templates](#) and [Creating a Project from a Template](#).

Once you have added a System Definer, the **Configuration** window has these settings tabs:

- Elements
- Species
- Phases and Phase Constitution
- Components
- Data Sources
- Description

## Define the System

This topic describes how to define a system, specifically how to select one or more databases and define the elements in the system with the **System Definer** activity node.

1. Right-click  **My Project** and from the menu select **Create New Activity → System Definer**. For many templates a System Definer is also added to the Project tree.
2. In the **Project** window, click the **System Definer** activity node.
3. In the **Configuration** window, choose **Databases** from the list or a database **Package**.

 [About the Database Files](#)

### OPTIONS:

- From the **Databases** list, select **<User database>** to define your own.
  - You can choose to add one or more databases by clicking the **Add a database**  button.
  - From the **Package** list, choose a predefined database package to add multiple databases at a time. Depending on your license, there are predefined database packages available to select.
4. Select the **Elements** tab to choose the elements in the **Periodic Table** or use the **Alphabetic List** to choose elements for the system.
  5. As required click the tabs to make edits: **Species**, **Phases and Phase Constitution**, and **Components**. The **Data Sources** and **Description** tabs provide information based on the database.

 [Configuration Window Settings](#)

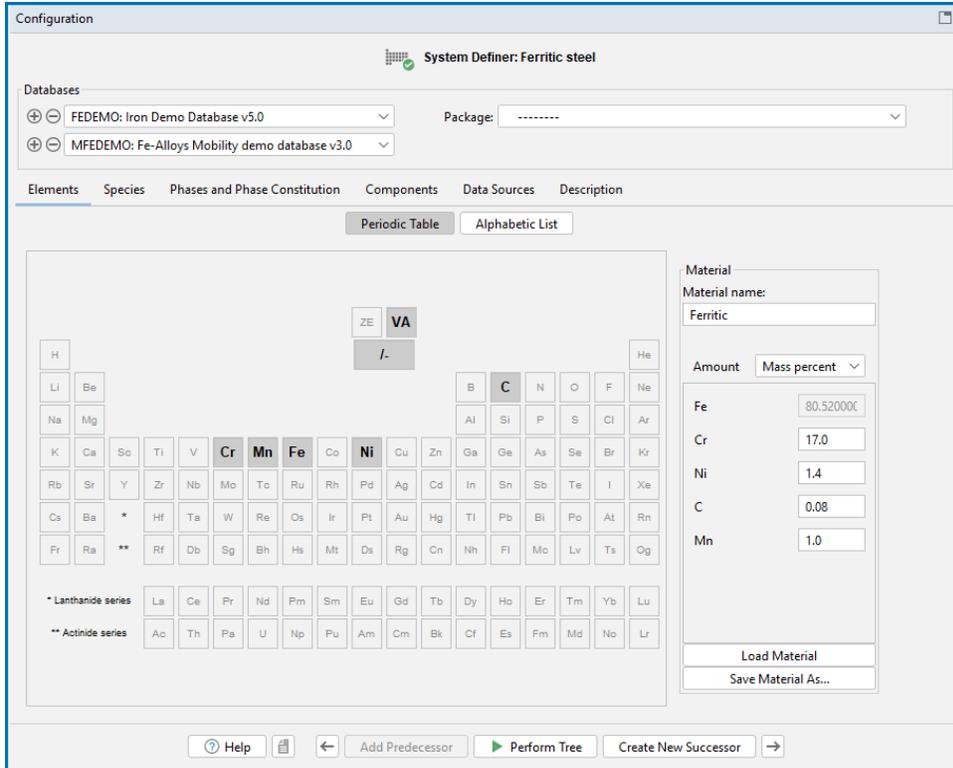
6. Click **Perform** to start the System Definer activity immediately and retrieve data about the system. Click the **Event Log** window to watch the progress of the job.

## Configuration Window Settings

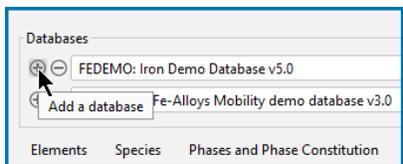
The **System Definer** Configuration window has many settings available, which are briefly described below.



This uses [PM\\_G\\_02: Coarsening and Interfacial Energy](#) to show these features.



## Choose Database(s) or a Database Package



Package: Demo: Steels and Fe-alloys (FEDEMO, MFEDEMO)

At the top of the **Configuration** window you can choose to add one or more **Databases** by clicking the **Add a database**  $\oplus$  button and choosing it from the list of available databases. You can also choose a predefined or custom database **Package** from the list. Depending on your license, there are predefined databases and packages available to select. After adding a database, click **Remove this database**  $\ominus$  button as required for your simulation.



If you have chosen a database package and then remove or change a database, a message displays *Selected databases and selected package may differ*. This means that the original package you selected, which added preset databases, does not match the databases you are using for the project. For information about how to clear the message, see [Global Settings: System Definer](#).

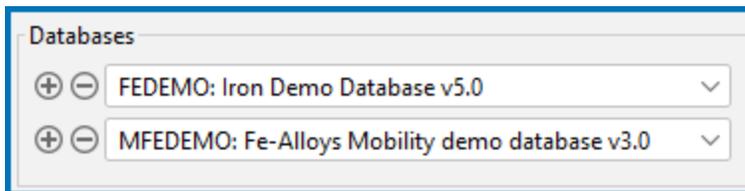


### [Global Settings: System Definer](#)

There are several settings tabs available for the System Definer activity.



The following example uses the FEDEMO and MFEDEMO databases and Graphical Mode example **PM\_G\_02**, which you can open from the **Help** → **Examples Files** menu.



## **Elements**

On the **Elements** tab, and based on the databases or database package selected, you can select and review elements using a **Periodic Table** or **Alphabetic List** format. Unavailable elements, which are based on the selected database, are greyed out.

### **Elements: Advanced Database Options**

Also on the **Elements** tab, there three special elements used in the databases.

On the **Periodic Table** view, there are three buttons. For the **Alphabetic List**, the buttons are available each as a checkbox. In both cases you toggle ON (click to select the button or checkbox) or toggle OFF (click to deselect the button or checkbox, in other words, you are *rejecting* the phase).



From this tab, you are choosing to include or reject the *whole system of phases*. Click the **Phase and Phase Constituent** tab if you want to select specific phases to reject.



This is an advanced feature. The details of these options are described for Console Mode and generally in the *Database Manager User Guide*. For example, see [ELEMENT](#) and [POURBAIX\\_DIAGRAM Module](#). For the purposes of Graphical Mode, the use of the buttons or checkboxes either include or reject (exclude) the vacancies (**VA**) or charged species (**/-**). It is recommended to keep the default after selecting the database(s). In general, these three special elements should not be rejected from the System Definer (Graphical Mode) or GES (Console Mode).



There is a training module related to this feature available via the Thermo-Calc online [Learning Hub](#). Subscriptions also include access to live monthly office hours, where you can ask our experts about any of the content on the Learning Hub or your own design simulation challenges.

- **VA**. This allows a vacancy as a species. Toggled ON to include all vacancies or OFF to reject all vacancies in the phase constitution. VA itself is electrical neutral. It is usually a constituent of a phase. The element VA does not take any degree of freedom in an equilibrium calculation.
- **/-**. This represents ELECTRON\_GAS with -1 net charge. It is implemented in the database which includes ionic-modeled phases, i.e. with net-charged constituents. The charged species can be formed by combining **/-** (for negative charge) or **/+** (for positive charge) with other ordinary elements, e.g. O<sup>-2</sup> from O1<sup>-2</sup> or FE<sup>+2</sup> from FE1<sup>+2</sup>. The element **/-** does not take any degree of freedom in an equilibrium calculation. You can combine the elements VA and **/-** to model charged vacancies, e.g. VA<sup>-2</sup> results from VA1<sup>-2</sup>. To reject all charged species in the system, click to deselect (i.e. turn off). This is relevant to such databases as TCS Metal Oxide Solutions Database (TCOX), TCS Aqueous Solution Database (TCAQ), and the

OXDEMO database.



For most databases, there is an extensive list of all the phase models included with the technical information.

- The element **ZE** represents UNIT\_CHARGE with +1 net charge. It is implemented and required only in the database that contains the AQUEOUS solution phase. ZE should never be rejected in GES. ZE itself is the only constituent of the phase REF\_ELECTRODE, i.e. the reference electrode for ZE potential. This phase (REF\_ELECTRODE) should always be suspended in the POLY module. Combining ZE with other ordinary elements forms aqueous species with net charge, e.g. the species H+1 results from H1ZE1 while OH-1 from H1O1ZE-1. The element ZE, like other ordinary elements, takes one degree of freedom in POLY in an equilibrium calculation. This is relevant to such databases as TCS Aqueous Solution Database (TCAQ) and when working with the Pourbaix module, which is only available with Console Mode and thus not available in Graphical Mode directly.

### ***Periodic Table***

In this example, using the demonstration databases FEDEMO and MFEDEMO, there are only a select number of elements to choose from when it is a DEMO database. These are selected for this example (Cr, Mn, Fe, Ni, C). The FEDEMO database is a subset of the main steel database TCS Steel and Fe-alloys Database (TCFE) and MFEDEMO is a subset of TCS Steels/Fe-Alloys Mobility Database (MOBFE).

The image shows a periodic table of elements. Several elements are highlighted in grey: ZE, VA, I-, C, Cr, Mn, Fe, and Ni. The table includes the Lanthanide series (La to Lu) and the Actinide series (Ac to Lr).

## Alphabetic List

Click **Alphabetic List** to view a list of available and used elements.

Ele... ^	FEDEMO	MFEDEMO
I-	<input checked="" type="checkbox"/>	<input type="checkbox"/>
C	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Cr	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Fe	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Mn	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Ni	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
VA	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

## Material Composition

After selecting the elements and in the **Periodic Table** view, under *Material*, you can enter a **Material name** and define the material composition **Amount** in **Mass percent**, **Mole percent**, **Mass fraction**, or **Mole fraction**.

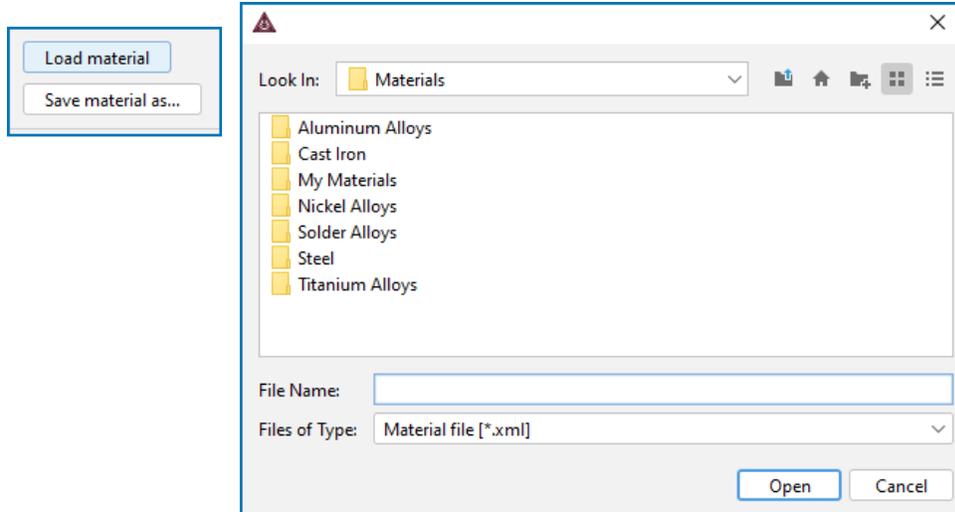
Element	Amount
Fe	80.52000
Cr	17.0
Ni	1.4
C	0.08
Mn	1.0

## How To Load or Save a Material

Click **Load material** to open a material file. The default folder opens to the **Materials** folder included with your installation.



[About the Material Files](#) and [Examples, Manuals, and Materials Folders](#)



You can click **Save material as** to save the material you defined, for example **Ferritic**. Add your own folders as needed.

### [Global Settings: System Definer](#)



For the **System Definer**, and if you load a material file and then change the composition, a message displays *The composition has changed*. This means that the original values/composition do not match the values you are using for the project. The message is cleared by either reloading the material file or editing the composition field to the original value.

## **Species**

On the **Species** tab for each database you can select and edit the *Species* and view the *Stoichiometry* for the selected database species.

Spe... ^	Stoichiometry	FEDEMO	MFEDEMO
/-	-1	<input checked="" type="checkbox"/>	<input type="checkbox"/>
C	C	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Cr	Cr	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Fe	Fe	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Mn	Mn	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Ni	Ni	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
VA	VA	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
C2	C <sub>2</sub>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
C3	C <sub>3</sub>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
C4	C <sub>4</sub>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
C5	C <sub>5</sub>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
C60	C <sub>60</sub>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
CR2	Cr <sub>2</sub>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
FE2	Fe <sub>2</sub>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
NI2	Ni <sub>2</sub>	<input checked="" type="checkbox"/>	<input type="checkbox"/>

At the bottom of the window, click **Add new species** and enter a **New species** name in the field.

 New species X

Enter the species:

## Phases and Phase Constitution

Elements   Species   **Phases and Phase Constitution**   Components   Data Sources

Phases

Status	Name ^	FEDEMO	MFEDEMO
Entered	BCC_A2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Entered	CBCC_A12	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	CEMENTITE	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Entered	CHI_A12	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	CUB_A13	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	DIAMOND_FCC_...	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	FCC_A1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Entered	FCC_A1#2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Entered	FCC_A1#3	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Entered	GAS	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	GRAPHITE	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	HCP_A3	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	KSI_CARBIDE	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	LAVES_PHASE_...	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	LIQUID	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	M23C6	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	M3C2	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	M5C2	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	M7C3	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	SIGMA	<input checked="" type="checkbox"/>	<input type="checkbox"/>

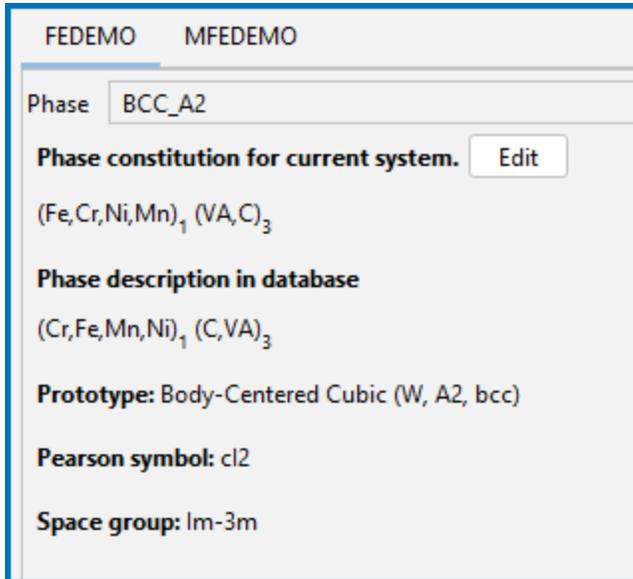
Check/uncheck all  

At the top of the **Phases and Phase Constitution** tab for each database you can review and edit the **Status (Entered or Dormant)** for the selected *Phases*. Select the **Check/uncheck all** checkbox to toggle between the options.



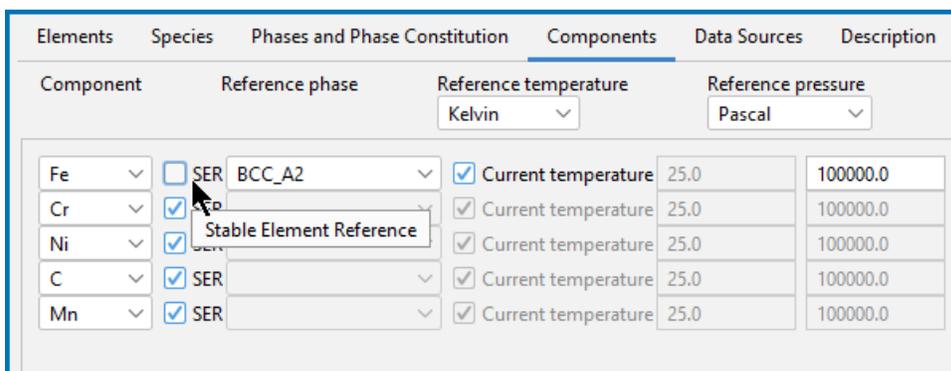
From this tab, you can also choose specific phases to include or exclude the **VA** and *l*- database species instead of doing it for the whole system as per [Elements: Advanced Database Options](#).

To review the Phase constitution for a specific phase, click it in the *Phase* list (for example **BCC\_A2**) and its details are displayed at the bottom of the window for the database selected. Click **Edit** then click an element to select or reject a constituent. The last remaining constituent on a sublattice cannot be rejected.



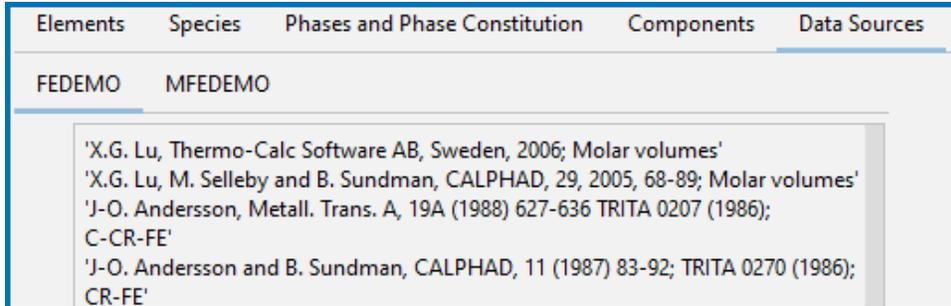
## Components

The **Components** tab is where you can review and edit the **Component**, **Reference phase**, **Reference temperature**, and **Reference pressure**. Click to toggle the **SER** (Stable Element Reference) checkbox on and off. Then choose a **Reference phase** from the list and edit accordingly. Choose a unit for the **Reference temperature** (**Kelvin**, **Celsius**, or **Fahrenheit**) and **Reference pressure** (**Pascal**, **Atmospheres**, or **Bar**).



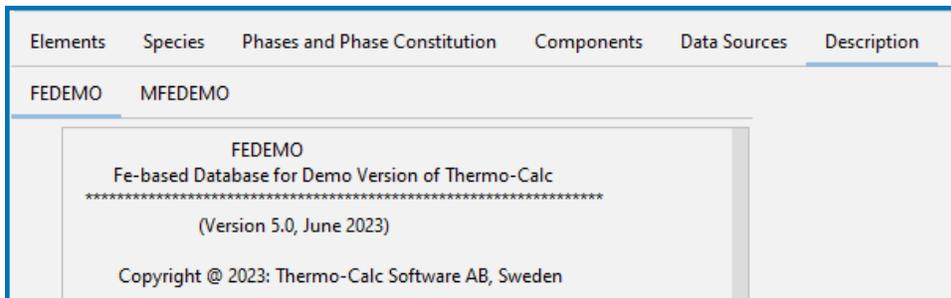
## Data Sources

Data source references are shown after performing the activity. Click **Perform** or **Perform Tree** at the bottom of the window when your system is set up.



## Description

The **Description** tab displays a description of the selected database(s). In this example, **FEDEMO**, or click **MFDEMO** to view that information.



## About the Material Files

When you are working with the System Definer or Material to Material Calculator in Graphical Mode you can open the default **Materials** folder included with your installation. The files are in .xml format and the data contained are from some common alloys. You can also save and create your own materials files or copy the xml files to the Materials folder.

 [Examples, Manuals, and Materials Folders](#)

## ***My Materials***

This is where you can add your own material files to be able to load it to the System Definer or Material to Material Calculator. When you are in the System Definer **Configuration** window, you define your material then click **Save material as**. You can save the profile to this folder or create a folder with a new name.

### ***Aluminum***

- UNS\_A95005
- UNS\_A96053

### ***Cast Iron***

- UNS\_F10007
- UNS\_F33101

### ***Nickel Alloys***

- UNS\_N06600
- UNS\_N6600
- UNS\_N06625
- UNS\_N06690
- UNS\_N07001
- UNS\_N07718
- UNS\_N08800
- UNS\_N08825

### ***Solder Alloys***

- ASTM\_25A
- ASTM\_96TS
- UNS\_L50134
- UNS\_L54321

## **Steel**

### *Austenitic Stainless Steel*

- ASTM\_304\_(UNS\_S30400)
- ASTM\_304L\_(UNS\_S30403)
- ASTM\_304LN\_(UNS\_S30453)
- ASTM\_316\_(UNS\_S31600)
- ASTM\_316L\_(UNS\_S31603)
- ASTM\_316LN\_(UNS\_S31653)
- ASTM\_317L\_(UNS\_S31703)
- UNS\_J92710
- UNS\_S30908

### *Duplex Stainless Steel*

- UNS\_S31803\_A
- UNS\_S32750\_A
- UNS\_S32750\_B

### *High Speed Steel*

- ASTM\_M10\_A5

### *High Temperature Steel*

- UNS\_S31008

### *HSLA Steel*

- UNS\_K12000

### *Low Carbon Steel*

- UNS\_H61180

### *Maraging Steel*

- UNS\_K92810

### *Martensitic Stainless Steel*

- ASTM\_420
- UNS\_J82090
- UNS\_J91639
- UNS\_S44004

### *Tool Steel*

- UNS\_T30106
- UNS\_T30109
- UNS\_T51620

### *Training Steel*

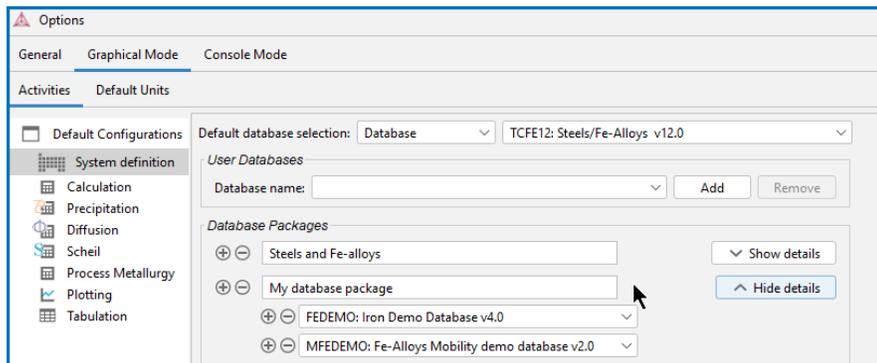
- Ferritic
- M7\_Steel

### ***Titanium Alloys***

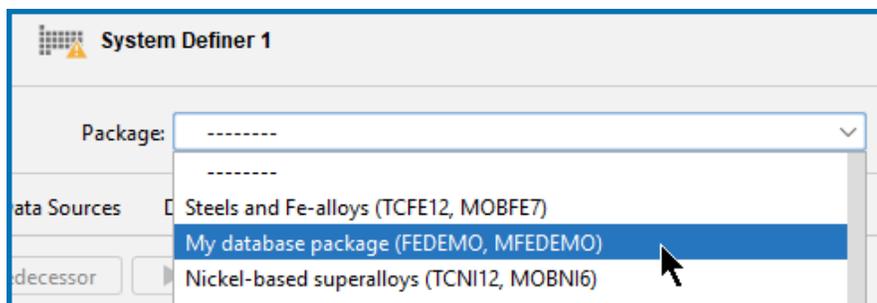
- UNS\_R54520
- UNS\_R56260

## Creating a Database Package

1. From the main menu select **Tools** → **Options** .
2. Click the **Activities** tab and click **System definition**.
3. Under *Database Packages*, click the **Add a database package**  $\oplus$  button.
4. Replace the default text *<Package #1>* with a name for the package, for example **My database package**.
5. Select a database from the **Choose database** list, for example, **FEDEMO Iron Demo Database**.
6. Click the **Add a database**  $\oplus$  button and from the **Choose database** list, select another database.
7. Click **OK**. The software needs to relaunch for the Database Package to be available to choose as the default for new System Definer activities.



The package is available for new System Definer nodes defined after adding the package.



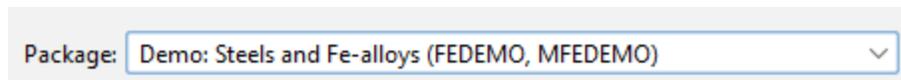
[Clearing a Database Package Message](#) and [Global Settings: System Definer](#)

## Clearing a Database Package Message

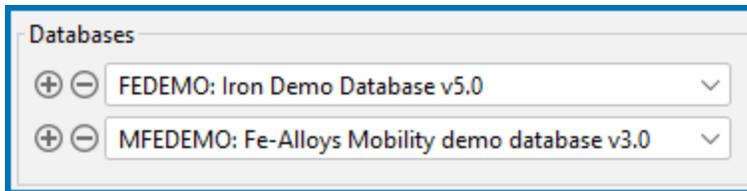
Sometimes when working on the System Definer **Configuration** window, you may add or delete Databases. If you originally added a Database Package, the addition or removal of a database means that the original package you selected does not now match the databases you are using. To clear the message *Selected databases and selected package may differ* and reset the databases you need to either:

- Change the newly added database back to the one that matches the database package, or
- Choose another package from the **Package** list, then select the package you want.

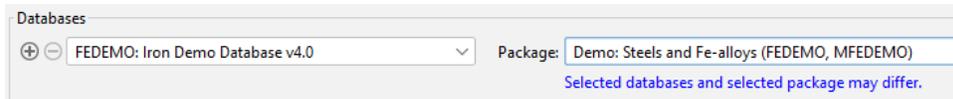
For example, from the **Package** list you select **Demo: Steels and Fe-alloys**.



This adds the **FEDEMO** and **MFEDEMO** databases.



If you add or remove **Databases**, in this example **MFEDEMO** is removed, then this does not match the **Package** and the message displays:



To clear the message, click the **Add a database**  button and from the **Choose database** list, select the database that belongs with the package, in this example, **MFEDEMO**. The message is cleared.

You can also choose another package from the list, which changes the databases, and then re-select the original package to reset the message.



[Creating a Database Package](#) and [Global Settings: System Definer](#)

## Equilibrium Calculator

An **Equilibrium Calculator** allows you to set the conditions for, and perform, a calculation. You can add the calculator to the System Definer directly (right-click and select it from the **Create New Successor** menu) or use a variety of the available templates.

Once you have added an Equilibrium Calculator, the **Configuration** window has these settings tabs. An Equilibrium Calculator that is the successor of another Equilibrium Calculator inherits all the settings of the predecessor.

- **Conditions:** Set the conditions for your calculation that define the stepping or mapping axis variables. Click the **Switch to Simplified Mode** and **Switch to Advanced Mode** buttons to toggle between those settings.
- **Functions:** Define your own quantities and functions, which you then can calculate and plot.
- **Options:** Modify numerical settings that determine how equilibria are calculated, as well as how stepping and mapping calculations are performed.



The **Options** tab is where you can locally define specific Equilibrium Calculator settings. These settings can also be applied globally. To open the **Options** window: From the main menu select **Tools** → **Options** and click the **Activities** tab. Then click **Calculation**.



There are these templates available under **Equilibrium** that use an Equilibrium Calculator: **Single Point**, **One Axis**, and **Phase Diagram**. Choose one of these to quickly set up a simulation. The **Quick Start** wizard is also an option. See [Getting Started Links and Templates](#).

### Define the Equilibrium Calculator

1. Add an Equilibrium Calculator node to the System Definer. If you used the **Single Point**, **One Axis**, or **Phase Diagram** templates (on the **My Projects Configuration** window under **Equilibrium**), click the **Equilibrium Calculator** node to display the **Configuration** settings window.
2. In the **Configuration** window, enter the settings described in [Conditions Tab Settings](#).

For the **Options** and **Functions** tab settings, see [Options Tab Settings](#) and [Calculating and Plotting Functions](#)

- Once you have finished defining the Equilibrium Calculator, you also choose settings on the [Plot Renderer](#).



### [Calculating with a Fixed Phase](#)



There are several examples that use an Equilibrium Calculator. See [T\\_01: Calculating a Single-Point Equilibrium](#), or for an example of how serially coupled Equilibrium Calculators can be used, see [T\\_06: Serially Coupled Equilibrium Calculators](#).

## Conditions Tab Settings

Conditions and axis variables for the **Equilibrium Calculator** are set in the **Conditions** tab on the **Configuration** window. The tab can be viewed in two modes: **Simplified** (the default) or **Advanced** (see [About Advanced Mode](#)). The conditions to set depend on the type of calculation.

Quantity	Min	Max	Step division	Type
Mass percent C	0.0	100.0	50.0	Linear - min no. of steps
Temperature	500.0	3000.0	50.0	Linear - min no. of steps



See examples [T\\_01: Calculating a Single-Point Equilibrium](#), [T\\_02: Stepping in Temperature in the Fe-C System](#) and [T\\_05: Stable and the Metastable Fe-C Phase Diagrams](#).



### [Options Tab Settings](#)



Depending on the calculation type selected, there are also a variety of [Plot Types](#) to choose from.

The following outlines what you can set on the **Conditions** tab in simplified mode.



Click **Switch to Advanced Mode** for more options.

## Composition Unit

Select the **Composition unit: Mass percent, Mole percent, Mass fraction, or Mole fraction.**

## Condition Definitions

Specify the values (and units) of the state variables: **Temperature, Pressure, and System size** or any other available lists under *Condition Definitions*.

## Calculation Type

Click to select a *Calculation Type*:

- **Single equilibrium** (no axes): To calculate a single point. The results from this calculation are displayed in the **Event Log**.
- **One Axis**: To vary a quantity on the X-axis.
- **Grid**: Computes the equilibrium of each grid point in a two-dimensional grid.
- **Phase diagram**: To calculate in a defined multicomponent heterogeneous system, from one or more initial equilibria. A phase diagram is usually mapped within a specific space that is constructed by two (or more) defined independent mapping axis-variables.



For **Single equilibrium**, no additional settings are required. Add a **Table Renderer** to perform the calculation and view the details in the **Visualizations** window.



When you use the [Equilibrium Property Model Settings](#) with the **Property Model Calculator**, it is the same calculation result as when you are using an **Equilibrium Calculator** with the **Single equilibrium** calculation type.

## Axis Definitions

- **One Axis:** Define the axis variable using these fields and menus: **Quantity**, **Min**, **Max**, **Step division**, **Type**, and **Step Method**.
- **Grid:** Define the two axis variables using these fields and menus: **Quantity**, **Min**, **Max**, and **Number of steps**.
- **Phase diagram:** Define the two axis variables using these fields and menus: **Quantity**, **Min**, **Max**, **Step division**, and **Type**.



If the Calculator already has a Plot Renderer as a successor and you change the stepping/mapping quantities, then the quantities represented on the X- and Y-axis are automatically updated in the Plot Renderer.



Depending on the calculation type selected, there are also a variety of [Plot Types](#) to choose from.

## Step Division and Type (One Axis and Phase Diagram)

Enter a **Step division** and select a **Type**: **Linear - min no. of steps**, **Linear - max step size**, or **Logarithmic - scale factor**.

- For **Linear - min no. of steps**, the **Step division** value specifies a minimum number of steps that is calculated between the minimum and maximum value during the stepping/mapping operation.
- For **Linear - max step size**, the **Step division** sets the maximum size of each step. For **Logarithmic - scale factor**, **Step division** specifies the scale factor of a logarithmic function that distributes the steps from the initial equilibrium to the minimum and maximum values defined.

## Step Method (One Axis)

From the **Step Method** list, choose **Normal** or **Separate phases**.

- **Normal** allows a stepping calculation with the chosen independently-varying equilibrium condition (axis-variable). Only the step axis value is changed between each step.
- **Separate phases** calculates how the Gibbs energy for a number of phases varies for different compositions. This is particularly useful to calculate Gibbs energies for complex phases with miscibility gaps and for an ordered phase that is never disordered (e.g. SIGMA-phase, G-phase, MU-phase, etc.).

## Number of Steps (Grid)

Set a fixed **Number of steps**. The number of steps along with the minimum and maximum values for the axes define a grid. For each grid point an equilibrium is calculated.

### *About Advanced Mode*

Click **Switch to Advanced Mode** to add and remove conditions as well as set additional axis definitions. However, the number and types of conditions set must still conform to the Gibbs' phase rule. The **Number of missing conditions is** field, at the top of the tab shows how many conditions that you have left to set. If the number is negative, then that number of conditions need to be removed.

## **List of Advanced Mode Condition Definitions**

- Activity of component
- Activity referred to a phase
- Amount of component
- Amount of component in a phase
- Amount of phase
- Chemical potential of a component
- Chemical potential referred to a phase
- Composition
- Enthalpy
- Entropy
- Fix phase
- Gibbs energy
- Helmholtz energy
- Internal energy

- $\ln(\text{activity of component})$
- $\ln(\text{activity referred to a phase})$
- Pressure
- Site fraction
- System size
- Temperature
- User-defined
- Volume

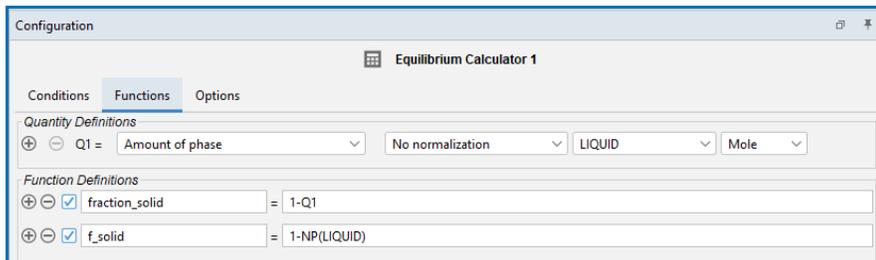
## Calculating and Plotting Functions

The **Functions** tab allows you to define your own functions that can be plotted in a Plot Renderer.



For an example of how you can use functions that you define, see [T\\_07: User-Defined Functions](#).

1. Add an **Equilibrium Calculator** node and click it in the **Project** window.
2. In the **Configuration** window click the **Functions** tab.
3. Under **Quantity Definitions**, define **Q1**. Click the add  $\oplus$  button to **Add a new quantity**. Then from the lists, choose the variable for each quantity to define. The quantities are called **Q1**, **Q2**, **Q3** and so on. In this example, the quantity **Q1** is defined as the **Amount of phase**, with **No normalization**, **LIQUID** in **Mole** units.



4. Under **Function Definitions**, click the add  $\oplus$  button to add a new function. In the left field edit the default name of each function and enter the function itself in the right field after the equal sign. In this example, the quantity defined as Q1 is renamed to **fraction\_solid**, which gives as its output the fraction of solid phase (**1-Q1**).

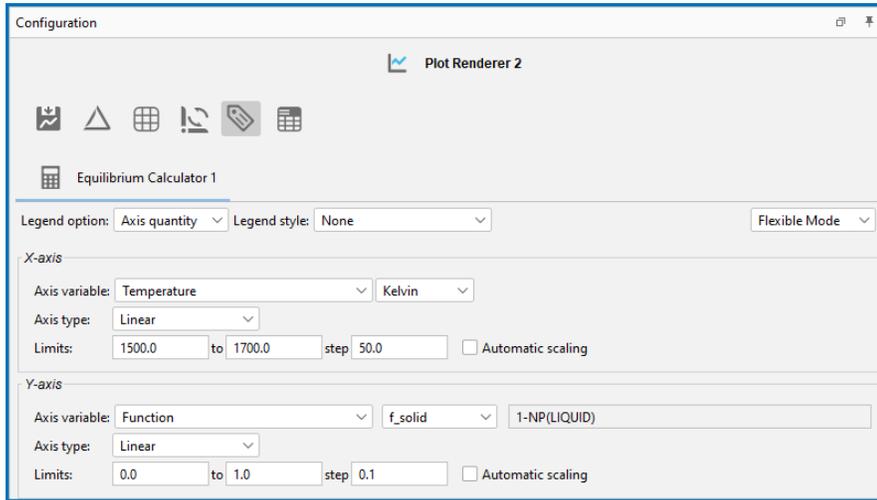


You can also use Console Mode syntax when entering the function. For example, the function above can be entered as `1-NP(LIQUID)`. See [Parameters, Functions, and Variables](#) for details.

5. To plot the defined function, right-click **Equilibrium Calculator** and choose **Create New Successor → Plot Renderer**.
6. Click the **Plot Renderer** node. In the tab for the Equilibrium Calculator that contains the function, set an **Axis variable** to **Function**, then from the menu select the defined function. In this example the Y-axis is set to plot the value of the function **fraction\_**

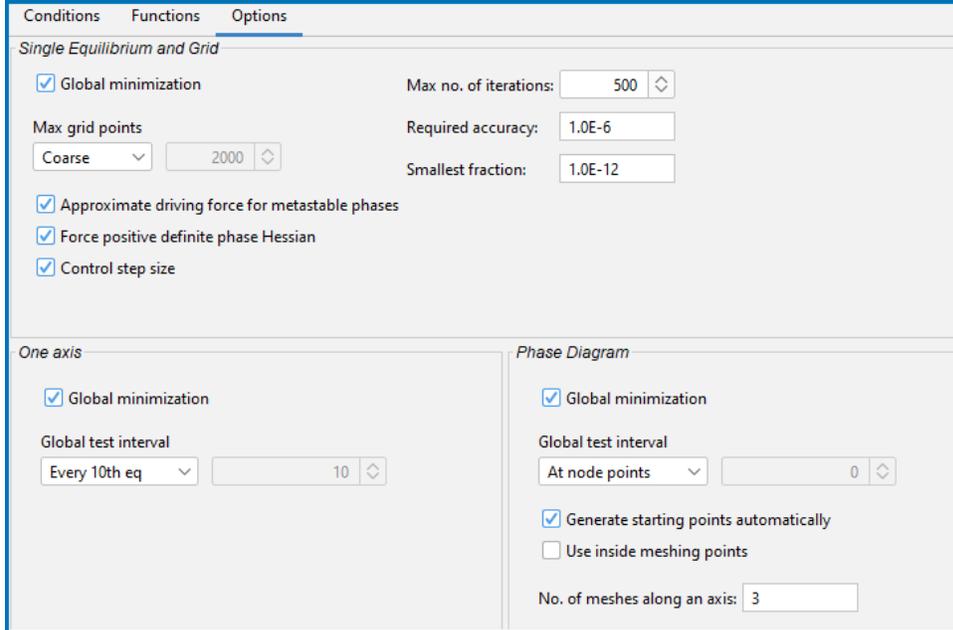
**solid.**

When run, the Plot Renderer plots the value of the function.



## Options Tab Settings

These settings are for the [Equilibrium Calculator](#) and are located on the **Configuration** window → **Options** tab.





You can change these settings locally for a specific Calculator or globally for some defaults in the **Options window** (a different location than the tab).

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **Graphical Mode** tab and the **Calculation** node in the tree.

### *Single Equilibrium and Grid*

Under *Single Equilibrium and Grid*, there are these *Calculation Type* setting options.

### **Global Minimization**

The **Global minimization** checkbox is selected by default to ensure that the most stable minimum under the specified conditions is computed.

### **Max Grid Points**

For the **Max grid points**, the default is **Coarse** (2000 grid points), or choose **Medium** (20,000 grid points), **Fine** (200,000 grid points), or **Custom** to set your own number of grid points.

### **Approximate Driving Force for Metastable Phases**

The **Approximate driving force for metastable phases** checkbox is selected by default. Click to clear the checkbox to change the default as required and based on the options described below.

This setting involves the convergence of metastable phases and affects their driving forces. It can also have an effect on when an equilibrium is considered successful.

The default is to allow an equilibrium with metastable phases that have not converged, as long as the stable phases have converged. This is efficient but often causes approximate values of the driving forces for the metastable phases.

---

If you change the default, it enforces metastable phases to converge. This gives accurate driving forces for metastable phases as well as stable phases. It can however take a slightly longer time, and if metastable phases do not converge it causes the equilibrium calculation to fail.

### Force Positive Definite Phase Hessian

The **Force positive definite phase Hessian** checkbox is selected by default and this setting determines how to reach the minimum of an equilibrium state.



[ADVANCED\\_OPTIONS](#)

### Control Stepsize

The **Control stepsize** checkbox is selected by default and this setting determines how to reach the minimum of an equilibrium state.



[ADVANCED\\_OPTIONS](#)

### Max No of Iterations

Enter or choose a value for the **Max no. of iterations**. By default, the program tries 500 iterations before it gives up. As some models give computation times of more than 1 CPU second/iteration, this number is also used to check the CPU time and the calculation stops if 500 CPU seconds/iterations are used.

### Required Accuracy

The default **Required accuracy** is  $1.0\text{E}-6$ . This is a relative accuracy, and the program requires that the relative difference in each variable must be lower than this value before it has converged. A larger value normally means fewer iterations but less accurate solutions. The value should be at least one order of magnitude larger than the machine precision.

### Smallest Fraction

The default **Smallest fraction** (or **Smallest fraction in a phase** for Property Models) is  $1.0\text{E}-12$ . This is the value assigned to constituents that are unstable. It is normally only in the gas phase you can find such low fractions.

The default value for the smallest site-fractions is  $1\text{E}-12$  for all phases except for IDEAL phase with one sublattice site (such as the GAS mixture phase in many databases) for which the default value is always as  $1\text{E}-30$ .



In Console Mode, the default of 1E-30 for IDEAL phases is true unless you are using the command SET\_NUMERICAL\_LIMITS to reset an even-lower value (e.g. 1E-45, that is naturally enforced to all the phases in the system.

## One Axis

Under *One Axis*, there are these *Calculation Type* setting options.

### Global Minimization

The **Global minimization** checkbox is selected by default to ensure that the most stable minimum under the specified conditions is computed.



#### ADVANCED OPTIONS

### Global Test Interval

The default **Every 10th eq** is the recommended **Global test interval** i.e. every tenth step and at each phase change during calculations. Or choose **Always** or **Custom**.

If you choose **Custom**, the integer number determines how often the global minimization should be used during step and mapping. Any other positive integer number,  $n$ , suggests it performs a global minimization test at every  $n$ -th step during calculations.

## Phase Diagram

Under *Phase diagram*, there are these *Calculation Type* setting options.

### Global Minimization

The **Global minimization** checkbox is selected by default to ensure that the most stable minimum under the specified conditions is computed.

### Global Test Interval

The default **At node points** is the recommended **Global test interval** and is used only at node points during the calculations.

- If you choose **Every 10th eq**, this is every tenth step and at each phase change during calculations.
- If you choose **Custom**, the integer number determines how often the global minimization should be used during step and mapping. Any other positive integer number,  $n$ , suggests it performs a global minimization test at every  $n$ -th step during calculations.

## Generate Starting Points Automatically

The **Generate starting points automatically** checkbox is selected by default.

## Use Inside Meshing Points

Click the **Use inside meshing points** checkbox as required. The initial mesh consists of initial equilibrium points added only along the edge/border of the diagram defined by the axis-variables. If you select the checkbox, then the initial equilibrium points added are also added to the inside edge.

## No of Meshes Along an Axis

The default **No of meshes along an axis** is 3, which represents how many intervals of initial equilibrium points are added along an edge in the diagram. For example, if set to 2 (two), initial equilibrium points are added at the beginning, center, and end of each axis-variable forming 2 intervals.

## Calculating with a Fixed Phase

This describes how to use the advanced mode to set a phase to be fixed at a certain amount. This allows you to, for example, find out at what temperature a material starts to melt. If you set the phase to be fixed to liquid phase at zero amount, and do not define the temperature state variable, then you can calculate at what temperature the material enters a state where the liquid phase is no longer zero (that is, when it starts to melt).



For a fixed phase equilibrium calculation, it is recommended to serially couple the Equilibrium Calculators. This is where one Equilibrium Calculator activity is the successor to another Equilibrium Calculator activity that performs an ordinary equilibrium calculation, and then, for the condition you are interested in, you make a rough estimate of the value. This gives the fixed phase calculation better starting values.

1. Right-click a node that can add an **Equilibrium Calculator** to your project.
2. In the **Project** window, click the **Equilibrium Calculator** activity node.
3. In the **Configuration** window, click **Switch to Advanced Mode**.

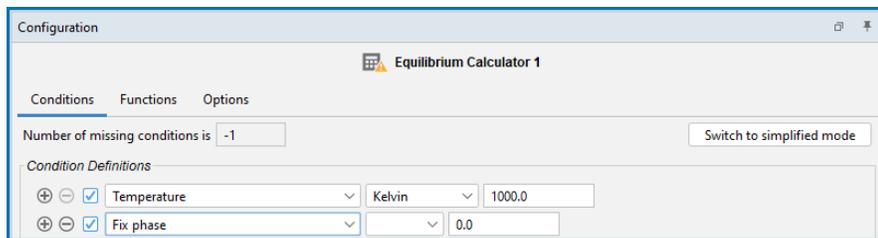


If the **Default calculation mode** is globally set to **Advanced** then by default the Configuration window is already in advanced mode.



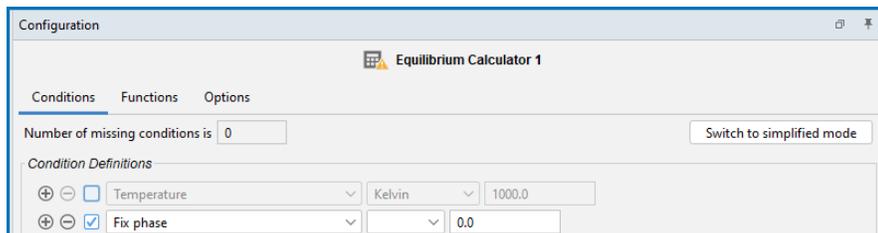
### [Global Settings](#)

4. Under *Condition Definitions*, click the **Add quantity**  $\oplus$  button.
5. Select **Fix phase** from the first menu.
6. Select the phase that you want to fix in the second menu, and set the amount of that phase you want to fix the phase to. For example, if you want to know when your material starts to melt, set the phase to **LIQUID** and the value to **0.0**:



Above the *Condition Definitions*, the **Number of missing conditions is** field is probably **-1**. This means that you must remove one condition.

7. To remove a condition, either click the **Remove**  $\ominus$  button, or click to clear the checkbox. For example, if you want to know at what temperature your material starts to melt, then you cannot have a fixed temperature, in which case the temperature condition is not selected:



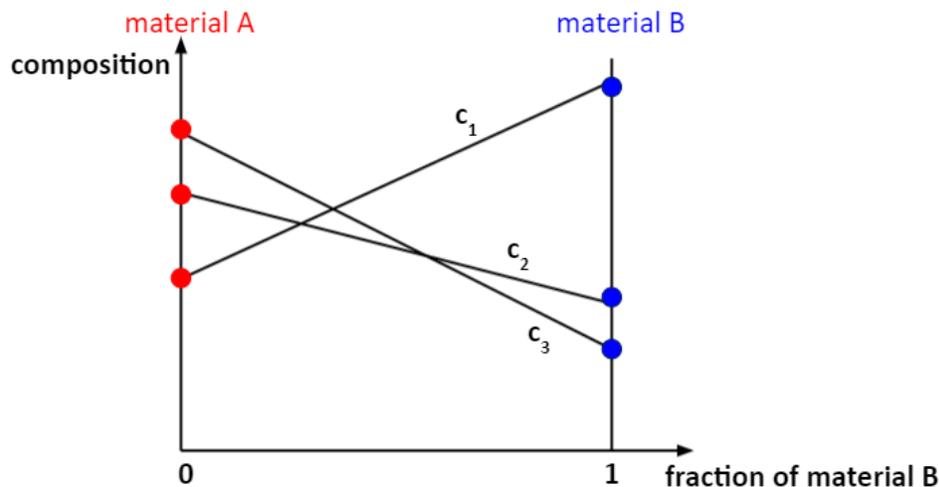
The **Number of missing conditions is** field displays **0**. When you have set all your conditions, you can perform the Equilibrium Calculator (now or later).

## Material to Material Calculator

A **Material to Material Calculator** allows you to set the conditions for, and perform, a calculation to examine how materials transition from one into the other. Typical applications are for graded materials, non-similar material joints (without requiring a diffusion calculation), or a reaction of two materials, e.g. influence of volcanic ash on a turbine component in an engine.

For example, you can look at the transitions of phase compositions ( $c_1$ ,  $c_2$ ,  $c_3$  in the diagram) for two materials, A and B and then calculate an equilibrium, one axis (similar to a property diagram), or phase diagram to examine, for example, what phases are forming at the interfaces or to see what kinds of reactions there are at specific points in the transition.

Composition of the elements is evolving versus the fraction of the second material.



You can add the calculator to the System Definer directly (right-click and select it from the **Create New Successor** menu).



There is also a **Material to Material** template available under **Equilibrium** that you can use to quickly set up a simulation. See [Getting Started Links and Templates](#).

Once you have added a Material to Material Calculator, the **Configuration** window has these settings tabs.

- **Conditions:** Set the conditions for your calculation that define the stepping or mapping axis variables.
- **Functions:** Define your own quantities and functions, which you then can calculate and plot.
- **Options:** Modify numerical settings that determine how equilibria are calculated, as well as how stepping and mapping calculations are performed.



Read more on our website about the [Material to Material Calculator](#). If you are in Thermo-Calc, press F1 to search the help.

## Define the Material to Material Calculator

1. Add a Material to Material Calculator node to the System Definer. If you used the **Material to Material** template (on the  **My Projects Configuration** window under **Equilibrium**), click the **Material to Material Calculator** node to display the **Configuration** settings window.
2. In the **Configuration** window, enter the settings described in [Material to Material Calculator Conditions Tab Settings](#).



For the **Options** and **Functions** tabs, the settings are the same as the Equilibrium Calculator. See [Options Tab Settings](#) and [Calculating and Plotting Functions](#).

3. Once you have finished defining the Material to Material Calculator, you also choose settings on the [Plot Renderer](#).



[T\\_14: Fe-Cr-Ni Transition Comparison Using the Material to Material Calculator](#).

## Material to Material Calculator Conditions Tab Settings

Conditions and axis variables for the Material to Material Calculator are set in the **Conditions** tab on the **Configuration** window. The conditions to set depend on the type of calculation.



See example [T\\_14: Fe-Cr-Ni Transition Comparison Using the Material to Material Calculator](#)

The **Material to Material** template (on the  **My Projects Configuration** window under **Equilibrium**) creates a project to perform calculations to examine how materials transition from one into the other. The project consists of a System Definer, Material to Material Calculator, and a Plot Renderer.



**Functions:** This is the same as the Equilibrium Calculator. See [Calculating and Plotting Functions](#)



**Options:** This is the same as the Equilibrium Calculator. See [Options Tab Settings](#)

The following outlines what you can set on the **Conditions** tab.

### Composition Unit

Select the **Composition unit**: **Mass percent**, **Mole percent**, **Mass fraction**, or **Mole fraction**.

### Condition Definitions

Under *Condition Definitions*:

- For both the **First material** and **Second material**, define the values (and units) of the state variables **Temperature** and **Pressure**.
- Enter a **Fraction of second material**.
  - For a **Single equilibrium** calculation this is used as condition for the calculation.
  - For a **One Axis** calculation this is used as the start value for the calculation.
  - For a **One Axis** or **Phase diagram** calculation, and if the **Fraction of second material** is chosen as the *Axis Definition>Quantity*, then it is stepping between the *Min* and *Max*.
- For both materials, select the **Dependent component**.
- Select the **Activity conditions** checkbox to define the **Activity of component** for one or more components. By default this is not selected. Click the **Add quantity**  and **Remove quantity**  buttons to add and remove conditions.



The reference state is defined on the System Definer.

## Condition Definitions > Define the First and Second Materials

For each **First material** and **Second material** component there are individual settings to define as follows.

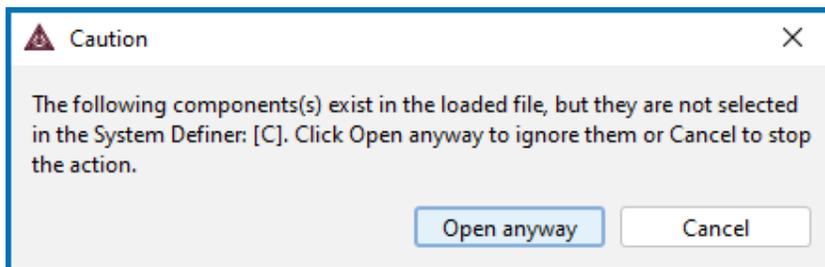
Enter a unique **Material name** for each. This name is used on the plots and in the quantity definitions so it is good to use as concise a name as possible that clearly identifies each material. For example, `Martensitic Steel` for the **First material** and `Alloy 800` for the **Second material**.

For each **Composition** field, enter numeric values for each material.

- Click **Save material as** after defining a composition to add it for reuse in other projects and calculations.
- Click **Load material** to add the composition of a predefined material from an xml file. Also see [How To Load or Save a Material](#)



If the user-defined material file includes one or more components not defined on the System Definer, then you are prompted to make a decision:

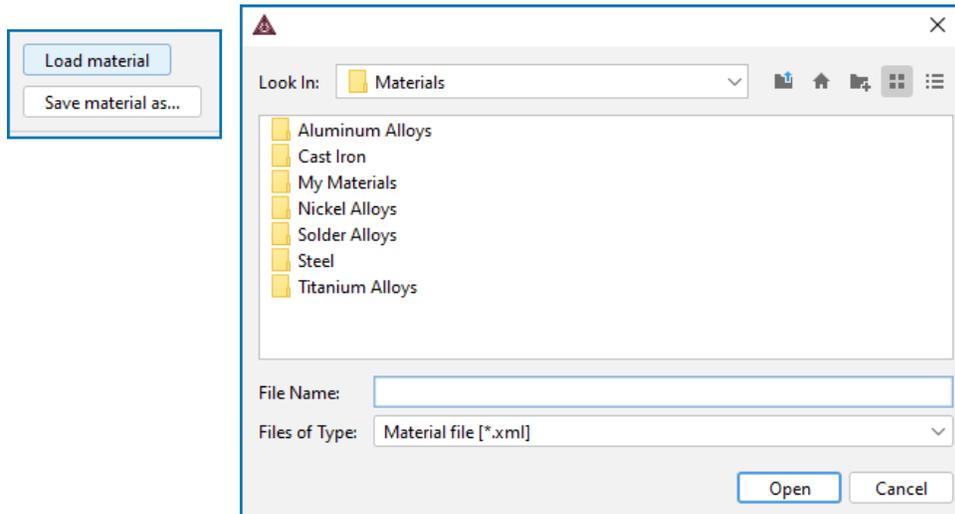


Click **Open anyway** to continue to load the material file even with the undefined component(s). However, this means the undefined component is excluded at first from calculations until it is further defined on the System Definer. Once the material is defined then it is automatically included the next time calculations are run. Otherwise, click **Cancel** to stop the action and either edit the material file or manually define the component(s) in the System Definer.

## How To Load or Save a Material

Click **Load material** to open a material file. The default folder opens to the **Materials** folder included with your installation.

 [About the Material Files](#) and [Examples, Manuals, and Materials Folders](#)



You can click **Save material as** to save the material you defined, for example **Ferritic**. Add your own folders as needed.

 [Global Settings: System Definer](#)



For the **System Definer**, and if you load a material file and then change the composition, a message displays *The composition has changed*. This means that the original values/composition do not match the values you are using for the project. The message is cleared by either reloading the material file or editing the composition field to the original value.

### Calculation Type

Click to select a *Calculation Type*:

- **Single equilibrium** (no axes): To calculate a single point. The results from this calculation are displayed in the **Visualizations** window if a Table Renderer is used.
- **One Axis**: To vary a quantity on the X-axis; the fraction of one of the materials or temperature.
- **Phase diagram**: To calculate a section in a multicomponent system between two materials, transitioning from one into the other. The phase diagram is mapped within a specific space that is constructed by the two mapping axis-variables; the fraction of one of the materials and temperature.

## Axis Definitions

- **One Axis**: Define the axis variable using these fields and menus: **Quantity**, **Min**, **Max**, **Step division**, **Type**, and **Step Method**.
- **Phase diagram**: Define the two axis variables using these fields and menus: **Quantity**, **Min**, **Max**, **Step division**, and **Type**.

Also see additional setting details for **Step division**, **Type**, and **Step Method**.



If the Calculator already has a Plot Renderer as a successor and you change the stepping/mapping quantities, then the quantities represented on the X- and Y-axis are automatically updated in the Plot Renderer.

## Step Division and Type (One Axis and Phase Diagram)

Enter a **Step division** and select a **Type**: **Linear - min no. of steps**, **Linear - max step size**, or **Logarithmic - scale factor**.

- For **Linear - min no. of steps**, the **Step division** value specifies a minimum number of steps that is calculated between the minimum and maximum value during the stepping/mapping operation.
- For **Linear - max step size**, the **Step division** sets the maximum size of each step. For **Logarithmic - scale factor**, **Step division** specifies the scale factor of a logarithmic function that distributes the steps from the initial equilibrium to the minimum and maximum values defined.

## Step Method (One Axis)

From the **Step Method** list, choose **Normal** or **Separate phases**.

- **Normal** allows a stepping calculation with the chosen independently-varying equilibrium condition (axis-variable).
- **Separate phases** calculates how the Gibbs energy for a number of phases varies for different compositions. This is particularly useful to calculate Gibbs energies for complex phases with miscibility gaps and for an ordered phase that is never disordered (e.g. SIGMA-phase, G-phase, MU-phase, etc.).

## Binary Calculator

---

A Binary Calculator can be used for some calculations involving two components. You can think of this activity as a combination of a System Definer and an Equilibrium Calculator, but designed to simplify setting up and performing calculations on binary systems.

The Binary Calculator relies on some specifications that are not supported by all databases. You need a specially designed database for the Binary Calculator such as the TCBIN database.



For an example see [T\\_03: Fe-C Phase Diagrams](#).



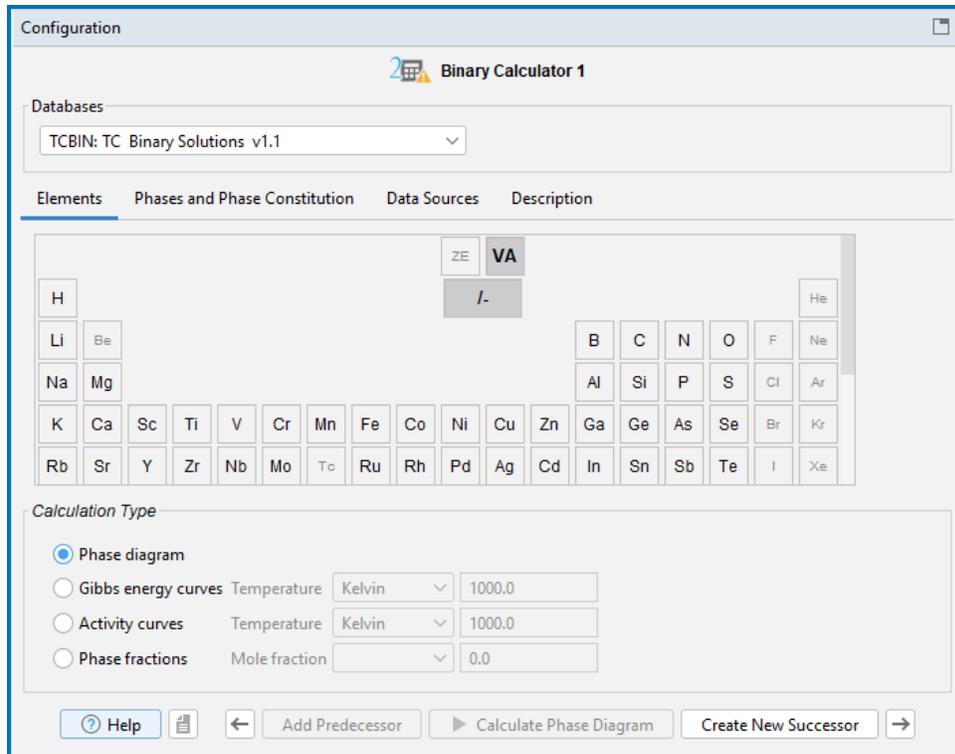
There is also a **Binary** template available on the  **My Projects Configuration** window under **Equilibrium** that you can use to quickly set up a simulation. See [Getting Started Links and Templates](#).

## Configuration Window Settings

The Binary Calculator **Configuration** settings window and the associated tabs are similar to what you see on a System Definer. The database TCBIN is added by default when you use the **Binary** template (on the  **My Projects Configuration** window under **Equilibrium**), but you can choose an installed and licensed database from the list.



The various versions of the TCFE (Thermo-Calc Steels and Fe-Alloys) databases are not compatible with the Binary Calculator.



## Elements

On the **Elements** tab, and based on the database selected, select and review elements from the periodic table. Unavailable elements, which are based on the selected database, are greyed out.

## Phases and Phase Constitution

At the top of the **Phases and Phase Constitution** tab, and based on the selected database, you can review the *Status* for the selected *Phases*. Select the **Check/uncheck all** checkbox to toggle between the options.

To review the **Phase constitution** for a specific phase, click it in the *Phase* list (for example **BCC\_A2**) and its details are displayed below the *Phases* list. Click **Edit** then click an element to select or reject a constituent. The last remaining constituent on a sublattice cannot be rejected.

## Data Sources

Data sources are not available until the activity is performed. Click **Perform Tree** at the bottom of the window to display available data sources.

## Description

The **Description** tab displays a description of the selected database.

## Calculation Type

Click to choose a *Calculation Type*.

- **Phase diagram:** To calculate in a defined multicomponent heterogeneous system, from one or more initial equilibria. A phase diagram is usually mapped within a specific space that is constructed by two (or more) defined independent mapping axis-variables.
- **Gibbs energy curves:** To calculate at a constant temperature over the whole composition range.
- **Activity curves:** To calculate at a constant temperature over the whole composition range.
- **Phase fractions:** To calculate as a function of temperature at a constant composition.

Then:

- For **Gibbs energy curves** and **Activity curves**, select a unit from the **Temperature** list: **Kelvin**, **Celsius**, or **Fahrenheit** and enter a value in the field.
- For **Phase fractions**, select and enter the **Mole fraction** component and enter a value in the field.



For the Plot Renderer, there is an option to use the Table View  button to convert plots to table data for all calculation types and calculators. See [Plot Renderer: Convert a Plot to a Table and Export the Data](#).

## Ternary Calculator

A **Ternary Calculator** is used for some calculations involving three components. You can think of this activity as a combination of a System Definer and an Equilibrium Calculator, but designed to simplify setting up and performing calculations on ternary systems.



For an example, see [T\\_04: Fe-Cr-C Ternary Phase Diagram at 1000 K](#).



There is also a **Ternary** template available under **Equilibrium** that you can use to quickly set up a simulation. See [Getting Started Links and Templates](#).

## Configuration Window Settings

Below lists the available settings that must be configured before performing the calculation and the image shows the **Configuration** window for a Ternary Calculator activity.

### Databases

Select a database that contains specifications dedicated to the Ternary Calculator.

---

## Elements

On the **Elements** tab, and based on the database selected, select and review elements from the periodic table. Unavailable elements, which are based on the selected database, are greyed out.

## Phases and Phase Constitution

At the top of the **Phases and Phase Constitution** tab, and based on the selected database, you can review the *Status* for the selected *Phases*. Select the **Check/uncheck all** checkbox to toggle between the options.

To review the **Phase constitution** for a specific phase, click it in the *Phase* list (for example **BCC\_A2**) and its details are displayed below the *Phases* list. Click **Edit** then click an element to select or reject a constituent. The last remaining constituent on a sublattice cannot be rejected.

## Data Sources

Data sources are not available until the activity is performed. Click **Perform Tree** at the bottom of the window to display available data sources.

## Description

The **Description** tab displays a description of the selected database.

## Calculation Type

Click to choose a *Calculation Type*.

- **Isothermal section:** To calculate the ternary system for the whole composition range.
- **Monovariant lines:** To calculate the variation of the liquid compositions with temperature.
- **Liquidus projection:** The projection liquid surface and the monovariant lines are calculated over the given temperature range.

Then for **Isothermal section** and **Liquidus projection**, select a unit from the **Temperature** list: **Kelvin**, **Celsius**, or **Fahrenheit** and enter a value in the field.



For the Plot Renderer, there is an option to use the Table View  button to convert plots to table data for all calculation types and calculators. See [Plot Renderer: Convert a Plot to a Table and Export the Data](#).

## Scheil-Gulliver Solidification Calculations



In Graphical Mode, you use the [Scheil Calculator](#) activity node to simulate the various models available in Thermo-Calc or with the Add-on Modules or additional databases.



Read more about [Scheil Solidification Simulations](#) on our website, including [how to select the right model for your simulation](#). If you are in Thermo-Calc, press F1 to search the help to learn about using Scheil.

Thermo-Calc is often used to perform equilibrium calculations, but some non-equilibrium transformations or partial-equilibrium transformations can also be simulated.

A well-known example of a non-equilibrium calculation is the *Scheil-Gulliver solidification simulation*. In Thermo-Calc, the available Scheil-Gulliver solidification simulations are *classic Scheil*, Scheil with *back diffusion* in the primary phase, or Scheil with *solute trapping*. There are also configuration options available with *fast diffusers* for the classic and back diffusion models.

### Classic Scheil Simulation

This is a Scheil simulation based on the well-known *Scheil-Gulliver* model. If this model is used without "fast diffusers" it is *Classic Scheil* simulation with the following assumptions:

- Diffusion in the liquid phase is assumed to be very fast, that is, infinitely fast.
- Diffusion in the solid phases is so slow that it can be ignored, that is, diffusion is assumed to be zero.
- The liquid/solid interface is in thermodynamic equilibrium.

In this Classic Scheil simulation the temperature is decreased step-by-step. When the temperature drops below the liquidus temperature the equilibrium amount and composition of solid and liquid phase is calculated. The solid phase is removed from the system and only the amount and composition of the liquid phase is used for the next calculation step at a lower temperature. Again, the equilibrium amount and composition of solid and liquid phase is calculated and again the solid phase is removed from the system for the next step. This procedure is repeated until the last liquid disappears.

This calculation procedure is equivalent to assuming that there is no diffusion in the solid phase and infinitely fast diffusion of all elements in the liquid phase. It has been shown to be a good approximation of the solidification of most alloys such as Ni-superalloys, Cu alloys, Al alloys, Mg alloys, and others. However, in materials with interstitial elements, such as carbon, ignoring diffusion in the solid causes discrepancies with experimental results. Interstitial elements have much faster diffusion rates. The assumption of no diffusion in the solid phase during solidification is thus not correct at most industrial or lab-scale solidification rates.

For this reason, a variant of the Scheil simulation was developed and implemented in Thermo-Calc, where one or more elements can be defined as *fast diffusers*. See [Other Options: Fast Diffusers and Scheil Simulations](#).

## Scheil Simulation with Back Diffusion in Primary Phase

The *Scheil simulation with back diffusion in primary phase* kinetic model quantitatively takes into account the real back diffusion of all elements in the primary solid phase (typically the FCC or BCC phase). This model requires the use of both a thermodynamic and mobility database. The calculation also requires the cooling rate to be specified; a fast cooling rate allows for less time for back diffusion and the simulation result is similar to the classic Scheil-Gulliver solidification simulation. A very slow cooling rate allows for almost complete back diffusion, the solidification simulation is then close to an equilibrium calculation. It also requires that the size of the solidification domain is specified. In most cases this domain size corresponds to the secondary arm spacing, as this is where typically the liquid is trapped during solidification.



The secondary arm spacing also depends on the cooling rate. For this reason, a simple empirical relation between cooling rate and secondary arm spacing is predefined in Thermo-Calc.

For the *Scheil simulation with back diffusion in primary phase* model the following assumptions are made.

- Diffusion of all elements in the liquid phase is infinitely fast.
- Diffusion of all elements in the primary solid phase are quantitatively calculated using mobility data, a cooling rate and a domain size (typically this is the secondary arm spacing).
- The liquid/solid interface is in thermodynamic equilibrium.

Also see [Other Options: Fast Diffusers and Scheil Simulations](#).

---

## Scheil with Solute Trapping

The *Scheil with solute trapping* model simulates deviation from local equilibrium for the primary phase. It is useful for high solidification speeds, such as those seen in additive manufacturing applications and is based on the Aziz [1982] model for solute trapping and extended to multicomponent systems.

The model is based on the main assumption that the primary phase is the only dendrite forming phase affected by solute trapping.

For equilibrium- and Scheil-type solidification simulations thermodynamic equilibrium is established at the solid-liquid interface. This means that solutes are partitioned between the solid and liquid phases according to solidus and liquidus lines of the phase diagram. The assumption of thermodynamic equilibrium at the solid-liquid interface however breaks down for very fast solidification rates, that are encountered for example during additive manufacturing. Aziz [1982, Aziz; 1988, Aziz], proposed a model that links partitioning at the solid-liquid interface to solidification rate / interface velocity. Basically, the faster the solidification rate, the lower is the partitioning at the interface. Put very simply, what is thought to happen, is that the solid-liquid interface moves so quickly, that the solutes in the liquid are trapped in the advancing solid phase, thus the name “solute trapping”.

Aziz [1982] proposed a model to account for solute trapping based on balancing the overall kinetics at the solid-liquid interface. The two terms to be balanced are the velocity of the interface, that is calculated based on the cooling rate and considerations of the solid-liquid geometry (dendrite arm spacing) the interface kinetics, that follows from the trans-interface diffusion (meaning how fast solutes travel from one side of the solid-liquid interface to the other) and the change of phase from liquid to solid without changing the composition.

For a *Scheil with solute trapping* model the following assumptions are made.

- Only one primary solid phase forms dendrite, NOT necessarily the first solid phase
- Solute trapping in primary solid phase only. Other solid phases have equilibrium compositions following Scheil model
- Amounts of solid phases are dependent on solute trapping and solidification speed
- Dynamic liquidus for primary solid phase is dependent on solute trapping and solidification speed

### **Solute Trapping References**

[1982Azi] Aziz, M. J. (1982). Model for solute redistribution during rapid solidification. *Journal of Applied Physics*, 53(2), 1158–1168.

[1988Azi] Aziz, M. J., & Kaplan, T. (1988). Continuous growth model for interface motion during alloy solidification. *Acta Metallurgica*, 36(8), 2335–2347.

## Other Options: Fast Diffusers and Scheil Simulations

If a *Classic Scheil* or *Scheil simulation with back diffusion in primary phase* model is selected plus one or more *Fast diffuser* checkboxes selected, then Thermo-Calc takes into account the concept of the *fast diffuser*. Typically, C is defined as a fast diffuser, but also other elements such as N, O, or others can also be defined.



*Fast diffuser* does not quite define this since diffusion is not actually simulated. It is simply assumed that the “fast diffusers” distribute throughout the solid and liquid phase according to thermodynamic equilibrium. This corresponds to *infinitely fast diffusion* for the selected elements.

For a *Classic Scheil* model with the compositions defined with *Fast diffuser* the following assumptions are made.

- Diffusion of all elements in the liquid phase is infinitely fast.
- Diffusion of all elements in the solid phases, except the ones defined as “fast diffusers”, is zero.
- Diffusion of the elements defined as “fast diffusers” is infinitely fast in the solid phase.
- The liquid/solid interface is in thermodynamic equilibrium.

For a *Scheil simulation with back diffusion in primary phase* model with the compositions defined with *Fast diffuser* the following assumptions are made.

- Diffusion of all elements in the liquid phase is infinitely fast.
- Diffusion of all elements in the primary solid phase are quantitatively calculated using mobility data, a cooling rate and a domain size (typically this is the secondary arm spacing).
- Diffusion of the elements defined as “fast diffusers” is infinitely fast in the solid phase.
- The liquid/solid interface is in thermodynamic equilibrium.

The Classic Scheil simulation and Classic Scheil simulation with fast diffuser models can be considered *thermodynamic* models. There are clear limitations as a certain amount of back diffusion always takes place in the solid phase at real solidification.

You can also use the more sophisticated *Scheil simulation with back diffusion in primary phase* kinetic model. Adding the fast diffuser option to this can be useful, for example for interstitials like C in steels where these then equilibrate in all phases and not just the primary phase.

## Homogenization, Segregation Profiles, and the Diffusion Module



Scheil calculations can be used in diffusion simulations with the Diffusion Module (DICTRA).

Homogenization is a process where you want to get rid of the composition segregation from solidification with a homogenization heat treatment. This composition segregation can be simulated with Scheil. Then with the Diffusion Module (DICTRA), you input the secondary dendrite arm spacing to get the right diffusion distances. With composition profile from file the idea is that you should be able to input for example a measured composition profile (from EDS/WDS, micro-probe etc). This could also be a segregation profile but also composition profiles over a weld, substrate/coating, diffusion couple, and so forth.

## Summary

The available Scheil-Gulliver solidification simulations are *classic* Scheil, Scheil with *back diffusion* in the primary phase, or Scheil with *solute trapping*. There are also configuration options available with *fast diffusers* for the classic and back diffusion models.



There is also connectivity to the Diffusion Module (DICTRA) and *homogenization* where you collect data from a Scheil solidification calculation, which is then used as part of the kinetic homogenization simulation of segregated composition profiles.



[Scheil in Graphical Mode vs Console Mode](#)



The Scheil with back diffusion feature is only available for systems with diffusion data, i.e. this model requires the use of a mobility database.

In Thermo-Calc, the Scheil calculation allows the calculation of:

- The solidification range of an alloy.
- Depression of the solidus temperature due to segregation.
- Composition of the last liquid to disappear in segregation pockets.
- Phases formed on final solidification in segregation pockets.
- The composition gradient in the primary solid phase(s) (segregation profile).
- Solute trapping to simulate deviation from local equilibrium for the primary phase
- Kinetic homogenization simulation of segregated composition profile (diffusion calculations)

## Scheil in Graphical Mode vs Console Mode

Type of Calculation	Console Mode	Graphical Mode
Classic Scheil	Use the START_WIZARD command to answer prompts	Scheil Calculator > select the Classic Scheil button on the Configuration window.  On the My Project Configuration window under Getting Started, click Quick Start and select Scheil solidification simulation.
Scheil with back diffusion in the primary phase	USE_BACK DIFFUSION command (turn on) then use the START_WIZARD command to answer prompts	Scheil Calculator > select the Scheil with back diffusion in the primary phase button on the Configuration window then enter the settings as needed. A mobility database is required.
Scheil with solute trapping	SOLUTE_TRAPPING command (turn on) then use the START_WIZARD command to answer prompts	Scheil Calculator > select the Scheil with solute trapping button on the Configuration window then enter the settings as needed.
Fast diffusers and Classic Scheil	Use the START_WIZARD command to answer the prompt <code>Fast diffusing components</code> where you enter the elements that diffuse fast in solid phases and can assume to be in equilibrium in the whole phase. E.g. C in a steel.	Scheil Calculator > select the Classic Scheil button plus one or more of the Fast diffuser checkboxes next to the composition(s) on the Configuration window then enter the settings as needed.
Fast diffusers and Scheil with back diffusion in the primary phase	USE_BACK DIFFUSION command (turn on) then use the START_WIZARD command to answer the prompt <code>Fast diffusing components</code> where you enter the elements that diffuse fast in solid phases and can assume to be in equilibrium in the whole phase. E.g. C in a steel.	Scheil Calculator > select the Scheil with back diffusion in the primary phase button plus one or more of the Fast diffuser checkboxes next to the composition(s) on the Configuration window then enter the settings as needed.

Type of Calculation	Console Mode	Graphical Mode
		window then enter the settings as needed. A mobility database is required.
Homogenization	Use the EVALUATE_SEGREGATION_PROFILE command before START_WIZARD to generate the segregation profile to file. In DICTRA_MONITOR use INPUT_SCHEIL_PROFILE for use of segregated profile in simulation. An additional license to the Diffusion Module (DICTRA) is needed to work with more than three components.	Use the <b>Homogenization</b> template under <b>Non-Equilibrium</b> to automatically set up the project tree with the defaults needed to work with this. An additional license to the Diffusion Module (DICTRA) is needed to work with more than three components.

## Scheil Calculator

In Graphical Mode, you use the **Scheil Calculator** to set up the various Scheil solidification simulations outlined in more detail in [Scheil-Gulliver Solidification Calculations](#).



The Scheil Calculator is also used in combination with the specialized Add-on Modules such as the Diffusion Module (DICTRA) and Additive Manufacturing (AM) Module.

The available Scheil-Gulliver solidification simulations are *classic* Scheil, Scheil with *back diffusion* in the primary phase, or Scheil with *solute trapping*. There are also configuration options available with *fast diffusers* for the classic and back diffusion models.



There is also connectivity to the Diffusion Module (DICTRA) and *homogenization* where you collect data from a Scheil solidification calculation, which is then used as part of the kinetic homogenization simulation of segregated composition profiles.



[Scheil in Graphical Mode vs Console Mode](#)



The Scheil with back diffusion feature is only available for systems with diffusion data, i.e. this model requires the use of a mobility database.

In Thermo-Calc, the Scheil calculation allows the calculation of:

- The solidification range of an alloy.
- Depression of the solidus temperature due to segregation.
- Composition of the last liquid to disappear in segregation pockets.
- Phases formed on final solidification in segregation pockets.
- The composition gradient in the primary solid phase(s) (segregation profile).
- Solute trapping to simulate deviation from local equilibrium for the primary phase
- Kinetic homogenization simulation of segregated composition profile (diffusion calculations)

There are several options to get started with a Scheil simulation using the Scheil Calculator.

- On the  **My Project** window under **Getting Started**, use the **Quick Start → Scheil Solidification Simulation** option to set up a Classic Scheil simulation. This adds a tree of nodes where you can then choose to simulate using the other options on the Scheil Calculator. See [Creating a Project with the Quick Start Wizard](#).
- Use the **Scheil Solidification** template under **Non-Equilibrium** to add a project tree consisting of **System Definer**, **Scheil Calculator**, and **Plot Renderer** nodes. See [Getting Started Links and Templates](#).
- Use the **Homogenization** template under **Non-Equilibrium** to add a project tree consisting of **System Definer**, **Scheil Calculator**, **Diffusion Calculator**, and two **Plot Renderer** nodes. See [Getting Started Links and Templates](#).



When working with the **Homogenization** template, this also includes a **Diffusion Calculator**, which may require additional licenses to run with more than three components. Information, such as the Scheil segregation profiles available with the Diffusion Calculator, is described with that content.

- Build your own tree by right-clicking and adding nodes from  **My Project**. Then go to [Configuration Window Settings](#) to enter settings directly.
- Open an example and use this as a template.
- When you are working with the Additive Manufacturing (AM) Module, a **Scheil Calculator** is also included with one of the [Additive Manufacturing Templates](#) or can be added as a predecessor to the **AM Calculator**, where it is used to generate the materials property data required for these simulations.



By default, a [Plot Renderer](#) successor to a Scheil Calculator has the plot axes defined as **Mole fraction of solid** (X-axis) and **Temperature Celsius** (Y-axis). When these default axes are used and the setting **Include one axis equilibrium calculation** is enabled (see [Advanced Options](#)), the results of solidification under equilibrium conditions are automatically plotted in the same diagram as the Scheil calculations.

## Define the Scheil Calculator

1. Add a **Scheil Calculator** node to the System Definer. If you used the **Scheil Solidification** or **Homogenization** templates on the  **My Projects Configuration** window under **Non-Equilibrium** (or opened an example), click the node to display the **Configuration** settings window.
2. Enter the settings described in [Configuration Window Settings](#). The **Configuration** tab has all the main settings and the permutations related to the types of Scheil simulations available such as back diffusion and solute trapping.



When working with the **Homogenization** template, this also includes a **Diffusion Calculator**, which may require additional licenses to run with more than three components. Information, such as the Scheil segregation profiles available with the Diffusion Calculator, is described with that content.

3. As needed you can also change default settings from the **Advanced Options** tab (see [Advanced Options](#))
4. Once you have finished defining the Scheil Calculator, you also choose settings and variables on the [Plot Renderer](#). For a list of available variables see [Scheil Calculator Available Plot Variables](#).

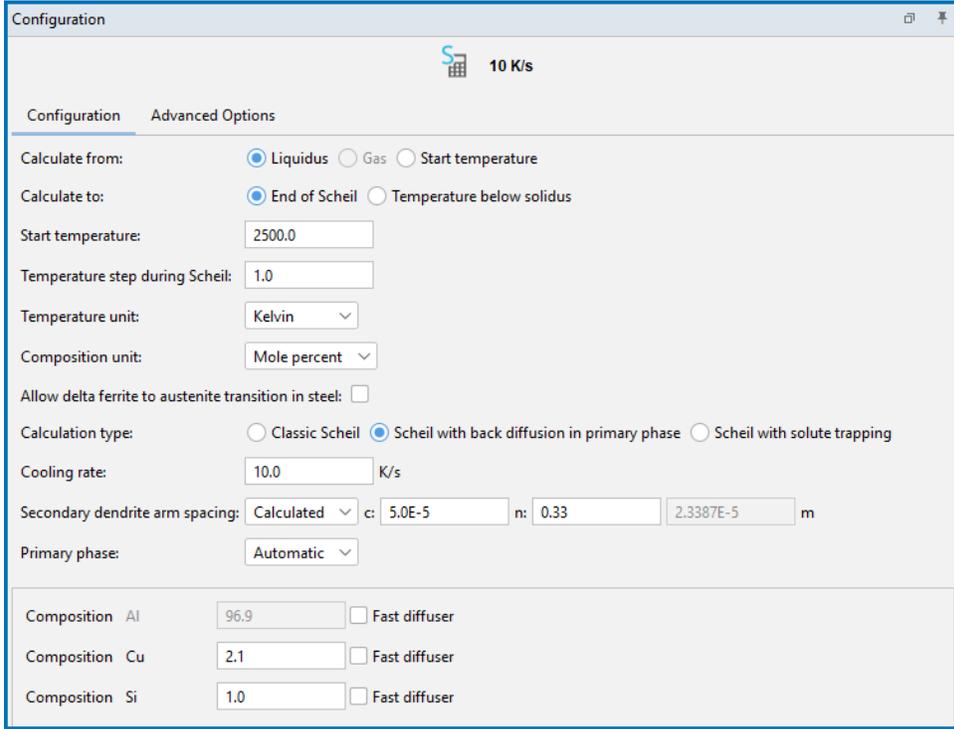


For examples using the Scheil Calculator, see [T\\_08: Scheil and Equilibrium Solidification](#), [T\\_10: Scheil Solidification with Back Diffusion](#), and [T\\_13: Scheil Solidification with Solute Trapping](#). There are also Console Mode, TC-Python, and Diffusion Module (DICTRA) examples described elsewhere.

## Configuration Window Settings

An example of the main **Configuration** window of a **Scheil Calculator** activity is shown below using the [T\\_10: Scheil Solidification with Back Diffusion](#) example.

### [Scheil-Gulliver Solidification Calculations](#)



Configuration window showing settings for Scheil-Gulliver Solidification Calculations. The window is titled "Configuration" and displays a cooling rate of 10 K/s. The settings are as follows:

- Calculate from:**  Liquidus  Gas  Start temperature
- Calculate to:**  End of Scheil  Temperature below solidus
- Start temperature:** 2500.0
- Temperature step during Scheil:** 1.0
- Temperature unit:** Kelvin
- Composition unit:** Mole percent
- Allow delta ferrite to austenite transition in steel:**
- Calculation type:**  Classic Scheil  Scheil with back diffusion in primary phase  Scheil with solute trapping
- Cooling rate:** 10.0 K/s
- Secondary dendrite arm spacing:** Calculated
- c:** 5.0E-5
- n:** 0.33
- m:** 2.3387E-5 m
- Primary phase:** Automatic

Composition settings for fast diffusers:

Element	Composition	Fast diffuser
Al	96.9	<input type="checkbox"/>
Cu	2.1	<input type="checkbox"/>
Si	1.0	<input type="checkbox"/>



There are additional settings found on the **Advanced Options** tab and based on the type of Scheil calculation. See [Advanced Options](#).

### Calculate From

Use the **Calculate from** setting to choose from where the calculation should start. With this setting it is possible to obtain properties of the gas and/or liquid phase before the solidification starts.

Some of the temperature definitions related to these choices are:

- *Liquidus*, the temperature where solidification starts.
- *Gas / Liquid*, the temperature above which the liquid phase is no longer stable.
- *Liquid / Gas*, the temperature below which the gas phase is no longer stable.

Click a button to choose **Liquidus**, **Gas**, or **Start temperature**.



Depending on the choice for this setting (**Liquidus**, **Gas**, or **Start temperature**), one or more of the following temperature values is calculated and collected in the **Event Log** at the end of the calculation: *Liquidus*, *Gas / Liquid*, or *Liquid / Gas*.

By default, **Liquidus** is selected. The results are calculated from the start of the solidification.

Select **Gas** (after a gas phase is selected on the System Definer) to perform single equilibrium calculations from the *Gas / Liquid* temperature.

- The equilibrium calculations are performed at 5 Kelvin (K) intervals in the two-phase region between the *Gas / Liquid* temperature and the *Liquid / Gas* temperature.
- In the fully liquid region below the *Liquid / Gas* temperature single equilibrium calculations are performed at 50 K intervals until the *Liquidus* temperature is reached. After that the Scheil solidification calculation proceeds as usual.

Select **Start temperature** if you want to obtain properties of the liquid phase (and gas phase if selected) before the solidification starts.

- Ordinarily the start temperature is only used as a start point in the search of the liquidus temperature, as the Scheil calculation starts from the *Liquidus* temperature.
- For this choice, single equilibrium calculations are performed at specific intervals from the temperature entered in the start temperature field, down to the *Liquidus* temperature, and after that the Scheil solidification calculation proceeds as usual.
- The equilibrium calculations are performed at 50 Kelvin (K) intervals in the fully liquid region and at 5 K intervals in a possible two-phase (*Liquid / Gas*) region.

## Calculate To

Use the **Calculate to** setting to choose from where the calculation should end, **End of Scheil** or **Temperature below solidus**, where a frozen structure is assumed below solidus.

- **End of Scheil** stops the calculation when the Scheil calculation is finished.
- **Temperature below solidus** calculates the properties in the solid state, for the phase compositions and fractions at the end of the Scheil calculation. The default **Final temperature** is room temperature. Click **Show advanced options** to adjust the *Temperature below solidus settings*: Change the **Final temperature** and/or define the **No. of points**.
- See [Advanced Options](#).

### Start Temperature

Enter a **Start temperature** that is higher than the liquidus temperature of the alloy, in other words, the temperature at which the alloy is completely melted.

### Temperature Step during Scheil

Enter a **Temperature step during Scheil**. The temperature step is given in the same unit as **Start temperature**. Decreasing the temperature step increases the accuracy, but the default value is usually adequate.

### Temperature Unit

Select the **Temperature unit**: **Kelvin**, **Celsius** or **Fahrenheit**.

### Composition Unit

Select the **Composition unit**: **Mass percent**, **Mole percent**, **Mass fraction**, or **Mole fraction**.

### Allow Delta Ferrite to Austenite Transition in Steel

Select the **Allow delta ferrite to austenite transition in steel** checkbox to enable BCC to FCC transition.



This feature should only be used for such steels for which it is reasonable to assume infinitely fast diffusion (equilibrium conditions) when delta ferrite is present. When the **Allow delta ferrite to austenite transition in steel** checkbox is selected in Graphical Mode or the DELTA\_FERRITE\_AUSTENITE\_TRANSITION command used in Console Mode, this essentially assumes that diffusion can be considered (infinitely) fast when delta ferrite is present as a solid phase, i.e. assumes an equilibrium condition. After delta ferrite is no longer stable, the usual Scheil assumptions for the selected calculation type are in effect. This setting is therefore not suitable for steels where you DO NOT have a complete BCC to FCC transition during solidification. This setting is also NOT available in combination with solute trapping.

## Calculation Type: Classic, Back Diffusion, Solute Trapping



The available settings are based on the chosen model. See [Scheil-Gulliver Solidification Calculations](#) for descriptions.

Click to select a **Calculation type: Classic Scheil, Scheil with back diffusion in primary phase, or Scheil with solute trapping**. Then based on this selection, additional settings are made available as indicated.



The back diffusion and solute trapping options are available to select after adding a mobility database and once components are defined on the System Definer.

## Composition

The **Composition** settings display after you add components to the System Definer.

## Fast Diffuser

The **Fast diffuser** checkboxes are available after you add components to the System Definer. Click the checkbox to allow redistribution of this component in both the solid and liquid parts of the alloy.

---

## Cooling Rate (Back Diffusion)

This field is available when **Scheil with back diffusion in primary phase** is selected.

Specify the **Cooling rate** in Kelvin per second (K/s). An increased value moves the result from equilibrium toward a Scheil-Gulliver calculation.

## Secondary Dendrite Arm Spacing (Back Diffusion)

This field is available when **Scheil with back diffusion in primary phase** is selected.

The **Secondary dendrite arm spacing** is the distance in meters between two secondary dendrite arms. It can be either a **Constant** or **Calculated** from the cooling rate as  $c^*(\text{cooling rate})^{-n}$  where  $c$  and  $n$  are entered in these fields.

Select **Calculated** or **Constant**.

- For **Calculated**, enter values for **c** and **n**.
- For **Constant**, enter a value (unit = m) in the field.

## Primary Phase (Back Diffusion and Solute Trapping)

This field is available when **Scheil with back diffusion in primary phase** or **Scheil with solute trapping** is selected.

The primary solidified phase is the phase where the back diffusion or solute trapping takes place.

If **Automatic** (Scheil Calculator) or **Any phase** (Property Models) is selected (or kept as the default), the program tries to find the phase which gives the most back diffusion or solute trapping.

To override this setting, choose a specific primary phase from the list:

- For back diffusion, only phases with diffusion data can be used as primary phases.
- For solute trapping, only phases that dissolve all elements in the system can be used as primary phases.

## Scanning Speed, Alpha, and Calculated Solidification Speed (Solute Trapping)

These fields are available when **Scheil with solute trapping** is selected.

As needed, enter **Scanning speed** and  $\alpha$  to define the **Calculated solidification speed**.



On the Scheil Calculator, the **Calculated solidification speed** is shown. For Property Models, below is how the values are calculated from the **Scanning speed** and  $\alpha$  values entered.

The **Calculated Solidification Speed** is  $V_s = V_{\text{scanning}} * \cos(\alpha)$  m/s, where  $V_{\text{scanning}}$  is the user specified **Scanning speed** in m/s and  $\alpha$  is the alpha angle,  $\alpha$ , between the solid/liquid boundary and scanning direction.

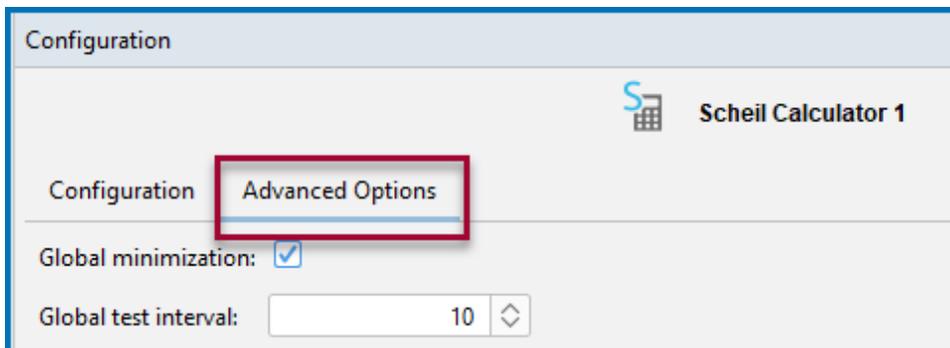
## Advanced Options

This topic has advanced settings information for the **Scheil Calculator Configuration** window. The main settings are on the **Configuration** tab.



[Configuration Window Settings](#)

Click the **Advanced Options** tab to adjust the following default settings as needed.



To make global default setting changes, see [Global Settings: Scheil Calculator](#).

### Global Minimization

Global minimization is enabled by default i.e. the **Global minimization** checkbox is selected.



It is recommended to keep global minimization on as it makes the calculation more accurate. However, turn it off if you find that the calculation is taking too long for any particular system, and accuracy is not the highest priority.

For this setting, a global equilibrium test is performed at selected intervals when an equilibrium is reached. By default it is at every 10 th Scheil step but this can be changed using the **Global test interval** setting. The global equilibrium test is also done every time the set of stable phases tries to change. This costs more computer time but the calculations are more robust.

### Global Test Interval

Use the **Global test interval** setting to change the interval at which a global equilibrium test is done, which by default is every 10th step as long as there are no changes in the set of stable phases. Every time the set of stable phases tries to change, a global equilibrium test is also done regardless of the test interval value chosen. A global equilibrium test means that the calculated equilibrium state obtained by the ordinary minimization calculation is tested against the Global Minimization Technique, and if the result is found unstable then the full global minimization calculation is performed instead.

### Liquid Phase

The default in the list is the **LIQUID** phase already defined in the chosen database.

### Gas Phase

The default in the list is the **Gas phase** already defined in the database

### Calculate Evaporation Properties

Select the **Calculate evaporation properties** checkbox to calculate the **Molar mass of gas**, **Driving force for evaporation**, and **Evaporation enthalpy** properties.

A gas phase is required to calculate these properties.

This option sets the gas phase to dormant, meaning that the thermodynamic properties of the gas are evaluated, without the gas being allowed to form. This enables the calculation of the following properties: **Molar mass of gas**, **Evaporation enthalpy**, and **Driving force for evaporation**. Driving force for evaporation is positive when the gas is stable (temperature above evaporation temperature). A negative driving force means the gas is metastable.

---

## Terminate Scheil On

This setting controls when a Scheil calculation should terminate. From the **Terminate Scheil on** list, choose **Fraction of liquid phase** or **Temperature** and enter a value in the field.

## Max No. of Iterations

Enter or choose a value for the **Max no. of iterations**. By default, the program tries 500 iterations before it gives up. As some models give computation times of more than 1 CPU second/iteration, this number is also used to check the CPU time and the calculation stops if 500 CPU seconds/iterations are used.

## Required Accuracy

The default **Required accuracy** is  $1.0\text{E-}6$ . This is a relative accuracy, and the program requires that the relative difference in each variable must be lower than this value before it has converged. A larger value normally means fewer iterations but less accurate solutions. The value should be at least one order of magnitude larger than the machine precision.

## Smallest Fraction

The default **Smallest fraction** (or **Smallest fraction in a phase** for Property Models) is  $1.0\text{E-}12$ . This is the value assigned to constituents that are unstable. It is normally only in the gas phase you can find such low fractions.

The default value for the smallest site-fractions is  $1\text{E-}12$  for all phases except for IDEAL phase with one sublattice site (such as the GAS mixture phase in many databases) for which the default value is always as  $1\text{E-}30$ .



In Console Mode, the default of  $1\text{E-}30$  for IDEAL phases is true unless you are using the command SET\_NUMERICAL\_LIMITS to reset an even-lower value (e.g.  $1\text{E-}45$ , that is naturally enforced to all the phases in the system).

## Approximate Driving Force for Metastable Phases

The **Approximate driving force for metastable phases** checkbox is selected by default. Click to clear the checkbox to change the default as required and based on the options described below.

This setting involves the convergence of metastable phases and affects their driving forces. It can also have an effect on when an equilibrium is considered successful.

The default is to allow an equilibrium with metastable phases that have not converged, as long as the stable phases have converged. This is efficient but often causes approximate values of the driving forces for the metastable phases.

If you change the default, it enforces metastable phases to converge. This gives accurate driving forces for metastable phases as well as stable phases. It can however take a slightly longer time, and if metastable phases do not converge it causes the equilibrium calculation to fail.

## Include One Axis Equilibrium Calculation

The **Include one axis equilibrium calculation** checkbox is selected by default and a **One Axis** (also called *step*) calculation is performed before the actual Scheil calculation. It produces the solidification curve under equilibrium conditions and it is shown on the plot if the default axis variables are used. This is sometimes referred to as the *equilibrium line*.

If a comparison to equilibrium is not necessary, change the default setting and then only a Scheil calculation is performed. This results in a much faster calculation, which for a complex system may be preferable.

## Temperature Below Solidus Settings

The **Temperature below solidus settings** are available when **Temperature below solidus** is selected for **Calculate to**. This calculates the properties for the given number of temperatures, down to the final temperature for the phase compositions and fractions at the solid phases at the end of the Scheil calculation.

- Enter the **No. of points**. The default is 50.
- Enter the **Final temperature**. The default is room temperature (298.15 K).

## Scheil Calculator Available Plot Variables

When working with Scheil calculations in Graphical Mode, there are several plot variables available with the Plot Renderer that is a successor to the Scheil Calculator.



For a comparable list in Console Mode, see [State Variables for Scheil](#).

The following list is specific to a Plot Renderer that is a successor to a Scheil Calculator. The software includes relevant variables in the *Axis variable* list based on the set up of the Scheil Calculator.



There may be some additional variables available if the calculation involves the Diffusion Module (DICTRA) and/or other specialized databases are used.

- Apparent heat capacity per gram
- Apparent heat capacity per mole
- Apparent volumetric CTE
- Average composition of solid phases
- Composition of a phase
- Density
- Distribution of component in phase
- Driving force for evaporation per mole
- Dynamic viscosity
- Electric conductivity
- Electric resistivity
- Evaporation enthalpy per mole
- Heat per gram
- Heat per mole
- Kinematic viscosity
- Latent heat per gram
- Latent heat per mole
- Mass fraction of liquid
- Mass fraction of solid
- Mass fraction of solid phase
- Molar mass of gas
- Molar volume
- Molar volume of phase
- Mole fraction of liquid
- Mole fraction of solid
- Mole fraction of solid phase
- Site fraction
- Surface tension

- Temperature
- Thermal conductivity
- Thermal diffusivity
- Thermal resistivity
- Volume fraction of liquid
- Volume fraction of solid
- Volume fraction of solid phase

## Scheil Calculations Frequently Asked Questions

Below are some frequently asked questions (FAQs) related to Scheil calculations in Thermo-Calc. It is not intended to cover all the scenarios that can occur with such complex calculations. A particular focus is on the use and definition of variables available for plotting latent heat, heat, and apparent heat capacity, and the differences between Graphical Mode and Console Mode.

### ***What is the latent heat evolution for a Scheil solidification simulation?***

Theory usually defines the latent heat as the isothermal enthalpy change of phase transformation at any chosen temperature. It is called *latent* because it is regarded as a potential heat release or uptake. Usually this is the transformation from liquid to solid.

However, most alloys, if not all, solidify non-isothermally. In the Scheil calculation the plotted quantity called *latent heat* might not follow the theoretical definition of latent heat. It is the cumulative enthalpy change of phase transformation.



There is no contribution from heat capacity, only from phase transformation.

### ***How can I plot the latent heat evolution after a Scheil simulation?***

In Graphical Mode, on the Plot Renderer node, select one of the **Axis variables**—**Latent heat per mole** or **Latent heat per gram**—to plot the quantity.

In Console Mode, the corresponding quantities are `NHR` or `BHR`. If you plot NH in Console Mode, you get two curves. The bottom curve is NHR, or the latent heat. See [State Variables for Scheil](#) for a list of the other variables.

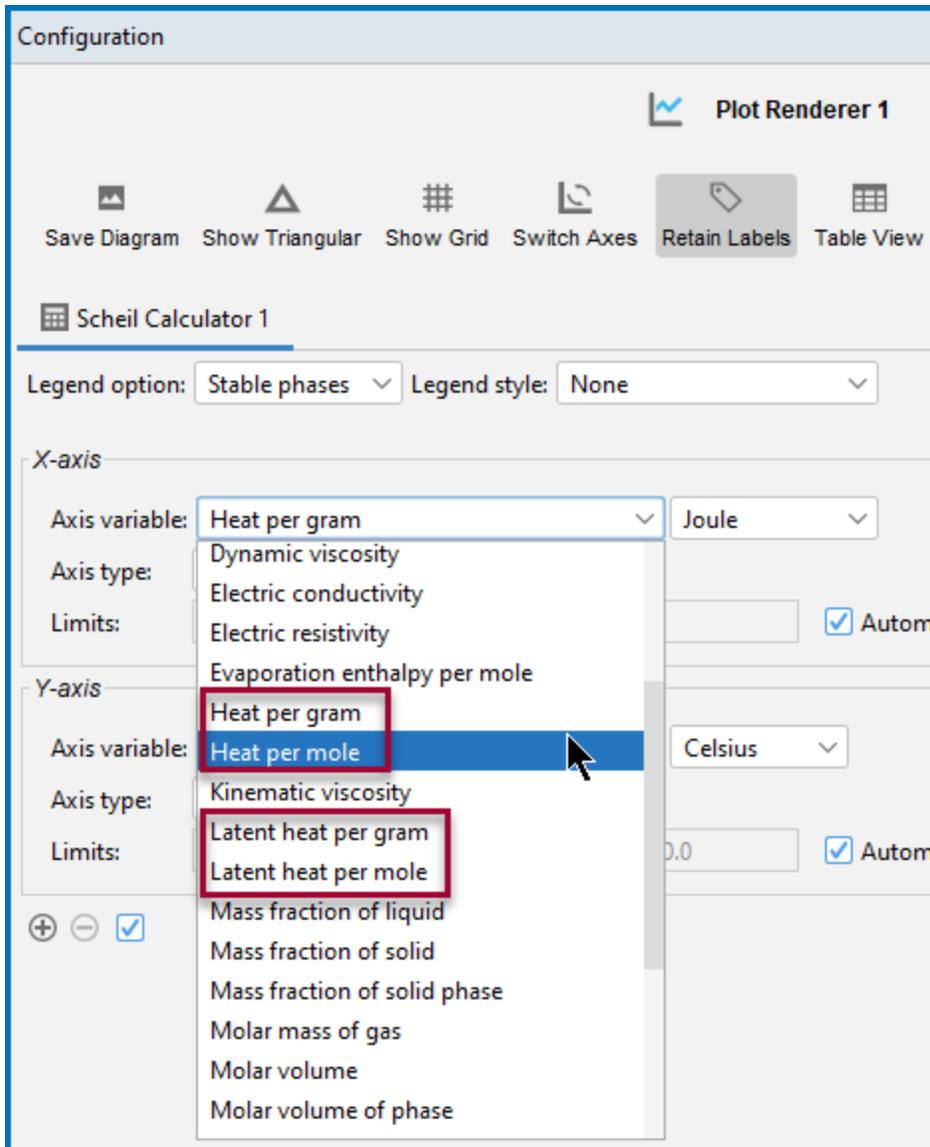


For more detail, see [Why in Console Mode are there two curves when plotting heat evolution after a Scheil simulation?](#)

### ***What is the difference between the Latent heat per mole or gram and just Heat per mole or gram?***

In Graphical Mode, on the Plot Renderer node, in addition to **Latent heat per mole** and **Latent heat per gram** you can also select one of the **Axis variables**—**Heat per mole** or **Heat per gram**. What is the difference?

**Heat per mole** or **Heat per gram** is the total molar (or specific) enthalpy of the system. This includes both the enthalpy change from phase transformations, and from the temperature change, which comes from the heat capacity.



### ***Why in Console Mode are there two curves when plotting heat evolution after a Scheil simulation?***

 The following is applicable to Console Mode.

The top curve is the total heat evolution and it is plotted with the quantity  $NHV$ . This is the total molar or specific enthalpy, including the contribution from both phase transformations and from heat capacity of the phases. The contribution from the solid phases are averaged.

The bottom curve is the axis quantity and it is plotted with the quantity  $N_{HR}$ . This means that this is only the contribution of enthalpy changes from phase transformation from liquid; therefore it starts at 0 at the onset of melting.

### ***How do you plot the distribution of component in phase?***

In Graphical Mode the axis quantity is called **Distribution of component in phase**. This gives the distribution of that element as a fraction of the total molar inventory of that element, from 0 to 1 (0 to 100 % of the overall composition). For example, the total amount of Fe in all phases should then sum up to 1.



Be aware that if using the fast diffuser option there may be issues summing up to 1.

In Console Mode, `NN(Phase, Element)` shows the fraction of this element relative to its total amount in the system, residing in that phase. Here a value of 0.3 means 30 % of all the Fe in the system is in that particular phase.

You can also use wildcards, e.g. `NN(Phase, *)`, which shows all elements' respective fractions of the total in that phase (this does not sum up to 1). This is useful to get an overview of preferential partitioning of elements during the solidification process.

Also useful is plotting `NN(*, Element)`, where you see the distribution of the element over all phases in the system; these curves should all sum up to 1. Here, 0.3 of Fe in Liquid and 0.7 in FCC, would mean that 30 % of the total Fe-content in this alloy is still in the liquid, etc.



Due to calculation issues when working with fast diffusers, this command is not available in Console Mode.

### ***What is the apparent heat capacity?***

The apparent heat capacity is what is going to be measured experimentally; heat capacity is theoretically the derivative of enthalpy with respect to temperature. This apparent heat capacity includes both contributions from phase transformation and from temperature change.

## ***Scheil Resources: Installed Examples***



For Graphical Mode examples, see [T\\_08: Scheil and Equilibrium Solidification](#), [T\\_10: Scheil Solidification with Back Diffusion](#), and [T\\_13: Scheil Solidification with Solute Trapping](#).



For Console Mode examples see [Examples of Scheil-Gulliver Calculations](#).

## ***Scheil Resources: Website Video Tutorials and Application Example***



Link to more information about [Scheil Solidification Simulations](#) on our website. There are also [video tutorials](#) related to some Scheil examples as part of a playlist on our [YouTube channel](#) and an [application example](#) using the Diffusion Module (DICTRA).

## ***Scheil Resources: Self-paced Learning Hub***

The Thermo-Calc Learning Hub has a module dedicated to Scheil Calculations.

Learn about Thermo-Calc and the Add-on Modules at your own pace with our comprehensive online [Learning Hub](#). The Learning Hub includes courses for Thermo-Calc, the Diffusion Module (DICTRA), the Precipitation Module (TC-PRISMA), and the TC-Python SDK. In addition to introductory lectures, instructors walk through and discuss real world examples that correlate calculations with microstructures and other metallurgical concepts, spanning a wide range of material types.

Subscriptions also include access to live monthly office hours, where you can ask our experts about any of the content on the Learning Hub or your own materials challenges.

## Property Model Calculator

---

A **Property Model Calculator** enables you to predict and optimize material properties based on their chemical composition and temperature. The available calculation types are **Single**, **One Axis**, **Grid**, **Min/Max**, **Uncertainty**, and **Batch**.

Using these, you can evaluate models simultaneously over a range of compositions and temperature and cross plot the results, then conduct an uncertainty analysis and plot the results as either a histogram or a probability plot. The **Batch** calculation further enables you to read input from a file, such as a spreadsheet.



[Configuration Window Settings](#)



There are **General Models**, **Steel Models**, and **Nickel Models** available. For more information, see [About the Property Models](#).



You can also create custom Property Models. Search the [TC-Python help](#) on the website for details about creating custom models. In general go to the **Property Model Framework** → **Best Practices** section.

### Property Model Calculator Workflow

At the start when you are designing a project, it is recommended you first test it by performing a **Single** calculation type. This verifies that the configuration of the model is valid, otherwise you will get a *Not a Number* result in the [Event Log Window](#), which indicates the calculation did not work correctly.

Once the Single calculation performs successfully, you can experiment with other calculation types such as **Grid**, **Min/Max**, **Uncertainty**, or **Batch**. Best practice for both Grid and Uncertainty is to start with a small number of steps (Grid) or samples (Uncertainty) to confirm that it works before increasing the number of steps or samples. For Batch calculations, it is good practice to first review the data file requirements described in [Working with Batch Calculations](#).

For a Grid calculation, you can additionally plot a *Heat map*, *Contour*, or *3D* diagram. This setting is done on the [Plot Renderer](#), which has several [Plot Types](#), [Plotting Options](#), and [Plot Types](#) available.

In general, the workflow for each model differs. A recommended start point is to do a **One Axis** calculation (using the [Equilibrium Calculator](#)) to determine what phases you get and where the transitions are before setting up the Property Model Calculator.



There are a variety of related templates both for the equilibrium one axis calculations and to add a **Property Model** node structure. In addition there are **Steel TTT** and **Steel CCT** templates that you can use to quickly set up a simulation for the Steel Model Library. For details, see [Getting Started Links and Templates](#).

## About the Property Models

Property Models are used to predict and optimize properties of materials using models stored within the software where the calculations are set up in the [Property Model Calculator](#).

### General Models

The following General Property Models are available with the Property Model Calculator.

The **General Models** are available to all users.



To run calculations with the **Steel Models** (as part of the Steel Model Library) requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. To run calculations with the **Nickel Models** (as part of the Nickel Model Library) requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases.



[About the Steel Model Library Property Models](#) and [About the Nickel Model Library Property Models](#)

### Coarsening

The **Coarsening** Property Model calculates the coarsening rate coefficient  $K$  ( $m^3/s$ ) of a spherical precipitate phase in a matrix phase.



See [About the Coarsening Property Model](#) and [Coarsening Property Model Settings](#) for background theory and input parameter details.

### Columnar to Equiaxed Transition (CET)

The **Columnar to Equiaxed Transition** Property Model calculates the fraction of equiaxed grains that correlates with a certain solidification condition, specifically thermal gradient ( $G$ ) and solidification growth rate ( $v$ ).



See [About the Columnar to Equiaxed Transition \(CET\) Property Model](#) and [Columnar to Equiaxed Transition \(CET\) Property Model Settings](#) for background theory and input parameter details.

---

## ***Crack Susceptibility Coefficient***

The **Crack Susceptibility Coefficient** Property Model calculates the hot tearing tendency during solidification.



See [About the Crack Susceptibility Coefficient Property Model](#) and [Crack Susceptibility Coefficient Property Model Settings](#) for background theory and input parameter details.

## ***Driving Force***

The **Driving Force** Property Model calculates the thermodynamic driving force for a phase.



See [Driving Force Property Model Settings](#) for input parameter details.

## ***Equilibrium***

The **Equilibrium** Property Model calculates the equilibrium for the given conditions. Optionally define additional function definitions.



See [Equilibrium Property Model Settings](#) for input parameter details.

## ***Equilibrium with Freeze-in Temperature***

The **Equilibrium with Freeze-in Temperature** Property Model calculates equilibrium at the freeze-in temperature and evaluates the properties at a different temperature. The assumption is that diffusion and phase transformations are negligible when changing from the freeze-in-temperature and, therefore, that the phase amounts and compositions of phases are kept at all other temperatures.



See [About the Equilibrium with Freeze-in Temperature Property Model](#) and [Equilibrium with Freeze-in Temperature Property Model Settings](#) for background information and input parameter details.

## ***Interfacial Energy***

The **Interfacial Energy** Property Model estimates the interfacial energy between a matrix phase and a precipitate phase using thermodynamic data from a CALPHAD database.



See [About the Interfacial Energy Property Model](#) and [Interfacial Energy Property Model Settings](#) for background information and input parameter details.

## **Liquid and Solidus Temperature**

The **Liquid and Solidus Temperature** Property Model is for doing a common calculation. For example, you can easily use uncertainty calculations, vary one or more conditions, and see how that affects the liquidus and solidus temperatures.



See [Liquid and Solidus Temperature Property Model Settings](#) for input parameter details.

## **Phase Transition**

The **Phase Transition** Property Model can be used to vary which condition to relax, and then it calculates the point when a new phase may form. It is useful to determine melting temperature, boiling temperature or solubility limits. It returns the phase transformation temperature, or composition, depending on the relaxed condition.



See [Phase Transition Property Model Settings](#) for input parameter details.

## **Scheil**

The **Scheil** Property Model is used to calculate solidification under the Scheil assumption. The majority of the Model input parameters are pulled from the Scheil Calculator to offer the outputs of the liquidus/solidus/solidification range so you can explore these in the calculation types of the Property Models.



See [Scheil Property Model Settings](#) and [Scheil Advanced Options: Property Models](#).

## **Spinodal**

The **Spinodal** Property Model calculates the spinodal line.



See [About the Spinodal Property Model](#) and [Spinodal Property Model Settings](#) for background information and input parameter details.

## **T-Zero Temperature**

The **T-Zero Temperature** Property Model calculates the so-called  $T_0$  line.



See [About the T0 Temperature Property Model](#) and [T0 Temperature Property Model Settings](#) for background information and input parameter details.

## Yield Strength

The **Yield Strength** Property Model calculates yield stress. It returns the calculated yield strength at room temperature for the specified material using the equilibrium values calculated at temperature  $T$  (i.e.  $T$  is the freeze-in temperature). Then you can further define the following contributions to the total yield stress: precipitation strengthening, solid solution, grain boundary strengthening, and constant addition of strength.



See [About the Yield Strength Property Model](#) and [Yield Strength Property Model: Simplified Mode Settings](#) or [Yield Strength Property Model: Advanced Mode Settings](#) for background information and input parameter details.

## Calculation Types

The Property Model Calculator includes these calculation types:

- **Single:** Calculates a single point.
- **One Axis:** Varies a quantity on the X-axis.
- **Grid:** Evaluates two axis variables of the selected quantities in the specified range and number of steps. You can then select to create heat map, contour and 3D plots on the Plot Renderer.
- **Min/Max:** Evaluates the Property Model(s) for all variations of the selected quantities at the given limits. The Mean field is as defined under *Condition Definitions* for the respective quantity. The total minimum and maximum of the Model(s) results are shown in the Event log.
- **Uncertainty:** Evaluates the Property Model(s) where the values of the quantities are sampled from Gaussian distributions. The Mean field is as defined under *Condition Definitions* for the respective quantity. The result is visualized as a histogram or normal probability plot by adding a Plot Render activity.
- **Batch:** Use this to allow for high throughput calculations and read data from a file, such as a spreadsheet. You can include experimental results in the data file, which then can be used to compare to the batch calculation results.



[About the Uncertainty Calculations](#) and [Plot Types](#)

## Examples and Video Tutorials



[Thermo-Calc General Property Models Examples Collection](#)



For tutorials about this feature, go to our [website](#) or browse the Property Model Calculator playlist on our [YouTube channel](#).

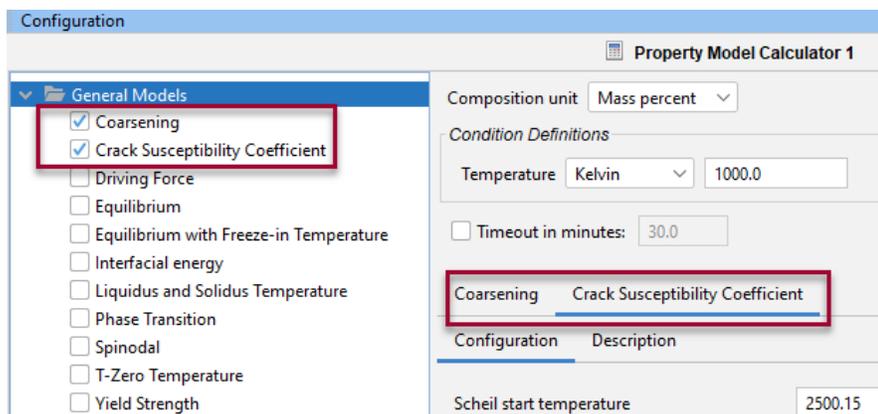
## Define the Property Model Calculator

1. Add a **Property Model Calculator** node to the System Definer. If you used the **Property Models** template, click the node to display the **Configuration** settings window.
2. From the left side of the **Configuration** window, click to select one or more of the checkboxes under the **General Models**, **Steel Models**, or **Nickel Models** folders.



Steel Models are only available if you have a license for the Steel Model Library. Nickel Models are only available if you have a license for the Nickel Model Library. If you are interested in learning more about the available models, go to [our website](#).

A tab with the same name as the Property Model opens in the right side of the window under *Condition Definitions*. When you select more than one Model at a time, a matching tab is added in the same order that the checkbox is selected.



3. For each Model on the **Configuration** tab, select or enter settings described in [Configuration Window Settings](#) and based on the *Calculation Type* chosen.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.

4. Once you have finished defining the Property Model Calculator, you also choose settings on the [Plot Renderer](#).

## Examples and Video Tutorials



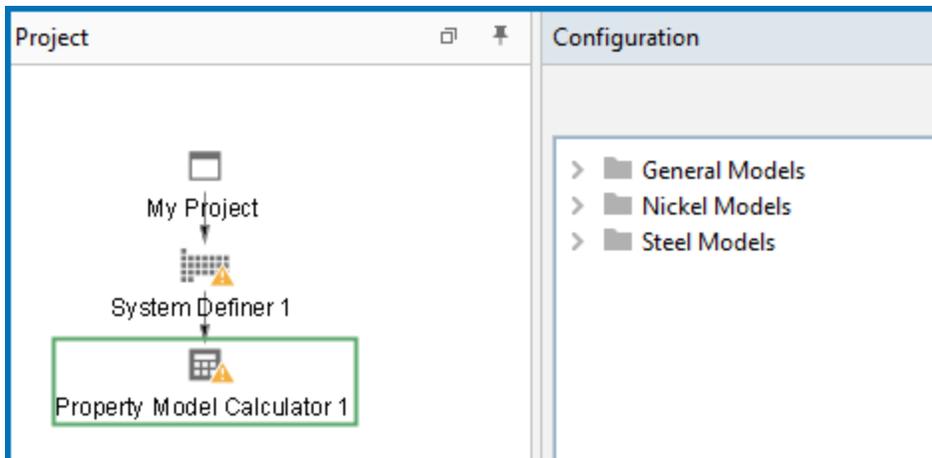
[Thermo-Calc General Property Models Examples Collection](#)



For tutorials about this feature, go to our [website](#) or browse the Property Model Calculator playlist on our [YouTube channel](#).

## Configuration Window Settings

Click the **Property Model Calculator** node in the **Project** window to view the **Configuration** window. The settings available vary based on such choices as the Property Model and calculation type selected.



### Composition Unit

Select the **Composition unit**: **Mass percent**, **Mole percent**, **Mass fraction**, or **Mole fraction**.

### Condition Definitions

Choose a unit (**Kelvin**, **Celsius** or **Fahrenheit**) and then enter a value for the **Temperature** and **Composition** for each component.

The values entered in *Condition Definitions* are used as the start values for the different calculation types.

## Timeout in Minutes

Select the **Timeout in minutes** checkbox to enter a time when a calculation will stop trying to complete the evaluation. This is useful if it is a complex or unknown calculation outcome and you want to prevent it from taking too much computer time or resources. The number entered in the field means that this is the maximum amount of time that each line can take for an individual calculation. This is useful in particular for Batch or Uncertainty calculations.

## Define the Property Model

For each Property Model chosen, select or enter **Configuration** settings as required.



For links to each Model with the settings details, see [About the Property Models](#).



When working in the **Configuration** window, click the **Description** tab for more information about the Model.

## Calculation Type

Click to select a *Calculation Type*:

- **Single** to calculate a single point. Any model can use this calculation type and no additional settings are required. The results from this calculation are displayed in the **Event Log** or if a **Table Renderer** is used, in the **Visualizations** window.
- **One Axis**: To vary a quantity on the X-axis.
- **Grid**: Evaluates two axis variables of the selected quantities in the specified range and number of steps.
- **Min/Max**: Evaluates the Property Model(s) for all variations of the selected quantities at the given limits. The **Mean** field is as defined under *Condition Definitions* for the respective quantity. The total minimum and maximum of the model(s) results are shown in the **Event log**.
- **Uncertainty**: Evaluates the Property Model(s) where the values of the quantities are sampled from Gaussian distributions. The **Mean** field is as defined under *Condition Definitions* for the respective quantity. The result is visualized as a histogram or normal probability plot by adding a Plot Renderer activity.

- **Batch:** Use this to allow for high throughput calculations and read data from a file, such as a spreadsheet, rather than, for example, having a fixed grid or random values. Results from these batch calculations can then be compared to experimental values (as long as this data is included), using a **Cross plot** on the Plot Renderer.

## Axis Definition

For **One Axis**, select a **Quantity** to vary along the X-axis, for example, **Temperature** then enter a **Min**, **Max** and **Number of steps**.

## Grid Definitions

For the **Grid** calculation type, define the two axes variables using the fields and menus: **Quantity**, **Min**, **Max**, and **Number of steps**. The number of steps along with the minimum and maximum values for the axes define a grid. For each grid point the selected models are evaluated.



For a **Grid**, on the Plot Renderer, you can choose to plot a *Heat map*, *Contour*, or *3D* diagram.



[Plot Type: Heat Map, Contour, and 3D](#)

## Min/Max Definitions

For the **Min/Max** calculation type, click to select the applicable checkboxes under **Quantity to enable or disable an axis**. The **Mean** field is as defined under *Condition Definitions* for the respective quantity.

Enter numerical values in the **ΔMin/Max** fields to evaluate the change to the quantity. The highest and lowest value of each selected model results are shown in the **Event Log**.

## Sampling of Data from Gaussian Distributions

For the **Uncertainty** calculation type, click to select the applicable checkboxes under **Quantity to vary this quantity**. The **Mean** field is for the distribution function as defined under *Condition Definitions* for the respective quantity. The default **ΔMin/Max** is a 95% confidence interval.



[About the Uncertainty Calculations](#)

## Sampling Parameters

For the **Uncertainty** calculation type, choose a number of **Standard deviations** where you want to **Truncate the distribution**. Choose the **Total number of samples**.

 [About the Uncertainty Calculations](#)

## Batch Controls

 For more details about the format of a data file used to define the compositions, parameters, experimental data, or skipping validation, see [Working with Batch Calculations](#).

In the **Data file** field, either enter a file path or click the load data file button () to navigate to a data file such as an Excel spreadsheet (with \*.xls or \*.xlsx extensions) or a \*.csv file. You can also use other programs (e.g. Google Sheets) to enter data and export to a CSV format that can be read by the software.

Click **Open** to import the data. In the **Data file** field you can see the file path to the selected file. The next time you click the load data file button () the program remembers the last location from where the file is opened.

Choose the delimiter—**Comma (,)**, **Colon (:)**, **Equals sign (=)**, **Semicolon (;)**, or **Tab**—that matches the delimiter in the data file and click the **Reload** () button.

The data is imported and displays on the Property Model Calculator:

- The data are used for the applicable fields that must match what already exists as a Property Model parameter (e.g. critical radius, grain size, etc.) as well as the composition, temperature, and units that you want to run the calculation for. Also, if the dependent element composition is not included in the data file, the batch calculation then sets the dependent element to be the same as defined in the user interface.
- The calculation runs once for every row of data included in the file.



When the data file is imported into Thermo-Calc, there is also a **Row** column autogenerated by Thermo-Calc.

Select the **Set zero amount to value (in mole-/mass-fraction)** to enter a small value for the mole/mass-fraction in cases where a Property Model has problems calculating zero amounts of elements.

Select the **Skip validation of data file** checkbox to allow a calculation to perform using the data file even when certain criteria are not met.

When selected, the validation of the data file is skipped:

- If the **Composition** (elements) do not match the system composition entered on the GUI in the System Definer or Property Model Calculator.
- If one or more **Parameters** are not correctly entered, not selected for the specific Property Model, or are unavailable with the selected Property Model, then these are ignored in the calculation.



If there are any data file validation error(s), a tooltip on the **Perform** button lists and describes these.

## Diffusion Calculator

---



The Diffusion Calculator is available with three elements if you do not have the additional Diffusion Module (DICTRA) license. With the Add-on Module you can use all available elements as per the licensed database selected. See [Available Graphical Mode Features](#) for details about this and other options available with additional licenses.

A **Diffusion Calculator** allows you to set the conditions for, and perform, a diffusion calculation. You can add the calculator to the System Definer directly (right-click and select it from the **Create New Successor** menu).



There are also templates available under **Non-Equilibrium** that you can use to quickly set up a simulation. See [Getting Started Links and Templates](#).

Once you have added a Diffusion Calculator, the **Configuration** window has **Conditions** and **Options** settings tabs where there are many available conditions to set and then continue the settings with the specialized **Plot Renderer**.

- **Conditions:** Set the conditions for your calculation that define the geometry, regions, phases, grids, composition profiles, and thermal profile.



The **Composition Profiles** and **Thermal Profile** are both set up on the **Configuration** window for a Diffusion Calculator. These are previewed in the **Visualizations** window.

- **Options:** Modify the simulation conditions such as the choosing the default solver (automatic or homogenization) and the time integration method. Also define the time step controls and if you choose homogenization as the solver, define further settings such as the interpolation scheme and whether to use global minimization or

not.

- **Plot Renderer:** There are also unique settings available for this calculator once you add a [Diffusion Calculator Plot Renderer](#).



Working with these simulations is an advanced topic. It is recommended you review the documentation and videos available or search for a specific topic.

## Available Graphical Mode Features

The Diffusion Module is an Add-on Module to the core Thermo-Calc software.



A separate license is required to perform calculations for more than three elements. Without it you are able to use the module in *Demo Mode*. This is not available in Console Mode.

### *Diffusion Templates*

Under **Non-Equilibrium Calculations**, the **Homogenization** and **Diffusion** templates are available to all Thermo-Calc users when in Graphical Mode.



If you are using the Diffusion Module in Demo Mode, see [Demonstration \(Demo\) Mode](#) for what is available to you. Even if you have a license, you may find yourself in Demo Mode if you have a network license and all the licenses are checked out.

### USING THE TEMPLATES

After opening Thermo-Calc in Graphical Mode, in the templates section under **Non-Equilibrium Calculations** click:

- **Homogenization** to add a *System Definer*, *Scheil Calculator*, *Diffusion Calculator*, and two *Plot Renderer* nodes to the **Project** tree.
- **Diffusion** to add a *System Definer*, *Diffusion Calculator*, and *Plot Renderer* to the **Project** tree.

### [Creating a Project from a Template](#)

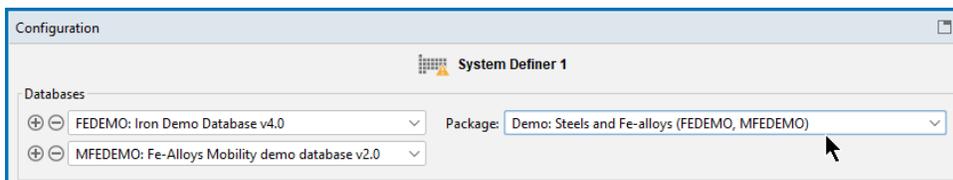
## **Diffusion Calculator**

A Diffusion Calculator allows you to set the conditions for your calculation that define the geometry, regions, phases, grids, composition profiles, and the thermal profile, plus additional options. Many of the settings are automatically set as you define your system. Details about this activity are described in [Diffusion Calculator](#).

## **Demo Database Packages for the Examples Collection**

Both a thermodynamic and mobility database are required to run the Diffusion Module simulation. If you have a Diffusion Module (DICTRA) license you can run all the examples because the demonstration database packages are included with your installation.

Select the database packages from the **System Definer Configuration** window to run a simulation.



## **Demonstration (Demo) Mode**

The Diffusion Module and some examples are available to all Thermo-Calc users but only for simulations with three elements. If you do not have a license then you are in *Demo Mode* when using the Diffusion Calculator, or either of the templates.

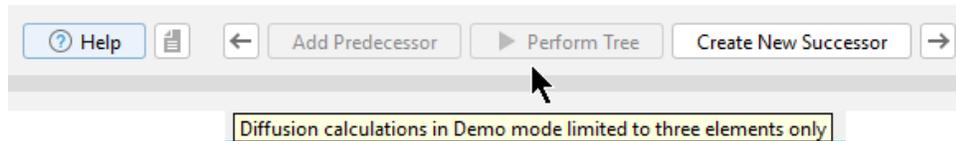
## **DIFFUSION TEMPLATES**

If you are in DEMO mode then this is indicated by the addition of **DEMO** text under the button.

## DIFFUSION CALCULATOR

If you are experimenting with the Diffusion Calculator in Demo Mode then you may have access to a variety of databases based on your license. However, you can only define three elements for a demo simulation.

If you define more than three elements on the System Definer, when you go to the Diffusion Calculator, the **Perform** button is unavailable and the tooltip explains why.



Even if you have more than three elements, the Plot Renderer **Perform** button is still available. However, if you click the button and try to run the simulation the **Event Log** displays an error message.

## Additive Manufacturing (AM) Module

There is connectivity between the Diffusion Module (DICTRA) or the Precipitation Module (TC-PRISMA) and the Additive Manufacturing (AM) Module when you have full licenses available for all relevant Add-on Modules and databases.

You can add a Diffusion Calculator or Precipitation Calculator as a successor to the AM Calculator and then when probes are included with the AM Calculator this data can be further used with the thermal profile.

Details are in the AM Module documentation. See [AM Calculator: Working with Probe Data](#).

## Define the Diffusion Calculator

Working with diffusion simulations is an advanced topic. It is recommended you review the Diffusion Module (DICTRA) documentation and videos available or search for a specific topic.



1. Add a **Diffusion Calculator** node to the System Definer. If you used the **Diffusion** template (on the  **My Projects Configuration** window under **Non-Equilibrium**), click the node to display the **Configuration** settings window.
2. In the Diffusion Calculator **Configuration** window, enter the settings described in [Conditions Tab Settings](#) and [Options Tab Settings](#).



The **Composition Profiles** and **Thermal Profile** are both set up on the **Configuration** window for a Diffusion Calculator. These are previewed in the **Visualizations** window.



When you also have licenses either for the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA), for transient simulations you can additionally include probe data from the AM Calculator in the set up of the thermal profile for diffusion and precipitation calculations, respectively.

3. Once you have finished defining the Diffusion Calculator, you also choose settings on the [Plot Renderer](#). Some additional settings are specific to the [Diffusion Calculator Plot Renderer](#).



[Available Graphical Mode Features](#)



[Diffusion Module \(DICTRA\) Examples Collection](#)

## Conditions Tab Settings

The following Configuration window is from the *Graphical Mode Quick Start Guide*. It is using project example file *D\_07: Diffusion Carburization Multiphase*.

Configuration Carburization multiphase

Conditions Options

Composition unit: Mass percent

Length unit: Meter

Geometry:  Planar

Austenite

Austenite

Left Interface Boundary

Region

Name: Austenite

Width: 0.003  m 100  points and type Geometric  Ratio: 1.02

Phase: FCC\_A1

Phase: M7C3

Phase: M3C2

Composition profiles:

Right Interface Boundary

Left Boundary Condition

Mixed zero flux and activity  Cr Zero flux

Activity

Tip: The reference state is defined in the System Definer

Right Boundary Condition

Closed system

Thermal Profile

Isothermal  Non-isothermal

Temperature: 1123.0  Kelvin

Simulation time: 1000.0  Hours



These are the settings available from the **Conditions** tab. To edit the global defaults for some settings, go to the **Options** window (different from the tab). See [Global Settings: Diffusion Calculator](#).



The Diffusion Module (DICTRA) in Console Mode has commands to perform some of the settings also available on the Diffusion Calculator. Where possible the name of the command is indicated and includes a link to the topic.

## General

### Composition Unit

Choose from **Mass percent**, **Mole percent**, **Mass fraction**, or **Mole fraction**.

### Length Unit

Choose **m**, **μm**, or **nm**.

### Geometry

Choose a **Geometry**: **Planar**, **Spherical**, or **Cylindrical**. For **Cylindrical**, also enter a value for the **First interface position**.

With a cylindrical or spherical geometry, the system's zero coordinate (left boundary) is at the center of the cylinder or sphere. The highest coordinate (right boundary) is defined by the cylinder or sphere radius.

<i>Geometry</i>	<i>Graphic</i>
Planar	
Spherical	
Cylindrical	



The **Composition Profiles** and **Thermal Profile** are both set up on the **Configuration** window for a Diffusion Calculator. These are previewed in the **Visualizations** window.

## Region Settings



There should be one unique phase in each region.

### Left and Right Interface Boundary

Toggle between hiding and showing the **Left interface boundary** or **Right interface boundary**.

Click the **Add quantity**  $\oplus$  or **Remove quantity**  $\ominus$  buttons as required to add a **Phase** and then enter a value for the **Driving force** as required. The default driving force is  $1.0E-5$ .

### Name

Enter a **Name** for the region in the field. For example, enter `Austenite`. This name is automatically updated on the tab and above the grid points graphic.

### Width

Enter a numerical value in the **Width** field. The unit is the one selected for the **Length unit**.

### Points

Enter a numerical value in the **points** field. The default is 50.

### Type (of Grid)

Select a type of grid, **Linear**, **Geometric**, or **Double geometric**.

Choose **Geometric** for a grid that yields a varying density of grid points in the region. Then enter a **Ratio** to determine the distribution. A geometrical factor larger than one yields a higher density of grid points at the lower end of the region and a factor smaller than one yields a higher density of grid points at the upper end of the region.

Choose **Double geometric** to have a high number of grid points in the middle or at both ends of a region. Then specify two geometrical factors, the **Lower ratio** for the distribution in the lower (left) part of a region, and **Upper ratio** for the distribution in the upper (right) part of a region.

## Phase

Choose a **Phase** from the list. The available options are defined in the System Definer. At least one phase that dissolves all system elements must be selected.

Click the **Add phase**  or **Remove phase**  buttons as required.



The **Composition Profiles** and **Thermal Profile** are both set up on the **Configuration** window for a Diffusion Calculator. These are previewed in the **Visualizations** window.

## Boundary Conditions

Boundary conditions are conditions that define how matter behaves at the boundaries of your system. By default, matter is not allowed to cross the system boundaries.

You can change the setting for both the lower boundary (left side/centre) and the upper boundary (right side/surface) of the system.



Examples D\_06 and D\_07 use boundary conditions, as described in [Diffusion Module \(DICTRA\) Examples Collection](#).

The boundary condition settings are done on the **Diffusion Calculator**. The following describes the options available for the *Left boundary* and *Right boundary* conditions.

### Closed System

By default it is a **Closed system**. Choose **Mixed zero flux and activity** or **Composition** if the system is not closed. Corresponds to a fix flux value, which is set to zero at all times. This is the default boundary condition.

### Composition

Allows you to set a fix composition at the boundary.



In Console Mode this is similar to the command STATE\_VARIABLE\_VALUE, but in Graphical Mode this condition is limited to setting a fixed composition.

---

## Mixed Zero Flux and Activity

The flux of selected components is either set to zero or a prescribed activity value.

For **Mixed zero flux and activity**, and for each independent element selected in the **Composition profile**, choose either **Zero flux** or **Activity** from the lists and enter a value in the field when **Activity** is selected. The reference state for the components are taken from the System Definer **Components** tab.

In the field for **Activity**, enter a formula that the software evaluates during the calculation.

The formula can be:

- A function of the variable TIME
- A constant

The formula must be written with these rules:

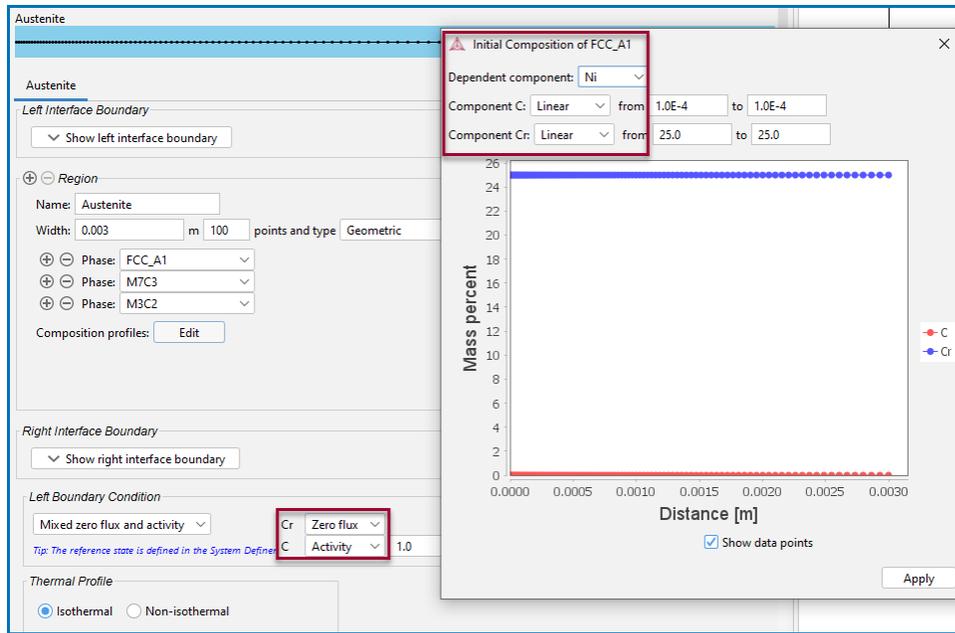
- A number must begin with a number (not a dot)
- A number must have a dot or an exponent (E)

The operators + , - , \* , / , \*\* (exponentiation) can be used and with any level of parenthesis.

As shown, the following operators must be followed by ( ) i.e. open and closed parentheses:

- SQRT(X) is the square root
- EXP(X) is the exponential
- LOG(X) is the natural logarithm
- LOG10(X) is the base 10 logarithm
- SIN(X), COS(X), TAN(X), ASIN(X), ACOS(X), ATAN(X)
- SINH(X), COSH(X), TANH(X), ASINH(X), ACOSH(X), ATANH(X)
- SIGN(X)
- ERF(X) is the error function

This image is from example *D\_07: Diffusion Carburization Multiphase*.



## Composition Profiles

The **Composition Profiles** are set up on the **Configuration** window for a Diffusion Calculator. These are viewed in the **Visualizations** window.

For most calculations, you can include data as follows.

- *Given by function* (linear, step, or function). See [Composition Profiles: Given by Function](#).
- *Table input* that can be either directly entered or imported. See [Composition Profiles: Table Input \(i.e. Data File Import\)](#).
- *Solidification data*. For a **Diffusion Calculator** that is a successor to the **Scheil Calculator**, you can perform a Scheil solidification calculation followed by kinetic homogenization simulation of segregated composition profile. In this case, there are different options on the **Configuration** window for this section. The profile then uses information from the Scheil Calculator. See [Composition Profiles: Scheil Segregation Profiles](#).



When using the **Table input** setting, you can directly enter data in the table or you can import a **Data file**. If you import a data file, even though it is a simple set of data, first prepare it based on the [Guidelines for Importing Data Files to Composition Profiles](#).

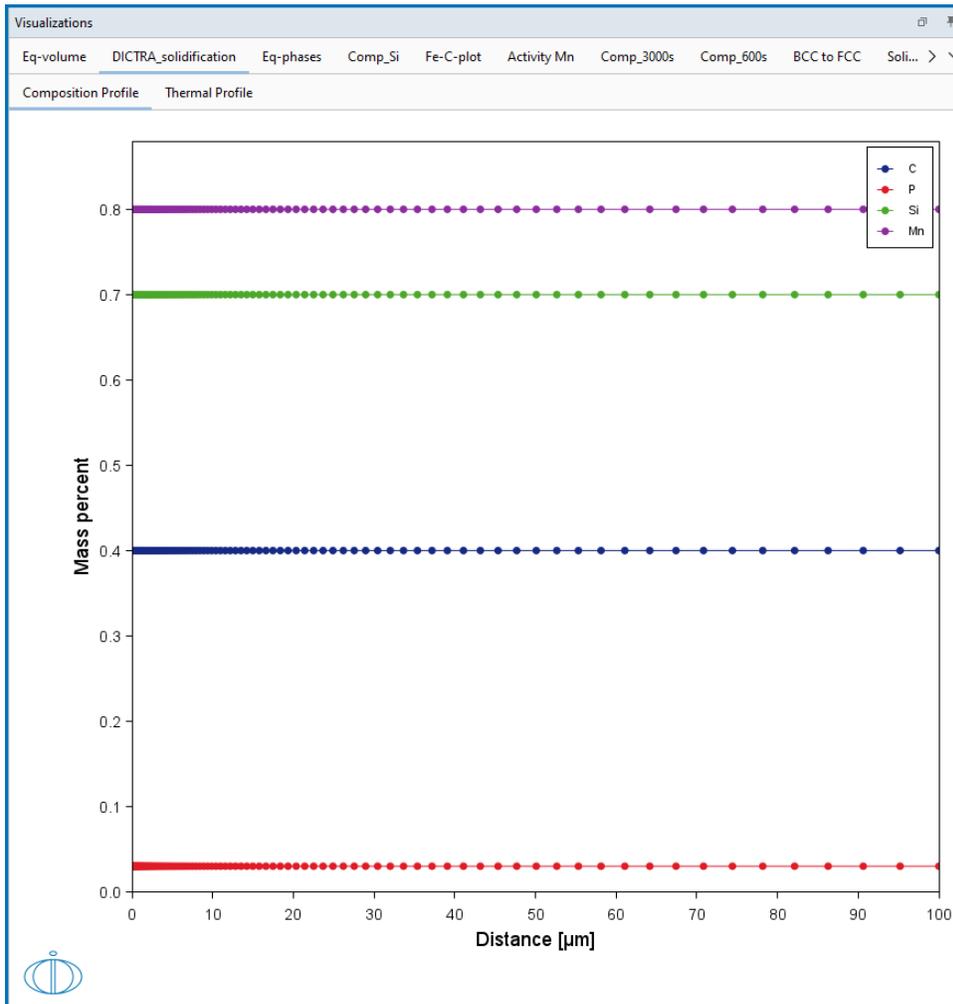


Figure 2: This composition profile is from example D\_08 Microsegregation During Solidification. Other examples using this feature are D\_04\_Diffusion\_Fe-C\_Moving\_Boundary\_Austenite\_to\_Ferrite and D\_09 Ni\_Post\_Weld\_Heat\_Treatment\_From\_File, which are all included with your installation.

## Composition Profiles: Given by Function

1. On the Diffusion Calculator **Configuration** window, first enter the required settings described in [Conditions Tab Settings](#) including the [Region Settings](#) and [Boundary Conditions](#).
2. Click to select the **Composition profiles** → **Given by function** option.
3. Select a **Dependent component** for which the composition does not need to be given. The available options depend on the phase description.
4. For each **Component** choose one of these options:
  - **Linear**: Enter numerical values in the **from** and **to** fields.
  - **Step**: Enter numerical values in the **from**, **to**, and **step at** fields. The unit is the same as the **Length unit** selected at the top of the Configuration window.
  - **Function**: Enter a composition function in the field using SI units. The syntax to use for writing functions is a Python syntax (a programming language). Any valid Python expression that is a function of the distance variable  $x$  will work.



You can use the standard Jython math library but you must prefix these functions with the text "math". For example: `10 + math.sin(x*2)`. The Jython math library is available here: <https://www.jython.org/>.

- The error function  $\text{erf}(x)$  is also available in the Jython math library and can be called with `math.erf(x)`.
  - Another useful function is the Heaviside step-function  $\text{hs}(x)$ . For example, `3+2hs(x-1e-4)` provides a concentration of 3 at the left side and 5 at the right side with a sharp step in the concentration profile at  $1\text{e-}4\text{m}=100\text{ mm}$ .
5. The **Composition Profile** is previewed in the **Visualizations** window as in [Figure 2](#). Make adjustments to the settings as needed and watch it dynamically change in the

window.

6. After completing this section, also continue with [Boundary Conditions](#) and the [Diffusion Calculator Thermal Profile](#).

### *Composition Profiles: Table Input (i.e. Data File Import)*

1. On the Diffusion Calculator **Configuration** window, first enter the required settings described in [Conditions Tab Settings](#) including the [Region Settings](#) and [Boundary Conditions](#).
2. Click to select the **Composition profiles → Table input** option.
3. Select a **Dependent component** for which the composition does not need to be given (i.e. it is excluded from the table). The available options depend on the phase description. The **Composition unit** and **Length unit** are taken from the settings already entered.
4. To enter data directly into the table:
  - a. Click in each cell under the applicable columns, e.g. **Distance** and then each individual component such as **Fe, Al**, etc.
  - b. Directly enter the values in the table. Press <Enter> to add rows and use the scroll bars to navigate the table.
5. To import a data file:
  - a. It is important to first prepare the data file as in [Guidelines for Importing Data Files to Composition Profiles](#).
  - b. When the data file is ready, to the right of the **Data file** field, click the file  button to load the file.
  - c. Navigate to a data file such as an MS Excel spreadsheet (with **\*.xls** or **\*.xlsx** extensions) or a **\*.csv** file. You can also use other programs (e.g. Google Sheets) to enter data and export to a CSV format that can be read by the software. Click **Open** to import the data to the **Configuration** window table.

- d. If there are issues with the above import, experiment with choosing the delimiter type that corresponds to the data in the file: **Comma** ( , ), **Colon** ( : ), **Equals sign** ( = ), **Semicolon** ( ; ), or **Tab**—that matches the delimiter in the data file. Click the **Reload** button . The data is imported to the **Configuration** window table.
6. The **Composition Profile** is previewed in the **Visualizations** window as in [Figure 2](#). Make adjustments to the settings as needed and watch it dynamically change in the window.
7. After completing this section, also continue with [Boundary Conditions](#) and the [Diffusion Calculator Thermal Profile](#).

### *Composition Profiles: Scheil Segregation Profiles*

This section is specific to when the **Homogenization** template is used, or generally when a **Diffusion Calculator** is a successor to the **Scheil Calculator**.

When this is the case, the *Composition profiles* take information from the Scheil calculation, and then you can choose to **Average over all solidified phases** (the default), where **All solidified phases** is further selected by default or you can define by **Custom phases**.

Alternatively, choose **Primary dendrite phase**, where it is assumed that other phases formed during solidification will not dissolve during homogenization. The **Primary dendrite phase** can only be BCC, FCC, or HCP.



Also when **Primary dendrite phase** is selected, the Scheil segregation profiles for fast diffusers are not available. The alloy composition for fast diffusers that is defined on the **Scheil Calculator Configuration** window next to the **Fast diffusers** checkbox is used instead.



The segregated composition profile can be assumed to be equal to a concentration profile for regular Scheil assumptions. i.e. the segregated composition profile as a function of solidified material can be directly mapped to a concentration profile given the secondary arm spacing. This direct relation between segregated and concentration profiles is no longer valid when fast diffusers are used. Then the fast diffusers equilibrate at all temperatures during the Scheil solidification and the concentration profiles for fast diffusers therefore change at every temperature. The average segregated profile for all elements (also fast diffusers) as a function of solidified material represent the composition profile at the final temperature at the end of solidification.



To learn about the **Homogenization** template, see [Getting Started Links and Templates](#).



Example [D\\_10: Iron \(Fe\) Homogenization in Scheil](#) uses the **Homogenization** template to demonstrate the use of the Scheil data with a diffusion calculation.

## Guidelines for Importing Data Files to Composition Profiles

The **Composition Profiles** settings are done on the **Configuration** window for a **Diffusion Calculator**. The profile is then previewed in the **Visualizations** window.

You can import data from a file. There are specific guidelines needed to correctly prepare the data in the file to ensure it imports without errors.

### *Data File Requirements*

The following are instructions about setting up the data file when **Table input** is selected for the [Composition Profiles](#).

The data in the spreadsheet or csv file can include the distance and the compositions, plus the parameters defining the dependent component, composition unit, and length unit. In order for the data to be included in the calculation, the column and row definitions in the file must adhere to certain rules of entry.



Only the columns described here are read / used in the calculation. It is important to enter the information exactly as described. It is recommended that the data file is cleaned up before importing / loading it and performing calculations.

	A	B	C	D	E	F	G
1	Distance	Mn	Cr	Ti	dependent_component=fe	composition_unit=mass_pct	length_unit=micrometer
2	0	0	0	0			
3	500	0.1	0.6	1.1			
4	1000	0.2	0.12	1.2			
5	1500	0.5	0.19	1.5			
6	2000	0.1	0.1	1.3			
7	3000	0.1	0.05	1.1			

Figure 3: This is an example of data entered into an Excel spreadsheet (which is then converted to csv format). Columns A to D show the data entered to create the Composition Profile itself. Columns E, F, and G are to define default settings although these can also be defined on the Configuration window.

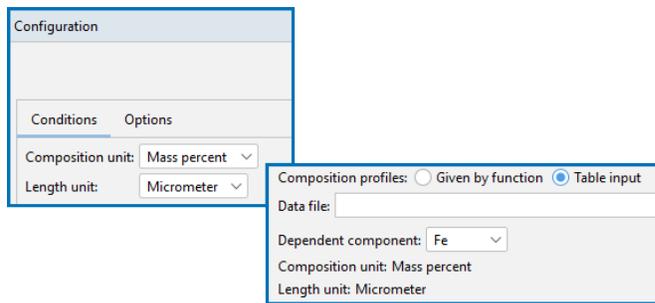


Figure 4: Three columns in the spreadsheet correspond to settings that can either be entered on the Configuration window, or defined in the data file. This is for the dependent component (column E), composition unit (column F), and length unit (column G). When this information is not specified in the data file, the default is taken from the settings defined on the Configuration window.

## Distance and Composition/Components Plus Values

### General Rules

- No abbreviations are allowed.
- All header text must be in English, i.e. no translations into other languages will be read correctly by the data file.

- The composition columns **MUST** match what is already entered on the System Definer or the file will not load.



In the table, `code font` indicates something entered into the data file (or it is an example) and any *<text in italic with angle brackets>* is a generic name for an entry.

Column Heading Requirements and Options	Row Cell Requirements and Options
<p><b>Distance</b></p> <p>This is column A as in <a href="#">Figure 3</a></p> <p>Enter the text <code>Distance</code>. No abbreviations allowed but it is not case sensitive.</p>	<p>Valid format and entries for each row in this column:</p> <ul style="list-style-type: none"> <li>• Any numerical input for the distance e.g. 500, 0.15, 8.7, 1.00E-05, etc.</li> </ul>
<p><i>&lt;Components / Composition&gt;</i></p> <p>These are columns B to D as in <a href="#">Figure 3</a></p> <p>Each element of the composition is in a separate heading column using the <i>&lt;element short name&gt;</i></p> <p>This is already included in the table header on the <b>Configuration</b> window and based on the entries selected on the System Definer.</p> <p>Enter an <i>&lt;element short name&gt;</i> into each column, row 1. e.g. in this example, <code>Mn</code>, <code>Cr</code>, and <code>Ti</code> are entered in columns B, C, and D, respectively. Entries are not case sensitive, so alternatives in this example for Mn can be <code>mn</code> or <code>MN</code>.</p>	<p>Valid format and entries for each row in these columns:</p> <ul style="list-style-type: none"> <li>• A number for the composition e.g. 0.15, 8.7, 1.00E-05, etc.</li> </ul> <div data-bbox="857 1024 1386 1388" style="border: 2px solid red; padding: 10px;"> <p> The elements in the file must match what is selected on the System Definer. (1) If it is in the file but not selected on the System Definer, then it is not added to the table, and (2) if the opposite is true, that an element is selected on the System Definer but not included in the file, then an empty column is added and manually you must enter the details.</p> </div>

## Composition Unit, Length Unit, and Dependent Component

### General Rules

- The defaults can be selected on the **Configuration** window, or can be entered in its own column in the data file. When this is not specified in the data file, the default is taken from the settings defined on the Configuration window. See [Figure 4](#).

- No abbreviations allowed but it is not case sensitive. For example, it is acceptable to enter `composition_unit` or `Composition_Unit`, or `COMPOSITION_UNIT`, etc.
- All header text must be in English, i.e. no translations into other languages will be read correctly by the data file. For example, do not add the accents to the A and O in the word Angstrom, e.g. Ångström cannot be read correctly.
- Do not enter anything in the rows in each of these columns.



In the table, `code font` indicates something entered into the data file (or it is an example) and any *<text in italic with angle brackets>* is a generic name for an entry.

Column Heading Description	Column Heading Requirements and Options
<p>The <b>Dependent component</b> in the data file is defined with the heading <code>dependent_component=&lt;element&gt;</code></p> <p>In <a href="#">Figure 4</a>, this is in column E.</p> <p>i.e. <code>dependent_component=fe</code>. Add the heading to its own column to set the <b>Dependent component</b> to <b>Fe</b>.</p>	<p>Valid format and entries of the <i>&lt;dependent_component&gt;</i> (the text is added after the = sign):</p> <ul style="list-style-type: none"> <li>• Any of the components/elements selected already on the System Definer can be entered.</li> </ul>
<p>The <b>Composition unit</b> in the data file is defined with the heading <code>composition_unit=&lt;unit&gt;</code>.</p> <p>In <a href="#">Figure 4</a>, this is in column F.</p> <p>i.e. <code>composition_unit=mass_pct</code>. Add the heading to its own column to set the <b>Composition unit</b> to <b>Mass percent</b>.</p>	<p>Valid format and entries of the <i>&lt;composition_unit&gt;</i> (the text is added after the = sign):</p> <ul style="list-style-type: none"> <li>• <code>mass_pct</code></li> <li>• <code>mass_frac</code></li> <li>• <code>mole_pct</code></li> <li>• <code>mole_frac</code></li> </ul>
<p>The <b>Length unit</b> for the distance in the data file is defined with the heading <code>length_unit=&lt;unit&gt;</code>.</p> <p>In <a href="#">Figure 4</a>, this is in column G.</p> <p>i.e. <code>length_unit=micrometer</code>. Add the heading to its own column to set the <b>Length unit</b> for the distance to <b>Micrometer</b>.</p>	<p>Valid format and entries of the <i>&lt;length_unit&gt;</i> (the text is added after the = sign):</p> <ul style="list-style-type: none"> <li>• <code>Meter</code></li> <li>• <code>Millimeter</code></li> <li>• <code>Micrometer</code></li> <li>• <code>Nanometer</code></li> <li>• <code>Angstrom</code></li> </ul>

Column Heading Description	Column Heading Requirements and Options
	<div style="border: 2px solid red; padding: 10px; display: inline-block;">  <p>Enter <code>Angstrom</code> without the accents on A and O, e.g. Ångström will not be read correctly.</p> </div>

## Diffusion Calculator Thermal Profile

The **Thermal Profile** is set up on the **Configuration** window for a **Diffusion Calculator**. The isothermal or non-isothermal profile is viewed in the **Visualizations** window. Choose **Isothermal** to do a simulation at constant temperature. Choose **Non-isothermal** to enter or import data for a user-defined temperature profile.



When connected to an AM Calculator (see [Including Probe Data from an AM Calculator](#)) the **Isothermal** and **Non-isothermal** options are not visible as it is not relevant to that view.

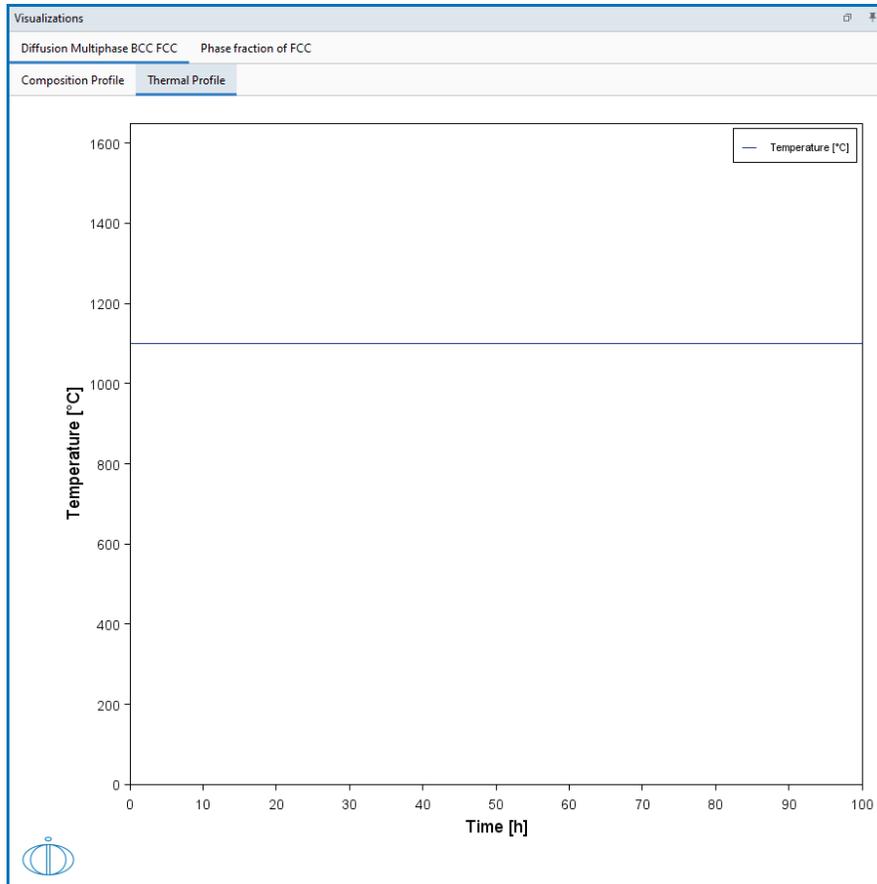
### *Isothermal Calculations*



Many of the Graphical Mode examples for this Add-on Module are isothermal calculations, e.g. *D\_03\_Diffusion\_Multiphase*.

1. On the Diffusion Calculator **Configuration** window, first enter the required settings described in [Conditions Tab Settings](#) including the [Region Settings](#), [Boundary Conditions](#), and [Composition Profiles](#).
2. Under **Thermal Profile** click to choose **Isothermal** to do a simulation at constant temperature.
3. Enter a **Temperature** and choose a unit (**Kelvin**, **Celsius**, or **Fahrenheit**).
4. Enter a **Simulation time** and choose a unit: **Seconds**, **Minutes**, **Hours**, or **Days**.

The **Thermal Profile** is displayed in the **Visualizations** window showing the constant temperature entered.



5. For this calculation type this completes the settings you need to enter for the Thermal Profile.

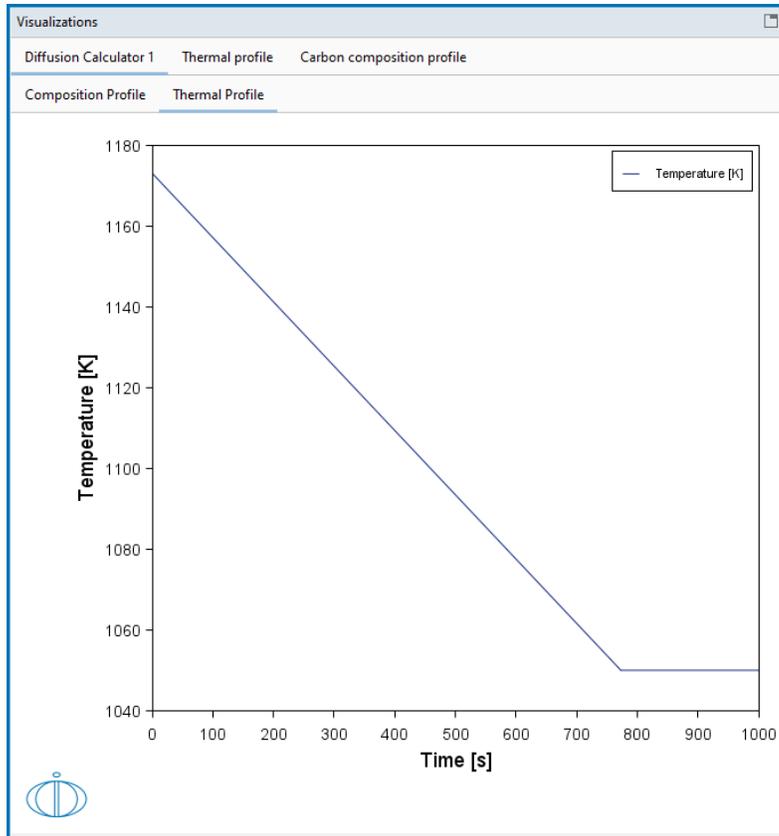
### *Non-isothermal Calculations*

1. On the Diffusion Calculator **Configuration** window, first enter the required settings described in [Conditions Tab Settings](#) including the [Region Settings](#), [Boundary Conditions](#), and [Composition Profiles](#).
2. Under **Thermal Profile** click to choose **Non-isothermal** to enter or import data for a user-defined temperature profile.
3. Select a **Temperature unit: Kelvin, Celsius, or Fahrenheit**.
4. Enter a total **Simulation time** and choose a unit: **Seconds, Minutes, Hours, or Days**.

---

You can next either directly enter **Time** and **Temperature** coordinates into the table or import the information from a file. A minimum of two points is required.

5. To enter data directly into the table:
  - a. Click in each cell under the applicable columns, **Time** and **Temperature** then directly enter the values in the table.
  - b. Press <Enter> to add rows and use the scroll bars to navigate the table.
6. To import from a file:
  - a. Although it is a simple set of data, prepare the data file as in [Guidelines for Importing Data Files to Thermal Profiles](#).
  - b. When the data file is ready, to the right of the **Import from file** field, click the file  button.
  - c. Navigate to a data file such as an MS Excel spreadsheet (with **\*.xls** or **\*.xlsx** extensions) or a **\*.csv** file. You can also use other programs (e.g. Google Sheets) to enter data and export to a CSV format that can be read by the software. Click **Open** to import the data to the **Configuration** window table.
  - d. If there are issues with the above import, experiment with choosing the delimiter type that corresponds to the data in the file: **Comma (,)**, **Colon (:)**, **Equals sign (=)**, **Semicolon (;)**, or **Tab**. Click the **Reload** button . The data is imported to the **Configuration** window table.
7. For this calculation type this completes the settings you need to enter for the **Thermal Profile**, which is displayed in the **Visualizations** window. Make adjustments to the settings as needed and watch it dynamically change in the window.



This is from example *D\_04: Diffusion Fe-C Moving Boundary Austenite to Ferrite*.

### *Including Probe Data from an AM Calculator*



When you also have a license for the Additive Manufacturing (AM) Module you can include probe data from the AM Calculator in the set up of the thermal profile.

There is connectivity between the Diffusion Module (DICTRA) or the Precipitation Module (TC-PRISMA) and the Additive Manufacturing (AM) Module when you have full licenses available for all relevant Add-on Modules and databases.

You can add a Diffusion Calculator or Precipitation Calculator as a successor to the AM Calculator and then when probes are included with the AM Calculator this data can be further used with the thermal profile.

Details are in the AM Module documentation. See [AM Calculator: Working with Probe Data](#).

---

## Guidelines for Importing Data Files to Thermal Profiles

The **Thermal Profile** settings are done on the **Configuration** window for a **Diffusion Calculator**. The profile is then previewed in the **Visualizations** window.

You can import data from a file. There are specific guidelines needed to correctly prepare the data in this file to ensure it imports without errors.

### *Data File Requirements*

For a Thermal Profile you can either directly enter **Time** and **Temperature** coordinates into the table on the **Configuration** window, or import the data from a file. A minimum of two points is required.

Although there are only two columns to define, it is still important to ensure the file is set up correctly. In order for the data to be included in the calculation, the column and row definitions in the file must adhere to certain rules of entry.

### GENERAL RULES

- No abbreviations are allowed in the header.
- A header is not actually required in this file but in order to know which column is which, it is recommended to include it.
- All header text must be in English, i.e. no translations into other languages will be read correctly by the data file.



Only the columns described here are read / used in the calculation. It is important to enter the information exactly as described. It is recommended that the data file is cleaned up before importing / loading it and performing calculations.

	A	B
1	Time	Temperature
2	0	1173
3	773	1050
4	1000	1075
5	1500	2000

Figure 5: This is an example of data entered into an Excel spreadsheet (which is then converted to csv format). Columns A and B show the data entered to create the Thermal Profile.

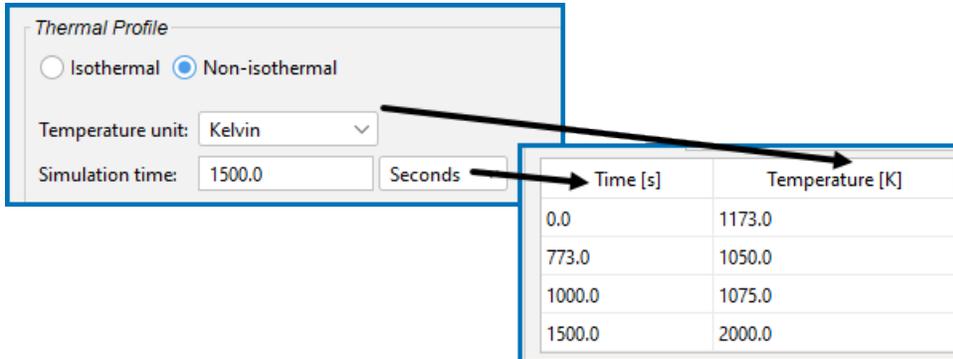


Figure 6: The units shown on the Configuration window for the Time and Temperature correspond to settings that can ONLY be selected on the Configuration window. The data file does not contain any information about the units.

Column Heading Requirements and Options	Row Cell Requirements and Options
<p><b>Time</b></p> <p>This is column A as in <a href="#">Figure 5</a></p> <p>Enter the text <code>Time</code>. No abbreviations allowed but it is not case sensitive. It is also not required but recommended to include so you know which column of data it applies to.</p>	<p>Valid format and entries for each row in this column:</p> <ul style="list-style-type: none"> <li>Any numerical input for the time that corresponds to the unit selected on the Configuration window e.g. 0.0, 773, 1000, 1500, are times in seconds, which is the unit chosen in the example shown in <a href="#">Figure 6</a>.</li> </ul>
<p><b>Temperature</b></p> <p>This is column B as in <a href="#">Figure 5</a></p> <p>Enter the text <code>Temperature</code>. No abbreviations allowed but it is not case sensitive. It is also not required but recommended to include so you know which column of data it applies to.</p>	<p>Valid format and entries for each row in these columns:</p> <ul style="list-style-type: none"> <li>Any numerical input for the temperature that corresponds to the unit selected on the Configuration window e.g. 1173, 1050, 1075, are temperatures in Kelvin, which is the unit chosen in the example shown in <a href="#">Figure 6</a></li> </ul>

## Options Tab Settings

The following settings are available on the **Options** tab for a [Diffusion Calculator](#).



You can change these settings locally for a specific Calculator or globally for some defaults in the **Options window** (a different location than the tab).

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

then click the **Graphical Mode** tab and the **Diffusion** node in the tree.



[Global Settings: Diffusion Calculator.](#)



The Diffusion Module (DICTRA) in Console Mode has commands to perform some of the settings also available on the Diffusion Calculator. Where possible the name of the command is indicated and includes a link to the topic.

## Simulation Conditions

### Default Solver

Choose the *Default solver*: **Automatic** or **Homogenization**. Select **Homogenization** to define the *Homogenization Model Specific* sections.



[Classic and Homogenization Model Settings](#)



The homogenization solver (called the homogenization model in Console Mode) is useful when you want to simulate long-range diffusion through a multiphase mixture, under the assumption that local equilibrium holds at each node point.

### Time Integration Method

**Automatic** , **Trapezoidal**, or **Euler backwards**.

The default is **Automatic** where the default degree of implicitity corresponds to trapezoidal (0.5) for classic and Euler Backward (1.0) for the homogenization model. If large fluctuations occur in the profiles it may be necessary to use the more stable but less accurate *Euler backwards*.

## Save Results on File

**Yes**, **No**, or **Every nth**.. Select **Every nth** to enter a value in the field.

This setting determines if the results are to be saved on file during the course of the simulation. Select:

- **Yes** to always save on file.
- **No** to never save on file.
- **Every nth**. to save every  $n^{\text{th}}$  time to file. Then enter an integer value ranging from 0 to 99.

## Use Forced Starting Values in Equilibrium Calculations

The default, **Automatic**, allows the software program to automatically use forced starting values to perform the equilibrium calculations. When a phase with an ordering contribution is entered into the simulation, for example the gamma-prime/FCC\_I12 phase in Ni-base superalloys, forced starting values are automatically turned on. Or choose **Yes** or **No**.

## Default Driving Force for Phases Allowed to Form at Interfaces

Enter a value in the field. The required driving force (evaluated as  $DGM_{(ph)}$ ) to be used for determining whether an inactive phase is stable. DGM is the variable for the driving force that is already divided by RT, and it is dimensionless. The default is  $1E-5$ .



[Suffixes](#)

## Timestep Control

### Max Relative Error

Enter a value in the field. The default is 0.05.

### Max Absolute Error

Enter a value in the field. The default is  $1.0E-5$ .

## Timestep

Enter values in the fields.

 This value is always in seconds.

- **Initial:** What is used as the initial time step. The default is  $1.0E-7$ .
- **Smallest allowed:** The smallest time step allowed during the simulation. This is required when using the automatic procedure to determine the time step. The default is  $1.0E-7$ .
- **Max % of simulation time:** The default is 10.

## Factor Specifying the Maximum Increase in the Time Step Taken from One Time Step to Another

For example, if 2 is entered the maximum time step is twice as long as the previous time step taken.

Enter a value in the field. The default is 2.

## The Timestep is to be Controlled by the Phase Interface Displacement During the Simulation

The default is **Automatic**. When a liquid phase is entered in a simulation this option is automatically turned on. Optionally choose **Yes** or **No** to set whether the time step is to be controlled by the phase interface displacement during the simulation.

## Classic and Homogenization Model Settings

### *Classic Model Specific*

### Activity or Potential

Use the [activity or potential] of a component in order to find the correct tieline at the phase interface.

Determines whether the program should use the potential or the activity of a component in order to find the correct tie-line at the phase interface. The potential or the activity is varied by the program and is set in order to reduce the degrees of freedom at the local equilibrium.

## Flux Balance Equation Accuracy

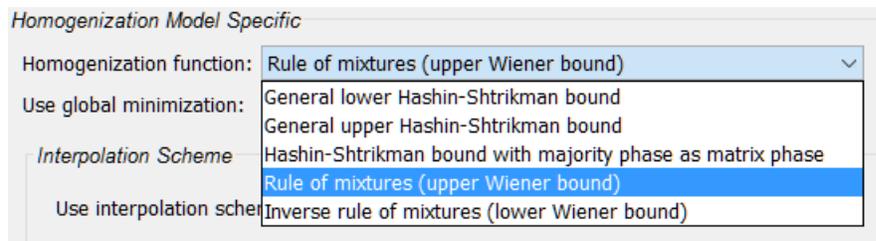
Required accuracy during the solution of the flux balance equations. The default is  $1.0E-16$ .

## Homogenization Model Specific

This section is available when **Homogenization** is selected (or automatically selected) as the **Default solver**. These settings are available on the **Options** tab.

## Homogenization Function

Several functions are available. The default is **Rule of mixtures (upper Wiener bound)**. See [About the Homogenization Functions](#).



## Use Global Minimization

Select the checkbox for global minimization used in equilibrium calculations. In general, using global minimization significantly increases the simulation time, but there is also a significantly reduced risk for non-converged equilibrium calculations.

## Interpolation Scheme

When the homogenization model is used, simulations are run using an implicit finite volume method to increase numerical stability. Because of this, and other factors, homogenization model simulations generally run slower than other comparable simulations. You can reduce simulation time with the **Use Interpolation scheme** checkbox. The checkbox is selected by default.

## Discretization

[**Logarithmic** or **Linear**] discretization with [*number of*] steps in each dimension.

- For the linear discretization the scheme is not used at node points where the content of one or more solutes fall below a certain critical value. For such cases, where composition span many orders of magnitude, the logarithmic discretization can be tried.
- Enter the number of steps in composition space. In the limit where an infinite number of steps are used, exactly the same solution is obtained as without the interpolation scheme. However, excellent results can be obtained with a reasonable discretization. The default is 10000 steps.

## Memory to Use

This is the fraction of free physical memory to be used by the interpolation scheme.

Select **Megabyte** or **Fraction of free** and then enter a value in the field.

## About the Homogenization Functions

The geometrical interpretation of the Hashin-Shtrikman bounds are concentric spherical shells of each phase. For the general lower Hashin-Shtrikman bound the outermost shell consists of the phase with the most sluggish kinetics and vice versa for the general upper bound. The geometrical interpretation of the Hashin-Shtrikman bounds suggest further varieties of the bounds, viz. *Hashin-Shtrikman bound with prescribed matrix phase* and *Hashin-Shtrikman bound with majority phase as matrix phase*, where the outermost shell consist of a prescribed phase or the phase with highest local volume fraction, respectively.

The geometrical interpretation of the Wiener bounds are continuous layers of each phase either parallel with (upper bound) or orthogonal to (lower bound) the direction of diffusion.



The labyrinth factor functions (described below) are available in Console Mode and when using TC-Python.

The labyrinth factor functions implies that all diffusion takes place in a single continuous matrix phase. The impeding effect on diffusion by phases dispersed in the matrix phase is taken into account by multiplying the flux with either the volume fraction (*Labyrinth factor  $f$  with prescribed matrix phase*), or the volume fraction squared (*Labyrinth factor  $f^{**2}$  with prescribed matrix phase*), of the matrix phase.



The varieties with excluded phases (described below) are available in Console Mode and when using TC-Python.

The varieties with excluded phases are useful in several respects. First, if a phase is modeled as having zero solubility for a component, the mobility of that component in that phase is undefined, which causes a (non-terminal) error. Setting a phase as excluded causes the mobility of all components in that phase to be set to zero. Second, often there are some major matrix solid solution phases and some minor precipitate phases. If the mobilities in the minor precipitate phases are zero the lower Hashin-Shtrikman bound is useless as it produces a kinetic coefficient of zero. However, using *General lower Hashin-Shtrikman bound with excluded phase(s)* the excluded phases are not considered when evaluating what phase has the most sluggish kinetics.

## Pause, Resume, and Cancel Diffusion Calculations

Diffusion calculations are often complex simulations that take some time to complete. Sometimes you may want or need to pause or resume a calculation, or make adjustments to your compositions and start again.

### **Graphical Mode**

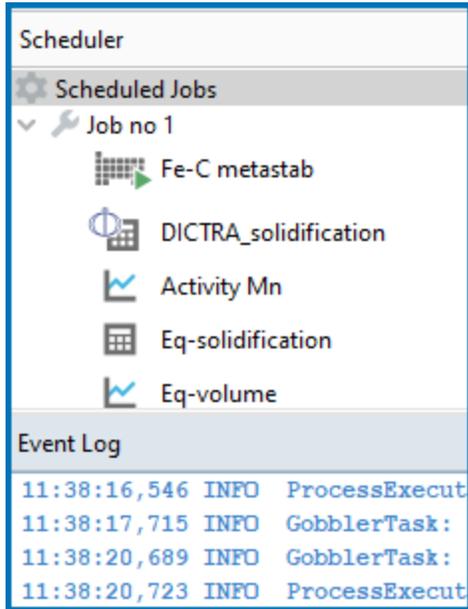
In Graphical Mode, you pause and resume when you are on the Diffusion Calculator **Configuration** window. When pausing a Diffusion Calculator calculation, you can make changes to the **Simulation time**, **Left Boundary Condition**, and **Right Boundary Condition**. This is useful, for example, when simulating carburization, where it is common to first have a step with high carbon activity in the furnace for the actual carburization, and then lower it and allow the carbon in the specimen to diffuse without further increased carbon content.



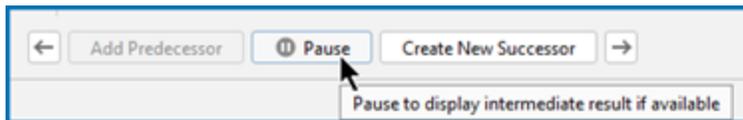
This is also available for the Precipitation Calculator (but not any other calculators). See [Pause, Resume, and Cancel Precipitation Calculations](#).

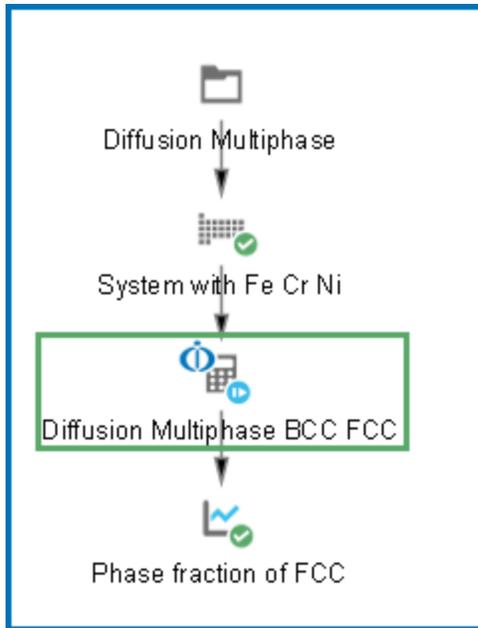
### HOW TO PAUSE AND RESUME A JOB

1. Run the job (i.e. click **Perform Tree**). In the **Scheduler** you can see the job listed and in the **Event Log** you can follow the progress of the calculation.

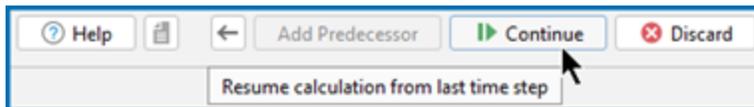


2. In the **Project** window, click the **Diffusion Calculator** node.
3. In the **Configuration** window at the bottom, click **Pause**. Observe that the **Diffusion Calculator** in the **Project** window has a pause indicator. If there are intermediate results available these are listed in the **Event Log**.



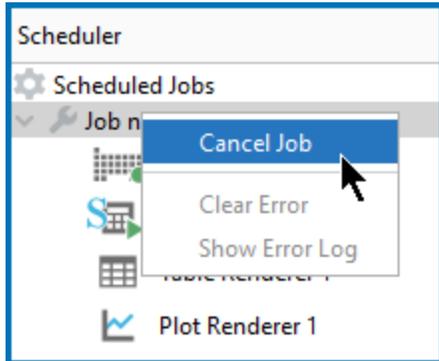


4. As required, make changes to the settings: *Left Boundary Condition*, *Right Boundary Condition*, *Temperature*, and/or *Simulation time*. The rest of the **Configuration** window settings are grayed out and no other editing can be done while paused.
5. When ready, click **▶ Continue** to resume your calculations from the last time step or click **✖ Discard** to discard the calculation (then click **Yes** or **No** on the window that opens).



## HOW TO CANCEL A JOB

- In the **Scheduler** window, right-click the job you want to cancel and select **Cancel Job**.



## Console Mode

In response to a command or a lengthy calculation, Thermo-Calc may display a lot of text in the Console window. Use these keyboard shortcuts to pause, resume, or stop a process.

- To pause the text on printing press <Ctrl+S>.
- To resume the printing on screen, press <Ctrl+Q>.
- When Thermo-Calc is performing a stepping or mapping operation, the results are continuously printed in the Console window. To stop the calculation of the current region of the mapping and stop the output, press <Ctrl-A> (Windows) or <Ctrl-C> (Linux).

## Diffusion Calculator Plot Renderer

The following is information about the settings available in both Simplified and Advanced Mode for a [Plot Renderer](#) when it is a successor to a **Diffusion Calculator**.

 [Conditions Tab Settings](#) and [Configuration Settings](#)

### Simple Mode

When you open the Diffusion Calculator Plot Renderer, it defaults to **Simple Mode**. Use the **Advanced Mode** to have more flexibility.

Use **Simple Mode** to define the most common axis variables. The software automatically sets the correct **Plot condition** and **Independent variable**.



When you select **Advanced Mode** as the *Plot type*, this expands the variable options available from both the *X-axis* and *Y-axis* lists. Otherwise the axis variables default to the most logical combination.

### X-axis Variable

Select the variable for the X-axis: **Distance** or **Time**.

- For **Distance** also choose a unit (**m**, **µm**, or **nm**) and what region to include. The list contains the regions defined on the Diffusion Calculator or you can choose **All regions**.
- For **Time** choose a unit (**Seconds**, **Minutes**, **Hours**, or **Days**).

### Y-axis Variable

Select the variable to plot on the Y-axis. The available options are based on choices such as whether **Distance** or **Time** is chosen for the X-axis, whether isothermal or non-isothermal is selected, whether there is a homogenization solver, or if there is one or more regions.

### Distance Along the X-axis

If **Distance** is plotted along the X-axis, then by default the **Composition** is plotted along the Y-axis.

- From the center list, choose the component to plot, or select **All**.
- From the next list, choose the unit to be **Mass percent**, **Mole percent**, **Mass fraction**, **Mole fraction**, or **u-fraction**.

### Time Along the X-axis (One Region)

If **Time** is plotted along the X-axis, then choose the Y-axis variable to plot plus the additional settings:

- **Total phase fraction: By volume** or **By mass**. Then select a phase from the list.
- **Total fraction of element: By mole** or **By mass**. Then select a component from the list.
- **Total fraction of element in phase: By mole** or **By mass**. Then select a component

and a phase from the lists.

- For a non-isothermal thermal profile **Temperature** is available: **Kelvin**, **Celsius**, or **Fahrenheit**.

### Distance Along the X-axis (Homogenization)

If **Distance** is plotted along the X-axis and a homogenization solver is being used, then choose **Composition** or **Phase fraction** to plot on the Y-axis. Then select the applicable settings.

- Choose the component to plot, or select **All** and the unit **Mass percent**, **Mole percent**, **Mass fraction**, **Mole fraction**, or **u-fraction**.
- Choose **By mole** or **By mass** then select a component from the list.

### Time Along the X-axis (Two or More Regions)

If **Time** is plotted along the X-axis, then choose the Y-axis variable to plot plus the additional settings:

- **Total phase fraction: By volume** or **By mass**. Then select a phase from the list.
- **Total fraction of element: By mole** or **By mass**. Then select a component from the list.
- **Total fraction of element in phase: By mole** or **By mass**. Then select a component and a phase from the lists.
- **Composition at interface: Mass percent**, **Mole percent**, **Mass fraction**, or **Mole fraction**. Then choose a **Region** and whether it is **Upper** or **Lower** and a component.
- **Position of interface**: Choose a unit (**m**, **µm**, or **nm**) and the interface.
- **Velocity of interface**: Choose a unit (**m**, **µm**, or **nm**) and the interface.
- **Width of region**: Choose a unit (**m**, **µm**, or **nm**) and the interface.
- For a **Non-isothermal** profile, **Temperature** is available. Choose a unit **Kelvin**, **Celsius**, or **Fahrenheit**.

### Axis Type

Select the type of axis scale: **Linear**, **Logarithmic**, **Logarithmic 10**, or **Inverse**.

## Limits

Specify the range along the axis to show in the plot. In the fields, enter the minimum and maximum values of the axis variable. You can also determine the **step** size between the tick marks along each axis.

Select the **Automatic scaling** checkbox to allow the program to set the limits.

## Time



An alternative **Time** setting is available in **Advanced Mode**.

When the X-axis variable is **Distance**, choose a **Time** unit (**Seconds**, **Minutes**, **Hours**, or **Days**). Enter a numerical value.

## Add and Remove Axes

Use the **Add an axis**  and **Remove this axis**  buttons to add additional X- and/or Y-axes to a plot or to merge plots (to overlay one plot with another in the same diagram).

### *Advanced Mode*

Use **Advanced Mode** to select any combination of axis variables, and then access further settings such as the **Plot condition**, **Region**, and **Independent variable**. Examples of advanced plot options are activity of a component at an interface, diffusivities, and diffusion paths. Also user-defined functions can be used as plot variables where Console Mode syntax is used to define functions. In some cases, sensible default values are selected, but they can be changed according to your needs.

### ***Plot Axis Settings in Advanced Mode***

- If any plot axis variable of the type **Integral** is selected, then the plot condition is set to **Integral** and the Independent variable is set to **Time**.
- If a plot axis variable is set to **Time** then the plot condition is set to **Time** and the Independent variable is also set to **Time**.
- If a plot axis variable is set to **Distance** then the plot condition is set to **Distance** or **Interface** and the Independent variable is set to **Distance**.

<i>Plot condition</i>	<i>Independent Variable settings based on the Plot condition</i>
<b>Time:</b> Choose a unit ( <b>Seconds, Minutes, Hours, or Days</b> ). Enter a numerical value.	The default is <b>Distance</b> : Choose a <b>Global</b> or <b>Local</b> distance. A <b>Global</b> distance is counted from the rightmost interface of the system. A <b>Local</b> distance is counted from the lower interface of the Region selected.  You can also choose <b>Time</b> , which requires no further settings.
<b>Distance:</b> Choose a specific <b>Region</b> from the list or use the default <b>All regions</b> .	The default is <b>Time</b> and there are no further settings.  You can also choose a <b>Global</b> or <b>LocalDistance</b> . A <b>Global</b> distance is counted from the rightmost interface of the system. A <b>Local</b> distance is counted from the lower interface of the Region selected.
<b>Interface:</b> From the lists choose a <b>Region</b> and whether it is the <b>Upper</b> or <b>Lower</b> region.	The default is <b>Distance</b> : Choose a <b>Global</b> or <b>Local</b> distance. A <b>Global</b> distance is counted from the rightmost interface of the system. A <b>Local</b> distance is counted from the lower interface of the Region selected.  You can also choose <b>Time</b> , which requires no further settings.
<b>Integral</b>	The default is <b>Time</b> and there are no further settings.  You can also choose a <b>Global</b> or <b>LocalDistance</b> . A <b>Global</b> distance is counted from the rightmost interface of the system. A <b>Local</b> distance is counted from the lower interface of the Region selected.

## Precipitation Calculator

---



The Precipitation Calculator is available with three elements if you do not have the additional Precipitation Module (TC-PRISMA) license. With the Add-on Module you can use all available elements. See [Available Options](#) for details about this and other options available with additional licenses.

A **Precipitation Calculator** allows you to set the conditions for, and perform, a precipitation calculation. You can add the calculator to the System Definer directly (right-click and select it from the **Create New Successor** menu)



There is also a **Precipitation** template available under **Non-Equilibrium** that you can use to quickly set up a simulation. See [Getting Started Links and Templates](#).

Once you have added a Precipitation Calculator, the **Configuration** window has **Conditions** and **Options** settings tabs where there are many available conditions to set and then continue the settings with the specialized **Plot Renderer**.

### Conditions

Set the conditions for your calculation that define the *Matrix phase* and *Precipitate phase*. Choose the *Calculation Type*.



[Matrix Phase Settings](#)



[Precipitate Phase Settings](#)



[Calculation Type Settings](#)

### Options

Modify *Numerical Parameters* that determine how the conditions are calculated.



[Numerical Parameters on the Options Tab](#)

## Plot Renderer

There are also unique settings available for this calculator once you add a [Precipitation Calculator Plot Renderer](#).



Working with these simulations is an advanced topic. It is recommended you review the documentation and videos available or search for a specific topic.



[Precipitation Module \(TC-PRISMA\) Examples Collection](#)

## Available Options

The Precipitation Module, previously referred to as TC-PRISMA, is an Add-on Module to the core Thermo-Calc software.



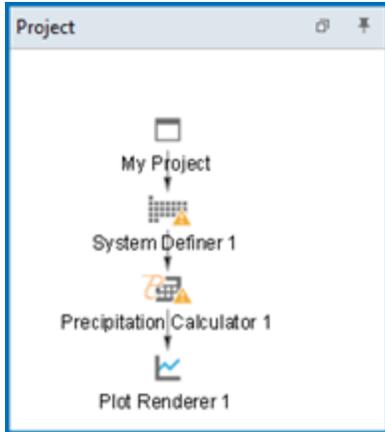
A separate license is required to perform calculations for more than three elements. Without it you are able to use the module in *Demo Mode*.

## Precipitation Template

A **Precipitation** template under **Non-Equilibrium Calculations** is available to all Thermo-Calc users when in Graphical Mode. If you are in DEMO mode then this is indicated by the addition of **DEMO** text.

### USING THE TEMPLATE

After opening Thermo-Calc in Graphical Mode, in the templates section under **Non-Equilibrium**, click the **Precipitation** button to add a *System Definer*, *Precipitation Calculator*, and *Plot Renderer* to the **Project** tree.



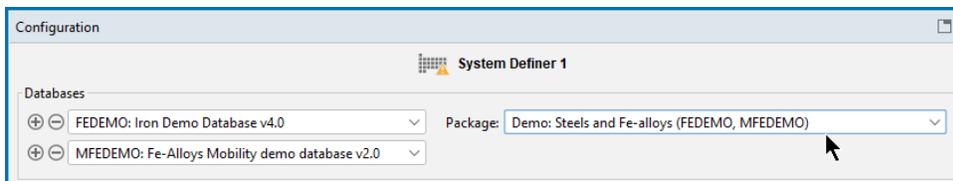
[? Creating a Project from a Template](#)

## ***Precipitation Calculator***

A Precipitation Calculator allows you to set the conditions for, and perform, a precipitation calculation.

## ***Demo Database Packages for the Examples Collection***

Both a thermodynamic and mobility database are required to run the Precipitation Module simulation. If you have a Precipitation Module (TC-PRISMA) license you can run all the examples as the demonstration database packages are included with your installation. Select the database packages from the **System Definer** → **Configuration** window to run a simulation.



## ***Demonstration (Demo) Mode***

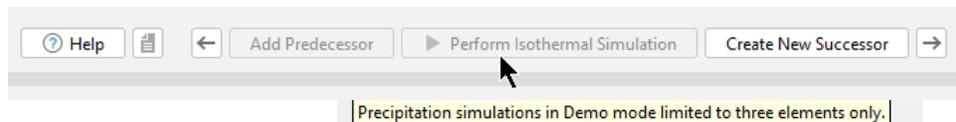
The Precipitation Module, and some examples, are available to all Thermo-Calc users but only for simulations of alloys with up to three elements. If you do not have a license for the Precipitation Module then you are in *Demonstration Mode* when using the Precipitation Calculator or Precipitation Simulation template.

A **Precipitation** template under **Non-Equilibrium Calculations** is available to all Thermo-Calc users when in Graphical Mode. If you are in DEMO mode then this is indicated by the addition of **DEMO** text.

## PRECIPITATION CALCULATOR

If you are experimenting with the Precipitation Calculator in Demo Mode, you may have access to a variety of databases based on your license. However, you can only define up to three elements for a demo simulation.

If you define more than three elements on the System Definer, when you go to the Precipitation Calculator, the **Perform** button is unavailable and the tooltip explains why. In this case one of the chosen elements needs to be removed and then the Perform button is made available.



Even if you have more than three elements, the Plot Renderer or Table Renderer **Perform** button is still available. However, if you click the button and try to run the simulation the Event Log displays an error message.

## Additive Manufacturing (AM) Module

There is connectivity between the Diffusion Module (DICTRA) or the Precipitation Module (TC-PRISMA) and the Additive Manufacturing (AM) Module when you have full licenses available for all relevant Add-on Modules and databases.

You can add a Diffusion Calculator or Precipitation Calculator as a successor to the AM Calculator and then when probes are included with the AM Calculator this data can be further used with the thermal profile.

Details are in the AM Module documentation. See [AM Calculator: Working with Probe Data](#).

## Define the Precipitation Calculator

1. Add a **Precipitation Calculator** node to the System Definer. If you used the **Precipitation** template (on the  **My Projects Configuration** window under **Non-Equilibrium**), click the node to display the **Configuration** settings window.

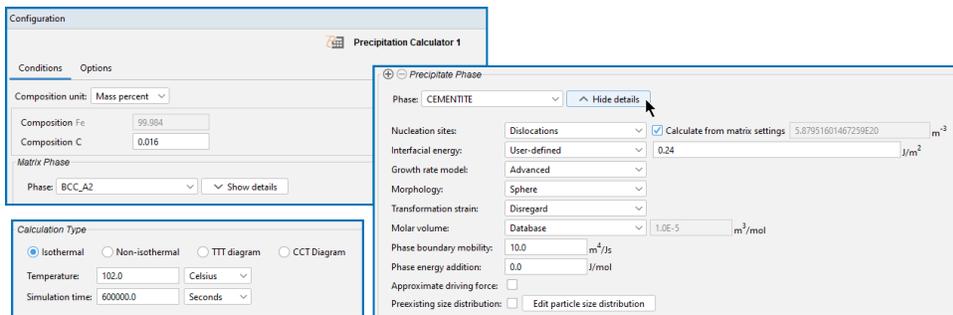
- In the Precipitation Calculator **Configuration** window, enter the settings on the **Conditions** and **Options** tabs. These are described separately:
  - [Matrix Phase](#)
  - [Precipitate Phase](#)
  - [Calculation Type](#) and [Precipitation Calculator Thermal Profile](#)
  - [Options Tab Advanced Settings](#)
- Once you have finished defining the Precipitation Calculator, you also choose settings on the **Plot Renderer**. Some additional settings are specific to the [Precipitation Calculator Plot Renderer](#).

### Matrix Phase Settings

Below are details about the settings available for the *Composition* and *Matrix Phase*, which is selected from the **Conditions** tab on the calculator **Configuration** window.



There are additional **Conditions** tab settings described for the [Precipitate Phase](#) and [Calculation Type](#).



This is the three distinct settings areas for the Configuration window for example *P\_04\_Precipitation\_Fe-C\_Cemetite.tcu*. In this case, Show Details is expanded to show the Precipitate Phase settings.



You can change these settings locally for a specific Calculator or globally for some defaults in the **Options window** (a different location than the tab).

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  on the keyboard.

then click the **Graphical Mode** tab and the **Precipitation** node in the tree.

## Show or Hide Details

Click **Show Details** to view some of the additional settings listed below. You can also set the default to display in **Phase view mode** on the **Options** window then click the **Graphical Mode** tab and the **Precipitation** node in the tree. Click **Hide Details** to hide the information.

## COMPOSITION SETTINGS

### Composition Unit

Select the **Composition unit**: **Mass percent**, **Mole percent**, **Mass fraction**, or **Mole fraction**.

## MATRIX PHASE SETTINGS



[Selecting the Disordered Phase as a Matrix Phase](#)

### Phase



Only phases with kinetic data can be selected as the matrix phase. If the list is empty, go to the System Definer to confirm that both thermodynamic and kinetic databases are selected and defined.

Choose a **Phase** from the list. The list is based on the settings for the System Definer. When setting up a system, choose a matrix phase with kinetic data available in the database.

### Elastic Properties

Choose **Disregard** to ignore the elastic properties.

Default elastic constants for **Isotropic** or **Cubic** are based on the major element of the alloy system. The elastic properties can affect nucleation rate, nuclei size, and particle shape.

- For **Isotropic**, enter values for **Shear modulus** (in GPa) and **Poisson's ratio** as required.
- For **Cubic**, enter values for **c11**, **c12**, and **c44** as required.  $c_{11}$ ,  $c_{12}$ ,  $c_{44}$  are the elastic constants.

## Molar Volume

Use the **Database** value (if the molar volume for the phase is defined in the thermodynamic database) or select **User-defined** to enter another value in m<sup>3</sup>/mol.



If you select the **Grain growth** checkbox, see the separate [Grain Growth Settings](#) section for details.

## Grain Size



If the **Grain growth** checkbox is NOT selected then this version of the **Grain size** section is available in order for the average grain size to be entered.

Grain size is the "diameter" of a grain. The **Grain size** value changes the available nucleation sites when **Grain boundaries**, **Grain edges**, or **Grain corners** is selected along with **Calculate from matrix settings** in the *Precipitate Phase*. Enter a numerical value and choose a unit from the list. The default is 1.0E-4 m.

## Grain Aspect Ratio

For an elongated grain with a minor axis and a major axis, one may use the minor axis as grain size and the major/minor ratio as the grain aspect ratio to characterize the grain. The **Grain aspect ratio** value also changes the available nucleation sites when **Grain boundaries**, **Grain edges**, or **Grain corners** is selected along with **Calculate from matrix settings** in the *Precipitate Phase*. Enter a numerical value. The default is 1.0.



If the **Grain growth** checkbox is selected, the aspect ratio is fixed at 1.0 and can not be changed. See [Grain Growth Settings](#).

---

## Dislocation Density

The **Dislocation density** value changes the available nucleation sites when **Dislocations** is selected along with **Calculate from matrix settings** in the *Precipitate Phase*. Enter a numerical value. The default is 5.0E12 m<sup>-2</sup>.

## Mobility Adjustment

The **Mobility adjustment** parameters modify the atomic mobility data from a database.

Choose **Same for all elements**, which applies the adjustment to all elements, or **Per element** to adjust to each individual element. Then for the following settings, enter one set of values for **Same for all elements**, or individually for **Per element**.

- **Prefactor** (unitless) is a parameter that multiplies to the mobility data from a database. This value scales the mobility by a constant amount. This can be useful, for example, when the material has a higher than normal vacancy concentration at the start of the precipitation simulation. (e.g. from a prior solutionizing and quenching treatment).
- The **Activation energy** (J/mol) is a value that adds to the activation energy of mobility data from a database. This value scales the mobility by a temperature-dependent amount. Similar usage as the **Mobility adjustment >Prefactor** setting.

## Grain Growth Settings

---

### Grain Growth

Select the **Grain growth** checkbox to use this to calculate the temporal evolution of grain size distribution (GSD). The grains are assumed of spherical morphology when modeling the growth rates. Nucleation is not considered, thus an initial GSD is necessary to start the simulation.

After selecting the checkbox, there are additional settings made available as described in this section.



This checkbox is not available when a **TTT diagram** or **CCT diagram** *Calculation Type* is selected.

### Grain Size >> Grain Size Distribution

When the **Grain growth** checkbox is selected, then this version of the **Grain size** settings section is available. This is where you access the grain *Size distribution* settings.

Click **Edit grain size distribution** to open the *Size distribution* settings window. In this window, you can edit the parameters and generate a graph comparing the radius and number density for initial grains.

Select a **Length unit**: **Meter**, **Micrometer**, **Nanometer**, or **Ångström**.

Select a **Distribution**: **Normal**, **Log normal**, or **From file**.

- If **Normal** or **Log normal** is selected, enter values for **Mean radius** and **Std** (standard deviation). The default Std is different for each choice.
- If **Hillert** is selected, enter a value for the **Mean radius**.
- If **From file** is selected, click **Import** and navigate to the file containing the required information and click **Open**. This file can be in .xls, .xlsx, .csv or .txt formats. The file should consist of two columns with values where the first column contains **radius** data and the second contains **number density** data.



A similar setup can also be found in the [Particle Size Distribution \(PSD\)](#).

Once the size distribution is defined, click **Generate** to view or update the graph comparing the radius and number density for initial grains.

---

## Grain Boundary Energy

This setting is available when the **Grain growth** checkbox is selected.

Enter a value for the **Grain boundary energy**. The default is 0.5 J/m<sup>2</sup>.

## Grain Boundary Mobility

This setting is available when the **Grain growth** checkbox is selected.

The **Prefactor** (m<sup>4</sup>/Js) is a parameter that represents the magnitude of the grain boundary motion.

The **Activation energy** (J/mol) is a parameter that describes the temperature dependence of the grain boundary mobility.

## Zener Pinning

This setting is available when the **Grain growth** checkbox is selected.

Select the **Zener pinning** checkbox to consider the effect of precipitates on inhibiting normal grain growth. When this checkbox is selected, there are additional settings available in the **Precipitate Phase** section.



[P\\_14: Grain Growth and the Zener Pinning Effect](#)

## Selecting the Disordered Phase as a Matrix Phase

The following information is about using disordered FCC as a matrix phase with the following thermodynamic and mobility database packages:

- TCCU and MOBCU (Cu-based alloys)
- TCNI and MOBNI (Ni-based alloys)

In the TCNI/MOBNI and TCCU/MOBCU packages, the well-known order/disorder two-sublattice model is used to describe the Gibbs energy of both FCC\_A1 and FCC\_L12. With this treatment, FCC\_L12 is becoming FCC\_A1 if the site fractions of each element on both sublattices are identical, which means that FCC\_A1 is only a special case of FCC\_L12. Therefore, FCC\_A1 is not shown in the phase list on the *Phases and Phase Constitution* tab on the System Definer activity and in subsequent equilibrium calculation results. Instead it is shown only as FCC\_L12. The real ordered FCC\_L12 is shown as FCC\_L12#2.

In precipitation simulations, the matrix phase is quite often the disordered FCC phase. You can directly select FCC\_L12 as the matrix phase and run a simulation. However, the speed is not optimal due to the sophisticated model used for both Gibbs energy and atomic mobilities. A better and more convenient way is to deselect FCC\_L12 and FCC\_L12#2 from the phase list on the *Phases and Phase Constitution* tab on the **System Definer** if the ordered phase is irrelevant in the alloy under investigation, such as in most Cu alloys. Once these are unchecked (i.e. not selected), the FCC\_A1 phase is available and can later be selected as the matrix phase.

For Ni-based superalloys using the TCNI/MOBNI package, the ordered FCC\_L12#2 (gamma prime) has to be included as the precipitate phase in most of calculations. In this case, you can select DIS\_FCC\_A1 from the phase list on the *Phases and Phase Constitution* tab and then select it as the matrix phase in the **Precipitation Calculator**.

### *Precipitate Phase Settings*

Below are details about the settings for the *Precipitate Phase*, which is selected from the **Conditions** tab on the Precipitation Calculator **Configuration** window.



There are additional **Conditions** tab settings described for the [Matrix Phase](#) and [Calculation Type](#).

To open the **Options** window:

- Windows: Select **Tools** → ☰ **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **Graphical Mode** tab and the **Precipitation** node in the tree.

### **Show or Hide Details**

Click **Show Details** to view some of the additional settings listed below. You can also set the default to display in **Phase view mode** on the **Options** window then click the **Graphical Mode** tab and the **Precipitation** node in the tree. Click **Hide Details** to hide the information.

## Phase



The phases available to choose have both thermodynamic and kinetic data. If the list is empty, go to the System Definer to confirm that both types of databases are selected and defined.

Choose a **Phase** from the list. The list is based on the System Definer settings.

## Nucleation Sites

The number of different nucleation sites is dependent on the shape and size of grains in the matrix. Grain size is the "diameter" of a grain. Choose one of the following from the list.

- **Bulk, Grain boundaries, Grain edges, Grain corners, or Dislocations.**
- For **Grain boundaries, Grain edges, and Grain corners**, enter the **Wetting angle (0-90)** in addition to the matrix settings. **Wetting angle** defines the deviation from spherical shape (or dihedral angle).

Click to select the **Calculate from matrix settings** checkbox to calculate the initial number density of sites from the matrix grain size or dislocation density.

To enter a specific value for the number of **Nucleation sites**, click to clear the checkbox.

## Interfacial Energy

Choose **Calculated** to use the estimated value. To adjust the estimate, enter a different **prefactor** or choose **User-defined** to enter a value in  $\text{J/m}^2$ . For the User-defined option, you can also enter it as a function of radius ( $r$ ) and temperature ( $T$ ).

## Growth Rate Model

Select **Simplified, General, Advanced, Para-eq, NPLE, or PE Automatic** (where Para-eq and PE are both abbreviations for *paraequilibrium*).



All models treat a particle (precipitate) of stoichiometric composition or with negligible atomic diffusivity. Except for the **PE Automatic** model, local equilibrium or paraequilibrium at the precipitate-matrix interface is assumed. The **PE Automatic** model enables smooth transition from paraequilibrium to local equilibrium.



In the documentation and on the GUI, the abbreviations *Para-eq* and *PE* are both used for paraequilibrium. There is also the use of *OE* as an abbreviation for *ortho-equilibrium* and *LE* for *local equilibrium*.

## Morphology

Choose the particle shape: **Sphere** (default), **Cuboid**, **Plate**, or **Needle**. Options are based on the **Elastic properties** selected for the **Matrix** phase:

- For a **Cubic** elastic property, **Sphere**, **Plate**, **Needle**, and **Cuboid** are available.
- For an **Isotropic** elastic property, **Sphere**, **Plate** and **Needle** are available.

For **Plate** or **Needle**, select the **Calculate aspect ratio from elastic energy** checkbox or enter a numerical value in the **Aspect ratio** field to provide a constant aspect ratio.

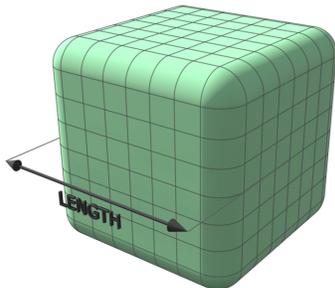


Figure 7: The cuboid shape is described by a supersphere. Cuboids have six faces, which form a convex polyhedron.

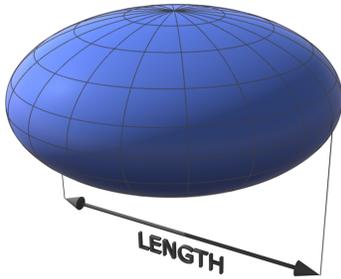


Figure 8: The plate is described as oblate spheroid. Oblate spheroids have rotational symmetry around an axis from pole to pole.

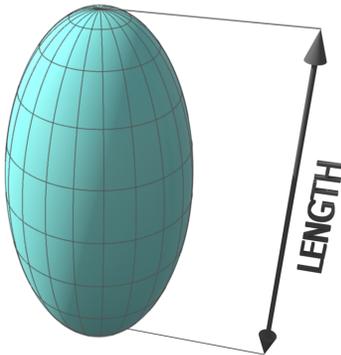


Figure 9: The needle shape is described as prolate spheroid. A prolate spheroid is a surface of revolution obtained by rotating an ellipse about its major axis.

## Transformation Strain

Choose **Disregard** to ignore the transformation strain. When **Isotropic** or **Cubic** is chosen in **Elastic properties** in Matrix Phase, you can also choose **Calculated from molar volume** to obtain a purely dilatational strain. If **Plate** or **Needle** is selected as the **Morphology**, you can alternatively choose **User-defined** and enter the properties for  $\epsilon_{11}$ ,  $\epsilon_{12}$ ,  $\epsilon_{13}$ ,  $\epsilon_{22}$ ,  $\epsilon_{23}$ , and  $\epsilon_{33}$ .

## Molar Volume

Use the **Database** value (if the phase molar volume is defined in the thermodynamic database) or select **User-defined** to enter another value.

---

## Zener Pinning Parameters

These parameters are available when the **Zener pinning** checkbox is selected under the **Matrix Phase** settings. These settings are to consider the effect of precipitates on inhibiting normal grain growth.

- **Cutoff size:** Precipitates with radius smaller than this value are neglected in pinning force calculation. Default value :  $7 \times 10^{-7}$  m.
- **Kinetic prefactor:** Dimensionless kinetic coefficient in Zener equation. Default value: 0.5.
- **Exponent:** Exponent of precipitate volume fraction in Zener equation. Default value: 0.93.



[P\\_14: Grain Growth and the Zener Pinning Effect](#)

## Phase Boundary Mobility

A parameter that accounts for interface-controlled growth. Only effective if a very small, positive value is used. Use with caution due to a tentative treatment.

## Trans-Interface Mobility Adjustment

This setting is available when **PE Automatic Growth rate model** is selected.

The **Trans-Interface mobility adjustment** parameters modify the atomic mobility data from a database.

Choose **Same for all elements**, which applies the adjustment to all elements, or **Per element** to adjust to each individual element.



The transition kinetics from Para-Equilibrium (PE) to Ortho-Equilibrium (OE) is controlled by the ratio of atomic mobility of substitutional elements to that of interstitial elements. This implies that the option of **Same for all elements**—while useful to the case that the atomic mobility data for all elements are needed to be scaled with the same factor to match experimental trans-interface mobility data—does not change the PE-OE transition kinetics. Therefore, the change of transition kinetics separately requires adjustment of individual elements, which is when the **Per element** setting is useful.

Then for the following settings, enter one set of values for **Same for all elements**, or individually for **Per element**:

- **Prefactor** (unitless) is a parameter that multiplies to the mobility data from a database. This value scales the mobility by a constant amount. This results in the trans-interface mobility that controls the kinetics of Para-Equilibrium (PE) to Ortho-Equilibrium (OE) transition.
- The **Activation energy** (J/mol) is a value that adds to the activation energy of mobility data from a database. This value scales the mobility by a temperature-dependent amount. Similar usage as the **Trans-Interface mobility adjustment >Prefactor** setting.

### Phase Energy Addition

An energy value that adds to the Gibbs free energy of the precipitate phase from a database.

### Approximate Driving Force

Select the checkbox to include this if simulations with several compositions sets of the same phase create problems.

### Preexisting Size Distribution

Select the checkbox to include this. Click **Edit particle size distribution** to open the **Preexisting Particle Size Distribution** window where you can edit the parameters and view a graph comparing the radius and number density for the selected component.



Press F1 and search for "Particle Size Distribution (PSD)" for detailed instructions.

## Precipitation Calculator Thermal Profile

The **Thermal Profile** is set up on the **Configuration** window for a **Precipitation Calculator**. The isothermal or non-isothermal profile is viewed in the **Visualizations** window. Choose **Isothermal** to do a simulation at constant temperature. Choose **Non-isothermal** to enter or import data for a user-defined temperature profile.



When connected to an AM Calculator (see [Including Probe Data from an AM Calculator](#)) the **Isothermal** and **Non-isothermal** options are not visible as it is not relevant to that view.

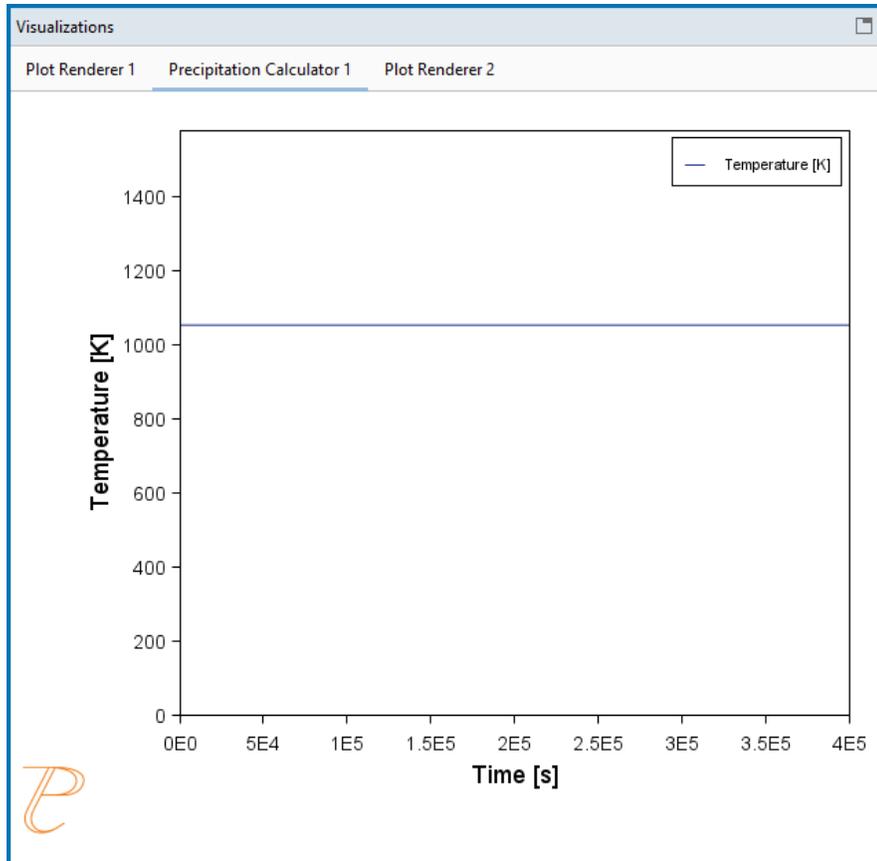
## ISOTHERMAL CALCULATIONS



Many of the Graphical Mode examples for this Add-on Module are isothermal calculations, e.g. *P\_02\_Precipitation\_Fe-C-Cr\_Cementite-M7C3-M23C6*.

1. On the Precipitation Calculator **Configuration** window, first enter the required *Matrix Phase* and *Precipitate Phase* settings.
2. Under **Calculation Type**, click to choose **Isothermal** to do a simulation at constant temperature.
3. Enter a **Temperature** and choose a unit (**Kelvin**, **Celsius**, or **Fahrenheit**).
4. Enter a **Simulation time** and choose a unit: **Seconds**, **Minutes**, **Hours**, or **Days**.

The profile is displayed on the **Precipitation Calculator** tab in the **Visualizations** window showing the constant temperature entered.



5. For this calculation type this completes the settings you need to enter for the Thermal Profile.

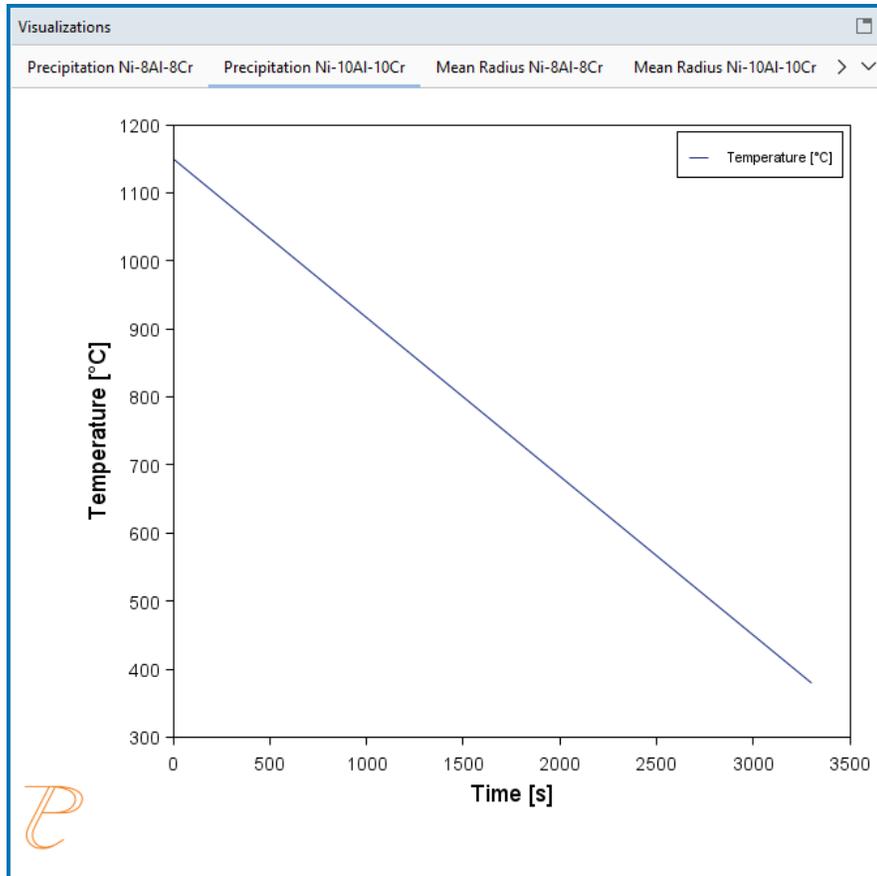
## NON-ISOTHERMAL CALCULATIONS

1. On the Precipitation Calculator **Configuration** window, first enter the required *Matrix Phase* and *Precipitate Phase* settings.
2. Under **Calculation Type**, click to choose **Non-isothermal** to enter or import data for a user-defined temperature profile.
3. Select a **Temperature unit: Kelvin, Celsius, or Fahrenheit**.
4. Enter a total **Simulation time** and choose a unit: **Seconds, Minutes, Hours, or Days**.

---

You can next either directly enter **Time** and **Temperature** coordinates into the table or import the information from a file. A minimum of two points is required.

5. To enter directly into the table:
  - a. Click in each cell under the applicable columns, **Time** and **Temperature** then directly enter the values in the table.
  - b. Press <Enter> to add rows and use the scroll bars to navigate the table.
  
6. To import a data file:
  - a. Although it is a simple set of data, prepare the data file as in [Guidelines for Importing Data Files to Thermal Profiles](#).
  - b. When the data file is ready, to the right of the **Import from file** field, click the file  button.
  - c. Navigate to a data file such as an MS Excel spreadsheet (with **\*.xls** or **\*.xlsx** extensions) or a **\*.csv** file. You can also use other programs (e.g. Google Sheets) to enter data and export to a CSV format that can be read by the software. Click **Open** to import the data to the **Configuration** window table.
  - d. If there are issues with the above import, experiment with choosing the delimiter type that corresponds to the data in the file: **Comma (,)**, **Colon (:)**, **Equals sign (=)**, **Semicolon (;)**, or **Tab**. Click the **Reload** button . The data is imported to the **Configuration** window table.
  
7. For this calculation type this completes the settings you need to enter for the profile, which is displayed on the **Precipitation Calculator** tab in the **Visualizations** window. Make adjustments to the settings as needed and watch it dynamically change in the window.



This is from the non-isothermal calculation example, *P\_06\_Precipitation\_Ni-Al-Cr\_Non-isothermal\_Gamma-Gamma\_prime*.

## INCLUDING PROBE DATA FROM AN AM CALCULATOR



When you also have a license for the Additive Manufacturing (AM) Module you can include probe data from the AM Calculator in the set up of the thermal profile.

There is connectivity between the Diffusion Module (DICTRA) or the Precipitation Module (TC-PRISMA) and the Additive Manufacturing (AM) Module when you have full licenses available for all relevant Add-on Modules and databases.

You can add a Diffusion Calculator or Precipitation Calculator as a successor to the AM Calculator and then when probes are included with the AM Calculator this data can be further used with the thermal profile.

Details are in the AM Module documentation. See [AM Calculator: Working with Probe Data](#).

### *Guidelines for Importing Data Files to Thermal Profiles*

The **Thermal Profile** settings are done on the **Configuration** window for a **Precipitation Calculator**. The profile is then previewed in the **Visualizations** window.

You can import data from a file. There are specific guidelines needed to correctly prepare the data in this file to ensure it imports without errors.

### **DATA FILE REQUIREMENTS**

For a Thermal Profile you can either directly enter **Time** and **Temperature** coordinates into the table on the **Configuration** window, or import the data from a file. A minimum of two points is required.

Although there are only two columns to define, it is still important to ensure the file is set up correctly. In order for the data to be included in the calculation, the column and row definitions in the file must adhere to certain rules of entry.

## General Rules

- No abbreviations are allowed in the header.
- A header is not actually required in this file but in order to know which column is which, it is recommended to include it.
- All header text must be in English, i.e. no translations into other languages will be read correctly by the data file.



Only the columns described here are read / used in the calculation. It is important to enter the information exactly as described. It is recommended that the data file is cleaned up before importing / loading it and performing calculations.

	A	B
1	Time	Temperature
2	0	1173
3	773	1050
4	1000	1075
5	1500	2000

Figure 10: This is an example of data entered into an Excel spreadsheet (which is then converted to csv format). Columns A and B show the data entered to create the Thermal Profile.

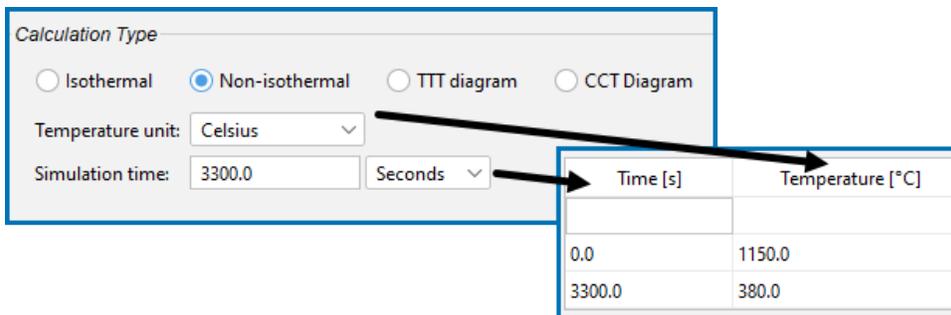


Figure 11: The units shown on the Configuration window for the Time and Temperature correspond to settings that can ONLY be selected on the Configuration window. The data file does not contain any information about the units.

Column Heading Requirements and Options	Row Cell Requirements and Options
<b>Time</b>	Valid format and entries for each row in this column:

Column Heading Requirements and Options	Row Cell Requirements and Options
<p>This is column A as in <a href="#">Figure 10</a></p> <p>Enter the text <code>Time</code>. No abbreviations allowed but it is not case sensitive. It is also not required but recommended to include so you know which column of data it applies to.</p>	<ul style="list-style-type: none"> <li>Any numerical input for the time that corresponds to the unit selected on the Configuration window e.g. 0.0, 773, 1000, 1500, are times in seconds, which is the unit chosen in the example shown in <a href="#">Figure 11</a>.</li> </ul>
<p><b>Temperature</b></p> <p>This is column B as in <a href="#">Figure 10</a></p> <p>Enter the text <code>Temperature</code>. No abbreviations allowed but it is not case sensitive. It is also not required but recommended to include so you know which column of data it applies to.</p>	<p>Valid format and entries for each row in these columns:</p> <ul style="list-style-type: none"> <li>Any numerical input for the temperature that corresponds to the unit selected on the Configuration window e.g. 1173, 1050, 1075, are temperatures in Celsius, which is the unit chosen in the example shown in <a href="#">Figure 11</a></li> </ul>

## Particle Size Distribution (PSD)



### [P\\_10: Initial Particle Size Distribution of Fe-Cr-C](#)

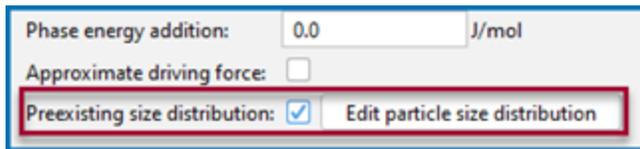
On the Precipitation Calculator you can set the pre-existing particle size distribution (PSD) parameters in the **Show Details** section. The size distribution can be entered as a pre-defined distribution, by importing a file, or by manually entering information into a table.

## Accessing the 'Pre-existing Size Distribution' Settings

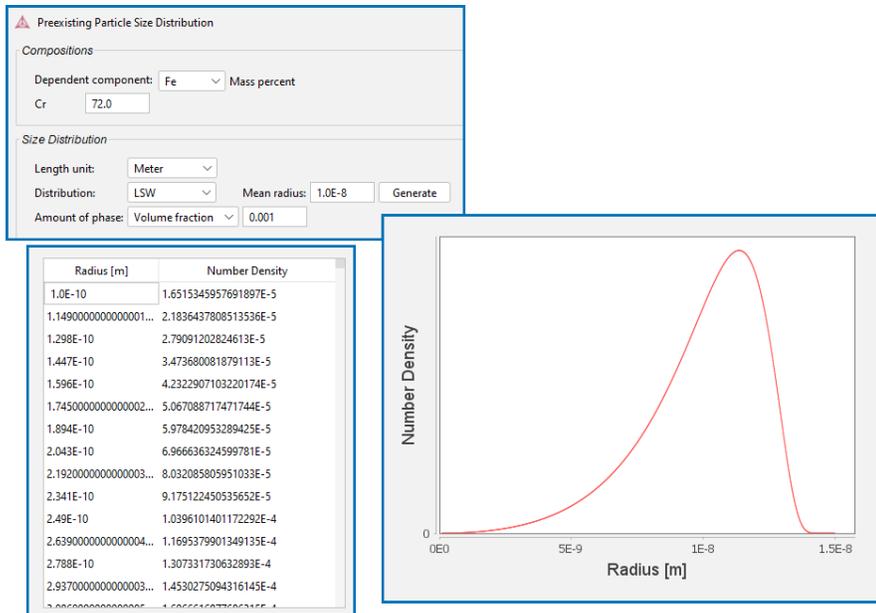
- To the right of the *Precipitate Phase* settings fields, click **Show Details**.



- At the bottom of the *Precipitate Phase* section, click to select the **Preexisting size distribution** checkbox. Then click **Edit particle size distribution**.



- The **Preexisting Particle Size Distribution** window opens as shown for example P\_10.



## Defining the Preexisting Size Distribution

1. Under *Compositions*, choose a **Dependent component** from the list and enter the composition for the other component.
2. Under *Size Distribution* define the following:
  - a. Choose a **Length unit: Meter, Micrometer, Nanometer, or Ångström.**
  - b. Choose a **Distribution: LSW (Lifshitz-Slyozov-Wagner), Normal, Log normal, Weibull, or From file.** You can also click in the table to manually enter numbers into the cells instead of importing a file.

Radius [m]	Number Density
<input type="text"/>	

- c. Then based on the **Distribution** selected, enter additional settings:
  - **Mean radius:** The mean particle radius.
  - **Sigma:** The standard deviation for the chosen length unit.
  - **Alpha:** A dimensionless shape factor that should vary between 1.5 and 10.
  - **Fraction offset:** The fraction offset applied to the mean radius to shift the start of the distribution. This must be greater than or equal to zero, and less than 0.95.
- d. If **From file** is selected, click **Import** and navigate to the file containing the required information and click **Open**. This file can be in .xls, .xlsx, .csv, or .txt formats. The file should consist of two columns with values where the first column contains **radius** data and the second contains **number density** data.
- e. For all options, choose an **Amount of phase: Volume percent or Volume fraction** and then enter a number in the field.
- f. Once the **Distribution** is defined, click **Generate**.

### Calculation Type Settings

Below are details about the settings for the *Calculation Type*, which is selected from the **Conditions** tab on the Precipitation Calculator **Configuration** window.



There are additional **Conditions** tab settings described for the [Matrix Phase](#) and [Precipitate Phase](#).

## Isothermal

Use an **Isothermal** calculation type to do a simulation at constant temperature.



These settings are described in the section for the [Precipitation Calculator Thermal Profile](#).

## Non-isothermal

Use a **Non-isothermal** calculation type to do a simulation with a user-defined temperature profile. You can enter information directly or import a data file.



These settings are described for the [Precipitation Calculator Thermal Profile](#) including information about importing data files.



When you also have a license for the Additive Manufacturing (AM) Module you can include probe data from the AM Calculator in the set up of the thermal profile.

## TTT Diagram

Use a **TTT-diagram** to calculate the time-temperature-transformation (TTT) curve for the formation of the precipitate phase.

- **Temperature:** Enter **Min**, **Max**, and **Step** numerical values and choose a temperature **Unit**.
- **Max annealing time:** Enter a numerical value and choose a time **Unit**.
- **Stop criteria:** Choose **Volume fraction of phase** or **% of equilibrium fraction** and then enter a numerical value. For each temperature, the simulation stops when the stop criteria is fulfilled or if the maximum annealing time is reached, whichever happens first.



See example [P\\_03: Stable and Metastable Carbides - TTT Diagram](#)

## CCT Diagram

Use a **CCT-diagram** to calculate the continuous-cooling-transformation (CCT) curve for precipitation.

- **Temperature:** Enter **Min** and **Max** numerical values and choose a temperature **Unit**.
- **Cooling rate(s):** Enter a range of values in the field, e.g. .01 .1 1 10 100. These values are equal to **K/s**, **°C/s** or **°F/s** per second based on the temperature **Unit** selected.
- **Stop criteria:** Enter a numerical value for the **Volume fraction of phase**. For each temperature, the simulation stops when the stop criteria is fulfilled.



See example [P\\_07: Continuous Cooling Transformation \(CCT\) Diagram of Ni-Al-Cr  \$\gamma\$ - \$\gamma'\$](#) .

### *Numerical Parameters on the Options Tab*

The Precipitation Calculator *Numerical Parameters* settings are located on the **Configuration** window → **Options** tab.



You can change these settings locally for a specific Precipitation Calculator or globally for some defaults in the **Options** window settings.

### Max Time Step Fraction

The **Max time step fraction** is the maximum time step allowed for time integration as fraction of the simulation time. The default is 0.1.

### No. of Grid Points Over One Order of Magnitude in Radius

The default number of grid points for every order of magnitude in size space. The default is 150.0.

### Max No. of Grid Points Over One Order of Magnitude in Radius

The maximum allowed number of grid points in size space. The default is 200.0.

**Min No. of Grid Points Over One Order of Magnitude in Radius**

The minimum allowed number of grid points in size space. The default is 100.0.

**Max Relative Volume Fraction of Subcritical Particles Allowed to Dissolve in One Time Step**

The portion of the volume fraction that can be ignored when determining the time step. The default is 0.01.

**Max Relative Radius Change**

The maximum value allowed for relative radius change in one time step. The default is 0.01.

**Relative Radius Change for Avoiding Class Collision**

Set a limit on the time step. The default is 0.5.

**Max Overall Volume Change**

This defines the maximum absolute (not ratio) change of the volume fraction allowed during one time step. The default is 0.001.

**Max Relative Change of Nucleation Rate in Logarithmic Scale**

This parameter ensures accuracy for the evolution of effective nucleation rate. The default is 0.5.

**Max Relative Change of Critical Radius**

Used to place a constraint on how fast the critical radius can vary, and thus put a limit on time step. The default is 0.1.

**Min Radius for a Nucleus to be Considered as a Particle**

The cut-off lower limit of precipitate radius. The default is 5.0E-10 m.

## Max Time Step During Heating Stages

The upper limit of the time step that has been enforced in the heating stages. The default is 1.0 s.

## Maximum Relative Solute Composition Change at Each Time Step

Set a limit on the time step by controlling solute depletion or saturation, especially at the isothermal stage. The default is 0.01.

## Pause, Resume, and Cancel Precipitation Calculations

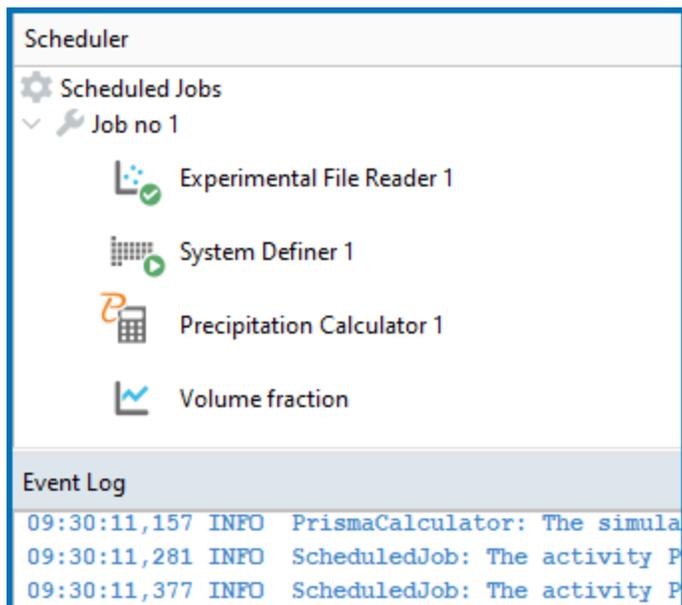
Precipitation calculations are often complex simulations that take some time to complete. Sometimes you may want or need to pause or resume a calculation, or make adjustments to your compositions and start again. You pause and resume from the Precipitation Calculator **Configuration** window.



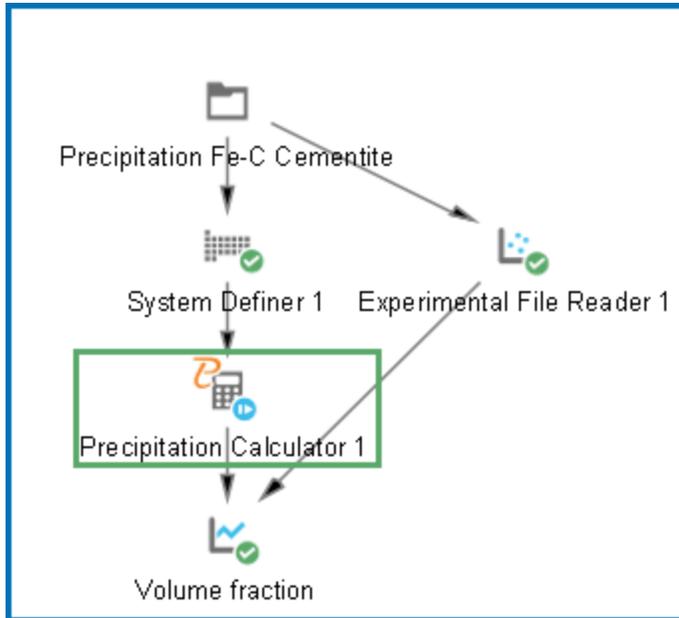
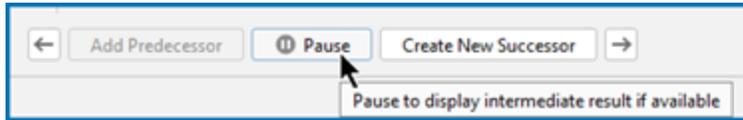
This is also available for the Diffusion Calculator (but not any other calculators). See [Pause, Resume, and Cancel Diffusion Calculations](#).

### To Pause and Resume a Job

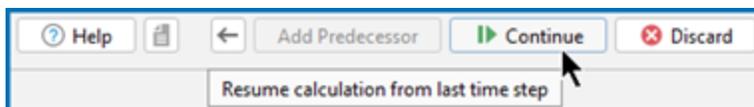
1. Run the job (i.e. click **Perform Tree**). In the **Scheduler**, you can see the job listed and in the **Event Log** you can follow the progress of the calculation.



- In the **Project** window, click the **Precipitation Calculator** node.
- In the **Configuration** window at the bottom, click **Pause**. Observe that the **Precipitation Calculator** in the **Project** window has a pause indicator. If there are intermediate results available these will be listed in the **Event Log**.

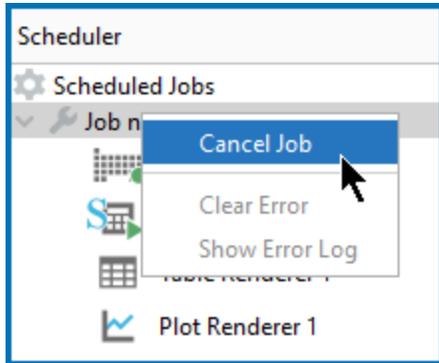


- When pausing the calculation, you can make changes to a variety of settings under the *Matrix Phase* or *Precipitate Phase* sections (click **Show Details**), plus the **Interfacial energy**, **Temperature** (for non-isothermal calculations), and **Simulation time**. The rest of the **Configuration** window settings are grayed out and no other editing can be done while paused.
- When ready, click **Continue** to resume your calculations from the last time step or click **Discard** to discard the calculation (then click **Yes** or **No** on the window that opens).



### To Cancel a Job

- In the **Scheduler** window, right-click the job you want to cancel and select **Cancel Job**.



## Precipitation Calculator Plot Renderer

The following is information about the settings available for a **Plot Renderer** when it is a successor to a **Precipitation Calculator**.

There are also specific settings related to [Non-Isothermal Simulations](#) that are detailed in this topic.

### Plot Settings

#### Legend Option

Select whether the diagram's legend displays **On** or **Off**.

#### Axis Variable

Set the state variable you want plotted along the X-axis and the Y-axis. The available variables in the list are based on how your system is set up.

Below are additional details related to the *Axis variable* chosen.

### SEPARATE MULTIMODAL PSD

If you select **Mean radius**, **Number density**, **Size distribution**, **Number density distribution**, or **Normalized number density distribution** then select the **Separate multimodal PSD** checkbox to enter settings for **Points**, **Excess kurtosis**, **Valley depth ratio** and **Minimum peak**.

 There are additional settings described for [Non-Isothermal Simulations](#).

### 3D → 2D

For a spherical morphology, and if you select **Mean radius**, **Size distribution**, **Number density distribution**, or **Normalized number density distribution** as an axis variable, you can also select the **3D → 2D** checkbox to convert the 3D property to the corresponding 2D property considering the resulting 2D size distribution obtained from sectioning a 3D spherical dispersion. See [3D to 2D Conversion](#) for settings.

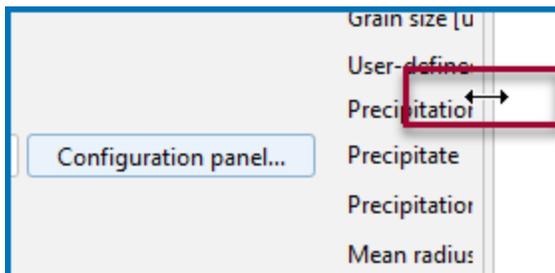
### YIELD STRENGTH

If you choose **Yield strength** as an *Axis variable*, you can further define the model. Click to expand the drop down list to the right of the *Axis variable*, then select any or all of the checkboxes listed below to plot the respective contributing elements to the yield strength.

Click **Configuration Panel** toggle between the settings options and to further fine-tune the model. These options are described for the [Yield Strength Property Model: Simplified Mode Settings](#) and [Yield Strength Property Model: Advanced Mode Settings](#). The greyed out sections (e.g. the **Matrix**, **Precipitate** phases, and **Grain size**) are defined on the **Precipitation Calculator** and cannot be changed.



To see these settings after clicking **Configuration Panel**, you need to expand the **Plot Renderer Configuration** window to the right. Either detach the window to resize, or hover with the mouse until you see the double arrow, then click and drag to expand the window.



- Total yield strength
- Intrinsic strength
- Solid solution strengthening
- Solid solution strengthening FCC
- Solid solution strengthening BCC
- Solid solution strengthening HCP
- Grain boundary strengthening
- Total precipitation strengthening
- Precipitation strengthening per phase
- Constant strength addition



The **P\_01: Isothermal Precipitation of Al3Sc** example demonstrates the use of this Yield Strength Model.

### 3D to 2D Conversion

When the *Precipitate PhaseMorphology* is set to **Sphere** on the Precipitation Calculator, the following options are available when certain axis variables are also selected.

Select the **3D → 2D** checkbox if you want to make a direct comparison between the predicted three-dimensional (3D) radius of the precipitates and the two-dimensional (2D) data obtained from a micrograph.



The stereological conversion is described in the theory for spherical particles.

These variables can be converted to 2D:

- **Mean radius:** The average circular radius of cross sections of particles obtained by taking a section through the dispersion, assuming spherical precipitates.
- **Size distribution:** The number density of particle cross sections that have a radius varying within a given size class in a unit area.
- **Number density distribution:** The number of particle cross sections that have a radius varying within a given size class in a unit area.

- **Normalized number density distribution:** The probability of finding particle cross sections with a cross sectional radius varying within the size class. The radius is normalized by the volumetric mean of the dispersion.

## Axis Type

Select the type of axis scale: **Linear**, **Logarithmic**, **Logarithmic 10**, or **Inverse**.

## Limits

Specify the range along the axis to show in the plot. In the fields, enter the minimum and maximum values of the axis variable. You can also determine the **step** size between the tick marks along each axis.

Select the **Automatic scaling** checkbox to allow the program to set the limits.

## Unit (*Time X Axis*)

Choose a **Unit**: **Seconds**, **Hours**, or **Days**.

## Add an Axis and Remove this Axis Button

Use the **Add an axis** ⊕ and **Remove this axis** ⊖ buttons to add additional X- and/or Y-axes to a plot or to merge plots (to overlay one plot with another in the same diagram). When merging plots, you define this variable as the Z-axis.

## Axis Variables

- **Mean radius:** Spherical radius of average volume of all particles for a specific phase and nucleation type, regardless of their actual shapes. If the matrix phase is selected, it outputs the average grain radius.
- **Critical radius:** Spherical radius of critical nuclei for a specific phase and nucleation type. If the matrix phase is selected, it outputs critical grain radius.
- **Yield strength:** To use the Yield Strength Property Model to calculate yield stress. For more information see [About the Yield Strength Property Model](#).
- **Matrix composition:** Instantaneous compositions of the matrix phase.
- **Precipitate composition:** To track the instantaneous composition of precipitate particles. In particular, it is useful to distinguish different composition sets of the same phase (for example, FCC\_A1#2 and FCC\_A1#3). Further choose **Solutes** or **All**.

- 
- **Number density:** Instantaneous number of particles per unit volume for a specific phase and nucleation type. If the matrix phase is selected, it outputs instantaneous number of grains per unit volume.
  - **Size distribution:** Number of particles, or grains if the matrix phase is selected, varying with their sizes per unit volume per unit length, for a specific phase and nucleation type, at a specific time.
  - **Number density distribution:** Retrieve the number density (number of particles per unit volume) of precipitates, or grains if the matrix phase is selected, distributed in different particle sizes.
  - **Normalized Number density distribution:** Retrieve the number density (number of particles per unit volume) of precipitates, or grains if the matrix phase is selected, normalized by total number of particles or grains per unit volume, distributed in different particle sizes normalized by average size.
  - **Volume fraction:** Instantaneous volume fraction for a specific phase and nucleation type.
  - **Nucleation rate:** Instantaneous number of nuclei per unit volume per unit time for a specific phase and nucleation type.
  - **Normalized driving force:** Instantaneous nucleation driving force per unit mole of elements for a specific phase and nucleation type. It is normalized with  $RT$  and therefore, dimensionless.
  - **Temperature**
  - **Mean cubic factor:** Average cubic factor of cuboid particles. Available only when cuboid is selected as the **Morphology**.

The following settings area available as indicated when **Needle**, **Plate**, or **Cuboid** are selected as the **Morphology** in the *Precipitate Phase* section on the Precipitation Calculator.

- **Cubic factor distribution:** Variation of cubic factor with particle size at a specific time. Available with a cuboid morphology.
- **Mean aspect ratio:** Average aspect ratio of non-spherical particles. Available with a needle or plate morphology. Note that this is always larger than 1, where 1 = a sphere.
- **Mean particle length:** Diameter of non-spherical particles along the longer axis. Available with a needle or plate morphology.
- **Aspect ratio distribution:** Variation of aspect ratio with particle size at a specific time. Available with a needle or plate morphology.

## Non-Isothermal Simulations

When doing non-isothermal simulations it is common that particles grow in different generations. This results in multimodal size distributions. To correctly estimate the properties of these different generations of particles you need to separate the peaks of multimodal distributions.

### Separate Multimodal PSD

When the **Separate multimodal PSD** checkbox is selected on a Plot Renderer activity for the Precipitation Calculator, the size distribution is evaluated at the given time steps and checked for multimodal peaks. These are separated and used to calculate the specified property.



#### P\_06: Precipitation of $\gamma'$ in Ni Superalloys - Non-isothermal

The screenshot shows the 'Precipitation Ni-8Al-8Cr' window. The 'Legend option' is set to 'On'. Under 'Y-axes', there are expand/collapse and refresh icons. The 'Axis variable' is set to 'Size distribution'. The 'FCC\_L12#2 (Bulk)' is selected in the dropdown. The 'Time' is '2100.0 3300.0' with a unit of 'Seconds'. The 'Separate multimodal PSD' checkbox is checked. Below these fields, there are input boxes for 'Excess kurtosis: 25.0', 'Valley depth ratio: 0.05', and 'Minimum peak: 100000.0'.

It can be applied on the following plot properties:

- Mean radius
- Size distribution
- Number density
- Volume fraction

### Points

This field is available when **Mean radius**, **Number density**, or **Volume fraction** is selected as the **Axis variable**. Since the evaluation of multimodality at each time step is costly, you can specify how many evaluation **Points** to use. The points are geometrically distributed over the time scale where nucleated particles are found. The default is 40 points.

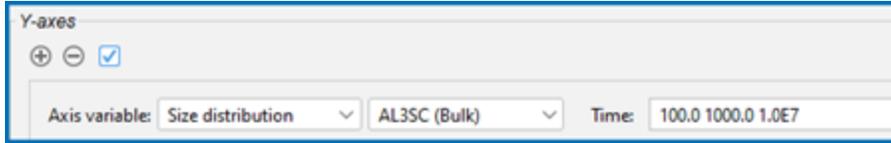
### Time

This field is available when **Size distribution** or **Number density distribution** is selected as the **Axis variable**. Enter one or a series of numbers in the field, separated by a space.

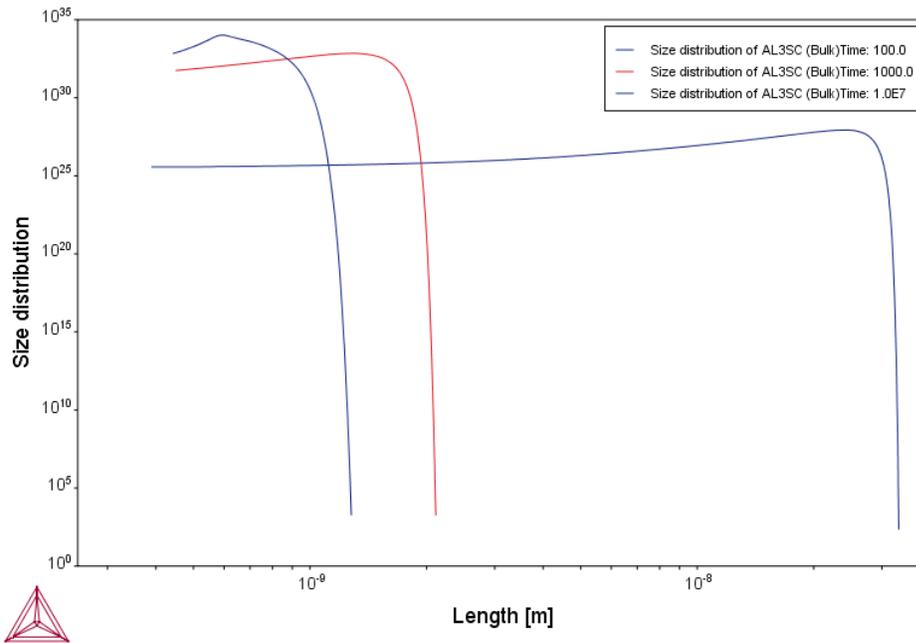


## P\_09: Precipitation of Al-Sc AL3SC with Assumption of Sphere and Cuboid Morphologies

For example, if you enter 100.0 1000.0 1.0E7 in the field:



When you click **Perform** the times are plotted:



### Excess Kurtosis

The **Excess kurtosis** number specifies the sensitivity when the program evaluates whether a curve is bimodal or not. The evaluation process calculates the excess kurtosis of the given size distribution and compares it with the input value. If the value is lower than the given value, the distribution is split. The excess kurtosis is a statistical measure that measures how *peaked* and *how heavy tail* a distribution has. For the definition used see [Wikipedia article](#). The default is 25.0.

### Minimum Separation Limit (Valley Depth Ratio)

The **Minimum separation limit** is a rough method to prevent that noise or wiggles are interpreted as different distributions. If a local minima is encountered above this value the following peak is not interpreted as a new distribution. The valley depth ratio is the ratio of the minimum separation limit to the peak value. The default is 0.01.

### Minimum Peak

Any separated peak below the entered **Minimum peak** value is removed. The default is 100000.0.

## Plot Renderer

The Plot Renderer is where you define the settings and output results in a diagram that displays in the **Visualizations** window. The available Plot Renderer settings are based on the System Definer and/or calculator activity node predecessor (i.e. Equilibrium Calculator, Scheil Calculator, Property Model Calculator, and so forth). Settings availability is also determined by the *Calculation Type* as well as if you have an Add-on Module. If you used any of the templates, the Plot Renderer is automatically added to the Project tree.

### [Configuration Settings](#)



There are additional details for more advanced settings and the Add-on Modules: [Plotting Options](#), [Diffusion Calculator Plot Renderer](#), [Precipitation Calculator Plot Renderer](#), and [AM Calculator Plot Renderer](#).



3D Plots with additional functionality are also available with the Add-on Additive Manufacturing (AM) Module. The information is described separately and included with the Module's documentation.

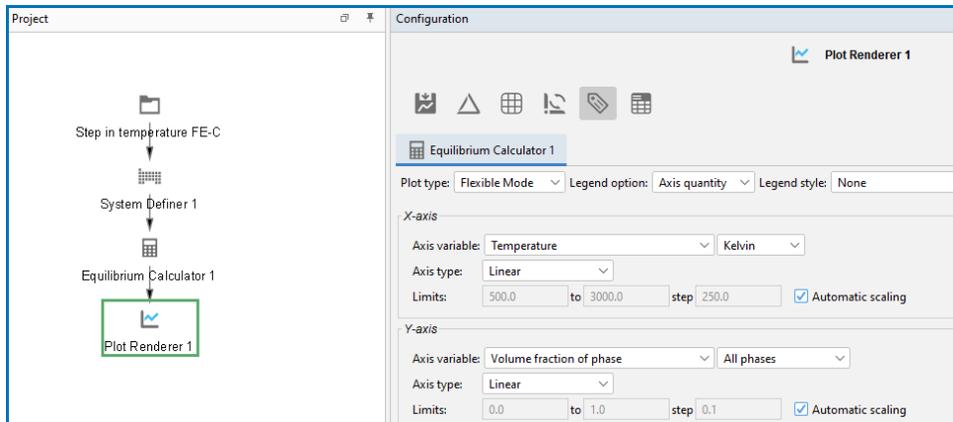
## Plot Toolbar

Button	Name	Description
	Save Diagram	Save the plot diagram. From the <b>Save</b> window, choose the file type—png (the default), jpg, ps, pdf, gif, svg, or emf. For PNG and JPG files you can also change the image quality.   <a href="#">Saving and Printing the Output (Graphical Mode)</a>
	Toggle Triangular/Regular Diagram	Display the diagram in triangular form, with the X-axis along the base and the Y-axis along the left side. Such a diagram is often useful if you want to plot the fractions of two components in a ternary diagram.
	Show Grid	Overlay a grid on the diagram.
	Switch Axes	Show the plotting X-axis variable on the diagram's Y-axis, and the Y-axis variable on the diagram's X-axis.

Button	Name	Description
	Retain Labels	Toggle the default to keep labels displaying on the plot. By default labels are retained.
	Table View	<p>Toggle to switch between displaying a Plot or the results of a Table in the <b>Visualizations</b> window. This is available for single equilibrium or stepping calculations. When you click to toggle ON, click <b>Perform</b> to regenerate as a Table. When you click to toggle OFF, click <b>Perform</b> to regenerate the Plot.</p> <p>For example, set up the Plot Renderer and click <b>Perform</b> to display the Plot. Then click <b>Table View</b> and <b>Perform</b> to see the results in a Table format in the <b>Visualizations</b> window.</p> <p>When toggled ON, click <b>Save table</b> to export the table data. Choose the <b>Decimal digits</b> and <b>Number format (Auto, Decimal, or Scientific)</b>.</p> <p> <a href="#">Saving and Printing the Output (Graphical Mode)</a></p>

## Configuration Settings

Click the **Plot Renderer** node in the **Project** window to view the **Configuration** window. The settings available vary based on such choices as the specific calculator used and calculation type selected.



### [Working with Plots and Tables](#)

For the Add-on Modules with additional specialized functionality also see:

- [Diffusion Calculator Plot Renderer](#)
- [Precipitation Calculator Plot Renderer](#)
- [AM Calculator Plot Renderer](#)

---

## Settings

The options along the top of the window are available depending on the calculation type.

### Plot Type

There are a variety of combinations of Calculator and *Calculation Type* that offer different [Plot Types](#). These are described in separate topics:

- [Plot Type: Cross Plot and Statistical](#)
- [Plot Type: Heat Map, Contour, and 3D](#)
- [Plot Type: Parallel Coordinates](#)
- [Plot Type: Flexible Mode and Grouped Mode](#)
- [Plot Type: CCT Mode](#)
- [Plot Type: TTT Mode](#)
- [Plot Type: Simple Mode and Advanced Mode](#)

### Tie-lines

Select how many tie-lines to be plotted in the diagram. Every  $n^{\text{th}}$  number of tie-line is plotted, where  $n$  is the number set here. Consequently, the higher the number, the fewer the number of plotted tie-lines in the diagram.



This setting is not available if the Plot Renderer is a successor to an Equilibrium Calculator where only one axis is defined and varied (that is, an Equilibrium Calculator that has performed a stepping calculation).

### Legend Option

Select whether the diagram's legend displays the **Stable phases**, the **Axis quantity**, or choose **Off** for no legend.

### Legend Style

You can select a **Legend style** for the **Stable phases** or **Axis quantity** selected for **Legend option**. Choose **None**, **All**, **Constitution description**, or **Ordering description**.



## [About Legend Styles](#)

### Filter

This is available with the **Property Model Calculator** and **Process Metallurgy Calculator**.

Use the **Filter** checkbox to add various filters to the plot or table based on the set up of the calculation. When selected, click the **Add quantity**  and **Remove quantity**  buttons to add filters. For example, enter a **Min** and **Max** and choose an axis variable such as **Ms Temperature** and the unit **Kelvin** to adjust the output in the **Visualizations** window. Click **Perform** each time you add or remove a filter for the results to update.

### Show Limits (Batch)

This is available for the **Batch** calculation type, **Cross plot**.

Click the **Show limits** checkbox and then choose **RMS** (root mean squared), **Absolute**, or **Relative in % limits**. For **Absolute** or **Relative in %** enter a value in the field (the default is 15.0). A "perfect fit" line is added (solid line) to the plot and then the upper and lower limits shown on the plot are based on the type of limit selected.

- When **RMS** is selected, a "perfect fit" line is added (solid line) plus the upper and lower limits (dashed lines), which are calculated for the experimental data points (read from the data file). The RMS error value is shown in the plot legend. The upper and lower limits are then this calculated value plus and minus from the perfect fit line.
- When **Absolute** is selected, the value entered (e.g. the default is 15), then this is the plus or minus value from the perfect fit line.
- When **Relative in %** is selected, the value entered (e.g. the default is 15%), then this is the upper and lower limit that is plotted and this value is plus or minus percent from the perfect fit line as is calculated and shows in the plot legend.

### Show Calculation Failures at Value (Batch)

This is available for the **Batch** calculation type, **Cross plot**.

Click the **Show calculation failures at value** checkbox and enter a value in the field. Use this checkbox to include any values that failed in the calculation and you want to see these on the plot. The value entered in the field replaces the non-existing calculated value(s) and can be cross plotted against experimental results from that data file.

## Interpolate (Grid)

From the **Interpolate** list, choose to interpolate data **Never** (the default), **Once**, **Twice** or **3 times**. This means points are put in between calculated points in the grid and interpolated instead of calculated in the calculation engine.

Choose how many points of data to interpolate to improve curve smoothness – **Never**, **20x20**, **50x50**, or **100x100**. Try different settings for your own set of data.

## Label Decimal Digits

Choose the number of decimal digits to display in the plot label. The default is **2**.

## Label Number Format

Choose from **Auto** (the default), **Decimal**, or **Scientific**.

## Axis Variable

Set the state variable you want plotted along the X-axis and the Y-axis (and Z-axis for Grid calculations).



If the stepping/mapping variables are changed in the Equilibrium Calculator that precedes the Plot Renderer, then the variables to be plotted along the diagram's X- and Y-axis are automatically updated to the appropriate quantities.



For a **Grid Calculation Type**, the X- and Y-axes always represent the quantities that define the calculation axes, but you need to set a variable for the Z-axis (the axis perpendicular to the plane defined by the X- and Y-axes).



For a **BatchCalculation Type**, the available options from the X- and Y- axes lists are based on the data file contents, i.e. if experimental data is included, then these are listed. See [Working with Batch Calculations](#) for details. Additional [Plot Types](#) are also available.

## Axis Type

Select the type of axis scale: **Linear**, **Logarithmic**, **Logarithmic 10**, or **Inverse**.

## Automatic Scaling

The **Automatic scaling** checkbox is selected by default to allow the program to set the limits.

## Limits

This setting is available when the **Automatic scaling** checkbox is cleared (the scaling is turned off).

Enter the lower and upper **Limits** of the variables to display on the axis. You can also determine the **step** size between the tick marks along each axis.



If the **Automatic scaling** is turned off (i.e. the checkbox is not selected) then it is important to enter a large enough number in the *step* field for the numerical values to display correctly on the axis. That is, if the step size is too small, the program ignores this setting and the axis labels will not be as expected.



For logarithmic axes, the step size is related to the exponent. For example, if the step entered is 1.0, then the exponent increases by one each time ( $10^{-9}$ ,  $10^{-8}$ ,  $10^{-7}$  and so forth).

## Contour Values and Use Custom Contour Values (Z-axis, Contour Plots)

The **Use custom contour values** checkbox is available when **Contour** is selected as the **Plot type**. Select this checkbox to enter one or a series of numbers in the **Contour values** field. Numbers can only be separated by a space.

---

## Add an Axis and Remove this Axis Buttons

Use the **Add an axis**  and **Remove this axis**  buttons to add additional X- and/or Y-axes to a plot or to merge plots (to overlay one plot with another in the same diagram). When merging plots, you define this variable as the Z-axis.

 [Plotting Options](#)

## Slag Property Options (Process Metallurgy)

For the **Process Metallurgy Calculator** when you are selecting axis variables on the Plot Renderer, there is the option to choose **Slag property**. The associated quantity that you select from the next drop-down list has these options to choose: **All**, **B2**, **B3**, **B4**, **Bas2**, **Bells ratio**, **Ls**, or **Log10 (Sulphur capacity)**.

Also choose **In liquid slag only**, **In solid slag only**, or **In all slag**.

 References and equations for these properties are described in [Process Metallurgy Slag Properties](#).

## Unit (Process Metallurgy)

For the **Process Metallurgy Calculator** select a **Unit** for the quantity to plot: **Tonne**, **Kilogram**, **Gram**, or **Pound**

## Table Renderer

A **Table Renderer** visualizes calculation results in a table. Several Table Renderer activities (as well as Plot Renderer activities) can be shown in the **Visualizations** window as different tabs.

You can add the Table Renderer directly (right-click a calculator node and select it from the **Create New Successor** menu) or use a variety of the available templates to automatically add the node. For example, on the  **My Projects Configuration** window under **Equilibrium**, choose the **Single Point** or **One Axis** templates.



For details about the templates, see [Getting Started Links and Templates](#) and [Creating a Project from a Template](#).



You can only successfully tabulate the results of a single equilibrium calculation or a stepping (i.e. property diagram) operation. The tabulated data from an equilibrium calculation is different from the tabulated data from a stepping calculation.



The settings are described in the section [Configuration Settings](#).

### Table Button

Button	Name	Description
	Save Table	Once a calculation is performed and there is data in the table, click to save it.   <a href="#">Saving and Printing the Output (Graphical Mode)</a>



For the Plot Renderer, there is an option to use the Table View  button to convert plots to table data for all calculation types and calculators. See [Plot Renderer: Convert a Plot to a Table and Export the Data](#).

## Results from an Equilibrium Calculation

The following tab shows the result of an equilibrium calculation (from example [T\\_01: Calculating a Single-Point Equilibrium](#)).

Table Renderer 1				
<b>System</b>				
Moles	1.00000			
Mass	55.64392	[g]		
Temperature	1000.00000	[K]		
Total Gibbs Energy	-42140.55264	[J]		
Enthalpy	24707.16903	[J]		
Volume	7.29746E-6	[m3]		
<i>Component</i>	<i>Mole Fraction</i>	<i>Mass Fraction</i>	<i>Activity</i>	<i>Potential</i>
Fe	0.99537	0.99900	0.00619	-42277.77189
C	0.00463	0.00100	0.21818	-12658.35450
<b>Stable Phases</b>				
	<i>Moles</i>	<i>Mass</i>	<i>Volume Fraction</i>	
<b>BCC_A2#1</b>	0.99609	55.59691	0.99711	Composition and constitution ▾
<b>Composition</b>				
<i>Component</i>	<i>Mole Fraction</i>	<i>Mass Fraction</i>		
Fe	0.99928	0.99984		
C	0.00072	0.00016		
<b>Constitution</b>				
	$(Fe)_1 (C, VA)_3$			
	<i>Constituent</i>	<i>Site Fraction</i>		
<i>Sublattice 1:</i>				
	Fe	1.00000		
<i>Sublattice 2:</i>				
	VA	0.99976		
	C	0.00024		

For each stable phase listed, from the **Composition and constitution** menu you can select the information to view. In this example, the maximum amount of information is shown and includes details about both composition and constitution for **BCC\_A2#1**.



The default for the Phase description detail can be changed globally from **Options** window.

## Results from a Stepping Operation

The **Table Render 1** tab in the example shows the results of a stepping calculation (where a Table Renderer is added to the example in [T\\_02: Stepping in Temperature in the Fe-C System](#)).

Visualizations			
Plot Renderer 1	Table Renderer 1		
Temperature [K]	Volume fraction of BCC_A2	Volume fraction of FCC_A1	Volume fraction of GRAPHITE
1000.00000	0.99711		0.00289
1010.00000	0.99718		0.00282
1011.17630	0.99719		0.00281
1011.17630	0.99719	0.00000	0.00281
1011.17630	0.87704	0.12296	0.00000
1011.17630	0.87704	0.12296	
1020.00000	0.86401	0.13599	
1030.00000	0.84702	0.15298	
1040.00000	0.82743	0.17257	

## Process Metallurgy Module Single Equilibrium

When you use a **Process Metallurgy Calculator** with a **Single** equilibrium calculation, there is a variety of additional information you can display in the table such as the **Slag properties** (see [Process Metallurgy Slag Properties](#) for equations and references) and the **Activities**.



**PMET\_01: Basic Oxygen Furnace (BOF).** Using version 01b, added a Table Renderer and did a **Single** calculation.

Configuration

Save table    Decimal digits: 5    Number format: Auto    Temperature unit

All stable phases  Composition

Slag properties  All    In liquid slag only

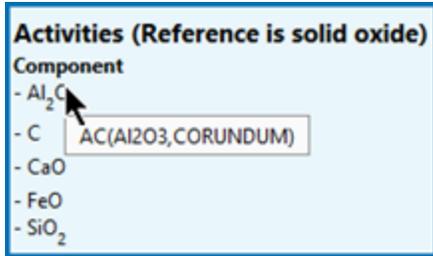
Activity  All    Reference is solid oxide

Visualizations

< liquid metal    Composition of slag    Amount of phase groups    Table Renderer 1

- Si	0.01037	[Mass percent]
- Al	0.16654	[Mass percent]
- O	0.00088	[Mass percent]
- Ca	0.00002	[Mass percent]
<b>Slag properties (All)</b>		
- B <sub>2</sub>	4.28222	
<b>Activities (Reference is solid oxide)</b>		
Component	Activity	Reference p

For the activities in the table, you can hover over the Component and the syntax of the activity displays.



## Yield Strength Property Model

The [Yield Strength Property Model: Simplified Mode Settings](#) is available with the Property Model Calculator. After defining the Model and adding a Table or Plot, all yield strength quantities are available to select from this list. These yield strength quantities are also accessible via a Table Renderer (or Plot Renderer) that is a successor to the Precipitation Calculator.



For details about the quantities, see [About the Yield Strength Property Model](#).



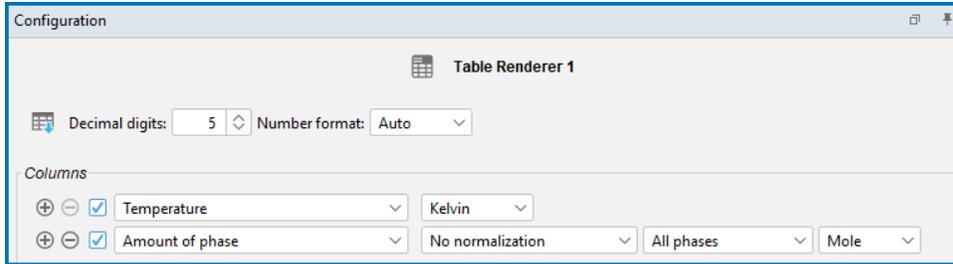
This uses the [PM\\_G\\_05: Yield Strength NiAlCr](#) example and then a Table Renderer is added. Alternatively, you can click the **Table View**  button and reperform the Plot Renderer for the same results.

Visualizations	
Plot Renderer 1	Table Renderer 1
<b>Total yield strength</b>	<b>Mean radius Reppich model</b>
117.79675	0.00000
117.79675	5.00000E-9
176.11187	1.00000E-8

An example of the results when yield strength is chosen as the Axis variable for a Table Renderer that is a successor to the Property Model Calculator.

## Configuration Settings

The **Table Renderer** settings are done on the **Configuration** window. Depending on what calculator node the table is a successor to, you can have different settings available. For most tables, this is where you set the level of detail for the columns as shown in the figure.



## Save the Table

To save all the data in a table as a Character separated (.csv), Hyper Text Markup (.html), or Excel (.xls), in the **Configuration** window click **Save table**  or in the **Visualizations** window right-click in the table and select **Save As**.

To copy the data from a single cell of a table to the clipboard, right-click the cell and select **Copy**. To copy the whole table to the clipboard, select **Copy all**.



The data saved is what is shown in the **Table Renderer** tab on the **Visualizations** window. If data is calculated (and not displayed) it is not saved.

## Number of Decimal Digits and Number Format

The table results can be presented in ordinary decimal number format or with normalized scientific notation. Choose **Scientific** or **Decimal** to increase or decrease the number of **Decimal digits** used in the table. The default is 5 digits. The **Number format** defaults to **Auto** to use the most appropriate format based on the tabulated data.



The defaults for these settings can be changed globally.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

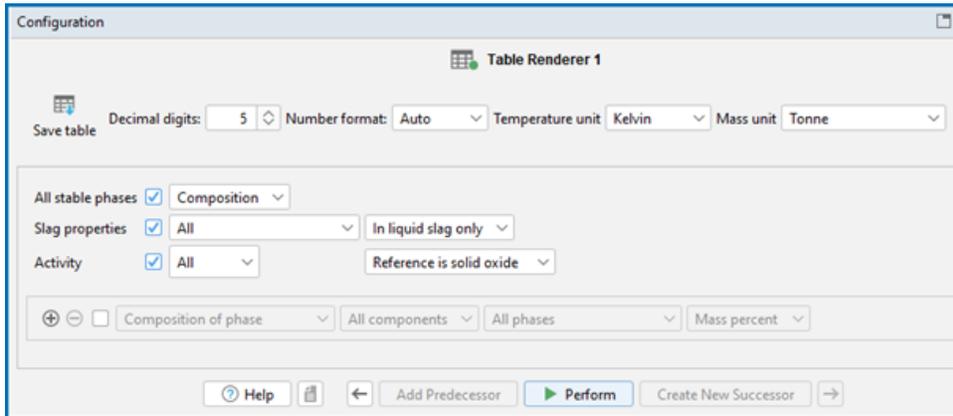
then click the **Activities** tab and the **Tabulation** node in the tree.

## Process Metallurgy Settings

For the **Process Metallurgy Calculator** there are additional settings available.



[PMET\\_01: Basic Oxygen Furnace \(BOF\)](#). Using version 01b, added a Table Renderer and did a **Single** calculation.



Choose a **Temperature unit**: **Kelvin**, **Celsius** or **Fahrenheit**.

Choose a **Mass unit**: **Tonne**, **Kilogram**, **Gram**, or **Pound**.

Select the **All stable phases** checkbox and choose **Composition** or **Condensed** from the list.

Select the **Slag properties** checkbox and choose the associated quantity from the next drop-down list: **All**, **B2**, **B3**, **B4**, **Bas2**, **Bells ratio**, **Ls**, or **Log10 (Sulphur capacity)**.

Also choose **In liquid slag only**, **In solid slag only**, or **In all slag**.



Details, including references and equations, are described in [Process Metallurgy Slag Properties](#).

Select the **Activity** checkbox then from the first list choose a component (or keep the default **All**). Then from the second drop-down list select **Reference solid oxide** or **Reference liquid slag**.

The activity is always given relative to a phase. The liquid slag is always the IONIQ\_LIQ phase. The composition set of ION\_LIQ depends on the calculation and the slag always contains a high amount of oxygen. The reference phase for the solid oxides depends on the components and it is always listed in the table.

## Precipitation Calculator with Yield Strength

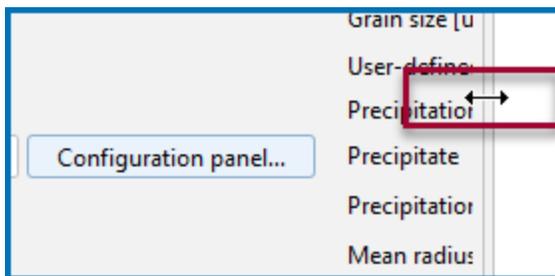
When you use a **Precipitation Calculator** and choose **Yield strength** as an *Axis variable*, there are additional settings available.

If you choose **Yield strength** as an *Axis variable*, you can further define the model. Click to expand the drop down list to the right of the *Axis variable*, then select any or all of the checkboxes listed below to plot the respective contributing elements to the yield strength.

Click **Configuration Panel** toggle between the settings options and to further fine-tune the model. These options are described for the [Yield Strength Property Model: Simplified Mode Settings](#) and [Yield Strength Property Model: Advanced Mode Settings](#). The greyed out sections (e.g. the **Matrix**, **Precipitate** phases, and **Grain size**) are defined on the **Precipitation Calculator** and cannot be changed.



To see these settings after clicking **Configuration Panel**, you need to expand the **Plot Renderer Configuration** window to the right. Either detach the window to resize, or hover with the mouse until you see the double arrow, then click and drag to expand the window.



- Total yield strength
- Intrinsic strength
- Solid solution strengthening
- Solid solution strengthening FCC
- Solid solution strengthening BCC
- Solid solution strengthening HCP
- Grain boundary strengthening

- Total precipitation strengthening
- Precipitation strengthening per phase
- Constant strength addition



The **P\_01: Isothermal Precipitation of Al3Sc** example demonstrates the use of this Yield Strength Model.



[Yield Strength Property Model: Simplified Mode Settings](#)

### Precipitation Calculator Separate Multimodal PSD and 3D > 2D

When certain Axis variables are selected on the Plot Renderer or Table Renderer that is a successor to the Precipitation Calculator, the **Separate multimodal PSD** and **3D > 2D** checkboxes are available for the Precipitation Module (TC-PRISMA).



Another requirement is that the *Precipitate Phase Morphology* is set to **Sphere** on the Precipitation Calculator.

See the specialized section for the Plot Renderer for details.



[Precipitation Calculator Plot Renderer](#)

## Experimental File Reader

---

The **Experimental File Reader** activity node is added as a predecessor from the top  **My Project** node. This allows you to use (or "read") the data contained in an experimental data file (with the file extension \*.EXP) and then plot the data on one or more Plot Renderers.



See the [Tips and Tricks: How to Plot Experimental Data](#) blog post on our website for useful guidance about this topic.



If you are a Console Mode user, this is the same concept as using the [Experimental Data Files \(\\*.POP, \\*.DOP\)](#).

The EXP file is a type of data file with information specifying a plotted diagram and it is written in the DATAPLOT graphical language. From this activity node you can plot the EXP file or save a copy of the file to another location.



[Creating Activities and Successors](#) and [Plotting Experimental Data Files](#) in the *DATPLOT User Guide* included with this documentation set.

### How to Plot Using an Experimental Data File

1. Right-click  **My Project** and from the **Create New Activity** submenu choose **Experimental File Reader**.
2. In the **Configuration** window in the *EXP file* field, enter the name of the EXP file to plot. Or click the **Select Exp file**  button to navigate to a location on the computer to add the dataset.
3. Once the .EXP file is added, under **Dataset Selection** you can choose to include or exclude specific datasets. Use the **Select/Deslect all datasets** checkbox as required.
4. Right-click the **Experimental File Reader** node and from the **Create New Successor** submenu choose **Plot Renderer**.

If the source EXP file is changed, you can also click the **Reload the selected EXP file from disk**  button to refresh the data. If you want to save or export the loaded file, click the **Save EXP file**  button, then navigate to where you want to save it and give it a file name.

5. In the **Configuration** window click **Perform** or right-click the **Plot Renderer** node and select **Perform**.

# General Property Models

---

 [Property Model Calculator](#)

## In this section:

---

About the Property Models .....	260
Scheil Advanced Options: Property Models .....	265
About the Coarsening Property Model .....	268
About the Columnar to Equiaxed Transition (CET) Property Model .....	272
About the Crack Susceptibility Coefficient Property Model .....	277
Driving Force Property Model Settings .....	290
About the Equilibrium with Freeze-in Temperature Property Model .....	292
Equilibrium Property Model Settings .....	297
About the Interfacial Energy Property Model .....	300
Liquid and Solidus Temperature Property Model Settings .....	303
Phase Transition Property Model Settings .....	306
Scheil Property Model Settings .....	309
About the Spinodal Property Model .....	314
About the T0 Temperature Property Model .....	317
About the Yield Strength Property Model .....	322

## About the Property Models

---

Property Models are used to predict and optimize properties of materials using models stored within the software where the calculations are set up in the [Property Model Calculator](#).

### General Models

The following General Property Models are available with the Property Model Calculator.

The **General Models** are available to all users.



To run calculations with the **Steel Models** (as part of the Steel Model Library) requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. To run calculations with the **Nickel Models** (as part of the Nickel Model Library) requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases.



[About the Steel Model Library Property Models](#) and [About the Nickel Model Library Property Models](#)

### Coarsening

The **Coarsening** Property Model calculates the coarsening rate coefficient  $K$  ( $\text{m}^3/\text{s}$ ) of a spherical precipitate phase in a matrix phase.



See [About the Coarsening Property Model](#) and [Coarsening Property Model Settings](#) for background theory and input parameter details.

### Columnar to Equiaxed Transition (CET)

The **Columnar to Equiaxed Transition** Property Model calculates the fraction of equiaxed grains that correlates with a certain solidification condition, specifically thermal gradient ( $G$ ) and solidification growth rate ( $v$ ).



See [About the Columnar to Equiaxed Transition \(CET\) Property Model](#) and [Columnar to Equiaxed Transition \(CET\) Property Model Settings](#) for background theory and input parameter details.

### ***Crack Susceptibility Coefficient***

The **Crack Susceptibility Coefficient** Property Model calculates the hot tearing tendency during solidification.



See [About the Crack Susceptibility Coefficient Property Model](#) and [Crack Susceptibility Coefficient Property Model Settings](#) for background theory and input parameter details.

### ***Driving Force***

The **Driving Force** Property Model calculates the thermodynamic driving force for a phase.



See [Driving Force Property Model Settings](#) for input parameter details.

### ***Equilibrium***

The **Equilibrium** Property Model calculates the equilibrium for the given conditions. Optionally define additional function definitions.



See [Equilibrium Property Model Settings](#) for input parameter details.

### ***Equilibrium with Freeze-in Temperature***

The **Equilibrium with Freeze-in Temperature** Property Model calculates equilibrium at the freeze-in temperature and evaluates the properties at a different temperature. The assumption is that diffusion and phase transformations are negligible when changing from the freeze-in-temperature and, therefore, that the phase amounts and compositions of phases are kept at all other temperatures.



See [About the Equilibrium with Freeze-in Temperature Property Model](#) and [Equilibrium with Freeze-in Temperature Property Model Settings](#) for background information and input parameter details.

---

## ***Interfacial Energy***

The **Interfacial Energy** Property Model estimates the interfacial energy between a matrix phase and a precipitate phase using thermodynamic data from a CALPHAD database.



See [About the Interfacial Energy Property Model](#) and [Interfacial Energy Property Model Settings](#) for background information and input parameter details.

## ***Liquid and Solidus Temperature***

The **Liquid and Solidus Temperature** Property Model is for doing a common calculation. For example, you can easily use uncertainty calculations, vary one or more conditions, and see how that affects the liquidus and solidus temperatures.



See [Liquid and Solidus Temperature Property Model Settings](#) for input parameter details.

## ***Phase Transition***

The **Phase Transition** Property Model can be used to vary which condition to relax, and then it calculates the point when a new phase may form. It is useful to determine melting temperature, boiling temperature or solubility limits. It returns the phase transformation temperature, or composition, depending on the relaxed condition.



See [Phase Transition Property Model Settings](#) for input parameter details.

## ***Scheil***

The **Scheil** Property Model is used to calculate solidification under the Scheil assumption. The majority of the Model input parameters are pulled from the Scheil Calculator to offer the outputs of the liquidus/solidus/solidification range so you can explore these in the calculation types of the Property Models.



See [Scheil Property Model Settings](#) and [Scheil Advanced Options: Property Models](#).

## ***Spinodal***

The **Spinodal** Property Model calculates the spinodal line.



See [About the Spinodal Property Model](#) and [Spinodal Property Model Settings](#) for background information and input parameter details.

## T-Zero Temperature

The **T-Zero Temperature** Property Model calculates the so-called  $T_0$  line.



See [About the T0 Temperature Property Model](#) and [T0 Temperature Property Model Settings](#) for background information and input parameter details.

## Yield Strength

The **Yield Strength** Property Model calculates yield stress. It returns the calculated yield strength at room temperature for the specified material using the equilibrium values calculated at temperature  $T$  (i.e.  $T$  is the freeze-in temperature). Then you can further define the following contributions to the total yield stress: precipitation strengthening, solid solution, grain boundary strengthening, and constant addition of strength.



See [About the Yield Strength Property Model](#) and [Yield Strength Property Model: Simplified Mode Settings](#) or [Yield Strength Property Model: Advanced Mode Settings](#) for background information and input parameter details.

## Calculation Types

The Property Model Calculator includes these calculation types:

- **Single:** Calculates a single point.
- **One Axis:** Varies a quantity on the X-axis.
- **Grid:** Evaluates two axis variables of the selected quantities in the specified range and number of steps. You can then select to create heat map, contour and 3D plots on the Plot Renderer.
- **Min/Max:** Evaluates the Property Model(s) for all variations of the selected quantities at the given limits. The Mean field is as defined under *Condition Definitions* for the respective quantity. The total minimum and maximum of the Model(s) results are shown in the Event log.
- **Uncertainty:** Evaluates the Property Model(s) where the values of the quantities are sampled from Gaussian distributions. The Mean field is as defined under *Condition Definitions* for the respective quantity. The result is visualized as a histogram or normal probability plot by adding a Plot Render activity.

- **Batch:** Use this to allow for high throughput calculations and read data from a file, such as a spreadsheet. You can include experimental results in the data file, which then can be used to compare to the batch calculation results.



[About the Uncertainty Calculations](#) and [Plot Types](#)

## Examples and Video Tutorials



[Thermo-Calc General Property Models Examples Collection](#)



For tutorials about this feature, go to our [website](#) or browse the Property Model Calculator playlist on our [YouTube channel](#).

## Scheil Advanced Options: Property Models

---

This topic has advanced settings information for the **Show advanced Scheil options** checkbox that is available for a Property Model **Configuration** windows. The main settings are on the **Configuration** tab.



These settings are based on the Scheil Calculator settings with minor differences in wording or application. See [Advanced Options](#).



For theory or background details about Scheil, start by reviewing [Scheil-Gulliver Solidification Calculations](#).

To view all of these settings, click the **Show advanced Scheil options** checkbox.

### Temperature Step during Scheil

Enter a **Temperature step during Scheil** for the Scheil simulation. Temperature step is given in the same unit as the temperature in the **Condition Definitions**.

### Global Minimization

Global minimization is enabled by default i.e. the **Global minimization** checkbox is selected.



It is recommended to keep global minimization on as it makes the calculation more accurate. However, turn it off if you find that the calculation is taking too long for any particular system, and accuracy is not the highest priority.

For this setting, a global equilibrium test is performed at selected intervals when an equilibrium is reached. By default it is at every 10 th Scheil step but this can be changed using the **Global test interval** setting. The global equilibrium test is also done every time the set of stable phases tries to change. This costs more computer time but the calculations are more robust.

## Global Test Interval

Use the **Global test interval** setting to change the interval at which a global equilibrium test is done, which by default is every 10th step as long as there are no changes in the set of stable phases. Every time the set of stable phases tries to change, a global equilibrium test is also done regardless of the test interval value chosen. A global equilibrium test means that the calculated equilibrium state obtained by the ordinary minimization calculation is tested against the Global Minimization Technique, and if the result is found unstable then the full global minimization calculation is performed instead.

## Liquid Phase

The default in the list is the **LIQUID** phase already defined in the chosen database.

## Solid Fraction for Evaluating Liquidus Temperature

Enter a value for the **Solid fraction for evaluating liquidus temperature**. A small fraction larger than zero can be used to avoid small amounts of high temperature stable phases for the evaluation of liquidus temperature.

## Terminate at Fraction of Liquid Phase Amount

Enter a value in the field for **Terminate at fraction of liquid phase amount**.



For **Crack Susceptibility Coefficient** model, the value should not be larger than the setting **Liquid fraction: Smallest for vulnerability**.

## Max No. of Iterations

Enter or choose a value for the **Max no. of iterations**. By default, the program tries 500 iterations before it gives up. As some models give computation times of more than 1 CPU second/iteration, this number is also used to check the CPU time and the calculation stops if 500 CPU seconds/iterations are used.

---

## Required Accuracy

The default **Required accuracy** is  $1.0\text{E}-6$ . This is a relative accuracy, and the program requires that the relative difference in each variable must be lower than this value before it has converged. A larger value normally means fewer iterations but less accurate solutions. The value should be at least one order of magnitude larger than the machine precision.

## Smallest Fraction in a Phase

The default **Smallest fraction** (or **Smallest fraction in a phase** for Property Models) is  $1.0\text{E}-12$ . This is the value assigned to constituents that are unstable. It is normally only in the gas phase you can find such low fractions.

The default value for the smallest site-fractions is  $1\text{E}-12$  for all phases except for IDEAL phase with one sublattice site (such as the GAS mixture phase in many databases) for which the default value is always as  $1\text{E}-30$ .

This default setting is used as a start value in the Scheil Calculator. During simulation, and if there are convergence issues, the value can be decreased.

## Smallest Fraction for alloy composition

The default **Smallest fraction for alloy composition** is  $1.0\text{E}-7$ . A smallest value assigned to alloy mole/mass fraction in case a smaller input value for an element is entered.

## Approximate Driving Force for Metastable Phases

The **Approximate driving force for metastable phases** checkbox is selected by default. Click to clear the checkbox to change the default as required and based on the options described below.

This setting involves the convergence of metastable phases and affects their driving forces. It can also have an effect on when an equilibrium is considered successful.

The default is to allow an equilibrium with metastable phases that have not converged, as long as the stable phases have converged. This is efficient but often causes approximate values of the driving forces for the metastable phases.

If you change the default, it enforces metastable phases to converge. This gives accurate driving forces for metastable phases as well as stable phases. It can however take a slightly longer time, and if metastable phases do not converge it causes the equilibrium calculation to fail.

## About the Coarsening Property Model

**Coarsening** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc.

### [Coarsening Property Model Settings](#)

In the coarsening regime where the driving force is capillarity, the change of mean particle radius evolves according to the equation:

$$\bar{r}^3 - \bar{r}_0^3 = Kt$$

Following Morral and Purdy [1994Mor], the rate constant of precipitating  $\beta$  phase in a multicomponent  $\alpha$  phase is

$$K = \frac{8\sigma V_m^\beta}{9} \left[ \left[ c_i^\beta - c_i^\alpha \right]^T [M]^{-1} \left[ c_k^\beta - c_k^\alpha \right] \right]^{-1}$$

The mobility matrix  $M$  is not the same as the mobilities that can be obtained directly from the Diffusion Module (DICTRA). The  $M$  matrix instead corresponds to the  $L''$  matrix as defined by Andersson and Ågren [1992And].

Using the  $u$ -fractions that is defined as

$$u_k = \frac{x_k}{\sum_{j \in \beta} x_j}$$

where the summation only is performed over the substitutional elements give the final equation for the coarsening rate coefficient.

$$K = \frac{8\sigma V_m^\beta}{9} \left[ \left[ u_i^\beta - u_i^\alpha \right]^T [L''_{jk}^\alpha]^{-1} \left[ u_k^\beta - u_k^\alpha \right] \right]^{-1}$$

The interfacial energy,  $\sigma$ , is calculated using the extended Becker's model (also available as the **Interfacial Energy** Property Model).

### References

[1992And] J.-O. Andersson, J. Ågren, Models for numerical treatment of multicomponent diffusion in simple phases. *J. Appl. Phys.* 72, 1350–1355 (1992).

[1994Mor] J. E. Morral, G. R. Purdy, Particle coarsening in binary and multicomponent alloys.  
Scr. Metall. Mater. 30, 905–908 (1994).

## Coarsening Property Model Settings

**Coarsening** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc and it calculates the coarsening rate coefficient  $K$  ( $\text{m}^3/\text{s}$ ) of a spherical precipitate phase in a matrix phase.

 [About the Coarsening Property Model](#)

 For an example, see [PM\\_G\\_02: Coarsening and Interfacial Energy](#).

The output using this Model is the coarsening rate coefficient [ $\text{m}^3/\text{s}$ ] for the precipitate phase and the interfacial energy [ $\text{J}/\text{m}^2$ ] between matrix and the precipitate phase.



This Model requires that the thermodynamic database used includes descriptions of molar volume of the matrix and precipitates phases as well as a kinetic database with mobilities for the matrix phase.

### Configuration Settings

The settings are found on the Property Model Calculator when **Coarsening** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Matrix Phase

Choose a **Matrix phase**.

---

## Precipitate Phase

Choose a **Precipitate phase** to calculate its coarsening rate. Click the **Add phase**  button to add another Precipitate phase. Click the **Remove phase**  button to remove a phase.

### *Plot Renderer Settings*

 [Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Temperature**
- **Coarsening rate coefficient Ni [m<sup>3</sup>/s]** for the precipitate phase.
- **Interfacial energy Ni [J/m<sup>2</sup>]** for it to be between the matrix and precipitate phase.

## About the Columnar to Equiaxed Transition (CET) Property Model

The **Columnar to Equiaxed Transition** Property Model calculates the fraction of equiaxed grains that correlates with a certain solidification condition, specifically thermal gradient ( $G$ ) and solidification growth rate ( $v$ ) (defined as the migration rate of the interface between liquid and primary solid), so that valuable information on the solidification microstructure can be obtained.

This General Model is available when using the [Property Model Calculator](#) in Thermo-Calc. The Model formulation consists of two essential steps (1) to determine the dendrite tip radius and tip undercooling, and (2) to apply the calculated undercooling to the CET Model. These are discussed below.



Use of this Property Model requires mobility data of liquid for the calculations. For all simulations both a thermodynamic and a mobility (kinetic) database must be selected on the System Definer.



[Columnar to Equiaxed Transition \(CET\) Property Model Settings](#)



[PM\\_G\\_15: Columnar to Equiaxed Transition \(CET\) of a NiAlCr Alloy](#), [PM\\_G\\_16: Columnar to Equiaxed Transition of a CMSX-4 Alloy](#), and [PM\\_G\\_17: Columnar to Equiaxed Transition of an IN718 Alloy](#)

### Determine the Dendrite Tip Radius and Tip Undercooling

The first step determines the dendrite tip radius and tip undercooling, which is strongly dependent on  $v$  while slightly on  $G$ . For this Model, only primary solid is assumed to be stable while other secondary solid phases are ignored. Mullins and Sekerka instability analysis [1964Mul] and marginal stability criterion [1978(a)Lan; 1978(b)Lan; 1978Mül] are used to determine the tip radius. Meanwhile, tip undercooling,  $\Delta T$ , is also calculated due to constitutional, curvature, and solute trapping effects. The multicomponent extension of this framework is achieved based on the approach developed by Hunziker [2001Hun]. It integrates with both thermodynamic and mobility databases so that key thermodynamic and kinetic properties for multicomponent alloy systems, such as liquidus, liquidus slope, solute partitioning coefficients, and diffusivities of liquid, etc., can be obtained directly from the

databases. Solute trapping effects on liquidus, liquidus slope, and solute partitioning are also considered based on the model developed by Aziz [1982Azi] and extended to multicomponent systems.

## Apply the Undercooling to the CET Model

The second step is to apply the calculated undercooling,  $\Delta T$ , to the CET Model. The model developed by Gäumann et al. [2001Gäu] is used, which extended Hunt's model [1984Hun] to include high solidification rate. A slight modification [2018Hai] enables elimination of an adjustment parameter by directly using  $\Delta T$ . The model correlates  $G$ ,  $\Delta T$  (mostly in terms of  $v$ ), and the fraction of equiaxed grains,  $\phi$ , as

$$G = \frac{1}{n+1} \sqrt[3]{\frac{-4\pi N_0}{3 \ln(1-\phi)}} \Delta T \left(1 - \frac{\Delta T_n^{n+1}}{\Delta T^{n+1}}\right)$$

Where three materials related parameters are introduced:

- $N_0$ : number of nucleation sites per unit volume, in the unit of  $1/\text{m}^3$
- $\Delta T_n$ : nucleation undercooling, in the unit of K
- $n$ : exponent

## References

- [1964Mul] W. W. Mullins, R. F. Sekerka, Stability of a Planar Interface During Solidification of a Dilute Binary Alloy. *J. Appl. Phys.* 35, 444–451 (1964).
- [1978(a)Lan] J. S. Langer, H. Müller-Krumbhaar, Theory of dendritic growth—I. Elements of a stability analysis. *Acta Metall.* 26, 1681–1687 (1978).
- [1978(b)Lan] J. S. Langer, H. Müller-Krumbhaar, Theory of dendritic growth—II. Instabilities in the limit of vanishing surface tension. *Acta Metall.* 26, 1689–1695 (1978).
- [1978Mül] H. Müller-Krumbhaar, J. S. Langer, Theory of dendritic growth—III. Effects of surface tension. *Acta Metall.* 26, 1697–1708 (1978).
- [1982Azi] M. J. Aziz, Model for solute redistribution during rapid solidification. *J. Appl. Phys.* 53, 1158–1168 (1982).
- [1984Hun] J. D. Hunt, Steady state columnar and equiaxed growth of dendrites and eutectic. *Mater. Sci. Eng.* 65, 75–83 (1984).
- [2001Gäu] M. Gäumann, C. Bezençon, P. Canalis, W. Kurz, Single-crystal laser deposition of superalloys: processing–microstructure maps. *Acta Mater.* 49, 1051–1062 (2001).

[2001Hun] O. Hunziker, Theory of plane front and dendritic growth in multicomponent alloys. Acta Mater. 49, 4191–4203 (2001).

[2018Hai] M. Haines, A. Plotkowski, C. L. Frederick, E. J. Schwalbach, S. S. Babu, A sensitivity analysis of the columnar-to-equiaxed transition for Ni-based superalloys in electron beam additive manufacturing. Comput. Mater. Sci. 155, 340–349 (2018).

## Columnar to Equiaxed Transition (CET) Property Model Settings

The **Columnar to Equiaxed Transition** Property Model calculates the fraction of equiaxed grains that correlates with a certain solidification condition, specifically thermal gradient ( $G$ ) and solidification growth rate ( $v$ ).

This General Model is available when using the [Property Model Calculator](#) in Thermo-Calc.



[About the Columnar to Equiaxed Transition \(CET\) Property Model](#)



[PM\\_G\\_15: Columnar to Equiaxed Transition \(CET\) of a NiAlCr Alloy](#), [PM\\_G\\_16: Columnar to Equiaxed Transition of a CMSX-4 Alloy](#), and [PM\\_G\\_17: Columnar to Equiaxed Transition of an IN718 Alloy](#)

### System Definer Configuration Settings



Use of this Property Model requires mobility data of liquid for the calculations. For all simulations both a thermodynamic and a mobility (kinetic) database must be selected on the System Definer.

### Property Model Calculator Configuration Settings

The settings are found on the [Property Model Calculator](#) when the **Columnar to Equiaxed Transition** Model is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

## Primary Phase

Select the **Primary phase** from the list. This is the primary solid phase to form the dendrite.

## Interfacial Energy

Enter a value for the **Interfacial energy [J/m<sup>2</sup>]**, which is between the liquid and primary phase. The default is 0.25 J/m<sup>2</sup>.

## Number of Nucleation Sites

Enter a value for the **Number of nucleation sites [/m<sup>3</sup>]** for equiaxed grains per cubic meter. The default is 2.0E15 / m<sup>3</sup>.

## Nucleation Undercooling

Enter a value for the **Nucleation undercooling [K]**, which is the critical undercooling for nucleation of equiaxed grains. The default is 2.5 K.

## Equiaxed Exponent

Enter a value for the **Equiaxed exponent**. The Equiaxed exponent is used in the equation that calculates the fraction of equiaxed grains. This is usually a value between 3 and 5.

## Solve For, Temperature Gradient, and Equiaxed Fractions

Select an option from the **Solve for** list (**Equiaxed fraction** or **Thermal gradient**) and then enter the required secondary information as indicated.

- **Equiaxed fraction:** Solve for the equiaxed fraction given the thermal gradient.

When **Equiaxed fraction** is selected, then enter a value for the **Temperature gradient**, which is the temperature gradient of the liquid ahead of the dendrite tip. The default is 1000 K/m.

- **Thermal gradient** [K/m]: Solve for the thermal gradient given the equiaxed fraction (s).

When **Thermal gradient** is selected, then enter a value for the **Equiaxed fractions**. This finds the thermal gradient for the given equiaxed fractions. The values entered should be between 0.001 and 0.99. If more than one fraction, separate the values with a space. The default is 0.0066 0.49 0.99.

## Growth Rate

Select a **Growth rate** from the list then enter a value in the field.

- If **v** is selected, then enter a value for the solidification growth rate, **v**. The default is 0.1 m/s.
- If **log10(v)** is selected, then enter a value for the logarithm of growth rate, **log10(v)**. The default is -1.0 m/s.

## Solute Trapping

Select the **Solute trapping** checkbox to enable solute trapping at the tip interface. By default this is not selected.

## Plot Renderer Settings

 [Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **log10(v) [m/s]**
- **Liquidus**
- **Fraction of equiaxed grains**
- **Tip radius**
- **Thermal gradient**
- **v [m/s]**
- **Dendrite tip undercooling [K]**

---

## About the Crack Susceptibility Coefficient Property Model

---

The **Crack Susceptibility Coefficient** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc. It is used to calculate the hot tearing tendency during solidification. The model is based on the publication by Yan and Lin [2006Yan].

Hot tearing is one of the most common and serious defects encountered during the casting of, for example, aluminum alloys. In general, it is defined by the formation of a macroscopic fissure in a casting as a result of stress and the associated strain, generated during cooling, at a temperature above the non-equilibrium solidus.

Crack susceptibility is also referred to as solidification cracking, hot cracking, hot shortness, supersolidus cracking, and shrinkage brittleness [2006Yan].

The Crack Susceptibility Coefficient (CSC) is calculated using Scheil results with three different models; Clyne and Davies [1981Cly; 2006Yan], Kou [1988Kou], and Easton [2014Eas].



[PM\\_G\\_07: Hot Crack Susceptibility](#)

### Input Parameters, Scheil Settings, and CSC Model Selection



The input parameters are entered on the **Configuration** window for the Property Model Calculator and described in [Crack Susceptibility Coefficient Property Model Settings](#).

There are these models to choose from as the Crack Susceptibility Coefficient (CSC) Model: **Clyne and Davies, Kou, or Easton.**

There are also several Scheil settings, including advanced options to choose from. Some additional theory and background is included below to supplement the settings input choices.

## Clyne and Davis CSC Model Theory and Background



The original model assumptions are from Clyne and Davis [1981 Clyne]. Later, the experimental hot cracking susceptibility was collected from several sources by Yan and Lin [2006 Yan] and given as a probability 0-100%. The CSC calculated with the current model cannot be compared by absolute values with the experimental number; instead it is best to compare the trends, to see if it is increasing or decreasing. The experimental values are therefore given with relative coordinates and do not map directly to the calculated plot axes.

The following information describes the background when **Clyne and Davies** is chosen as the **CSC Model**.

The **Crack Susceptibility Coefficient (CSC)** was originally proposed by Clyne and Davies [1981Cly] to describe the effects of alloy composition on hot tearing.

The current model is based on the publication by Yan and Lin [2006Yan]. In Thermo-Calc, the CSC is calculated using Scheil and uses assumptions of cooling rates under the thermal conditions that the heat flow is proportional to  $\frac{1}{\sqrt{time}}$ .

The CSC is defined as  $CSC = \frac{tV}{tR}$  where:

- $tV$  is the time during solidification where the casting is vulnerable to cracking, and
- $tR$  is the time available for the stress relief process.

The values for  $tV$  and  $tR$  are given by the thermal condition, the Scheil results, and the input parameters for the liquid fraction (defined on the **Configuration** window):

- Start of relaxation
- Transition to vulnerability
- Smallest for vulnerability

These liquid fraction parameters are elaborated on next using the results from a simulation using the TCS Al-based Alloy Database (TCAL) and the alloy Al-0.9Cu-10Mg-0.9Ni-12.2Si mass%.

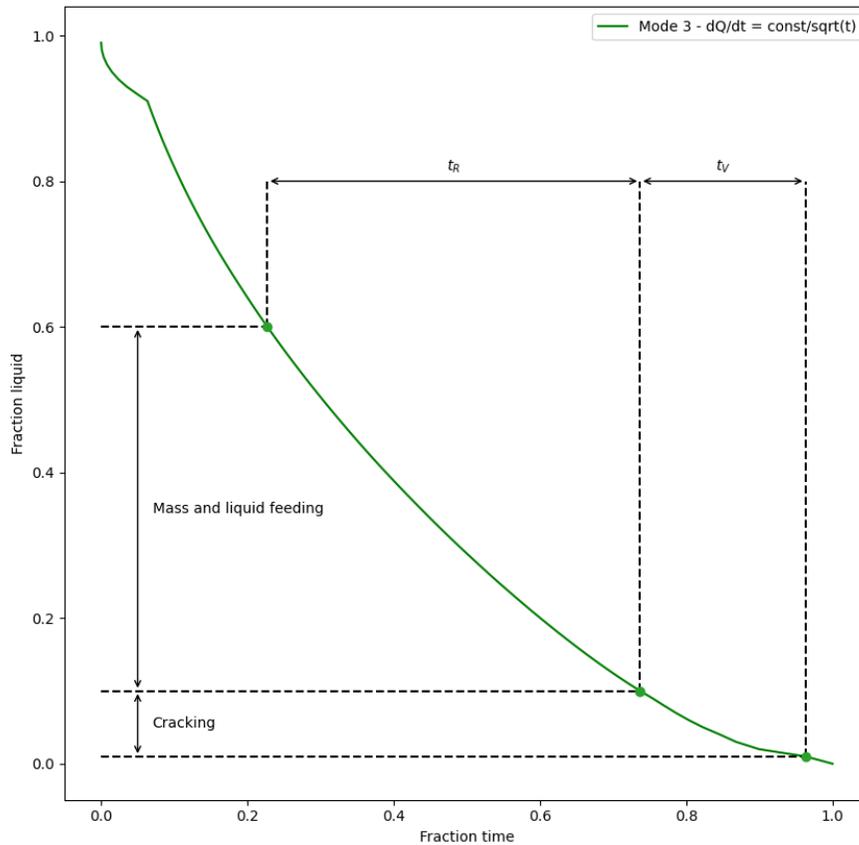


Figure 12: The results of the thermal condition in Mode 3, fraction of liquid, including the addition of lines and arrows to the plot for clarity. See the text for more detail.

### Liquid Fraction Parameters

The **Crack Susceptibility Coefficient** Model is based on the concept of the existence of critical time periods during the solidification process when the structure is most vulnerable to cracking.

Clyne and Davies concluded that the mass and liquid feeding readily occurs at liquid volume fractions between about 0.6 and 0.1, so the time spent in these ranges was defined as time-relaxation ( $t_R$ ). At very low volume fractions of liquid, the material is too strong to crack. The authors chose a fraction of liquid between 0.1 and 0.01 as the vulnerable regime, and the time spent defined as  $t_V$ . See [Figure 12](#) for the results from a simulation using the TCS Al-based Alloy Database (TCAL) and the alloy Al-0.9Cu-10Mg-0.9Ni-12.2Si mass%.

The Scheil simulation takes temperature steps and does not regard time at all. See [Figure 13](#) for the typical Scheil results of amount of solid phases as function of temperature. The simulation is made with the TCS Al-based Alloy Database (TCAL) and the alloy Al-0.9Cu-10Mg-0.9Ni-12.2Si mass%.

A thermal condition is assumed to connect the amount of liquid fraction to fraction of time.

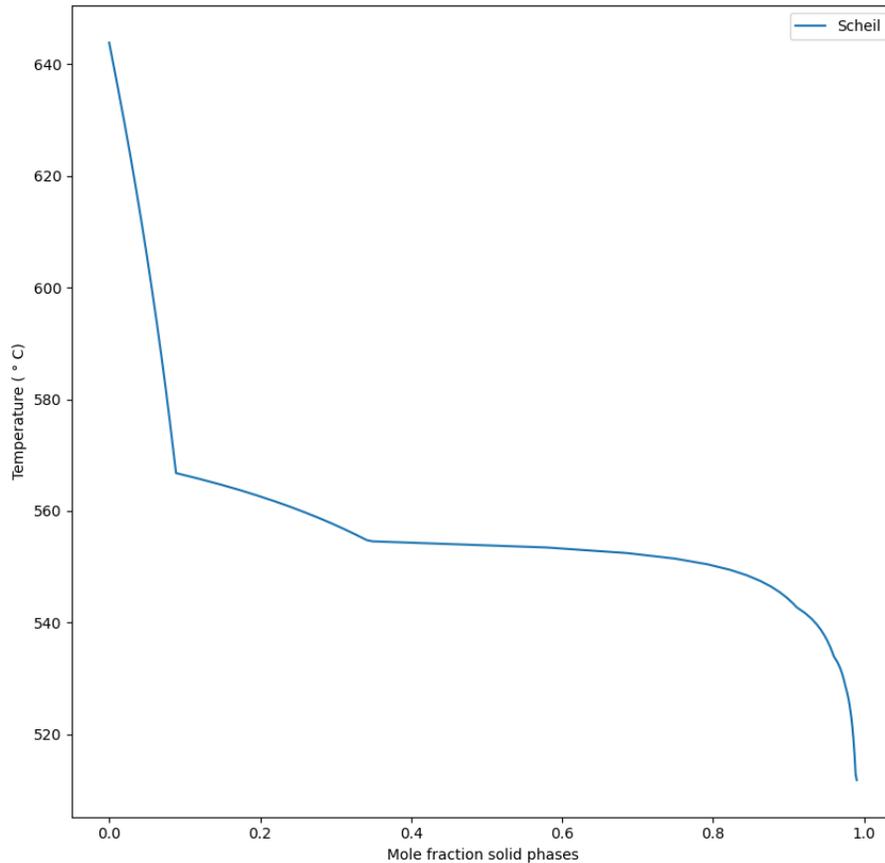


Figure 13: A Scheil simulation result of the mole fraction solid phases vs temperature (°C). See the text for more detail.

Following Clyne and Davies' approach, the cooling rates were then estimated under three different thermal conditions:

- Mode 1 with a constant cooling rate,  $dT/dt = \text{constant}$
- Mode 2 with a constant heat flow,  $dQ/dt = \text{constant}$ , and
- Mode 3 with a heat flow proportional to  $1/\sqrt{\text{time}}$

Then:

- Liquid fraction: Start of relaxation
- Liquid fraction: Transition to vulnerability
- Liquid fraction: Smallest for vulnerability

The crack index can then be obtained from the Scheil simulation results: fraction liquid vs T curve and the thermal condition.



1-fraction solid phases = fraction liquid phase.

[Figure 14](#) can be thought of as the 'master curve' for the connection between fraction of liquid and fraction of time. The limits of the amount of liquid 0.01, 0.1, and 0.6 and the corresponding fraction time are shown for each thermal condition. In [Figure 12](#), which is a less cluttered plot, the time spent in the relaxation and in the vulnerable regions are marked with lines and arrows for a single thermal mode.

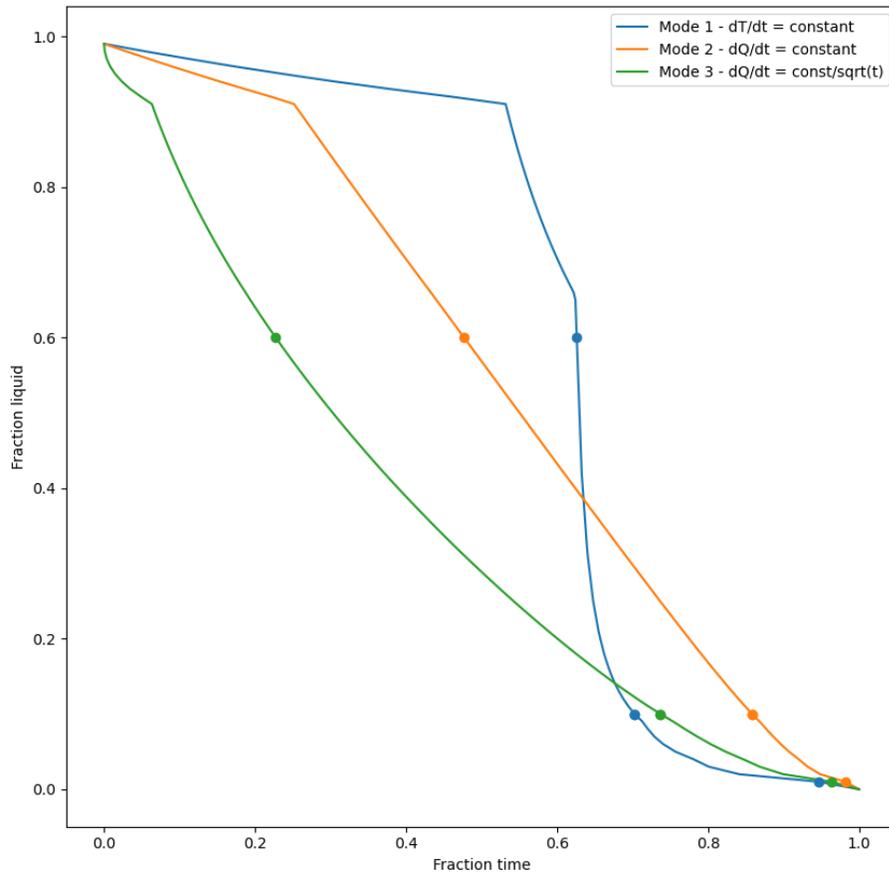


Figure 14: The results of the thermal condition in Modes 1-3, the fraction of time vs the fraction of liquid. See the text for more detail.

## Kou CSC Model

When **Kou** is chosen as the **CSC Model** [1988Kou], the cracking susceptibility coefficient is defined as the absolute value of the derivative of the  $T$  vs  $f_s^{0.5}$  curve at the end of the solidification, where

- $T$  = Temperature
- $f_s$  = fraction solid

## Easton CSC Model

When **Easton** is chosen as the **CSC Model** [2014Eas], the cracking susceptibility coefficient is defined as the integral of  $f_s(T)$  between  $T_0$  and  $T_{co}$  where

- $f_s$  = fraction solid
- $T_0$  = temperature for the solid fraction of a coherent dendrite structure.
- $T_{co}$  = The temperature for the solid fraction of coalescence of dendrite structure.  
The user inputs are the solid fraction for coherency and coalescence.

## **References**

[1981Cly] T. W. Clyne, G. J. Davies, The Influence of Composition on Solidification Cracking Susceptibility in Binary Alloy Systems. Br. Foundrym. 74, 65–73 (1981).

[1988Kou] S. Kou, A criterion for cracking during solidification. Acta Mater. 88, 366–374 (2015).

[2006Yan] X. Yan, J. C. Lin, Prediction of hot tearing tendency for multicomponent aluminum alloys. Metall. Mater. Trans. B. 37, 913–918 (2006).

[2014Eas] M. A. Easton, M. A. Gibson, S. Zhu, T. B. Abbott, An A Priori Hot-Tearing Indicator Applied to Die-Cast Magnesium-Rare Earth Alloys. Metall. Mater. Trans. A. 45, 3586–3595 (2014).

## Crack Susceptibility Coefficient Property Model Settings

The **Crack Susceptibility Coefficient** Property Model calculates the hot tearing tendency during solidification.

This General Model is available when using the [Property Model Calculator](#) in Thermo-Calc.

When the default **Clyne and Davies** is selected as the **CSC Model**, the calculations use Scheil and assumptions of cooling rates under three different thermal conditions:

- Constant cooling rate,  $dT/dt = \text{constant}$
- Constant heat flow,  $dQ/dt = \text{constant}$
- Heat flow proportional to  $1/\sqrt{\text{time}}$



For details about this more advanced model, see [About the Crack Susceptibility Coefficient Property Model](#).



The advanced Scheil settings are described with [Scheil Advanced Options: Property Models](#).



For an example, see [PM\\_G\\_07: Hot Crack Susceptibility](#).

### Configuration Settings

These settings are found on the Property Model Calculator when **Crack Susceptibility Coefficient** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### SCHEIL PARAMETERS



These parameters are the same as those used for the Scheil Calculator although some setting names are slightly different.

## Scheil Start Temperature

Enter a **Scheil start temperature** for the Scheil simulation that is higher than the liquidus temperature of the alloy. Is given in the same unit as the temperature in the **Condition Definitions**.

## Scheil Calculation Type

Click to select a **Calculation type**: **Classic Scheil**, **Scheil with back diffusion in primary phase**, or **Scheil with solute trapping**. Then based on this selection, additional settings are made available as indicated.



See [Additional Scheil Calculation Type Settings](#)



The back diffusion and solute trapping options require the selection of a mobility database on the System Definer.

## Fast Diffusers

Enter the element/component to make as **Fast diffusers**. Include a space between each entry, C N, for example, is the default entered for Carbon and Nitrogen to be fast diffusers.

## Allow Delta Ferrite to Austenite Transition in Steel

This is available when **Classic Scheil** or **Scheil with back diffusion in primary phase** is selected.

Select the **Allow delta ferrite to austenite transition in steel** checkbox to enable BCC to FCC transition.



This feature should only be used for such steels for which it is reasonable to assume infinitely fast diffusion (equilibrium conditions) when delta ferrite is present. When the **Allow delta ferrite to austenite transition in steel** checkbox is selected in Graphical Mode or the DELTA\_FERRITE\_AUSTENITE\_TRANSITION command used in Console Mode, this essentially assumes that diffusion can be considered (infinitely) fast when delta ferrite is present as a solid phase, i.e. assumes an equilibrium condition. After delta ferrite is no longer stable, the usual Scheil assumptions for the selected calculation type are in effect. This setting is therefore not suitable for steels where you DO NOT have a complete BCC to FCC transition during solidification. This setting is also NOT available in combination with solute trapping.

## Show Advanced Scheil Options

Click the **Show advanced Scheil options** checkbox to enter several settings related to Scheil calculations. See [Scheil Advanced Options: Property Models](#).

## CSC MODEL SELECTION

### CSC Model

Choose a **CSC Model**: **Clyne and Davies** (the default), **Kou**, or **Easton**.

Then continue with these additional settings:

- For **Clyne and Davies** go to [Thermal Mode and Liquid Fraction Parameters \(Clyne and Davies\)](#).
- For **Easton**, go to [Solid Fraction for Coherency and Solid Fraction for Coalescence \(Easton\)](#)



For details and references about these models, see [About the Crack Susceptibility Coefficient Property Model](#).

## ADDITIONAL SCHEIL CALCULATION TYPE SETTINGS

---

## Cooling Rate (Back Diffusion)

This field is available when **Scheil with back diffusion in primary phase** is selected.

Specify the **Cooling rate** in Kelvin per second (K/s). An increased value moves the result from equilibrium toward a Scheil-Gulliver calculation.

## Secondary Dendrite Arm Spacing (Back Diffusion)

This field is available when **Scheil with back diffusion in primary phase** is selected.

The **Secondary dendrite arm spacing** is the distance in meters between two secondary dendrite arms. It is calculated from the cooling rate as  $c \cdot (\text{cooling rate})^{-n}$  where  $c$  and  $n$  are entered in these fields.

Enter values for **Secondary dendrite arm spacing  $c$**  and **Secondary dendrite arm spacing  $n$** .

## Primary Phase (Back Diffusion and Solute Trapping)

This field is available when **Scheil with back diffusion in primary phase** or **Scheil with solute trapping** is selected.

The primary solidified phase is the phase where the back diffusion or solute trapping takes place.

If **Automatic** (Scheil Calculator) or **Any phase** (Property Models) is selected (or kept as the default), the program tries to find the phase which gives the most back diffusion or solute trapping.

To override this setting, choose a specific primary phase from the list:

- For back diffusion, only phases with diffusion data can be used as primary phases.
- For solute trapping, only phases that dissolve all elements in the system can be used as primary phases.

## Scanning Speed and $\alpha$ (Solute Trapping)

These fields are available when **Scheil with solute trapping** is selected.

As needed, enter **Scanning speed** and  **$\alpha$**  to define the **Calculated solidification speed**.



On the Scheil Calculator, the **Calculated solidification speed** is shown. For Property Models, below is how the values are calculated from the **Scanning speed** and  $\alpha$  values entered.

The **Calculated Solidification Speed** is  $V_s = V_{\text{scanning}} * \cos(\alpha)$  m/s, where  $V_{\text{scanning}}$  is the user specified **Scanning speed** in m/s and  $\alpha$  is the alpha angle,  $\alpha$ , between the solid/liquid boundary and scanning direction.

## Thermal Mode and Liquid Fraction Parameters (Clyne and Davies)

These settings are available when **Clyne and Davies** is selected as the **CSC Model**.

### Thermal Mode

Select a **Thermal mode** for the solidification simulation:

- **Constant cooling rate,  $dT/dt = \text{constant}$**
- **Constant heat flow,  $dQ/dt = \text{constant}$**
- **Heat flow proportional to  $1/\text{sqrt}(\text{time})$**  (i.e. heat flow is proportional to  $\frac{1}{\sqrt{\text{time}}}$  )

### Liquid Fraction for Start of Relaxation

Enter a value for the **Liquid fraction for start of relaxation**.

### Liquid Fraction Transition to Vulnerability

Enter a value for the **Liquid fraction transition to vulnerability**.

### Liquid Fraction Smallest to Vulnerability

Enter a value for the **Liquid fraction smallest to vulnerability**.

## Solid Fraction for Coherency and Solid Fraction for Coalescence (Easton)

These settings are available when **Easton** is selected as the **CSC Model**.

## Solid Fraction for Coherency

Enter a value in the **Solid fraction for coherency** field, which is the solid fraction for a coherent dendrite structure. The default is 0.7.

## Solid Fraction for Coalescence

Enter a value in the **Solid fraction for coalescence** field, which is the solid fraction for coalescence of a dendrite structure. The default is 0.98.

## *Plot Renderer Settings*

 [Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Composition**
- **Temperature**
- **Crack Susceptibility coefficient.** The tendency for the material to crack during solidification.
- **Liquidus temperature**
- **Solidus temperature**
- **Freezing range [K].** The solidification freezing range (Liquidus - Solidus) given in Kelvin.

## Driving Force Property Model Settings

**Driving Force** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc and it calculates the thermodynamic driving force for a phase divided by RT.



For an example, see [PM\\_G\\_03: Driving Force and Interfacial Energy](#).

### Configuration Settings

The settings are found on the Property Model Calculator when **Driving force** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Matrix Phase

Choose a **Matrix phase**, the single phase region to start in. Select **Any phase** to use the phases that take part of the initial equilibrium or choose a specific phase.

### Precipitate Phase

Choose a **Precipitate phase** to calculate its driving force.

### Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Composition**
- **Interfacial energy Ni [J/m<sup>2</sup>]**
- **Driving force per mole** and then choose a unit: **Joule, Calorie, Electron volt, Gigajoule, Watt hour, or Kilowatt hour.**
- **Normalized driving force**

## About the Equilibrium with Freeze-in Temperature Property Model

---

The **Equilibrium with Freeze-in Temperature** Property Model calculates equilibrium at the freeze-in temperature and evaluates the properties at a different temperature.

The assumption is that diffusion and phase transformations are negligible when changing from the freeze-in-temperature and, therefore, that the phase amounts and compositions of phases are kept at the evaluation temperatures.

Typically, the freeze-in occurs during cooling where equilibrium can be assumed above the freeze-in temperature. This is the default setting and the **Equilibrium above freeze-in temperature** checkbox is automatically selected for this model. Change this default setting (i.e. click to clear the checkbox) when the frozen structure should be evaluated at a higher temperature than the freeze-in temperature.

The electrical resistivity due to phase interface scattering is evaluated as the scattering constant times sum of the interaction between the volume fraction of all the phases. The default value for the constant is  $4.0e-8 \Omega m$ . The contribution to thermal conductivity is assumed to be related to that of electrical resistivity, following the Wiedemann-Franz law.

A selection of homogenization functions is available for the evaluation of thermal- and electric-properties. Thermal and electric properties depend on the microstructure and the geometrical representation of phases in the microstructure can be linked to different homogenization functions. The homogenization function is applied on thermal- and electrical-resistivity. The electric conductivity, thermal conductivity, and thermal diffusivity are evaluated from the resistivities.



For examples, see [PM\\_G\\_10: Freeze-in Thermal Conductivity](#) and [PM\\_G\\_11: Freeze-in Electrical Resistivity](#).



[Equilibrium with Freeze-in Temperature Property Model Settings](#)



This Property Model uses thermophysical properties that are included with some Thermo-Calc databases such as the TCS AI-based Alloy Database (TCAL), which is used in the two examples included with your installation. For more information about the availability and the theory, see [About Thermophysical Properties Data Parameters](#), [Electrical Resistivity Model](#), and [Thermal Conductivity Model](#).

## Equilibrium with Freeze-in Temperature Property Model Settings

The **Equilibrium with Freeze-in Temperature** Property Model calculates equilibrium at the freeze-in temperature and evaluates the properties at a different temperature. The assumption is that diffusion and phase transformations are negligible when changing from the freeze-in-temperature and, therefore, that the phase amounts and compositions of phases are kept at all other temperatures.



[About the Equilibrium with Freeze-in Temperature Property Model](#)



For examples, see [PM\\_G\\_10: Freeze-in Thermal Conductivity](#) and [PM\\_G\\_11: Freeze-in Electrical Resistivity](#).

### Configuration Settings

The settings are found on the [Property Model Calculator](#) when the **Equilibrium with Freeze-in Temperature** Model is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Freeze-in Temperature

Enter a **Freeze-in temperature** value in the field. The unit matches those selected under *Condition Definitions*. This is the temperature where the equilibrium is calculated.

The default is 623.15 K (350 °C).

### Equilibrium Above Freeze-in Temperature

The **Equilibrium above freeze-in temperature** checkbox is selected by default and it calculates equilibrium above freeze-in temperature. Click to clear the checkbox when evaluating the frozen-in structure at higher temperatures than the freeze-in temperature.

---

## Evaluate for Single Phase Only

Select the **Evaluate for a single phase only** checkbox to evaluate the properties of a specific phase or deselect to evaluate the properties for the entire system.

### Phase for Evaluation

This setting is available when the **Evaluate for a single phase only** checkbox is selected.

From the **Phase for evaluation** list, select any available phase or **None**.

### Equilibrium Minimization Strategy

Select an **Equilibrium minimization strategy**. The default uses the **Local minimization preferred** option.

The **Minimization Strategy** setting is used to ensure that the most stable minimum under the specified conditions is computed.

With either the *Global test preferred* or *Local minimization preferred* setting, the program cycles through options before it gives up:

- For **Global test preferred** the minimization starts with a global test and if that fails it runs full global minimization.
- For **Local minimization preferred** the minimization starts with a local minimization, in case of failure it tries a global test and finally a full global minimization.

If you choose **Local minimization only** or **Global minimization only**, the program just tries the one setting and gives up if it fails.



For general information about global minimization, see the topic related to the Console Mode command, [GLOBAL\\_MINIMIZATION](#).

### Homogenization Function

The **Homogenization function** is used for the evaluation of the systems thermal and electric properties. The function is applied on thermal- and electrical-resistivity. The electric conductivity, thermal conductivity, and thermal diffusivity are evaluated from the resistivities.

---

The geometrical interpretation of the Hashin-Shtrikman bounds are concentric spherical shells of each phase.

- **Rule of mixtures (upper Wiener bound):** the geometrical interpretation are continuous layers of each phase parallel with the direction of evaluation of the property.
- **Inverse rule of mixtures (lower Wiener bound):** The geometrical interpretation are continuous layers of each phase orthogonal to the direction of evaluation of the property.
- **General lower Hashin-Shtrikman bound:** The outermost shell consists of the phase with the most sluggish property.
- **General upper Hashin-Shtrikman bound:** The innermost shell consists of the phase with the most sluggish property.
- **Hashin-Shtrikman bound with majority phase as matrix phase:** The outermost shell consists of the phase with the highest local volume fraction.

## Account for Phase Interface Scattering

The **Account for phase interface scattering** checkbox is selected by default.

The electrical resistivity due to phase interface scattering is evaluated as the scattering constant times sum of the interaction between the volume fraction of all the phases. The default value for the constant is  $4.0e-8 \Omega m$ . The contribution to thermal conductivity is assumed to be related to that of electrical resistivity, following the Wiedemann-Franz law.

## Phase Interface Scattering Constant

This setting is available when the **Account for phase interface scattering** checkbox is selected.

The **Phase interface scattering constant** default value is  $4.0e-8 \Omega m$ .

## Set Reference Temperature for Technical CTE

This setting is available when the **Account for phase interface scattering** checkbox is selected.

In the field, enter the **Reference temperature for technical CTE**, where CTE is the coefficient of thermal expansion. Typically room temperature is used as the default 293.15 K (20 °C).

## Define User Functions

Select the **Define user functions** checkbox to enter up to two **User-defined functions** using the Console Mode syntax.



For an example of user-defined functions, see [T\\_07: User-Defined Functions](#).

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Temperature**
- **Electric resistivity (ohm m) ( $\Omega\text{m}$ )**
- **Electric conductivity (S/m)**
- **Thermal conductivity (W/(mK))**
- **Thermal resistivity (mK/W)**
- **Thermal diffusivity ( $\text{m}^2/\text{s}$ )**
- **Heat capacity (J/mol K)**
- **Volume ( $\text{m}^3/\text{mol}$ )**
- **Density ( $\text{g}/\text{cm}^3$ )**
- **Linear CTE (technical) (1/K):** Fractional change of length per unit temperature change. Calculated as  $(L1-L0)/(T1-T0)/L0$ , where  $L0$  is the length at a reference temperature  $T0$ .  $T0$  is by default equal to room temperature. Assumed Isotropic material.
- **Linear CTE (physical) (1/K):** Fractional change of length per unit temperature change. Assumed Isotropic material.
- **Volumetric CTE (physical) (1/K):** Fractional change of volume per unit temperature change.
- **User-defined function and 2nd user-defined function:** Available when the **Define user functions** checkbox is selected and the functions are defined.

## Equilibrium Property Model Settings

---

**Equilibrium** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc and it calculates the equilibrium for the given conditions.



When you use this Model with the **Property Model Calculator**, it is the same calculation result as when you are using an **Equilibrium Calculator** with the **Single equilibrium** calculation type.

### Configuration Settings

The settings are found on the Property Model Calculator when **Equilibrium** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Equilibrium Minimization Strategy

Select an **Equilibrium minimization strategy**. The default uses the **Global test preferred** option. For this Property Model, and for either Global option, also enter the **Max number of global gridpoints**, where the default is 2000.

The **Minimization Strategy** setting is used to ensure that the most stable minimum under the specified conditions is computed.

With either the *Global test preferred* or *Local minimization preferred* setting, the program cycles through options before it gives up:

- For **Global test preferred** the minimization starts with a global test and if that fails it runs full global minimization.
- For **Local minimization preferred** the minimization starts with a local minimization, in case of failure it tries a global test and finally a full global minimization.

If you choose **Local minimization only** or **Global minimization only**, the program just tries the one setting and gives up if it fails.



For general information about global minimization, see the topic related to the Console Mode command, [GLOBAL\\_MINIMIZATION](#).

## Quantity Definitions

For *Quantity Definitions* define **Q1**. Click the **Add**  button to define additional items and the **Remove**  button to delete items.

Then from the lists, choose the variable for each quantity to define. The quantities are called **Q1, Q2, Q3** and so on.

All quantities that normally can be obtained from an equilibrium calculation can also be obtained when plotting the results.

## Function Definitions

For *Function Definitions*, define **Function 1**. Click the **Add**  button to define additional items and the **Remove**  button to delete items. In the left field edit the default name of each function and enter the function itself in the right field after the equal sign.



To learn about using variables and functions, in Graphical Mode, see [T\\_07: User-Defined Functions](#) and in Console Mode, see [example 44](#).



You can also use Console Mode syntax when entering the function. See [Parameters, Functions, and Variables](#) for details.

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

All quantities that normally can be obtained from an equilibrium calculation can also be obtained when plotting the results as well as the defined functions.



[Conditions Tab Settings](#)

---

## About the Interfacial Energy Property Model

---

**Interfacial Energy** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc.

 [Interfacial Energy Property Model Settings](#)

Interfacial energy is an important parameter used in precipitation simulations to calculate the rates of nucleation, growth/dissolution, and coarsening. The value of interfacial energy can vary dramatically (usually between 0.01 to 2.0 J/m<sup>2</sup>).

The extended Becker's model functions to estimate coherent interfacial energy by using thermodynamic data from existing CALPHAD thermodynamic databases:

$$\sigma_c = \frac{n_s z_s}{N_A z_l} \Delta E_s$$

where  $\sigma_c$  is the coherent interfacial energy,  $n_s$  is the number of atoms per unit area at the interface,  $z_s$  is the number of cross bonds per atom at the interface,  $z_l$  is the coordination number of an atom within the bulk crystal lattice, and  $\Delta E_s$  is the energy of solution in a multicomponent system involving the two phases being considered [1938Bec].

### Reference

[1938Bec] R. Becker, Die Keimbildung bei der Ausscheidung in metallischen Mischkristallen. Ann. Phys. 424, 128–140 (1938).

## Interfacial Energy Property Model Settings

**Interfacial Energy** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc and is used to estimate the interfacial energy between a matrix phase and a precipitate phase using thermodynamic data from a CALPHAD database. The approximation model is based on Becker's bond energy approach and is also available in Precipitation Module (TC-PRISMA).



In the actual calculation, the coefficients taken distinguish between only bcc and fcc structure for the (110) and (111) lattice plane, respectively.



[About the Interfacial Energy Property Model](#)



For an example, see [PM\\_G\\_02: Coarsening and Interfacial Energy](#).

For other planes and/or other phases, the estimate is approximate. In most situations, it is recommended to perform calibrations against some experimental data because the interfacial energy is dependent on so many factors that are ignored in the estimation, for example entropy, incoherency, orientation, and curvature, etc. Uncertainty factors must also be considered as thermodynamic data in CALPHAD databases are not always accurate.

### Configuration Settings

The settings are found on the Property Model Calculator when **Interfacial energy** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Matrix Phase

Choose a **Matrix phase** to calculate the interfacial energy [J/m<sup>2</sup>].

## Precipitate Phase

Choose a **Precipitate phase** to calculate the interfacial energy [J/m<sup>2</sup>].

### *Plot Renderer Settings*

 [Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Temperature**
- **Interfacial energy Ni [J/m<sup>2</sup>]** for it to be between the matrix and precipitate phase.

# Liquid and Solidus Temperature Property Model Settings

---

**Liquidus and solidus temperature** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc. It calculates the equilibrium solidification range in terms of temperature. If only the liquidus temperature is required, the calculation times are reduced.

## Configuration Settings

The settings are found on the Property Model Calculator when **Liquidus and solidus temperature** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

## Only Calculate Liquidus Temperature

Select the **Only calculate liquidus temperature** checkbox as required. When selected, this improves the calculation speed but the solidus is not calculated.

## Liquid Phase

Choose the **Liquid phase** of the system. This is a fallback choice. Normally this should be handled automatically, even if multiple liquid phases are present.

## Upper Temperature Search Limit

Enter an **Upper temperature search limit** to calculate for a temperature above liquidus for any of the considered compositions. This needs to be in the fully liquid state.

## Max. Number of Iterations

Enter a **Max. number of iterations**, which is the maximum number of phase transformation iterations performed. The default is 10.

## Global Minimization (Liquidus and Solidus)

Select the **Global minimization for liquidus** and/or **Global minimization for solidus** checkboxes to find all phases present in the equilibria.



Selecting one or both checkboxes can significantly slow down the calculations but ultimately it produces more reliable results.



Global minimization recommendation: First try to calculate without global minimization, then if inaccuracies occur, switch global minimization on. Problems mostly occur if many phases are present at a given temperature.

## Suspend Gas Phase

Select the **Suspend gas phase** checkbox as required. The occurrence of a gas phase may mislead the liquidus temperature determination. If the presence of the gas phase is absolutely required, you can activate it and then choose a user-defined phase.

## Gas Phase

Choose a **Gas phase** from the list.

## Print Results in Event Log

Select the **Print results in Event Log** checkbox to display/include all results in the Event Log window. This includes the **First solidified phase**, **Liquidus (K)**, **Solidus (K)**, and **Composition** information. Choose the **Output type**: **Single row** or **Full list for each iteration**.

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Composition**
- **Temperature**
- **Liquidus temperature**
- **Solidus temperature**

## Phase Transition Property Model Settings

---

The **Phase Transition** Property Model is available when using the [Property Model Calculator](#) in Thermo-Calc. By varying which condition to relax, it calculates the point when a new phase may form. It is useful to determine melting temperature, boiling temperature or solubility limits. It returns the phase transformation temperature, or composition, depending on the relaxed condition.



For an example, see [PM\\_G\\_01: Phase Transition](#).

### Configuration Settings

The settings are found on the Property Model Calculator when **Phase transition** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Matrix Phase

Choose a **Matrix phase**, the single phase region to start in. Select **Any phase** to use the phases that take part of the initial equilibrium or choose a specific phase.

### Phase to Form

For the new **Phase to form**, select **Any phase** or choose a specific phase.

### Condition to Vary

Choose a composition or temperature **Condition to vary** when calculating the phase transition.

## Suspend All Other Phases

Select the **Suspend all other phases** checkbox to suspend the phases that are not of interest, which allows for a faster calculation. However, this could mean that metastable equilibria are in fact calculated.



It is best practice to suspend phases on the **System Definer** by selecting the applicable phases on the **Phases and Phase Constitution** tab.

## Search Direction

For the composition or temperature condition being varied, choose the **Search direction** to be in a **Positive**, **Negative**, or **Positive and Negative** direction.



The start value is taken from *Condition Definitions*. The closer the start value is to the actual transition point, the more likely it is that the model will find the transition. Note that the phase transition for a given phase may actually occur in the opposite direction from the selected one.

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Composition**
- **Temperature**
- **Relaxed condition: Composition**
- **Relaxed condition: Temperature**



Since only one variable can serve as a **Relaxed condition** per calculation, the quantity that is not relaxed has a NaN result in its corresponding result quantity. The **Temperature** result quantity inherits the unit set in the conditions while the **Composition** result quantity inherits the normalization (mole or mass) but always returns a fraction.

## Scheil Property Model Settings

---

**Scheil** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc. It is used to calculate solidification under the Scheil assumption. The majority of the Model input parameters are taken from the Scheil Calculator to offer the outputs of the liquidus/solidus/solidification range so you can explore these in the calculation types of the Property Models.



For theory or background details about Scheil, start by reviewing [Scheil-Gulliver Solidification Calculations](#).

### Configuration Settings

The settings are found on the Property Model Calculator when **Scheil** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).



The advanced Scheil settings are described with [Scheil Advanced Options: Property Models](#).



These parameters are the same as those used for the Scheil Calculator although some setting names are slightly different.

## Scheil Start Temperature

Enter a **Scheil start temperature** for the Scheil simulation that is higher than the liquidus temperature of the alloy. Is given in the same unit as the temperature in the **Condition Definitions**.

## Calculation Type

Click to select a **Calculation type**: **Classic Scheil**, **Scheil with back diffusion in primary phase**, or **Scheil with solute trapping**. Then based on this selection, additional settings are made available as indicated.



The back diffusion and solute trapping options require the selection of a mobility database on the System Definer.

## Fast Diffusers

Enter the element/component to make as **Fast diffusers**. Include a space between each entry, C N, for example, is the default entered for Carbon and Nitrogen to be fast diffusers.

## Calculate To

Use the **Calculate to** setting to choose from where the calculation should end, **End of Scheil** or **Temperature below solidus**.

- **End of Scheil** stops the calculation when the Scheil calculation is finished.
- **Temperature below solidus** is where a frozen structure is assumed below solidus. When this is selected also enter a **Temperature below solidus** in the field. The default is 298.15.

## Allow Delta Ferrite to Austenite Transition in Steel

This is available when **Classic Scheil** or **Scheil with back diffusion in primary phase** is selected.

Select the **Allow delta ferrite to austenite transition in steel** checkbox to enable BCC to FCC transition.



This feature should only be used for such steels for which it is reasonable to assume infinitely fast diffusion (equilibrium conditions) when delta ferrite is present. When the **Allow delta ferrite to austenite transition in steel** checkbox is selected in Graphical Mode or the DELTA\_FERRITE\_AUSTENITE\_TRANSITION command used in Console Mode, this essentially assumes that diffusion can be considered (infinitely) fast when delta ferrite is present as a solid phase, i.e. assumes an equilibrium condition. After delta ferrite is no longer stable, the usual Scheil assumptions for the selected calculation type are in effect. This setting is therefore not suitable for steels where you DO NOT have a complete BCC to FCC transition during solidification. This setting is also NOT available in combination with solute trapping.

## Show Advanced Scheil Options

Click the **Show advanced Scheil options** checkbox to enter several settings related to Scheil calculations. See [Scheil Advanced Options: Property Models](#).

### Cooling Rate (Back Diffusion)

This field is available when **Scheil with back diffusion in primary phase** is selected.

Specify the **Cooling rate** in Kelvin per second (K/s). An increased value moves the result from equilibrium toward a Scheil-Gulliver calculation.

### Secondary Dendrite Arm Spacing (Back Diffusion)

This field is available when **Scheil with back diffusion in primary phase** is selected.

The **Secondary dendrite arm spacing** is the distance in meters between two secondary dendrite arms. It is calculated from the cooling rate as  $c \cdot (\text{cooling rate})^{-n}$  where  $c$  and  $n$  are entered in these fields.

Enter values for **Secondary dendrite arm spacing c** and **Secondary dendrite arm spacing n**.

### Primary Phase (Back Diffusion and Solute Trapping)

This field is available when **Scheil with back diffusion in primary phase** or **Scheil with solute trapping** is selected.

The primary solidified phase is the phase where the back diffusion or solute trapping takes place.

If **Automatic** (Scheil Calculator) or **Any phase** (Property Models) is selected (or kept as the default), the program tries to find the phase which gives the most back diffusion or solute trapping.

To override this setting, choose a specific primary phase from the list:

- For back diffusion, only phases with diffusion data can be used as primary phases.
- For solute trapping, only phases that dissolve all elements in the system can be used as primary phases.

### Scanning Speed and $\alpha$ (Solute Trapping)

These fields are available when **Scheil with solute trapping** is selected.

As needed, enter **Scanning speed** and  $\alpha$  to define the **Calculated solidification speed**.



On the Scheil Calculator, the **Calculated solidification speed** is shown. For Property Models, below is how the values are calculated from the **Scanning speed** and  $\alpha$  values entered.

The **Calculated Solidification Speed** is  $V_s = V_{\text{scanning}} * \cos(\alpha)$  m/s, where  $V_{\text{scanning}}$  is the user specified **Scanning speed** in m/s and  $\alpha$  is the alpha angle,  $\alpha$ , between the solid/liquid boundary and scanning direction.

## Plot Renderer Settings

[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Temperature**
- **Liquidus temperature, Scheil**
- **Solidus temperature, Scheil**
- **Freezing range, Scheil [K]**. The solidification freezing range (Liquidus - Solidus) given in Kelvin.

- **Volume fraction of phases:** The volume fraction of solid phases (evaluated at the lowest temperature from the simulation)
- **Mole fraction of solid phases:** The mole fraction of solid phases (evaluated at the lowest temperature from the simulation)
- **Total fraction of solid phases:** The total mole fraction of solid phases (evaluated at the lowest temperature from the simulation)
- **Eutectic fraction:** Eutectic fraction at eutectic point (evaluated using Gibbs phase rule).
- **Electric resistivity (ohm m)**
- **Electric conductivity (S/m)**
- **Thermal conductivity (W/mK)**
- **Thermal resistivity (MK/W)**
- **Thermal diffusivity (m<sup>2</sup>/s)**

## About the Spinodal Property Model

**Spinodal** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc and it calculates the spinodal line.

The spinodal is defined by the condition where the second derivative of Gibbs free energy is zero ( $d^2G/dx^2 = 0$ ). The locus of these points is known as the *spinodal curve*.

With this Model up to two spinodal solutions can be found. When searching for the spinodal you can vary the temperature or a composition in mass percent, mole percent, mass fraction, or mole fraction (e.g. mass percent Fe, mole percent Fe, and so forth).

When within the miscibility gap, but above the spinodal curve, phase transformation occur by classical nucleation and growth of the spinodal phase. When within the miscibility gap, but below the spinodal curve, the phase transformation occur by spinodal decomposition of the spinodal phase.



The input parameters are entered on the Configuration window for the Property Model Calculator and described in [Spinodal Property Model Settings](#).



For an example, see [PM\\_G\\_08: Spinodal](#).



The microstructure evolution of spinodal decomposition can be calculated with the phase field method. For some background information you can read more about it in chapter 2 of this doctoral thesis: T. Barkar, "Modelling phase separation in Fe-Cr alloys: A continuum approach," PhD thesis, KTH Royal Institute of Technology, 2018, <https://urn.kb.se/resolve?urn=urn:nbn:se:kth:diva-235394>.

## Spinodal Property Model Settings

**Spinodal** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc and it calculates the spinodal line.



[About the Spinodal Property Model](#)



For an example, see [PM\\_G\\_08: Spinodal](#).

### Configuration Settings

These settings are found on the Property Model Calculator when **Spinodal** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Spinodal Phase

From the **Spinodal phase** list select the applicable phase you want to search for the spinodal.

### Condition to Vary

From the **Condition to vary** list select an option to vary when searching for the spinodal. Up to two spinodals can be found. These options are **Temperature** or those based on the **Composition** (e.g. Fe, Cr, and so forth) and **Composition unit** (i.e. Mass percent, Mole percent, Mass fraction, or Mole fraction).

### Manual Search Limits

The **Manual search limits** checkbox is not selected by default and this sets automatic search limits.

---

Select the checkbox to enter user-defined search limits in the fields. Enter the **Lower condition limit** and **Upper condition limit**. Then enter the number of steps when estimating the position(s) of the spinodal(s) in the **Steps when estimating spinodal** field.

### *Plot Renderer Settings*

#### [Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Composition**
- **Spinodal temperature:** The first spinodal when the search condition is temperature.
- **Spinodal temperature - 2nd solution:** The second spinodal when the search condition is temperature.
- **Spinodal:** The first spinodal regardless of search condition type.
- **Spinodal - 2nd solution:** The second spinodal regardless of search condition type.

## About the $T_0$ Temperature Property Model

**T-Zero Temperature** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc and it calculates the so-called  $T_0$  line.

The  $T_0$  temperature is defined as the temperature where two phases of identical chemical composition have the same molar Gibbs free energy. This temperature is an important quantity in the field of diffusionless phase transformations, e.g. martensitic transformation, since it is the upper limit where diffusionless phase transformations can occur.



For examples, see [PM\\_G\\_09: T-Zero Temperature](#) and [PM\\_G\\_14: Ti-Fe T-Zero Martensite](#)



The input parameters are entered on the Configuration window for the Property Model Calculator and described in [T0 Temperature Property Model Settings](#).

### Vary Temperature or Conditions

With this Model up to two solutions can be found. When searching for  $T_0$  you can vary the condition of temperature or element composition in units of mass percent (e.g. mass percent Fe). This is managed using the settings **Condition to vary** and in some cases, **Manual search limits**.



When **Temperature** is the varied condition,  $T_0$  is obtained as a solution. When a specific composition is selected as the condition to vary, then the temperature(s) is calculated when the  $T_0$  condition is fulfilled. So the resulting temperature is equal to  $T_0$ .



In most cases the curvature of the Gibbs energies of the two phases is different and results in multiple solutions for  $T_0$ . If you keep the default settings (*Condition to vary* → *Temperature* and with no manual search limits) the Property Model Calculator can store up to two solutions. If the solution of interest is not calculated, including **Manual search limits** and/or changing the condition to vary to one of the listed compositions, allows you to find the solution of interest.

---

## Gibbs Energy Addition

There is also an option to include a Gibbs energy addition to the second phase, which allows for inclusion of an energy barrier for martensitic phase transformations. This is done using the **Energy addition second phase [J/mol]** setting.



The energy addition field in Graphical Mode corresponds to the Console Mode command `ADVANCED_OPTIONS PHASE_ADDITION` (see [EQUILIBRIUM\\_CALCUL](#) for information). However, the energy addition can also be composition- and/or temperature-dependent, which is not possible in Console Mode. In addition, Property Models are not available in Console Mode. However, You can access the functionality using methods available with an applicable API such as TC-Python or TC-Toolbox for MATLAB®

## T<sub>0</sub> Temperature Property Model Settings

**T-Zero Temperature** is a Property Model available when using the [Property Model Calculator](#) in Thermo-Calc and it calculates the so-called T<sub>0</sub> line. For more details see [About the T<sub>0</sub> Temperature Property Model](#).



For examples, see [PM\\_G\\_09: T-Zero Temperature](#) and [PM\\_G\\_14: Ti Fe T-Zero Martensite](#)

### Configuration Settings

These settings are found on the Property Model Calculator when **T-Zero Temperature** is selected under **General Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### First Phase

From the **First phase** list select the applicable phase you want to search for the evaluation of Gibbs energy.

### Second Phase

From the **Second phase** list select the applicable phase you want to search for the evaluation of Gibbs energy.

### Energy Addition Second Phase

Use the **Energy addition second phase [J/mol]** field to include a Gibbs energy addition to the second phase. This allows for inclusion of an energy barrier for martensitic phase transformations.



For an example of this see [PM\\_G\\_14: Ti\\_Fe\\_T-Zero\\_Martensite](#).

## Condition to Vary

From the **Condition to vary** list select an option to vary when searching for the solution (or root). Up to two solutions can be found. These options are **Temperature** or those based on the *Composition + Composition unit* for example **Mass percent Fe**, **Mass percent Cr**, **Mass percent Ni**, and so forth.



When **Temperature** is the varied condition,  $T_0$  is obtained as a solution. When a specific composition is selected as the condition to vary, then the temperature(s) is calculated when the  $T_0$  condition is fulfilled. So the resulting temperature is equal to  $T_0$ .



In most cases the curvature of the Gibbs energies of the two phases is different and results in multiple solutions for  $T_0$ . If you keep the default settings (*Condition to vary* → *Temperature* and with no manual search limits) the Property Model Calculator can store up to two solutions. If the solution of interest is not calculated, including **Manual search limits** and/or changing the condition to vary to one of the listed compositions, allows you to find the solution of interest.

## Manual Search Limits

The **Manual search limits** checkbox is not selected by default and this sets automatic search limits.

Select the checkbox to enter user-defined search limits in the fields. Enter the **Lower condition limit** and **Upper condition limit**. Then in the **Steps when estimating root(s)** field, enter the number of steps in a linear grid when estimating the position(s) of the conditions.



*Root* means a solution of  $T_0$ , so a composition and temperature where the Gibbs energies are the same. In other words, it finds the "root(s)" where the two selected phases have equal Gibbs energy. Think of the "root" as the conditions (temperature, composition) where the Gibbs energy of the two selected phases intersect, i.e. have equal Gibbs energies.

### Plot Renderer Settings



#### [Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Composition**, when the search condition is temperature
- **Temperature**, when the search condition is composition (in this case temperature is equal to T-Zero).
- **T-Zero**: The first root (solution) when the search condition is temperature.
- **T-Zero - 2nd solution**: The second root (solution) when the search condition is temperature.
- **Varied condition**: The first varied condition regardless of search condition type.
- **Varied condition - 2nd solution**: The second varied condition regardless of search condition type.

## About the Yield Strength Property Model

The **Yield Strength** Model is a general model to estimate the yield stress of a material that considers these contributions to the overall yield stress of the material:

- Intrinsic strength ( $\sigma_i$ ) for the pure elements,
- Grain boundary strengthening ( $\sigma_{gb}$ ),
- Solid solution strengthening ( $\sigma_{ss}$ ),
- Precipitation strengthening ( $\sigma_p$ ), and
- A user set, constant strength contribution ( $\sigma_c$ ).

The temperature ( $T$ ) is used to evaluate the equilibrium state of the system, and the resulting compositions and phase fractions are subsequently used in the evaluation of mechanical properties.

It returns the calculated yield strength at room temperature for the specified material using the equilibrium values calculated at temperature  $T$  (i.e.  $T$  is the freeze-in temperature).

Below follows a short summary for each of the contributions.



The settings for the contributions are found on the Property Model Calculator when **Yield Strength** is selected under **General Models**.



Yield strength is also available as a variable when working with Plots and Tables that are successors to the **Precipitation Calculator**.



The settings details, some of which are mentioned in this topic, are entered on the **Configuration** window, see [Yield Strength Property Model: Simplified Mode Settings](#) or [Yield Strength Property Model: Advanced Mode Settings](#).

### Intrinsic Strength

The *intrinsic strength* model formulation for this Property Model corresponds to a surface of reference with end-members. For example, for FCC\_A1 modeled as (Fe,Ti)(C,Va), the intrinsic strength is a linear combination (as given by the composition) of Fe:Va (i.e. pure FCC Fe), Ti:Va (i.e. pure FCC Ti), Fe:C (i.e., FCC FeC) and Ti:C (i.e., FCC TiC). With such an

implementation, it is possible to calculate yield strength for materials with a significant fraction of hard phases, such as carbides, embedded in a soft matrix and the intrinsic strengthening effect of interstitial elements.

The general form of the intrinsic strength model follows in the spirit of the Compound Energy Formalism (CEF) surface of reference term, where the intrinsic strength for, e.g., a two-sublattice phase model would correspond to:

$$\sigma_{\text{intrinsic}} = \sum_{i,j} \sigma_{i:j} y'_i y''_j$$

Such a formulation means that the intrinsic strength depends on the calculated site fractions from, e.g., an equilibrium calculation.

### Phases with Assessed Intrinsic Strength



Not all compound phases are yet assessed in terms of intrinsic strength. The following phases are assessed: BCC\_A2, CEMENTITE\_D011, FCC\_A1, FCC\_L12, HCP\_A3, M23C6\_D84, M3C2\_D510, M6C\_E93, M7C3\_D101, MC\_SHP.

### Temperature Dependence for Pure Elements

The following pure elements can be used to evaluate yield strength at non-room temperature conditions:

- Ag, Al, Au, Be, Ce, Co, Cr, Cu, Dy, Er, Fe, Gd, Hf, Ho, Ir, La, Mg, Mo, Nb, Nd, Ni, Pd, Pt, Rh, Sm, Ta, Ti, U, V, W, Y, Zn, and Zr

The temperature dependence is of the form:

$$y_{s_{\text{pure}}} = A * \exp(-B/T)$$

and is available for the listed pure elements where  $A$  and  $B$  are fitted to data from [1968Sam; 2017Wu]. This improves the results for evaluated yield strength at non-room temperature conditions.



The calculated solid solution strength can differ from versions of Thermo-Calc prior to 2022b where the pure element strength from Cordero et al. was used [2016Cor]. This is especially noticeable for pure element or dilute solution calculations. The pure element strength is only evaluated for FCC, BCC, and HCP.



Compound end-members of the intrinsic model are not modeled as temperature dependent, however, this should have only a small effect on most alloys.

## Constant Addition of Strength

The *constant strength contribution* parameter,  $\sigma_c$ , is set by the user as a constant strength addition to the material.

## Grain Boundary Strengthening

The *grain boundary strengthening*,  $\sigma_{gb}$ , is the contribution to the total strength stemming from the grain boundary's ability to hinder dislocation movement described by the Hall-Petch relation [2004Han].

The parameters entered are the Hall-Petch constant,  $k_{h-p}$ , in MPa  $\mu\text{m}^{1/2}$ , and the average grain diameter in  $\mu\text{m}$ .

## Solid Solution Strengthening

The *solution strengthening*,  $\sigma_{ss}$ , is the contribution to total strength due to the elastic strains in the crystal lattice caused by alloying elements of a lattice parameter differing from the main constituent. For the solid solution strengthening the option of evaluating the strength at any temperature is available. The result is normalized with respect to the sum of all mole fractions for FCC, BCC, and HCP. This option uses the phase fractions and compositions calculated at the overall equilibrium temperature. The model by Walbrühl et al. [2017Wal] is implemented.



[PM\\_G\\_06: Yield Strength HEA](#) shows how this model is applied.

## Precipitation Strengthening

The *precipitation strengthening*,  $\sigma_p$ , is the contribution to total strength due to precipitation of a secondary phase, e.g. gamma prime ( $\gamma'$ ) in Ni-base alloys or  $\text{Al}_3\text{Sc}$  precipitates in Al-base alloys. This model is available for any number of precipitating phases. The total particle strength is calculated by the square mean of the individual particle strength contributions:

$$\sigma_p = \sqrt{\sigma_{p1}^2 + \sigma_{p2}^2 + \dots + \sigma_{pn}^2}$$

The **Simplified model (general)**, **Seidman model (Al-base)**, **Reppich model (Ni-based)**, and **Deschamps model (Al-base)** are available as settings on the **Configuration** window to estimate the precipitation strengthening contribution. These are briefly described next.

### Simplified Model (General)

The **Simplified model (general)** is based on the work by Zander et al. [2008Zan] and considers the general mechanisms of cutting and looping without regard to any detailed dislocation mechanisms. The default parameters are, however, optimized against experimental data for the Al-0.3wt%Sc alloy described by Seidman et al. [2002Sei]. Also see [Mean Precipitate Radius and Particle Size Distribution \(PSD\)](#) below.



[PM\\_G\\_04: Yield Strength](#) compares the **Simplified model (general)**, **Seidman model (Al-base)**, and **Deschamps model (Al-base)** yield strength versus precipitate radius to experimental data for an Al-0.3wt%Sc alloy homogenized at 648 °C for 24 hours and subsequently aged at 350 °C.

### Seidman Model (Al-base)

The **Seidman model (Al-base)** by Seidman et al. [2002Sei] considers several contributions to the overall precipitation strengthening - ordering, coherency stresses, shear modulus mismatch, and Orowan looping. The Seidman model is suitable for Al-base alloys with a low volume fraction of spherical precipitates. Also see [Mean Precipitate Radius and Particle Size Distribution \(PSD\)](#) below.



[PM\\_G\\_04: Yield Strength](#) compares the **Simplified model (general)**, **Seidman model (Al-base)**, and **Deschamps model (Al-base)** yield strength versus precipitate radius to experimental data for an Al-0.3wt%Sc alloy homogenized at 648 °C for 24 hours and subsequently aged at 350 °C.

## Reppich Model (Ni-base)

The **Reppich model (Ni-base)** by Reppich et al. [1982aRep] considers both weak and strong coupling of dislocation as they propagate through an ordered secondary phase and is thus suitable for Ni-base alloys with relatively high volume fractions of gamma prime or gamma double prime. Also see [Mean Precipitate Radius and Particle Size Distribution \(PSD\)](#) below.



The Reppich model is not valid for very small values of volume fraction precipitate and precipitate radius.



[PM\\_G\\_05: Yield Strength NiAlCr](#) is an example of this model.

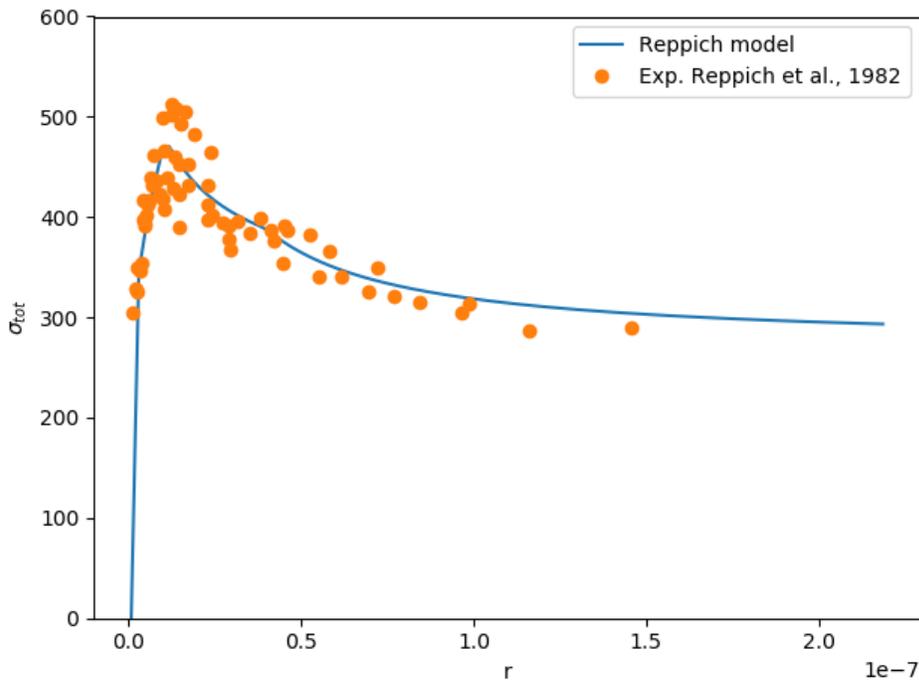


Figure 15: Reppich model plotted with results and experimental data for PE16 [1982bRep], all data is normalized with the square root of the volume fraction of precipitate.

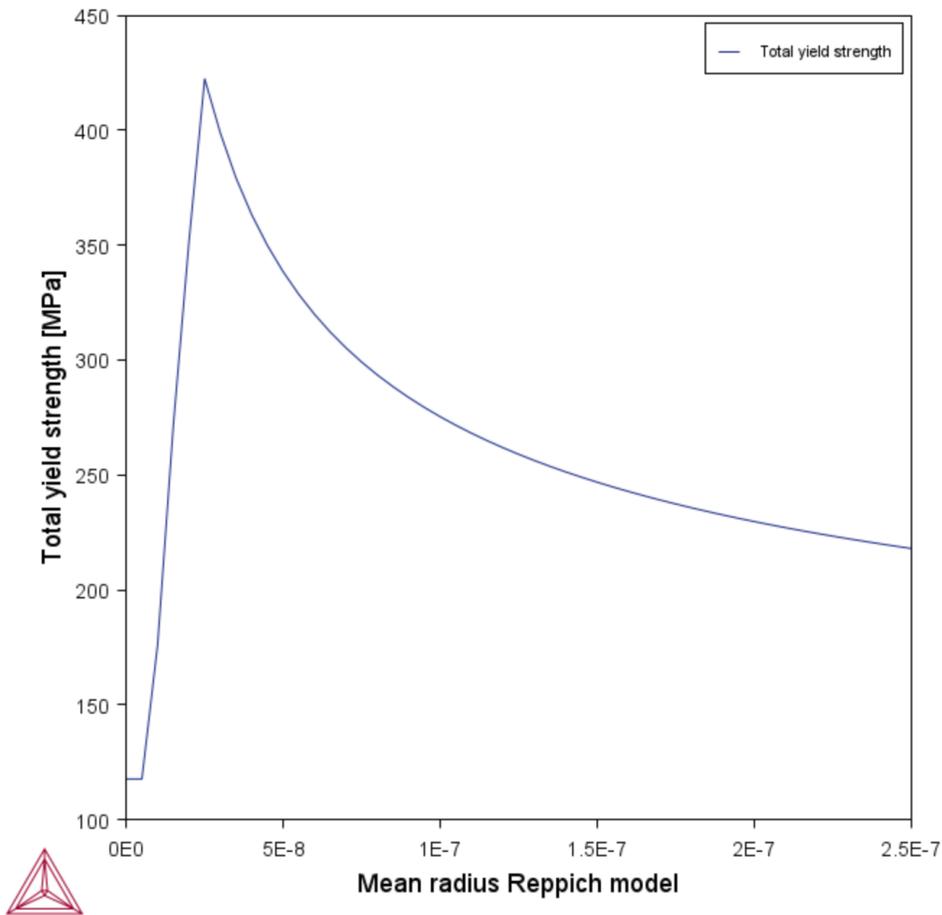


Figure 16: Reppich model yield strength vs mean precipitate radius for Ni-10Cr-10Al.

### Deschamps Model (Al-base)

The **Deschamps model (Al-base)** by Deschamps and Brechet [1998Des] considers contributions to the overall precipitation strengthening precipitate from particle cutting and Orowan looping. The Deschamps model is suitable for Al-base alloys with a low volume fraction of spherical precipitates. However, it has been successfully validated against other alloy types, such as Fe-based ferritic materials with TiC precipitates. Also see [Mean Precipitate Radius and Particle Size Distribution \(PSD\)](#) below.

The Deschamps model considers Friedel statistics as a default setting, with the option to instead apply Kock's statistics, as shown in the PM\_G\_04 example.



[PM\\_G\\_04: Yield Strength](#) compares the **Simplified model (general)**, **Seidman model (Al-base)**, and **Deschamps model (Al-base)** yield strength versus precipitate radius to experimental data for an Al-0.3wt%Sc alloy homogenized at 648 °C for 24 hours and subsequently aged at 350 °C.

## Mean Precipitate Radius and Particle Size Distribution (PSD)

The mean precipitate radius entered on the **Configuration** window for all precipitation strengthening models is transformed to a Lifshitz-Slyozov-Wagner (LSW) particle size distribution [2010Tir]. The distribution is stepwise integrated and the mean radius for all sections are used to calculate a partial precipitate strengthening contribution. The partial contributions are added as

$$\sigma_{tot}^2 = \sum(\sigma_p^2)$$

The use of a PSD smooths the transition between the cutting and Orowan regions. This change is implemented for the Simplified model (general) and Seidman model (Al-base).

When using the Yield Strength Model with the Precipitation Module (TC-PRISMA) the PSD as calculated by Precipitation Module (TC-PRISMA) is used for each time step. Also see [Precipitation Calculator Plot Renderer](#) for the settings input related to this.

### References

- [1968Sam] Samsonov, G. V. " Handbook of the Physicochemical Properties of the Elements," (Springer US, Boston, MA, 1968, 2012 American Edition).
- [1982aRep] Reppich, B. "Some new aspects concerning particle hardening mechanisms in  $\gamma'$  precipitating Ni-base alloys—I. Theoretical concept," Acta Metall. 30, 87–94 (1982).
- [1982bRep] Reppich, B., Schepp, P., and Wehner, G. "Some new aspects concerning particle hardening mechanisms in  $\gamma'$  precipitating nickel-base alloys—II. Experiments," Acta Metall. 30, 95–104 (1982).
- [1998Des] A. Deschamps, Y. Brechet, Influence of predeformation and ageing of an Al–Zn–Mg alloy—II. Modeling of precipitation kinetics and yield stress. Acta Mater. 47, 293–305 (1998).

- 
- [2002Sei] Seidman, D. N., Marquis, E. A., and Dunand, D. C. "Precipitation strengthening at ambient and elevated temperatures of heat-treatable Al(Sc) alloys," *Acta Mater.* 50, 4021–4035 (2002).
- [2004Han] Hansen, N. "Hall–Petch relation and boundary strengthening," *Scr. Mater.* 51, 801–806 (2004).
- [2008Zan] Zander, J. and Sandström, R. "One parameter model for strength properties of hardenable aluminium alloys," *Mater. Des.* 29, 1540–1548 (2008).
- [2010Tir] M. Tiryakioğlu, G. Ökten, D. Hudak, R. T. Shuey, J. P. Suni, On evaluating fit of the Lifshitz–Slyozov–Wagner (LSW) distribution to particle size data. *Mater. Sci. Eng. A* 527, 1636–1639 (2010).
- [2016Cor] Cordero, Z. C., Knight, B. E., and Schuh, C. A. "Six decades of the Hall–Petch effect – a survey of grain-size strengthening studies on pure metals," *Int. Mater. Rev.* 61, 495–512 (2016).
- [2017Wal] Walbruhl, M., Linder, D., Ågren, J., and Borgenstam, A. "Modelling of solid solution strengthening in multicomponent alloys," *Mater. Sci. Eng. A* 700, 301–311 (2017).
- [2017Wu] Z. Wu, M. C. Tropsky, Y. F. Gao, J. R. Morris, G. M. Stocks, H. Bei, Phase stability, physical properties and strengthening mechanisms of concentrated solid solution alloys. *Curr. Opin. Solid State Mater. Sci.* 21, 267–284 (2017).

## Yield Strength Property Model: Simplified Mode Settings

This section has information about the **Simplified** mode configuration settings available for the **Yield Strength** Property Model.

The **Yield Strength** Model is available with the Property Model Calculator and calculates yield stress.



Yield strength is also available as a quantity when working with Plots and Tables that are successors to the **Precipitation Calculator**.

The Yield Strength Model returns the calculated yield strength at room temperature (per default) for the specified material using the equilibrium values calculated at temperature  $T$  (i.e.  $T$  is the freeze-in temperature). The Model is defined by further setting these contributions:

- Intrinsic strength ( $\sigma_i$ )
- Solid solution strengthening ( $\sigma_{ss}$ )
- Grain boundary strengthening ( $\sigma_{gb}$ )
- Precipitation strengthening ( $\sigma_p$ )
- Constant strength addition ( $\sigma_c$ )



These contributions are defined in [About the Yield Strength Property Model](#).



For examples using the Property Model Calculator, see [PM\\_G\\_04: Yield Strength](#), [PM\\_G\\_05: Yield Strength NiAlCr](#), and [PM\\_G\\_06: Yield Strength HEA](#).



For an example when it is available as an axis variable with the Precipitation Calculator, see [P\\_01: Isothermal Precipitation of Al3Sc](#).

### General Configuration Settings

The settings for these contributions are found on the [Property Model Calculator](#) when the **Yield Strength** Model is selected under **General Models**.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).



There are settings described elsewhere when **Yield Strength** is selected as an axis variable for a Plot Renderer or Table Renderer that are successors to the **Precipitation Calculator**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.

### *Simplified Mode Settings*

Select **Simplified** as the **Mode**.



If the following set up is not sufficient for your simulation, see the [Yield Strength Property Model: Advanced Mode Settings](#) for more detailed options.

For all calculations in Simplified Mode, the automatically applied settings are as follows:

- Equilibrium minimization strategy is Global test preferred.
- The matrix phase is automatically identified.
- Burgers vector is calculated.
- Shear modulus for the matrix is calculated.

### **Solid Solution Strengthening (Simplified Mode)**

Select the **Solid solution strengthening** checkbox to include solid solution strengthening in the calculation.

### **Grain Boundary Strengthening and Grain Size (Simplified Mode)**

Click to select the **Grain boundary strengthening** checkbox to include calculated grain boundary strengthening ( $\sigma_{gb}$ ). Then enter the **Grain size [um]**, which is the grain diameter in  $\mu\text{m}$ .

## Precipitation Strengthening (Simplified Mode)

Click to select the **Precipitation strengthening** checkbox to include calculated precipitation strengthening ( $\sigma_p$ ).



In Simplified Mode, the default **Precipitation strength model** is automatically the **Deschamps model (Al-base)**. However, if the major element is Ni and gamma prime is present in the system, the **Reppich model (Ni-base)** is used instead.

Then enter the following settings.

- Select a **Precipitate** from the list.
- Enter a **Mean precipitate radius**. The unit is meter (m).
- Enter a **Critical radius** ( $r_c$ ), which determines the mean precipitate radius where transition between cutting and looping occurs. The unit is meter (m).

## Constant Strength Addition (Simplified Mode)

Enter a value for the **Constant strength addition**,  $\sigma_c$ . The unit is MPa.

### *Additional Settings*

- [Yield Strength Property Model: Calculation Type and Grid Definitions](#)
- [Yield Strength Property Model: Calculation Type and Grid Definitions](#)
- [Yield Strength Property Model: Plot and Table Settings](#)

## Yield Strength Property Model: Advanced Mode Settings

This section has information about the **Advanced** mode configuration settings available for the **Yield Strength** Property Model.

The **Yield Strength** Model is available with the Property Model Calculator and calculates yield stress.



Yield strength is also available as a quantity when working with Plots and Tables that are successors to the **Precipitation Calculator**.

The Yield Strength Model returns the calculated yield strength at room temperature (per default) for the specified material using the equilibrium values calculated at temperature  $T$  (i.e.  $T$  is the freeze-in temperature). The Model is defined by further setting these contributions:

- Intrinsic strength ( $\sigma_i$ )
- Solid solution strengthening ( $\sigma_{ss}$ )
- Grain boundary strengthening ( $\sigma_{gb}$ )
- Precipitation strengthening ( $\sigma_p$ )
- Constant strength addition ( $\sigma_c$ )



These contributions are defined in [About the Yield Strength Property Model](#).



For examples using the Property Model Calculator, see [PM\\_G\\_04: Yield Strength](#), [PM\\_G\\_05: Yield Strength NiAlCr](#), and [PM\\_G\\_06: Yield Strength HEA](#).



For an example when it is available as an axis variable with the Precipitation Calculator, see [P\\_01: Isothermal Precipitation of Al3Sc](#).

## General Configuration Settings

The settings for these contributions are found on the [Property Model Calculator](#) when the **Yield Strength** Model is selected under **General Models**.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).



There are settings described elsewhere when **Yield Strength** is selected as an axis variable for a Plot Renderer or Table Renderer that are successors to the **Precipitation Calculator**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.

## Advanced Mode Settings

Select **Advanced** as the **Mode**. Available settings are displayed based on the choice.



For the specific **Simplified** settings, go to [Yield Strength Property Model: Simplified Mode Settings](#).

### Set Matrix Phase Explicitly

Select the **Set matrix phase explicitly** checkbox and choose a **Matrix** phase from the list. This allows forcing a specific phase to be considered as the matrix phase.



This is an important parameter as it impacts the evaluation of the shear modulus and affects the result for the precipitation strengthening contribution.

The solid solution strengthening contribution is not affected by choice of matrix phase, since all stable FCC, BCC, and HCP phases always are evaluated for solid solution strength contributions. If this option is not selected, the matrix phase is automatically determined by the model.

### Solid Solution Strengthening (Advanced Mode)

Select the **Solid solution strengthening** checkbox to include solid solution strengthening in the calculation.

#### Evaluate at Higher Temperature

This checkbox is available when the **Solid solution strengthening** checkbox is selected. By default the checkbox is not and the simulation is evaluated at room temperature.

Select the **Evaluate at higher temperature** checkbox to evaluate the contribution  $\sigma_{ss}$  at temperatures higher than room temperature.

When selected, enter a **Solid solution strengthening temperature** in the field. The unit of temperature is the same as that selected above under *Condition Definitions*.



If no temperature is entered, the default, room temperature, is used.

---

## Grain Boundary Strengthening and Grain Size (Advanced Mode)

Click to select the **Grain boundary strengthening** checkbox to include calculated grain boundary strengthening ( $\sigma_{gb}$ ). Then enter the **Grain size [um]**, which is the grain diameter in  $\mu\text{m}$ .

## User-defined Hall-Petch Coefficient

By default, a suitable default **Hall-Petch coefficient** is automatically calculated by the model. Select the **User defined Hall-Petch coefficient** checkbox to enter a different **Hall-Petch coefficient**:  $k_{h-p}$  in  $\text{MPa } \mu\text{m}^{1/2}$ .

## Precipitation Strengthening (Advanced Mode)

Click to select the **Precipitation strengthening** checkbox to include calculated precipitation strengthening ( $\sigma_p$ ). Then enter the settings that follow under the **Precipitation Strengthening Model** section.

The **Precipitate** list defaults to **Any phase**.

- If **Any phase** is selected as a Precipitate phase, the non-matrix-like phase with the highest volume fraction is selected. If more than one precipitate is defined and set to **Any phase**, non-matrix-like phases are selected in order of descending volume fraction.
- If more precipitates with **Any phase** are defined than there are stable non-matrix-like phases in the system, the redundant precipitates are ignored.
- Graphite is prohibited to be set as a precipitate phase.

Otherwise, select another phase from the list. Click the  $\oplus$  and  $\ominus$  buttons to add and remove additional Precipitates that can have different settings.

## ***Precipitation Strengthening Model***

The precipitation strengthening contribution ( $\sigma_p$ ) is calculated by one of these models, which are further described below.

Select a **Precipitation strengthening model** from the list:

- **Simplified model (general)**
- **Seidman model (Al-base)**: Preferentially used for Al-based alloys.
- **Reppich model (Ni-base)**: Preferentially used for Ni-based alloys.



The Reppich model is not valid for very small values of volume fraction precipitate and precipitate radius.

- **Deschamps model (Al-base)**: Preferentially used for Al-based alloys.

### Simplified Model (General) Settings

For the **Simplified model (general)**, enter values in the fields for the following. The default values of  $k_p$  and  $r_c$  are optimized against experiments for Al-Sc.

- **Mean radius Simplified model**: The precipitate mean radius. The unit is meter (m). Also see [Mean Precipitate Radius and Particle Size Distribution \(PSD\)](#)
- **Critical radius ( $r_c$ )**, which determines the mean precipitate radius where transition between cutting and looping occurs. The unit is meter (m).
- $k_p$ : The prefactor for the **Simplified model**, which adjusts the overall height of the yield precipitation strength curve.

### Seidman Model (Al-base) Settings

For the **Seidman model (Al-base)**, enter values in the fields for the following. The default parameters are optimized against experiments.

- **Mean radius Seidman model**: The precipitate mean radius. The unit is meter (m). Also see [Mean Precipitate Radius and Particle Size Distribution \(PSD\)](#)
- **Anti-phase boundary energy**:  $\gamma_{apb}$ , which is the average anti-phase boundary energy for the precipitate phase. The unit is  $\text{Jm}^{-2}$ .
- **m**: A Seidman exponent, it is a constant in the expression for strengthening by modulus mismatch. See [1985Ard] for details.
- $\alpha$ : A Seidman prefactor. It is a fitting constant in the expression for coherency strengthening. See [1985Ard] for details.

## Reppich Model (Ni-base) Settings

For the **Reppich model (Ni-base)**, enter values in the fields for the following. The default parameters are optimized against experiments.

- **Mean radius Reppich model:** The precipitate mean radius. The unit is meter (m).
- **Anti-phase boundary energy:**  $\gamma_{apb}$ , which is the average anti-phase boundary energy for an L12 ordered precipitate phase. The unit is  $\text{Jm}^{-2}$ .
- **Calculate anti-phase boundary energy:** Select the checkbox to use calculated anti-phase boundary energy in the yield strength calculation. The software uses the standalone APBE model in the Ni-library from within the Yield Strength Model for the calculation.



The APBE can only be calculated for gamma prime ( $\gamma'$ ). If the selected precipitate phase is not  $\gamma'$ , the user-defined value in the configuration window is used.

- **w:** A parameter that accounts for elastic repulsion between strongly coupled dislocations.
- **A:** Numerical factor reflecting particle morphology (0.72 for spherical). See [1982Rep] for details.

## Dechamps Model (Al-base) Settings

For the **Dechamps model (Al-base)** enter values in the fields for the following. The default parameters are optimized against experiments.

- **Mean radius Dechamps model:** Precipitate mean radius. The unit is meter (m).
- **Critical radius ( $r_c$ ),** which determines the mean precipitate radius where transition between cutting and looping occurs. The unit is meter (m).
- $\beta$ : A parameter used to evaluate the dislocation line tension. Usually close to 1/2 but can deviate from this value.
- **Use Kock's statistics:** By default, Friedel statistics are used. Select the checkbox to use Kock's statistics instead. See [1998Des].

## ADDITIONAL PRECIPITATION PARAMETERS (ADVANCED MODE)

---

The following are available for all models and when the **Additional precipitation parameters** checkbox is selected.

### Taylor Factor

Enter a value for the **Taylor factor**.

The value of the Taylor factor accounts for texture in the material. Normally the value is between 2.24 and 3.06 for random orientations.

### Shear Modulus

Enter a value for the **Shear modulus** of the matrix phase in Pa.

### Burgers Vector

Enter a value for the **Burgers vector**. The unit is meter (m).

### Constant Strength Addition (Advanced Mode)

Select the **Constant strength addition** checkbox and enter a value for the **Constant strength addition**,  $\sigma_c$ . The unit is MPa This accounts for any contributions not accounted for in the Yield Strength model, e.g. martensite strengthening.

### Equilibrium Minimization Strategy

Select an **Equilibrium minimization strategy**. The default uses the **Global test preferred** option.

The **Minimization Strategy** setting is used to ensure that the most stable minimum under the specified conditions is computed.

With either the *Global test preferred* or *Local minimization preferred* setting, the program cycles through options before it gives up:

- For **Global test preferred** the minimization starts with a global test and if that fails it runs full global minimization.
- For **Local minimization preferred** the minimization starts with a local minimization, in case of failure it tries a global test and finally a full global minimization.

If you choose **Local minimization only** or **Global minimization only**, the program just tries the one setting and gives up if it fails.



For general information about global minimization, see the topic related to the Console Mode command, [GLOBAL\\_MINIMIZATION](#).

## Precipitation Calculator Setting - Density of Points



This setting is only available for a precipitation calculation using the Precipitation Module (TC-PRISMA).

Select the **Density of points: Low** (the default), **Medium**, or **High**. This is a measure of how many time steps yield strength is calculated.

### References

[1982Rep] Reppich, B. "Some new aspects concerning particle hardening mechanisms in  $\gamma'$  precipitating Ni-base alloys—I. Theoretical concept," Acta Metall. 30, 87–94 (1982).

[1998Des] A. Deschamps, Y. Brechet, Influence of predeformation and ageing of an Al–Zn–Mg alloy—II. Modeling of precipitation kinetics and yield stress. Acta Mater. 47, 293–305 (1998).

### Additional Settings

- [Yield Strength Property Model: Calculation Type and Grid Definitions](#)
- [Yield Strength Property Model: Plot and Table Settings](#)

## Yield Strength Property Model: Calculation Type and Grid Definitions

This section has information about the *Calculation Type* and *Grid Definitions Configuration* settings available for the **Yield Strength** Property Model. It is a continuation from **Simplified** or **Advanced Mode** setting entries:

- [Yield Strength Property Model: Simplified Mode Settings](#)
- [Yield Strength Property Model: Advanced Mode Settings](#)

---

## Calculation Type Settings

### Calculation Type

- **Single** to calculate a single point. Any model can use this calculation type and no additional settings are required. The results from this calculation are displayed in the **Event Log** or if a **Table Renderer** is used, in the **Visualizations** window.
- **One Axis**: To vary a quantity on the X-axis.
- **Grid**: Evaluates two axis variables of the selected quantities in the specified range and number of steps.
- **Min/Max**: Evaluates the Property Model(s) for all variations of the selected quantities at the given limits. The **Mean** field is as defined under *Condition Definitions* for the respective quantity. The total minimum and maximum of the model(s) results are shown in the **Event log**.
- **Uncertainty**: Evaluates the Property Model(s) where the values of the quantities are sampled from Gaussian distributions. The **Mean** field is as defined under *Condition Definitions* for the respective quantity. The result is visualized as a histogram or normal probability plot by adding a Plot Renderer activity.
- **Batch**: Use this to allow for high throughput calculations and read data from a file, such as a spreadsheet, rather than, for example, having a fixed grid or random values. Results from these batch calculations can then be compared to experimental values (as long as this data is included), using a **Cross plot** on the Plot Renderer.

### Axis Definition

For **One Axis**, select a **Quantity** to vary along the X-axis, for example, **Temperature** then enter a **Min**, **Max** and **Number of steps**.

### Grid Definition Settings

#### Grid Definitions

For the **Grid** calculation type, define the two axes variables using the fields and menus: **Quantity**, **Min**, **Max**, and **Number of steps**. The number of steps along with the minimum and maximum values for the axes define a grid. For each grid point the selected models are evaluated.



For a **Grid**, on the Plot Renderer, you can choose to plot a *Heat map*, *Contour*, or *3D* diagram.



[Plot Type: Heat Map, Contour, and 3D](#)

## Min/Max Definitions

For the **Min/Max** calculation type, click to select the applicable checkboxes under **Quantity to enable or disable an axis**. The **Mean** field is as defined under *Condition Definitions* for the respective quantity.

Enter numerical values in the  $\Delta$ **Min/Max** fields to evaluate the change to the quantity. The highest and lowest value of each selected model results are shown in the **Event Log**.

## Sampling of Data from Gaussian Distributions

For the **Uncertainty** calculation type, click to select the applicable checkboxes under **Quantity to vary this quantity**. The **Mean** field is for the distribution function as defined under *Condition Definitions* for the respective quantity. The default  $\Delta$ Min/Max is a 95% confidence interval.



[About the Uncertainty Calculations](#)

## Sampling Parameters

For the **Uncertainty** calculation type, choose a number of **Standard deviations** where you want to **Truncate the distribution**. Choose the **Total number of samples**.



[About the Uncertainty Calculations](#)

## Additional Settings

- [Yield Strength Property Model: Plot and Table Settings](#)

## Yield Strength Property Model: Plot and Table Settings

This section has information about the Plot Renderer and Table Renderer Configuration settings available for the **Yield Strength** Property Model. It is a continuation from **Simplified** or **Advanced Mode** setting entries:

- [Yield Strength Property Model: Simplified Mode Settings](#)
- [Yield Strength Property Model: Advanced Mode Settings](#)
- [Yield Strength Property Model: Calculation Type and Grid Definitions](#)

For more information see:

-  [Plot Renderer](#) and [Configuration Settings](#)
-  [Table Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.



Calculations can be performed over a range for the model parameters available in each sub-model. If, for example, the mean precipitate radius in a precipitation strengthening model is set as the variable to step over, it becomes available as a plot quantity in the Plot Renderer.

Selectable result quantities:

- Total yield strength
- Intrinsic strength
- Solid solution strengthening
- Solid solution strengthening FCC
- Solid solution strengthening BCC
- Solid solution strengthening HCP
- Grain boundary strengthening
- Total precipitation strengthening
- Precipitation strengthening per phase
- Constant strength addition
- Total hardness
- Intrinsic hardness
- Total solid solution hardening
- Solid solution hardening FCC
- Solid solution hardening BCC

- Solid solution hardening HCP
- Grain boundary hardening
- Total precipitation hardening
- Precipitation hardening per phase
- Constant hardness addition
- Volume fraction of precipitate phase
- Volume fraction of stable phase(s)

---

# Scheduler and Event Log Windows

---

**In this section:**

---

Using the Scheduler .....	345
Event Log Window .....	347

## Using the Scheduler

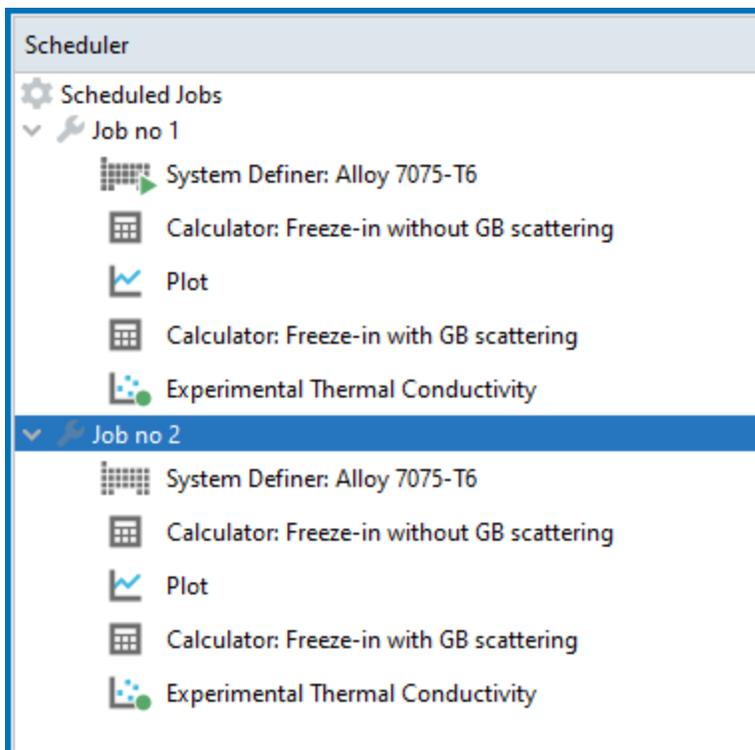
The **Scheduler** window has activity information about current or scheduled jobs. A *job* is any activity you have selected to be performed. The Scheduler also indicates if there is an error during a job and when a job has stopped. If you have set an activity, tree or project to be performed at a later time then this is also listed.

### About the Scheduler Jobs

- Whenever you trigger the performance of a whole project, tree or individual activity, all the activities to be performed are shown under a job heading.
- When all the activities belonging to a job are completed the job and its activities disappear from the Scheduler window.
- If there is an error during the performance of an activity, icons for that job and activity remain in the Scheduler window until removed.



#### [Node or Window Status Markers](#)



## ***Managing the Schedule***

In the Scheduler window, you can cancel scheduled jobs, remove errors and show information about errors.

- To cancel all jobs: Right-click the **Scheduled Jobs** label or the cogwheel icon, and select **Cancel All Jobs**
- To remove (clear) all failed activities and jobs: Right-click the **Scheduled Jobs** label or the cogwheel icon, and select **Clear All Errors**
- To cancel a specific job: Right-click the job label (for example, **Job no 1**) and select **Cancel Job**
- To remove (clear) a specific failed job: Right-click the job label (for example, **Job no 1**) and select **Cancel Job**
- To open a window with error details: Right-click the label for the failed job (for example, **Job no 1**), and select **Show Error Log**

## Event Log Window

The **Event Log** window is closed by default. Information about what Thermo-Calc is processing or doing is in blue text, an error message is in red text, and caution/warning information is in yellow text.



### Global Settings

```

Event Log
14:11:08,366 INFO GoblerTask: Gobbled: 14:11:08,366 [pool-5-thread-1] INFO AbstractQuantity: condition T the value 1773.15
14:11:08,472 INFO GoblerTask: Gobbled: 14:11:08,472 [pool-5-thread-1] INFO GeneralEquilibriumCalculator: Calculating equilibrium
14:11:08,472 INFO GoblerTask: Gobbled: 14:11:08,472 [pool-5-thread-1] INFO AbstractQuantity: condition W(Mn) the value 0.5
14:11:08,473 INFO GoblerTask: Gobbled: 14:11:08,472 [pool-5-thread-1] INFO AbstractQuantity: condition T the value 1773.15
14:11:08,570 INFO ScheduledJob: Execution cancelled
14:11:08,571 INFO ScheduledJob: Cancelling activity: Equilibrium Calculator 1
14:11:08,570 INFO ScheduledJob: Cancelling activity: Plot Renderer 1
14:11:08,584 INFO GoblerTask: Gobbled: 14:11:08,583 [pool-5-thread-1] INFO GeneralEquilibriumCalculator: Calculating equilibrium
14:11:09,813 WARN ProcessExecutorAllocator: The subprocess completed with status code 2: the execution was cancelled by the user

```

- To open the Event Log window: Click the **Event Log** window once to expand it or select it from the **Window → Event Log** menu. Double-clicking the window maximizes it.
- To clear all messages in the window, right-click in the text area and select **Clear**.
- To specify the level of detail in the Event Log, open the **Options** window:
  - Windows: Select **Tools → ☰ Options**.
  - Mac: Select **Thermo-Calc <version> → Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **General** tab and set the **Log level** slide bar to anything between **Debug** (most detailed) and **Error** (least detailed).

---

## Working with Plots and Tables

---

**In this section:**

---

Performing Projects, Trees and Activities .....	349
Saving and Printing the Output (Graphical Mode) .....	350
Plotting Options .....	353
Common Quantities Available for Plots and Tables .....	358
Plot Types .....	362
Plot Type: Cross Plot and Statistical .....	369
Plot Type: Parallel Coordinates .....	374
Plot Type: Heat Map, Contour, and 3D .....	380
Plot Type: Flexible Mode and Grouped Mode .....	385
Plot Type: CCT Mode .....	387
Plot Type: TTT Mode .....	389
Plot Type: Simple Mode and Advanced Mode .....	391
Zooming In and Out .....	393
Plotting Several Calculations in One Diagram .....	394
Plot Renderer: Convert a Plot to a Table and Export the Data .....	396
Merging Plots into One Diagram .....	398
Adding Axes to a Diagram .....	400
Rotating 3D Plots .....	402

## Performing Projects, Trees and Activities

---

You can trigger the performance of an activity (also called a *job*) and all the activities below it in the same tree in either the **Project** or **Configuration** window. In the Project window you can also perform a single activity without performing its successors or perform the whole project (all the activity trees).

In the **Configuration** window, click **Perform Tree** to perform the currently selected activity and all the activities below it. If there are no activities below the selected activity, then the button says **Perform** instead.



All activities that must be completed as prerequisites are automatically performed first. For example, if you Perform Tree for a Plot Renderer, then the predecessors, Equilibrium Calculator and System Definer, are automatically performed before the Plot Renderer.

In the Project window, you can do any of the following:

- Perform an activity: Right-click the activity and select **Perform Now**, or **Perform Later** and schedule a time to perform the activity
- Perform an activity and all activities below it in the tree: Right-click any node in the tree and select **Perform Tree Now**. Or select **Perform Tree Later** and schedule a time for all activities in the tree to be performed
- Perform a project: Right-click  **My Project** and select **Perform Now** or **Perform Later** and schedule a time to perform all project activities in the tree



If there is an error during the performance of an activity, icons display status markers, which are described in [Node or Window Status Markers](#).

## Saving and Printing the Output (Graphical Mode)



### [Global Settings: Graphical and Console Mode Plotting](#)



Output can refer to results in the form of tabulated data in a table or a diagram.



The terms *diagram* and *plot* are interchangeable. The commands in Console Mode generally use the term diagram and in Graphical Mode it is also called a plot (e.g. the Plot Renderer).

In Graphical Mode the output is displayed in the **Visualizations** window. The output can be in the form of a table or a diagram (plot). In both cases you can save from the **Configuration** or **Visualizations** windows. You can print the table or diagram from the **Visualizations** window.



Saving a Parallel coordinates plot from the Visualizations window has less file type options. See [Plot Type: Parallel Coordinates](#).

## Saving Diagrams

You can save the output in a variety of file formats. For JPG and PNG images you can also adjust the image quality.



In Console Mode, the diagram is output to the **Console Results** window after using the PLOT\_DIAGRAM command.

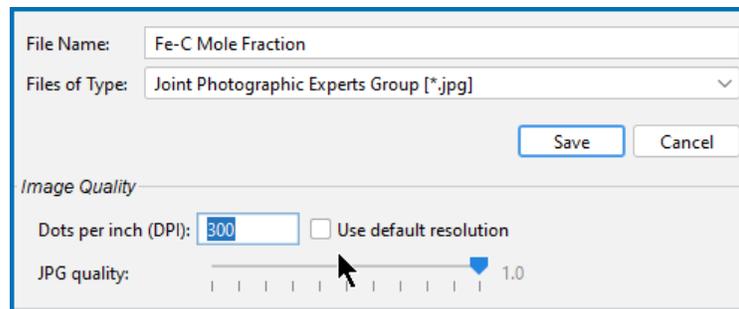


In Graphical Mode, the diagram is output to the **Visualizations** window after performing a tree that includes a Plot Renderer.

1. In the **Visualizations** window, right-click the diagram and select **Save as**.

 In the Graphical Mode **Configuration** window you can also click the **Save Diagram**  (or **Save table** ) buttons to open the **Save** window.

2. In the **Save** window:
  - a. Navigate to where you want to save the diagram.
  - b. Enter a **File name**.
  - c. From the **Files of type** list choose **png** (the default), **jpg**, **ps**, **pdf**, **gif**, **svg**, or **emf**.
  - d. For PNG and JPG files only, under the **Image Quality** section, choose to improve the resolution of the image. For JPG files you can also adjust the file size.
    - To change the resolution of a PNG or JPG image, click to clear the **Use screen resolution** checkbox. Then enter a numerical value in the **Dots per inch (DPI)** field.
    - For JPEG files, you can also use the slider to choose a number between 0 and 1 for the **JPG quality**. The default is 0.9. The lower the number, the smaller the file.



- e. When you are ready, click **Save**.

## Saving (Exporting) Tables

You can save the table output in different file formats from the [Table Renderer](#) and [Plot Renderer Configuration](#) and **Visualizations** windows.



In Graphical Mode, a table is output to the **Visualizations** window after performing a tree that includes a Table Renderer. It is also output using the **Table View**  button located on the Plot Renderer **Configuration** window.

1. Generate a table. Then depending on which window you are in:
  - In the **Visualizations** window, right-click the table and select **Save as**, or
  - In the **Configuration** window click the **Save table**  button.
2. In the **Save As** window:
  - a. Navigate to where you want to save the table.
  - b. Enter a **File Name**.
  - c. From the **Files of Type** list choose **Text [\*.txt]**(the default), **Hyper Text Markup [\*.html]**, or **Excel [\*.xls]**.
  - d. When you are ready, click **Save**.

## **Printing Output**

In the **Visualizations** window, right-click the table or diagram and choose **Print**. Select additional settings as required on the **Print** window.

## Plotting Options

---

The [Plot Renderer](#) is a dynamic activity node with options such as being able to:

- add more than one Plot Renderer to its predecessor node (i.e. one of the calculator activities),
- add axes to the plots,
- merge results in one plot,
- filter quantities (only for a Parallel coordinates plot), and
- choose from a variety of plot types based on the selected combination of Calculator and Calculation Type.



Additional settings are also described in [Configuration Settings](#).



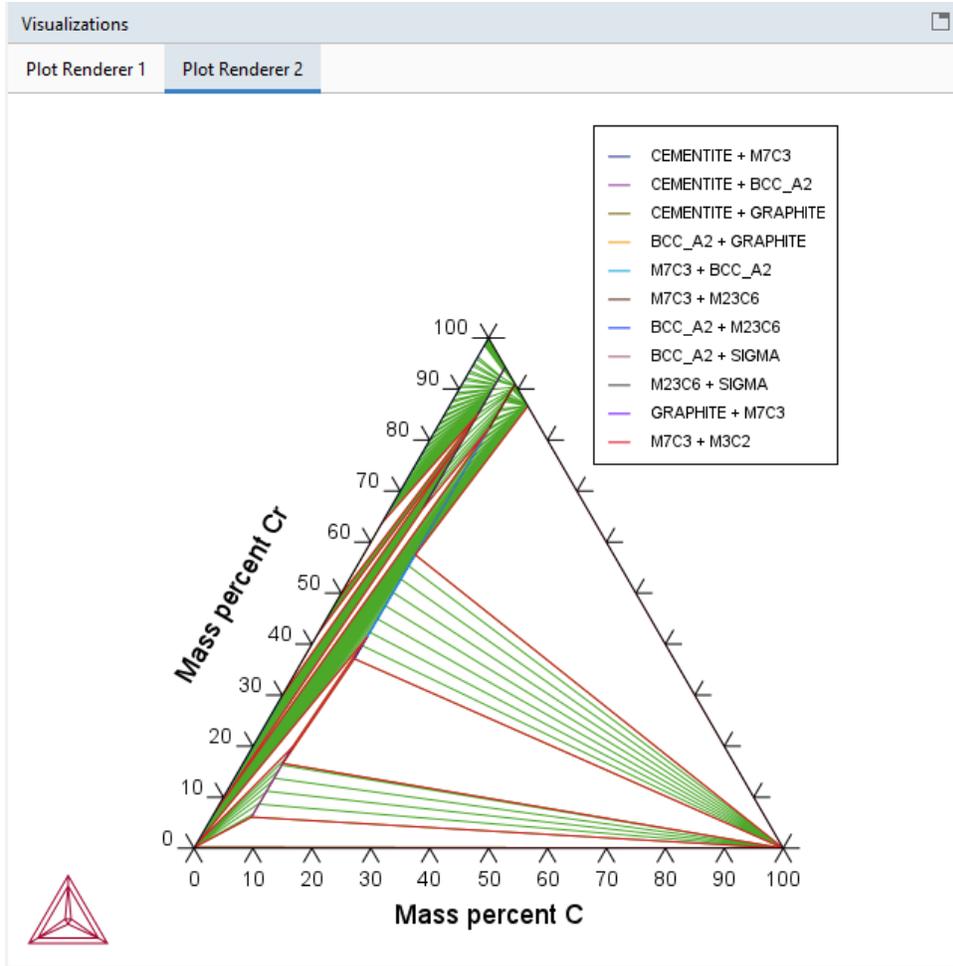
The specialized Add-on Additive Manufacturing (AM) Module has additional functionality. The information is described separately and included with the Module's documentation.

### ***Plots Can Have More Than One Predecessor***

A Plot Renderer can have several predecessors (if you want to plot several calculations in the same diagram), in which case there are several tabs in the Configuration window, one for each predecessor.



[Plotting Several Calculations in One Diagram](#)

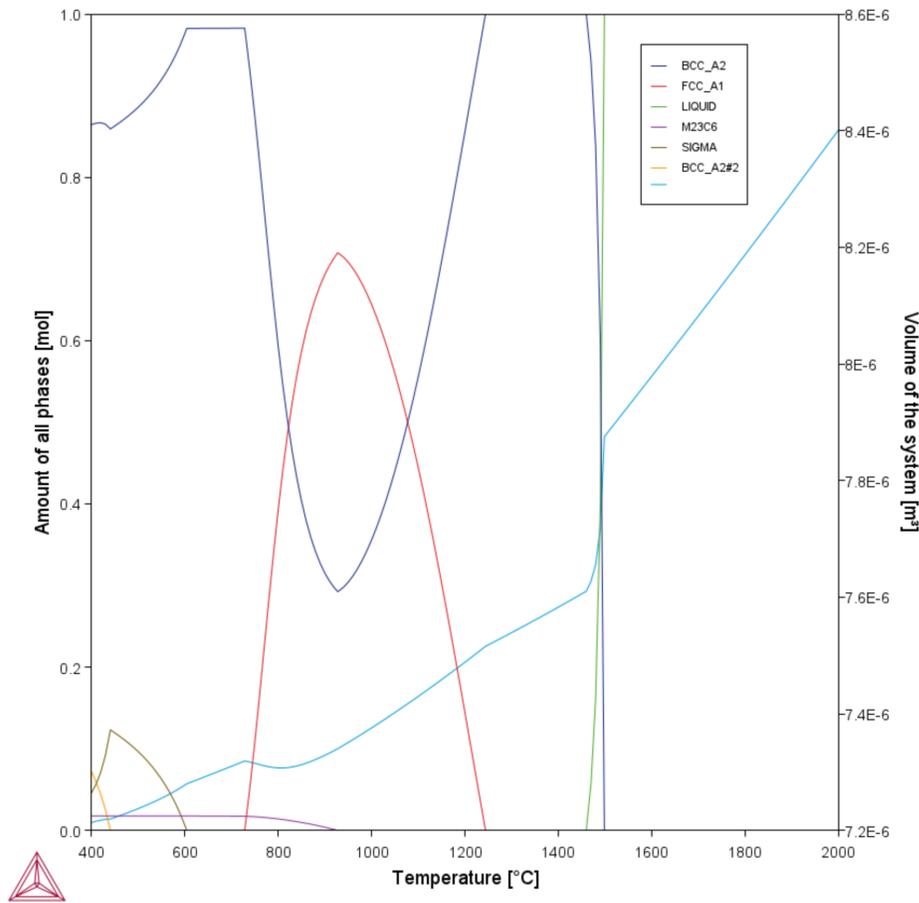


Several Plot Renderer activities (as well as Table Renderer activities) can be shown in the Visualizations window as different tabs. The figure is from Graphical Mode example T\_04.

## Add Axes to Plots

You can add additional X- and/or Y-axes to a plot to compare two or more variables at the same time as in the figure.

 [Adding Axes to a Diagram](#)



Two variables plotted on the Y-axis.

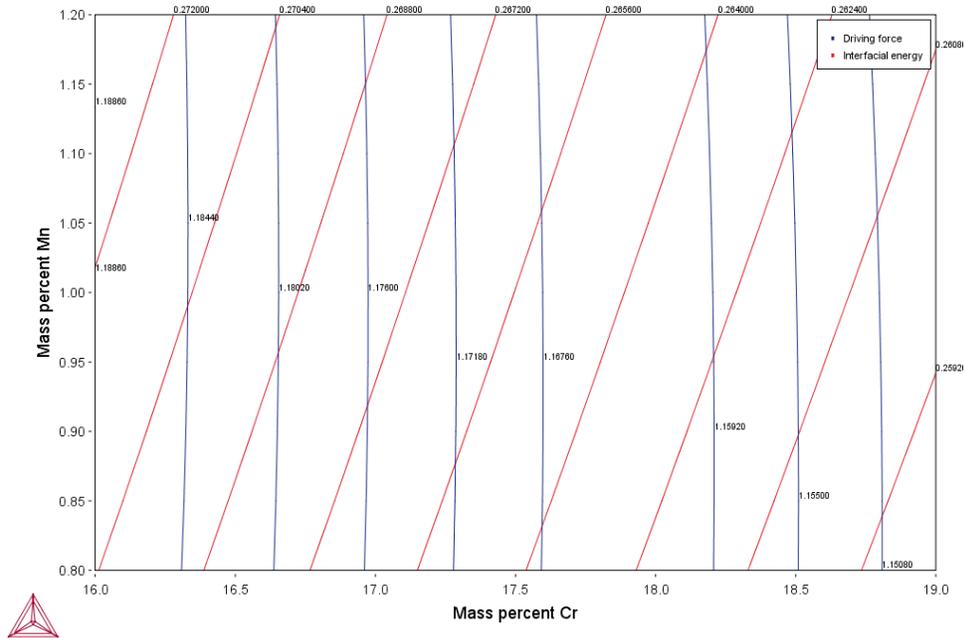
## Merge Results in One Plot

You can merge (or overlay) calculated results for several variables into one plot. This is similar to adding an axis, but instead of displaying different variables along the X- and/or Y-axes, the same X and Y variables are used to compare the third variable (defined in the Plot Renderer as the Z-axis).

As in the figure (from example PM\_G\_03 in Graphical Mode Examples), the driving force and interfacial energy results are combined into one diagram. The X- and Y-axis is the same for both (Mass percent Cr/Mass percent Mn).



[Merging Plots into One Diagram](#)



A contour plot showing both the driving force and interfacial energy calculation results comparing Mass percent Cr to Mass percent Mn.

## Choose a Plot Type

When you set up most Calculators, there are calculation types to choose from on the **Configuration** window. The selection of the *Calculation Type* also provides options to choose from the *Plot type* menu for some of the associated **Plot Renderer** windows.



For a list of calculations for each Calculator, see [The Calculation Types in Graphical Mode](#).

There are a variety of combinations of Calculator and *Calculation Type* that offer different [Plot Types](#). These are described in separate topics:

- [Plot Type: Cross Plot and Statistical](#)
- [Plot Type: Heat Map, Contour, and 3D](#)
- [Plot Type: Parallel Coordinates](#)
- [Plot Type: Flexible Mode and Grouped Mode](#)

- [Plot Type: CCT Mode](#)
- [Plot Type: TTT Mode](#)
- [Plot Type: Simple Mode and Advanced Mode](#)

### ***Filter Quantities on a Parallel Coordinates Plot***

Use the **Filter** checkbox on a Plot Renderer to add various filters to the **Parallel coordinates** plot or table and based on the set up of the calculation.

When selected, click the **Add quantity**  and **Remove quantity**  buttons to add filters. For example, enter a **Min** and **Max** and choose an axis variable such as **Composition Cr** and the unit **Mass percent** to adjust the output in the **Visualizations** window. Click **Perform** each time you add or remove a filter for the results to update.

## Common Quantities Available for Plots and Tables

The following table lists the common quantities available as axis variables or columns when working with plots and tables in Graphical Mode. All the quantities are listed alphabetically in Equilibrium Calculator conditions, quantity function definitions, and table and plot results (specifically for these types of calculations-one axis equilibrium, phase diagram, binary and ternary).



These quantities are also available in Console Mode but are entered manually.

Each quantity is selected on the **Configuration** window as required either as **Axis variables** for the Plot Renderer or when choosing the **Columns** for a Table Renderer.

<i>Quantity</i>	<i>Available Units</i>	<i>Comment or Link to More Information</i>
Activity of component	dimensionless	
Activity referred to a phase	dimensionless	
Amount of component	Mole, Gram, Kilogram, Tonne, Pound	
Amount of component in phase	Mole, Gram, Kilogram, Tonne, Pound	
Amount of phase	Mole, Gram, Kilogram, Tonne, Pound	
Bohr magneton number	dimensionless	
Chemical diffusion coefficient	m <sup>2</sup> /s	
Chemical potential of component	Joule, Calorie, Electron volt, Gigajoule, Watt hour, Kilowatt hour	
Chemical potential referred to a phase	Joule, Calorie, Electron volt, Gigajoule, Watt hour, Kilowatt hour	
Composition	Mass percent, Mole percent, Mass fraction, Mole fraction	

Quantity	Available Units	Comment or Link to More Information
Composition of a phase	Mass percent, Mole percent, Mass fraction, Mole fraction	
Curie temperature	Kelvin, Celsius, Fahrenheit	
Density	Gram per cubic meter, Kilogram per cubic meter, Gram per cubic centimeter, Pound per cubic inch	
Dynamic viscosity	Pascal-second, milliPascal-second, Poise, centiPoise	 <a href="#">About Thermophysical Properties Data Parameters</a>
Electric conductivity	dimensionless	 <a href="#">About Thermophysical Properties Data Parameters</a>
Electric resistivity	dimensionless	 <a href="#">About Thermophysical Properties Data Parameters</a>
Enthalpy	Joule, Calorie, Electron volt, Gigajoule, Watt hour, Kilowatt hour	
Entropy	Joule per Kelvin, Calorie per Kelvin, Electron volt per Kelvin	
Function	No units.	Select <b>Function</b> when you want to choose a user-defined function. For example, if <code>fraction_solid</code> is predefined defined on the <b>Functions</b> tab for an Equilibrium Calculator and you want to plot this, choose <b>Function</b> .
Gibbs energy	Joule, Calorie, Electron volt, Gigajoule, Watt hour, Kilowatt hour	
Heat capacity	Joule per mole Kelvin, Calorie per mole Kelvin, Joule per kilogram Kelvin, BTU per pound Fahrenheit	
Helmholtz energy	Joule, Calorie, Electron volt, Gigajoule, Watt hour, Kilowatt hour	
Internal energy	Joule, Calorie, Electron volt,	

Quantity	Available Units	Comment or Link to More Information
	Gigajoule, Watt hour, Kilowatt hour	
Intrinsic diffusion coefficient	dimensionless	
Isothermal compression	dimensionless	
Kinematic viscosity	Square meter per second, Stokes, Centistokes	 <a href="#">About Thermophysical Properties Data Parameters</a>
L''	dimensionless	
Mobility	dimensionless	
Normalized driving force	dimensionless	
Phase stability function	dimensionless	
Pressure	Pascal, Atmospheres, Bar	
Site fraction	dimensionless	
Surface tension	Joule per m <sup>2</sup> , Millijoule per m <sup>2</sup>	 <a href="#">About Thermophysical Properties Data Parameters</a>
System size	Mole, Gram, Kilogram, Tonne, Pound	
Temperature	Kelvin, Celsius, Fahrenheit	
Thermal conductivity	dimensionless	 <a href="#">About Thermophysical Properties Data Parameters</a>
Thermal diffusivity	dimensionless	 <a href="#">About Thermophysical Properties Data Parameters</a>
Thermal expansivity	per Kelvin, per Fahrenheit	

Quantity	Available Units	Comment or Link to More Information
Thermal resistivity	dimensionless	 <a href="#">About Thermophysical Properties Data Parameters</a>
Thermodynamic factor	dimensionless	
Tracer diffusion coefficient	dimensionless	
User-defined		Enter another axis variable.
Volume	Cubic meter, Cubic decimeter, Cubic centimeter, Normal cubic meter	
Volume fraction of phase		
ln(activity of component)		
ln(activity referred to a phase)		
log10(Chemical diffusion coefficient)		
log10(Intrinsic diffusion coefficient)		
log10(Mobility)		
log10(Tracer diffusion coefficient)		
u-fraction		 <a href="#">The u-Fraction Variable</a>

## Plot Types

---

The [Plot Renderer](#) is a dynamic activity node with options such as being able to choose from a variety of *Plot types* based on the Calculator and, in most cases, *Calculation Type* selected.



The *Mode* plot types are related to groups of settings made available (or hidden) on the Plot Renderer, rather than a particular plot output type such as contour, cross plot, or heat map, for example.

The tables below list the *Plot types* available with the various Calculator and *Calculation Type* combinations, either [By Calculator](#), [By Plot Type](#), or [By Calculation Type](#).



### [The Calculation Types in Graphical Mode](#)

There are a variety of combinations of Calculator and *Calculation Type* that offer different [Plot Types](#). These are described in separate topics:

- [Plot Type: Cross Plot and Statistical](#)
- [Plot Type: Heat Map, Contour, and 3D](#)
- [Plot Type: Parallel Coordinates](#)
- [Plot Type: Flexible Mode and Grouped Mode](#)
- [Plot Type: CCT Mode](#)
- [Plot Type: TTT Mode](#)
- [Plot Type: Simple Mode and Advanced Mode](#)



The AM Calculator plotting is more advanced and is described separately. Go to [AM Calculator Plot Renderer](#) as a starting point.



The following require additional licenses: Diffusion Calculator, Process Metallurgy Calculator, CCT Mode, and TTT Mode.

## By Calculator

<i>Calculator</i>	<i>Calculation Type</i>	<i>Plot Types</i>	<i>Notes</i>
Diffusion Calculator	n/a	Simple Mode; Advanced Mode	There are no <i>Calculation Types</i> to choose when setting up a Diffusion Calculator.
Equilibrium Calculator	One Axis Phase diagram	Flexible Mode Grouped Mode	
	Grid	Heat Map Contour 3D	
Material to Material Calculator	One Axis Phase diagram	Flexible Mode Grouped Mode	
Process Metallurgy Calculator (Equilibrium calculations)	One Axis	Flexible Mode Grouped Mode	
	Grid	Heat Map Contour 3D	
Process Metallurgy Calculator (Process simulation)	n/a	Flexible Mode Grouped Mode	There are no <i>Calculation Types</i> to choose when setting up a Process Metallurgy Calculator as a <i>Process simulation</i> .
Property Model Calculator	One Axis	Flexible Mode Grouped Mode	
		CCT Mode	Also select <b>Log10(cooling rate [K/s])</b> as the axis <i>Quantity</i> .
		TTT Mode	Also select <b>Temperature</b> as the axis <i>Quantity</i> .
	Grid	Heat Map Contour 3D	
		Parallel coordinates	At least three quantities under <i>Grid Definitions</i> must be selected on the Property Model

Calculator	Calculation Type	Plot Types	Notes
			Calculator; then this plot is available on the Plot Renderer.
	Uncertainty	Cross Plot Parallel coordinates Statistical	
		Statistical - Histogram	Choose <b>Frequency</b> as the <i>Axis variable</i> and <b>Histogram</b> as the <i>Axis type</i>
		Statistical - Probability	Choose <b>Normal probability</b> as the <i>Axis variable</i> and <b>Probability</b> as the <i>Axis type</i>
	Batch	Statistical Cross Plot Parallel coordinates	
Ternary Calculator	Isothermal section Monovariant lines Liquidus projection	Flexible Mode Grouped Mode	

## By Plot Type

Plot Type	Calculation Type	Calculator(s)	Notes
3D	Grid	Equilibrium Calculator Property Model Calculator Process Metallurgy Calculator	 <p>3D Plots with additional functionality are also available with the Add-on Additive Manufacturing (AM) Module. The information is described separately and included with the Module's documentation.</p>
Advanced Mode	n/a	Diffusion Calculator	There are no <i>Calculation Types</i> to choose when setting up a Diffusion Calculator.

<i>Plot Type</i>	<i>Calculation Type</i>	<i>Calculator(s)</i>	<i>Notes</i>
CCT Mode	One Axis	Property Model Calculator	Also select <b>Log10(cooling rate [K/s])</b> as the axis <i>Quantity</i> .
Contour	Grid	Equilibrium Calculator Property Model Calculator Process Metallurgy Calculator	
Cross Plot	Batch Uncertainty	Property Model Calculator	
Flexible Mode or Grouped Mode	One Axis	Equilibrium Calculator Material to Material Calculator Process Metallurgy Calculator (Equilibrium calculation) Property Model Calculator	
	Isothermal section Monovariant lines Liquidus projection	Ternary Calculator	
	n/a	Process Metallurgy (Process simulation)	There are no <i>Calculation Types</i> to choose when setting up a Process Metallurgy Calculator as a <i>Process simulation</i> .
	Phase diagram	Equilibrium Calculator Material to Material Calculator	
Heat Map	Grid	Equilibrium Calculator Property Model	

<i>Plot Type</i>	<i>Calculation Type</i>	<i>Calculator(s)</i>	<i>Notes</i>
		Calculator Process Metallurgy Calculator	
Parallel coordinates	Batch Grid Uncertainty	Property Model Calculator	For the <b>Grid</b> calculation at least three quantities under <i>Grid Definitions</i> must be selected on the Property Model Calculator, then this plot type is available on the Plot Renderer.
Simple Mode	n/a	Diffusion Calculator	There are no <i>Calculation Types</i> to choose when setting up a Diffusion Calculator.
Statistical	Batch Uncertainty	Property Model Calculator	
Statistical - Histogram	Batch Uncertainty	Property Model Calculator	Choose <b>Frequency</b> as the <i>Axis variable</i> and <b>Histogram</b> as the <i>Axis type</i> Batch is only available with the Property Model Calculator
Statistical - Probability	Batch Uncertainty	Property Model Calculator	Choose <b>Normal probability</b> as the <i>Axis variable</i> and <b>Probability</b> as the <i>Axis type</i> Batch is only available with the Property Model Calculator
TTT Mode	One Axis	Property Model Calculator	Also select <b>Temperature</b> as the axis <i>Quantity</i> .

## By Calculation Type

<i>Calculation Type</i>	<i>Plot Types</i>	<i>Calculator (s)</i>	<i>Notes</i>
Batch	Statistical Parallel coordinates	Property Model Calculator	
Grid	Heat Map Contour 3D	Equilibrium Calculator Property Model Calculator	

Calculation Type	Plot Types	Calculator (s)	Notes
		Process Metallurgy Calculator	
	Parallel coordinates	Property Model Calculator	At least three quantities under <i>Grid Definitions</i> must be selected on the Property Model Calculator, then the plot is available on the Plot Renderer.
Isothermal section Liquidus projection Monovariant lines	Flexible Mode Grouped Mode	Ternary Calculator	
Min/Max	Flexible Mode Grouped Mode	Property Model Calculator	
One Axis	Flexible Mode Grouped Mode	Equilibrium Calculator Material to Material Calculator Process Metallurgy Calculator Property Model Calculator	
	CCT Mode TTT Mode	Property Model Calculator	CCT Mode: Also select <b>Log10(cooling rate [K/s])</b> as the axis <i>Quantity</i> . TTT Mode: Also select <b>Temperature</b> as the axis <i>Quantity</i> .
Phase diagram	Flexible Mode Grouped Mode	Equilibrium Calculator Material to Material Calculator	
Uncertainty	Parallel coordinates	Property Model Calculator	
	Statistical		

---

<i>Calculation Type</i>	<i>Plot Types</i>	<i>Calculator (s)</i>	<i>Notes</i>
	Statistical - Histogram		Choose <b>Frequency</b> as the <i>Axis variable</i> and <b>Histogram</b> as the <i>Axis type</i>
	Statistical - Probability		Choose <b>Normal probability</b> as the <i>Axis variable</i> and <b>Probability</b> as the <i>Axis type</i>

## Plot Type: Cross Plot and Statistical

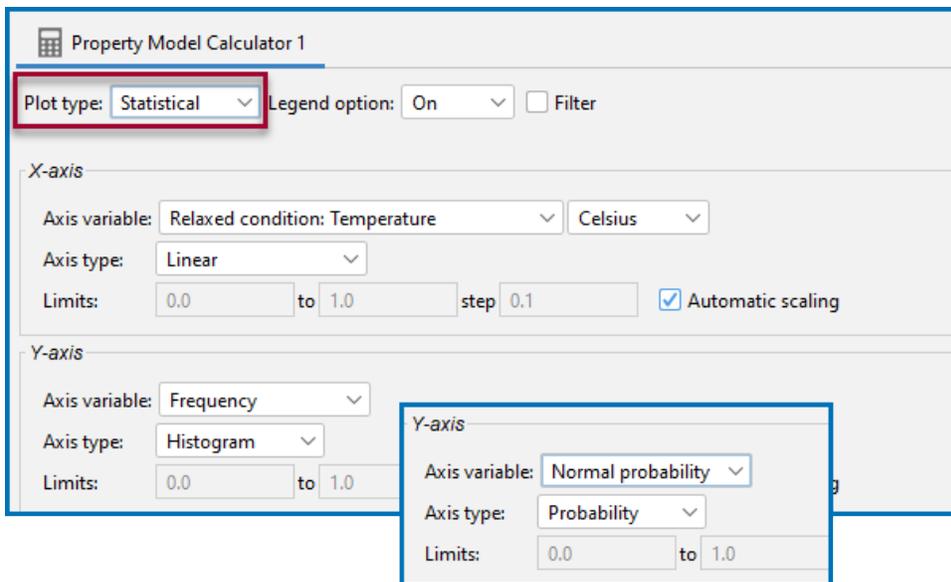
The [Plot Renderer](#) is a dynamic activity node with options such as being able to choose from a variety of *Plot types* based on the Calculator and Calculation Type selected.

- ❓ [The Calculation Types in Graphical Mode](#)
- ❓ See [Plot Types](#) detailing availability *By Calculator*, *By Plot Type*, or *By Calculation Type*.

### Statistical (with a Histogram or Probability Axis Type)

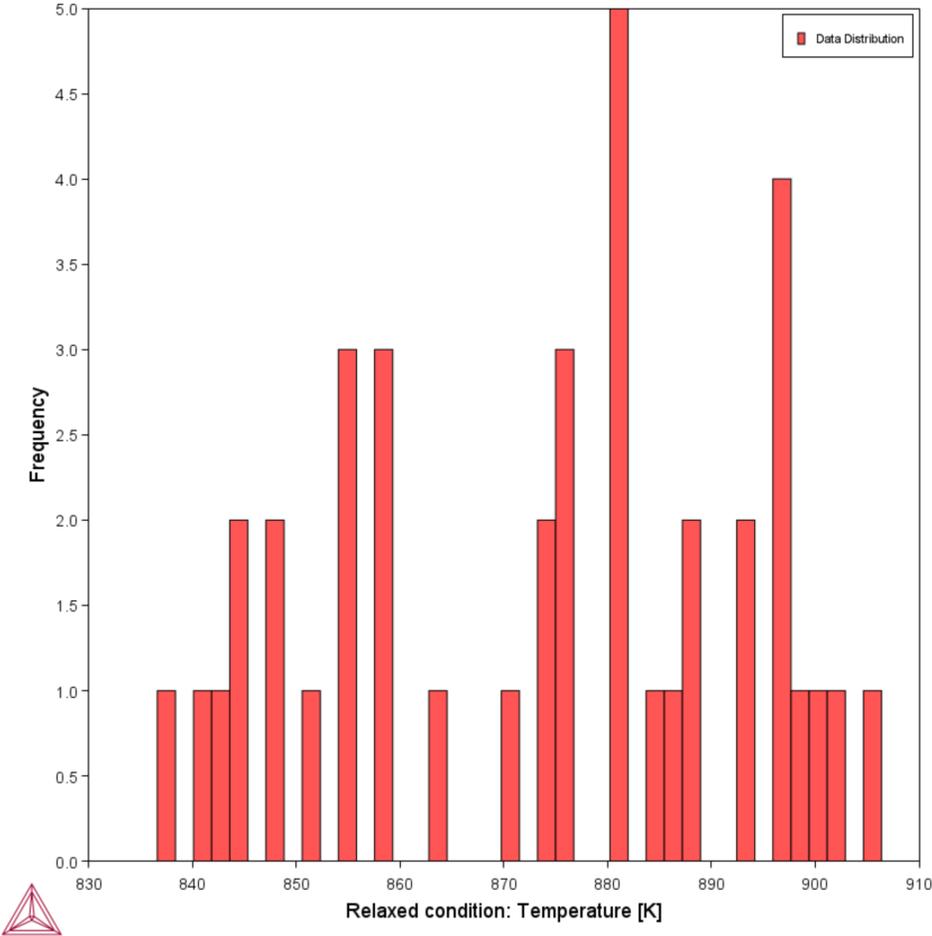
Use a **Statistical** *Plot type* to select **Frequency** or **Normal probability** as the *Axis variable*. The *Axis type* is then set to **Histogram** or **Probability**, respectively.

 The following uses a General Property Model example: [PM\\_G\\_01: Phase Transition](#).

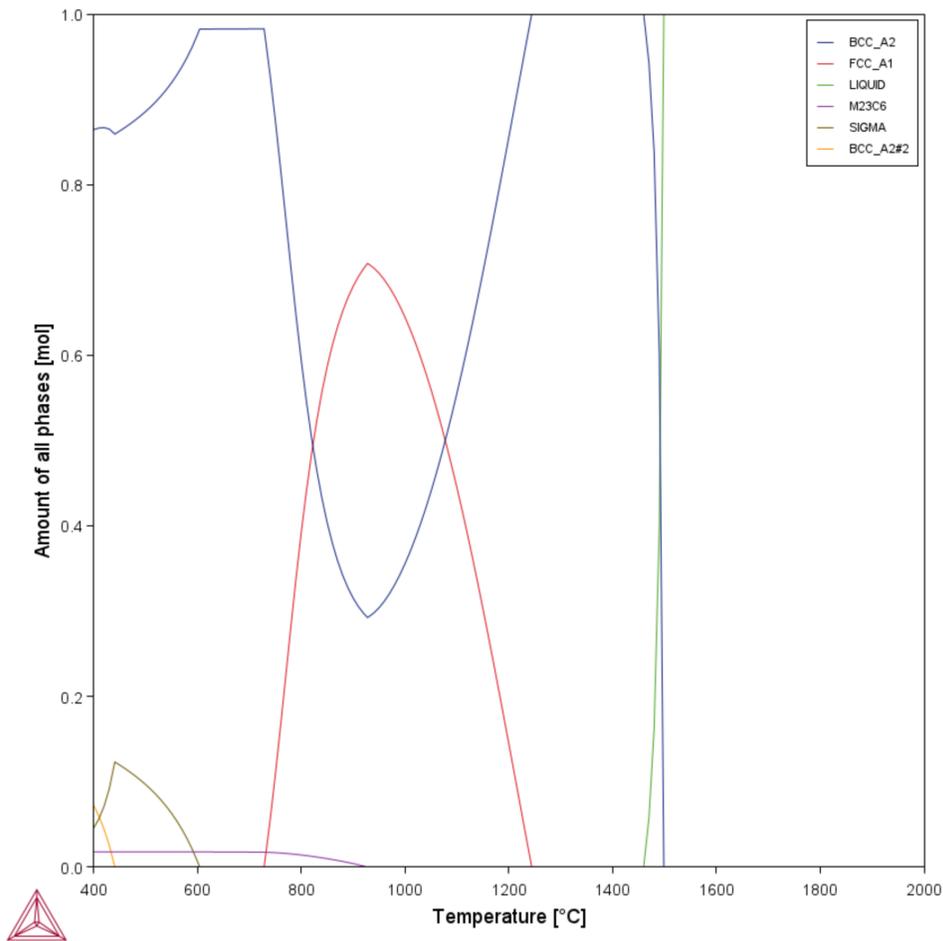


The screenshot shows the 'Property Model Calculator 1' interface. The 'Plot type' dropdown is set to 'Statistical' and is highlighted with a red box. Below it, the 'X-axis' section is configured with 'Axis variable: Relaxed condition: Temperature', 'Axis type: Linear', and 'Limits: 0.0 to 1.0 step 0.1'. The 'Y-axis' section is configured with 'Axis variable: Frequency' and 'Axis type: Histogram'. A blue box highlights a secondary 'Y-axis' configuration with 'Axis variable: Normal probability' and 'Axis type: Probability'.

*The two combination options, Frequency → Histogram or Normal probability → Probability, selected with Statistical on a Plot Renderer as a successor to a Property Model Calculator.*



A Statistical Plot type, with the Frequency Axis variable and Histogram Axis type selected.



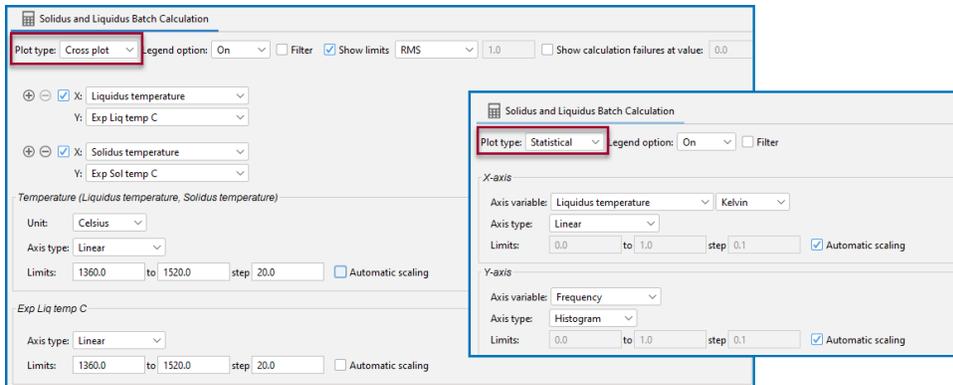
A Statistical Plot type, with the Normal probability Axis variable and Probability Axis type selected.

## Cross Plot (Parity Plot)

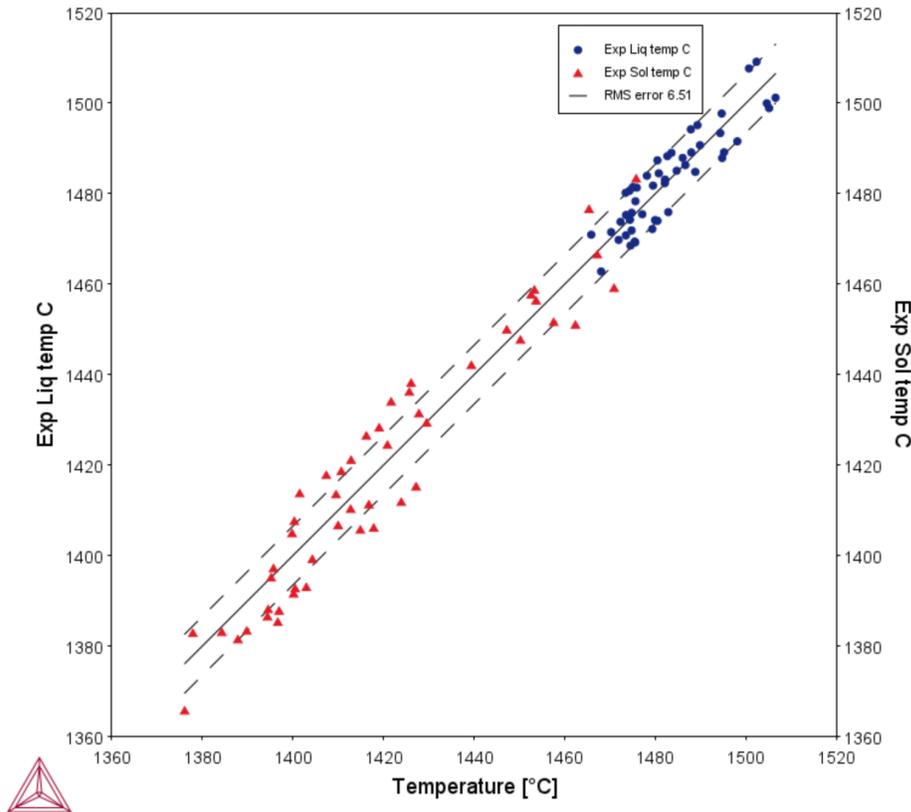
Use **Cross plot** to plot the experimental data from a Batch calculation with the calculated result, e.g. you might be looking at liquidus temperature and now can plot both to compare the experimental data against the Thermo-Calc calculated results. You choose the experimental data value (which is read from the data file) as an X- or Y-axis variable and then the calculated result as the other X- or Y-axis variable. This is also known as a *parity* plot.



The following uses a General Property Model example: [PM\\_G\\_12: Solidus and Liquidus Batch Calculation](#).



*For a Batch calculation, and when the data file contains experimental (Exp) data, you can use Cross plot on a Plot Renderer to compare it to the simulated results.*



*A comparison of experimental liquidus and solidus data using a Cross plot with a Batch calculation. A root mean square (RMS) setting shows the distribution of the data points.*



There is a blog post related to [ICME Workflows](#) that provides an additional example about the use of cross plots.



Another example showing the use of cross plots is [PM\\_G\\_03: Driving Force and Interfacial Energy](#), which also has a video to accompany it.

## Plot Type: Parallel Coordinates

---

The [Plot Renderer](#) is a dynamic activity node with options such as being able to choose from a variety of *Plot types* based on the Calculator and Calculation Type selected.

 [The Calculation Types in Graphical Mode](#)

 See [Plot Types](#) detailing availability *By Calculator, By Plot Type, or By Calculation Type*.

The **Parallel coordinates** plot type is available when using the Property Model Calculator with a **Grid, Batch, or Uncertainty Calculation Type**.

The **Parallel coordinates** plot is useful when interpreting multidimensional data and to compare how different parameters affect each other. This is especially useful in materials design after performing a batch or uncertainty set of evaluations when multiple inputs are varied at the same time and multiple model outputs are given as a result.

For this plot type, there are some differences compared to other plots as to what can be done on the **Visualizations** window and the **Plot Renderer Configuration** window.

- **Visualizations** window: After a plot is generated, the options available when you right-click are slightly different. You can choose to **Select/Deselect a line** (to display additional information), and to **Save As** and **Copy** plot images.



Other options such as Plot Themes and plot formatting options are not available.

- **Plot Renderer Configuration** window: You can select the **Filter** checkbox and then filter one or more quantities.



To access this plot type on the Plot Renderer for a **Grid** calculation, at least three quantities under *Grid Definitions* must be selected on the Property Model Calculator.

## Tips to Interpret the Plot Data

- When most lines between two parallel axes are somewhat parallel to each other, there is a positive relationship between these two dimensions.
- When lines cross in a kind of superposition of X-shapes, that's a negative relationship.
- When lines cross randomly it means that there is no particular relationship.

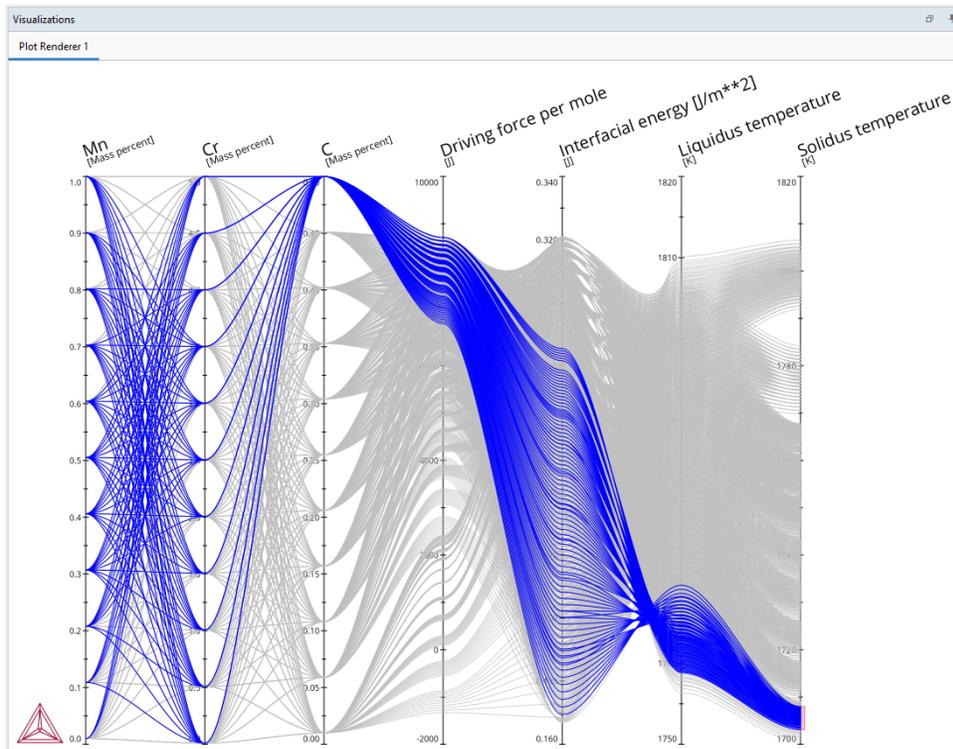


Figure 17: Parallel coordinates plot showing multidimensional data.

## Visualization Window Plot Options

Once the plot is generated in the Visualizations window, you can right-click and choose one of these options:

- **Select/Deselect a line** (or Ctrl + right-click). For this option, you must right-click directly on an axis line to view the details and highlight the relevant information.
- **Save As** to save the plot (in its current state) as a **jpg**, **png**, or **gif** file.
- **Copy** to copy the plot (in its current state) to a clipboard. Then you can paste it into any supported application. The pasted image defaults to a **png** format.



For both **Save As** and **Copy** actions, the image is saved in the current state that the both the plot and Visualizations window is in, i.e. if you have selected one or more lines to display information, then this is what is included in the saved file. It is also saved based on the current size of the Visualizations window, i.e. to ensure that the entire plot is saved, you may need to expand the window to the full view you want to capture. Sometimes it is also useful to detach the window and drag it to full size before saving the image.



For an example using this plot type, see the Nickel Model Library example [PM\\_Ni\\_04: Strain Age Cracking \(SAC\)](#).

### *Select and Deselect Lines*

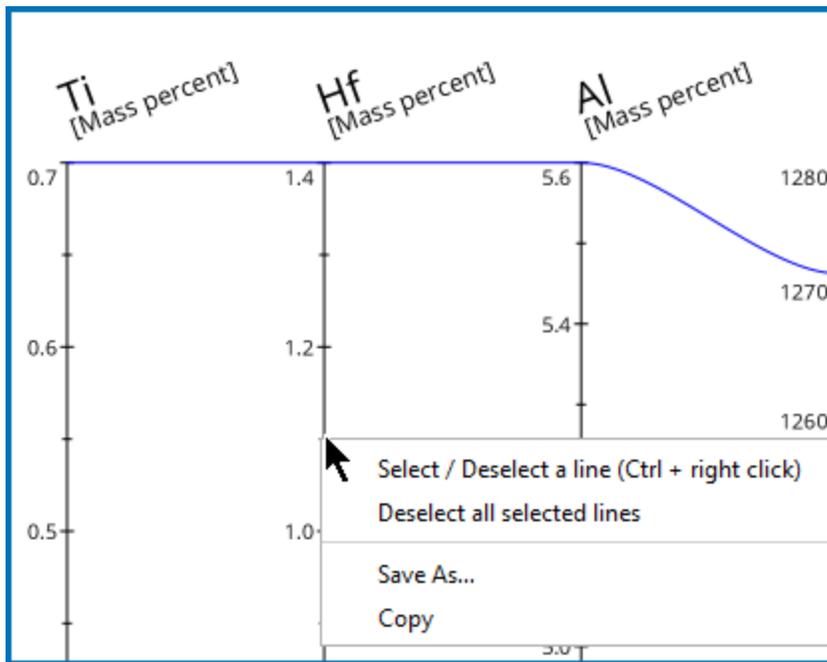
In the **Visualizations** window, you can highlight and display details about any plot axis, which is useful particularly with complex plots.



You must directly right-click the axis line to turn it on (select) or off (deselect); it does not work to just right-click anywhere in the Visualizations window for this action.

To toggle the **Select/Deselect a line** option on (select a line) and off (deselect a line), hover over an axis line then:

- Ctrl + right-click or
- Right-click and choose **Select/Deselect a line**.



To deselect all lines, either do this one at a time or when several lines are selected, you can right-click to choose **Deselect all selected lines**.

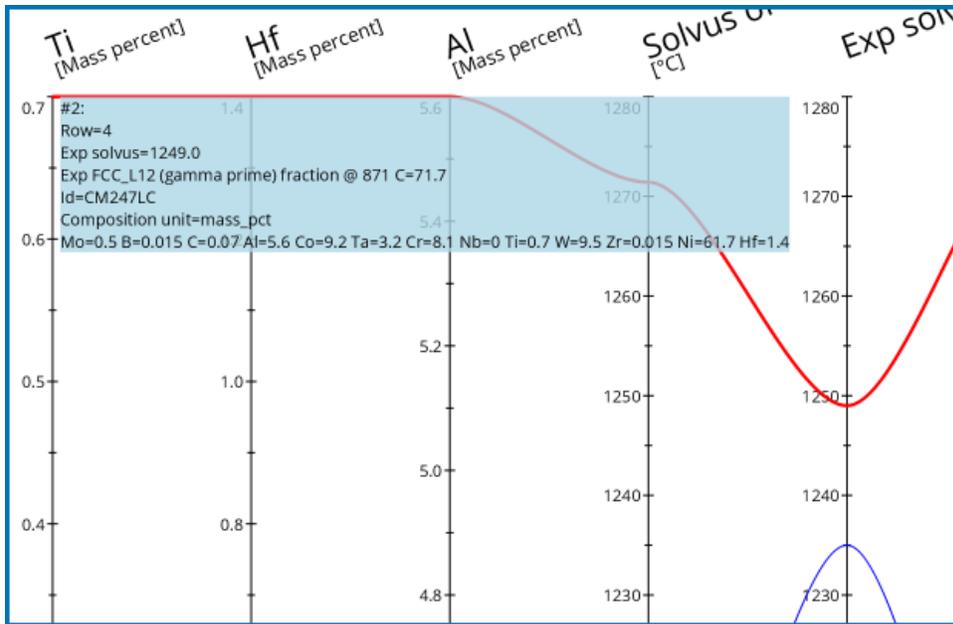


Figure 18: Choosing to display specific axes lines and the associated information on the Visualizations window for a Parallel coordinates plot type.

### Save As

1. Once the plot is generated in the Visualizations window, right-click anywhere in the window and choose **Save As**.
2. In the **Save** window:
  - a. Navigate to where you want to save the plot.
  - b. Enter a **File Name**.
  - c. From the **Files of Type** list choose **png** (the default), **jpg**, or **gif**.
  - d. When you are ready, click **Save**.

### Copy

1. Once the plot is generated in the Visualizations window, right-click anywhere in the window and choose **Copy**.
2. Navigate to where you want to paste the image (i.e. Word, Slack, email, and so forth) and Ctrl + V to paste it. The pasted image defaults to a **png** format.

---

## Plot Renderer Configuration Window Options

### *Filter Quantities*

Use the **Filter** checkbox on a Plot Renderer to add various filters to the **Parallel coordinates** plot or table and based on the set up of the calculation.

When selected, click the **Add quantity**  and **Remove quantity**  buttons to add filters. For example, enter a **Min** and **Max** and choose an axis variable such as **Composition Cr** and the unit **Mass percent** to adjust the output in the **Visualizations** window. Click **Perform** each time you add or remove a filter for the results to update.

## Plot Type: Heat Map, Contour, and 3D

---

The [Plot Renderer](#) is a dynamic activity node with options such as being able to choose from a variety of *Plot types* based on the Calculator and Calculation Type selected.



[The Calculation Types in Graphical Mode](#)



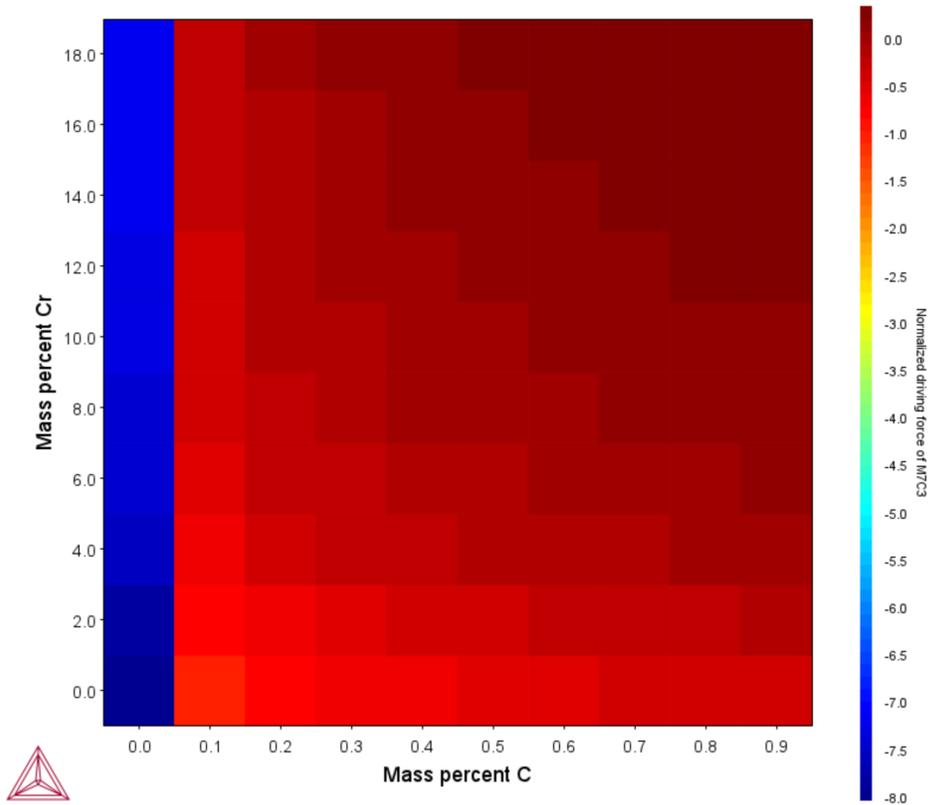
See [Plot Types](#) detailing availability *By Calculator, By Plot Type, or By Calculation Type*.



Visit the website [Application Examples → Process Metallurgy](#) for examples of these Plot types using the Process Metallurgy Calculator.

### Heat Map

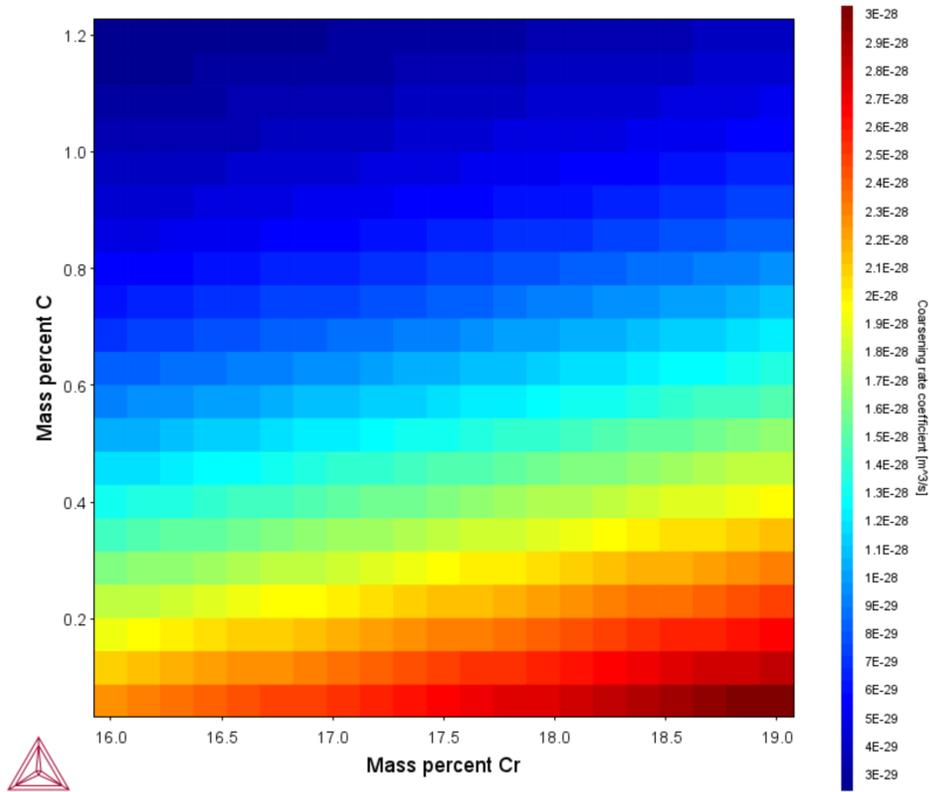
For a **Heat map** plot, each Z-axis variable value in the grid is represented by a color-coded area, where dark red represents the highest Z-axis variable value and dark blue represents the lowest Z-axis variable value.



Thermo-Calc example T\_09 uses an Equilibrium Calculator with a Grid Calculation Type to generate a Heat map plot.



[T\\_09: Carbide Driving Force Heat Map](#)



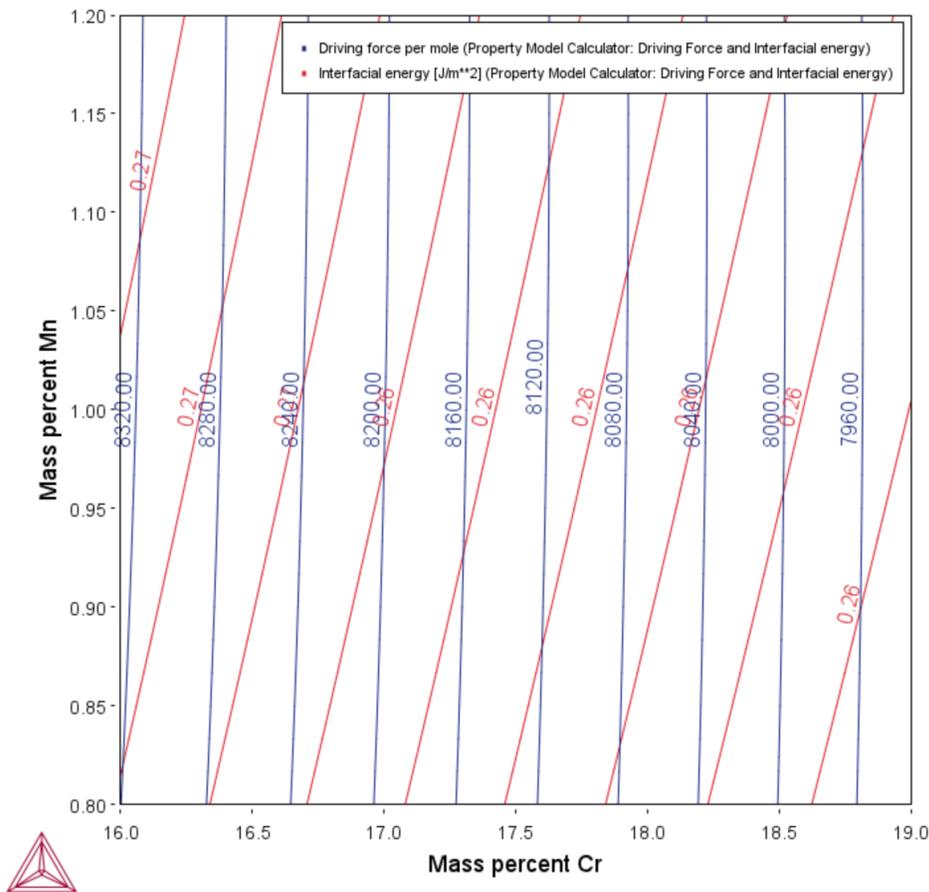
Thermo-Calc General Property Model example *PM\_G\_02* uses a Property Model Calculator with a Grid Calculation type to plot the coarsening rate coefficient as a Heat map.



[PM\\_G\\_02: Coarsening and Interfacial Energy](#)

## Contour

For a **Contour** plot, Z-axis variable values are traced with curves in the same way that height values are traced with curves on a topographical map. The colors used originate from the different data series.



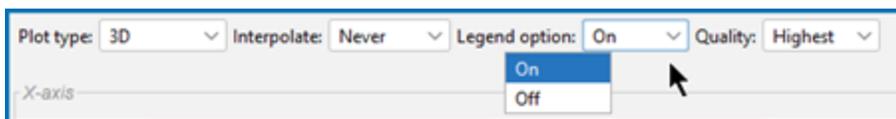
Thermo-Calc General Property Model example PM\_G\_03 uses a Property Model Calculator with a Grid Calculation type to compare the driving force and interfacial energy on an overlaid contour plot.



### [PM\\_G\\_03: Driving Force and Interfacial Energy](#)

## 3D

For a **3D** plot, the 3D surface is created by Delaunay tessellation of the calculated data. The surface uses a cold to warm color map where red represents the highest Z-axis variable value and blue represents the lowest Z-axis variable value. The color map color legend can be turned on or off with the legend option.

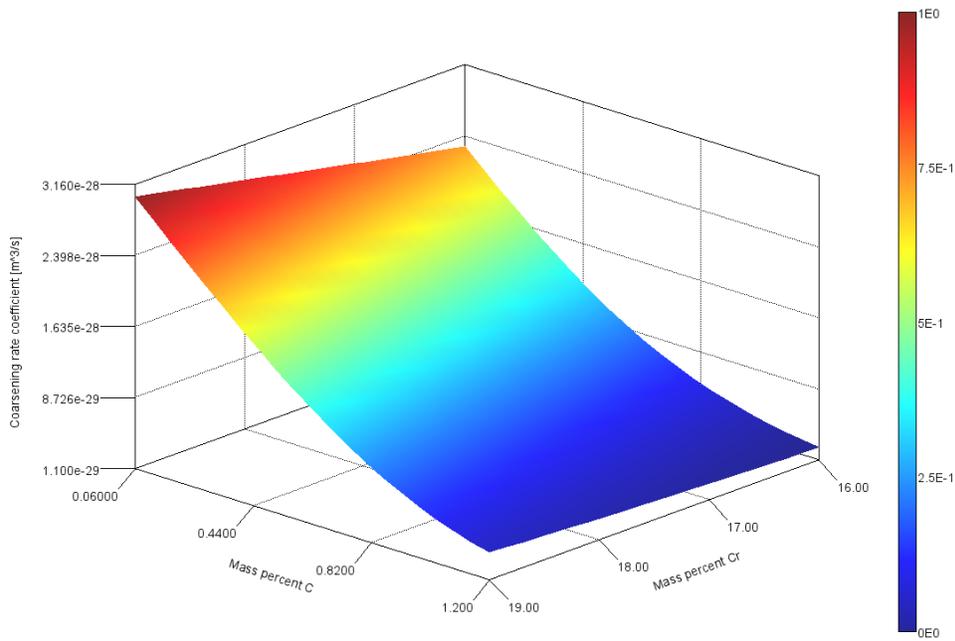




3D Plots with additional functionality are also available with the Add-on Additive Manufacturing (AM) Module. The information is described separately and included with the Module's documentation.



When in the Thermo-Calc help (press F1), go to [Rotating 3D Plots](#) to watch a short video showing you how to rotate this plot in the Visualizations window.



*Thermo-Calc General Property Model example PM\_G\_02 uses a Property Model Calculator with a Grid Calculation type to plot the coarsening rate coefficient as a 3D plot.*



[PM\\_G\\_02: Coarsening and Interfacial Energy](#)

---

## Plot Type: Flexible Mode and Grouped Mode

---

The [Plot Renderer](#) is a dynamic activity node with options such as being able to choose from a variety of *Plot types* based on the Calculator and Calculation Type selected.



[The Calculation Types in Graphical Mode](#)



See [Plot Types](#) detailing availability *By Calculator, By Plot Type, or By Calculation Type*.

### Flexible Mode

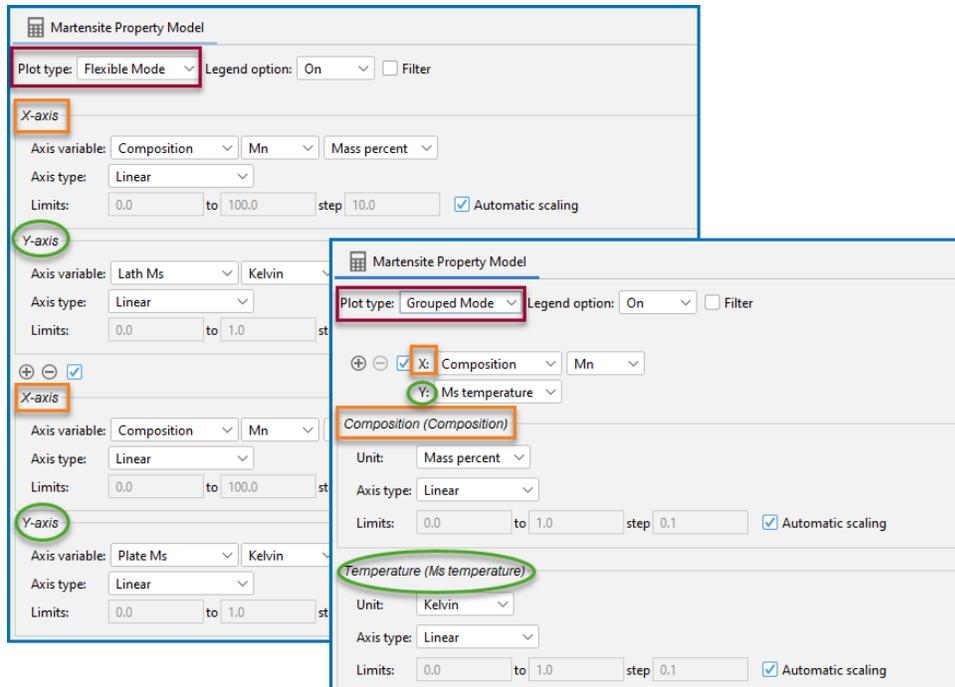
Use **Flexible Mode** to individually add and configure axes quantities and change the units or other associated settings one at a time.

### Grouped Mode

Use **Grouped Mode** to add and configure groups of axis quantities at the same time and of the same type. For example, change units or settings in groups with similar settings like temperature units or axis types.

### Switching Between Modes

If you have a license for the Steel Model Library, you can try yourself to switch between Flexible Mode and Group Mode and then change the settings to how you want it using the **Steels** example [PM\\_Fe\\_02: Fe-Mn Martensite Morphologies](#).



Comparing Flexible Mode (left) to Grouped Mode (right) on a Plot Renderer as a successor to a Property Model Calculator. The advantage of Grouped Mode is that you can, for example, configure or edit settings in one place, making it easier to manage and work with the plot quantities.



To experiment using the **Equilibrium Calculator**, see Thermo-Calc example [T\\_02: Stepping in Temperature in the Fe-C System](#).



To experiment using the **Material to Material Calculator**, see Thermo-Calc example [T\\_14: Fe-Cr-Ni Transition Comparison Using the Material to Material Calculator](#).

## Plot Type: CCT Mode

---

The [Plot Renderer](#) is a dynamic activity node with options such as being able to choose from a variety of *Plot types* based on the Calculator and Calculation Type selected.

 [The Calculation Types in Graphical Mode](#)

 See [Plot Types](#) detailing availability *By Calculator, By Plot Type, or By Calculation Type*.

Use **CCT Mode** to make a CCT diagram from a **One Axis** calculation with a *Quantity* of **Log10 (cooling rate [K/s])** selected for the CCT Diagram Property Model. The X- and Y-axes are *Time* and *Temperature*, respectively. Result quantities defined as time-temperature pairs are identified and plotted. Cooling curves are also automatically added in the plot.



This plot type is available with the Steel Model Library, which requires specific maintenance and database licenses. See [About the Steel Model Library Property Models](#).

In general, this plot type is available from the drop-down list in these situations:

- When you add the **Steel - CCT** template (on the  **My Projects Configuration** window under **Property Models**), it is the default type on the Plot Renderer (renamed CCT in the template).
- If a user-defined Property Model has both **Time** and **Temperature** quantities defined in a coupled manner, for example if you create your own Property Model with TC-Python. That is, if you select or create any model where the resulting combination of stepping-quantity and result-quantities has both time and temperature definitions.

 [Configuration Window Settings](#) and [Configuration Settings](#)

 [Steel Library CCT Diagram Template](#)

The settings example below uses Property Model example [PM\\_Fe\\_08: CCT Diagram for Fe-C-Mn-Si-Cr-V](#).

CCT calculator

Plot type: CCT Mode Legend option: On Filter

Ferrite start (2%) (T)

Pearlite start (2%) (T)

Bainite start (2%) (T)

Martensite start (T)

Austenite transformed 50% (T)

Austenite transformed 98% (T)

Temperature (Ferrite start (2%) (T), Pearlite start (2%) (T), Bainite start (2%) (T), Martensite start (T), Austenite transformed 50% (T), Austenite transformed 98% (T))

Unit: Celsius

Axis type: Linear

Limits: 0.0 to 1000.0 step 100.0 Automatic scaling

Time (Ferrite start (2%) (t), Pearlite start (2%) (t), Bainite start (2%) (t), Martensite start (t), Austenite transformed 50% (t), Austenite transformed 98% (t))

Unit: Seconds

Axis type: Logarithmic 10

Limits: 1.0 to 1000000.0 step 1.0 Automatic scaling

*CCT Mode on a Plot Renderer as a successor to a Property Model Calculator.*

## Plot Type: TTT Mode

The [Plot Renderer](#) is a dynamic activity node with options such as being able to choose from a variety of *Plot types* based on the Calculator and Calculation Type selected.

 [The Calculation Types in Graphical Mode](#)

 See [Plot Types](#) detailing availability *By Calculator, By Plot Type, or By Calculation Type*.

Use **TTT Mode** to make a TTT diagram from a **One Axis** calculation with temperature. The X- and Y-axes are *Time* and *Temperature*, respectively. Time quantities (such as *Pearlite start*, *Bainite start*) are plotted as functions of temperature. Temperature quantities (such as *Martensite start*, *Martensite 50%*) are independent of time and plotted as horizontal lines.



This plot type is available with the Steel Model Library, which requires specific maintenance and database licenses. See [About the Steel Model Library Property Models](#).

In general, this plot type is available from the drop-down list in these situations:

- When you add the **Steel - TTT** template (on the  **My Projects Configuration** window under **Property Models**), it is the default type on the Plot Renderer (renamed TTT in the template).
- When you are using at least one of these Steel Property Models—**Martensite Temperatures**, **Pearlite**, or **Bainite** with a **One Axis** calculation and when both time and temperature quantities are selected on the Plot Renderer—the TTT Mode is available.
- If a user-defined Property Model has both time and temperature quantities defined in a coupled manner, for example if you create your own Property Model with TC-Python. That is, if you select or create any model where the resulting combination of stepping-quantity and result-quantities has both time and temperature definitions.

 [Configuration Window Settings](#) and [Configuration Settings](#)

 [Steel Library TTT Diagram Template](#)

The settings example below uses Property Model example [PM\\_Fe\\_06: TTT Diagram for Fe-C-Mn-Si-Cr-V](#).

TTT calculator

Plot type: **TTT Mode** Legend option: On  Filter

Ferrite start (2%)

Pearlite start (2%)

Bainite start (2%)

Austenite transformed 50%

Martensite 50%

Martensite 98%

Temperature (Martensite start, Martensite 50%, Martensite 98%, Temperature)

Unit: Celsius

Axis type: Linear

Limits: 0.0 to 1000.0 step 100.0  Automatic scaling

Time (Ferrite start (2%), Pearlite start (2%), Bainite start (2%), Austenite transformed 50%, Austenite transformed 98%)

Unit: Seconds

Axis type: Logarithmic 10

Limits: 1.0 to 1000000.0 step 1.0  Automatic scaling

*TTT Mode on a Plot Renderer as a successor to a Property Model Calculator.*

## Plot Type: Simple Mode and Advanced Mode

The [Plot Renderer](#) is a dynamic activity node with options such as being able to choose from a variety of *Plot types* based on the Calculator and Calculation Type selected.



[The Calculation Types in Graphical Mode](#) and [Plot Types](#)



**Simple Mode** and **Advanced Mode** as a *Plot type* are only available for the Diffusion Module (DICTRA) in Graphical Mode. Unlike many other Calculators, the Diffusion Calculator does not require the selection of a *Calculation Type* before a Plot Type is available on the [Diffusion Calculator Plot Renderer](#).

### Simple Mode

Use **Simple Mode** to define the most common axis variables. The software automatically sets the correct **Plot condition** and **Independent variable**.



For one example using Simple Mode, see [D\\_04: Fe-C Moving Boundary: Austenite to Ferrite](#).

### Advanced Mode

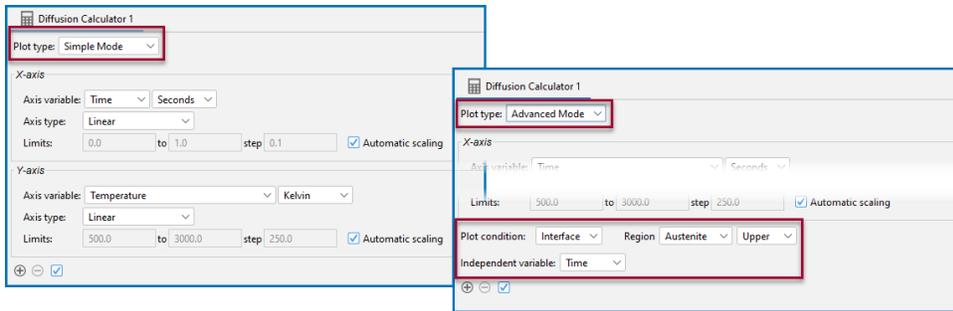
Use **Advanced Mode** to select any combination of axis variables, and then access further settings such as the **Plot condition**, **Region**, and **Independent variable**. Examples of advanced plot options are activity of a component at an interface, diffusivities, and diffusion paths. Also user-defined functions can be used as plot variables where Console Mode syntax is used to define functions. In some cases, sensible default values are selected, but they can be changed according to your needs.



For one example using Advanced Mode, see [D\\_10: Iron \(Fe\) Homogenization in Scheil](#).

### Switching Between Modes

You can try yourself to switch between Simple Mode and Advanced Mode and then change the settings to how you want it using example [D\\_04: Fe-C Moving Boundary: Austenite to Ferrite](#).



*Comparing Simple Mode (left) to Advanced Mode (right) on a Plot Renderer as a successor to a Diffusion Calculator. For Advanced Mode, there are additional settings made available based on the combination of axis variables.*

---

## Zooming In and Out

---

In the **Visualizations** window (where the plots are):

- Use the mouse scroll wheel to zoom in and out.
- Click and drag the mouse from left to right to highlight an area of the plot to zoom into a specific point. Use the scroll wheel to continue zooming in or out.

### Resetting a Plot to the Default View

- For 2D plots, click and drag the mouse from right to left.
- For 2D and 3D plots, click **Perform Tree** to regenerate the plot in the **Visualizations** window.



#### [Rotating 3D Plots](#)

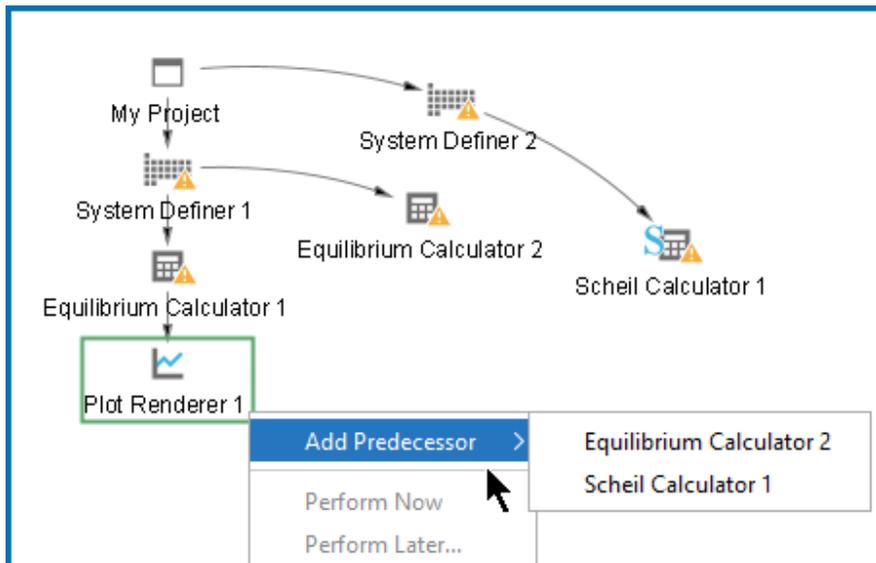
In the **Project** window (where the activity nodes are):

- Right-click and select **Zoom In** or **Zoom Out**.
- To go back to the default zoom, select **Reset Zoom**.

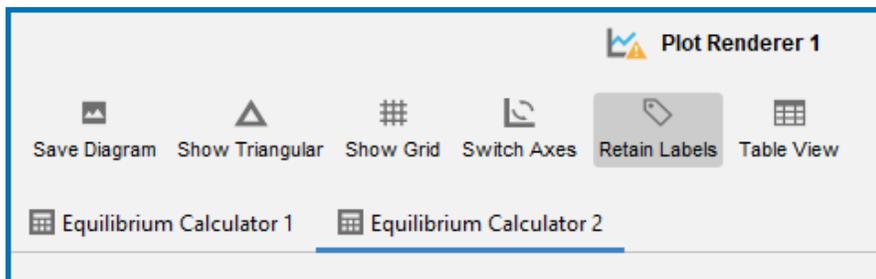
## Plotting Several Calculations in One Diagram

To plot several calculations in one diagram, you must have, in addition to the Plot Renderer's original predecessor, at least one available Plot Renderer predecessor in your project, for example, the additional predecessor can be an Equilibrium Calculator, Binary Calculator, Ternary Calculator, Scheil Calculator, or Experimental File Reader.

1. In the Project window click the **Plot Renderer** node.
2. In the **Configuration** window click **Add Predecessor**. Or right-click the **Plot Renderer** and choose **Add Predecessor**.

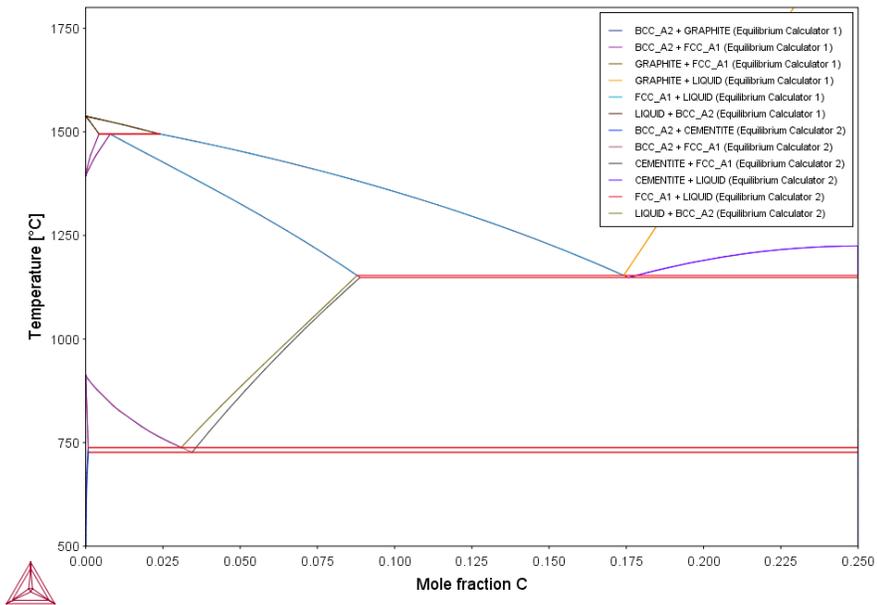


3. Select a predecessor. When added, the extra predecessor is linked to the Plot Renderer. On the Plot Renderer's Configuration window, the predecessors are shown as two tabs:



- Configure how you want each calculation plotted on the tabs in the **Configuration** window, then click to **Perform** the Plot Renderer activity.

The calculations are plotted in the same diagram in a tab in the **Visualizations** window. If any of the axis variables of the plots differ, then the label and scale of all calculations are shown parallel to each other. The following shows two calculations where different axes variables are plotted.



## Plot Renderer: Convert a Plot to a Table and Export the Data

For all calculation types and calculators, the associated Plot Renderer has a **Table View**  button you can use to convert a plot to table data. Once the plot data is converted, you can toggle between displaying the plot or table in the **Visualizations** window as well as save the table and export to file.



For the [Table Renderer](#) only single equilibrium and property diagram (stepping) calculations are available.

### Converting a Plot to a Table on the Plot Renderer

1. Set up the Plot Renderer for your particular calculation and click **Perform** to display the plot in the **Visualizations** window.
2. On the Plot Renderer **Configuration** window, click the **Table View**  button (toggled ON).
3. Choose the **Decimal digits** and **Number format (Auto, Decimal, or Scientific)** to display in the table.
4. Click **Perform**. This converts the plot data to display table data in the **Visualizations** window.

### Switching Between a Plot and a Table in the Visualizations Window

Once the plot data is converted, you can toggle between displaying the plot or table in the **Visualizations** window

- To display the Table: On the Plot Renderer **Configuration** window, click the **Table View**  button to toggle it ON and then click **Perform** to generate the table.
- To display the Plot: On the Plot Renderer **Configuration** window, click the **Table View**  button to toggle it OFF and then click **Perform** to generate the plot.

### Saving the Table (Exporting the Data)

Once the plot data is converted to a table, you can save/export this to file.

- 
1. Convert the plot to a table (the **Table View**  button is toggled ON).
  2. Click **Save table** to export the table data.
  3. In the **Save As** window navigate to where you want to save the table.
  4. Enter a **File Name**.
  5. From the **Files of Type** list choose **Text [\*.txt]**(the default), **Hyper Text Markup [\*.html]**, or **Excel [\*.xls]**.
  6. Click **Save**.

 [Saving and Printing the Output \(Graphical Mode\)](#) for more options.

## Merging Plots into One Diagram

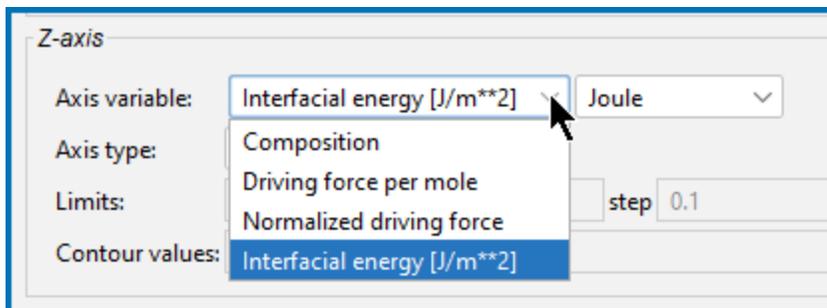
You can merge (or overlay) calculated results for several variables into one diagram. The same X and Y variables are used to compare the third variable (defined in the Plot Renderer as the Z-axis).



You cannot overlay heat map or frequency plots.

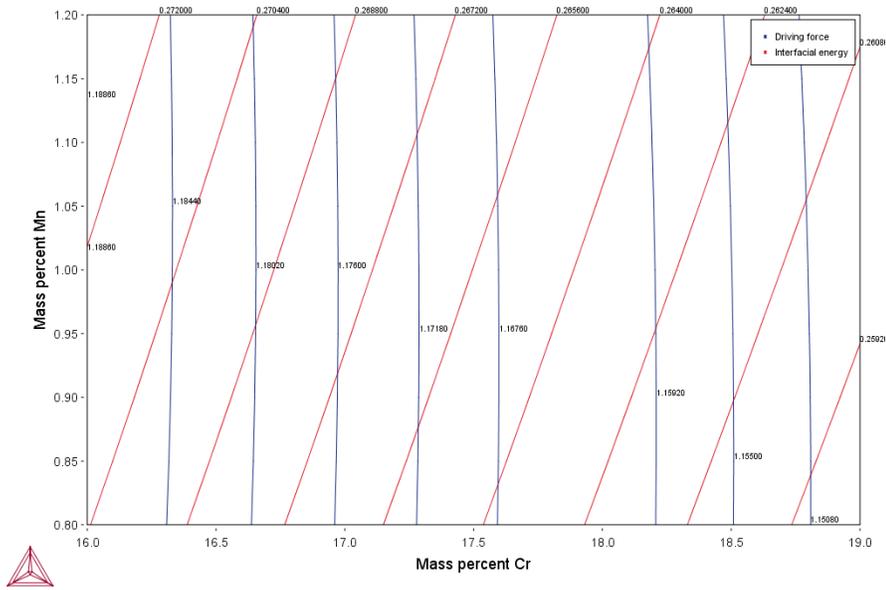
The following uses example [PM\\_G\\_03: Driving Force and Interfacial Energy](#) to describe the steps to overlay two plots: the driving force and interfacial energy results are combined into one diagram. The X- and Y-axis is the same for both (Mass percent Cr/Mass percent Mn).

1. In the **Project** window right-click the calculator node (e.g. Property Model Calculator) and add a **Plot Renderer** node from the **Create New Successor** menu.
2. Click the **Plot Renderer** node.
3. In the **Configuration** window, define the plot. Available settings are based on the calculation type. In this example the Property Model Calculator is set to a *Grid* calculation type. Choose **On** for the **Legend option** and **Contour** as the **Plot type**.
4. Click the **Add an axis**  button. A new axes settings section is added underneath.
5. From the **Z Axis** section choose the **Axis variable**. In this example, **Interfacial energy**.



6. Click **Perform** to plot both on the same diagram.

In this case, a contour plot showing both the driving force and interfacial energy calculation results comparing Mass percent Cr to Mass percent Mn.



## Adding Axes to a Diagram

You can add additional X- and/or Y-axes to a plot to compare two or more variables at the same time.



The following uses example *PM\_G\_03\_Driving\_force\_and\_Interfacial\_energy.tcu* and its Equilibrium Calculator as the starting point to create the plot shown in the figure.

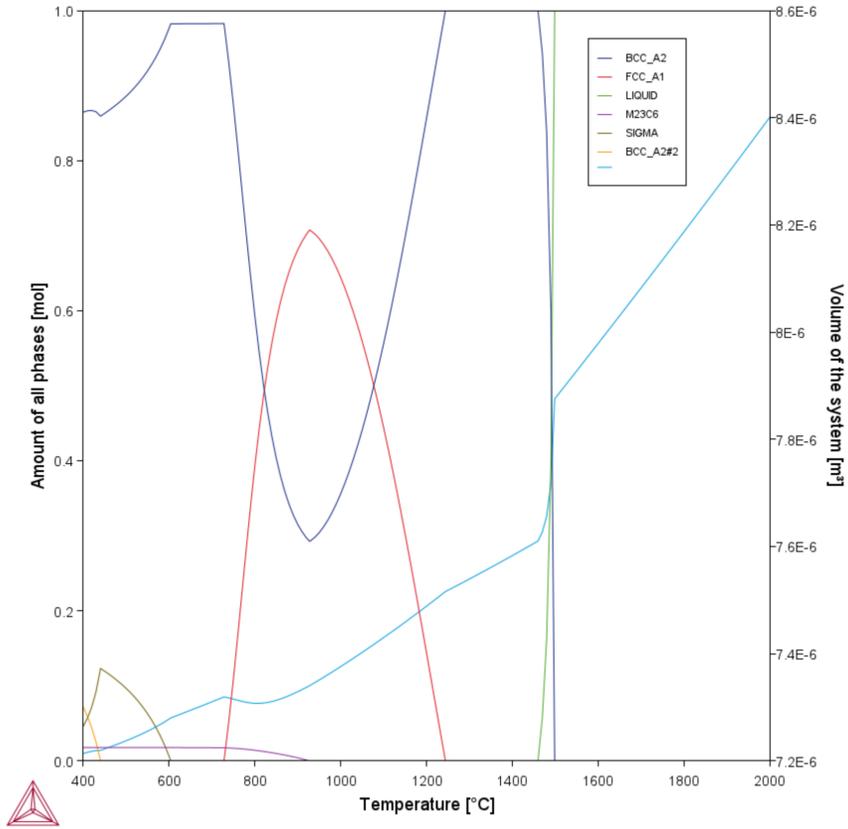
1. In the **Project** window right-click the calculator node (e.g. an Equilibrium Calculator) and add a **Plot Renderer** node from the **Create New Successor** menu.
2. Click the **Plot Renderer** node.
3. In the **Configuration** window, define the plot. Available settings are based on the calculation type.
4. Click the **Add an axis**  button. A new settings section is added underneath.
5. From the *X Axis* and/or *Y Axis* sections define another **Axis variable** and choose an **Axis type**.
6. Click **Perform** to plot both variables on the selected axis, in this example, the Y-axis.

In this example using **Amount of phase** and **Volume** are selected as the Y-axis variables and **Temperature** as the X-axis variable.

The screenshot shows the configuration window for a plot with the following settings:

- Top Y-axis:**
  - Axis variable: Amount of phase
  - Axis type: Linear
  - Limits: 0.0 to 1.0, step 0.1
  - Automatic scaling:
- Bottom Y-axis:**
  - Axis variable: Volume
  - Axis type: Linear
  - Limits: 0.0 to 1.0, step 0.1
  - Automatic scaling:
  - Additional settings:  SER, System, Cubic meter
- X-axis:**
  - Axis variable: Temperature
  - Axis type: Linear
  - Limits: 500.0 to 3000.0, step 250.0
  - Automatic scaling:

Between the Y-axis sections, there are control buttons: , , and .



The final plot comparing the additional variables.

---

## Rotating 3D Plots

---

When you use a *Grid* calculation type you can plot the diagram in 3D.



[Plot Type: Heat Map, Contour, and 3D](#) and [The Calculation Types in Graphical Mode](#)



3D Plots with additional functionality are also available with the Add-on Additive Manufacturing (AM) Module. The information is described separately and included with the Module's documentation.

In the **Visualizations** window after you create the 3D plot:

- Rotate the diagram automatically. Double-click in the **Visualizations** window and the diagram rotates clockwise. Left-click once to stop the automatic rotation.
- Rotate the diagram manually. Left-click and hold the mouse and move it around the **Visualizations** window.
- Scroll the mouse wheel/use your fingers on a trackpad to zoom on the Z-axis values.
- Right-click and move the mouse to shift the surface up or down without zooming.
- To reset the plot to its default view click **Perform Tree** to regenerate the plot.

---

# Plot and Table Formatting

---

**In this section:**

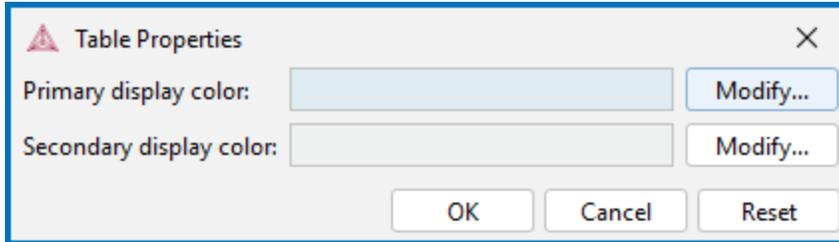
---

Setting Background Colours .....	404
Changing the Plot Properties .....	405
Plot Labels .....	407
Plot Labels: Entering a Mixture of Plain Text and Equations .....	408
Plot Labels: Rotation Angle and Show Anchor .....	410
Moving Plot Labels .....	411
About Legend Styles .....	412
Legend and Label Styles .....	413

## Setting Background Colours

---

To change the background colours of the table, in the **Visualizations** window right-click a Table Renderer tab, and select **Properties**. Modify as required.



The defaults for these settings can be changed globally from the **Options** window.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

then click the **Activities** tab and the **Tabulation** node in the tree.

## Changing the Plot Properties

To make local changes to the appearance of a specific plot, in the **Visualizations** window right-click a plot and select **Properties**.

The available settings are the same as what you can globally set under the **Tools** → **Options** menus but are only applied to the specific plot. Below includes details about features that are unique to the local plot properties options.

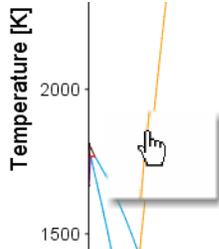


To make global plot setting default changes, see the details in [Global Settings: Graphical and Console Mode Plotting](#).

### Edit Individual Plot Lines

You can also edit some properties for individual plot lines (the color, the line width and type, and whether data points are included).

In the **Visualizations** window, hover the mouse over a plot line. When it is over a line that can be clicked the crosshair cursor turns into a pointing finger. Click the line and then you can make changes for that specific line.



To prevent unintended edits, hold down <Ctrl> while moving the cursor around the plot to only display it as a crosshair.

### Color Switching Plots Between Light and Dark Mode

Depending on whether you are using a **Light** or **Dark** look and feel for the user interface (see [Global General Settings](#)), you can also switch the default colors (lines, background, and

more) of a plot using this local **Plot Properties** option. Alternatively, to set the global defaults for future project files (and plots), see [Global Plot Settings: Design the Look and Feel](#).



Use the **Color switching** option to override the look and feel settings that may not be optimal. For example, when preparing a plot for publication, the **Dark** mode plot or line colors may not work well or do not follow a journal's guidelines. In other words, you can view and export the plot with **Light** mode color settings even though the interface is using Dark mode, and vice versa.

### ***How to Use the Color Switching for Local Plots***

Perform your calculation. Right-click in the **Visualizations** window and select **Plot Properties**. Select a **Color switching** option:

- **Switch automatically between Light mode and Dark mode:** The default, which matches the current look and feel for the GUI.
- **Always use Light mode colors:** If you are in **Dark** mode, this immediately switches the plot to **Light** default color settings.
- **Always use Dark mode colors:** If you are in **Light** mode, this immediately switches the plot to **Dark** default color settings.



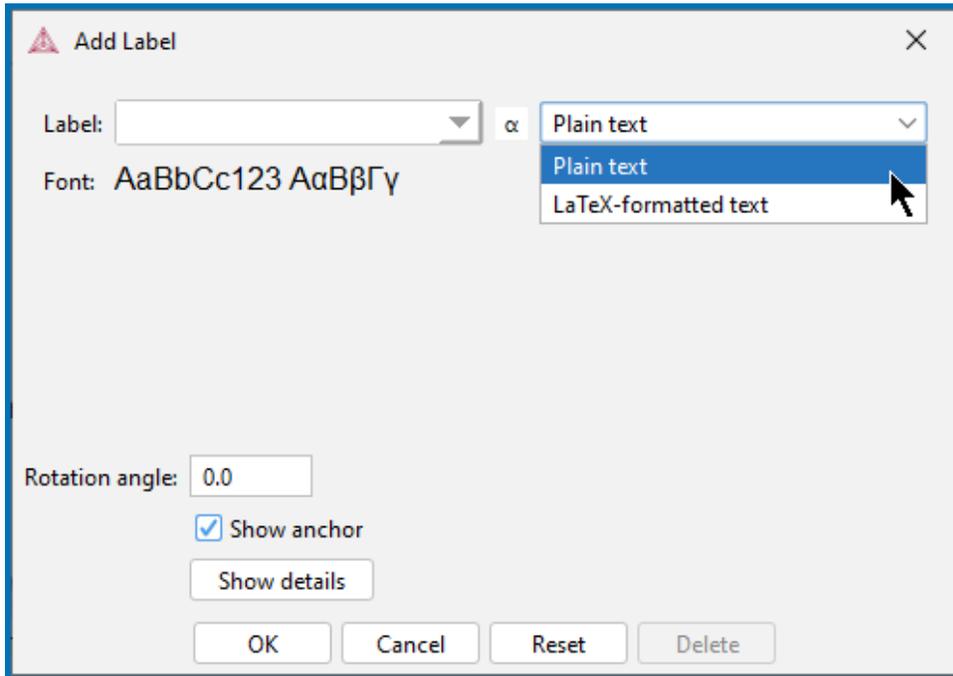
Tie lines and invariant lines are the same color for both Light and Dark modes, i.e. these colors do not change.

### ***Additional Plot Color Changes are Locked to the Original Mode***

When applying any of these settings, you can make further changes to the colors for the local plot but the change is locked to the mode. For example, you are in **Dark** mode and you change the plot background color to yellow. Then you switch to a **Light** look and feel mode. The background color for that plot is the default white background that is set for Light mode. Then if you change the background to blue, but now switch to a **Dark** look and feel, the background is yellow as this is the color applied in Dark mode to this particular plot. If you switch back to Light mode then the plot background is blue again.

## Plot Labels

To add a label text at a certain point in a plot, right-click the point, and select **Add Label**. For phase diagrams, the program suggests a label that you can keep or edit. The suggested label specifies the names of the stable phases at the coordinate where you clicked. For all other plot types, you enter your own text, which can either be Plain text or LaTeX formatted text.



You can enter the label as **Plain text** or in **LaTeX-formatted text**. For **LaTeX-formatted text** enter or edit the text as required. For example, underscores (`_`) are added before the numbers to make subscripts in the label.



For a list of supported LaTeX commands, see:

<https://www2.ph.ed.ac.uk/snuggletex/documentation/math-mode.html>.



By default when using the LaTeX command, you are in math mode and Thermo-Calc automatically interprets the entered text as math. The above link uses the dollar sign (\$) in all the examples but this is NOT required. For example, if you enter `$E=mc^2$` this is displayed including the \$ instead of the equation.

## Plot Labels: Entering a Mixture of Plain Text and Equations

When you enter text using LaTeX formatted text it displays the text as an equation.

When using the **Plain text** format, the label does not display correctly because the label is a mixture of text (**Mole-Fraction**) and an equation needing subscripts (**Al<sub>2</sub>O<sub>3</sub>**).

Mole-fraction Al<sub>2</sub>O<sub>3</sub>

The 'Edit Label' dialog box shows the following settings:

- Label: Mole-fraction Al<sub>2</sub>O<sub>3</sub>
- Format: Plain text
- Font: AaBbCc123 AαBβΓγ
- Buttons: Modify...

To add a mixture of plain text and equation formatted text, exit math mode by using the string `\textsf` followed by the text to display enclosed in curly brackets `{}`.

For example `\textsf{Mole-fraction }Al2O3` results in this label. In this example, you need to add a space at the end of fraction in order to separate the text from the equation part of the label. You can also choose a color to apply to the label text.

Mole-fraction Al<sub>2</sub>O<sub>3</sub>

The 'Edit Label' dialog box shows the following settings:

- Label: `\textsf{Mole-fraction }Al2O3`
- Format: LaTeX-formatted text
- Text size: 16
- Color: Black
- Buttons: Modify...



When working in Console Mode, you can also use the above windows on the Console Results window to add and edit labels. In addition, you can use the command.

It is possible to change the text style used in LaTeX labels, either for a section of the text or for the whole text (unless something else is specified).

<i>Section of Text Within Curly Brackets</i>	<i>All Text After the Command</i>	<i>Resulting Text Style</i>
<code>\textsf{}</code>	<code>\sf</code>	Sans serif (default)
<code>\textrm{}</code>	<code>\rm</code>	Roman
<code>\textbf{}</code>	<code>\bf</code>	Bold
<code>\textit{}</code>	<code>\it</code>	Italic
<code>\textem{}</code>	<code>\em</code>	Emphasis
<code>\texttt{}</code>	<code>\tt</code>	Monospace

---

## Plot Labels: Rotation Angle and Show Anchor

---

By default the **Show anchor** checkbox is selected, which includes an arrow from the label point to the coordinates.

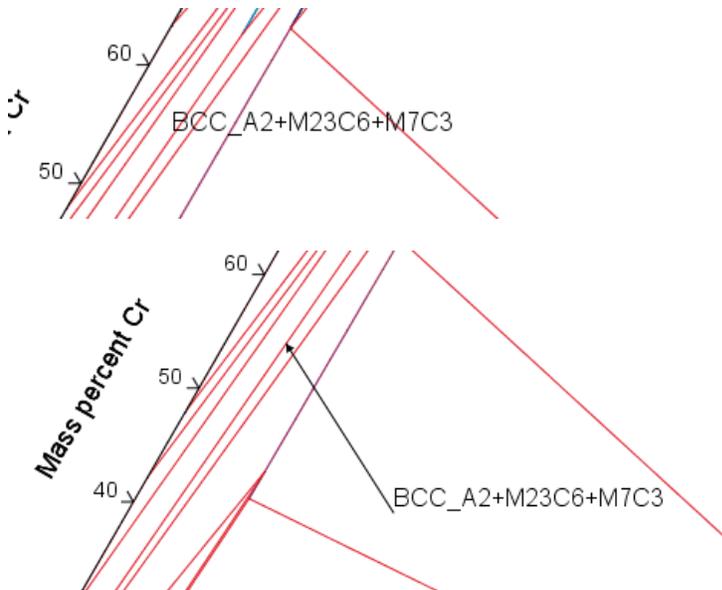
If you do not want the label oriented horizontally in the Plot Renderer tab, then specify the number of degrees of rotation in the **Rotation angle** field.

Click **OK** to add the label to the plot. You can also move a label by clicking and dragging it. The default anchor arrow is added to indicate the point that belongs to the label.

## Moving Plot Labels

---

Drag and drop labels anywhere on the plot. The anchor arrow points to the point of origin.



Close up of a plot label before being moved (top) and after being moved (bottom). The arrow in the lower image points to the label's original location on the plot.

## About Legend Styles

---

There are a variety of legend styles you can use to display detail about the phases. The same legend styles are available for both Graphical Mode and Console Mode plots.



For Console Mode, use the command `CHANGE_LEGEND`.

In Graphical Mode the label styles are defined when using the [Plot Renderer](#). You can select a **Legend style** for the **Stable phases** or **Axis quantity** selected for *Legend option*.

When there is a legend in the diagram you can append constitution and/or ordering descriptions to the phase name.

If constituent description is included in the legend, then for each equilibria along a line in the diagram, the largest constituent fractions are summed until they are equal or greater than 0.6. This is done separately for each sublattice. If the equilibria along the line gets different sets of constituents these are all appended to the list for that sublattice.

The ordering description is only printed for partitioned phases (phases that have a disordered contribution) that begins with at least two identical sublattices (sublattices where the set of constituents are identical). `DISORD` (disordered) is shown if site fractions are identical between identical constituents on all identical sub lattices (with an error margin) otherwise `ORD` (ordered) is shown in the legend.

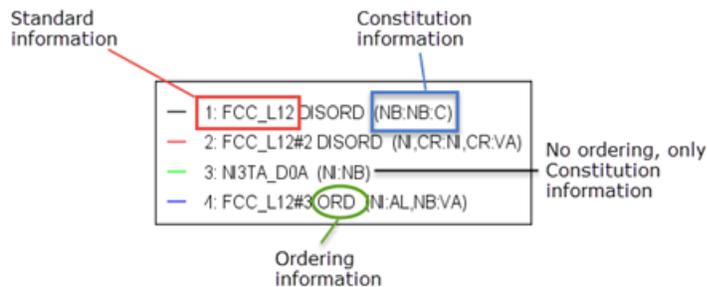
If ordering between equilibria in a line are different, then `ORD + DISORD` is shown in the legend.

## Legend and Label Styles

### Legend Styles

To show the differences between labels, the following uses the same example file that creates Figure 6 in TCNI database extended information document (found on the Thermo-Calc [website](#)).

When **All** is selected for this specific example the label includes the following:



Specifically:

- There are two disordered FCC phases (matrix and carbide), one ordered FCC (gamma prime), plus a phase NI3TA\_D0A (called Delta in Ni-alloys).
- ORDERING shows the different FCC phases.
- CONSTITUTION shows that FCC\_L12#1 is NbC, FCC\_L12#2 is matrix with Ni and Cr as major constituent, FCC\_L12#3 is Ni3Al with Nb addition and NI3TA\_D0A is Nb-rich (eg. Ni3Nb).

### Label Styles

These examples are for each label style choice - **None**, **All**, **Constitution**, or **Ordering**. The instructions are for when you are setting the options using the Plot Renderer. In Console Mode, you choose the options (**ALL**, **NONE**, **CONSTITUTION**, or **ORDERING**) after the command **CHANGE\_LEGEND**.

#### **None**

When **None** is selected the legend includes the standard information about the phases.

- 1: FCC\_L12
- 2: FCC\_L12#2
- 3: N3TA\_D0A
- 4: FCC\_L12#3

## All

When you select **All** the **Constitution description** and the **Ordering description** (when there is ordering) are included.

- 1: FCC\_L12 DISORD (NB:NB:C)
- 2: FCC\_L12#2 DISORD (NI,CR,NI,CR:VA)
- 3: N3TA\_D0A (NI:NB)
- 4: FCC\_L12#3 ORD (NI:AL,NB:VA)

## Constitution Description

When you select **Constitution description**, this is added to the standard description.

- 1: FCC\_L12(NB:NB:C)
- 2: FCC\_L12#2(NI,CR,NI,CR:VA)
- 3: N3TA\_D0A(NI:NB)
- 4: FCC\_L12#3(NI:AL,NB:VA)

Here is an example with a brief explanation of what you might see when choosing this option.



Only major constituents are listed.

—	C14_LAVES (CR:NB)
—	C15_LAVES (V,CR:TI)
—	CR3SI_A15 (NB,V:MO,TI:VA)
—	CR3SI_A15#2 (NB,V:MO:VA)
—	LIQUID (CR,MO)
—	C15_LAVES#2 (CR:NB)
—	ALPHA_B19 (V:V)
—	ALTI_L10 (TI:NB)

First there is the phase name then (NB, V : MO, Ti : VA) meaning the first sub-lattice contains mostly NB and V. The second sub-lattice contains mostly Mo and TI, the third sub-lattice contains mostly VA= Vacancy= No atom/molecule at all.

Mostly" means Thermo-Calc sums the largest site fractions in a sub-lattice until it reaches 0.6. E.g. If sorted values are FE=0.5, AL=0.15, CR=0.15, NI=0.09 + smaller values then you will get "(Fe,AL,CR)" because FE+AL > 0.6 and then CR because it has same size as AL.

### **Ordering Description**

When you select **Ordering description** and when there is ordering, this is added to the standard description.

—	1: FCC_L12 DISORD
—	2: FCC_L12#2 DISORD
—	3: Ni3TA_D0A
—	4: FCC_L12#3 ORD

---

# Additive Manufacturing (AM) Module

---

**In this section:**

---

About the Additive Manufacturing (AM) Module .....	417
Available Options .....	420
Additive Manufacturing (AM) Module Help Resources .....	425
Network License Restrictions .....	426

## About the Additive Manufacturing (AM) Module



This guide is a supplement to the full Thermo-Calc documentation set. Access the help in Thermo-Calc by pressing F1, or choose **Help** → **Online help**, or click **Online Help** on the main **My Project Configuration** window. See [Additive Manufacturing \(AM\) Module Help Resources](#) to learn other ways to access information.

The Additive Manufacturing (AM) Module (also referred to as the AM Module) is an Add-on Module to Thermo-Calc and it is available in Graphical Mode as the AM Calculator. The aim of the Additive Manufacturing Module is to better understand the laser powder bed fusion (LPBF) process by predicting the temperature distribution and melt pool geometry as a function of process parameters and using material properties from the Thermo-Calc thermodynamic and properties databases. Special focus is to have a unified treatment of alloy-dependent material properties and process parameters when solving the multiphysics problem of a moving heat source that melts and solidifies metal powder. The multiphysics simulation involves thermal conduction, fluid flow, evaporation-, radiation-, and convective-heat loss.

The Module can simulate the transition from conduction- to keyhole-mode. With experimental melt pool data you can calibrate the heat source. The calibrated heat source can be used to predict additional process conditions and/or to construct process maps (through the TC-Python Software Development Kit).

Furthermore, once you have the temperature distribution, for instance as a function of time and space, you can also use this information for other Thermo-Calc Add-on-Module simulations such as with the Diffusion Module (DICTRA) or the Precipitation Module (TC-PRISMA), or for input to other external finite element programs.

Evaluation of the following is possible depending on the calculation type:

### ***Steady-State***

- Size of melt pool
- Peak temperature
- Velocity of fluid flow
- Property variations through the melt pool (temperature, viscosity, thermal conductivity, density) or any selected line

- 2D sectioning in any plane
- Heat source calibration
- Keyhole mode (with a Gaussian heat source)

### ***Transient and Transient with Heat Source from Steady-state***

- Temperature vs. time response at selected points of the build and how this changes with process parameters
- Time-dependence of the properties listed above under steady-state
- Connect the above temperature vs. time response with the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA)
- Keyhole mode (with a Transient from Steady-state calculation using a Gaussian heat source)

## **Overview of the Content**

There are [Additive Manufacturing \(AM\) Module Help Resources](#) and [Network License Restrictions](#) to consider and access.

Theory is covered in a summary of the [Additive Manufacturing Simulation Types](#) and then some basic equations pertaining to the logic behind the simulation and settings are discussed in [Additive Manufacturing Module Theory](#).

The workflow for the AM Module is outlined in [Setting Up the Additive Manufacturing Simulation](#) and there are two default [Additive Manufacturing Templates](#) you can use to start defining your project. You can also add nodes individually to build your own tree. There are two workflows, and in either workflow, defining the AM Calculator involves choosing one of the available simulation types: Steady-state, Transient, or Transient with heat source from Steady-state.

Then you can start building the simulation using the activity nodes and depending on which workflow you start with.

This involves the standard **System Definer** and **Scheil Calculator** settings, which are the same as what is used with a standard Thermo-Calc configuration. When you add an Additive Manufacturing Template (and need to generate some materials properties or do not have a material library to use), then some unique default settings for the Scheil Calculator and these particular types of simulations are already set by default. These are discussed in [Setting Up the Additive Manufacturing Simulation](#).

This guide specifically includes details about the settings for the [AM Calculator](#) and the specialized [AM Calculator Plot Renderer](#). There is also information about the different [AM Plot Types](#).

Then finally you can visualize and examine your results using the [AM Visualizations Window Plot Toolbar](#) and learn how to [Rotate, Zoom, and Pan 3D Plots: AM Calculations](#). There are many details related to visualizing the plots discussed in [Working with AM Visualizations](#) as well as a lot of specialized information for heat source calibrations discussed in [Visualizing Heat Source Calibrations](#).

## Databases

For compatibility with the Additive Manufacturing Module, a database needs the following properties added in addition to the thermodynamics: surface tension for liquid, viscosity for liquid, thermal conductivity, molar volume, and a complete gas description.

The Additive Manufacturing Module is currently available together with the following databases, starting with the version indicated:

- Steel and Fe-alloys Database (TCFE13)
- Nickel-based Superalloys Database (TCNI12)
- Magnesium-based Alloys Database (TCMG6 and newer)
- Copper-based Alloys Database (TCCU6)
- Titanium and TiAl-based Alloys Database (TCTI5)
- Noble Metal Alloys Database (TCNOBL3)
- High Entropy Alloys Database (TCHEA6 and newer)
- Aluminum-based Alloys Database (TCAL9)



Read more about the [Additive Manufacturing \(AM\) Module](#) on our website including the details about database compatibility or to watch an [introductory webinar](#). You can also use the [Getting Started Guide](#) to learn about the key features available.

## Available Options

---

The Additive Manufacturing (AM) Module is an Add-on Module to the core Thermo-Calc software.



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. Without a license you are in *Demo Mode* where you can, for example, open and view example set ups, run some examples, add templates and nodes to the Project window, adjust some Configuration settings, and preview some functionality on the Visualizations window.

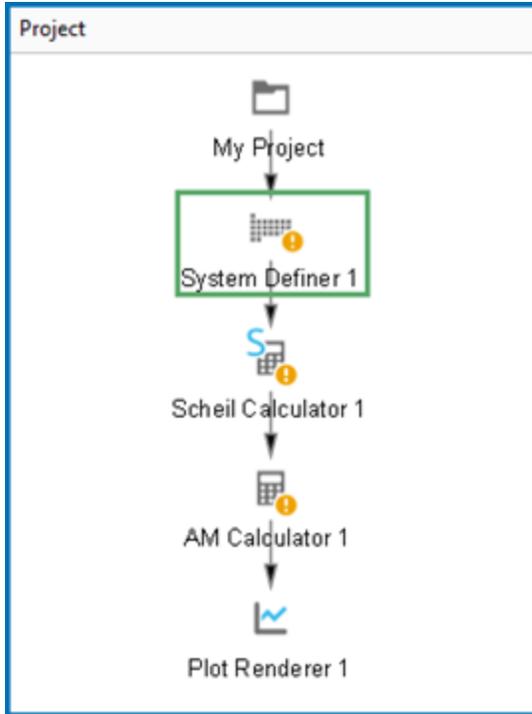
### **Additive Manufacturing Templates**

Two types of **Additive Manufacturing** templates are under **Applications** and these are available to all Thermo-Calc users when in Graphical Mode. If you are in DEMO mode then this is indicated by the addition of **DEMO** text. See [Figure 19](#).

#### *Using the Templates*

After opening Thermo-Calc in Graphical Mode, in the templates section under **Applications**, click the **Additive Manufacturing** button to add a *System Definer*, *Scheil Calculator*, *AM Calculator*, and *Plot Renderer* to the **Project** tree.

This template is used when you need to generate and use data from a Scheil calculation.



An alternate template is available when you already have data or can directly use or import a Material Library.

After opening Thermo-Calc in Graphical Mode, in the templates section under **Applications**, and to the right of **Additive Manufacturing**, click the **With Material Library** button to add an *AM Calculator* and *Plot Renderer* to the **Project** tree.

### ***Demonstration (Demo) Mode***

The Additive Manufacturing (AM) Module is available in a demonstration (DEMO) mode without a license. It includes access to the templates, some examples, and some access to the embedded predefined material properties libraries. You also have access to the fluid flow settings and a steady-state calculation can be done. However, there are limitations such as the ability to change settings and to fully run calculations.

- Some examples (i.e. AM\_01, AM\_02, AM\_03, and AM\_06b) use embedded and predefined Material Properties libraries that are included for all users. Although these examples can be run without a license, you cannot make any changes to the settings. If changes are made, the **Perform** button can still be clicked but the calculation will fail and a message displays above the **Perform Tree** button to inform

you of the need for a license. In addition, the **Event Log** displays an error message, ERROR AM Calculation: AMCalculation in Demo mode has limited functionality. See [Figure 20](#).

- Generally, when in DEMO mode and using the **With Materials Library** template, you can perform a calculation with default values. You cannot change any setting, including the calculation types (i.e. Steady-state, Transient, or Transient with heat source from Steady-state) as well as the Material Properties library i.e. meaning only Steady-state with the IN625 library is available with the default setup is possible.
- All plot configurations are possible to perform in the above cases.
- The following examples require a full license to the Additive Manufacturing (AM) Module to run, plus the indicated additional licenses. For those without the applicable licenses, you can still open the examples to review the settings and then refer to the documentation for discussions of what it is showing.
  - AM\_04 also requires a license for TCS Ti/TiAl-based Alloys Database (TCTI5 and newer). You can also go to our *Getting Started Guide* [on the website](#), which uses AM\_04 in the example.
  - AM\_05 also requires a license for the Diffusion Module (DICTRA), plus additional database licenses for the TCS Steel and Fe-alloys Database (TCFE13 and newer), and the TCS Steels/Fe-Alloys Mobility Database (MOBF8 and newer).
  - AM\_06a, which calibrates heat sources, requires an Additive Manufacturing (AM) Module license, but its companion AM\_06b, where you use these calibrated heat sources, does not require an additional license.



Figure 19: The templates indicate that you are in DEMO mode and would need a license to fully use the templates or run AM\_04, AM\_05, and AM\_06a.



```
Event Log
06:09:11,203 INFO GobblersTask: Gobbled: 06:09:11,202 [main] INFO CalculationEngine: Starting Thermo-Calc 2023b, Java-v
06:09:14,052 INFO GobblersTask: Gobbled: 06:09:14,052 [main] INFO SoftwareLicenseHub: Database directory: C:\Program Fi
06:09:16,071 ERROR GobblersTask: Gobbled: 06:09:16,071 [pool-7-thread-1] ERROR AMCalculation: AMCalculation in Demo mode
```

Figure 20: When in DEMO mode and you click on the AM Calculator node, the above note is included at the bottom of the Configuration window to remind you that a license is needed for full functionality. The Perform Tree button is not always available and sometimes even if it is available to click, the calculation itself cannot complete due to lack of a license. There will also be an error message in the Event Log to indicate the need for a license.

## Material Properties and Libraries

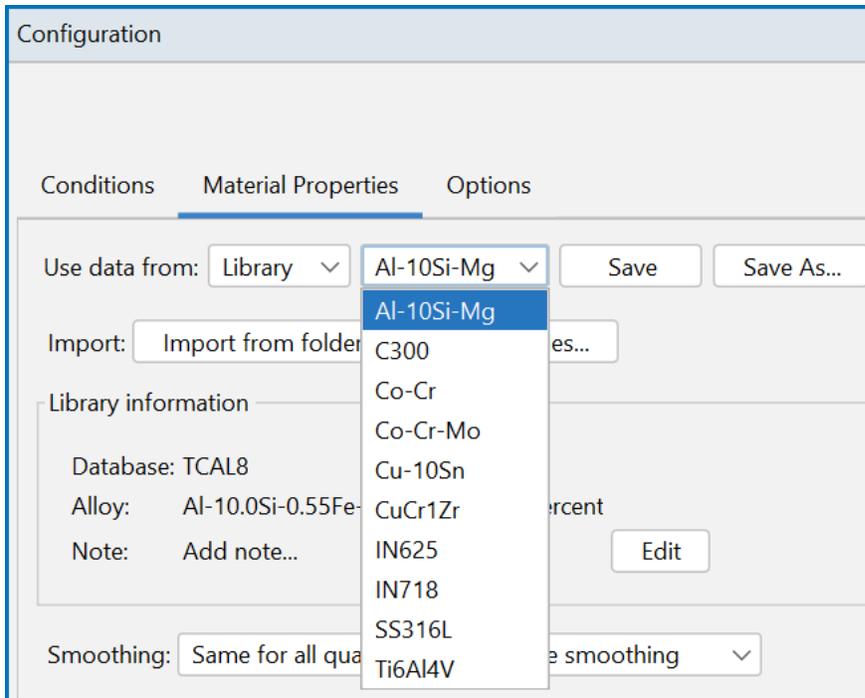


All predefined Material Properties libraries are available if you have a license for the Additive Manufacturing (AM) Module, even if you do not have the alloy database it is created with.

### Create or Import Your Own Material Library

The use of the Scheil Calculator with an AM Calculator provides you with a way to generate the data you need to feed into any AM calculation. When this Scheil calculation is completed, and if you have a full license, you can save the results to the Material Properties Library for later use. From the AM Calculator there is an option to use either Scheil data (when it is connected to the Scheil Calculator) or predefined library data (as long as you have a license to the Additive Manufacturing (AM) Module the calculations can be run fully). With the applicable license, you can also import your own material library for a particular alloy of interest and then you can just use the AM Calculator (i.e. the **With Material Library** template setup) without needing the additional Scheil calculation step.

Users with a full Additive Manufacturing license can access all the predefined libraries for use in calculations (and also can create and import their own libraries and save these for reuse).



## Using Data with other Add-on Modules



When you also have licenses either for the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA), for transient simulations you can additionally include probe data from the AM Calculator in the set up of the thermal profile for diffusion and precipitation calculations, respectively.



See [AM Calculator: Working with Probe Data](#).

## Additive Manufacturing (AM) Module and TC-Python



When you also have a license for the TC-Python Software Development Kit, you can use the Additive Manufacturing (AM) Module functionality. Some examples are available with just a TC-Python license. Learn More about [TC-Python on our website](#).

---

## Additive Manufacturing (AM) Module Help Resources

---

### 'Online' Help: No Internet Required

There are several ways to access online help in a browser. The content opens in a browser but uses local content so you do not need an Internet connection except for links to external websites.

First open Thermo-Calc, then choose one of these options:

- Press <F1> on the keyboard.
- Select **Help** →  **Online Help**.
- On the **My Project Configuration** window, click  **Online Help**.
- At the bottom of any **Configuration** window, click  **Help**.



There are several resources available on our website to help you learn how to use Thermo-Calc and other Add-on Modules. Go to the [Getting Started Guides](#) page and choose one of the guides that provides new users an introduction to setting up simple calculations in Graphical Mode. The information there is also in this help documentation.

### Additive Manufacturing (AM) Module Specific Information



Read more about the [Additive Manufacturing \(AM\) Module](#) on our website including the details about database compatibility or to watch an [introductory webinar](#). You can also use the [Getting Started Guide](#) to learn about the key features available.

## Network License Restrictions

---

The Additive Manufacturing (AM) Module requires a separate license. If you are using a network client installation of Thermo-Calc, then you may not be able to use it even if you have access to a license server with a valid network license file. The reason for this is because other clients who are part of your network installation may have checked out all instances of the network license allowed to run simultaneously.



For users with a network license, you must exit Thermo-Calc to release the license for other users. The license is checked out as soon as you add an AM Calculator and remains unavailable to other users until you exit the program.



With a network license, and if as per above you temporarily do not have access to a license, you are automatically put into *Demo Mode*. Then the AM Calculator is available.



Search the online help or see the *Thermo-Calc Installation Guide* for more about network licenses.

## General Theory and Background

---

**In this section:**

---

Additive Manufacturing Simulation Types .....	428
Additive Manufacturing Module Theory .....	430
About Heat Source Calibration .....	441
About the Keyhole Model .....	444

## Additive Manufacturing Simulation Types

Three simulation types are available on the [AM Calculator](#) – *Steady-state*, *Transient*, and *Transient with heat source from Steady-state*.



[Additive Manufacturing Module Theory](#)



The **Heat source calibration** *Calculation Type* is available for *Steady-state* simulations, and described more in [About Heat Source Calibration](#).



The keyhole model is available when a Gaussian heat source is used for either the *Steady-state* or *Transient with heat source from Steady-state* types of simulation (see [About the Keyhole Model](#)).

### *Steady-state*

In the *Steady-state* mode it is assumed that the temperature distribution and the fluid flow around the heat source is in steady state and does not change with time. This is useful to get an estimation of the temperature distribution and size of the melt pool when you assume that the heat source is moving at a constant speed in a given path. In the single-track experiments, temperature distribution around the heat source and fluid flow inside the melt pool reach a steady state very quickly, and you should then perform steady state simulations to predict melt pool geometry and cooling rates around the melt pool. In a sense, Steady-state simulations give you an overall picture of the process but in order to get more precise details and predict temperature distribution in a multi-layer build, as a function of time, you should perform Transient simulations. The benefit of the steady state option is that these simulations are quick, and you get a solution typically within 1-5 minutes, depending on the process parameters and your computational resources. For the Steady-state simulations, you can include or exclude fluid flow inside the melt pool due to the Marangoni effect. For the given processing conditions, if convection is the dominant mode of heat transfer, inclusion of fluid flow is crucial to enhance the accuracy of the model by correctly capturing the underlying physical behavior of melting and solidification of material. Furthermore, you can also perform simulations with a powder layer on the top of the substrate having different material properties than the bulk material. The Steady-state simulations are performed on a symmetric domain where you specify only the height of the substrate and the thickness of the powder layer, if present. The length and width of the computational

domain are determined automatically based on the process parameters. The temperature distribution is computed using the energy equation while the fluid flow is modeled using the Navier-Stokes equation.

### *Transient*

In the *Transient* mode, you can perform full-scale transient simulations in a 3D rectangular build part and have the possibility to specify a scanning strategy comprising multiple tracks and multiple layers. Here you can enter the height, width, and length of the entire build part or a representative segment of the build part and configure a scanning strategy either for a single track or for multiple tracks (bidirectional or unidirectional). You can also add multiple layers of powder and rotate the scanning pattern between layers.

Similar to the Steady-state mode, here you can also choose to include fluid flow inside the melt pool to correctly capture the underlying physical phenomena of melting and solidification of the material and thereby increasing the accuracy of the model. The inclusion of fluid flow requires coupling the Navier-Stokes equations together with the energy equation which comes at the cost of increased numerical complexity resulting in longer simulation times.

### *Transient with Heat Source from Steady-state*

In order to perform full scale 3D simulations in an efficient manner, with multitracks and multilayers, including fluid flow in the melt pool or with powder layer(s) having different properties than the solid material, you can use the *Transient with heat source from Steady-state* mode. This mode develops a novel approach where the effect of fluid flow due to Marangoni convection in the melt pool is incorporated without solving for Navier-Stokes equations in the full-scale 3D domain.

The concept here involves the assumption that the temperature distribution and fluid flow inside the melt pool instantly reach steady state. We first solve for temperature distribution, using the energy equation and fluid flow due to Marangoni convection, using the Navier-Stokes equations for the given process parameters in the Steady-state mode. Once we have the steady state solution, we use this solution inside the melt pool and map it as a “heat source” in the transient simulations. Using this approach, we do not have to solve for the complicated Navier-Stokes equations at each time step in the transient simulation, yet we still capture the effect of fluid flow on the shape of the melt pool. Furthermore, we also reduce the computational time by solving for the most non-linear region in the energy equation i.e., inside the melt pool, only once, and then reusing the solution as a boundary condition in the transient simulations. This approach has considerably decreased the numerical complexity of the full-scale transient simulations consequently reducing the simulation time without making a significant compromise on the accuracy of the solution.

## Additive Manufacturing Module Theory

This section is an overview of the underlying fundamental theory used and applied to the calculations when you are setting up the [AM Calculator](#) using one of the [Additive Manufacturing Simulation Types](#).



The settings related to the following are defined for the [AM Calculator](#) starting with the [AM Calculator Conditions Settings](#) section.

### Energy Equation: The Enthalpy Formulation

The following formulation is based on a simple assumption that each volume element, which may contain several dendrites or a considerable part of a dendrite, is approximated as homogeneous, i.e., it is sufficient to characterize the whole volume with a single set of state variables. This assumption thus excludes the direct account of varying composition caused by microsegregation within the element.

The enthalpy formulation is based on the conservation of energy and is essentially equivalent with Fourier's second law:

$$[\text{Eq. 1}] \quad \dot{H}_V = -\text{div}(J_Q) + \dot{Q}$$

$H_V$  is the enthalpy density and  $\dot{H}_V$  is its time derivative.

The enthalpy density is related to the molar enthalpy  $H_m$  and molar volume  $V_m$  by:

$$[\text{Eq. 2}] \quad H_V = \frac{H_m}{V_m}$$

$\text{div}$  denotes the divergence operator and  $J_Q$  is the heat flux given by Fourier's law:

$$[\text{Eq. 3}] \quad J_Q = -\kappa \text{grad}(T)$$

where  $\kappa$  is the heat conductivity,  $\text{grad}$  denotes the gradient operator and  $T$  is the temperature.  $\dot{Q}$  is a source term and represents the heat added from the outside at a certain point. In modeling of a process like welding or additive manufacturing this is a key term and requires a lot of modeling.

As state functions the enthalpy and the molar volume depend on temperature, state of phase, and composition of the individual phases. In a volume element approximated as homogeneous, one thus has:

$$[\text{Eq. 4}] \quad H_m = \sum_{\beta} f^{\beta} H_m^{\beta} (T, x_k^{\beta})$$

$$[Eq. 5] \quad V_m = \sum_{\beta} f^{\beta} V_m^{\beta} (T, x_k^{\beta})$$

where  $f^{\beta}$  is the mole fraction of the phase  $\beta$  and  $x_k^{\beta}$  is the mole fraction of  $k$  in  $\beta$ . The time derivatives thus are:

$$[Eq. 6] \quad \dot{H}_m = \sum_{\beta} \dot{f}^{\beta} H_m^{\beta} (T, x_k^{\beta}) + \sum_{\beta} f^{\beta} \left( c_P^{\beta} (T, x_k^{\beta}) + \sum_{k=1}^n \frac{\partial H_m^{\beta}}{\partial x_k^{\beta}} \frac{\partial x_k^{\beta}}{\partial T} \right) \dot{T}$$

$$[Eq. 7] \quad \dot{V}_m = \sum_{\beta} \dot{f}^{\beta} V_m^{\beta} (T, x_k^{\beta}) + \sum_{\beta} f^{\beta} \left( V_m^{\beta} \alpha^{\beta} (T, x_k^{\beta}) + \sum_{k=1}^n \frac{\partial V_m^{\beta}}{\partial x_k^{\beta}} \frac{\partial x_k^{\beta}}{\partial T} \right) \dot{T}$$

Here  $c_P^{\beta}$  is the molar heat capacity of the  $\beta$  phase and  $\alpha^{\beta}$  its volumetric thermal expansion. For a given volume element the rates can be calculated from Thermo-Calc for the temperature, fraction of phases and their composition with the input of the rates  $\dot{f}^{\beta}$  and  $\dot{T}$ .

The rate of change of the enthalpy density, i.e., the left-hand side of [Eq. 1](#) thus is obtained as:

$$[Eq. 8] \quad \dot{H}_V = \frac{1}{V_m} \left( \dot{H}_m - H_m \frac{\dot{V}_m}{V_m} \right)$$

The first term inside the brackets represents the change for a constant number of atoms due to a change in temperature or phase. The second term represents a change in volume for a constant number of atoms. However, the second term is balanced by a corresponding contribution in the heat flux, which we do not usually account for, and we should thus leave it out.

$$[Eq. 9] \quad \dot{H}_V = \frac{\dot{H}_m}{H_m}$$

The fraction of phases sums up to unity and in the case of only one solid phase:

$$\dot{f}^L = -\dot{f}^{\beta}$$

In most treatments the positive quantity

$$\Delta H_m = \left( H_m^L(T, x_k^L) - H_m^{\beta}(T, x_k^{\beta}) \right)$$

treated as constant and referred to as the latent heat. It is also common to neglect the difference in heat capacity between solid and liquid and the variation of  $\Delta H_m$  with composition. [Eq. 6](#) then is simplified into

$$[Eq. 10] \quad \dot{H}_m = \dot{f}^L \Delta H_m + c_P \dot{T}$$

By the same token

$$\Delta V_m = \left( V_m^L(T, x_k^L) - V_m^\beta(T, x_k^\beta) \right)$$

is often approximated as constant and so [Eq. 7](#) is simplified into

$$[\text{Eq. 11}] \quad \dot{V}_m = \dot{f}^L \Delta V_m + V_m \alpha \dot{T}$$

where  $V_m$  is given by [Eq. 5](#) and yields

$$V_m = f^L V_m^L + (1 - f^L) V_m^\beta = V_m^\beta + f^L \Delta V_m$$

[Eq. 9](#) now becomes:

$$[\text{Eq. 12}] \quad \dot{H}_V = \frac{1}{V_m} \left( \dot{f}^L \Delta H_m^L + c_P \dot{T} \right)$$

and [Eq. 1](#) becomes:

$$[\text{Eq. 13}] \quad \frac{1}{V_m} \left( \dot{f}^L \Delta H_m^L + c_P \dot{T} \right) = \text{div}(\kappa \text{grad}(T)) + \dot{Q}$$

### *The Equivalent Heat Capacity Method Using Thermo-Calc*

If it is further assumed that the fraction liquid  $f^L$  is a unique function of temperature one has:

$$[\text{Eq. 14}] \quad \dot{f}^L = \frac{df^L}{dT} \dot{T}$$

and [Eq. 13](#) may be further simplified to:

$$[\text{Eq. 15}] \quad \frac{c_p^{eff}}{V_m} \dot{T} = \text{div}(\kappa \text{grad}(T)) + \dot{Q}$$

where the effective heat capacity  $c_p^{eff}$  is given by:

$$[\text{Eq. 16}] \quad c_p^{eff} = \left( \frac{df^L}{dT} \Delta H_m + c_P \right)$$

[Eq. 15](#) is a normal heat-flow equation and the complication of the liquid/solid phase transformation is taken into account by means of an effective heat capacity. This method is referred to as the equivalent heat-capacity method by Rappaz [1989Rap].

It then remains to be discussed how to determine  $f^L(T)$ . It must be given by some extra information. Traditionally it was taken from experimental information but more recently it has been taken from the Scheil equation. The original equation was based on a number of approximations which are unnecessary today when thermodynamic and properties databases and efficient software such as Thermo-Calc are available. In fact, when using Scheil in Thermo-Calc one can directly calculate

$c_p^{eff}(T)$ , referred to as apparent heat capacity, for a given initial composition of the liquid. A possible approach thus is to calculate  $c_p^{eff}(T)$  for the alloy under consideration and then store the result as a table and use that in the solution of [Eq. 15](#). The molar volume may be extracted from the same calculation. An example for a 316L type of stainless steel is shown in the plots below.

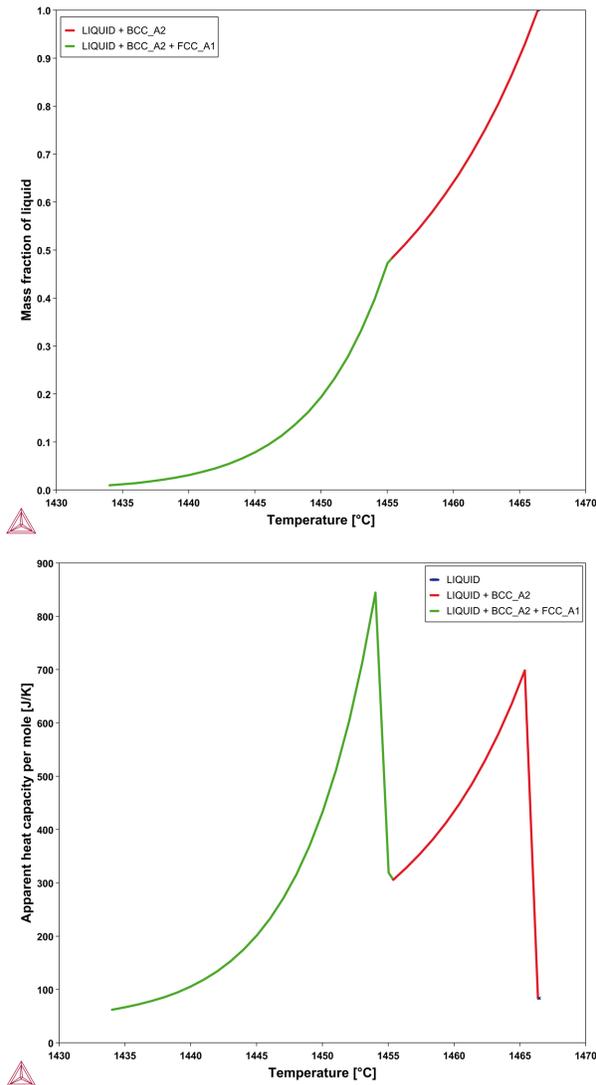


Figure 1: Results of a Scheil simulation of Fe-18Cr-10Ni alloy showing the mass fraction of the liquid (left) and the apparent heat capacity (right) both as a function of temperature.

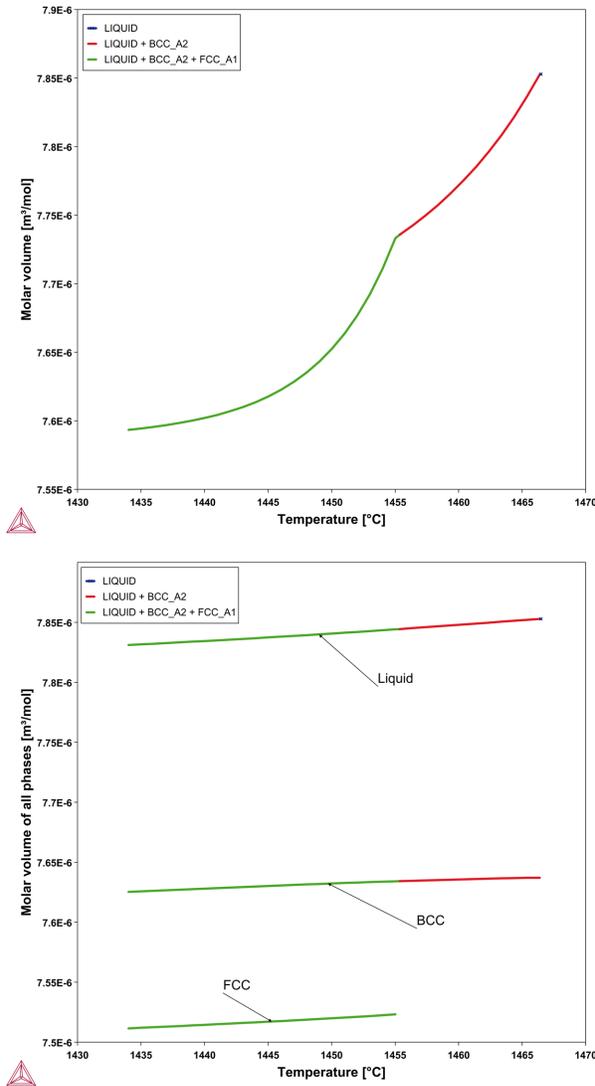


Figure 2: Molar volume from a Scheil simulation of Fe-18Cr-10Ni alloy. The right plot shows all phases as a function of temperature.

### The Enthalpy Method Using Thermo-Calc

In the enthalpy method we keep [Eq. 1](#) and use [Eq. 9](#) to write

$$[\text{Eq. 17}] \quad \frac{\dot{H}_m}{V_m} = -\text{div}(J_Q) + \dot{Q}$$

As pointed out by Rappaz [1989Rap] there is no mathematical difference between the enthalpy method and the equivalent heat capacity method because the former is simply the integral of the heat capacity over  $T$ . For numerical reasons the cusps in the heat capacity may be disadvantageous and the enthalpy has a nicer behavior as shown in the plot below, where it gives temperature as a unique function of enthalpy.

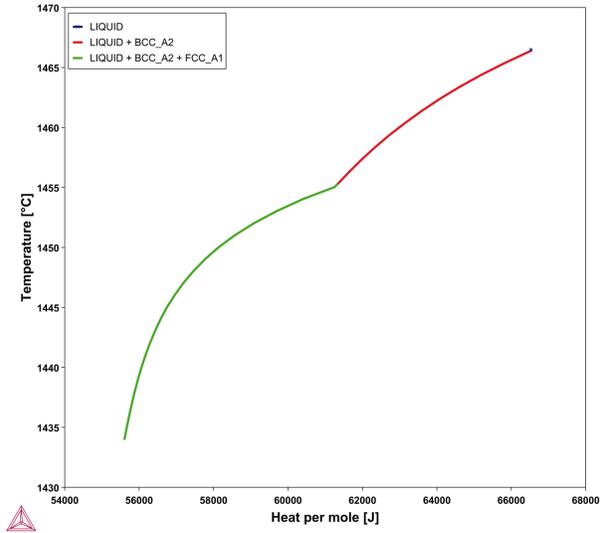


Figure 3: Enthalpy from a Scheil simulation of Fe-18Cr-10Ni alloy.

In the Additive Manufacturing Module, we therefore use [Eq. 17](#) to predict the evolution of temperature during the laser powder bed fusion (LPBF) process for the given material properties and process parameters.

### Steady-state Formulation

An important approximation is when one can solve the stationary heat-flow equation for a volume element moving with the heating source. In that case, we neglect the left-hand side of [Eq. 17](#) and add a translation term as follows:

$$[\text{Eq. 18}] \quad \text{div}[\kappa \text{grad}(T)] - \frac{v_b}{V_m} \text{grad}(H_m) + \dot{Q} = 0$$

Where  $v_b$  is the velocity of the heat source. In case of additive manufacturing,  $v_b$  is the scanning speed.

### Fluid Flow

When there is also fluid flow involved, a fluid transport term has to be added to the heat-flow equation and one obtains:

$$[\text{Eq. 19}] \quad \frac{\dot{H}_m}{V_m} = \text{div}[\kappa \text{grad}(T)] - \frac{v}{V_m} \text{grad}(H_m) + \dot{Q}$$

where  $v$  is the velocity vector field and may be imposed or obtained from the solution of the Navier-Stokes equation:

$$[\text{Eq. 20}] \quad \rho \dot{v} + \rho(v \text{grad})v - \mu(\text{divgrad})v = F_V - \text{grad}P$$

where  $\rho$  is the density related to the molar volume by means of

$$[\text{Eq. 21}] \quad \rho = \frac{1}{V_m} \sum_k x_k M_k$$

$M_k$  is the molar weight in  $\text{kg mol}^{-1}$ .  $\mu$  is the viscosity,  $F_V$  is a volume force, e.g., gravitational forces, and  $P$  is the internal pressure. The equation is often combined with approximating the liquid as incompressible.

$$[\text{Eq. 22}] \quad \text{div}(v) = 0$$

### Modeling Heat Source

It is crucially important to correctly model the heat source in AM simulations since the heat source not only affects the size of the melt pool but also determines the temperature gradients around the melt pool. In the Additive Manufacturing Module, there are three types of heat source models: *Gaussian*, *Double-ellipsoidal*, and *Conical*.



Also see additional theory related to heat sources in [About Heat Source Calibration](#).

### GAUSSIAN HEAT SOURCE

The first model is a surface heat source model which was initially proposed by Pavelic et al. [1969Pav]. By surface heat source model, it is meant that the power input from the heat source is only defined on the top surface of the workpiece, as a Neumann boundary condition on the computational domain, and the heat energy then diffuses into the workpiece depending on the thermal diffusivity of the selected material. As suggested by Pavelic et al., the heat flux ( $\text{W/m}^2$ ) is defined as Gaussian distribution around the center of the heat source, deposited on the top surface of the domain, as given in the equation below. A schematic representation of the heat flux, as defined by Gaussian heat source is shown in the figure below.

$$[\text{Eq. 23}] \quad Q = \frac{2\varepsilon P}{\pi r^2} \exp\left[-2 \frac{(x-X_p)^2 + (y-Y_p)^2}{r^2}\right]$$

Where  $Q$  is the power density deposited on the top surface ( $\text{W/m}^2$ ),  $\varepsilon$  denotes absorptivity of the laser beam,  $P$  is the power of the heat source (W),  $r$  is the laser spot radius, and  $x$  and  $y$  denote coordinates of the computational domain while  $X_p$  and  $Y_p$  represent the location of the heat source.

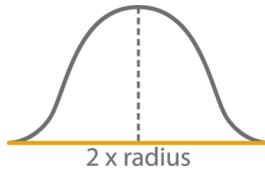


Figure 4: Schematic representation of energy distribution in Gaussian heat source model.



Also see additional theory related to using a Gaussian heat source with the keyhole option in [About the Keyhole Model](#).

## DOUBLE ELLIPSOIDAL (GOLDAK) HEAT SOURCE

The second heat source model in the AM Module is a volumetric heat source model which was first proposed by Goldak et al. [1984Gol], and is now commonly used in LBPf simulations. In contrast to the surface heat source model, in the volumetric heat source model, the power input is defined not only on the surface of the workpiece but also inside the powder bed, along the depth of the powder bed. The heat distribution in the double-ellipsoidal heat source model is given by a combination of two ellipsoids; one in the front quadrant of the heat source and the other in the rear quadrant, as shown in the figure below. The power density is thus given by two different expressions, one for each quadrant of the heat source.

$$[Eq. 24] \quad Q_f = f_f \frac{6\sqrt{3}\epsilon P}{\pi^{\frac{3}{2}} a_f b c} \exp \left[ -2 \frac{(x - X_p)^2}{a_f^2} + \frac{(y - Y_p)^2}{b^2} + \frac{(z - Z_p)^2}{c^2} \right]$$

$$Q_r = f_r \frac{6\sqrt{3}\epsilon P}{\pi^{\frac{3}{2}} a_r b c} \exp \left[ -2 \frac{(x - X_p)^2}{a_r^2} + \frac{(y - Y_p)^2}{b^2} + \frac{(z - Z_p)^2}{c^2} \right]$$

Where  $Q_f$  and  $Q_r$  are the power densities ( $W/m^3$ ) in the front and rear quadrant of the heat source, while  $a_f$  and  $a_r$  are the semi-axes of the front and rear ellipsoids, respectively.  $b$  and  $c$  are the semi-axes along the width and depth of the melt pool.  $f_f$  and  $f_r$  are the proportional coefficients for the front and rear ellipsoid of the heat source, with the condition  $f_f + f_r = 2 \cdot f_f$  is then given as follows:

$$[Eq. 25] \quad f_f = \frac{2a_f}{a_f + a_r}$$

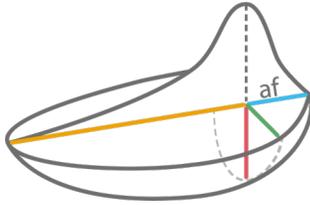


Figure 5: Schematic representation of energy distribution in double ellipsoidal heat source model.

## CONICAL HEAT SOURCE

The third heat source model in the AM Module is also a volumetric heat source model, which was first employed for welding simulations but due to inherent similarities between welding and LPBF processes, this model is also commonly used for AM simulations. The conical heat source model is defined by a Gaussian heat distribution on the top surface of the workpiece and a conical distribution along the depth of the powder bed. A schematic representation of the energy distribution for conical heat source model is shown below.

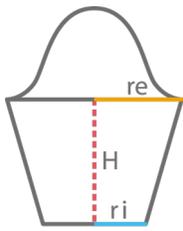


Figure 6: Schematic representation of energy distribution in conical heat source model.

The power density for the conical heat source is given as follows [2019Zha]:

$$[Eq. 26] \quad Q = \frac{6\varepsilon P}{\pi H(r_e^2 + r_e r_i + r_i^2)} \exp \left[ -2 \frac{(x-X_p)^2 + (y-Y_p)^2}{r_o^2} \right]$$

with

$$r_o = r_e + (z - ZP) \frac{(r_e - r_i)}{H}$$

where  $r_e$  and  $r_i$  are the cone radius at the top and bottom, respectively, while  $H$  is the height of the cone.

## Numerical Boundary Conditions

In order to obtain temperature distribution during the LPBF process in the Additive Manufacturing (AM) Module, incorporating fluid flow inside the melt pool, we couple energy equations with the Navier-Stokes equations, as explained earlier. In order to accurately reflect the physical process, both of these equations are subjected to a certain set of boundary conditions, which are given as follows.

## BOUNDARY CONDITIONS FOR ENERGY EQUATION

The energy equation is subjected to four different boundary conditions. The first boundary condition is defined on the top surface (either the powder bed surface or the solid substrate) and represents the energy lost due to convection and radiation to the surrounding gas.

$$[Eq. 27] \quad Q_L = h(T - T_{ambient}) + \epsilon\sigma(T^4 - T_{ambient}^4)$$

where  $h$  is the convective heat transfer coefficient,  $\epsilon$  is the surface radiation emissivity and

$$\sigma = 5.669 \times 10^{-8} W/m^2 K^4$$

is the Stefan-Boltzmann constant.  $T_{ambient}$  is the ambient temperature or the temperature of the surrounding gas in the build chamber and can be set in the AM Calculator configuration.

The second boundary condition represents the heat lost through the vertical walls of the domain and for simplicity we have chosen adiabatic boundary conditions for these surfaces. The boundary condition for the vertical walls of the computational domain is thus given as:

$$[Eq. 28] \quad Q_N = 0$$

The third boundary condition reflects the temperature of the base plate and is therefore applied to the bottom of the computational domain. Here we have assumed that the base plate of the build part is kept at a constant temperature and the boundary condition is therefore given as:

$$[Eq. 29] \quad T = T_{baseplate}$$

$T_{baseplate}$  is the base plate temperature and can be set in the AM Calculator configuration.

The fourth and the most important boundary condition is the evaporation boundary condition that represents the heat lost due to evaporation of the material. Evaporation is implemented using a physics-based approach where the heat loss due to evaporation ( $Q_E$ ) is given as:

$$[Eq. 30] \quad Q_E = 0.82J_E\Delta H_v$$

where  $J_E$  is the evaporation flux and  $\Delta H_E$  is the evaporation enthalpy. For a multicomponent system, the evaporation flux is given as follows:

$$[Eq. 31] \quad J_E = \frac{1}{\sqrt{2\pi MRT}} \left( P_o \exp\left(-\frac{\Delta G_v}{RT}\right) - P \right)$$

where  $\Delta G_v$  is the driving force for evaporation,  $M$  is the molar mass of the gas,  $P$  is the gas pressure inside the chamber,  $P_o$  is the atmospheric pressure, and  $R$  is the universal gas constant. The driving force for evaporation, the evaporation flux, and the molar mass of the

gas are all temperature-dependent material properties which are obtained from Thermo-Calc databases and are included in the material library whereas the gas pressure inside the chamber can be set from the AM Calculator configuration.

## BOUNDARY CONDITIONS FOR NAVIER-STOKES EQUATIONS

For the velocity, a no-slip boundary condition is employed on all surfaces of the computational domain. A no-slip boundary assumes that at a solid boundary, the fluid will have zero velocity relative to the boundary. For the pressure, a pinned boundary condition is used.

The most important boundary condition in Navier-Stokes setup is the Marangoni boundary condition that reflects the tangential force on the surface of the melt pool due to temperature variation of surface tension or the so-called Marangoni effect. In AM, large temperature gradients on the surface of the melt pool, leads to the Marangoni effect and is the primary cause of convective flow inside the melt pool. The Marangoni shear stress ( $\tau$ ) is modeled using the following equation:

$$[\text{Eq. 32}] \quad \tau = \frac{\partial \gamma}{\partial T} \nabla_s T$$

where  $\gamma$  is the surface tension and  $\nabla_s$  represents the tangential component of the spatial derivative.

Surface tension is also a temperature-dependent material property which is obtained from Thermo-Calc databases and is included in the material library.

### *Thermophysical Properties of Powder Material*

When the option “Use separate material properties for powder” is selected, a different thermal conductivity, density and molar volume is used for the powder material as compared to the solid substrate. The above mentioned properties depend on the porosity ( $\phi$ ) of the powder. The density ( $\rho_{\text{powder}}$ ) and molar volume ( $Vm_{\text{powder}}$ ) of the powder are then simply given as follows:

$$\rho_{\text{powder}} = \rho_{\text{solid}}(1 - \phi)$$

$$Vm_{\text{powder}} = Vm_{\text{solid}} \frac{1}{1 - \phi}$$

For thermal conductivity of powder  $k_{\text{powder}}$ , an empirical expression is used which was suggested by Xue and Barlow [1991Xue]. The thermal conductivity of powder thus also depends on the conductivity of the surrounding gas. For simplicity, the surrounding gas is assumed to be air. The thermal conductivity of powder is given as follows:

$$[\text{Eq. 33}] \quad k_{\text{powder}} = (6.3 + 22\sqrt{0.09k_{\text{solid}} - 0.016}) \frac{k_{\text{solid}}(1 - \phi)}{(k_{\text{solid}}/k_{\text{air}})(10^{0.523 - 0.594\phi}) - 1}$$

where  $k_{\text{air}} = 0.05784$ .

## References

- [1969Pav] V. Pavelic, Experimental and computed temperature histories in gas tungsten arc welding of thin plates. *Weld. J. Res. Suppl.* 48, 296–305 (1969).
- [1984Gol] J. Goldak, A. Chakravarti, M. Bibby, A new finite element model for welding heat sources. *Metall. Trans. B.* 15, 299–305 (1984).
- [1989Rap] M. Rappaz, Modelling of microstructure formation in solidification processes. *Int. Mater. Rev.* 34, 93–124 (1989).
- [1991Xue] S. Xue, J.W. Barlow, Models for the prediction of the thermal conductivities of powders, *Solid Freeform Fabrication Symposium Proceedings*, Center for Materials Science, University of Texas at Austin, Austin, TX (1991), pp. 62-69.
- [2019Zha] Z. Zhang, Y. Huang, A. Rani Kasinathan, S. Imani Shahabad, U. Ali, Y. Mahmoodkhani, E. Toyserkani, 3-Dimensional heat transfer modeling for laser powder-bed fusion additive manufacturing with volumetric heat sources based on varied thermal conductivity and absorptivity. *Opt. Laser Technol.* 109, 297–312 (2019).

## About Heat Source Calibration

There are these models available to define the heat source in the AM process: **Gaussian**, **Double ellipsoidal**, or **Conical**. All these heat sources rely on different sets of parameters to fully define the heat input to the powder and solid substrate.

The standard Gaussian heat source model (a **Single point** calculation) requires defining the **Absorptivity** and the **Beam radius** on the AM Calculator Configuration window. The keyhole model takes into account the effect of evaporation recoil pressure that produces a keyhole, if present for the given processing conditions. Multiple reflections are also included in the keyhole model to improve the calculation of the keyhole shape. However, the absorptivity of the material at printing conditions is often unknown and can be regarded as a fitting parameter. The beam radius of the heat source of the printer is, however, most often known but does not always match the definition of the beam size for the Gaussian heat source used in the simulation. Leveraging the experimental melt pool data and the relation between melt pool size and energy density can help to calibrate the absorptivity and beam radius for the Gaussian heat source model.



The keyhole model is available for Gaussian heat sources. See [About the Keyhole Model](#) for some additional information.

Similarly, for volume heat sources, i.e. **Double ellipsoidal** and **Conical**, it is necessary to enter four and three parameters, respectively, along with the absorptivity. Since volume heat sources are a numerical way to correctly capture the shape of the melt pool, it is not always possible to know the values of these parameters for the processing conditions.

However, according to the literature [2021Gra] you can find a relationship between the size i.e., width and depth of the melt pool, and the energy density (defined as the ratio of power and scanning speed) and based on that, you can also find a relationship between the heat source parameters and the energy density.



See [Additive Manufacturing Module Theory](#) for a general overview including details about the heat sources.

This means that if a set of experimental data for an alloy can be provided in the form of power, scanning speed, width of the melt pool, and depth of the melt pool, then you can perform a heat source calibration to calibrate the heat source parameters for the given material. This is done by running an optimization algorithm for the selected experimental data and then choosing an appropriate function for each heat source parameter. As a result, the output includes expressions of the given parameters as a function of the energy density, which can be saved and is available later in heat source list as a **Users heat source**. This is for any given process parameter (i.e. the saved heat source can be used for any process parameter).



Once the **Users heat source** is available to choose in the heat source list, then only the **Power** of the heat source is needed to be entered and the rest of the parameters, including the absorptivity, are automatically included in the simulation based on the selected expressions when originally saving the heat source calibration data.

In order to set up the calibration of the heat source, you select the **Steady-state** simulation type and then select **Heat source calibration** as the *Calculation Type*. The calibration is performed assuming steady-state conditions so most of the configurations are done in the same way as the **Single point** Steady-state simulation. The difference is that you need to choose which heat source to use, i.e. **Gaussian**, **Double ellipsoidal**, or **Conical**, and no other heat source parameters need to be entered.



For more details about including the **Heat source calibration** *Calculation Type* during the simulation set up, go to [AM Calculator Conditions Settings](#). Then go to [Visualizing Heat Source Calibrations](#) for details about navigating the various previews and results.

### *Heat Source Calibration with Experiment Data*

When the **Heat source calibration** is selected in the **Steady-state** simulation type, there is an option to import a file with experimental data points that can then be used in the simulation. Data can also be entered directly into the table on the AM Calculator Configuration window. However, if an **Experiment file** is imported it needs to be set up in a certain way so that the software can read the data.



#### [Heat Source Calibration Experiment File Requirements](#)

Once the experiment data is entered, you then can view the plot of melt pool dimensions vs. energy density (P/v) in the Visualizations window, and make a selection of the experiments to be used for calibration by checking the corresponding checkbox in the **Use experiment** column. When this is done, click **Perform Tree** to perform the calibration.

### *Examples Related to Heat Source Calibration*

There are two examples included with the installation to help you understand how to calibrate a heat source and then use the calibration in simulations.



[AM\\_06a: Calibrating a Heat Source for a 316L Steel](#): This example demonstrates the calibration of the heat source using the **Gaussian** as well as **Double ellipsoidal** heat sources.



[AM\\_06b: Using the Calibrated Heat Source for a 316L Steel](#): The purpose of this example is to use the saved heat sources from AM\_06a and then validate the results by comparing the melt pool size from the simulations with the experimental melt pool dimensions. It also uses the keyhole model.

### *Reference*

[2021Gra] D. Grange, A. Queva, G. Guillemot, M. Bellet, J.-D. Bartout, C. Colin, Effect of processing parameters during the laser beam melting of Inconel 738: Comparison between simulated and experimental melt pool shape. J. Mater. Process. Technol. 289, 116897 (2021).

## About the Keyhole Model

In additive manufacturing a laser beam melts the powder, and at high energy density of the laser also evaporation of the liquid metal occurs. The evaporation causes transfer of momentum from the metal to the vapour, i.e. a recoil. The conservation of total momentum then leads to a recoil pressure that pushes down the liquid/gas interface to form a cavity, also known as a keyhole. However, this free surface is computationally expensive to calculate due to the very dynamic movements of the keyhole. To reduce the complexity, the Additive Manufacturing (AM) Module has an analytic model [1994Kap] to pre-compute the keyhole shape and the corresponding mesh.



Example [AM\\_06b: Using the Calibrated Heat Source for a 316L Steel](#) includes the use of the keyhole model. The different processing conditions are selected to simulate both the conduction mode as well as the keyhole mode.

The analytical keyhole model approximates the heat conduction by a moving line source where the keyhole shape is obtained by heat balance at the keyhole wall using average material properties. A point-by-point scheme compares absorbed power and conduction losses to obtain the local inclination angle of the keyhole walls.

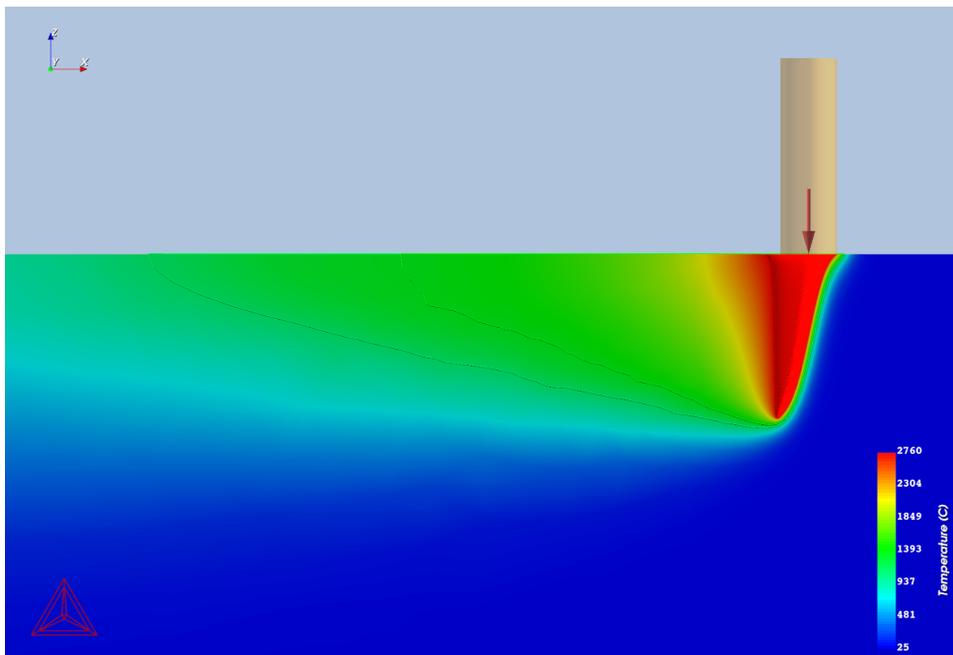


Figure 7: Side-view of a keyhole and melt pool for the material SS316L with Gaussian beam radius of  $40\ \mu\text{m}$  and the absorptivity set to 40%.

When a keyhole is formed, the energy is transmitted into the material through numerous reflections within the keyhole (see [Figure 8](#)).

For metals with an absorption factor of around 30% the reflections can increase the effective absorption to values around 80% or higher of the total laser beam energy. In the Additive Manufacturing (AM) Module, multiple reflections are implemented in the analytical keyhole mode according to the Jahn publication [2020Jah]. This strategy allows for computation of a keyhole shape, including multi reflections, without implementing a numerically expensive ray tracing algorithm.

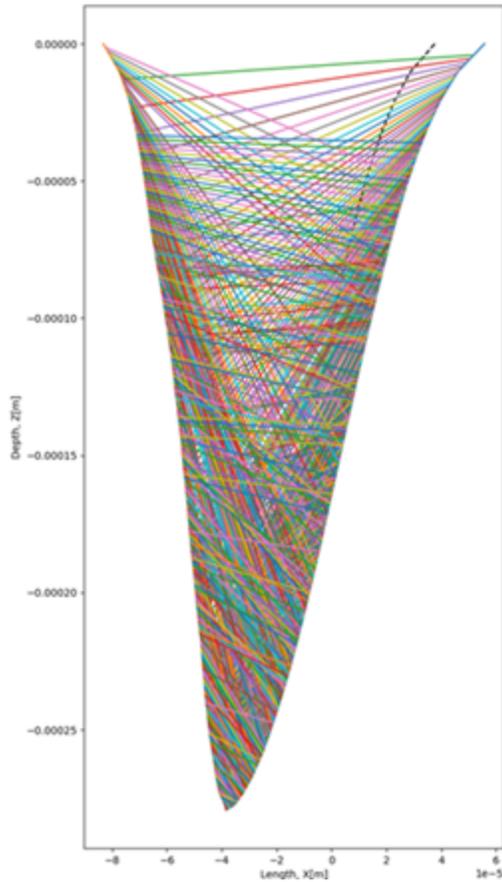


Figure 8: Ray tracking of the laser beam reflected at the wall of the keyhole.

The keyhole model is limited to the Gaussian heat source without fluid flow and without separate material properties for powder. An additional input is the Rayleigh length of the laser beam where the default value is 2.5 mm. The Rayleigh length is related to the depth of focus of the heat source. The length is defined as the distance along the propagation direction of a beam from the waist to the place where the area of the cross section is doubled.



To include the keyhole feature, go to [AM Calculator Conditions Settings](#) and in the *Heat Source* section, you can enable this feature. See [Keyhole Model Settings](#).



See [Additive Manufacturing Module Theory](#) for a general overview including details about the heat sources.

## References

- [1994Kap] A. Kaplan, A model of deep penetration laser welding based on calculation of the keyhole profile. *J. Phys. D. Appl. Phys.* 27, 1805–1814 (1994).
- [2020Jah] M. Jahn, J. Montalvo-Urquizo, Modeling and simulation of keyhole-based welding as multi-domain problem using the extended finite element method. *Appl. Math. Model.* 82, 731–747 (2020).

## Additive Manufacturing Workflow

---

**In this section:**

---

Setting Up the Additive Manufacturing Simulation .....	448
Additive Manufacturing Templates .....	452

---

## Setting Up the Additive Manufacturing Simulation

Setting up an Additive Manufacturing (AM) Module simulation can be done in one of two workflows.

### *General Overview*

The general procedure of a simulation using the Additive Manufacturing Module is the following:

1. Define the system (Workflow 1 only):
  - a. Choose a database.
  - b. Select elements and composition.
2. Retrieve the materials data (Workflow 1 or 2):
  - a. Either with a Scheil calculation (Workflow 1) or using the materials data library (Workflow 2).
  - b. Apply data smoothing.
3. Set up the AM Calculator (Workflow 1 or 2):
  - a. Choose a simulation mode *Steady-state*, *Transient*, or *Transient with heat source from Steady-state*.
  - b. For *Steady-state* you can also choose a *Heat source calibration Calculation Type* to calibrate a heat source based on the provided experimental data instead of using just a *Single point* calculation.
  - c. Set the simulation conditions.
4. Run the simulation and visualize the results (Workflow 1 or 2):
  - a. *3D Plot*
  - b. *Plot Over Line (2D)*
  - c. *Probe 2D* plot at a probe position (only available for transient calculations). You can also use and plot probe data with the Add-on Diffusion Module (DICTRA) and/or Precipitation Module (TC-PRISMA).
  - d. *Heat source calibration parameters* and *Melt pool dimensions*: These plots are available for **Heat source calibration** calculations.

## *Workflow 1: When You Need to Obtain the Material Properties Data for the Calculation*

Use a System Definer and Scheil Calculator to extract some Scheil data, then feed the data into the AM Calculator. Then add the Plot Renderer to define the specific AM Calculator Plot Renderer features and visualize the results. In this workflow, you can also save the results from the Scheil Calculator into the Material Library and reuse this data in the future.



When you also have licenses either for the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA), for transient simulations you can additionally include probe data from the AM Calculator in the set up of the thermal profile for diffusion and precipitation calculations, respectively.

## *Workflow 2: When You Already Have the Material Properties Data Available - Material Library*

When you know you have the available material library data available, either by using the provided libraries, importing your own external data, or by saving your own material library after running a Scheil calculation, then you can directly start defining the AM Calculator and add and define one or more plots to visualize the results.



The Scheil Calculator and System Definer function the same as with a regular Thermo-Calc simulation. There are features available on the Scheil Calculator that are particularly useful to these types of calculations. The Plot Renderer is also unique to this calculator but is still based on the standard activity node.

The various activity nodes for each workflow are briefly described below.

### *System Definer*

The first step of the set-up is to select which database to use and define the material for the simulation. This is done in the System Definer.

In a **System Definer** activity, you select the database to use to retrieve thermodynamic data and define which elements the system has as components. You can also select the species to include as well as change the reference temperature and pressure for your components.

A **System Definer** node can be added either directly to the  **My Project** node or by default when the **Additive Manufacturing** template is added.



See [System Definer](#) and then to learn more about the many settings go to [Configuration Window Settings](#).

### *Scheil Calculator (Workflow 1)*

Once you have defined your system, you need to retrieve the materials data necessary for the AM calculations. This is done using the Scheil Calculator when you are working in Workflow 1.

A **Scheil Calculator** node can be added either directly to the **System Definer** node or by default when the **Additive Manufacturing** template is added.

### **About the Scheil Calculator Settings**

The Scheil Calculator in the Additive Manufacturing Module template is configured to generate the data necessary for the AM calculation. It is configured to start the simulation at a temperature of 5000 degrees and capture the evaporation and calculate the material properties down to room temperature. If you add a Scheil Calculator manually from the system definer, you will need to change these settings yourself. If you are working with different materials, different settings might be more suitable to your material.

### *Materials Libraries (Workflow 1 or 2)*

A Scheil calculation using the Scheil Calculator (Workflow 1) can be saved for reuse in the Materials Libraries. Alternatively, if there is external data available to use, you can skip the Scheil Calculator and just use a library (Workflow 2). There are also some predefined materials libraries included with the installation that can be used, and are used, in the examples (e.g. AM\_01, AM\_02, and AM\_03 and others) so that users without an AM license and necessary databases can see how this works.



[Available Options](#)

### *AM Calculator*

Add an **AM Calculator** node to the Scheil Calculator. If you used the **Additive Manufacturing** template (on the **My Projects Configuration** window under **Applications**), click the node to display the **Configuration** settings window.



Cuboid geometries are currently only available for transient simulations. Both Steady-state and Transient models use adaptive mesh refinement to decrease computational cost. You have the option to either choose from the predefined mesh settings that defines the maximum and minimum element size based on the process parameters, or specify the maximum and minimum element size yourself.

You can use this either with or without the Scheil and System Definer if you already have the materials properties calculation you want.



[AM Calculator](#) and [AM Calculator Conditions Settings](#)

### ***Choose Materials Data Source***

Once the Scheil simulation is complete, click on the AM Calculator 1 node and then the Materials Properties tab to see the material property data obtained from the Scheil calculation. In the Material Properties tab, you can plot the properties required for the AM simulation. This is also where you select the data source that will be used in the simulation. In the Use data from: drop-down list you can choose either Scheil Calculator or Library. The Scheil Calculator option uses the Scheil results that you just calculated, while the Library option uses the data that was shipped with the software or previously calculated data that you have saved to the Library.

### ***Visualizations Window***



[Working with AM Visualizations](#) and [Visualizing Heat Source Calibrations](#)

### ***Apply Data Smoothing***

Before you run the AM Calculator, it is important that the data you will base the AM calculation on does not have any sharp peaks or curves to be able to solve the numerical problem. To avoid this, you can apply smoothing to your data. The default setting is **Little smoothing** but this can be changed depending on your simulation.

To determine how much smoothing to apply, you can plot the different properties. In the plot drop-down list you can select which property you want to plot to check the data and if there are any sharp peaks or curves in the plot. The plot appears immediately when you select a property. It is recommended to try running the simulations with only applying **Little smoothing**. If the calculation fails, you can increase the smoothing and try again. It is also possible to apply different levels of smoothing on different properties.

## Additive Manufacturing Templates

There are two templates you can use to access the Additive Manufacturing Module settings and features in Thermo-Calc.



You can also add nodes individually to build your own tree.



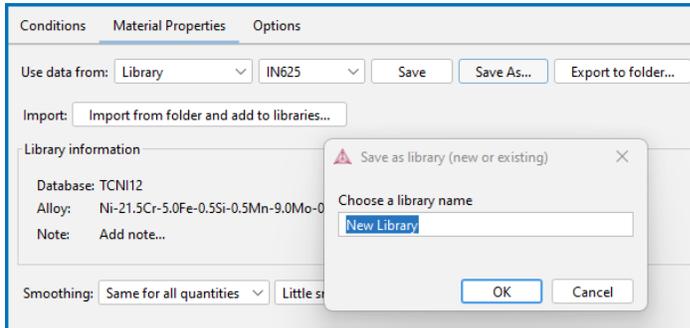
When you also have licenses either for the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA), for transient simulations you can additionally include probe data from the AM Calculator in the set up of the thermal profile for diffusion and precipitation calculations, respectively.

### **Additive Manufacturing Template**

On the main **Project Configuration** window, under **Applications**, click the **Additive Manufacturing** icon. This adds the following activity nodes as a tree in the Project window – **System Definer**, **Scheil Calculator**, **AM Calculator**, and **Plot Renderer**. When you add this Additive Manufacturing template (and need to generate some materials properties or do not have a material library to use), then some unique default settings for the Scheil Calculator and these particular types of simulations are already set by default. Then you continue defining the AM Calculator by choosing one of the available simulation types: Steady-state, Transient, or Transient with heat source from Steady-state.

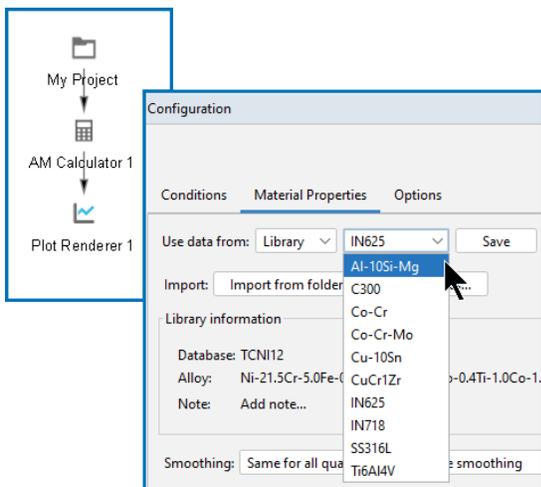


There is also an option on the AM Calculator to save the material properties in a material library, and use these later for a new AM Calculator where you want to make calculations related to the same alloy.



### ***With Material Library Template***

Click **With Material Library** to create an activity tree that only includes the AM Calculator and Plot Renderer nodes. The tree setup does not need the System Definer or Scheil Calculator. Instead you can select (or import) pre-saved Materials Properties data from a library and use these for your AM calculations. For example, if you have already run calculations for the Scheil Calculator for your alloy, and you want to perform a different AM calculation for the same alloy, this data can now be reused where you start the AM Calculator using the second option and use the material property data from the material library.



---

## Working with the Activity Nodes for an AM Simulation

---

### In this section:

---

AM Calculator .....	455
AM Calculator Conditions Settings .....	456
Heat Source Calibration Experiment File Requirements .....	467
AM Calculator Materials Properties Settings .....	470
AM Calculator Options Settings .....	472
AM Calculator: Working with Probe Data .....	474
AM Plot Types .....	475
AM Calculator Plot Renderer .....	478
Working with AM Visualizations .....	484
Visualizing Heat Source Calibrations .....	489
AM Visualizations Window Plot Toolbar .....	498
Rotate, Zoom, and Pan 3D Plots: AM Calculations .....	500

## AM Calculator

Depending how you set up your simulation, the **AM Calculator** can either be part of the workflow after a **Scheil Calculator** or directly at the root below the **My Project** node.



There is an **Additive Manufacturing** template available under **Applications** that you can use to quickly set up either one of these workflow options for the AM simulation. See [Additive Manufacturing Templates](#) and [Setting Up the Additive Manufacturing Simulation](#).

Once you have added an AM Calculator, the **Configuration** window has these tabs where there are many available features to set on the **Conditions**, **Materials Properties**, and **Options** sections and with the specialized **Plot Renderer** and toolbar on the **Visualizations** window.

There are these simulation type pathways to choose: **Steady-state**, **Transient**, or **Transient with heat source from Steady-state**.



In cases where convection is the dominant mode of heat transfer, fluid flow inside the melt pool plays a significant role in determining the correct shape of the melt pool.



For more details about these see [Additive Manufacturing Simulation Types](#). Also review the [Additive Manufacturing Module Theory](#) as needed.

- **Steady-state**: Compute temperature distribution in a steady-state environment, either on a bare metal substrate or with a powder layer on the top, with the possibility to add fluid flow inside the melt pool.
  - For this simulation type, you can also choose the **Heat source calibration Calculation Type** to calibrate a heat source based on the provided experimental data instead of performing a single point calculation. All types of heat sources can be used.

- **Transient:** Compute temperature distribution in a transient case with the given scanning strategy including multiple paths and layers and the possibility to add fluid flow inside the melt pool.
- **Transient with heat source from Steady-state:** Compute temperature distribution in a transient case with the given scanning strategy including multiple paths and layers.
  - A volume heat source is used in the transient simulations. This is based on the solution of a steady-state calculation.
- For a Gaussian heat source (*Steady-state* or *Transient with heat source from Steady-state*) you can also include a **Keyhole model**. This is available only when you exclude fluid flow or separate material properties for powder.

There are several settings and these are described separately for each tab.

- [AM Calculator Conditions Settings](#)
- [AM Calculator Materials Properties Settings](#)
- [AM Calculator Options Settings](#)

Also as a successor to the **AM Calculator**, you can right-click the node to add one or more **Plot Renderer** nodes to visualize your results. Although the node works the same as with the other calculators in Thermo-Calc, there are additional features and settings unique to this Module. Useful information is included throughout the documentation. See [AM Calculator Plot Renderer](#) as a starting point.



When setting up an AM simulation, the **Visualizations** window shows both active configuration changes and other design states as set up on the **AM Calculator** as well as the results generated by the **Plot Renderer** calculations. See [Working with AM Visualizations](#) and [Visualizing Heat Source Calibrations](#) for more information.

## AM Calculator Conditions Settings

Once you have added an [AM Calculator](#), the **Configuration** window has these settings tabs where there are many available conditions to set on the **Conditions**, **Materials Properties**, and **Options** tabs and with the specialized **Plot Renderer**.

This topic describes the available main settings on the **Conditions** tab.



[AM Calculator Materials Properties Settings](#) and [AM Calculator Options Settings](#)



[Additive Manufacturing Module Theory](#)

### *Calculation Type (Steady-state)*



Once a **Steady-state** simulation type is chosen, you can also choose to run a **Heat source calibration** calculation instead of only a **Single point**.

### ***Single Point***

By default for **Steady-state** calculations, **Single point** is selected. This relates to what is entered for the *Heat Source*, where settings for each heat source type (**Gaussian**, **Double-ellipsoidal**, and **Conical**) are entered as described in [Heat Source](#).

### ***Heat Source Calibration***



For some background, see [About Heat Source Calibration](#). Also see [Visualizing Heat Source Calibrations](#) for various options related to visualizing the calibration settings.

Select **Heat source calibration** to import or enter experimental data. See below for additional guidance.

## Experiment Data



It is important to prepare the experiment data file correctly so the simulation performs smoothly. See [Heat Source Calibration Experiment File Requirements](#).

In the **Experiment file** field, either enter a file path or click the file button () to navigate to a data file such as an Excel spreadsheet (with \*.xls or \*.xlsx extensions) or a \*.csv file. You can also use other programs (e.g. Google Sheets) to enter data and export to a CSV format that can be read by the software.

Click **Open** to import the data. In the **Experiment file** field you can see the file path to the selected file. The next time you click the file button () the program remembers the last location from where the file is opened.

Choose the delimiter—**Comma** (,), **Colon** (:), **Equals sign** (=), **Semicolon** (;), or **Tab**—that matches the delimiter in the experiment file and click the **Reload** () button.

## Global Settings

### Gas Pressure

Pressure is used to calculate the heat loss due to evaporation from the top surface of the given material.

**Gas pressure:** The pressure of the gas inside the build chamber. The default is 100000 **Pascal**. Select **Atmospheres** or **Bar**.



When you select a different unit the default value does not change.

Enter a different value in the field.

### Temperature Unit

**Temperature unit:** Select a unit **Kelvin**, **Celsius**, or **Fahrenheit**.

## Base Plate Temperature



If you change the **Temperature unit** then double check that the value in this field is also updated or correct as expected if you are keeping the default. This field does not update automatically when the unit changes.

**Base plate temperature:** The base plate of the build part is assumed to have a constant temperature. This temperature is used as a boundary condition for the base of the computational domain.

The default is 303.15 K.

## Ambient Temperature

The **Ambient temperature** is used to calculate the heat loss from the top surface due to radiation, convection and evaporation.

This setting is the temperature of the air in the immediate surrounding environment.



If you change the **Temperature unit** then double check that the value in this field is also updated or correct as expected if you are keeping the default. This field does not update automatically when the unit changes.

## Fluid Flow including Marangoni Effect

This section becomes available when the **Use separate material properties for powder** checkbox is not selected.



Also see the theory section, [Fluid Flow](#).

Choose to use **Fluid flow including Marangoni effect** if you want to calculate and include for the effect of fluid flow due to gradient of surface tension in the melt pool. The checkbox is selected by default. This solves the Navier-Stokes equation to include fluid flow inside the melt pool due to the Marangoni effect.

If you want to use separate properties for the powder layer, click to clear the checkbox to enable **Use separate material properties for powder** fields and then continue with the settings below.

---

## ***Use Separate Material Properties for Powder***

This option is available when the **Fluid flow including Marangoni effect** checkbox is NOT selected.

**Use separate material properties for powder:** Select to use different density and thermal conductivity for the powder layer; otherwise by default the same properties as for the solid are used.

**Powder density:** The powder density as a percentage (0-100%) of the solid material. The default is 80 % of the solid material.

## ***Geometry***

For a **Steady-state** simulation, it is just the height needed. For **Transient** or **Transient with heat source from Steady-state** these are volumetric and you also define the width and length. For all simulation types also define the coarseness of the mesh.

## ***Height for Steady-state Simulation***

**Height:** Enter the build height (mm). Length and width of the computational domain is automatically chosen as a function of the heat source parameters and the scanning speed.

## ***Height, Width, and Length for Transient Simulations***

- **Height:** Enter the initial build height (mm). Powder layer(s) with the given layer thickness will be added on the top. In mm.
- **Width:** Enter the build width (along y direction). In mm.
- **Length:** Enter the build length (along x direction). In mm.

## ***Mesh***

**Mesh:** The initial mesh size —**Coarse, Medium, Fine, or Custom**. It is adaptive and refined as required.

If **Custom** is selected, enter the maximum and minimum element sizes:

- **Maximum element size:** The initial element size (in  $\mu\text{m}$ ) in the mesh.
- **Minimum element size:** The minimum element size (in  $\mu\text{m}$ ) that the adaptive mesh can refine to.

## Heat Source

The heat source model has either a **Gaussian**, **Double ellipsoidal**, or **Conical distribution**.



[Modeling Heat Source](#) in the theory section. Also see [About Heat Source Calibration](#) for additional details.

**Gaussian** is a surface heat source, and it is applied to the top surface of the domain. Then the heat is diffused based on the diffusivity of the material. This is most commonly used heat source in the industry

But in cases where the melt pool tends to form in a keyhole formation then you need to include more multiphysics phenomena to get the correct shape of the melt pool. Then the volumetric heat sources give you better control of the shape of the melt pool and the temperature gradient around the heat source.



With a **Gaussian** heat source, you can also use the keyhole model. See [Keyhole Model Settings](#) below.

**Double ellipsoidal**, also referred to as Goldak, is a volume heat source model where the heat intensity is in the front and the back of the heat source is defined with the help of two different ellipsoids. The axis of the each ellipsoid is needed.

The **Conical distribution** is also a volumetric heat source where heat intensity is described as Gaussian distribution at the top surface, defined by a top radius, that varies linearly in the depth of the workpiece until it reaches a bottom radius within the prescribed depth.

After choosing the type of heat source, then enter parameters as required.

For all models enter:

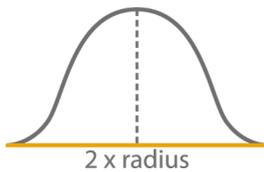
- **Power:** Power for the selected heat source model. The default is 120 W.
- **Absorptivity:** Absorptivity % for the selected heat source model. The default is 60 %.



Once the **Users heat source** is available to choose in the heat source list, then only the **Power** of the heat source is needed to be entered and the rest of the parameters, including the absorptivity, are automatically included in the simulation based on the selected expressions when originally saving the heat source calibration data.

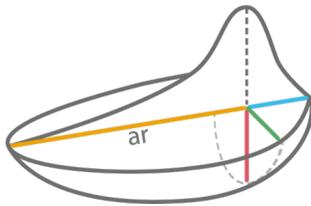
For **Gaussian**, also enter:

- **Beam radius:** Beam radius size for the **Gaussian** heat source model. The default is 110  $\mu\text{m}$ . Hover over the field in the GUI to view this example of it.

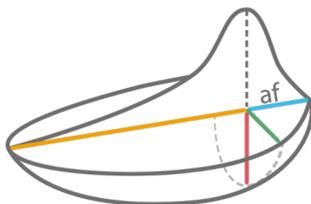


For **Double ellipsoidal**, also enter as required. For each field you can hover over it in the GUI to see these images that show you the location.

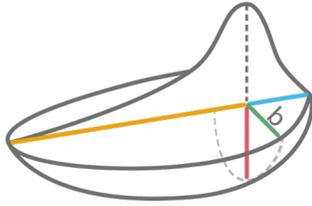
- **ar** (default is 70  $\mu\text{m}$ ).



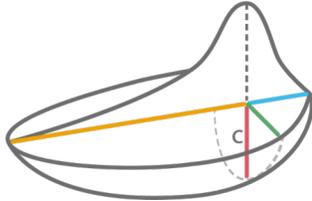
- **af** (default is 70  $\mu\text{m}$ ).



- **b** (default is 85  $\mu\text{m}$ ).

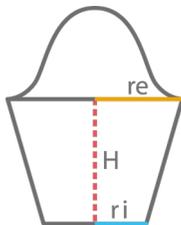


- **c** (default is 200 μm).



For **Conical distribution**, also enter the following. For each field you can hover over it in the GUI to see these images that show you the location.

- **re** (default is 100 μm).
- **ri** (default is 60 μm).
- **H** (default is 100 μm).



## KEYHOLE MODEL SETTINGS



[About the Keyhole Model](#)

To use the keyhole feature requires:

- A model without fluid flow, and without separate material properties for powder.
- Gaussian heat source type.

---

Select the **Use keyhole model** checkbox to enable the feature. Then enter a **Rayleigh length** of the laser beam in the field (unit = mm). The default is 2.5 mm.

### *Scanning Strategy*

The scanning strategy determines the build geometry for one or multiple layers. Green layer on the top of the geometry is the powder and the grey part is the solid substrate.

For **Steady-state** simulations, you only need to define the **Scanning speed** and **Layer thickness**.

- **Scanning speed:** The velocity of the moving heat source. The default is 500 mm/s.
- **Layer thickness:** The thickness of the powder layer. The default is 40 mm. When you change the layer thickness, the meshing size changes too.

For both transient simulations, you continue to define the settings below.

### **Pattern**

**Pattern:** Select the scanning pattern—**Single track**, **Bidirectional**, or **Unidirectional**.

- Select **Unidirectional** to use same scanning direction of the heat source for all tracks.
- Select **Bidirectional** to flip scanning direction of the heat source between alternate tracks.

### **Single Track**

- **Margin:** Offset of the laser scanning path from the sides of the computational domain. In case of Single track, offset is placed from the sides transverse to the scanning direction, whereas the scanning path is always in the middle of the domain going along the length of the workpiece.
- **Number of layers:** Enter the number of layers or scroll using the arrows.
- **Powder fill time:** Powder recoating time (unit = s) between two consecutive layers. This does not include lift time.

## Bidirectional and Unidirectional

- **Margin:** Offset of the laser scanning path from the sides of the computational domain.
- **Hatch spacing:** Separation (unit = mm) between two consecutive tracks.
- **Lift time:** Time (unit = s) between two tracks where the heat source is inactive.
- **Number of layers:** Enter the number of layers or scroll using the arrows.
- **Powder fill time:** Powder recoating time (unit = s) between two consecutive layers. This does not include lift time.
- **Rotation between layers:** Rotation (unit = degrees) of the scanning direction between two consecutive layers. The scanning pattern of the first layer is always aligned to the X axis then the second layer will be rotated (if you choose this setting) with the specified angle then the next layer will be rotated at the same angle as compared to the previous layer and so on. This way you can change the rotation of the layers.

### *Top Boundary Conditions*



[Numerical Boundary Conditions](#) in the theory section.

### ***Radiation Emissivity***

**Radiation emissivity:** To include the effect of radiation from the top surface to the surrounding gas, enter a value between 0 and 1. If you want to neglect the heat loss, then enter 0 to disable radiation.

### ***Convective Heat Transfer Coefficient***

**Convective heat transfer coefficient:** The convective heat transfer coefficient for the top surface to the surrounding gas. If you want to ignore this then enter 0 to disable convective heat transfer.

## Evaporation

**Evaporation:** Select the checkbox to include the effect of evaporation heat loss due to heating of the powder layer or the metallic surface close to the evaporation temperature.



The **Evaporation** heat loss is calculated based on the settings on the **Materials Properties** tab, i.e. evaporation enthalpy per mole, the driving force for evaporation per mole quantity and the molar mass of gas. These are also calculated by the databases.

## Probe Positions (Transient Simulations Only)



It is important to add the probes locations before the simulation is run or the action is not recorded.

You can add probes in the domain that record temperature and all other temperature related properties when the simulation is being run.

Click the Add  and Remove  buttons to add each probe and then you can either manually enter the probe coordinates (enter the **x**, **y**, and **z** coordinates in the fields) or click **Pick coordinate** to choose each of the points specifically on the **Geometry** tab in the **Visualizations** window.



[AM Calculator: Working with Probe Data](#)

## USING PROBE DATA IN DIFFUSION AND PRECIPITATION CALCULATIONS



When you also have licenses either for the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA), for transient simulations you can additionally include probe data from the AM Calculator in the set up of the thermal profile for diffusion and precipitation calculations, respectively.

## Heat Source Calibration Experiment File Requirements



It is important that the experiment data file contents are correctly entered in order for the calculation to perform smoothly.

When the **Heat source calibration** *Calculation Type* is selected in the **Steady-state** mode, there is an option to import a file with experimental data points that are used to calibrate the heat source.

Data can also be entered directly into the table on the AM Calculator Configuration window. However, if an **Experiment file** is imported it needs to be set up in a certain way so that the software can read the data.



There are these examples that use experiment data, key holing, and heat source calibration: [AM\\_06a: Calibrating a Heat Source for a 316L Steel](#) and [AM\\_06b: Using the Calibrated Heat Source for a 316L Steel](#).

Experiment Data							
Experiment file						delimiter	Comma
#	Power (W)	Speed (mm/s)	P/v (J/mm)	Width (μm)	Depth (μm)	Use experiment	
1	60.000000	2400.000000	0.025000	36.842110	9.444444	<input checked="" type="checkbox"/>	
2	60.000000	2000.000000	0.030000	39.009290	10.555560	<input type="checkbox"/>	
3	50.000000	1200.000000	0.041667	51.075950	9.303797	<input type="checkbox"/>	
4	60.000000	1600.000000	0.037500	46.439630	11.666670	<input type="checkbox"/>	

Figure 1: From example *AM\_06a*, this shows the data imported from an experiment file into the table. The "Use experiment" column allows you to choose the points to include or exclude for the simulation. Here row #1 is selected to show the data point. Further down in this list, rows 9, 14, 20, and 21 are also selected to be included.

### Experiment File Requirements



Only the columns described here are read / used in the calculation. It is important to enter the information exactly as described. All header text (the first line in the file) must also be written in English, i.e. no translations into other languages will be read correctly by the experiment file.

The experiment file is set up as follows.

- The first row in the file is the header.
- Include only four columns in the file and these must be in the following order and must use the indicated units.



Default Global units (i.e. set via the **Tools** → **Options** menu) are not applicable. The data must be in the indicated units.

- **Power** (unit = W)
  - Scanning **Speed** (unit = mm/s)
  - **Width** of the melt pool (unit =  $\mu\text{m}$ )
  - **Depth** of the melt pool (unit =  $\mu\text{m}$ )
- All of the values entered or imported to the table must be positive and greater than zero.

Do not include the following in the file:

- **#** this is the experiment number that is automatically associated to each imported or entered set of data. It is used to identify the applicable rows by a unique number so that you can easily identify these points (i.e. points are identified by labels on the plots), or when choosing to include or exclude the experiment point from the simulation or later from a plot.
- **P/v (J/mm)**. This is the energy density and is defined as the ratio of power and scanning speed. This is not included in any data file. This is automatically calculated based on the Power and Speed entered or included in the file (i.e it is defined as the ratio of power and scanning speed).
- **Use experiment**. This column is not part of the text file, it is available once the data is imported or entered in the table. Then you can choose to include or exclude specific data points as you examine this output in the Visualizations window. See [Working with AM Visualizations](#) for examples.

	A	B	C	D
1	Power	Speed	Width_avg	Depth_avg
2	50	800	57.27	27.93
3	100	2800	47.82	0
4	40	400	58.46	14.55
5	60	800	68.42	31.11

#	Power (W)	Speed (mm/s)	P/v (J/mm)	Width ( $\mu\text{m}$ )	Depth ( $\mu\text{m}$ )	Use experiment
1	50.000000	800.000000	0.062500	57.270000	27.930000	<input checked="" type="checkbox"/>
2	100.000000	2800.000000	0.035714	47.820000	0.000000	<input checked="" type="checkbox"/>
3	40.000000	400.000000	0.100000	58.460000	14.550000	<input checked="" type="checkbox"/>
4	60.000000	800.000000	0.075000	68.420000	31.110000	<input checked="" type="checkbox"/>

Figure 2: An example of the original experiment data file, with the four headings in the correct order and these do not need the units included here. When the file is imported, there is a standard header that is kept, plus the additional columns (i.e. #, P/V, and Use experiment), which are automatically available and updated as needed.

### Working with the Experiment Data Table



Once a set of data is imported to the table (or entered manually), expand the **Configuration** window to reveal the scroll bars. Then you can scroll up or down, left or right, within the table to select checkboxes for the relevant data points you want to include or exclude. This is applicable to the tables found on the AM Calculator as well as those on the Plot Renderer for the heat calibration data.

Experiment Data

Experiment file  delimiter

#	Power (W)	Speed (mm)	Depth ( $\mu\text{m}$ )	Use experiment
1	60.000000	2400.000000	9.444444	<input checked="" type="checkbox"/>
2	60.000000	2000.000000	10.555560	<input type="checkbox"/>
3	50.000000	1200.000000	9.303797	<input type="checkbox"/>

Melt pool

#	Power (W)	Speed (mm)	Depth ( $\mu\text{m}$ )	Use experiment
7	60.000000	1200.000000	12.109700	<input type="checkbox"/>
8	50.000000	800.000000	27.931030	<input type="checkbox"/>
9	100.000000	2800.000000	14.364410	<input checked="" type="checkbox"/>
10	40.000000	400.000000	14.545450	<input type="checkbox"/>
11	70.000000	1200.000000	23.776370	<input type="checkbox"/>
12	60.000000	800.000000	31.111110	<input type="checkbox"/>
13	80.000000	1200.000000	29.683540	<input type="checkbox"/>
14	50.000000	400.000000	56.103900	<input checked="" type="checkbox"/>
15	90.000000	1200.000000	43.565400	<input type="checkbox"/>

You can edit the table after importing data, or manually enter information. To add a row, scroll to the bottom of the table and press Enter.

## AM Calculator Materials Properties Settings

Once you have added an [AM Calculator](#), the **Configuration** window has these settings tabs where there are many available conditions to set on the **Conditions**, **Materials Properties**, and **Options** tabs and with the specialized **Plot Renderer**.

This section describes the settings for the **Materials Properties** tab.

 [AM Calculator Conditions Settings](#) and [AM Calculator Options Settings](#)

### Use Data From

If there is a Scheil Calculator predecessor to the AM Calculator, you can import the material properties from a **Library** or **Scheil Calculator**. When the AM template is added, the default is to import material properties from Scheil calculations.

When the **with Material Library** template is used, then the AM Calculator > Materials properties are available to import from a **Library** only.

---

In this case you also can **Save**, **Save as**, **Export to folder**, **Delete**, and **Rename** the material data both from Scheil and from material library. With Save or Save as, original data is saved i.e. without any smoothing. With Export to folder, the data is saved with Smoothing. the material data both from Scheil and from material library.

## ***Import***

You can import material properties contained in a folder.

Click the **Import from folder and add to libraries** button and navigate to the folder location where the material properties are located.

## ***Smoothing***

From the **Smoothing** list, select **Same for all quantities** (where **Little smoothing** is the default) or **Per quantity**. Then either define the smoothing once (**Same for all quantities**) or individually for each quantity. Options are **No smoothing**, **Little smoothing**, **Medium smoothing**, **Large smoothing**, or **Linear**.

Available quantities to define smoothing:

- Heat
- Apparent heat capacity per kg
- Density
- Thermal conductivity
- Dynamic viscosity
- Surface tension
- Molar volume
- Molar mass of gas
- Driving force for evaporation per mole
- Evaporation enthalpy per mole

## ***Plot***

From the **Plot** list, choose what quantity to plot

- Heat
- Apparent heat capacity per kg
- Density
- Thermal conductivity
- Dynamic viscosity
- Surface tension
- Molar volume
- Molar mass of gas
- Driving force for evaporation per mole
- Evaporation enthalpy per mole

### ***Show Transition Temperatures***

You can also choose to **Show transition temperatures**:

- Solidus, Liquidus, Evaporation
- Solidus
- Liquidus
- Evaporation
- Custom: If selected, also enter a value in the field. The default unit is the same as what is selected globally on the **Tools → Options → Graphical Mode → Default Units** tab.
- None

## **AM Calculator Options Settings**

Once you have added an [AM Calculator](#), the **Configuration** window has these settings tabs where there are many available conditions to set on the **Conditions**, **Materials Properties**, and **Options** tabs and with the specialized **Plot Renderer**.

This section describes the settings for the **Options** tab.



[AM Calculator Conditions Settings](#) and [AM Calculator Materials Properties Settings](#)

---

## ***Streamline Upwind Petrov-Galerkin (SUPG)***

Select the **Streamline upwind Petrov-Galerkin (SUPG)** checkbox to enable SUPG stabilization scheme for the energy and Navier-Stokes equations.

## ***Damping Factor for the Damped Newton Iteration***

Select the **Damping factor for the damped Newton iteration** checkbox to add a damping factor for the damped Newton iteration and enter a value between 0 and 1. The default is 0.7.

It may improve stability but it slows down convergence of the simulation.

## ***Save Interval for Transient Simulation***

Select the interval to save results from transient simulations. If the number of files saved are specified, choose if the step size should increase linearly or exponentially.

From the **Save interval for transient simulation** list choose **Automatic**, **Every Nth time step**, or **Time in seconds**. Then for each also enter as indicated:

- **Automatic**
  - **Max number of files:** The default is 100. Choose **Linear**.
  - Then choose **Linear** or **Exponentially increasing**. Enter another number of files in the field as needed.
- **Every Nth time step:** The default is 1 or enter or use the up and down arrows to choose another number.
- **Time in seconds:** The default is 0.01 or enter another number in the field.

## ***Number of Cores***

Select or enter the **Number of cores** available for the simulations. The default value is the total number of physical cores available in the system.

## AM Calculator: Working with Probe Data



When you also have licenses either for the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA), for transient simulations you can additionally include probe data from the AM Calculator in the set up of the thermal profile for diffusion and precipitation calculations, respectively.

### *Workflow to Include Probe Data from Diffusion or Precipitation Simulations*



You can use an installed example to visualize how the probes are set up for a Diffusion Calculator. See [AM\\_05: Using AM Calculator Probe Data with the Diffusion Module \(DICTRA\)](#). The process is similar for a Precipitation Calculator.

1. Set up the probes on the **AM Calculator**. Probes are available for transient calculations only. See [AM Calculator Conditions Settings](#). This must be done before you can use the probe data in the calculations connected to a Diffusion Calculator or Precipitation Calculator.
2. If not already done, add a **Diffusion Calculator** or **Precipitation Calculator** as a successor to the **AM Calculator**.
3. Set up your Diffusion or Precipitation Calculator as needed. Follow the basic settings details for the **Thermal Profile** section as described in [Diffusion Calculator Thermal Profile](#) and [Precipitation Calculator Thermal Profile](#).
4. Once the standard settings are completed, and as long as probes are set up, on the **Configuration** window, the section **Probes** is available. Until the AM Calculator is run, the list is empty. After the AM simulation is completed, the data is imported into the table you can choose how to **Condense time-temperature data**. These settings are described below.

### *Probes*

The list of **Probes** is available as long as *Probe Positions* are set up on the AM Calculator and the AM Calculator is performed to generate the data, which is fed into the applicable Calculator (Diffusion or Precipitation). Select the probe you want to plot, for example as a Segregation Profile as shown in example AM\_05.

## Condense Time-Temperature Data

Use the **Condense time-temperature data** setting to adjust how many time-temperature points are included in the **Thermal Profile** displayed in the **Visualizations** window. For example, for a complex calculation you can choose a **Medium** or **Large** setting to save fewer time steps at this stage of design, which in turn can ensure that a complex diffusion or precipitation calculation is able to finish.

Choose **None**, **Small**, **Medium**, **Large**, or **Custom**.

- **None** (the default) includes all points, i.e. no points are condensed together and all point data are included in the **Thermal Profile** preview. When using the **Custom** setting, this is equivalent to entering a value greater than 1 in the field.
- **Small** condenses a minimum number of points, which means that a *large* number of time steps are used in the calculation and thus lots of time steps are shown.
- **Medium** is when that a medium number of points are condensed together, which means that a *medium* number of time steps are used in the calculation.
- **Large** is when a large number of points are condensed together, which means that there are few, or a *small* number of time steps used in the calculation.
- When **Custom** is selected you can fine-tune the number of points to include. Enter a value between 0 and 1, where 0 maximizes condensing the points (the same as choosing **Large**), 1 minimizes condensing the points (the same as choosing **Small**), and 0.5 is a medium number of points being condensed (the same as choosing **Medium**). If you enter a value greater than 1, it is the same as choosing **None**, i.e. all data points are included. The larger the value, the more points are condensed, until only two points are shown.

## AM Plot Types



The AM Calculator visualization using the Plot Renderer is different from the other Calculators available in Thermo-Calc.



See [Plot Types](#) for more information for the Scheil Calculator and Diffusion Calculator, which are also used with the Additive Manufacturing (AM) Module.

## AM Calculator Plot Renderer Successors

When a Plot Renderer is added as a successor to the AM Calculator, the default layout of the Configuration window relates to how the AM Calculator is set up. The following provides information about each of the plot types (3D Plot, Plot Over Line, and Probe) that have tabs to define settings (on the Configuration window) as well as associated tabs to view the output (on the Visualizations window).



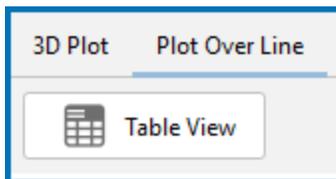
See [Working with AM Visualizations](#) for information about how these tabs are connected between the windows.

Plot Type	Simulation Type	Notes
3D Plot, Plot Over Line (2D)	Steady-state	No probe positions or time steps. Defined and available from a specific a tab on the Plot Configuration window.
3D Plot, Plot Over Line (2D), Probe (2D)	Transient, Transient with heat source from Steady-state	Probe positions and time steps available. Defined and available from a specific a tab on the Plot Configuration window.
Heat source calibration parameters and Melt pool dimensions	Steady-state plus Heat source calibration	These plots are available once the calibration is completed.

In addition to the Plot Types listed in the table, for 3D plots there are additional **Display** options available via the settings on the individual tabs on the Configuration window. For 2D plots you can also switch between a plot and a Table View using the button available on the Visualizations window. For a **Heat source calibration** calculation, there are also plots that become available after the experiment data is included and the simulation run.

### Table View (2D)

For 2D plots (Plot Over Line and Probe), and once the plot is performed, you can switch between a plot and a table by clicking the **Table View** button on the Visualizations window. It is available with all calculation types (Steady-state, Transient, and Transient with heat source from Steady-state).



## 3D Plot

The **3D Plot** type has its own tab on the Configuration and Visualizations windows. This plot type is further visualized using additional **Display** options or you can choose to define it as a **Slice**, also with additional settings. These settings for the 3D plot are defined on the Configuration window for the Plot Renderer. The 3D Plot is available with all calculation types (Steady-state, Transient, and Transient with heat source from Steady-state).

### 3D PLOT DISPLAY OPTIONS

The **Display** options available are also based on the **Axis variable** selected as listed in the table.

<i>3D Plot Display Type</i>	<i>Axis Variable</i>
Surface colormap	Available when Temperature, Surface tension, Thermal conductivity, or Dynamic viscosity is selected as an Axis variable.
Contour	Available when Temperature, Surface tension, Thermal conductivity, or Dynamic viscosity is selected as an Axis variable.
Iso-surface	Available when Temperature, Surface tension, Thermal conductivity, or Dynamic viscosity is selected as an Axis variable.
Arrow, 2D glyph, Cone, or Line	Available when Flow velocity field is selected as an Axis variable. Requires that the <b>Fluid flow including Marangoni effect</b> checkbox is selected on the AM Calculator.

### 3D SLICE PLOT

The **Slice** plot is a unique 3D plot available when the checkbox is selected. Further settings are then defined.

#### *Plot Over Line (2D)*

The **Plot Over Line** plot type is a 2D plot. On the Visualizations window it is available on the **Plot Over Line** tab and on the Configuration window on the tab of the same name.

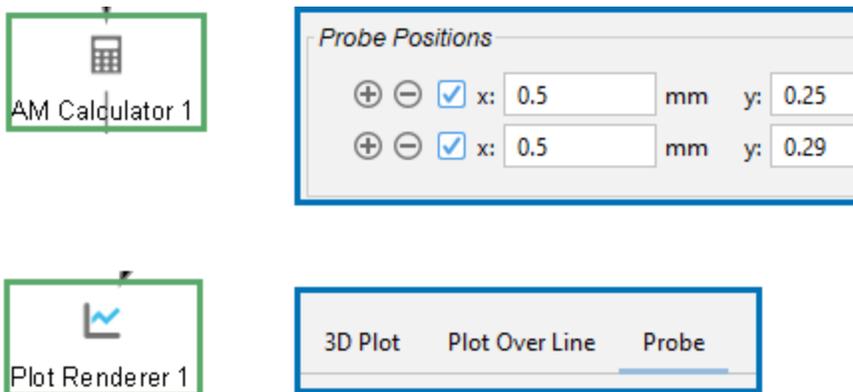
The **Plot Over Line** plot type is available with all calculation types (Steady-state, Transient, and Transient with heat source from Steady-state).

## Probe (2D)

The **Probe** plot type is a 2D plot. On the Visualizations window it is available on the **Probe** tab and on the Configuration window on the tab of the same name.

The **Probe** plot type is available with the Transient and Transient with heat source from Steady-state calculation types.

To define a Probe Plot requires that Probe Positions are selected on the AM Calculator. Then when a Plot Renderer is added as a successor, a Probe tab is available on the Configuration window where you can further define the plot.



Probe data can also be used with the Diffusion Calculator and Precipitation Calculator, when these Calculators are added as successors to the AM Calculator.

## Heat Source Calibration

When a heat source calibration calculation is performed there are additional plots available: *Heat source calibration parameters* and *Melt pool dimensions*.



[Visualizing Heat Source Calibrations](#)

## AM Calculator Plot Renderer

Once you have added an [AM Calculator](#), you can add the specialized **Plot Renderer**. The following is information about the settings available for a **Plot Renderer** when it is a successor to an **AM Calculator**.

The settings described below are made available based on a variety of parameters, e.g. the type of calculation (**Steady-state**, **Transient**, or **Transient with heat source from Steady-state**), whether fluid flow is included, whether probes are used, and so on. The software guides you by only providing the relevant settings to define.



Due to the many options, the following is not in any specific order and you may need to scroll or search for a particular setting as its availability can be for multiple plot types.

You can plot the material properties (2D) so you can specify the quantities and add as many lines as you want directly to the plot in the Visualizations window. What you see is **Distance vs Temperature**, first is first layer, second is second layer, and so forth on the 2D plots. You can select the different times from the list at the top at different points and see it on the plot, for example it might be at 26 minutes where you can see it is as the peak of the melt pool max temperature.



[Working with AM Visualizations and Visualizing Heat Source Calibrations](#)



[Rotate, Zoom, and Pan 3D Plots: AM Calculations](#)



[AM Visualizations Window Plot Toolbar](#)

## *Plot Settings*

### ***Time***

Once a Transient simulation plot is calculated, the **Time** step results are listed in a drop-down list. You can choose to look at a specific time point, or click the buttons to the right of the drop-down to navigate the time steps. Hover over the buttons to see the tooltips or see [Time Step Navigation Buttons](#).

### ***Time Step Navigation Buttons***



<i>Button</i>	<i>Action</i>
⏪	Move to the first time step
<	Move to the previous time step
▶	Play from the current time step to the last one
>	Move to the next time step
⏩	Move to the last time step

### ***Rescale Plotting Data Range Over All Time Steps***

This is available when a plot has time steps.

Select the **Rescale plotting data range over all time steps** checkbox to scale min and max values of legends and plots over all time steps.

### ***Filter by Material Types***

This option is only available when **Use separate material properties for powder** is selected on the AM Calculator.

Select **Filter by material types** to visualize a specific material type(s) from the list.

### ***Heat Source Calibration Experiments***

The **Heat source calibration experiments** list is available for a **Heat source calibration** calculation. Experiment data and other settings must be configured for this to be available.

The points available from the list are selected on the **AM Calculator** under *Experiment Data* and then you can include or exclude points by selecting or deselecting the relevant checkboxes in the **Use experiment** column.



See [AM Calculator Conditions Settings](#) and [Visualizing Heat Source Calibrations](#) for more detailed information about heat source calibration visualizations.

---

## Add a Quantity and Remove a Quantity Buttons

Use the **Add quantity**  and **Remove quantity**  buttons to add additional setting groups that include all of the available settings described below. Click the checkbox next to these buttons to enable or disable any particular settings group.

## Axis Variable

Set the state variable you want plotted along the X-axis and the Y-axis. The available variables in the list are based on how your system is set up.

Choose an **Axis variable** and an applicable unit (indicated in the brackets) from each of the drop-down lists.

- **Temperature** (Kelvin, Celsius, or Fahrenheit), **Surface tension** (Joule per m<sup>2</sup> or Millijoule per m<sup>2</sup>), **Thermal conductivity** (unitless).
- Available with fluid flow: **Dynamic viscosity** (Pascal-second, milliPascal-second, Poise, or centiPoise), or **Flow velocity field** (m/s, mm/s, μm/s, or nm/s).

## Display

Based on the **Axis variable** selected, choose the way to **Display** it.

- For **Temperature**, **Surface tension**, **Thermal conductivity**, or **Dynamic viscosity**, choose **Contour**, **Surface colormap**, or **Iso-surface**.
- For **Flow velocity field**, choose the object shape to display velocity vectors: **Arrow**, **2D glyph**, **Cone**, or **Line**.

The settings are further described in separate sections below.

## Tubes

Available for a **Contour** Display.

The **Tubes** checkbox is selected by default and this displays contour plots as 3D tubes. Click to clear the checkbox and instead display the contours as 2D lines.

---

## Values

Available for display types **Contour** and **Iso-surface**. For Iso-surface you can also specify the opacity of surface plot in %.

Choose the **Values: Automatic** (the default), **Custom values**, **Custom range**, **Solidus**, **Liquidus**, or **Solidus and Liquidus**.

- For **Automatic**, enter the **Number of steps**.
- For **Custom values** enter a number in the field.
- For **Custom range** enter the **Min** and **Max** number of lines, then **Number of steps**.
- Select **Solidus** to plot contour/iso-surface at solidus temperature.
- Select **Liquidus** and **Solidus and Liquidus** to plot contour/iso-surface at both solidus and liquidus temperatures.

Some settings are further described in separate sections below.

## Number of Steps

Available for **Contour** and **Iso-surface** Display types, and when **Automatic** or **Custom range** Values are selected.

Enter or choose the **Number of steps**.



In some cases for a **Custom range** selection, you need to expand the Configuration window to the right to access this setting.

## Opacity

Available for **Surface colormap** and **Iso-surface** (available for all **Values** options).

Enter a value between 0-100 as the percent (%) of **Opacity**.

---

## ***Slice***

Select the **Slice** checkbox to visualize the **Axis variable** on a cross-sectional plane. You can choose the normal of the cross-sectional plane and the distance of the plane from the center of the heat source.

Then choose **X-direction**, **Y-direction**, or **Z-direction**, for the **Distance from center of heat source** and for the **beam radius**.

## ***Coloring and Range***

Choose **Rainbow**, **Cool to warm**, **Purple Orange**, **Green Purple**, **Blue Brown**, **Green Red**, **Solid color**.

For all but **Solid color**, choose a **Range**—**Automatic** or **Custom**. Then for **Custom** enter a **Min** and **Max** number.

## ***Legend***

Click the **Legend** checkbox to include or click to clear to exclude a legend.

## ***Scale Glyph by Velocity Magnitude***

This setting is for a **Flow velocity field Axis variable**. It is available for all **Display** types.

Select the **Scale glyph by velocity magnitude** checkbox to scale the velocity display object (glyph) according to the velocity magnitude. Otherwise the glyphs are scaled uniformly.

## ***Glyph Scale Factor***

This setting is for a **Flow velocity field Axis variable**.

Enter a numerical value for the **Glyph scale factor**.

## ***Show Glyph in Every***

This setting is for a **Flow velocity field Axis variable**.

For the **Show glyph in every setting**, enter or select the number of grid **points** where velocity glyphs are displayed.

## Show Transition Temperatures

Available with 2D plot types, such as Probe or Plot Over Line. It is a dynamic list that updates the plot in the Visualizations window after the 2D plot is performed.

Select the different options to better visualize the transition temperature results. From the **Show transition temperatures** list, choose **Solidus**, **Liquidus**, **Evaporation**, **Solidus**, **Liquidus**, **Evaporation**, **Custom**, or **None**. For **Custom**, also enter a specific temperature value in the field. Then observe the changes on the plot, where the legend updates and the lines for each option are added or removed based on the selection.

## Working with AM Visualizations



The AM Calculator visualization using the Plot Renderer is different from the other Calculators available in Thermo-Calc.



See [Plot Types](#) for more information for the Scheil Calculator and Diffusion Calculator, which are also used with the Additive Manufacturing (AM) Module.

### *How the Project Nodes are Connected to the Configuration and Visualizations Tabs*

The following examples use a mix of adding an AM template and example AM\_01 to provide an overview of how the nodes in the **Project** window are connected to the **Configuration** window settings tabs and to the two levels of tabs in the **Visualizations** window. Try this yourself either by adding an AM template or open any of the examples and click around to understand how the geometry, heat source calibration (when applicable), and various plots are connected between all the windows.

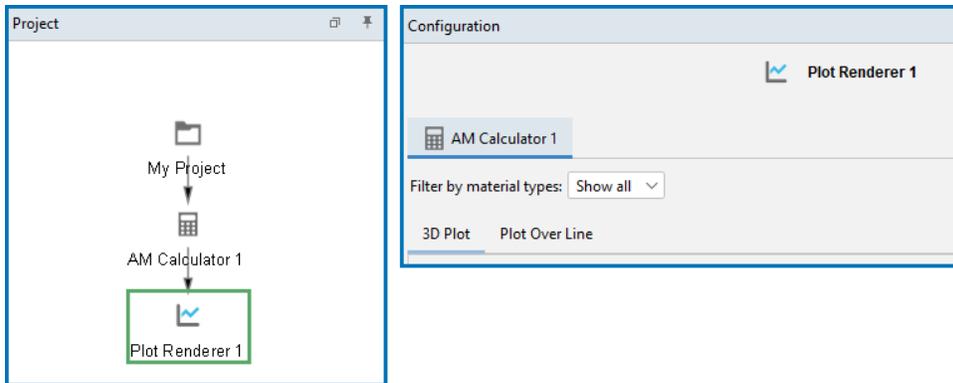
In general:

- In the Project window, the AM Calculator, Diffusion Calculator, and Precipitation Calculator nodes, plus all the associated successor Plot Renderer or Table Renderer nodes, are listed in the Visualizations window on the top level tabs using the same name as the node.
- It is recommended you rename the nodes in the Project window to meaningful names that make it easier to navigate the visualizations.

- In the Visualizations window, a **Geometry** tab is always associated to the **AM Calculator** node in the Project window. In addition, when a *Heat source calibration* calculation is chosen on the **AM Calculator**, there is also a **Heat source calibration** secondary level tab associated to this (at the same level as the **Geometry** tab).
- An AM Calculator and all its Plot Renderer successors are listed on the top level tab. The secondary level tab has visualizations related to geometry, heat source calibration, and the available plot types (3D Plot, Plot Over Line, Probe Plot, and the plots associated to heat source calibration).
- The Diffusion Calculator and any Plot Renderer or Table Renderer successors are all listed in the Visualizations window on the top level tab. The secondary level tab has visualizations related to the Composition Profile and Thermal Profile.
- The Precipitation Calculator and any Plot Renderer or Table Renderer successors are all listed in the Visualizations window on the top level tab. The secondary level tab has visualizations related to the Thermal Profile.
- The Scheil Calculator is not listed on the top level tab in the Visualizations window, but any Plot Renderer or Table Renderer successor for this Calculator is listed on the top level tab.
- Renaming any node in the Project window also renames the top level tabs in the Visualizations window.
- The second level tabs on the Visualizations window cannot be renamed as these represent the visualization type named by the system e.g. a geometry, plot type (3D Plot, Plot Over Line, or Probe Plot), or a Composition Profile (diffusion simulations) or Thermal Profile (diffusion or precipitation simulations).
- For 2D plots (Plot Over Line 2D Plot, Probe 2D Plot), click the **Table View** button on the applicable tab in the Visualizations window to convert the values into a table format.

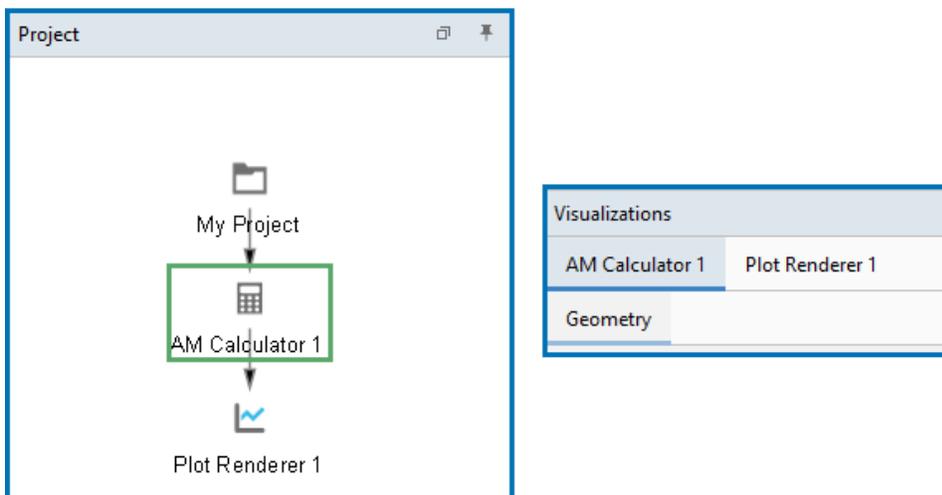
## CONFIGURATION WINDOW

When a Plot Renderer is added to an AM Calculator in the Project window, there are default tabs added to the Configuration window where you can further set up the available plot types. In this example, two tabs, **3D Plot** and **Plot Over Line**, are provided for you to set up these plot types.



## VISUALIZATIONS WINDOW

In the Visualizations window, the upper tab level is connected to a second level of tabs that represent the available visualizations. A **Geometry** tab is always associated to the **AM Calculator** node. Then for any **Plot Renderer** node, all of the plotting types such as a **3D Plot**, **Plot Over Line**, or **Probe**, are listed in the secondary tab level.



There is not always something to visualize at the second tab level. Keep an eye on the underlined tabs on both levels, i.e. click the upper level tab and look for the underlined second level tab associated to it and that is what is displayed. Also note that clicking the upper level tabs in the Visualizations window does not change the focus of the node clicked in the Project window, although the opposite is true.

Each node in the Project window is represented across the top set of tabs in the Visualizations window. Then the associated tab on the Visualizations window displays below as a secondary level. For example, in the **Project** window, if you click the AM Calculator node (in AM\_01 it is renamed to **Transient, DE HS**), then on the Visualizations window the **Transient, DE HS** tab is highlighted and has a blue underline. On the second tab level, the **Geometry** tab is also underlined to indicate it is associated to the **Transient, DE HS** tab. What is actually displayed in the Visualizations window is based on the second tab.

Then for each Plot Renderer added and defined, the secondary tab level is connected to what is visualized. For example, a Plot Renderer is added to the AM Calculator in the Project window. In this case, the node is renamed to **3D plot**. When this node is clicked in the Project window, the **3D plot** tab is highlighted and underlined in the Visualizations window.



You can also click each of the top level tabs in the Visualizations window at any time to navigate the visualizations, but the node focus in the Project window does not change to match.

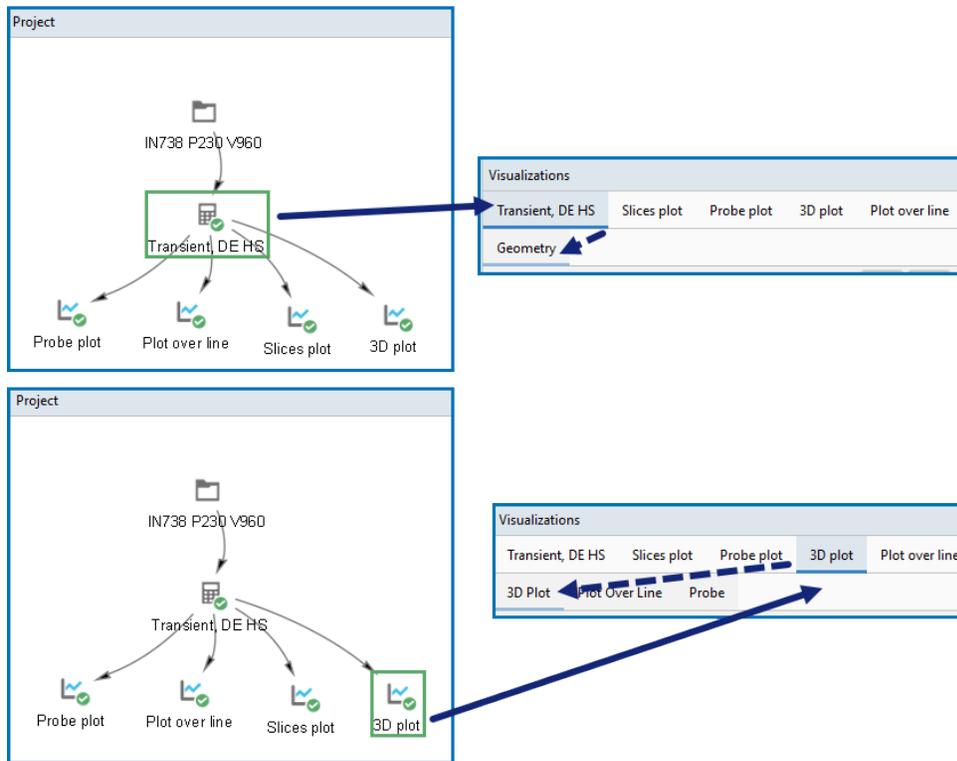
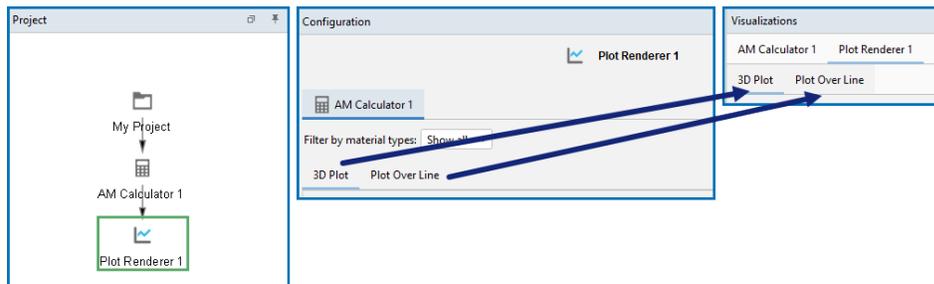


Figure 3: When you click a node in the Project window, the associated tab is highlighted and underlined in the Visualizations window. Then the second level of tabs are further underlined to indicate what is associated to the selected top level tab, and this is what actually displays in the Visualizations window.



In the Visualizations window, the top level tab for Plot Renderer is further connected to the **3D Plot** and **Plot Over Line** tabs. You can determine what you are actually viewing in the Visualizations window based on the upper level highlighted and underlined tab and then further by the second level underlined tab.

### Visualizing Heat Source Calibrations

The following is available for a **Steady-state** → **Heat source calibration** Calculation Type.

 [About Heat Source Calibration](#)

During the set up of the **Heat source calibration**, experiment data is either entered or imported to the AM Calculator. You can then view the *Heat source calibration* on a second level tab in the **Visualizations** window. This is associated to the AM Calculator.

Then once the experiment data is included and the heat source calibration is performed, you can further create plots on the Plot Renderer to examine the *Heat Calibration Parameters* and *Melt Pool Dimensions*.



More details about this are described in a separate topic. See [Visualizing Heat Source Calibrations](#).

## Visualizing Heat Source Calibrations

This topic describes both the AM Calculator and the Plot Renderer visualization outcomes related to the heat source calibration calculations for a Steady-state simulation.



Also review [Working with AM Visualizations](#), which provides an overview of how all the windows are connected.



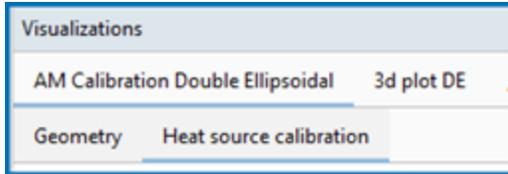
[AM\\_06a: Calibrating a Heat Source for a 316L Steel](#) and [AM\\_06b: Using the Calibrated Heat Source for a 316L Steel](#)

### AM Calculator Experiment Data

During the set up of the **Heat source calibration**, experiment data is either entered or imported to the **AM Calculator** table where it is visible on its associated **Configuration** window.

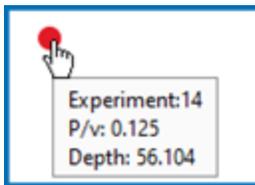
Experiment Data						
Experiment file				delimiter	Comma	
#	Power (W)	Speed (mm/s)	P/v (J/mm)	Width (µm)	Depth (µm)	Use experiment
1	60.000000	2400.000000	0.025000	36.842110	9.444444	<input checked="" type="checkbox"/>
2	60.000000	2000.000000	0.030000	39.009290	10.555560	<input type="checkbox"/>
3	50.000000	1200.000000	0.041667	51.075950	9.303797	<input type="checkbox"/>
4	60.000000	1600.000000	0.037500	46.439630	11.666670	<input type="checkbox"/>

Click the relevant AM Calculator node in the **Project** window to preview the table data via a **Heat source calibration** tab on the **Visualizations** window.



Use the table's **Use experiment** column on the AM Calculator Configuration window to select the relevant checkboxes for which **Width** and **Depth** points to include (i.e. *use*) or exclude (i.e. *not use*).

You can click or hover over any point on the **Heat source calibration** preview to display the label, which also includes the corresponding table row (i.e. *Experiment*) number for the point.



On the **Heat source calibration** window :

- For this calibration view, **Width** uses blue squares (solid color means used, hollow with same color outline means not used)
- For this calibration view, **Depth** uses red circles (solid color means used, hollow with same color outline means not used)
- The preview shows the energy density (P/v) on the X-axis and melt pool dimensions on the Y-axis.

Click to select or deselect checkboxes in the table to dynamically watch the selection change on the Visualizations window. The points that are selected in this table are used in the simulation results and are later labeled **Exp.Width** and **Exp.Depth** in the Plot Renderer Configuration table (see [Figure 6](#)). Then the calculated results are added to new columns labeled **Sim.Width** and **Sim.Depth** in the data table on the Plot Renderer Configuration window (see below).



Sometimes an experiment data point is selected in the table on the AM Calculator but after running the simulation it is not available in the corresponding Plot Renderer table (i.e. it is a greyed out row). This indicates that the calibration could not be calculated for that particular data point.

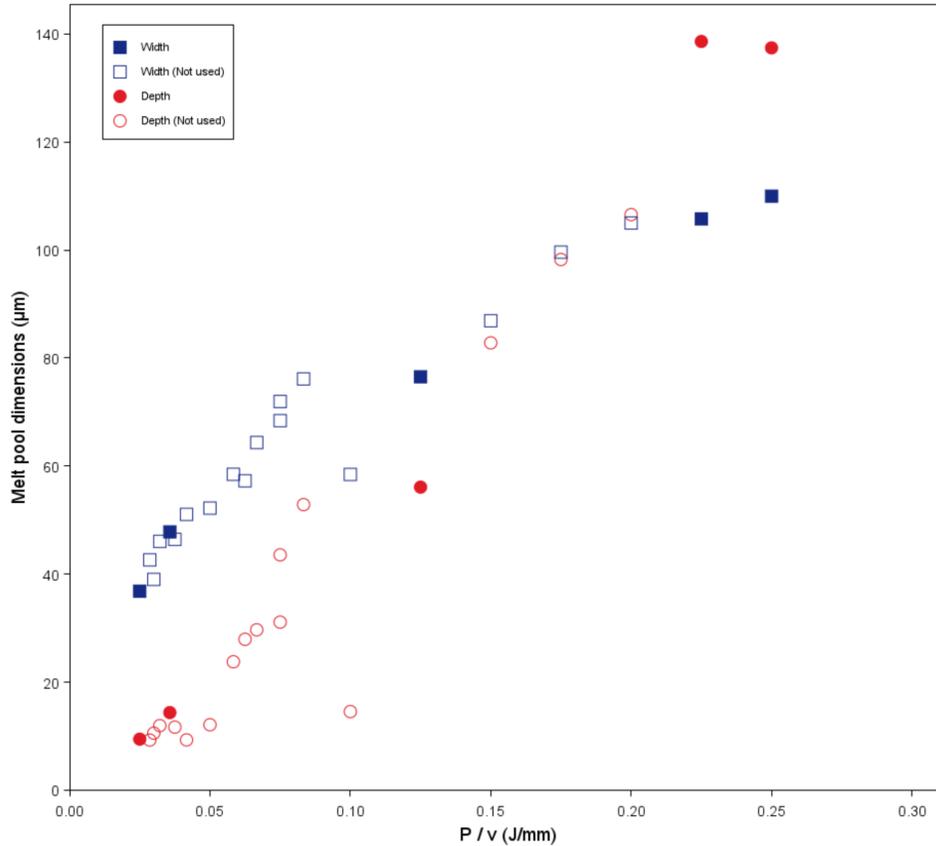


Figure 4: From example AM\_06a, this is the heat source calibration preview for the data imported to the AM Calculator node (renamed to AM Calibration Double Ellipsoidal). As per the legend, both "used" (solid color) and "not used" (hollow with outline color) points are included in this preview.

## Working with the Experiment Data Table



Once a set of data is imported to the table (or entered manually), expand the **Configuration** window to reveal the scroll bars. Then you can scroll up or down, left or right, within the table to select checkboxes for the relevant data points you want to include or exclude. This is applicable to the tables found on the AM Calculator as well as those on the Plot Renderer for the heat calibration data.

Experiment Data					
Experiment file <input type="text"/>				delimiter	Comma
#	Power (W)	Speed (mm/min)	Depth (μm)	Use experiment	
1	60.000000	2400.000000	9.444444	<input checked="" type="checkbox"/>	
2	60.000000	2000.000000	10.555560	<input type="checkbox"/>	
3	50.000000	1200.000000	9.303797	<input type="checkbox"/>	

#	Power (W)	Speed (mm/min)	Depth (μm)	Use experiment
7	60.000000	1200.000000	12.109700	<input type="checkbox"/>
8	50.000000	800.000000	27.931030	<input type="checkbox"/>
9	100.000000	2800.000000	14.364410	<input checked="" type="checkbox"/>
10	40.000000	400.000000	14.545450	<input type="checkbox"/>
11	70.000000	1200.000000	23.776370	<input type="checkbox"/>
12	60.000000	800.000000	31.111110	<input type="checkbox"/>
13	80.000000	1200.000000	29.683540	<input type="checkbox"/>
14	50.000000	400.000000	56.103900	<input checked="" type="checkbox"/>
15	90.000000	1200.000000	43.565400	<input type="checkbox"/>

### Plot Renderer Configuration Window

After the calibration is completed on the AM Calculator and the simulation run, then on the **Plot Renderer Configuration** window you can start to use the data.

### ADDING THE HEAT SOURCE CALIBRATION PLOTS

1. Add a Plot Renderer to the AM Calculator.
2. Choose an experiment point from the **Heat source calibration experiments** list. The points in this list match the points selected in the **Use experiment** column on the AM Calculator.
3. Click **Perform**.
4. Click the **Heat source calibration** tab on the Plot Renderer Configuration window to (1) view the table of data and (2) view the *Heat source calibration experiments* and *Melt pool dimensions* plots in the **Visualizations** window on the secondary level of

tabs.



## NAVIGATING THE HEAT SOURCE CALIBRATION TABLE

On the **Heat source calibration** tab the table includes these columns:

- The original experiment data for **Power** and **Speed**, plus the energy density (**P/v**).
- The original experiment data for **Width** and **Depth**, now with a prefix **Exp.** to indicate this is the experimental data.
- The calculated **Width** and **Depth** points, with the prefix **Sim.** to indicate it is the result of the simulation.



Only the experiment data points selected for inclusion when setting up the AM Calculator have data in these two **Sim.** columns.

- The **Use in fit** column has a checkbox available to include or exclude those points on the *Heat source calibration parameters* and *Melt pool dimensions* plots. Furthermore, if a simulation is not used in fit, the results from that simulation are not included to generate functions for **Heat source calibration** parameters.



Only points that were originally included in the simulation can be selected or deselected from the **Use in fit** column. If an experiment data point needs to be added, return to the AM Calculator, choose the point (s) to include there, and run the simulation again. Once the simulation is complete, the points are included in this table.

Heat source calibration		3D Plot	Plot Over Line					
#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp.Width ( $\mu\text{m}$ )	Exp.Depth ( $\mu\text{m}$ )	Sim.Width ( $\mu\text{m}$ )	Sim.Depth ( $\mu\text{m}$ )	Use in fit
1	60.0	2400.0	0.025	36.8421	9.4444	36.9718	9.6821	<input checked="" type="checkbox"/>
2	60.0	2000.0	0.03	39.0093	10.5556			<input type="checkbox"/>
3	50.0	1200.0	0.0417	51.076	9.3038			<input type="checkbox"/>
4	60.0	1600.0	0.0375	46.4396	11.6667			<input type="checkbox"/>

Figure 5: From example AM\_06a, the Heat source calibration tab on the Plot Renderer for a double ellipsoidal heat source calibration for the double ellipsoidal heat source calculation. Here experiment from row #1 is included in the simulation as indicated by the white row and the check mark in the Use in fit column. Rows 2-4 were not included in this simulation and are greyed out.

## WORKING WITH THE HEAT SOURCE CALIBRATION PARAMETERS

Based on the type of heat source selected on the AM Calculator, below the *Heat source calibration* table there are additional parameters you can choose to include or exclude from the *Heat source calibration parameters* and *Melt pool dimensions* plots.

Once the selection of experiments is made in the **Configuration** window for the **Heat source calibration**, you can choose appropriate functions for each heat source parameter including the absorptivity. See [Figure 6](#) for an example of the available parameters for a double ellipsoidal heat source.

	R2	Below P/v	Above P/v	Plot
Absorptivity:	0.96	Constant	Constant	<input checked="" type="checkbox"/>
a:	0.00	Constant	Constant	<input checked="" type="checkbox"/>
af:	0.00	Constant	Constant	<input checked="" type="checkbox"/>
b:	0.00	Constant	Constant	<input checked="" type="checkbox"/>
c:	1.00	Constant	Extrapolate linear	<input checked="" type="checkbox"/>

Figure 6: From example AM\_06a, these are the available parameters for a double ellipsoidal heat source simulation. The available parameters are based on the heat source chosen on the AM Calculator during set up.



Click to select or deselect the **Plot** checkbox and watch the change on the **Visualizations** window for both the *Heat source calibration parameters* and *Melt pool dimensions* plots. There are several options to choose from in this section that each change the plots dynamically.

The heat source parameters available to plot for each heat source type:

- **Gaussian: Absorptivity** and **Beam radius**.
- **Double ellipsoidal: Absorptivity, ar, af, b,** and **c**
- **Conical: Absorptivity, re, ri,** and **H**.



There is more information about using these parameters in [AM Calculator Conditions Settings](#) → [Heat Source](#) section.

For all heat source parameters, you can also choose to select:

- A **Constant** value i.e. the average value from all experiments or a function of energy density, which could be a **Linear function**, **Cubic function**, or a **Sigmoid function**.
- A **2d cubic function** of *Power (P)* and *Scanning speed (v)* (choose **2d cubic function P and v** from the list).
- For extrapolating outside the range of energy density as given from the experimental data, select a **Constant** value, **Extrapolate linear**, or choose **Extrapolate** for extrapolation using the same function as chosen for interpolation.
- Set the range of energy density to interpolate the heat source parameters by entering the appropriate values in the **Below P/v** and **Above P/v** fields.



There are **R2** values (the coefficient of determination) included that say how well the selected function fits the data. R2 ranges from zero to one, where 1 is a perfect match. E.g. A linear function between two points.

Once all the parameter adjustments are made, you can also retain this information for future reuse (click **Save As** or **Save**). You can also **Rename** or **Delete** these from the list as needed.



The name of the file defaults to using the *Heat Source type* + the *material type*. For example, Double ellipsoidal - 316L, or Gaussian - 316L.

## Visualizations Window

### HEAT SOURCE CALIBRATION PARAMETERS PLOT

A **Heat source calibration parameters** plot shows the calibrated heat source parameters as a function of energy density ( $P/v$ ) (unit J/mm). From the Plot Renderer Configuration window, you can also continue to select which experiments to be used to fit the calibration.

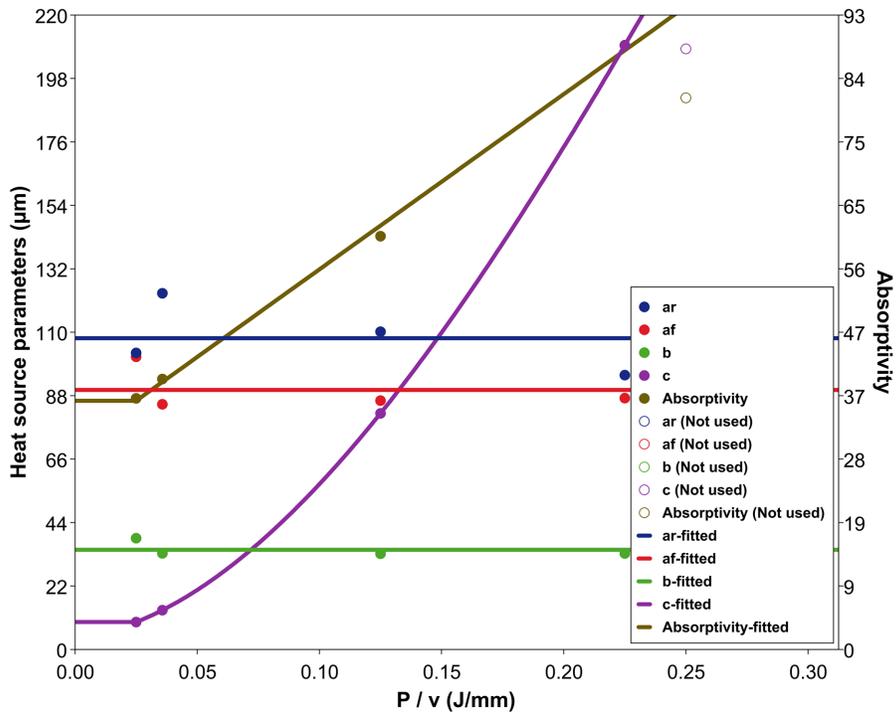


Figure 7: From example AM\_06a, the "Heat calibration parameters" plot for the double ellipsoidal calibration.

For instance, you can deselect the points that seem to be outliers or where the error bar in experimental data is large. The selection is done via the checkboxes available in the **Use in fit** column as in [Figure 5](#).

## MELT POOL DIMENSIONS PLOT

A plot of melt pool size, both from experiments and simulations, as a function of energy density ( $P/v$ ). This plot can be used to compare the results from the experiments and optimization simulations.



## AM Visualizations Window Plot Toolbar

The **Visualizations** window by default has tabs included to help you visualize both the set up using the available Calculators (e.g. AM Calculator, Diffusion Calculator, and Precipitation Calculator) and the results that are calculated via a **Plot Renderer** or **Table Renderer**. For 2D plots, you can also view data by clicking the **Table View** button.

 [Rotate, Zoom, and Pan 3D Plots: AM Calculations](#)

 [Working with AM Visualizations](#)

Button	Name	Action
	Zoom to heat source position	Click to zoom to the heat source.
	Set view to 3D	Click to reset the view where you zoom out to see the whole geometry.
  	Set view direction to HZ, XZ or XY	Click to orient to a side view in the <ul style="list-style-type: none"> <li>• YZ-axes direction.</li> <li>• XZ-axes direction.</li> <li>• XY-axes direction.</li> </ul>
	Azimuth rotation, Ctrl+ for invert rotation	Click to rotate around the free axis. Press and hold CTRL + click the mouse to control the direction turned.
	Elevation rotation, Ctrl+ for invert rotation	Click to rotate around the free axis. Press and hold CTRL + click the mouse to control the direction turned.
	Roll rotation, Ctrl+ for invert rotation	Rotate around the free axis Click Ctl+button to control the direction turned
	Save camera view angle and zoom level	There are two attributes that are saved when the <b>Save camera view angle and zoom level</b> button is clicked. The view angle of the camera and the zoom level. Then when you click the <b>Apply saved camera view and zoom level</b> , it mirrors the

Button	Name	Action
		<p>view angle of the camera and the zoom level. This is available for any 3D plot tab on the Visualizations window.</p> <p>Only one camera position can be saved/retained at any time for use.</p>
	Apply saved camera view angle and zoom level	<p>First use the <b>Save camera view angle and zoom level</b> to save the desired view. Then click the <b>Apply saved camera view and zoom level</b> button to mirror the view angle of the camera and the zoom level. This is available for any 3D plot tab on the Visualizations window.</p> <p>Only one camera position can be saved/retained at any time for use.</p>
	Show grid	Click to turn on and off the mesh grid.
	Show heat source	Click to show and hide the heat source.
	Mirror geometry	When in steady state you only see half the geometry. Click to show the whole geometry, click again to show half the geometry.
—	Hide layers	Enter a number of layers to hide
	Show Outer Box	Click to turn on and off the outer box geometry.
	Show Size of the Melt Pool	Click to show or hide the melt pool. This is the same thing as showing the size of the liquidus iso-surface plot. When clicked you also see the measurements in the <b>Event Log</b> window and shows on the <b>Visualizations</b> window.
	Show Size of Melt Pool plus Mushy Zone	Click to show or hide the melt pool and mushy zone. This is the same thing as showing the size of the solidus iso-surface plot. When clicked you also see the measurements in the <b>Event Log</b> window and shows on the <b>Visualizations</b> window.
	Show Manual Ruler	<p>Click to use the manual ruler to measure the melt flow.</p> <ul style="list-style-type: none"> <li>• Double click each time to add the start and end points.</li> <li>• To align a ruler on the x-, y-, or z- axis: Hold CTRL when adding or moving the end point. The ruler snaps to the closest point on the same axis.</li> <li>• To move a ruler: Click and hold a point, then drag to reposition.</li> <li>• Right-click a point to delete it.</li> </ul>

## Rotate, Zoom, and Pan 3D Plots: AM Calculations

When working with the Additive Manufacturing (AM) Module and the results from an [AM Calculator](#) simulation, two general plot types are added to the **Visualizations** window: a *3D plot* and a *Plot over line 2D plot*. The 2D plot has the same functionality as for other calculations but the 3D plot is specific to this Module.

In addition to the **Visualizations** window for an AM 3D plot having its own set of buttons (see [AM Visualizations Window Plot Toolbar](#)) there are also different ways to work with the 3D plots with respect to rotating, zooming, and panning the geometry around the window.

- In combination with the following actions use these buttons on the toolbar:

- **Set View to 3D** 

- **Set View Direction** 

- **Rotate the geometry** : Left-click and hold the mouse, then move it around the window to rotate in any direction. Hold CTRL + left mouse button to rotate on a specific axis.
- **Zoom in and out**: Scroll the mouse wheel or right-click and move the mouse up and down in the window.
- **Pan the geometry**: Hold Shift + left mouse button then move the mouse up or down, left or right, to move the geometry around the window without zooming.



Go to the Help (press F1 in Thermo-Calc) to view a short video showing the use of these actions.

# The Process Metallurgy Module

---



## [Process Metallurgy Module Examples Collection](#)

### In this section:

---

About the Process Metallurgy Module .....	502
Available Options with the Process Metallurgy Module .....	504
Network License Restrictions .....	506
Setting Up a Process Metallurgy Simulation .....	507
Working with the Process Metallurgy Module .....	510
Working with the Process Schedule .....	519
About Dynamic Time Stepping .....	521
About Pressure-Dependent Calculations .....	523
Working with the Material Manager .....	524
Process Metallurgy Slag Properties .....	526
Adiabatic Calculations: Material Enthalpy .....	529
Process Metallurgy Calculator .....	530
Defining the Equilibrium Simulation .....	533
Equilibrium Simulation: Conditions Tab .....	538
Defining the Process Simulation .....	544
Process Simulation: Conditions Tab .....	548
Process Simulation: Process Model Settings .....	550
Process Simulation: Materials Tab .....	554
Process Simulation: Process Schedule Tab .....	556
Options Tab .....	558

---

## About the Process Metallurgy Module

---

The Process Metallurgy Module is an Add-on Module to Thermo-Calc and it is available in Graphical Mode as the *Process Metallurgy Calculator*. The Add-on Module is designed to model reactions that occur in metallurgical processes. Although primarily used in steelmaking and steel refining processes—such as basic oxygen furnaces, electric arc furnaces, ladle furnace metallurgy, and so forth—applications are not limited to steelmaking.

There are two main branches of calculations possible: **Equilibrium** and **Process simulation**. Equilibrium calculations do not consider any kinetics, the process simulation includes reaction kinetics.

In general, for both types, the main difference compared to using a standard Equilibrium Calculator is that it is easy to handle the different materials present in a metallurgical process. The materials used in the process can be pre-defined, saved and used for the equilibrium calculations. Metallic materials can be defined in element weight percent, oxide materials in weight percent of oxide components, the gas phase in volume percent of gas components and its amount can be defined in normal cubic meters, and so forth. In principle this can be done in a standard Equilibrium Calculator. However, it is much easier when using the Process Metallurgy Calculator as this is designed for this specific purpose.

When using the **Equilibrium** branch of calculations, both isothermal and adiabatic calculations are possible. Adiabatic calculations assume no heat and mass exchange with the environment during the equilibrium reaction, meaning that the temperature changes as a result of exothermal or endothermal reactions taking place.

For a **Process simulation** branch of calculation, the reaction kinetics of the process are considered. This is done by dividing the system into zones. Typically one would have one steel zone containing liquid metal and a slag zone containing liquid oxide (slag). The kinetic model assumes that only a fraction of the steel zone reacts with a fraction of the slag zone per time step. This reacting fraction of the whole system is termed the reaction zone (in literature it has become known as Effective Equilibrium Reaction Zone or EERZ).

You can use the Process Metallurgy Module to calculate the following:

- The equilibrium between custom-defined steels, slags, and gasses.
- The equilibrium between other metallic and non-metallic phases.
- Desulfurization, dephosphorization, and decarburization.
- Any partition coefficient, for example the partitioning of sulfur between the liquid steel and slag phase.
- Slag characteristics, such as slag basicity or sulfur capacity.

- The fraction of liquid and solid in the slag.
- The temperature change in an adiabatic process.
- Kinetics of the reaction between phases. Typically this is the reaction between a liquid metal and slag phase, but it could also be the reaction between a solid oxide and slag (simulation of refractory wear) or between a solid metal and liquid metal (simulation of dissolution of alloy), etc.



The Process Metallurgy Module requires both a valid Maintenance and Support Subscription (M&SS) and a license for the TCS Metal Oxide Solutions Database (TCOX8 or newer).



All users can test the Process Metallurgy Module with the included OXDEMO database, which is limited to these elements: Al, C, Ca, Fe, O, S, and Si. For more information about this and other products [visit our website](#).

## Available Options with the Process Metallurgy Module

The Process Metallurgy Module is an Add-on Module to the core Thermo-Calc software.



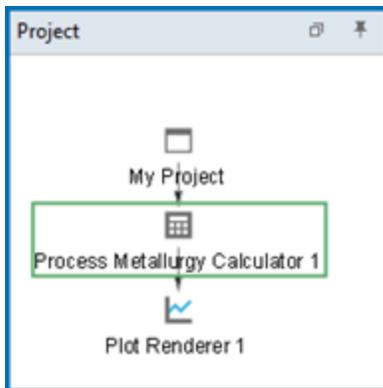
A separate license is required to perform calculations for more than three elements. Without it you are able to use the module in *Demo Mode*.

### Process Metallurgy Template

A **Process Metallurgy** template under **Applications** is available to all Thermo-Calc users when in Graphical Mode. If you are in DEMO mode then this is indicated by the addition of **DEMO** text.

#### Using the Template

After opening Thermo-Calc in Graphical Mode, in the templates section under **Applications** click **Process Metallurgy** to add a *System Definer*, *Process Metallurgy Calculator*, and *Plot Renderer* to the **Project** tree.



[Creating a Project from a Template](#)

### Process Metallurgy Calculator

A Process Metallurgy Calculator allows you to set the conditions for, and perform, a process metallurgy calculation.

## ***Demonstration (Demo) Mode***

The Process Metallurgy Module, and some examples, are available to all Thermo-Calc users but only for simulations of alloys with up to three elements. If you do not have a license for the Process Metallurgy Module then you are in *Demonstration Mode* when using the Process Metallurgy Calculator or its template.

### *Process Metallurgy Template*

A **Process Metallurgy** template under **Applications** is available to all Thermo-Calc users when in Graphical Mode. If you are in DEMO mode then this is indicated by the addition of **DEMO** text.

### *Process Metallurgy Calculator*

If you are experimenting with the Process Metallurgy Calculator in Demo Mode, you may have access to a variety of databases based on your license. However, you can only define up to three elements for a demo simulation.

If you define more than three elements and try to run the simulation the **Event Log** displays an error message.

## Network License Restrictions

---

The Process Metallurgy Module requires a separate license. If you are using a network client installation of Thermo-Calc, then you may not be able to use it even if you have access to a license server with a valid network license file. The reason for this is because other clients who are part of your network installation may have checked out all instances of the network license allowed to run simultaneously.



For users with a network license, you must exit Thermo-Calc to release the license for other users. The license is checked out as soon as you add a Process Metallurgy Calculator and remains unavailable to other users until you exit the program.



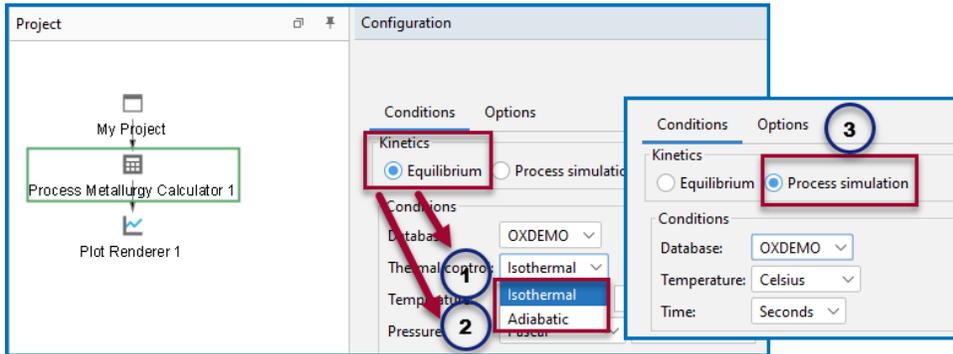
With a network license, and if as per above you temporarily do not have access to a license, you are automatically put into *Demo Mode*. Then the Process Metallurgy Calculator is available with three elements.



Search the online help or see the *Thermo-Calc Installation Guide* for more about network licenses.

## Setting Up a Process Metallurgy Simulation

The following basic steps need to be carried out to set up calculations in the **Process Metallurgy Calculator**. There are three branches of simulation to choose from on the **Configuration** window when setting up the simulation an isothermal equilibrium calculation, an adiabatic equilibrium calculation, and a process simulation.

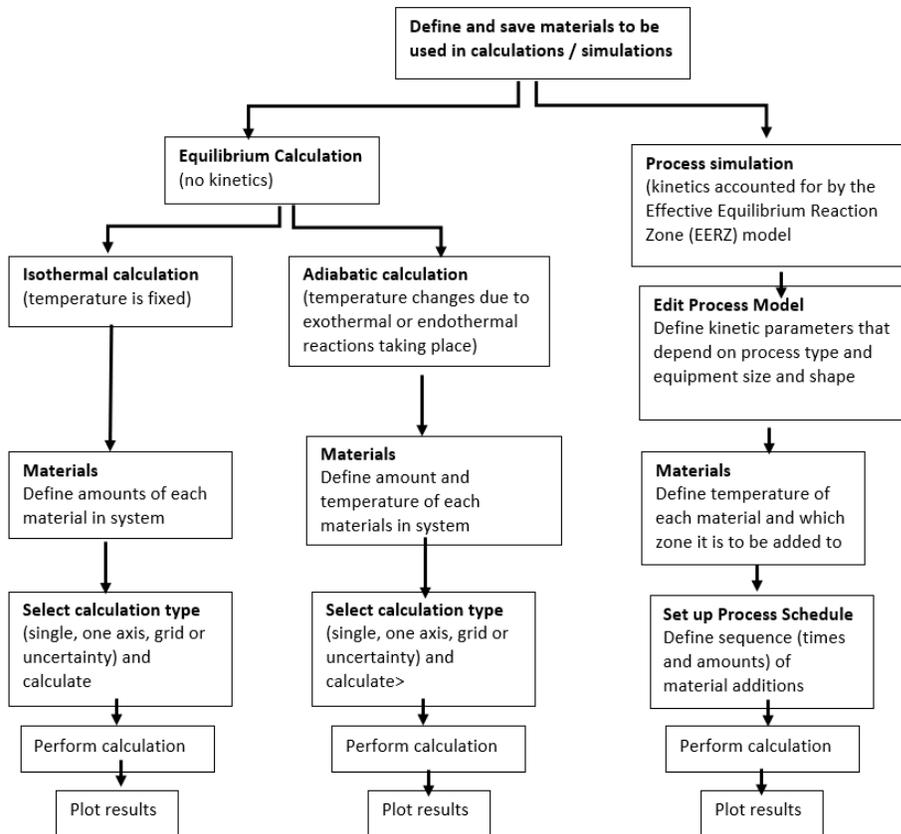


Examples of using the Process Metallurgy Calculator for Basic Oxygen Furnace (BOF) (with and without kinetics), a Ladle Furnace (LF) simulation, an Argon Oxygen Decarburization (AOD), and a Vacuum Oxygen Decarburization (VOD) simulation are included with your installation. See the [Process Metallurgy Module Examples Collection](#) for links.



You can find several learning resources on the [Process Metallurgy](#) page of our website (or in Thermo-Calc, go to **Help** → **Thermo-Calc website**). You can also [subscribe to the Thermo-Calc newsletter](#) to be kept up-to-date about upcoming releases, training, examples, videos and much more.

## Isothermal and Adiabatic Equilibrium Calculations and Process Simulation Options



The three branches of simulations available for the Process Metallurgy Calculator (1) an isothermal equilibrium calculation, (2) an adiabatic equilibrium calculation, and (3) a process simulation.

## Equilibrium Simulations

Two **Equilibrium** simulation types are available: **Isothermal** and **Adiabatic**. In general for the equilibrium simulation, the three most common material groups—*Steel*, *Slag*, and *Gas*—are automatically included with the default composition input types that are relevant to each material group. You can add materials and save your material compositions directly from the Process Metallurgy Calculator and quickly access these for other simulations. For example, you might want to predefine compositions of steels, ferro-alloys, slags, and slag additions. The overall composition of the steel and slag is then obtained by adding amounts of these predefined materials.

Each group of materials is automatically added to the similar group where it is defined. You can manage these groups using the *Material Manager* (see [Working with the Material Manager](#)).



For adiabatic calculations, the selection of material type (steel, slag, gas) has an influence on the calculation. For isothermal calculations, the results are identical.

The reason the selection of materials for adiabatic calculations has an influence is because there are different strategies to calculate the enthalpy of the materials at the temperature at which these are added. Therefore, defining FeO as a *steel* containing 50 at% Fe and 50 at% O or as a *slag* with 50 at% Fe and 50 at% O or as *slag* with 100% FeO, all result in different enthalpies assigned to the component FeO, which in turn results in different equilibrium temperatures.



[Defining the Equilibrium Simulation](#) and [Equilibrium Simulation: Conditions Tab](#)



[Adiabatic Calculations: Material Enthalpy](#)

## Process Simulations

When you choose to do a **Process simulation** it includes the reaction kinetics. Using the EERZ (Effective Equilibrium Reaction Zone) model, you can first define your materials as you do with an **Equilibrium** simulation. You also need to define the **Process model** where you define the kinetic parameters of the simulation by setting up reaction zones, reaction rates, heat transfer coefficients, and so forth. You also need to define the **Process schedule** where the sequence and amounts of added materials are defined.



[Defining the Process Simulation](#) and [Process Simulation: Conditions Tab](#)

## Calculation Types

The calculations available for an **Equilibrium** simulation, for both isothermic and adiabatic calculations, are *Single equilibrium*, *One Axis*, *Grid*, and *Uncertainty*. After adding a **Plot Renderer** or **Table Renderer**, you can choose a variety of quantities, including slag properties such as Bells ratio and B2, to output results that provide useful visual representations of the steelmaking systems being analyzed.



[Process Metallurgy Calculator](#) and [Defining the Equilibrium Simulation](#)

---

## Working with the Process Metallurgy Module

---

Working with the **Process Metallurgy Calculator** allows you to easily define complex metallurgical reactions. The calculator streamlines how you set up these steel and slag systems but there are details about various inputs and outputs that are different from other calculators. This topic is a general collection of information related to the Process Metallurgy Module and is intended to help you better understand how to interpret and make choices when you are setting up your steelmaking simulation.

### *Database Dependency*



It is not possible to use custom user databases with the Process Metallurgy Module. This is because several internal calculations, such as determining the component compositions of ionic phases, are dependent on the database. Therefore, only TCS Metal Oxide Solutions Database (TCOX) versions 8 and newer, and to a limited extent, OXDEMO, can be used with this Module.

### *Material Manager*

For information about saving and deleting materials used with the Process Metallurgy Calculator, see [Working with the Material Manager](#). Also see the next section.

### *Working with Material Groups (Steel, Slag, and Gas)*



For more details about these fields, see [Defining the Equilibrium Simulation](#). For information about saving and deleting materials, see [Working with the Material Manager](#).

There are three groups to define: **Steel** (or another user-defined **Metal** group), **Slag**, and **Gas**.



It is recommended you open the [PMET\\_01: Basic Oxygen Furnace \(BOF\)](#) example to follow along with the descriptions. Version b is used in the screenshots.



Open the example project file(s) from Thermo-Calc **Help** → **Example Files** → **Process Metallurgy** → **PMET\_01\_Basic\_Oxygen\_Furnace.tcu**. Choose a, b, or c versions of the example based on whether you have a license for the TCOX database.

In the image below, the red-highlighted area is where the Steel material group is defined. The drop-down menu that displays *Steel\_BOF* shows a user-defined material. This can be defined and then saved to the material library.

Configuration

Basic Oxygen Furnace - fixed temperature

Conditions Options

Kinetics

Equilibrium  Process simulation

Conditions

Database: OXDEMO

Thermal control: Isothermal

Temperature: Celsius 1650.0

Pressure: Pascal 100000.0

Material: Steel Steel\_BOF

Amount: Tonne 100.0

Input type: Mass percent Element Major component

Major component:  Fe 94.5

C 4.5

Si 1.0

Total: 100.0

Material: Slag Slag\_BOF

Amount: Tonne 3.0

Material: Gas Oxygen

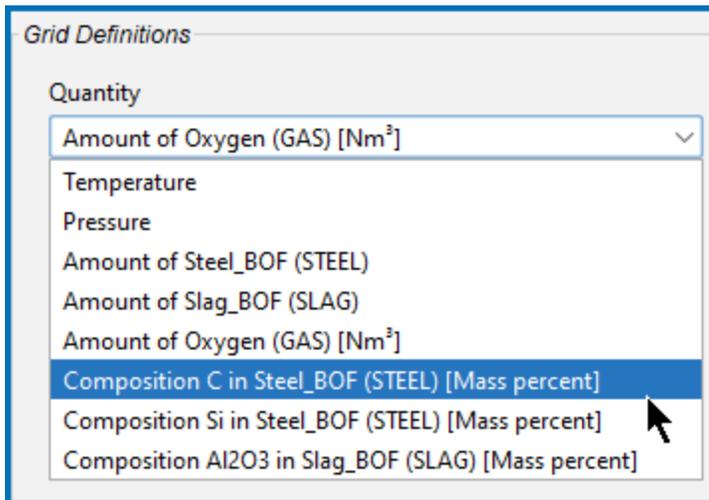
Amount: Normal cubic meter 0.0

The three material group areas on the Process Metallurgy Calculator where you can define components and save materials. You can create your own material groups using the *Material Manager*.

When you click **Show composition** next to any material group, additional fields display to give composition and input type where you can further define each material group. You can remove materials and add and remove material groups using the *Material Manager* (available on the **Configuration** window → **Options** tab).

In the example, click **Show composition** next to the **Slag** and **Gas** groups to see how the *Slag\_BOF* and *Oxygen* materials are defined.

From this composition area you can click the **Save material** button to add the new material to the group's list. The newly named material composition is then available for use with that material group. In this example, the *Steel\_BOF* material was defined first and then named at the end. It is now available in the lists wherever you select the **Steel** material or need to choose quantities to plot.



After defining and saving a material composition, the named material is available for selection in a variety of places such as when choosing quantities on the calculator and defining axis variables on the *Plot Renderer*.



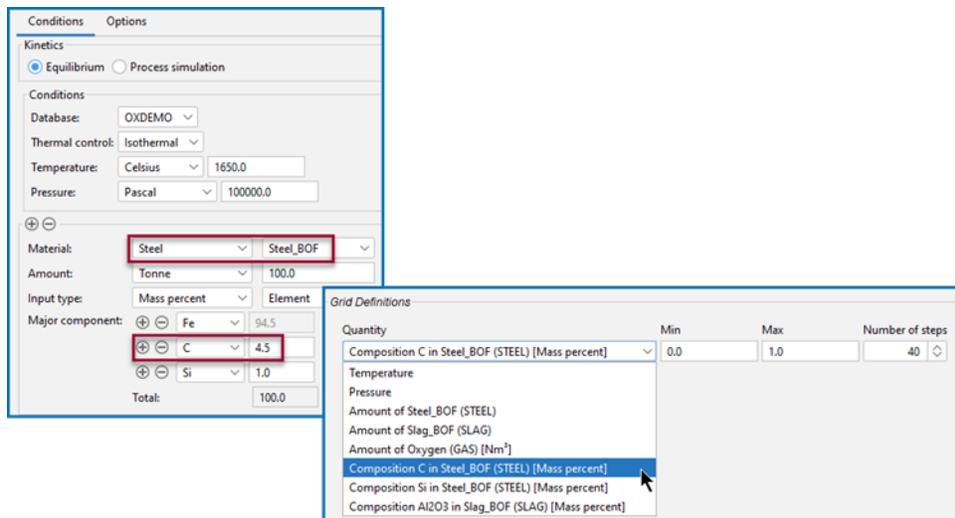
To add one or more material groups click the add  $\oplus$  button. Click the minus  $\ominus$  button to remove a material group. The new material inherits the default settings of that group.

## Understanding the Grid Definitions Quantities and How These Relate to the Composition

When you choose a **One Axis** or **Grid Calculation Type**, you also choose a **Quantity** under the **Grid Definitions**. The relationship between these quantities and how the material group composition is defined as follows.

The following uses **Composition of C in Steel\_LF (STEEL)** as an example quantity for a **One Axis** calculation.

Under **Grid Definitions**, the availability of the **Quantity** in the list is based on the material (**Steel\_BOF**) that is defined and saved for the **Steel** group (or any user-defined metal, slag, or gas group).

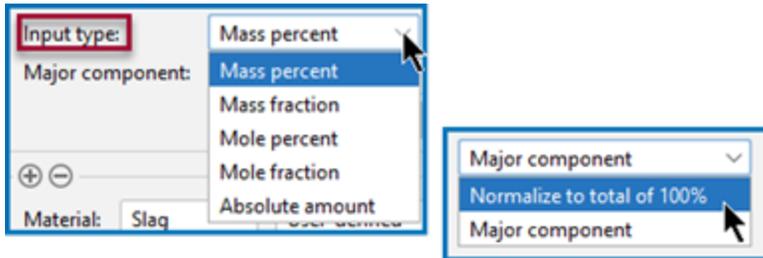


## Defining the Material Composition

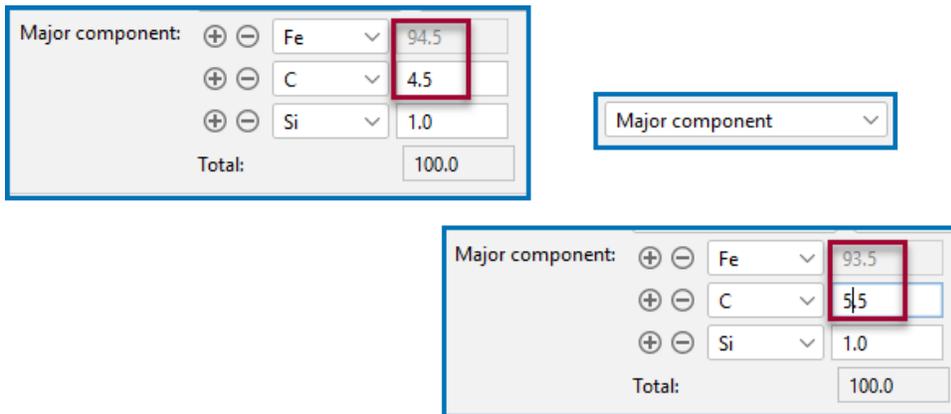
Click **Show composition** so that when you choose the **Input type** for a steel the composition of the elements can be set up using a combination of settings starting with **Mass-** or **Mole percent**, **Mass-** or **Mole fraction**, or for Equilibrium calculations, **Absolute amount**. Then you further choose to define the composition using either the **Major component** or **Normalize to total of 100%** options.



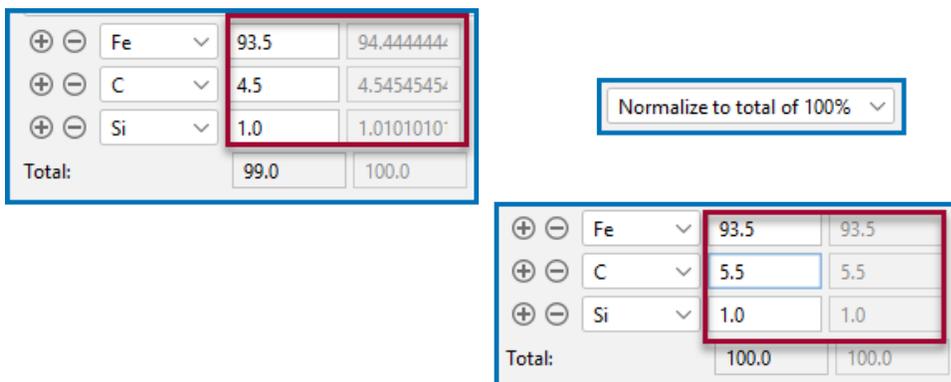
The *Amount of X in Y* quantities you can choose from the various drop-down lists (such as under *Grid Definitions*) are defined as Min-Max variation of the component X in Y. If a dependent element is present that is adapted as usual.



When you use **Major component**, as you add other components or adjust the composition, the major one is adjusted to be the rest of the composition.



When you use **Normalize to total of 100%**, the sum is adapted to 100%. This is useful if you need to work with a given composition that does not sum up to 100%.



Sometimes its preferable to use input of amount of each component **Amount** by weight (**Tonne, Kilogram, Gram, or Pound**), it can useful to choose **Absolute amount** as the **Input type**. In this case, the total amount of material automatically adjusts the total.

The first screenshot shows the following configuration:

- Material: Steel
- Amount: Tonne
- Input type: Absolute amount
- Element amounts: Fe (94.5), C (4.5), Si (1.0)
- Total: 100.0

The second screenshot shows the following configuration:

- Material: Steel
- Amount: Tonne
- Input type: Absolute amount
- Element amounts: Fe (94.5), C (5.5), Si (1.0)
- Total: 101.0

The third screenshot shows the 'Absolute amount' dropdown menu.

For the **Gas** component, if you choose **Normal cubic meter** as the **Amount**, then beside regular mass units, the amount of gas can also be provided in Nm<sup>3</sup>.

The screenshot shows the 'Configuration' window for 'Argon Oxygen Decarburization - calculator'. The 'Conditions' tab is active. The 'Kinetics' section has 'Equilibrium' selected. The 'Conditions' section shows 'Database: TCOX11', 'Thermal control: Adiabatic', 'Temperature: Celsius', and 'Pressure: Pascal'. The 'Material' section shows 'Steel' and 'Steel scrap' with a temperature of 1550.0 and an amount of 40.0. The 'Gas' section shows 'Gas' and 'Ar-O2 gas' with a temperature of 20.0 and an amount of 1000.0.

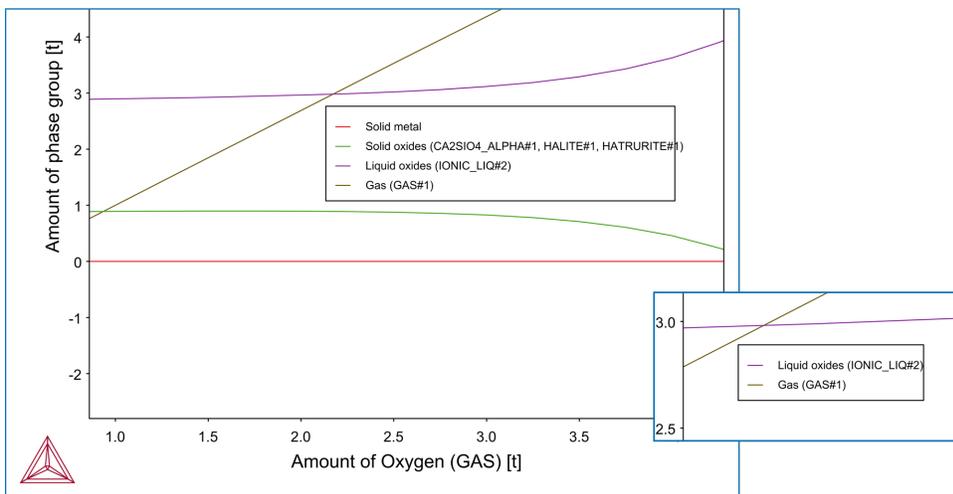
## Phase Groups

The phases that are plotted after you run a simulation are grouped together on the plot to make the interpretation of the results easier.

Phases of similar type are bundled together in the groups *liquid metal*, *solid metal*, *solid oxides*, *liquid oxides*, and *gas*. Except for the gas phase, all the groups can have more than one phase. This grouping is particularly useful for liquid metal and liquid oxide because they are both described by the same phase, IONIQ\_LIQUID, and because there is typically two or more compositions sets of IONIQ\_LIQUID. This is also useful for solid oxides, because a large number of phases can be stable, making the interpretation of results difficult without this grouping.



The term *oxides* in the Process Metallurgy Module can also mean carbides, sulfides, and fluorides.



Example of the plot labels with the phases grouped together. As you zoom in, only the phases that you can see are listed in the label, making it easier to examine the results.

## Slag Property Quantities

The slag properties describe the capability of a slag to take up sulfur from the metal melt. The Process Metallurgy Module defines so-called phase groups. These contain all phases of a certain kind, for example all solid slag or all slag phases. The component composition of such a phase group is calculated by weighting the component composition of each phase with

their fraction in the group.

This allows to calculate slag properties for the phase groups, for example for the overall slag.

On the Plot Renderer (for axis quantities) and Table Renderer there is an option to choose or define the **Slag property**. The associated quantity that you select has these options: **All**, **B2**, **B3**, **B4**, **Bas2**, **Bells ratio**, **Ls**, or **Log10 (Sulphur capacity)**.



Details, including references and equations, are described in [Process Metallurgy Slag Properties](#).

## ***Adiabatic Calculations***

When setting up the Process Metallurgy Calculator under the **Equilibrium Kinetics** branch, you choose the thermal control to be either **Isothermal** or **Adiabatic**. Adiabatic calculations assume no heat and mass exchange with the environment during the equilibrium reaction. The adiabatic process is implemented by an isenthalpic process with constant pressure (i.e. H=CONST), which is equivalent.



[Adiabatic Calculations: Material Enthalpy](#)

## ***Gas Volume Units***

Beside regular mass units, the amount of gas can also be provided in **Normal cubic meter** (Nm<sup>3</sup>). The applied reference conditions are 1013.25 hPa and 15.0 °C (according to ISO 2533).



There are slightly deviating definitions present in the industry, which need to be converted for precise results. For most applications of the Process Metallurgy Module this difference has a minor influence.

## ***Calculation of Component Compositions***

The composition of all ionic phases is provided as elemental as well as component composition (i.e. Fe<sub>2</sub>O<sub>3</sub>, FeO, CaO, ...). In order to provide a representation similar to that usually used in the industry, the content of S is provided as elemental content and the content of P is given as component content of P<sub>2</sub>O<sub>5</sub>. This is a simplification because P and S can also be present as sulphates and phosphates in the slag and can bind to different metal cations. This can cause deviations in the overall component composition if the content of S

---

and P is high in the phases. In doubt, the elemental composition can be used, it will always be exactly correct.

For non-ionic phases such as ALABANDITE, always the elemental composition will be shown only. This is also true if the IONIC\_LIQ composition cannot be described by components only due to constraints of the overall composition (for example if the material has been oxidized completely and forces a certain composition of the ionic liquid slag).

### ***Definition of Partial Pressure of Oxygen***

The oxygen partial pressure in the gas phase is calculated with the following formula:

$$p_{O_2} = p \cdot AC(O_2, GAS)$$

In the formula,  $p$  describes the total system pressure and  $AC$  the activity of oxygen in the gas phase.

### ***Activities of Slag Components***

The chemical activity of each component  $AC(COMPONENT, REFERENCE\_PHASE)$  is available either with reference to the liquid or the solid slag phase. It can only be calculated for components being present in the thermodynamic database.

### ***Process Schedule***

When you are working with the **Process simulation** branch you set up a process schedule. There are some useful details about this described in [Working with the Process Schedule](#).

### ***Dynamic Time Stepping***

The nature of a **Process simulation** requires a dynamic choice of the time step lengths. To understand this better, see [About Dynamic Time Stepping](#). These settings are done on the [Options Tab](#).

### ***Pressure-Dependent Calculations***

It is possible to apply variable pressure during process metallurgical calculations, which is relevant in various processes such as Vacuum Oxygen Decarburization (VOD) or Ruhrstahl-Heraeus (RH). See [About Pressure-Dependent Calculations](#).

## Working with the Process Schedule

- ② [Defining the Process Simulation](#)
- ② [Process Simulation: Process Schedule Tab](#)
- ② [About Dynamic Time Stepping](#)

Materials and heat can be added at any time in any amount. They can be either added to a:

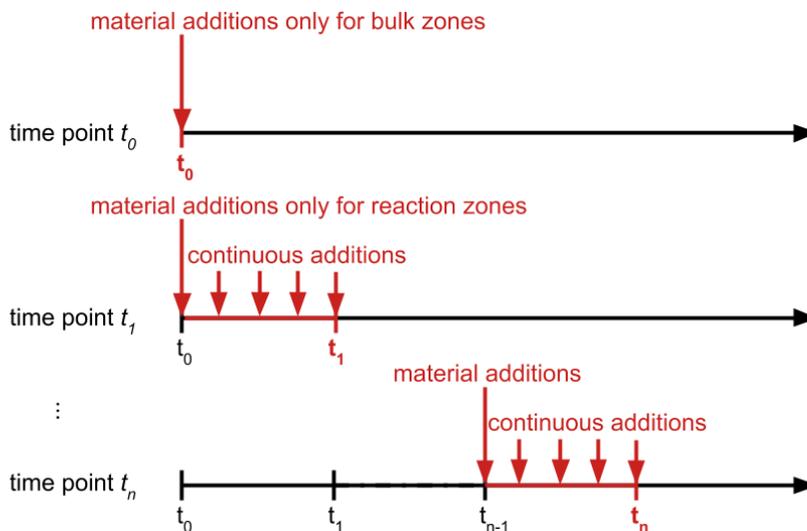
- bulk zone (such as steel, slag, etc.), or
- reaction zone (interface between two bulk zones).

These types of materials are used in the process schedule set up:

- Material addition (in tonne, kilogram, etc.)
- Continuous material addition (in kilogram per minute, etc.)
- Thermal / electrical power (i.e. a continuous heat addition in MW)

A material addition is typically used to model some event like emptying a bucket into the melt. A continuous material addition is instead used to model a continuous material feeding into the melt, such as wire feeding. Finally, heat can be added or removed during a time step and this is defined as power.

### A Process Schedule Additions Scheme



Generally the addition of materials and heat is modeled in the process simulations for any time point  $t_n$  as follows:

- Material additions happen immediately after the previous time point  $t_{n-1}$ , i.e. at the beginning of the time step.
- Continuous material and heat additions (power) are conceptually happening during the complete time step between  $t_{n-1}$  and  $t_n$ .

The data presented in the results is always referring to the time point, i.e. to the end of the time step. This means that reactions and heat transfer have happened between adding materials and heat at the beginning of the time step and the end of the time step. If for example a strongly cooling material is added, a possibly quite large heating of the zone might have happened until the end of the time step.

The first two time points are different:

- The first time point  $t_0$  at  $t=0$  does not represent a time step, instead only a mixing and equilibrium reaction of the material additions in each bulk zone is taking place. There are no reaction zones existing at this time point.
- The second time point  $t_1$  represents the first time step in the simulation where the different bulk zones react between each other via the reaction zones. Nevertheless it is conceptually identical to any other time point.

---

## About Dynamic Time Stepping

---

- [!\[\]\(d35f2658bfcf125b2b9e250fb63f5f16\_img.jpg\) Defining the Process Simulation](#)
- [!\[\]\(2abb56241840c1629171ffeabede10bf\_img.jpg\) Working with the Process Schedule](#)
- [!\[\]\(17c8c5357fcfc65351f9b7e5117b72ad\_img.jpg\) Options Tab](#)

The nature of a process simulation requires a dynamic choice of the time step lengths. On the one hand the simulation needs to capture very fast changes of temperature and composition, for example after addition of cold material. On the other hand it should calculate processes that take several hours in reality within an appropriate calculation time.

The time step control algorithm is primarily controlling the change of the temperature in each zone (including the reaction zones) such that it stays below a specified limit but is always as large as possible.

The dynamic time stepping can be switched off by choosing a very large maximum allowed temperature change per step (such as 1.0e6 K). In this case the maximum time step is always used, however this is only recommended in special situations.

If a temperature change is too large, the algorithm reverts the time step and returns to the previous time point in order to try a smaller time step. Therefore you can see in the Event Log output that the time points can jump back. Except for larger additions this is a rare event.

The parameters of the time-step control algorithm are:

- initial time step at  $t=0$ ,
- smallest allowed time step,
- largest allowed time step, and
- maximum allowed temperature change of a zone within a time step.

The initial time step as well as the largest allowed time step are defined in percent of the total duration of the process while the smallest allowed time step is given in seconds. This allows that these parameters normally need no change even if switching between processes with very short and very long duration.

Making the largest allowed time step bigger can reduce the number of steps and consequently the calculation time, but only if the time step is not limited by the maximum temperature change per time step. This means that both parameters need to be considered together.

---

Taking a too large smallest allowed time step can cause numerical difficulties, especially if large cold material additions are part of the process schedule.

The initial time step is usually of limited relevance for the overall calculation time, but should usually not be larger than the default value of 1.0%.

As different processes show very different temperature changes, it is probably most common to adapt the maximum temperature change per time step in order to have a good balance between accuracy of the results and calculation time. Usually it should be possible to change this parameter without adapting the other algorithm parameters.

The effects of the dynamic time step algorithm can be inspected by activating the data points in the plots.



To show the data points, in the **Visualizations** window in Thermo-Calc, right-click in the plot area and choose **Properties**, then click **Show data points**.

---

## About Pressure-Dependent Calculations

---

### [Defining the Process Simulation](#)

It is possible to apply variable pressure during process metallurgical calculations, which is relevant in various processes such as Vacuum Oxygen Decarburization (VOD) or Ruhrstahl-Heraeus (RH).

The implementation includes all relevant features for a realistic modeling of the pressure-dependency:

- the set pressure at a time-step is valid for all bulk and reaction zones of the system
- the exhaust gas zone has always atmospheric pressure
- the added material (only relevant for gas) has initially always atmospheric pressure
- the material transferred to and from the zones (i.e. the added material and the exhaust gas from the zones) is undergoing an adiabatic expansion / compression because it is moved between different pressures

Generally there is no effect of pressure change on anything else than gas (due to physics and the implementation of the database).



### [PMET\\_07: Vacuum Oxygen Decarburization Kinetics](#)

## Working with the Material Manager

---

Use the *Material Manager* to create and delete groups of materials and delete materials you have saved directly from the **Process Metallurgy Calculator**. It is useful for frequently used materials.



The Material Manager is only available for the Process Metallurgy Calculator.

Material groups created in the Material Manager display in all of the Process Metallurgy Calculator drop down lists, and these are also globally removed. Deleted group data directories are moved to the computer's trash bin.

A material saved in any of the material configuration panels is directly accessible globally in the Process Metallurgy Calculator Configuration window. When a material is removed in the Material Manager panel it is globally removed from the Process Metallurgy Calculator.



The settings are available on the **Configuration** window [Options Tab](#).

### ***Saving a Material***

This is done directly from the **Process Metallurgy Calculator** → **Configuration** window.

1. Add the Process Metallurgy Calculator to  **My Project** directly (right-click and select it from the **Create New Activity** menu) or use the available Process Metallurgy template on the  **My Projects Configuration** window under **Applications**.
2. On the **Configuration** window, next to the **Material** group you want to define, for example **Steel** (or another user-defined metal group), **Slag**, or **Gas**. Click **Show composition**.
3. Enter the composition and the **Major component** of the material you want to save for future use. You can alternatively select **Normalize to total of 100%** and enter the percentages for each element.
4. Click **Save material**.
5. In the window that opens, enter a **Material name**. Click **OK**.

When you go to the *Material Manager*, you can select this saved material from the applicable list if you need to delete it.



It is not possible to rename a saved material—you need to recreate/redefine the material and save it again with the new name.

## ***Deleting a Material***

This is done with the *Material Manager*, located on the **Configuration** window, **Options** tab.

1. After [Saving a Material](#), go to the *Material Manager*.
2. Select the **Material** group associated to the material saved previously, for example, **Steel** (or another user-defined metal group), **Slag**, or **Gas**.
3. From the **Material** list, select the material you want to delete and click **Delete material**.
4. In the window that opens, click **Yes** to delete the material and **No** to cancel.

## ***Creating a Material Group***

This is done with the *Material Manager*, located on the **Configuration** window, **Options** tab.

1. Go to the *Material Manager*.
2. Click **Create group**.
3. In the window that opens, select the **Material group type** (**Metal**, **Slag**, or **Gas**) you want to create.
4. Enter a **Material group name**. The new group is now available to select from the **Material group** list.

## ***Deleting a Material Group***

This is done with the *Material Manager*, located on the **Configuration** window, **Options** tab.

1. Go to the *Material Manager*.
2. From the **Material group** list, select the group you want to delete and click **Delete group**.
3. In the window that opens, click **Yes** or **No**.

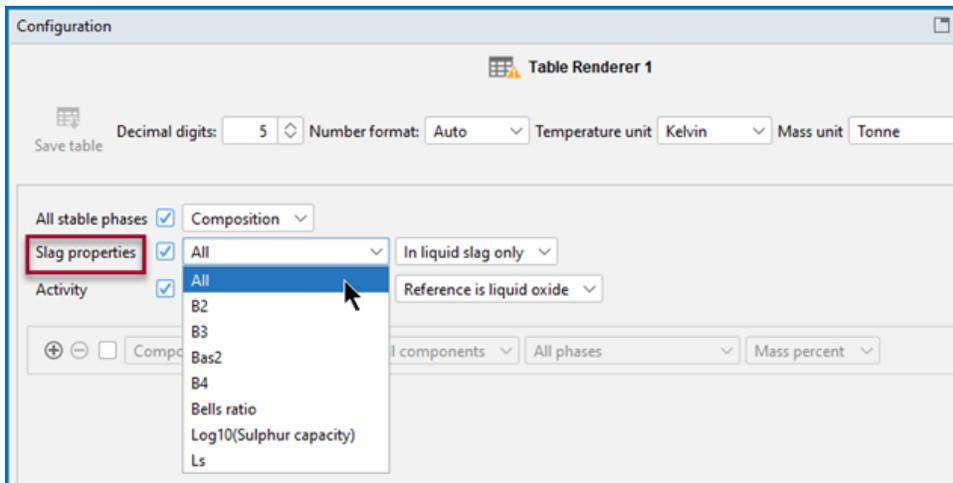
## Process Metallurgy Slag Properties

The slag properties describe the capability of a slag to take up sulfur from the metal melt. The Process Metallurgy Module defines so-called phase groups. These contain all phases of a certain kind, for example all solid slag or all slag phases. The component composition of such a phase group is calculated by weighting the component composition of each phase with their fraction in the group.

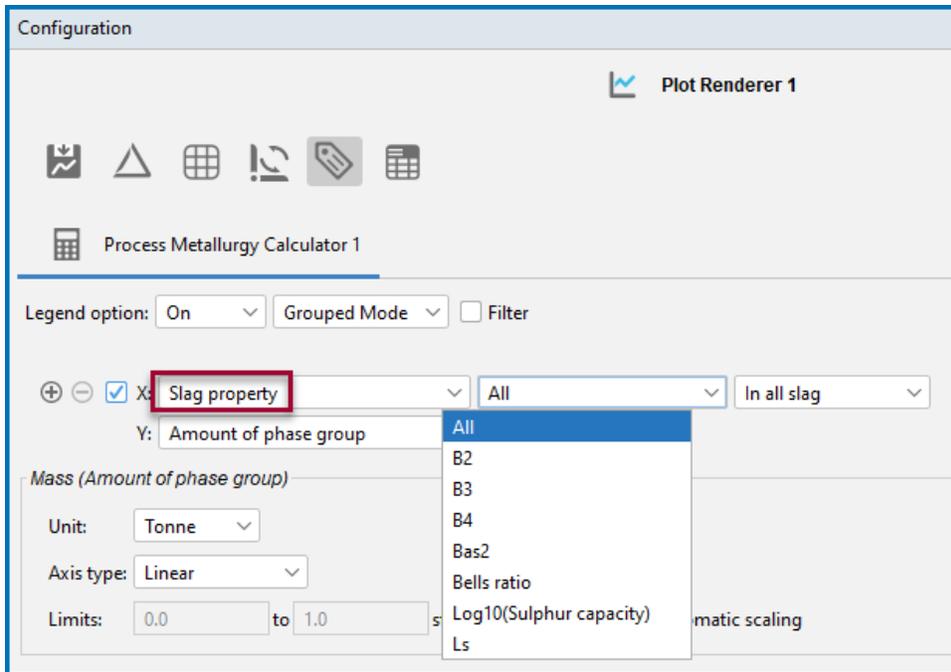
This allows to calculate slag properties for the phase groups, for example for the overall slag.

Various predefined slag properties such as different definitions of slag basicity, sulfur capacity and so forth are available for plotting or tabulating. On the **Process Metallurgy Calculator** when selecting axis variables on the **Plot Renderer** or choosing the properties to list with the **Table Renderer**, there is an option to include these slag properties.

The associated quantity selected from the next drop-down list has these options: **All**, **B2**, **B3**, **B4**, **Bas2**, **Bells ratio**, **Ls**, or **Log10(Sulphur capacity)**. Also choose **In liquid slag only**, **In solid slag only**, or **In all slag**. Depending on this setting, the composition averaged over liquid, solid, or all slag is applied as a basis for the slag property calculation.



*Slag properties for a Table Renderer.*



*Slag properties for a Plot Renderer.*

## Equations and References

The equations and references for the slag properties are defined below. Composition can be given in mass (% or fraction).

The formulas are given in Thermo-Calc console-like syntax (B: mass, W: weight fraction, AC: activity). A slag property is available if all of its components are present in the phase group. Steel and slag are the actual phase groups chosen depending on the user request, i.e. the slag property can be calculated for the liquid slag only or for the overall slag (liquid and solid).

Basicity is defined as:

$$B2 = \frac{B(CaO)}{B(SiO_2)}$$

$$B3 = \frac{B(CaO) + B(MgO)}{B(SiO_2)}$$

$$B4 = \frac{B(CaO) + B(MgO)}{B(SiO_2) + B(Al_2O_3)}$$

$$Bas2 = \frac{B(CaO) + B(MgO)}{B(SiO_2) + B(Al_2O_3) + B(TiO_2)}$$

$$Bells\ ratio = \frac{B(CaO) + 0.7 \cdot B(MgO)}{0.94 \cdot B(SiO_2) + 0.18 \cdot B(Al_2O_3)}$$

Sulfur capacity is defined as:

$$L_S = \frac{W_{(slag)}(S)}{W_{(steel)}(S)}$$

$$\text{Log}_{10}(C_s) = \text{log}_{10}(W_{(slag)}(S) \cdot 100 \cdot \left(\frac{AC(O_2, GAS)}{AC(S_2, GAS)}\right)^{0.5})$$

## References

**Basicity and MgO Saturation:** For definitions of the basicity and MgO saturation properties B2, B3, B4, and Bells ratio see [2015Kes]. Also see the Slag Atlas [1995Ver].

**Sulfur Distribution and Capacity:** For definitions of the sulfur distribution properties  $L_S$  see [1999And]. For the sulfur capacity, see Slag Atlas [1995Ver].

[1995Ver] Verein Deutscher Eisenhüttenleute (VDEh), ed. 1995. Slag Atlas 2nd Edition. Dusseldorf, Germany: Verlag Stahleisen GmbH.

[1999And] Andersson, Margareta A. T., Par G. Jönsson, and Mselly M. Nzotta. 1999. "Application Ladle Slags of the Used Sulphide Capacity Concept on High-Basicity in Bearing-Steel Production." *ISIJ International* 39 (11): 1140–49.

[2015Kes] Keskitalo, Björn. 2015. "Model Studies on the Effects of Composition Differences of Direct Reduction Pellets and an Adaptive Addition of Slag Formers for the EAF Process." Master's thesis, KTH Royal Institute of Technology, <http://urn.kb.se/resolve?urn=urn:nbn:se:kth:diva-168296>.

---

## Adiabatic Calculations: Material Enthalpy

---

An adiabatic calculation requires the enthalpy of the input materials as an initial condition. The Process Metallurgy Module uses different approaches for the different material types metal, gas, and slag. Any mechanical mixture is only applied for 1 bar / 1013 hPa.

### ***Metal***

Frozen-in global equilibrium calculation at  $T = 700\text{ °C}$ , taking the resulting enthalpy at the actual initial material temperature. This corresponds to a frozen-in microstructure. At initial material temperatures above  $700\text{ °C}$  the equilibrium is instead calculated directly.

### ***Gas***

Mechanical mixture between the enthalpy of the gas components. This is no limitation because all gas components in the databases are assumed to be ideal gas.

### ***Slag***

- **Mechanical mixture:** If the temperature is below  $T=1000\text{ °C}$  and every single material component (such as MgO, CaO, ...) is fully solid at the requested material temperature, mechanical mixture is assumed. This means that the enthalpy of every component is calculated separately and summed up (weighted by mass fraction). This corresponds to the assumption that different separate components have been added to the slag (such as burnt lime and magnesite). If any component forms gas or liquid at the initial material temperature, this mode is not used. Note that mechanical mixture is also applied if pure elements are added to the slag material, i.e. it is not the same in respect to the material enthalpy to define CaO and Ca + O.
- **Equilibrium:** If no mechanical mixture of the material components is possible or if the temperature is above  $T=1000\text{ °C}$ . This corresponds to the assumption of a fully homogenized slag material.

## Process Metallurgy Calculator

---



For information about licenses and availability, see [About the Process Metallurgy Module](#).

Use a **Process Metallurgy Calculator** to set up calculations for steel and slag used in steelmaking. Define the composition of a steel (or other metal) and slag system using material groups: one each for steel, slag, and gas. You can save your material compositions directly from the calculator and quickly access and manage the materials and groups—it collects groups of saved materials according to type, thus making it easy to retrieve them.

You can add the calculator to  **My Project** directly (right-click and select it from the **Create New Activity** menu) or use the [Process Metallurgy Template](#).



Examples of using the Process Metallurgy Calculator for Basic Oxygen Furnace (BOF) (with and without kinetics), a Ladle Furnace (LF) simulation, an Argon Oxygen Decarburization (AOD), and a Vacuum Oxygen Decarburization (VOD) simulation are included with your installation. See the [Process Metallurgy Module Examples Collection](#) for links.



You can find several learning resources on the [Process Metallurgy](#) page of our website (or in Thermo-Calc, go to **Help** → **Thermo-Calc website**). You can also [subscribe to the Thermo-Calc newsletter](#) to be kept up-to-date about upcoming releases, training, examples, videos and much more.

### ***Kinetics - Equilibrium or Process Simulation***

Once you have added a Process Metallurgy Calculator, the **Configuration** window is further branched based on the **Kinetics** option you choose, **Equilibrium**, where kinetics are not considered, or **Process simulation**, where kinetics are considered using the Effective Equilibrium Reaction Zone (EERZ) model.

All settings, no matter what kinetics branch, are defined on the **Configuration** window **Conditions** and **Options** tabs.

---

## Conditions Tab

### *Equilibrium*



For information about the individual settings on this tab, see [Equilibrium Simulation: Conditions Tab](#).

Set the conditions for your calculation that define the slag, metal, and gas. The topic, [Defining the Equilibrium Simulation](#), has instructions that walk you through the steps that represent a simplified steelmaking process.

### *Process Simulation*



For information about the individual settings on this tab, see [Process Simulation: Conditions Tab](#).

Set the conditions for your calculation that best describes the process you want to simulate. The topic, [Defining the Process Simulation](#), has instructions that walk you through the steps. Also the description of the kinetics examples are useful to help you better understand the settings.

## Options Tab



For information about the individual settings, see [Options Tab](#).

Use the **Material Manager** to create and delete groups of materials and delete materials you have saved directly from the Process Metallurgy Calculator. It is useful for frequently used materials. Also see [Working with the Material Manager](#).

The settings in **Process Material Equilibrium and Grid** define certain numerical settings used by the Thermo-Calc calculation engine for equilibrium calculations such as whether tests for global minimization should always be performed (this significantly slows down the calculations) or never be performed (this speeds up calculations but miscibility gaps might not be found), and so forth.



Calculations of oxide systems are numerically challenging, and they can often fail. The default values are chosen to give the best general performance and reliability and should only be changed by an expert user. Also, a large amount of logic is implemented in the Process Metallurgy Module to improve the convergence and calculation speed of equilibrium calculations. The timestep control is related to how finely the times defined in the **Process schedule** are resolved. See [About Dynamic Time Stepping](#) for more information.

### ***Local versus Global Options Settings***

The **Options** tab is where you can locally define specific Process Metallurgy Calculator settings. Settings can also be applied globally from the **Options** window.



To open the **Options** window: From the main menu select **Tools** → **Options** → **Graphical Mode** and click the **Activities** tab. Then click **Process Metallurgy**. See [Global Settings: Process Metallurgy](#).

## **Plots and Tables**

Right-click the node to add a [Plot Renderer](#) or [Table Renderer](#) to visualize your results. Although both nodes work the same as with the other calculators in Thermo-Calc, there are additional features and settings unique to the Process Metallurgy Module. Useful information is included throughout the documentation. You can also read more about the unique features in [Working with the Process Metallurgy Module](#).

## Defining the Equilibrium Simulation

After adding a **Process Metallurgy Calculator** to your project, you can choose to conduct an **Equilibrium** simulation, where you define the steel, slag, and gas groups of materials, or a **Process simulation**, where you set up a process using the EERZ (Effective Equilibrium Reaction Zone) model.



The following steps are for **Equilibrium** simulations. If you choose to do a **Process simulation**, see [Defining the Process Simulation](#).

This is a general overview of the setup based on the [PMET\\_01: Basic Oxygen Furnace \(BOF\)](#) example, which walks you through the steps that represent a simplified steelmaking process.

### 1: Add the Calculator and Choose the Simulation Type

1. Add a **Process Metallurgy Calculator** node to **My Projects** node. If you used the **Process Metallurgy** template on the **My Projects Configuration** window under **Applications**), click the node to display the **Configuration** settings window.
2. Under **Kinetics**, choose the simulation type, **Equilibrium** or **Process simulation**. For the following steps, choose **Equilibrium**.

### 2: Select a Database and Enter a Temperature

1. In the **Configuration** window, for the *Database* selection, the program automatically defaults to the highest available version of the TCOX database.



The Process Metallurgy Module requires both a valid Maintenance and Support Subscription (M&SS) and a license for the TCS Metal Oxide Solutions Database (TCOX8 or newer).



It is not possible to use custom user databases with the Process Metallurgy Module. This is because several internal calculations, such as determining the component compositions of ionic phases, are dependent on the database. Therefore, only TCS Metal Oxide Solutions Database (TCOX) versions 8 and newer, and to a limited extent, OXDEMO, can be used with this Module.



All users can test the Process Metallurgy Module with the included OXDEMO database, which is limited to these elements: Al, C, Ca, Fe, O, S, and Si. For more information about this and other products [visit our website](#).

2. Set the **Thermal Control** as being **Isothermal** or **Adiabatic**.
3. Select a **Temperature** unit (**Kelvin**, **Celsius**, or **Fahrenheit**) and enter a numerical value in the field.
4. Select a **Pressure** unit (**Pascal**, **Atmospheres**, or **Bar**) and then enter a numerical value in the field.

### ***3: Define the Conditions for the Steel, Slag, and Gas Material Groups***

Next define each material group—**Steel** (or any user-defined **Metal** group), **Slag**, and **Gas**.



See [Equilibrium Simulation: Conditions Tab](#) for an overview of the layout of this section. There is also an overview of these material groups in [Working with the Process Metallurgy Module](#).

1. Choose a **Material** to set up first, **Steel** (or any user-defined metal), **Slag**, or **Gas**.
2. Select an **Amount** from the list—**Tonne**, **Kilogram**, **Gram**, or **Pound**. Enter a numerical value in the field.
3. If this is the first time you are defining a material, or at any time when **User-defined** is selected, click **Show composition** to set the composition for the material (steel, slag, or gas or another user-defined metal).



The composition for each material is hidden by default because the Module allows you to easily save materials and access these from the material menu. Once you save a collection of materials, you will rarely have to click **Show composition**. See [Working with the Material Manager](#) for details about saving and deleting materials.

4. To the right of the **Material** list (e.g. for the **Steel** group), choose the material composition you want to use for this simulation. Predefined **Example Steel** and **Example Slag** compositions are included with the installation.
  - If you chose **Adiabatic** as the **Thermal Control**, also enter a **temperature** in the field for each material. This is the initial temperature before the material is added to the process.
5. From the first **Input type** list select **Mass percent**, **Mass fraction**, **Mole percent**, **Mole fraction**, or **Absolute amount**. For **Absolute amount**, you can directly enter values in the fields in the unit selected above for **Amount (Tonne, Kilogram, Gram, or Pound)**.
6. For the next list, the default is based on which group you are defining. For the Steel (Metal) group it is **Element**, for Slag it is **Component**, and for Gas it is **Gas component**.
7. Choose **Major component** so that as you add other components, the major one is adjusted to be the rest of the composition. Choose **Normalize to total of 100%** so that the sum is adapted to 100%. This is useful if you need to work with a given composition that does not sum up to 100%. If you select **Absolute amount**, **Normalize to total of 100%** is not available.
8. Based on which material group you are working with, add the **Element**, **Component**, or **Gas component**.



To add one or more material groups click the add **+** button. Click the minus **-** button to remove a material group. The new material inherits the default settings of that group.

9. After you have added and defined the element, component, or gas component it is recommended you save and name the material. This makes it easier to work with because the name of the material is automatically updated in the lists (and available as part of your library) instead of being named *User-defined* everywhere.

The material is automatically saved in the relevant material group, so only steel materials are shown in the steel menu, only slag materials are shown in the slag menu, and only gas in the gas menu. You can also create additional material groups if there are groups you use regularly, such as scrap compositions, ferro-alloys or slag additions. See [Working with the Material Manager](#) for more details about creating and deleting material groups and deleting materials.

10. Continue adding and defining the material groups (e.g. Steel, Slag, and Gas). You could, for example, add another Steel group if you wanted to define a composition of a steel scrap to add to a basic oxygen furnace (BOF).

#### 4: Select the Calculation Type and Associated Definitions

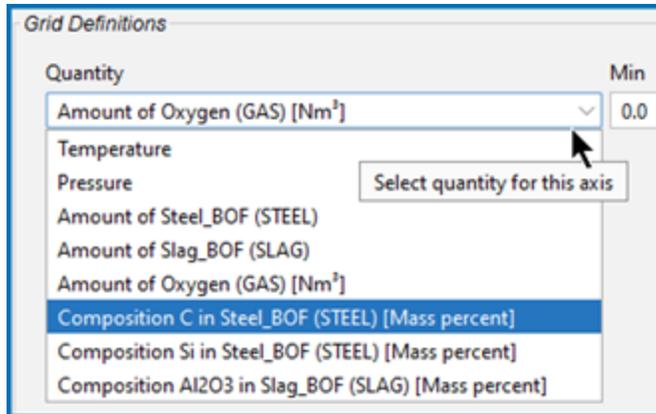
 These settings are described more in [Equilibrium Simulation: Conditions Tab](#).

1. Under *Calculation Type*, click to choose **Single** (no axes), **One Axis**, **Grid**, or **Uncertainty**.



For more complex calculations, it is good practice to start with a **Single** equilibrium calculation to make sure the calculation converges. Also see the settings described in [Options Tab](#). These can be useful if your calculation fails.

2. Based on the calculation type, define either the *Grid Definitions* (for **One Axis** or **Grid** calculations) or the *Sampling of Data from Gaussian Distributions* (for **Uncertainty** calculations). The **Quantity** displays all the stepping options set up with the material groups for **Steel**, **Slag**, and **Gas**. The example shows the steel group quantities.



The *Amount of X in Y* quantities you can choose from the various drop-down lists (such as under *Grid Definitions*) are defined as Min-Max variation of the component X in Y. If a dependent element is present that is adapted as usual.



How these quantities relate to the material definition is described in [Working with the Process Metallurgy Module](#).

## 5: Add and Define a Plot or Table and Run the Simulation

Once you have finished defining the Process Metallurgy Calculator, you also define the plot or table settings. To add a Plot Renderer or Table Renderer, right-click the calculator node and select an option from the **Create New Successor** menu.



The settings are generally described for the individual nodes. See [Plot Renderer](#) and [Table Renderer](#). Some useful information about the plots is also described in [Working with the Process Metallurgy Module](#).

## Equilibrium Simulation: Conditions Tab

The following outlines what you can set on the **Configuration** window **Conditions** tab when working with the **Process Metallurgy Calculator** and an **Equilibrium** simulation.



Also see [Defining the Equilibrium Simulation](#) for instructions that walk you through the steps that represent a simplified steelmaking process. A more complex process is described in [Defining the Process Simulation](#).

The screenshot displays the 'Configuration' window for 'Process Metallurgy Calculator 1'. The 'Conditions' tab is active, showing the following settings:

- Kinetics:** Equilibrium (selected), Process simulation
- Conditions:**
  - Database: TCOX11
  - Thermal control: Isothermal
  - Temperature: Celsius, 1700.0
  - Pressure: Pascal, 100000.0
- Calculation Type:** Single, One axis (selected), Grid, Uncertainty
- Grid Definitions:**

Quantity	Min	Max	Number of steps
Amount of User-defined (SLAG)	0.0	1.0	10

A detailed view of the material list shows three entries:

- Material:** Steel, User-defined, Amount: 0.0, Input type: Mass percent, Major component: Fe, Total: 100.0
- Material:** Slag, User-defined, Amount: 0.0
- Material:** Gas, User-defined, Amount: 0.0

An example of the different settings areas for a default Configuration window for a Process Metallurgy Calculator when an Equilibrium simulation type is selected. There are many settings available depending on the parameters.

### Database

The most recent available version of TCS Metal Oxide Solutions Database (TCOX) is selected by default.



The Process Metallurgy Module requires both a valid Maintenance and Support Subscription (M&SS) and a license for the TCS Metal Oxide Solutions Database (TCOX8 or newer).



It is not possible to use custom user databases with the Process Metallurgy Module. This is because several internal calculations, such as determining the component compositions of ionic phases, are dependent on the database. Therefore, only TCS Metal Oxide Solutions Database (TCOX) versions 8 and newer, and to a limited extent, OXDEMO, can be used with this Module.



All users can test the Process Metallurgy Module with the included OXDEMO database, which is limited to these elements: Al, C, Ca, Fe, O, S, and Si. For more information about this and other products [visit our website](#).

## Thermal Control

Choose **Isothermal** or **Adiabatic**. When **Adiabatic** is selected, no global temperature is defined (as this changes during the process). Instead the initial temperature has to be defined separately for each material. Adiabatic calculations assume no heat and mass exchange with the environment during the equilibrium reaction. Certain types of processes run at almost constant temperatures. For such processes isothermal conditions are acceptable. But in most metallurgical processes the temperature will change sometimes dramatically in function of time due to heating or cooling of the system and due to exothermal or endothermal reactions taking place. For such processes adiabatic conditions are a better choice.

## Temperature

Choose a **Temperature** unit (**Kelvin**, **Celsius**, or **Fahrenheit**) and then enter a numerical value in the field.

## Pressure

Choose a **Pressure** unit (**Pascal**, **Atmospheres**, or **Bar**) and then enter a numerical value in the field.

## Material Groups (Steel, Slag, and Gas)



For more details about these fields, see [Defining the Equilibrium Simulation](#). See also [Working with the Material Manager](#).

There are three groups to define: **Steel** (or another user-defined **Metal** group), **Slag**, and **Gas**.

In the image below, the red-highlighted area is where the **Steel** material group is defined. The drop-down menu that displays *Steel\_BOF* is a user-defined material. This can be defined and then saved to the material library.

*The three material group areas on the Process Metallurgy Calculator where you can define components and save materials.*

When you click **Show composition** next to any material group, additional fields display (e.g. **Input type**, **Major components**, and so forth) where you can further define each material group. You can add and remove materials using the *Material Manager* (available via the **Configuration** window → **Options** tab).



You can open the example project [PMET\\_01: Basic Oxygen Furnace \(BOF\)](#) from Thermo-Calc (**Help** → **Example Files** → **Process Metallurgy** → **PMET\_01\_Basic\_Oxygen\_Furnace.tcu**). Then click **Show composition** next to the **Slag** and **Gas** groups to see how the *Slag\_BOF* and *Oxygen* materials are defined.

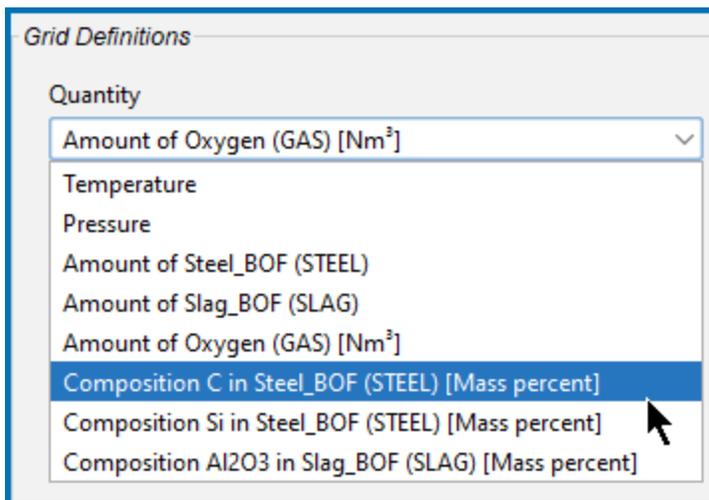
From this composition area you can click the **Save material** button to add the new material to the group's list. The newly named material composition is then available for use with that material group. In this example, the *Steel\_BOF* material was defined first and then named at the end. It is now automatically available wherever you select the Steel material or need to choose quantities to plot.



The *Amount of X in Y* quantities you can choose from the various drop-down lists (such as under *Grid Definitions*) are defined as Min-Max variation of the component X in Y. If a dependent element is present that is adapted as usual.



To add one or more material groups click the add  $\oplus$  button. Click the minus  $\ominus$  button to remove a material group. The new material inherits the default settings of that group.



After defining and saving a material composition, the named material is available for selection in a variety of places such as when choosing quantities on the calculator and defining axis variables on the Plot Renderer.

---

## Calculation Type

Click to select a *Calculation Type*.

- **Single:** To calculate a single point. The results from this calculation can be displayed in the **Visualizations** window using a **Table Renderer**.
- **One Axis:** To vary a quantity on the X-axis.
- **Grid:** Evaluates two axis variables of the selected quantities in the specified range and number of steps.
- **Uncertainty:** Evaluates calculations where the values of the quantities are sampled from Gaussian distributions. The **Mean** field is as defined under *Condition Definitions* for the respective quantity. The result is visualized as a histogram or normal probability plot by adding a Plot Render activity.

 [Plot Types](#)

### Grid Definitions (One Axis or Grid)

The **Quantity** menu displays all the stepping options set up with the material groups for **Steel** (or any user-defined metal group), **Slag**, and **Gas**.

For **One Axis**, select a **Quantity** to vary along the X-axis, for example, **Temperature** then enter a **Min**, **Max**, and **Number of steps**.

For the **Grid** calculation type, define the two axes variables using the fields and menus: **Quantity**, **Min**, **Max**, and **Number of steps**. The number of steps along with the minimum and maximum values for the axes define a grid. For each grid point the selected equilibrium is evaluated.

 Depending on the calculation type selected, there are also a variety of [Plot Types](#) to choose from.

### Sampling of Data from Gaussian Distributions (Uncertainty)

For the **Uncertainty** calculation type, click to select the applicable checkboxes under **Quantity**. The **Mean** field is for the distribution function as defined under *Condition Definitions* for the respective quantity. The default  $\Delta$ Min/Max is a 95% confidence interval.

 [About the Uncertainty Calculations](#)

## Sampling Parameters (Uncertainty)

For the **Uncertainty** calculation type, choose a number of **Standard deviations** where you want to **Truncate the distribution**. Choose the **Total number of samples**.



[About the Uncertainty Calculations](#)

## Defining the Process Simulation

---

After adding a **Process Metallurgy Calculator** to your project, you can choose to conduct an **Equilibrium** simulation, where you define the steel, slag and gas groups of materials, or a **Process simulation**, where you set up a process using the EERZ (Effective Equilibrium Reaction Zone) model.

 [Setting Up a Process Metallurgy Simulation](#)



The following steps are for **Process simulation**. If you choose to do an **Equilibrium**, see [Defining the Equilibrium Simulation](#).

### 1: Add the Calculator and Choose the Simulation Type

1. Add a **Process Metallurgy Calculator** node to  **My Projects** node. If you used the **Process Metallurgy** template on the  **My Projects Configuration** window under **Applications**), click the node to display the **Configuration** settings window.
2. Under **Kinetics**, choose the simulation type, **Equilibrium** or **Process simulation**. For the following steps, choose **Process simulation**.



Only adiabatic conditions are available for process simulations as this corresponds best to most practical processes.

### 2: Select a Database and Enter a Temperature

 [Process Simulation: Conditions Tab](#)

1. In the **Configuration** window, for the *Database* selection, the program automatically defaults to the highest available version of the TCOX database.



The Process Metallurgy Module requires both a valid Maintenance and Support Subscription (M&SS) and a license for the TCS Metal Oxide Solutions Database (TCOX8 or newer).



It is not possible to use custom user databases with the Process Metallurgy Module. This is because several internal calculations, such as determining the component compositions of ionic phases, are dependent on the database. Therefore, only TCS Metal Oxide Solutions Database (TCOX) versions 8 and newer, and to a limited extent, OXDEMO, can be used with this Module.



All users can test the Process Metallurgy Module with the included OXDEMO database, which is limited to these elements: Al, C, Ca, Fe, O, S, and Si. For more information about this and other products [visit our website](#).

2. Select a **Temperature** unit (**Kelvin**, **Celsius**, or **Fahrenheit**).
3. Select a **Time** unit (**Seconds**, **Minutes**, **Hours**, or **Days**).

### 3: Define the Process Model



#### [Process Simulation: Process Model Settings](#)

1. Enter a **Name** for the process and the total **Duration** of the process.
2. Choose a **Process model** from the list or click **Edit Process Model** to define a new model or edit an existing one.
3. In the **Process model** the following is defined:

- a. **Pressure.** This is can be either **Constant** throughout the process (enter a constant via the Edit Process Model) or be changed during the process (select **Table input** via the Edit Process Model) and then enter pressure in the **Process schedule** table.
- b. **Zones of the Effective Equilibrium Reaction Zone model.** Only two zones can be defined. The densities need to be given manually. They are used to calculate the volume of the reaction zones, which is in turn used for the calculation of the reaction kinetics. Choose to allow degassing or not.
- c. **Area of the Reaction zone and mass transfer coefficients.** The area is typically the area of the interface between the two zones. The mass transfer coefficients state how fast material is moved into the reaction zone. Large values result in fast kinetics, small numbers in slower kinetics. These values strongly depend on the type of material (mass transfer in slags is slower than in liquid steel due to its higher viscosity) and type of process (the violent reactions in a basic oxygen furnace have higher mass transfer coefficients compared to the gentle stirring performed in a ladle furnace. These parameters are basically fitting parameters that need to be determined based on comparison with experimental data. Typical values can also be found in the literature. Choose to allow degassing or not.
- d. **Transfer of phase group.** This defines how fast “foreign” phases are moved from one zone to the other. Foreign phases could be solid or liquid oxide inclusions in the liquid steel, or metallic droplets in the slag phase. A typical application is the simulation of the flotation of oxide inclusions out of the liquid steel into the slag phase during refining in a ladle furnace.
- e. **The Heat entering or leaving the system and being transferred between the zones.** Constant cooling defines how much heat is lost from the system by radiation and convection. Separate values can be given to each zone if needed. Heat is used when the system is actively heated by an external heat source such as an electric arc or induction heater. Here only the heating efficiency is given, which is the percentage of for example electrical power that is transferred to the system as heat. The times and power are defined in the **Process schedule**. Different amounts of heat and material are added to the zones resulting in the zones having different temperatures. Heat transfer defines how heat flows from one zone to the other. A high value results in a quick equalization of temperature, a small value results in the zones keeping their different temperatures.

## 4: Define the Materials



[Process Simulation: Materials Tab](#)

The materials can be defined, saved and retrieved in the same way as for an equilibrium simulation. For a process simulation, it must be defined to which zone the material is added. Typically, oxide materials are added to the slag zone and metallic materials are added to the steel zone. Gas or other injected materials (such as carbon or lime powder) are typically added to the reaction zone. But this choice depends on the process to be simulated and must be carefully chosen. By defining the same material twice or even three times, it can be split, so that part of the material is added to one zone and part of it to another zone.

## 5: Define the Process Schedule



[Working with the Process Schedule](#)



[Process Simulation: Process Schedule Tab](#)

On the **Process schedule** tab, the sequence and amounts of additions during the duration of the process are defined. All the materials defined in the **Materials** tab automatically are included in the table. If a heat addition is defined in the Process Model this is also included in the table. If the units **Tonne**, **Kilogram**, or **Pound** are selected, then the additions are defined as one-time additions at a certain time. If a rate is chosen as unit of the addition (such as **Kilogram per minute** or **Kilogram per second**, for gas also **Normal cubic meter per minute** or per second can be given) then the rate of addition is entered into the table. The additions are visualized in a plot for clarity.

## 6: Run the Simulation and Add and Define a Plot or Table

When the simulation is set up, click **Perform Tree** to run it. Plots and tables can be added in the same way as for equilibrium calculations.

## Process Simulation: Conditions Tab



See [Defining the Equilibrium Simulation](#) for instructions that walk you through the steps that represent a simplified steelmaking process. A more complex process is described in [Defining the Process Simulation](#) and where the settings in the following topics are described.



Examples of using the Process Metallurgy Calculator for Basic Oxygen Furnace (BOF) (with and without kinetics), a Ladle Furnace (LF) simulation, an Argon Oxygen Decarburization (AOD), and a Vacuum Oxygen Decarburization (VOD) simulation are included with your installation. See the [Process Metallurgy Module Examples Collection](#) for links.

The following outlines what you can set on the **Configuration** window **Conditions** tab when working with the **Process Metallurgy Calculator**.

Below is based on selecting a **Process simulation** under **Kinetics**. The settings to complete are for the **Process Model**, the **Materials**, and the **Process Schedule**.

The screenshot displays the Configuration window for Process Metallurgy Calculator 1. The **Conditions** tab is active, showing the **Kinetics** section with **Process simulation** selected. The **Materials** tab is also visible, showing a list of materials added to the process. The **Process schedule** tab is also visible, showing a table of process parameters and a bar chart of additions.

**Materials** Process schedule

Material	Temperature	Added to
Steel BOF_HotMetal	1350.0	Zone Steel
Steel BOF_Scrap	25.0	Zone Steel
Slag BOF_Ore	25.0	Zone Steel

**Materials** Process schedule

Plot	Time [Minutes]	Value
<input checked="" type="checkbox"/> BOF_HotMetal (STEEL)	Tonne	87.0
<input checked="" type="checkbox"/> BOF_Scrap (STEEL)	Kilogram per minute	2000.0
<input checked="" type="checkbox"/> BOF_Ore (SLAG)	Tonne	1.0
<input checked="" type="checkbox"/> BOF_Lime (SLAG)	Tonne	3.0
<input checked="" type="checkbox"/> Oxygen_RZ (GAS)	Normal m <sup>3</sup> per minute	200.0
<input checked="" type="checkbox"/> Oxygen_steel (GAS)	Normal m <sup>3</sup> per minute	100.0

**Additions**

Bar chart showing additions over time (0.0 to 3.0 minutes). The y-axis is labeled 'Rate' and ranges from 70 to 90. The chart shows a single bar at 0.0 minutes with a value of 2,000.0.

An example of the different sections on the Configuration window for a Process Metallurgy Calculator when a Process simulation type is selected. This screen shot is from the example PMET\_04c. There are many settings available depending on the parameters.

## Database

The most recent available version of TCS Metal Oxide Solutions Database (TCOX) is selected by default.



The Process Metallurgy Module requires both a valid Maintenance and Support Subscription (M&SS) and a license for the TCS Metal Oxide Solutions Database (TCOX8 or newer).



It is not possible to use custom user databases with the Process Metallurgy Module. This is because several internal calculations, such as determining the component compositions of ionic phases, are dependent on the database. Therefore, only TCS Metal Oxide Solutions Database (TCOX) versions 8 and newer, and to a limited extent, OXDEMO, can be used with this Module.



All users can test the Process Metallurgy Module with the included OXDEMO database, which is limited to these elements: Al, C, Ca, Fe, O, S, and Si. For more information about this and other products [visit our website](#).

## Temperature

Select a **Temperature** unit (**Kelvin**, **Celsius**, or **Fahrenheit**).

## Time

Select a **Time** unit (**Seconds**, **Minutes**, **Hours**, or **Days**).

---

## Process Simulation: Process Model Settings

---



The following outlines what you can set on the **Configuration** window under **Process** and when you click **Edit Process Model**. The overall steps to set this up are described in [Defining the Process Simulation](#).

### Process

Under *Process*, enter a **Name** and **Duration**.



For more complex processes, you could, for example, enter the process reference as its *Name* as is done for the EERZ example **PMET\_06\_Ladle\_Furnace\_Kinetics.tcu**.

### Process Model

Choose a *Process model*. Click **Edit Process Model** to enter details as below.



You can create your own **Process model** and name them so these are available to choose from the list in the future. This is done using the **Edit Process Model** window.

## Edit Process Model

To open the **Edit Process Model** window, click **Edit Process Model** to access the following settings.

### **Pressure**

For **Pressure**, this can be either **Constant** throughout the process or be changed during the process (using **Table input** here and then entering values in the Process Schedule). Choose:

- **Constant** to choose a unit (**Pascal**, **Atmospheres**, or **Bar**) and then enter a numerical value in the field.



Only constant pressure throughout the process can be used. Also, the pressure only influences reactions involving the gas phase.

- **Table input** to add or use the values entered in the **Process Schedule** table.

## Zones

In this section define the material zone(s), which are added to a **Zone** or **Reaction zone** on the **Materials** tab.

Enter a **Name** and **density** (in kg/m<sup>3</sup>) for the zones.

For example, enter `Steel` and `Slag`, which in the next section are available to define for **Zone 1** or **Zone 2** in the Reaction zone.

Select the **Allow degassing** checkbox to be able to simulate vacuum degassing, for example.

Click the **Add**  button to define additional items and the **Remove**  button to delete items.

## Reactions

In this section you are defining either a **Reaction zone** and/or **Transfer of phase group**.

Select the **Allow degassing** checkbox to be able to simulate vacuum degassing, for example.

Click the **Add**  button to define additional items and the **Remove**  button to delete items.

## Reaction Zone

If you choose **Reaction zone** from the list then enter an *area* (in m<sup>2</sup>).

Next choose from each **Zone 1** and **Zone 2** list the Zone as defined above, for example **Steel** or **Slag**.

---

For **mass transfer coefficient** choose:

- **Constant** to enter a numeric value in the field in m/s for each.
- **Table input** to add or use the values entered in the **Process Schedule** table.

Reaction zone area and mass transfer coefficient defines the kinetics of the reaction between the two zones. A large area and fast mass transfer results in fast kinetics.

## Transfer of Phase Group

If you choose to define a **Transfer of phase group**, then choose the state as **Liquid metal**, **Solid metal**, **All metal**, **Solid oxides**, **Liquid oxides**, **All oxides**, or **Gas**.

Then choose the two zones (e.g. Steel or Slag) to define **from zone** and **to zone**. Then enter a value and choose a time from the list, **Percent per minute** or **Percent per second**.

This parameter defines how fast “foreign” phases are transferred from one zone to another. A typical example is the flotation of oxide inclusions in liquid steel up into the slag phase.

## Heat

Choose the type of heat to apply to the process: **Constant cooling**, **Heat transfer**, and/or **Heat**.

Click the **Add** ⊕ button to define additional items and the **Remove** ⊖ button to delete items.

## Constant Cooling

If you choose **Constant cooling**, enter a **Name**, choose either **Zone** or **Reaction Zone** and enter a numerical value in the field in MW (megawatts).

If you choose **Zone**, you can further choose either Zone defined above (e.g. **Steel** or **Slag**). If you choose **Reaction Zone**, then heat is lost from the EERZ, the size of which is calculated at each timestep depending on the kinetic parameters defined in **Reaction Zone**.

This parameter defines how much heat is lost from a zone (or the reaction zone) by radiation or convection and results in an overall cooling of the system.

## Heat Transfer

If you choose **Heat transfer**, select two zones between where heat should flow and enter a **heat transfer coefficient** in  $Wm^2/K$ .

As different amounts of material and also different amounts of heat can be added to the different zones, these temperature are different. As the two zones are in contact heat flows from the hotter zone to the cooler zone thus equalizing the temperature. The amount of heat flowing from one zone to the other is defined by the **Heat transfer**. A high heat transfer coefficient leads to fast temperature equalization, a small heat transfer coefficient results in the zones keeping different temperatures.

## Heat

If you choose **Heat**, enter a *Name*, then select **Zone** or **Reaction zone**, choose a Zone (e.g. **Steel** or **Slag**) and enter a percentage of **MW efficiency** in the field.

Select **Heat** if the system is actively heated by an external heat source such as an electric arc or induction coil. Here only the efficiency is defined, which is the percentage of power (for example electrical power) that is transformed into heat that enters the system and heats it up. The profile of power versus time is defined in the **Process Schedule**.

Click **Save** when the Process Model is defined and enter a **Process model name**. Each time you save a process model it adds it to the list on the main **Configuration** window, which you can use in future set ups.

## Process Simulation: Materials Tab

---



The following outlines what you can set on the **Configuration** window **Materials** tab. The overall steps to set this up are described in [Defining the Process Simulation](#).



This section is similar to how you define the materials for the Equilibrium branch. Some information is explained in more detail for [Equilibrium Simulation: Conditions Tab](#) and not repeated here.

### Material

Materials are grouped into **Steel** (or another user-defined **Metal** group), **Slag** (or another user-defined non-metallic group), and **Gas**.

Some of what you choose from lists in this section depend on what you have entered as the **Process Model** above. For example, the **Zone** or **Reaction zone** options are based on those entries (e.g **Steel** or **Slag**).



To add one or more material groups click the add **+** button. Click the minus **-** button to remove a material group. The new material inherits the default settings of that group.

After choosing the Material for the section being defined, also enter a value of the *temperature* in the field. Choose what **Zone** or **Reaction zone** it is to be **Added to**.

The times and amounts of additions are defined in the **Process schedule**.

### Show Composition

When you click **Show composition** next to any material group, additional fields display where you can further define each material group.

From this composition area you can click the **Save material** button to add the new material to the group's list. The newly named material composition is then available for use with that material group.

From the first **Input type** list select **Mass percent**, **Mass fraction**, **Mole percent**, or **Mole fraction**.

For the next list, the default is based on which group you are defining. For the Steel (Metal) group it is **Element**, for Slag it is **Component**, and for Gas it is **Gas component**.

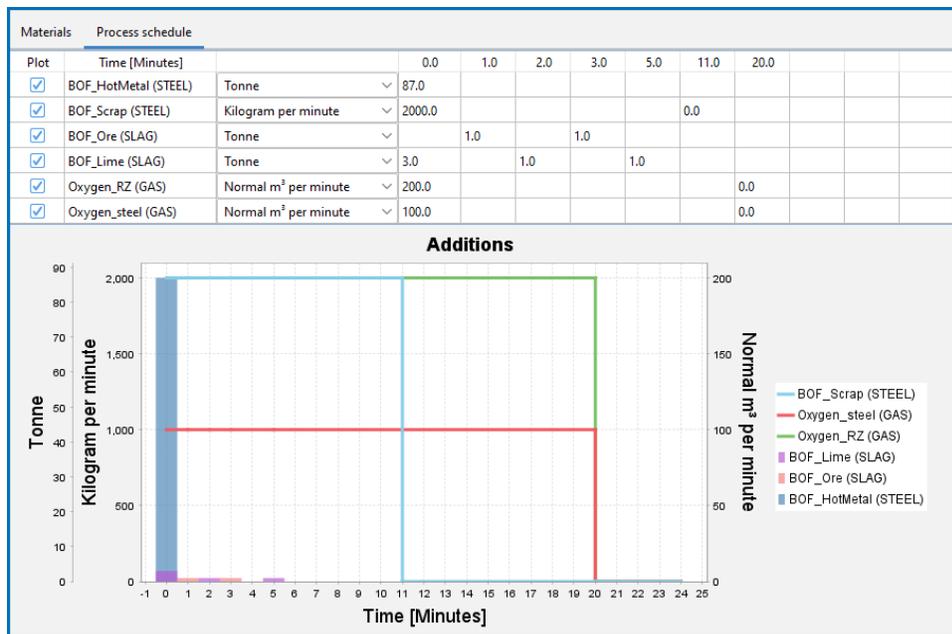
Choose **Major component** so that as you add other components, the major one is adjusted to be the rest of the composition. Choose **Normalize to total of 100%** so that the sum is adapted to 100%. This is useful if you need to work with a given composition that does not sum up to 100%.

## Process Simulation: Process Schedule Tab



The following outlines what you can set on the **Configuration** window **Process Schedule** tab. The overall steps to set this up are described in [Defining the Process Simulation](#). Also see [Working with the Process Schedule](#).

On the **Process schedule** tab, the sequence and amounts of additions during the duration of the process are defined. All the materials defined in the **Materials** tab automatically are included in the table. If a heat addition is defined in the *Process Model* this is also included in the table.



### Plot

Select the **Plot** checkbox to include or exclude the components in the process schedule plot area, for example, to help you examine the changes in the plot window, after defining the Steel, Scrap, Slag, or Gas rows, see what happens when you remove one or more items from the plot.

## Unit

Enter a value and choose a unit **Tonne**, **Kilogram**, **Pound**, **Kilogram per second**, or **Kilogram per minute**. For **Gas** you can also choose **Normal m3 per minute** as the unit.

If a unit **Tonne**, **Kilogram**, or **Pound** is selected, then the additions are defined as one-time additions at a certain time. If a rate is chosen as unit of the addition (such as Kilogram per minute or second, for gas Normal cubic meter per minute or second) then the rate of addition is entered into the table and it stays constant until a different value is entered.

## Options Tab

---

These settings are for the **Process Metallurgy Calculator** and are located on the **Configuration** window → **Options** tab. Only unique settings are described below.



You can change these settings locally for a specific Calculator or globally for some defaults in the **Options window** (a different location than the tab).

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

then click the **Graphical Mode** tab and the **Process Metallurgy** node in the tree.



[Equilibrium Simulation: Conditions Tab](#)

## Material Manager

Use the **Material group** and **Material** lists to create and delete material groups, as well as delete material compositions that are saved from the Process Metallurgy Calculator. It is useful for frequently used materials.



See [Working with the Material Manager](#) for details.

- **Material group:** To create or delete a group, choose it from this list, e.g. **Steel**, **Slag**, or **Gas**. The *Steel* group is simply a generic **Metal** group. You can create other user-defined metal groups and then these are listed. Click **Create group** or **Delete group** as required to manage the list.
- **Material:** To delete a material, choose the group type then select the material composition to delete. For example, select the **Slag** material group, then choose the **Example Slag** material and click **Delete material** to remove it from all lists.

## Process Material Equilibrium and Grid

### Minimization Strategy

Under *Process Material Equilibrium and Grid*, the default uses the **Global test preferred** option for **Equilibrium** simulations and the **Local minimization preferred** option for **Process simulations**.

The **Minimization Strategy** setting is used to ensure that the most stable minimum under the specified conditions is computed.

With either the *Global test preferred* or *Local minimization preferred* setting, the program cycles through options before it gives up:

- For **Global test preferred** the minimization starts with a global test and if that fails it runs full global minimization.
- For **Local minimization preferred** the minimization starts with a local minimization, in case of failure it tries a global test and finally a full global minimization.

If you choose **Local minimization only** or **Global minimization only**, the program just tries the one setting and gives up if it fails.



For general information about global minimization, see the topic related to the Console Mode command, [GLOBAL\\_MINIMIZATION](#).

### Max No. of Iterations

Under *Process Material Equilibrium and Grid*, the default for the **Max no. of iterations** is that the program tries 2000 iterations before it gives up.

### Required Accuracy

The default **Required accuracy** is  $1.0E-6$ . This is a relative accuracy, and the program requires that the relative difference in each variable must be lower than this value before it has converged. A larger value normally means fewer iterations but less accurate solutions. The value should be at least one order of magnitude larger than the machine precision.

## Smallest Fraction

Under *Process Material Equilibrium and Grid*, the default **Smallest fraction** is  $1.0E-16$ . If this value fails, the program tries other values. This is the value assigned to constituents that are unstable.



The default value is for all phases except for the IDEAL phase with one sublattice site (such as the GAS mixture phase in many databases) where the default value is always as  $1E-30$ .

## Max Grid Points (in Global Minimization)

Under *Process Material Equilibrium and Grid*, from the **Max grid points** list choose:

- **Coarse** for 2000 grid points, the default,
- **Medium** for 20,000 grid points,
- **Fine** for 200,000 grid points, or
- **Custom** to set your own number of grid points.

## Approximate Driving Force for Metastable Phases

Under *Process Material Equilibrium and Grid*, the **Approximate driving force for metastable phases** checkbox is selected by default, which means the metastable phases are included in all iterations. However, these may not have reached their most favorable composition and thus these driving forces may be only approximate.

## Force Positive Definite Phase Hessian

Under *Process Material Equilibrium and Grid*, the **Force positive definite phase Hessian** checkbox is selected by default and this setting determines how to reach the minimum of an equilibrium state.

## Control Step Size

Under *Process Material Equilibrium and Grid*, the **Control step size** checkbox is selected by default and this setting determines how to reach the minimum of an equilibrium state.

---

## Timestep Control



This section is used with the **Process simulation** calculations. Read [About Dynamic Time Stepping](#) to better understand these settings.

### Timestep

Under *Timestep control*, enter numerical values in the fields for the **Timestep**:

- **Initial %**
- **Smallest allowed** in seconds (s)
- **Max % of duration**

### Max Temperature Change Per Time Step

Under *Timestep control*, in the field enter the **Max temperature change per time step** in Kelvin (K).

---

# Nickel Model Library Property Models

---

 [Property Model Calculator](#)

## In this section:

---

About the Nickel Model Library Property Models .....	563
About the Antiphase Boundary Energy Property Model .....	565
About the Coarsening Nickel Property Model .....	571
About the Equilibrium with Freeze-in Temperature Nickel Property Model .....	575
About the Solvus for Ordered Phase Property Model .....	583
About the Strain-Age Cracking Property Model .....	587

---

## About the Nickel Model Library Property Models

---

These **Nickel Models** are available with the [Property Model Calculator](#) and as part of the Nickel Model Library.



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).

### ***Antiphase Boundary Energy - Ni***

The **Antiphase Boundary Energy - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates the antiphase boundary energy for the gamma prime ( $\gamma'$ ) phase in Ni-base alloys.

See [About the Antiphase Boundary Energy Property Model](#) and [Antiphase Boundary Energy Property Model Settings](#) for background theory and input parameter details.

### ***Coarsening - Ni***

The **Coarsening - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates the coarsening rate coefficient  $K$  of one or several precipitate phases in a matrix phase, assuming spherical geometry of the precipitating phase (s).

See [About the Coarsening Nickel Property Model](#) and [Coarsening Nickel Property Model Settings](#).

### ***Equilibrium with Freeze-in Temperature - Ni***

The **Equilibrium with Freeze-in Temperature - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates equilibrium at the freeze-in temperature and evaluates the properties at a different temperature.

See [About the Equilibrium with Freeze-in Temperature Nickel Property Model](#) and [Equilibrium with Freeze-in Temperature Nickel Property Model Settings](#).

### ***Solvus for Ordered Phase - Ni***

The **Solvus for Ordered Phase - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates the solvus of ordered phases such as FCC\_L12, BCC\_B2, NI3TA\_D0A, NI3TI\_D024, TCP-phases (Sigma), and so forth.

See [About the Solvus for Ordered Phase Property Model](#) and [Solvus for Ordered Phase Property Model Settings](#).

### ***Strain-Age Cracking - Ni***

The **Strain-Age Cracking - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, is used in applications related to welding or additively manufactured components, where there are interface strains between  $\gamma'$  (gamma prime) and the matrix.

See [About the Strain-Age Cracking Property Model](#) and [Strain-Age Cracking Property Model Settings](#).

## About the Antiphase Boundary Energy Property Model

The **Antiphase Boundary Energy - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates the antiphase boundary energy for the gamma prime ( $\gamma'$ ) phase in Ni-base alloys.



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).



For an example, see [PM\\_Ni\\_02: Antiphase Boundary Energy of  \$\gamma'\$](#)



The input parameters are entered on the **Configuration** window for the Property Model Calculator. There are also additional axis variables available on the Plot Renderer. See [Antiphase Boundary Energy Property Model Settings](#) for details.

Alloy strength can be assumed [2014Cru] to be proportional to the product of the antiphase boundary energy and the square root of the fraction of the strengthening gamma-prime phase.

Equilibrium is calculated with the selected phases and conditions. The antiphase boundary energy is then evaluated for the gamma prime phase given the equilibrium composition.

### References

- [1995Mio] A. P. Miodownik, N. J. Saunders, "The calculation of APB Energies in LI, Compounds Using a Thermodynamic Database," in Applications of Thermodynamics in the Synthesis and Processing of Materials, P. Nash, B. Sundman, Eds. (TMS, Warrendale, PA, 1995), pp. 91–104.
- [2000Sau] N. Saunders, M. G. Fahrman, C. J. Small, in Superalloys 2000 (TMS, Warrendale, Pa., 2000; pp. 803–811.
- [2014Cru] D. J. Crudden, A. Mottura, N. Warnken, B. Raeisnia, R. C. Reed, Modelling of the influence of alloy composition on flow stress in high-strength nickel-based superalloys. Acta Mater. 75, 356–370 (2014).

[2018Liu] Y.-X. Liu, Y. . Lin, A Yield Stress Model for a Solution-Treated Ni-Based Superalloy during Plastic Deformation. High Temp. Mater. Process. 37, 849–856 (2018).

## Antiphase Boundary Energy Property Model Settings

The **Antiphase Boundary Energy - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates the antiphase boundary energy for the gamma prime ( $\gamma'$ ) phase in Ni-base alloys.



[About the Antiphase Boundary Energy Property Model](#)



For an example, see [PM\\_Ni\\_02: Antiphase Boundary Energy of  \$\gamma'\$](#)

### Configuration Settings

These settings are found on the Property Model Calculator when **Antiphase Boundary Energy - Ni** is selected under **Nickel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Freeze-in Temperature

Enter a **Freeze-in temperature** value in the field. The unit matches those selected under *Condition Definitions*. This is the temperature where the equilibrium is calculated.

The default is 623.15 K (350 °C).

### Subset of Phases

The **Subset of phases** setting allows you to select a subset of phases relevant to the material type under investigation. The purpose is to consider only phases relevant to typical commercial Ni-base alloys, for example it excludes the eta and delta phases that are normally thermodynamically stable at heat treatment temperatures but rarely form in practice due to their slow formation. Select one of these options to include the specified phases in the calculation:

- **All phases:** To include all phases in the current system.
- **Gamma and gamma prime only ( $\gamma$  and  $\gamma'$  only):** To include only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For the **Coarsening - Ni** Property Model, you select **Gamma and selected precipitate(s)** instead and it still includes only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For **Solvus for Ordered Phases - Ni** and **Strain-Age Cracking - Ni**, this option is **Gamma + Gamma Prime**.

- **Typical Ni-base superalloy:** To include phases typically present in Ni-base superalloys, i.e. liquid, gamma ( $\gamma$ ), gamma prime ( $\gamma'$ ), gamma double prime ( $\gamma''$ ) and carbides (FCC\_L12#3, HCP\_A3#1, HCP\_A3#2, M23C6, M6C, M7C3) along with some additional phases (BCC\_A2, BCC\_A2#2, BCC\_B2, BCC\_B2#2, BCT\_D022, NI3B\_D011, M2B\_TETR, D5A\_M3B2, M3B2, MB\_B33, MB2\_C32, G\_PHASE, NI5ZR, NI7ZR2, CR3NI5SI2, CR3NI5SI2, SPINEL, ALPHA\_SPINEL, CORUNDUM).
- **Typical Ni-base superalloy plus eta and delta:** To include the phases listed above for a **Typical Ni-base superalloy** plus eta ( $\eta$ ) (NI3TI\_D024) and delta ( $\delta$ ) (NI3TA\_D0A) phases.
- **Typical Ni-base superalloy plus TCP phases:** To include the phases listed above for a **Typical Ni-base superalloy** plus topologically close packed (TCP) phases. TCP phases are laves (C14\_LAVES), sigma ( $\sigma$ ) (SIGMA), mu ( $\mu$ ) (MU\_PHASE), R (R\_PHASE), P (P\_PHASE), Z (Z\_PHASE) and CR3SI (CR3SI\_A15).

## Equilibrium Minimization Strategy

This setting is not available if **Gamma and gamma-prime only** is selected as the **Subset of phases**.

Select an **Equilibrium minimization strategy**. The default uses the **Global test preferred** option. For this Property Model, and for either Global option, also enter the **Max number of global gridpoints**, where the default is 20000.

The **Minimization Strategy** setting is used to ensure that the most stable minimum under the specified conditions is computed.

With either the *Global test preferred* or *Local minimization preferred* setting, the program cycles through options before it gives up:

- For **Global test preferred** the minimization starts with a global test and if that fails it runs full global minimization.
- For **Local minimization preferred** the minimization starts with a local minimization, in case of failure it tries a global test and finally a full global minimization.

If you choose **Local minimization only** or **Global minimization only**, the program just tries the one setting and gives up if it fails.



For general information about global minimization, see the topic related to the Console Mode command, [GLOBAL\\_MINIMIZATION](#).

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities and then choose the unit Joule per m<sup>2</sup> or Millijoule per m<sup>2</sup>:

- **Temperature**
- **APBE for 111 plane (1/2)**: The antiphase boundary energy for the 111-plane for the first  $\gamma'$  phase.
- **APBE for 100 plane (1/2)**: The antiphase boundary energy for the 100-plane for the first  $\gamma'$  phase.
- **APBE average (1/2)**: The average of the antiphase boundary energy for the 100- and 111-plane for the first  $\gamma'$  phase.
- **APBE for 111 plane (2/2)**: The antiphase boundary energy for the 111-plane for the second  $\gamma'$  phase in the case more than one FCC\_L12 composition set is identified as  $\gamma'$ .
- **APBE for 100 plane (2/2)**: The antiphase boundary energy for the 100-plane for the second  $\gamma'$  phase in the case more than one FCC\_L12 composition set is identified as  $\gamma'$ .

- **APBE average (2/2):** The average of the antiphase boundary energy for the 100- and 111-plane for the first  $\gamma'$  phase in the case more than one FCC\_L12 composition set is identified as  $\gamma'$ .
- **Delta APBE rel.pure Ni3Al:** w.r.t. Ni3Al - The antiphase boundary energy of the 111-plane for the current  $\gamma'$  relative to 'pure' Ni3Al  $\gamma'$ .

---

## About the Coarsening Nickel Property Model

---

The **Coarsening - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates the coarsening rate coefficient  $K$  of one or several precipitate phases in a matrix phase, assuming spherical geometry of the precipitating phase (s).

This model is based on the **General Models → Coarsening** Property Model, the primary difference that with the **Nickel Models** version you are guided to the selection of phases during the set up.



See [About the Coarsening Property Model](#) for the background theory described for the **General Models** version.

## Coarsening Nickel Property Model Settings

The **Coarsening - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates the coarsening rate coefficient  $K$  of one or several precipitate phases in a matrix phase, assuming spherical geometry of the precipitating phase (s).

 [About the Coarsening Nickel Property Model](#)



This Property Model requires a kinetic database with mobilities for the matrix phase.

### Configuration Settings

The settings are found on the Property Model Calculator when **Coarsening - Ni** is selected under **Nickel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Subset of Phases

The **Subset of phases** setting allows you to select a subset of phases relevant to the material type under investigation. The purpose is to consider only phases relevant to typical commercial Ni-base alloys, for example it excludes the eta and delta phases that are normally thermodynamically stable at heat treatment temperatures but rarely form in practice due to their slow formation. Select one of these options to include the specified phases in the calculation:

- **All phases:** To include all phases in the current system.
- **Gamma and gamma prime only ( $\gamma$  and  $\gamma'$  only):** To include only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For the **Coarsening - Ni** Property Model, you select **Gamma and selected precipitate(s)** instead and it still includes only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For **Solvus for Ordered Phases - Ni** and **Strain-Age Cracking - Ni**, this option is **Gamma + Gamma Prime**.

- **Typical Ni-base superalloy:** To include phases typically present in Ni-base superalloys, i.e. liquid, gamma ( $\gamma$ ), gamma prime ( $\gamma'$ ), gamma double prime ( $\gamma''$ ) and carbides (FCC\_L12#3, HCP\_A3#1, HCP\_A3#2, M23C6, M6C, M7C3) along with some additional phases (BCC\_A2, BCC\_A2#2, BCC\_B2, BCC\_B2#2, BCT\_D022, NI3B\_D011, M2B\_TETR, D5A\_M3B2, M3B2, MB\_B33, MB2\_C32, G\_PHASE, NI5ZR, NI7ZR2, CR3NI5SI2, CR3NI5SI2, SPINEL, ALPHA\_SPINEL, CORUNDUM).
- **Typical Ni-base superalloy plus eta and delta:** To include the phases listed above for a **Typical Ni-base superalloy** plus eta ( $\eta$ ) (NI3TI\_D024) and delta ( $\delta$ ) (NI3TA\_D0A) phases.
- **Typical Ni-base superalloy plus TCP phases:** To include the phases listed above for a **Typical Ni-base superalloy** plus topologically close packed (TCP) phases. TCP phases are laves (C14\_LAVES), sigma ( $\sigma$ ) (SIGMA), mu ( $\mu$ ) (MU\_PHASE), R (R\_PHASE), P (P\_PHASE), Z (Z\_PHASE) and CR3SI (CR3SI\_A15).

## Equilibrium Minimization Strategy

This setting is NOT available if **Gamma and selected precipitate(s)** is selected.

Select an **Equilibrium minimization strategy**. The default uses the **Global test preferred** option. For this Property Model, and for either Global option, also enter the **Max number of global gridpoints**, where the default is 20000.

The **Minimization Strategy** setting is used to ensure that the most stable minimum under the specified conditions is computed.

With either the *Global test preferred* or *Local minimization preferred* setting, the program cycles through options before it gives up:

- For **Global test preferred** the minimization starts with a global test and if that fails it runs full global minimization.
- For **Local minimization preferred** the minimization starts with a local minimization, in case of failure it tries a global test and finally a full global minimization.

If you choose **Local minimization only** or **Global minimization only**, the program just tries the one setting and gives up if it fails.



For general information about global minimization, see the topic related to the Console Mode command, [GLOBAL\\_MINIMIZATION](#).

## Matrix Phase

Choose a **Matrix phase**. The default is **Gamma** ( $\gamma$ ).

## Precipitate Phase

Choose a **Precipitate phase** to calculate its coarsening rate. Click the **Add phase**  $\oplus$  button to add another Precipitate phase. Click the **Remove phase**  $\ominus$  button to remove a phase.

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Temperature**
- **Coarsening rate coefficient Ni [m<sup>3</sup>/s]** for the precipitate phase.
- **Interfacial energy Ni [J/m<sup>2</sup>]** for it to be between the matrix and precipitate phase.

## About the Equilibrium with Freeze-in Temperature Nickel Property Model

---

The **Equilibrium with Freeze-in Temperature - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates equilibrium at the freeze-in temperature and evaluates the properties at a different temperature.

This Property Model is based on the **General Models** version but specifically implemented for use with nickel alloys.

The assumption is that diffusion and phase transformations are negligible when changing from the freeze-in-temperature and, therefore, that the phase amounts and compositions of phases are kept at the evaluation temperatures.

Typically, the freeze-in occurs during cooling where equilibrium can be assumed above the freeze-in temperature. This is the default setting and the **Equilibrium above freeze-in temperature** checkbox is automatically selected for this model. Change this default setting (i.e. click to clear the checkbox) when the frozen structure should be evaluated at a higher temperature than the freeze-in temperature.

The electrical resistivity due to phase interface scattering is evaluated as the scattering constant times sum of the interaction between the volume fraction of all the phases. The default value for the constant is  $4.0e-8 \Omega m$ . The contribution to thermal conductivity is assumed to be related to that of electrical resistivity, following the Wiedemann-Franz law.

A selection of homogenization functions is available for the evaluation of thermal- and electric-properties. Thermal and electric properties depend on the microstructure and the geometrical representation of phases in the microstructure can be linked to different homogenization functions. The homogenization function is applied on thermal- and electrical-resistivity. The electric conductivity, thermal conductivity, and thermal diffusivity are evaluated from the resistivities.



For an example see [PM\\_Ni\\_01: Lattice Parameter of  \$\gamma/\gamma'\$](#) .



[Equilibrium with Freeze-in Temperature Nickel Property Model Settings](#)



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).

## References

- [1985Nat] M. V. Nathal, R. A. Mackay, R. G. Garlick, Temperature dependence of  $\gamma$ - $\gamma'$  lattice mismatch in Nickel-base superalloys. *Mater. Sci. Eng.* 75, 195–205 (1985).
- [2019Goo] A. J. Goodfellow, L. R. Owen, K. A. Christofidou, J. Kelleher, M. C. Hardy, H. J. Stone, The Effect of Temperature and Mo Content on the Lattice Misfit of Model Ni-Based Superalloys. *Metals (Basel)*. 9, 700 (2019).

## Equilibrium with Freeze-in Temperature Nickel Property Model Settings

The **Equilibrium with Freeze-in Temperature - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates equilibrium at the freeze-in temperature and evaluates the properties at a different temperature.



[About the Equilibrium with Freeze-in Temperature Nickel Property Model](#)



For an example, see [PM\\_Ni\\_01: Lattice Parameter of  \$\gamma/\gamma'\$](#)

### Configuration Settings

The settings are found on the [Property Model Calculator](#) when the **Equilibrium with Freeze-in Temperature - Ni** Model is selected under **Nickel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Freeze-in Temperature

Enter a **Freeze-in temperature** value in the field. The unit matches those selected under *Condition Definitions*. This is the temperature where the equilibrium is calculated.

The default is 1273.15 K (1000 °C).

### Equilibrium Above Freeze-in Temperature

The **Equilibrium above freeze-in temperature** checkbox is selected by default and it calculates equilibrium above freeze-in temperature. Click to clear the checkbox when evaluating the frozen-in structure at higher temperatures than the freeze-in temperature.

## Subset of Phases

The **Subset of phases** setting allows you to select a subset of phases relevant to the material type under investigation. The purpose is to consider only phases relevant to typical commercial Ni-base alloys, for example it excludes the eta and delta phases that are normally thermodynamically stable at heat treatment temperatures but rarely form in practice due to their slow formation. Select one of these options to include the specified phases in the calculation:

- **All phases:** To include all phases in the current system.
- **Gamma and gamma prime only ( $\gamma$  and  $\gamma'$  only):** To include only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For the **Coarsening - Ni** Property Model, you select **Gamma and selected precipitate(s)** instead and it still includes only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For **Solvus for Ordered Phases - Ni** and **Strain-Age Cracking - Ni**, this option is **Gamma + Gamma Prime**.

- **Typical Ni-base superalloy:** To include phases typically present in Ni-base superalloys, i.e. liquid, gamma ( $\gamma$ ), gamma prime ( $\gamma'$ ), gamma double prime ( $\gamma''$ ) and carbides (FCC\_L12#3, HCP\_A3#1, HCP\_A3#2, M23C6, M6C, M7C3) along with some additional phases (BCC\_A2, BCC\_A2#2, BCC\_B2, BCC\_B2#2, BCT\_D022, NI3B\_D011, M2B\_TETR, D5A\_M3B2, M3B2, MB\_B33, MB2\_C32, G\_PHASE, NI5ZR, NI7ZR2, CR3NI5SI2, CR3NI5SI2, SPINEL, ALPHA\_SPINEL, CORUNDUM).
- **Typical Ni-base superalloy plus eta and delta:** To include the phases listed above for a **Typical Ni-base superalloy** plus eta ( $\eta$ ) (NI3TI\_D024) and delta ( $\delta$ ) (NI3TA\_D0A) phases.
- **Typical Ni-base superalloy plus TCP phases:** To include the phases listed above for a **Typical Ni-base superalloy** plus topologically close packed (TCP) phases. TCP phases are laves (C14\_LAVES), sigma ( $\sigma$ ) (SIGMA), mu ( $\mu$ ) (MU\_PHASE), R (R\_PHASE), P (P\_PHASE), Z (Z\_PHASE) and CR3SI (CR3SI\_A15).

---

## Evaluate for a Single Phase Only

Select the **Evaluate for a single phase only** checkbox to evaluate the properties of a specific phase or deselect to evaluate the properties for the entire system.

## Phase for Evaluation

This setting is available when the **Evaluate for a single phase only** checkbox is selected.

From the **Phase for evaluation** list, select any available phase or **None**.

## Equilibrium Minimization Strategy

This setting is NOT available if **Gamma and selected precipitate(s)** is selected.

Select an **Equilibrium minimization strategy**. The default uses the **Global test preferred** option. For this Property Model, and for either Global option, also enter the **Max number of global gridpoints**, where the default is 20000.

The **Minimization Strategy** setting is used to ensure that the most stable minimum under the specified conditions is computed.

With either the *Global test preferred* or *Local minimization preferred* setting, the program cycles through options before it gives up:

- For **Global test preferred** the minimization starts with a global test and if that fails it runs full global minimization.
- For **Local minimization preferred** the minimization starts with a local minimization, in case of failure it tries a global test and finally a full global minimization.

If you choose **Local minimization only** or **Global minimization only**, the program just tries the one setting and gives up if it fails.



For general information about global minimization, see the topic related to the Console Mode command, [GLOBAL\\_MINIMIZATION](#).

---

## Homogenization Function

The **Homogenization function** is used for the evaluation of the systems thermal and electric properties. The function is applied on thermal- and electrical-resistivity. The electric conductivity, thermal conductivity, and thermal diffusivity are evaluated from the resistivities.

The geometrical interpretation of the Hashin-Shtrikman bounds are concentric spherical shells of each phase.

- **Rule of mixtures (upper Wiener bound):** the geometrical interpretation are continuous layers of each phase parallel with the direction of evaluation of the property.
- **Inverse rule of mixtures (lower Wiener bound):** The geometrical interpretation are continuous layers of each phase orthogonal to the direction of evaluation of the property.
- **General lower Hashin-Shtrikman bound:** The outermost shell consists of the phase with the most sluggish property.
- **General upper Hashin-Shtrikman bound:** The innermost shell consists of the phase with the most sluggish property.
- **Hashin-Shtrikman bound with majority phase as matrix phase:** The outermost shell consists of the phase with the highest local volume fraction.

## Account for Phase Interface Scattering

The **Account for phase interface scattering** checkbox is selected by default.

The electrical resistivity due to phase interface scattering is evaluated as the scattering constant times sum of the interaction between the volume fraction of all the phases. The default value for the constant is  $4.0e-8 \Omega m$ . The contribution to thermal conductivity is assumed to be related to that of electrical resistivity, following the Wiedemann-Franz law.

## Phase Interface Scattering Constant

This setting is available when the **Account for phase interface scattering** checkbox is selected.

The **Phase interface scattering constant** default value is  $4.0e-8 \Omega m$ .

## Set Reference Temperature for Technical CTE

This setting is available when the **Account for phase interface scattering** checkbox is selected.

In the field, enter the **Reference temperature for technical CTE**, where CTE is the coefficient of thermal expansion. Typically room temperature is used as the default 293.15 K (20 °C).

## Define User Functions

Select the **Define user functions** checkbox to enter up to two **User-defined functions** using the Console Mode syntax.



For an example of user-defined functions, see [T\\_07: User-Defined Functions](#).



Dot derivatives (e.g. Hm.T for heat capacity) should NOT be used since these are not consistent with freezing in the amount of phases.

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Temperature**
- **Lattice parameter Gamma** ( $\gamma$ ) ( $a_p$ ): Choose a unit Meter, Micrometer, Nanometer, or Ångström.
- **Lattice parameter Gamma-Prime** ( $\gamma'$ ) ( $a_{gp}$ ): Choose a unit Meter, Micrometer, Nanometer, or Ångström.
- **Lattice mismatch Gamma/Gamma-Prime** ( $\gamma/\gamma'$ ): This is an unconstrained misfit between Gamma/Gamma-prime lattice parameters:

$$100 * \frac{2*(a_{gp}-a_g)}{(a_{gp}+a_g)}$$

- **Electric resistivity (ohm m)** ( $\Omega m$ )
- **Electric conductivity (S/m)**

- **Thermal conductivity (W/(mK))**
- **Thermal resistivity (mK/W)**
- **Thermal diffusivity (m<sup>2</sup>/s)**
- **Heat capacity (J/mol K)**
- **Volume (m<sup>3</sup>/mol)**
- **Density (g/cm<sup>3</sup>)**
- **Linear CTE (technical) (1/K):** Fractional change of length per unit temperature change. Calculated as  $(L1-L0)/(T1-T0)/L0$ , where  $L0$  is the length at a reference temperature  $T0$ .  $T0$  is by default equal to room temperature. Assumed Isotropic material.
- **Linear CTE (physical) (1/K):** Fractional change of length per unit temperature change. Assumed Isotropic material.
- **Volumetric CTE (physical) (1/K):** Fractional change of volume per unit temperature change.
- **User-defined function and 2nd user-defined function:** Available when the **Define user functions** checkbox is selected and the functions are defined.

## About the Solvus for Ordered Phase Property Model

The **Solvus for Ordered Phase - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, calculates the solvus of ordered phases such as FCC\_L12, BCC\_B2, NI3TA\_D0A, NI3TI\_D024, TCP-phases (Sigma), and so forth.

The Model chooses the correct composition set by minimizing the Gibbs energy, which means that the correct composition set is automatically detected by the software for you.

The remarkable properties of Ni-base superalloys often rely on the formation of finely dispersed ordered phases. In order to obtain the correct microstructure, and therefore also the properties you want, it is necessary to heat treat the alloy accordingly. One important aspect of the heat treatment is the solvus temperature for the phase or phases that is expected and wanted in the microstructure. An appropriate heat treatment aims to dissolve the ordered phases in the as-cast or as-printed structure, which must be done above the respective phases' solvus temperature.

By combining the **Solvus for Ordered Phase - Ni** model with the latest TCS Ni-based Superalloys Database (TCNI) version, you can predict the solvus temperature for any ordered phase, provided that it is present in the database and that it is stable for the given alloy chemistry. As such, appropriate heat treatments may readily be designed with guidance from thermodynamic calculations, which efficiently cuts down on time spent using trial-and-error approaches.



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).



For an example, see [PM\\_Ni\\_03: Critical Temperatures of Alloy 718](#).



The input parameters are entered on the **Configuration** window for the Property Model Calculator. There are also additional axis variables available on the Plot Renderer. See [Solvus for Ordered Phase Property Model Settings](#) for details.

## Solvus for Ordered Phase Property Model Settings



[About the Solvus for Ordered Phase Property Model](#)

## Configuration Settings

The settings are found on the Property Model Calculator when **Solvus for Ordered Phase- Ni** is selected under **Nickel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

## Naming Convention

Select **Conventional phase names** (the default) or **Explicit phase names**.

Select **Conventional phase names** to limit the list to commonly occurring ordered phases, which are **Gamma prime (FCC\_L12)**, **Gamma double prime (BCT\_D022)**, **Eta (NI3TI\_D024)**, **Delta (NI3TA\_D0A)**, and **Sigma (SIGMA)**. You can alternatively select a phase from the full list of **Explicit phase names**; phases available in the list are based on the selected database.

## Phase to Calculate Solvus For

From the **Phase to calculate solvus for** list, choose a phase. When **Conventional phase names** is selected **Gamma prime (FCC\_L12)** is the default.

Phases such as FCC\_L12 and BCC\_B2 normally exhibit ordering (when used with the TCS Ni-based Superalloys Database (TCNI)).

## Subset of Phases

The **Subset of phases** setting allows you to select a subset of phases relevant to the material type under investigation. The purpose is to consider only phases relevant to typical commercial Ni-base alloys, for example it excludes the eta and delta phases that are normally thermodynamically stable at heat treatment temperatures but rarely form in practice due to their slow formation. Select one of these options to include the specified phases in the calculation:

- **All phases:** To include all phases in the current system.
- **Gamma and gamma prime only ( $\gamma$  and  $\gamma'$  only):** To include only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For the **Coarsening - Ni** Property Model, you select **Gamma and selected precipitate(s)** instead and it still includes only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For **Solvus for Ordered Phases - Ni** and **Strain-Age Cracking - Ni**, this option is **Gamma + Gamma Prime**.

- **Typical Ni-base superalloy:** To include phases typically present in Ni-base superalloys, i.e. liquid, gamma ( $\gamma$ ), gamma prime ( $\gamma'$ ), gamma double prime ( $\gamma''$ ) and carbides (FCC\_L12#3, HCP\_A3#1, HCP\_A3#2, M23C6, M6C, M7C3) along with some additional phases (BCC\_A2, BCC\_A2#2, BCC\_B2, BCC\_B2#2, BCT\_D022, NI3B\_D011, M2B\_TETR, D5A\_M3B2, M3B2, MB\_B33, MB2\_C32, G\_PHASE, NI5ZR, NI7ZR2, CR3NI5SI2, CR3NI5SI2, SPINEL, ALPHA\_SPINEL, CORUNDUM).
- **Typical Ni-base superalloy plus eta and delta:** To include the phases listed above for a **Typical Ni-base superalloy** plus eta ( $\eta$ ) (NI3TI\_D024) and delta ( $\delta$ ) (NI3TA\_D0A) phases.
- **Typical Ni-base superalloy plus TCP phases:** To include the phases listed above for a **Typical Ni-base superalloy** plus topologically close packed (TCP) phases. TCP phases are laves (C14\_LAVES), sigma ( $\sigma$ ) (SIGMA), mu ( $\mu$ ) (MU\_PHASE), R (R\_PHASE), P (P\_PHASE), Z (Z\_PHASE) and CR3SI (CR3SI\_A15).

### Advanced Settings

Click the **Advanced settings** checkbox to edit the following settings.

#### Upper and Lower Temperature Search Limits (K)

The search for the solvus temperature is performed between the upper and lower limits. Enter values for the **Upper temperature search limit (K)** (default is 2000 K) and **Lower temperature search limit (K)** (default is 500 K).

---

## First and Second Sublattices for Ordering

Depending on the phase model for the selected phase, different sets of sublattices can be used to model ordering. For example, the ordering of FCC\_L12 in the TCS Ni-based Superalloys Database (TCNI) occurs on sublattice 1 and 2. Enter a number for the **First sublattice for ordering** (default is 1) and **Second sublattice for ordering** (default is 2).

## Number of Decimals

Ordering reactions may be continuous and it is necessary to set a numerical precision. This is because the difference in site occupancy in ordered and disordered phases may be small. Enter a **Number of decimals** at which to truncate the site fractions (default is 7).

## Max Number of Iterations

Limits the number of iterations. Enter any value in the **Max. number of iterations** field that is greater than 0. The default is 100.

## *Plot Renderer Settings*



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Select from these plot quantities:

- **Composition**
- **Solvus temperature**

## About the Strain-Age Cracking Property Model

The **Strain-Age Cracking - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, is used in applications related to welding or additively manufactured components, where there are interface strains between  $\gamma'$  (gamma prime) and the matrix.



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).



For best results with this Property Model and the features it uses, it is recommended to use TCNI version 12 and newer.



The input parameters are entered on the **Configuration** window for the Property Model Calculator. There are also additional axis variables available on the Plot Renderer. See [Strain-Age Cracking Property Model Settings](#) for details.



For an example, see [PM\\_Ni\\_04: Strain Age Cracking \(SAC\)](#).

Strain-age cracking (SAC) may occur in welded or additively manufactured components due to interface strains between  $\gamma'$  (gamma prime) and the matrix. The interface strain may be quantified as a misfit parameter, i.e.

$$\delta = \frac{2(a_{\gamma'} - a_{\gamma})}{(a_{\gamma'} + a_{\gamma})}$$

where  $a$  is the lattice parameter of each respective phase ( $\gamma$  matrix or  $\gamma'$ ) in conjunction with residual thermal stresses from welding or additive manufacturing. Strain-age cracking may occur at negative values of the misfit parameter in combination with thermal stresses and the risk for cracking is also related to the  $\gamma'$  volume fraction.

Misfit strain naturally varies with temperature (thermal expansion) and phase composition (molar volume depends on composition and temperature) and therefore an accurate

assessment of the strain-age cracking risk must account for these variables. When defining this Model on the Configuration window, this is managed using either a **Single temperature** or **Temperature interval** mode.

## Single Temperature Mode

When using the **Single temperature** mode, the model evaluates the misfit parameter at the temperature and composition specified. By stepping in temperature or composition (or both), it is possible to get an idea of how the misfit varies. Misfit values calculated using this mode can be compared with experiment but it is necessary to remember that comparisons are only valid if experimental data are taken from fully equilibrated alloys.

## Temperature Interval Mode

The **Temperature interval** mode circumvents the issue of having to compare with fully equilibrated alloys. It provides a single parameter to describe the tendency for strain-age cracking. When using this mode, the Model searches for a temperature interval, between the lower limit (this is the **Temperature** set under *Condition Definitions*) and the upper limit (by default the calculated  $\gamma'$  solvus, which can also be changed by using the **Set maximum temperature** checkbox), where the misfit parameter becomes negative.

The misfit value, and the  $\gamma'$  volume fraction is then integrated over the interval where the misfit is negative and the risk value for strain-age cracking is thereafter calculated.

Values of the SAC risk factor close to or below 0 indicate low or no risk for strain-age cracking. Positive values indicate risk for cracking and the more positive the value, the higher the risk. The risk factor, as defined here, also incorporates all thermodynamically related factors for strain-age cracking and makes it possible to differentiate the SAC risk for various alloys using a single number for any given temperature interval.

## Strain-Age Cracking Property Model Settings

The **Strain-Age Cracking - Ni** Property Model, available with the Property Model Calculator and the Nickel Model Library, is used in applications related to welding or additively manufactured components, where there are interface strains between  $\gamma'$  ( $\gamma'$  prime) and the matrix.



[Strain-Age Cracking Property Model Settings](#)



For an example, see [PM\\_Ni\\_04: Strain Age Cracking \(SAC\)](#).

## Configuration Settings

These settings are found on the Property Model Calculator when **Strain-Age Cracking - Ni** is selected under **Nickel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

## Subset of Phases

The **Subset of phases** setting allows you to select a subset of phases relevant to the material type under investigation. The purpose is to consider only phases relevant to typical commercial Ni-base alloys, for example it excludes the eta and delta phases that are normally thermodynamically stable at heat treatment temperatures but rarely form in practice due to their slow formation. Select one of these options to include the specified phases in the calculation:

- **All phases:** To include all phases in the current system.
- **Gamma and gamma prime only ( $\gamma$  and  $\gamma'$  only):** To include only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For the **Coarsening - Ni** Property Model, you select **Gamma and selected precipitate(s)** instead and it still includes only disordered FCC ( $\gamma$ ) and L12 ordered FCC ( $\gamma'$ ).



For **Solvus for Ordered Phases - Ni** and **Strain-Age Cracking - Ni**, this option is **Gamma + Gamma Prime**.

- **Typical Ni-base superalloy:** To include phases typically present in Ni-base superalloys, i.e. liquid, gamma ( $\gamma$ ), gamma prime ( $\gamma'$ ), gamma double prime ( $\gamma''$ ) and carbides (FCC\_L12#3, HCP\_A3#1, HCP\_A3#2, M23C6, M6C, M7C3) along with some additional phases (BCC\_A2, BCC\_A2#2, BCC\_B2, BCC\_B2#2, BCT\_D022, NI3B\_D011, M2B\_TETR, D5A\_M3B2, M3B2, MB\_B33, MB2\_C32, G\_PHASE, NI5ZR, NI7ZR2, CR3NI5SI2, CR3NI5SI2, SPINEL, ALPHA\_SPINEL, CORUNDUM).
- **Typical Ni-base superalloy plus eta and delta:** To include the phases listed above for a **Typical Ni-base superalloy** plus eta ( $\eta$ ) (NI3TI\_D024) and delta ( $\delta$ ) (NI3TA\_D0A) phases.
- **Typical Ni-base superalloy plus TCP phases:** To include the phases listed above for a **Typical Ni-base superalloy** plus topologically close packed (TCP) phases. TCP phases are laves (C14\_LAVES), sigma ( $\sigma$ ) (SIGMA), mu ( $\mu$ ) (MU\_PHASE), R (R\_PHASE), P (P\_PHASE), Z (Z\_PHASE) and CR3SI (CR3SI\_A15).

## Choose Mode

The Strain-Age Cracking (SAC) risk can be evaluated for a single temperature or for a specified temperature interval.



For background information about these settings, see [Strain-Age Cracking Property Model Settings](#).

From the **Choose mode** list, select **Temperature interval** or **Single temperature**.

For **Temperature interval** the following limits are applied and defined with additional settings:

- The lower limit **Temperature** is set under *Condition Definitions*.
- The upper limit is by default the  $\gamma'$  solvus temperature. You can also select the **Set maximum temperature** checkbox to enter a user defined maximum temperature.

## Set Maximum Temperature

Available when **Temperature interval** is selected as the mode.

The Strain-Age Cracking (SAC) risk is evaluated between the temperature specified in conditions and the  $\gamma'$  solvus.

Select the **Set maximum temperature** checkbox if a lower maximum temperature than the  $\gamma'$  solvus is required and thus to skip the  $\gamma'$  solvus calculation.

## Maximum Temperature

Available when **Temperature interval** is selected as the mode and the **Set maximum temperature** checkbox is selected.

Limit the search for a negative misfit to the **Maximum temperature** entered in the field.



This should be below the  $\gamma'$  solvus temperature.

## Initial Step Size for Search

Available when **Temperature interval** is selected as the mode.

The search for a negative misfit parameter is initiated with the step size entered in the **Initial step size for search** field.



If convergence issues arise, it can be useful to adjust this parameter and increase (or decrease) the lower temperature limit.

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Depending on the set up, there are a variety of plot quantities available to choose from. For example, select from these plot quantities:

- **Temperature**
- **Composition**
- **SAC risk factor**
- **Solvus of FCC\_L12 (gamma prime)**
- **Misfit value at current temperature**

---

# Steel Model Library Property Models

---



[Property Model Calculator](#)

## In this section:

---

About the Steel Model Library Property Models .....	593
Recommended Composition Ranges for Steel Models .....	596
About the Bainite Property Model .....	599
About the CCT Diagram Property Model .....	603
About the Critical Transformation Temperatures Property Model .....	612
About the Ferrite Property Model .....	616
About the Martensite Fractions Property Model .....	622
About the Martensite Temperatures Property Model .....	625
About the Martensitic Steel Strength Property Model .....	628
About the Pearlite Property Model .....	637
About the TTT Diagram Property Model .....	642

## About the Steel Model Library Property Models

These **Steel Models** are available with the [Property Model Calculator](#). The following are available with the Steel Model Library.



Also see [Steel Library CCT Diagram Template](#), [Steel Library TTT Diagram Template](#), [Plot Type: CCT Mode](#), and [Plot Type: TTT Mode](#).



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.



For the **Ferrite**, **Bainite**, and **Pearlite** Property Models, it is also useful to review the [Recommended Composition Ranges for Steel Models](#).

### **Bainite**

The **Bainite** Property Model describes the thermodynamics and isothermal kinetics of bainite transformation from austenite. In this Model, bainite is modeled as ferrite or a ferrite-cementite mixture.

See [About the Bainite Property Model](#) and [Bainite Property Model Settings](#) for background theory and input parameter details.

### **CCT Diagram**

The **CCT Diagram** Property Model calculates the concurrent transformations to ferrite, pearlite, bainite, and martensite under continuous cooling conditions and is used to construct a continuous-cooling-transformation (CCT) diagram.

See [About the CCT Diagram Property Model](#) and [CCT Diagram Property Model Settings](#) for background theory and input parameter details.

---

## ***Critical Transformation Temperatures***

The **Critical Transformation Temperatures** Property Model calculates critical transformation temperatures for steels.

See [About the Critical Transformation Temperatures Property Model](#) and [Critical Transformation Temperatures Property Model Settings](#) for background theory and input parameter details.

## ***Ferrite***

The **Ferrite** Property Model calculates the thermodynamics and isothermal kinetics of proeutectoid ferrite forming from austenite in hypoeutectoid steels.

See [About the Ferrite Property Model](#) and [Ferrite Property Model Settings](#) for background theory and input parameter details.

## ***Martensite Fractions***

The **Martensite Fractions** Property Model calculates the fraction of athermal martensite based on available driving force.

See [About the Martensite Fractions Property Model](#) and [Martensite Fractions Property Model Settings](#) for background theory and input parameter details.

## ***Martensite Temperatures***

The **Martensite Temperatures** Property Model calculates the martensite start (Ms) temperature based on modeling of the transformation barrier. Martensite half and finish temperatures are also given according to the Martensite Fractions Property Model.

See [About the Martensite Temperatures Property Model](#) and [Martensite Temperatures Property Model Settings](#) for background theory and input parameter details.

## ***Martensitic Steel Strength***

The **Martensitic Steel Strength** Property Model calculates the strength and/or hardness of martensitic steels.

See [About the Martensitic Steel Strength Property Model](#) and [Martensitic Steel Strength Property Model Settings](#) for background theory and input parameter details.

## ***Pearlite***

The **Pearlite** Property Model calculates the thermodynamics and kinetics of pearlite formation from austenite during isothermal heat treatment.

See [About the Pearlite Property Model](#) and [Pearlite Property Model Settings](#) for background theory and input parameter details.

## ***TTT Diagram***

The **TTT Diagram** Property Model calculates the concurrent transformations to ferrite, pearlite, bainite, and martensite under isothermal conditions and is used to construct a time-temperature-transformation (TTT) diagram.

See [About the TTT Diagram Property Model](#) and [TTT Diagram Property Model Settings](#) for background theory and input parameter details.

---

## Recommended Composition Ranges for Steel Models

---

The Steel Property Models (a.k.a. Steel Models or just Models) included with the Steel Model Library in Thermo-Calc calculate the process of austenite decomposition into ferrite, pearlite, bainite, and martensite. These Models are developed to satisfy the needs for a wide range of steels and are built on the thermodynamic and kinetic descriptions in the TCS Steel and Fe-alloys Database (TCFE) and TCS Steels/Fe-Alloys Mobility Database (MOBFE) databases, which are widely validated. However, the Steel Models also include their own modeling approach and parameters, which may limit the applicability to certain ranges of compositions and temperature, often due to some underlying assumptions of the modeling approaches and/or availability of calibration data.

In general, the Models for ferrite, bainite, and pearlite are for low-alloy, low-to-medium carbon steels involving austenite (Fe-rich disordered FCC\_A1) transforming into ferrite (Fe-rich disordered BCC\_A2) and cementite. For steels with higher Cr contents, it is also possible to select M7C3 and M23C6 to be the carbide in pearlite, and M7C3 for bainite. Otherwise, the Models are not designed for other phase relations, for example:

- Austenite transforming into a mixture of ferrite and another type of carbide (e.g. kappa carbide for high-Mn high-Al alloys).
- Precipitation (e.g. FCC-Cu, FCC-carbides due to microalloying elements Ti, V, Nb) taking place concurrently while austenite is transforming into ferrite (and possibly cementite).

If some other phases coexist with austenite prior to its transformation, the Steel Models assume these do not participate in or affect the transformations by any means (e.g. by providing extra nucleation sites). The only effect of these phases is to determine the austenite composition (which is now different from the nominal composition). In this case, the setting **Equilibrium composition at austenitizing temperature** is selected from the **Austenite composition from** list on the **Configuration** window.



For an explanation of **Austenite composition from** see the individual configuration settings for the different Steel Models (**Ferrite**, **Bainite**, and **Pearlite**). Links to these are listed in [About the Steel Model Library Property Models](#).

Ferrite and bainite calculations rely on carbon partitioning from ferrite to austenite. Therefore, the Models are *not* suitable for the ferrite transformed from austenite in a "massive" manner (composition invariant), in some "interstitial-free" ferrous alloys with very low carbon contents.

Pearlite is assumed to be steady-state, i.e. growth rate and lamellar spacing are constant over time under isothermal conditions. This is reasonable if composition and transformation temperature lie in the ferrite-cementite two-phase region of a phase diagram. The Pearlite Model does *not* cover the so-called "divergent pearlite", or "partitioned pearlite", which usually forms in the austenite-ferrite-cementite three-phase region with growth rate and lamellar spacing changing over time. Also, pearlite nucleation is modeled to be on austenite grain boundaries only. In practice, however, pearlite nucleation at grain edges and corners may also be considerable if the undercooling is small.



Parameters within the Steel Models are calibrated against experimental data on nucleation, growth, and overall transformation kinetics. However, the composition space covered by the calibration data, on which the Steel Models calculations are checked, is limited. As a result, together with the limitations from modeling approach discussed above, some recommended composition ranges are listed below. The Models may not give reasonable results when the composition in question is far outside the recommended ranges.

Element	Ferrite	Bainite		Pearlite <sup>3</sup>
		Bainite Start Temperature [2018Lea] <sup>1</sup>	Bainite Growth, Nucleation, Transformation Kinetics <sup>2</sup>	
C	Max. 0.7	Max. 0.7	0.2-1.5	0.6-1
Mn	Max. 3	Max. 5.5	Max. 0.6	Max. 1.8
Ni	Max. 5	Max. 9	Max. 5.3	Max. 3
Si	Max. 2	Max. 2	Max. 1	Max. 2
Al	Max. 1.4	–	–	(unknown, assuming similar to Si)
Co	Max. 3.2	–	–	Max. 2.2
Cr	Max. 3	Max. 5.7	Max. 5	Max. 13 <sup>[4]</sup>
Mo	Max. 2	Max. 4	(unknown)	Max. 0.6

Element	Ferrite	Bainite		Pearlite <sup>3</sup>
		Bainite Start Temperature [2018Lea] <sup>1</sup>	Bainite Growth, Nucleation, Transformation Kinetics <sup>2</sup>	
W	–	–	–	Max. 0.37

<sup>1</sup> The model is from [2018Lea]. The supporting bainite start data is compiled and published in [2019Lea].

<sup>2</sup> Parameters are not assessed to be composition dependent. The listed composition ranges come from the data of growth considered in a “universal” fitting.

<sup>3</sup> References in [2020Yan].

[4] Requires calculation setting **Custom** for type of carbide. Estimated boundaries: CEMENTITE for <5 wt.% Cr, M7C3 for 5~10 wt. % Cr, M23C6 for >10 wt.% Cr.

## References

[2018Lea] L. Leach, P. Kolmskog, L. Höglund, M. Hillert, A. Borgenstam, Critical Driving Forces for Formation of Bainite. Metall. Mater. Trans. A. 49, 4509–4520 (2018).

[2019Lea] L. Leach, P. Kolmskog, L. Höglund, M. Hillert, A. Borgenstam, Use of Fe-C Information as Reference for Alloying Effects on BS. Metall. Mater. Trans. A. 50, 4531–4540 (2019).

[2020Yan] J.-Y. Yan, J. Ågren, J. Jeppsson, Pearlite in Multicomponent Steels: Phenomenological Steady-State Modeling. Metall. Mater. Trans. A. 51, 1978–2001 (2020).

---

## About the Bainite Property Model

---

The **Bainite** Property Model, available with the Property Model Calculator and the Steel Model Library, describes the thermodynamics and isothermal kinetics of bainite transformation from austenite. In this Model, bainite is modeled as ferrite or a ferrite-cementite mixture.

For a given austenite composition, the bainite start temperature,  $WB_s$ , is calculated following Leach et al. [2018Lea] (see Eqs. 8 and 9 and Table 1 in the paper). At  $WB_s$ , the driving force for ferrite to precipitate out of austenite under the “para” condition (i.e. no partitioning of substitutional elements, carbon in equilibrium) is balanced by a barrier, which is modeled as a function of composition.

At a temperature below  $WB_s$ , there is a net driving force (over the barrier) for the growth of bainitic ferrite. The steady-state lengthening rate of ferrite plate is calculated based on the method from Leach et al. [2019Lea] with modifications. In the Bainite Model, unlike in [2019Lea], local equilibrium of carbon at the ferrite/austenite is sought, and a finite interfacial friction term is added to the dissipation terms. Otherwise the two methods are both based on a driving-force–dissipation balance and the Zener-Hillert equation, with a unique lengthening rate determined by maximizing it with respect to radius of curvature at the plate tip.

The Bainite Model also considers the case where cementite (under the “para” condition) can precipitate on the ferrite/austenite interface and grow into ferrite, forming a ferrite-cementite mixture. The formation of para-cementite usually accelerates lengthening, by enlarging the driving force of transformation while reducing carbon partitioning from bainite to austenite. Carbon partitioning to austenite vanishes if para-cementite forms in bainite to its maximum fraction allowed by mass balance.

Thickening of bainitic plates is usually much slower than lengthening. In the Bainite Model, it is assumed that whenever there is a positive driving force of precipitating para-cementite on the ferrite/austenite interface, the final state of the bainite transformation should be a mixture of ferrite and cementite, with austenite being completely transformed. Otherwise, the final state is a ferrite-austenite mixture with no cementite but a finite fraction of retained austenite (incomplete transformation).

The overall kinetics of bainite transformation is calculated in the extended-volume approach devised by Cahn [1956Cah] and generalized for ellipsoids. Both grain boundary (GB) nucleation and volume nucleation are considered, the former being very temperature-sensitive, the latter temperature-insensitive. This is used to model the kinks at about 350 °C in the TTT diagrams of some medium- and high-carbon steels—At higher temperatures GB nucleation is predominant, whereas at lower temperatures it is effectively suppressed, and volume nucleation becomes significant.



For an example of the Bainite Model, see [PM\\_Fe\\_05: Fe-C-Mn-Si-Ni-Cr-Mo Bainite](#)



The input parameters are entered on the **Configuration** window for the Property Model Calculator. There are also additional axis variables available on the Plot Renderer. See [Bainite Property Model Settings](#) for details.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

## References

- [1956Cah] J. W. Cahn, “The kinetics of grain boundary nucleated reactions”, Acta Metallurgica 4, 449–459 (1956).
- [2018Lea] L. Leach, P. Kolmskog, L. Höglund, M. Hillert, and A. Borgenstam, “Critical driving forces for formation of bainite”, Metallurgical and Materials Transactions A 49, 4509–4520 (2018).
- [2019Lea] L. Leach, J. Ågren, L. Höglund, and A. Borgenstam, “Diffusion-controlled lengthening rates of bainitic ferrite a part of the Steel Genome”, Metallurgical and Materials Transactions A 50, 2613–2618 (2019)

## Bainite Property Model Settings

The **Bainite** Property Model, available with the Property Model Calculator and the Steel Model Library, describes the thermodynamics and isothermal kinetics of bainite transformation from austenite. In this Model, bainite is modeled as ferrite or a ferrite-cementite mixture.



Considered elements: Fe, C, Mn, Si, Cr, Ni, Mo. Other elements in the system are neglected for bainite by mass percent.



For the **Ferrite**, **Bainite**, and **Pearlite** Property Models, it is also useful to review the [Recommended Composition Ranges for Steel Models](#).



[About the Bainite Property Model](#)



For an example, see [PM\\_Fe\\_05: Fe-C-Mn-Si-Ni-Cr-Mo Bainite](#).

### Configuration Settings

These settings are found on the Property Model Calculator when **Bainite** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Austenite Composition From

Select an option from the **Austenite composition from** list:

- **Nominal composition** uses the system composition as austenite composition.
- **Equilibrium composition at austenizing temperature.** Select to enter an **Austenitizing temperature**. The austenite composition is determined from an equilibrium calculation at the austenitizing temperature (if austenite is stable). This option is useful when austenite coexists with other phase(s), e.g. primary carbide, before the transformation(s) being calculated starts. But the Model only considers the effect of other phase(s) on austenite composition.

## Austenitizing Temperature

This setting is available when **Equilibrium composition at austenitizing temperature** is selected from the **Austenite composition from** list.

Enter an **Austenitizing temperature** in the field. The unit is the same as that selected under *Condition Definitions*.

## Grain Size

Enter the **Grain size** of the austenite in micrometers ( $\mu\text{m}$ ). The default is 100  $\mu\text{m}$ .

## Plot Renderer Settings



[Plot Renderer](#), [Configuration Settings](#), and [Plot Type: TTT Mode](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

These plot quantities are available:

- **Temperature**
- **Start time (2% bainite)**
- **Half time (50% bainite)**
- **Finish time (98% bainite)**
- **Bainite start temperature**
- **Lengthening rate [m/s]**
- **Cementite fraction in bainite**
- **Retained austenite fraction**

## About the CCT Diagram Property Model

The **CCT Diagram** Property Model, available with the Property Model Calculator and the Steel Model Library, can be used to calculate a Continuous-Cooling-Transformation (CCT) diagram, with a **One Axis** calculation on **Log10(cooling rate[K/s])**. This Model calculates the kinetics of concurrent transformation of austenite into ferrite, pearlite, bainite, and martensite. The algorithms and parameters for ferrite, pearlite, bainite, and martensite are the same as in their individual Property Models (**Ferrite**, **Pearlite**, **Bainite**, **Martensite Temperatures**, and **Martensite Fractions**), but the competitive occupation of prior austenite grain boundaries and bulk volume is handled in the **CCT Diagram** Property Model.



For an example, see [PM\\_Fe\\_08: CCT Diagram for Fe-C-Mn-Si-Cr-V](#)



The input parameters are entered on the **Configuration** window for the Property Model Calculator. There are also additional axis variables available on the Plot Renderer.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

## CCT Diagram Property Model Settings

The **CCT Diagram** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the concurrent transformations to ferrite, pearlite, bainite, and martensite under continuous cooling conditions and is used to construct a continuous-cooling-transformation (CCT) diagram.



[About the CCT Diagram Property Model](#) and [Steel Library CCT Diagram Template](#)



For an example, see [PM\\_Fe\\_08: CCT Diagram for Fe-C-Mn-Si-Cr-V](#)

## Configuration Settings

These settings are found on the Property Model Calculator when **CCT Diagram** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

## Austenite Composition From

Select an option from the **Austenite composition from** list:

- **Nominal composition** uses the system composition as austenite composition.
- **Equilibrium composition at austenizing temperature**. Select to enter an **Austenitizing temperature**. The austenite composition is determined from an equilibrium calculation at the austenitizing temperature (if austenite is stable). This option is useful when austenite coexists with other phase(s), e.g. primary carbide, before the transformation(s) being calculated starts. But the Model only considers the effect of other phase(s) on austenite composition.

## Austenitizing Temperature

This setting is available when **Equilibrium composition at austenitizing temperature** is selected from the **Austenite composition from** list.

Enter an **Austenitizing temperature** in the field. The unit is the same as that selected under *Condition Definitions*.

## Grain Size

Enter the **Grain size** of the austenite in micrometers ( $\mu\text{m}$ ). The default is 100  $\mu\text{m}$ .

## Start Temperature

Enter a **Start temperature**, which is the temperature at time zero where cooling starts.

## End Temperature

Enter an **End temperature**, which is the lowest possible temperature where cooling stops. Cooling may also stop before reaching this temperature if austenite has fully transformed (>98%).

## Log10(cooling rate [K/s])

Enter a **Log10(cooling rate [K/s])**, which is the 10-base logarithm of cooling rate in K/s.



The unit must be Kelvin per second, and it does NOT follow the unit selection in the GUI. This is the *Axis Quantity* if a CCT diagram is to be constructed.

## Calculation Setting

Select **Quick** (the default), **Accurate**, or **Custom** to balance accuracy and calculation time. Choose **Custom** to define the numerical settings yourself, which are otherwise set in the background when **Quick** or **Accurate** is selected. See the separate section below about the details for the **Custom** settings.

## Quick Calculation Setting

**Quick** is used for a relatively quick and rough calculation to reveal the general feature of a CCT diagram. Interpolation is used when necessary, in order to avoid calculating the growth rates and nucleation rates of pearlite and bainite at every time step. For simplicity, **Ferrite mode** is **PE** (paraequilibrium), and **Pearlite mode** is **Para-pearlite**.



Calculation times may vary between 30 minutes and several hours.

## Accurate Calculation Setting

**Accurate** evaluates the growth rates and nucleation rates at every time step accurately (without interpolation). The time step control is also tighter than when using the **Quick** setting.

- **Ferrite mode** is **PE if possible, otherwise OE** so as to handle any ortho-para transition.
- **Pearlite mode** is **Optimal pearlite**, but its rate calculation becomes slower if austenite contains many alloying elements.



Calculation times may vary between several hours and tens of hours.

## Details about the Quick and Accurate Calculation Settings

When you choose **Quick** or **Accurate** as the calculation setting, the following is what is automatically set in the background for each of the sections. These sections are visible to you and also can be edited when **Custom** is chosen:

<i>Setting</i>	<i>Quick</i>	<i>Accurate</i>
<b>Ferrite selected</b>	checkbox selected (yes)	checkbox selected (yes)
<b>Pearlite selected</b>	checkbox selected (yes)	checkbox selected (yes)
<b>Bainite selected</b>	checkbox selected (yes)	checkbox selected (yes)
<b>Martensite selected</b>	checkbox selected (yes)	checkbox selected (yes)
<b>Ferrite mode</b>	PE only	Faster start (TTT); PE if possible otherwise OE (CCT)
<b>Pearlite criterion</b>	Maximize growth rate	Maximize growth rate
<b>Pearlite mode</b>	Para-pearlite	Optimal pearlite
<b>Use interpolation when necessary</b>	checkbox selected (yes)	checkbox NOT selected (no)
<b>Interpolation error tolerance</b>	0.1	(does not apply)
<b>Maximum phase fraction change (absolute) in</b>	0.005	0.005

Setting	Quick	Accurate
a time step		
Maximum phase fraction change (relative) in a time step	0.05	0.05
Error tolerance for austenite fraction	1E-3	1E-4

### Custom Calculation Setting

The following settings are available when **Custom** is selected as the *Calculation Setting*. Otherwise these settings are handled in the background as described above for when **Quick** or **Accurate** is selected.

### Ferrite/Pearlite/Bainite/Martensite Selected Checkboxes

By default the **Ferrite selected**, **Pearlite selected**, **Bainite selected**, and **Martensite selected** checkboxes are selected. For ferrite and pearlite, additional settings are then available.

Click to deselect as required to exclude one or more from the calculation.

### Ferrite Mode

This setting is available when the **Ferrite selected** checkbox is selected.

From the **Ferrite mode** list choose an option.

- **OE only** and **PE only** are the same as in the **Ferrite Property Model**. i.e.
  - *Orthoequilibrium (OE)*: The ferrite and austenite compositions at the interface are fixed to the final equilibrium compositions
  - *Paraequilibrium (PE)*: Only carbon is allowed to partition, reaching PE at the interface
- **PE if possible, otherwise OE** means ferrite grows in PE mode if this mode has a positive driving force, otherwise ferrite grows in OE mode. This is usually useful to handle an ortho-to-para transition as temperature decreases.

### Pearlite Criterion

This setting is available when the **Pearlite selected** checkbox is selected.

Select an option from the **Criterion** list to determine the pearlite growth condition. This list is called **Pearlite criterion** for the CCT Diagram and TTT Diagram Property Models.

- For **Maximize growth rate**, pearlite growth rate is maximized with respect to lamellar spacing (and for optimal pearlite, partition coefficients of substitutional alloying elements between ferrite and cementite). The calculation gives maximal growth rate and minimal lamellar spacing. This is the default criterion.
- For **Maximize Gibbs energy dissipation rate**, Gibbs energy dissipation rate is maximized with respect to lamellar spacing (and for optimal pearlite, partition coefficients of substitutional alloying elements between ferrite and cementite). The calculation gives a smaller growth rate than the maximal, and a larger lamellar spacing than the minimal, which may better represent the average growth rate and lamellar spacing of all pearlite colonies in the material.

## Pearlite Mode

This setting is available when the **Pearlite selected** checkbox is selected. It is used to determine how substitutional alloying elements partition between ferrite and cementite in pearlite. Select an option from the **Pearlite mode** list.

- For **Optimal pearlite**, the partitioning of substitutional alloying elements between ferrite and cementite is optimized according to the **Criterion** while keeping carbon equilibrium between the two phases. This is the default mode. This mode realizes a smooth transition between ortho-pearlite and para-pearlite.
- For **Para-pearlite**, ferrite and cementite are in para-equilibrium where substitutional alloying elements do not partition and carbon reaches equilibrium.
- For **Ortho-pearlite**, ferrite and cementite are in ortho-equilibrium.

## Carbide in Pearlite

From the **Carbide in pearlite** list, select the carbide to be present in pearlite: **CEMENTITE** (default), **M7C3**, or **M23C6**. For M23C6, this usually depends on Cr content.



When M23C6 is selected, parameters optimized for stainless steels are invoked.



## Recommended Composition Ranges for Steel Models

### Carbide in Bainite

From the **Carbide in bainite** list, select the carbide to be present in pearlite: **CEMENTITE** (default) or **M7C3**. For M7C3, this usually depends on Cr content.

### Use Interpolation When Necessary

If the **Use interpolation when necessary** checkbox is selected, and depending on alloy composition and temperature, if the code determines that an interpolation is necessary and possible to implement, then a composition grid (TTT diagram) or composition-temperature grid (CCT diagram) is generated and refined as needed.

The growth rates and nucleation rates of pearlite and bainite, as functions of composition (TTT diagram) or composition and temperature (CCT diagram), are determined by interpolation (if sufficiently accurate). This is intended to avoid direct calculations of pearlite and bainite rates for every time step, which is usually time-consuming.

### Interpolation Error Tolerance

This setting is available when the **Use interpolation when necessary** checkbox is selected.

The growth and nucleation rates of pearlite and bainite, if determined from interpolation, are accepted only when the relative interpolation error estimated is below the value entered here. A large number implies few grid points necessary (shorter calculation time) and lower accuracy, and a small number implies the opposite.

### Maximum Phase Fraction Change (Absolute/Relative) in a Time Step

The **Maximum phase fraction change (absolute) in a time step** and **Maximum phase fraction change (relative) in a time step** settings are used to control time step size during time integration.

The phase fraction ( $X$ ) of ferrite, pearlite, and bainite cannot change more than  $atol + X * rtol$  during a time step, where  $atol$  and  $rtol$  are the absolute and relative maximum change, respectively. If the change is too large, the new time step is rejected, and time integration is retried with a smaller time step.

---

## Error Tolerance for Austenite Fraction

The **Error tolerance for austenite fraction** setting is used to control the accuracy for austenite fraction during time integration. Finite time steps cause an error in austenite volume fraction. When the error is larger than the tolerance, time integration is backed up and retried with a smaller time step.

### *Plot Renderer Settings*



[Plot Renderer](#), [Configuration Settings](#), [Plot Type: CCT Mode](#), and [Plot Type: TTT Mode](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Each quantity is defined in both time (**t**) and temperature (**T**):

- **Ferrite start (2%)**, time (**t**), and temperature (**T**)
- **Pearlite start (2%)**, time (**t**), and temperature (**T**)
- **Bainite start (2%)**, time (**t**), and temperature (**T**)
- **Total ferrite start (2%)**, time (**t**), and temperature (**T**): The start of the sum of polygonal ferrite and bainitic ferrite.
- **Total ferrite+cementite start (2%)**, time (**t**), and temperature (**T**): The start of the sum of pearlite and cementite-containing bainite.
- **Total ferrite + cementite finish (98%)**, time (**t**), and temperature (**T**): 98% austenite transformed to ferrite, pearlite, and bainite.
- **Martensite start**, time (**t**), and temperature (**T**)
- **Austenite transformed 2%**, time (**t**), and temperature (**T**)
- **Austenite transformed 10%**, time (**t**), and temperature (**T**)
- **Austenite transformed 50%**, time (**t**), and temperature (**T**)
- **Austenite transformed 90%**, time (**t**), and temperature (**T**)
- **Austenite transformed 98%**, time (**t**), and temperature (**T**)

Fractions of transformation products at the last time step (unit-free) are also available:

- **Terminal fraction of ferrite**
- **Terminal fraction of pearlite**
- **Terminal fraction of bainite**

- Terminal fraction of martensite
- Terminal fraction of total ferrite
- Terminal fraction of total ferrite+cementite



The **CCT Mode** on the Plot Renderer the time-temperature pairs are automatically identified and arranged in the plot to give a CCT diagram. Selected cooling curves are also given in the CCT Mode.

## About the Critical Transformation Temperatures Property Model

---

The **Critical Transformation Temperatures** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates critical transformation temperatures for steels under equilibrium conditions.

This Property Model can output the transformation temperatures for the following:

- Liquidus: First austenite or ferrite transformation from the liquid
- Solidus: Liquid fully transformed to solid
- A0: Magnetic transition temperature (Curie temperature) of cementite. The cementite is paramagnetic above A0 and ferromagnetic below
- A1: Austenite (FCC\_A1) transforms to ferrite (BCC\_A2) + carbide (cementite or graphite or M23C6)
- A2: Magnetic transition temperature (Curie temperature) of ferrite (BCC\_A2). Paramagnetic above A2 and ferromagnetic below
- A3: Austenite (FCC\_A1) transforms to ferrite (BCC\_A2)
- A4: Delta-ferrite (BCC\_A2) transforms to austenite (FCC\_A1)
- Acem: Austenite (FCC\_A1) transforms to cementite. Requires that the **Suspend graphite** checkbox is selected on the Configuration window, which it is by default.
- Agraph: Austenite (FCC\_A1) transforms to graphite. Requires that the **Suspend graphite** checkbox is NOT selected on the Configuration window, i.e. graphite is not suspended.



For an example, see [PM\\_Fe\\_04: Critical Temperatures](#).



The input parameters are entered on the Configuration window for the Property Model Calculator and described in [Critical Transformation Temperatures Property Model Settings](#).



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

## Critical Transformation Temperatures Property Model Settings

The **Critical Transformation Temperatures** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates critical transformation temperatures for steels under equilibrium conditions.



[About the Critical Transformation Temperatures Property Model](#)



For an example, see [PM\\_Fe\\_04: Critical Temperatures](#).



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

### Configuration Settings

These settings are found on the Property Model Calculator when **Critical Transformation Temperatures** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Calculate Solidus and Liquidus

Select the **Calculate solidus and liquidus** checkbox to calculate these temperatures.

---

## Calculate Ax Temperatures

Select the **Calculate Ax temperatures** checkbox to calculate A0, A1, A3, Acem, and Agraph temperatures. Typically, only one temperature of A3, Acem, and Agraph exists. See below for additional details about these temperatures.

- A0: Magnetic transition temperature (Curie temperature) of cementite. The cementite is paramagnetic above A0 and ferromagnetic below
- A1: Austenite (FCC\_A1) transforms to ferrite (BCC\_A2) + carbide (cementite or graphite or M23C6)
- A2: Magnetic transition temperature (Curie temperature) of ferrite (BCC\_A2). Paramagnetic above A2 and ferromagnetic below
- A3: Austenite (FCC\_A1) transforms to ferrite (BCC\_A2)
- A4: Delta-ferrite (BCC\_A2) transforms to austenite (FCC\_A1)
- Acem: Austenite (FCC\_A1) transforms to cementite (requires that the **Suspend graphite** checkbox is selected, which it is by default)
- Agraph: Austenite (FCC\_A1) transforms to graphite (requires that the **Suspend graphite** checkbox is NOT selected, i.e. graphite is not suspended)

## Suspend Graphite

By default the **Suspend graphite** checkbox is selected and suspends the graphite phase. Click to clear the checkbox as needed to enable the graphite.

## Suspend Gas

By default the **Suspend gas** checkbox is selected and suspends the gas phase. Click to clear the checkbox as needed.

## Manual Temperature Search Limits

By default the temperature search limits are automatically defined. Select the **Manual temperature search limits** checkbox to manually change the limits for the **Lower temperature search limit**, **Upper temperature search limit**, and **Max temperature step**. Is given in the same unit as the temperature in the **Condition Definitions**.

---

## Lower and Upper Temperature Search Limits

These settings are available when the **Manual temperature search limits** checkbox is selected.

In the **Lower temperature search limit** and **Upper temperature search limit** fields enter temperature values to search the lower and upper limits of transformation temperatures.

## Max Temperature Step

This setting is available when the **Manual temperature search limits** checkbox is selected.

In the **Max temperature step** field, enter the largest temperature step. This is most often reduced when changes in driving force or the amount of the phases indicate a phase transformation at a shorter temperature step.

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

Output the transformation temperatures:

- **Liquidus:** First austenite or ferrite transformation from the liquid
- **Solidus:** Liquid fully transformed to solid
- **A0:** Magnetic transition temperature (Curie temperature) of cementite. The cementite is paramagnetic above A0 and ferromagnetic below
- **A1:** Austenite (FCC\_A1) transforms to ferrite (BCC\_A2) + carbide (cementite or graphite or M23C6 or M7C3)
- **A2:** Magnetic transition temperature (Curie temperature) of ferrite (BCC\_A2). Paramagnetic above A2 and ferromagnetic below
- **A3:** Austenite (FCC\_A1) transforms to ferrite (BCC\_A2)
- **A4:** Delta-ferrite (BCC\_A2) transforms to austenite (FCC\_A1)
- **Acem:** Austenite (FCC\_A1) transforms to cementite (only possible if graphite is suspended)
- **Agraph:** Austenite (FCC\_A1) transforms to graphite (only possible if graphite is enabled)

## About the Ferrite Property Model

---

The **Ferrite** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the thermodynamics and isothermal kinetics of proeutectoid ferrite forming from austenite in hypoeutectoid steels.



For an example, see [PM\\_Fe\\_09: Fe-C-Ni Ferrite](#).

This model is for the form of proeutectoid ferrite which forms without distinct crystallographic preferences at austenite grain boundaries. This type of ferrite is referred to as *grain boundary allotriomorph*, or *polygonal ferrite*. Another type, plate-shaped ferrite (*Widmanstätten ferrite*), is treated in the **Bainite** Property Model as a special case of bainite without cementite.



[About the Bainite Property Model](#)

### Growth Models

Growth is modeled based on a simplified diffusion-control theory with the ferrite/austenite interfacial composition determined by:

- *Orthoequilibrium (OE)*: The ferrite and austenite compositions at the interface are fixed to the final equilibrium compositions
- *Paraequilibrium (PE)*: Only carbon is allowed to partition, reaching PE at the interface
- *Negligible-partitioning local equilibrium (NPLE)*
- *Orthoprecipitate (OP)*: Austenite interfacial composition being the bulk composition, ferrite interfacial composition from the maximum driving force condition (parallel tangent construction in Gibbs energy-composition diagram)
- *Paraprecipitate (PP)*: Only carbon is allowed to partition, otherwise the same as OP

The growth is parabolic if only diffusion-controlled. The parabolic rate constant can be modified by a *phase addition* because the driving force and interfacial compositions are changed. With a finite interfacial mobility and solute drag effects, the growth is then mixed-controlled. The effect is handled in the overall transformation kinetics but not reflected in the parabolic rate constant.

### Nucleation Rate and Interfacial Mobility

Nucleation rate of ferrite allotriomorph on austenite grain boundaries is calculated from classical nucleation theory assuming a pillbox shape of the ferrite nucleus [1988Lan].

When you are setting up the model, parameters for nucleation rate (site density, edge energy, and epsilon, see [1988Lan] for definition) and interfacial mobility (pre-exponential factor and activation energy) can be customized. Otherwise, default values assessed by Thermo-Calc are used. The overall kinetics of isothermal transformation is calculated in the extended-volume framework by numerical integration over time. Austenite composition changes over time due to partitioning of elements, and its effects on growth and nucleation are handled.



The input parameters are entered on the **Configuration** window for the Property Model Calculator. There are also additional axis variables available on the Plot Renderer. See [Ferrite Property Model Settings](#) for details.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

## Reference

[1988Lan] W. F. Lange, M. Enomoto, H. I. Aaronson, The kinetics of ferrite nucleation at austenite grain boundaries in Fe-C alloys. Metall. Trans. A. 19, 427–440 (1988).

## Ferrite Property Model Settings

The **Ferrite** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the thermodynamics and isothermal kinetics of proeutectoid ferrite forming from austenite in hypoeutectoid steels.



[About the Ferrite Property Model](#)



For an example, see [PM\\_Fe\\_09: Fe-C-Ni Ferrite](#).

### *Included Elements and Important Notes*

- Austenite must contain at least Fe and C for this model to function.
- Elements that are quite soluble in both austenite and ferrite—Mn, Ni, Si, Al, Co, Cr, and Mo—have their effects on overall kinetics assessed.

- Cu and strong carbide-formers—W, V, Nb, Ta, Ti, and Zr—can be included, but only to a certain extent not to cause phases other than austenite and ferrite to form at the calculation temperature. Otherwise, the model calculation may give unrealistic results or even fail. If carbide is expected to form at an austenitizing temperature, use the setting **Equilibrium composition at austenitizing temperature** for **Austenite composition from**, then enter an austenitizing temperature.



Elements not listed above are excluded in mass percent from the austenite composition used to calculate the transformation.



For the **Ferrite**, **Bainite**, and **Pearlite** Property Models, it is also useful to review the [Recommended Composition Ranges for Steel Models](#).

### Configuration Settings

These settings are found on the Property Model Calculator when **Ferrite** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Austenite Composition From

Select an option from the **Austenite composition from** list:

- **Nominal composition** uses the system composition as austenite composition.
- **Equilibrium composition at austenizing temperature**. Select to enter an **Austenitizing temperature**. The austenite composition is determined from an equilibrium calculation at the austenitizing temperature (if austenite is stable). This

option is useful when austenite coexists with other phase(s), e.g. primary carbide, before the transformation(s) being calculated starts. But the Model only considers the effect of other phase(s) on austenite composition.



As per above in [Included Elements and Important Notes](#), use the **Equilibrium composition at austenitizing temperature** setting if carbide is expected to form at an austenitizing temperature.

## Austenitizing Temperature

This setting is available when **Equilibrium composition at austenitizing temperature** is selected from the **Austenite composition from** list.

Enter an **Austenitizing temperature** in the field. The unit is the same as that selected under *Condition Definitions*.

## Grain Size

Enter the **Grain size** of the austenite in micrometers ( $\mu\text{m}$ ). The default is 100  $\mu\text{m}$ .

## Growth Mode

Select a **Growth mode**: **Orthoequilibrium (OE)**, **Paraequilibrium (PE)**, **Negligible-partitioning local equilibrium (NPLE)**, **Orthoprecipitate (OP)**, or **Paraprecipitate (PP)**. This defines the compositions on the two sides of the ferrite-austenite interface and therefore affects growth rate.



For definitions of these modes, see [About the Ferrite Property Model](#).

## Phase Addition

Enter a **Phase addition** in J/mol. This applies a Gibbs energy shift to ferrite.

## Skip Time Integration

Click to select the **Skip time integration** checkbox to skip this step in the calculations. Time integration is required for calculating ferrite start/half/finish times, but is usually time-consuming. Skipping time integration allows you to get the other result quantities more quickly.

---

## Custom Parameters

Select the **Custom parameters** checkbox to choose to set more advanced settings as listed below.

- **Site density**, **Edge surface energy**, and **Epsilon** are for calculating grain-boundary nucleation rate (see [1988Lan] for definition and details).
- **M0** and **Q** are to define an interfacial mobility in the Arrhenius equation.

### Site Density

This setting is available when **Custom parameters** is selected. Enter a **Site density** in  $1/m^2$ .

### Edge Surface Energy

This setting is available when **Custom parameters** is selected. Enter an **Edge surface energy** in  $J/m^2$ .

### Epsilon

This setting is available when **Custom parameters** is selected. Enter a value for **Epsilon** in  $J/m^2$ .

### M0

This setting is available when **Custom parameters** is selected. Enter a value for **M0** in  $m \text{ mol}/(Js)$ . This is the pre-exponential factor for interfacial mobility.

### Q

This setting is available when **Custom parameters** is selected. Enter a value for **Q** in  $J/mol$ . This is the activation energy for interfacial mobility.

### Maximum Phase Fraction Change (Absolute) in a Time Step

This setting is available when **Custom parameters** is selected, and is used for time integration.

Enter the **Maximum phase fraction change (absolute) in a time step**.

---

## Maximum Phase Fraction Change (Relative) in a Time Step

This setting is available when **Custom parameters** is selected, and is used for time integration.

Enter the **Maximum phase fraction change (relative) in a time step**

### *Plot Renderer Settings*



[Plot Renderer](#), [Configuration Settings](#), and [Plot Type: TTT Mode](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

- **Ferrite molar fraction (OE or PE):** The ferrite molar fraction at the end of the transformation, determined from orthoequilibrium (for **Ferrite mode** “OE” or “OP”) or paraequilibrium (for **Ferrite mode** “PE”, “NPLe”, or “PP”).
- [Growth Models](#)
- **Parabolic growth rate constant [m/s(1/2)]:** The ratio of half-thickness to square root of time. This is calculated from interfacial composition and diffusion coefficients in austenite.
- **Driving force [J/mol]**
- **Nucleation rate [1/(m<sup>2</sup> s)]:** Steady-state nucleation rate of ferrite per unit austenite grain boundary area.
- **Ferrite start, Ferrite half, or Ferrite finish:** The times at ferrite fraction being 0.02, 0.50, and 0.98, respectively.
- **Ferrite start (relative), Ferrite half (relative), or Ferrite finish (relative):** The times at 2%, 50%, and 98% of the final ferrite fraction, respectively.

### *Reference*

[1988Lan] W. F. Lange, M. Enomoto, H. I. Aaronson, The kinetics of ferrite nucleation at austenite grain boundaries in Fe-C alloys. Metall. Trans. A. 19, 427–440 (1988).

## About the Martensite Fractions Property Model

The **Martensite Fractions** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates fraction of athermal martensite based on an analytical equation derived in Huyan et al. [2016Huy]. It is assumed that the first forming martensite morphology is the only forming one. The first morphology is determined based on the Ms-temperatures of all morphologies.



For an example see [PM\\_Fe\\_01: Fe-Cr-C Martensite with Intercritical Annealing](#).



The input parameters are entered on the Configuration window for the Property Model Calculator and described in [Martensite Fractions Property Model Settings](#).



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

## Reference

[2016Huy] F. Huyan, P. Hedström, L. Höglund, A. Borgenstam, A Thermodynamic-Based Model to Predict the Fraction of Martensite in Steels. Metall. Mater. Trans. A. 47, 4404–4410 (2016).

## Martensite Fractions Property Model Settings

The **Martensite Fractions** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the fraction of athermal martensite based on available driving force.

 [About the Martensite Fractions Property Model](#)

 For an example see [PM\\_Fe\\_01: Fe-Cr-C Martensite with Intercritical Annealing](#).

### Configuration Settings

The settings are found on the Property Model Calculator when **Martensite Fractions** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Intercritical Annealing

Click to select the **Intercritical annealing** checkbox to assume intercritical annealing. Click to clear the checkbox if the specified composition is used directly. When selected, an equilibrium calculation is performed at the specified temperature and the composition of the austenite phase is used for the martensite temperature calculation.

### Annealing Temperature

This setting is available when the **Intercritical annealing** checkbox is selected.

Enter an **Annealing temperature** in the field. The unit is the same as that selected under *Condition Definitions*.

### Grain Size [ $\mu\text{m}$ ]

Enter the **Grain size** of the austenite in micrometers ( $\mu\text{m}$ ). The default is 100  $\mu\text{m}$ .

## Parent Phase Gibbs Energy Addition

In the **Parent phase Gibbs energy addition [J/mol]** field enter a value in J/mol to add a Gibbs energy offset to the parent austenite phase. This can be used, for example, to correct database offsets. The default is 0 J/mol.

### *Plot Renderer Settings*

 [Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

These plot quantities are available:

- **Total martensite percentage:** The martensite percentage at these conditions.
- **Retained austenite percentage:** The retained austenite percentage at these conditions.

## About the Martensite Temperatures Property Model

The **Martensite Temperatures** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the martensite start temperature ( $M_s$ ) based on modeling the transformation barrier with fitted analytical equations. It is based on Stormvinter et al. [2012Sto] with subsequent update and extension by Gulapura Hanumantharaju [2017Gul] and Thermo-Calc internal assessment. The partitionless equilibrium temperature  $T_0$  is calculated using the thermodynamic database. The Model also gives temperatures corresponding to 50%, 90%, and 99% transformations.

Martensite fractions are calculated with the same algorithm as in the **Martensite Fractions** Property Model.



For an example see [PM\\_Fe\\_01: Fe-Cr-C Martensite with Intercritical Annealing](#) and [PM\\_Fe\\_02: Fe-Mn Martensite Morphologies](#).



The input parameters are entered on the Configuration window for the Property Model Calculator and described in [Martensite Temperatures Property Model Settings](#).



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

## References

[2012Sto] A. Stormvinter, A. Borgenstam, J. Ågren, Thermodynamically Based Prediction of the Martensite Start Temperature for Commercial Steels. Metall. Mater. Trans. A. 43, 3870–3879 (2012).

[2017Gul] A. K. Gulapura Hanumantharaju, Master's thesis, KTH Royal Institute of Technology (2017).

## Martensite Temperatures Property Model Settings

The **Martensite Temperatures** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the martensite start (Ms) temperature based on modeling of the transformation barrier. Martensite half and finish temperatures are also given according to the Martensite Fractions Property Model.



[About the Martensite Temperatures Property Model](#)



For an example see [PM\\_Fe\\_01: Fe-Cr-C Martensite with Intercritical Annealing](#) and [PM\\_Fe\\_02: Fe-Mn Martensite Morphologies](#).

### Configuration Settings

The settings are found on the Property Model Calculator when **Martensite Temperatures** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Intercritical Annealing

Click to select the **Intercritical annealing** checkbox to assume intercritical annealing. Click to clear the checkbox if the specified composition is used directly. When selected, an equilibrium calculation is performed at the specified temperature and the composition of the austenite phase is used for the martensite temperature calculation.

### Annealing Temperature

This setting is available when the **Intercritical annealing** checkbox is selected.

Enter an **Annealing temperature** in the field. The unit is the same as that selected under *Condition Definitions*.

---

## Grain Size [ $\mu\text{m}$ ]

Enter the **Grain size** of the austenite in micrometers ( $\mu\text{m}$ ). The default is 100  $\mu\text{m}$ .

## Parent Phase Gibbs Energy Addition

In the **Parent phase Gibbs energy addition [J/mol]** field enter a value in J/mol to add a Gibbs energy offset to the parent austenite phase. This can be used, for example, to correct database offsets. The default is 0 J/mol.

## Plot Renderer Settings



[Plot Renderer](#), [Configuration Settings](#), and [Plot Type: TTT Mode](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

- **Ms temperature:** Martensite start temperature
- **Lath Ms:** Martensite start temperature of lath alpha or alpha-prime martensite
- **Plate Ms:** Martensite start temperature of plate alpha or alpha-prime martensite
- **Epsilon Ms:** Martensite start temperature of epsilon martensite
- **M50:** 50% martensite temperature
- **M90:** 90% martensite temperature
- **M99:** 99% martensite temperature

## About the Martensitic Steel Strength Property Model

The **Martensitic Steel Strength** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the strength and/or hardness of martensitic steels.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

Steels undergoing austenitization, or at least partial austenitization (i.e., intercritical annealing), may form martensite upon quenching. The mechanical properties of martensite depend heavily on the content of interstitial elements in the parent austenite but are of course also affected by other elements and secondary phases. In general, a martensitic steel exhibits the following contributions to strength (or hardness):

$$[\text{Eq. 1}] \quad \sigma_{tot} = \sigma_i + \sigma_{ss} + \Delta\sigma_{mart.}$$

where  $\sigma_i$  is the total intrinsic strength of the phases in the material,  $\sigma_{ss}$  is the contribution from solid solution strengthening from each phase, and  $\Delta\sigma_{mart.}$  is the additional strength due to formation of martensite from the parent austenite.



The  $\sigma_i$  and  $\sigma_{ss}$  terms are modeled in the same way as in the **Yield Strength** Model. See [About the Yield Strength Property Model](#).

The last term ( $\Delta\sigma_{mart.}$ ) in [Eq. 1](#) can further be decomposed into:

$$[\text{Eq. 2}] \quad \Delta\sigma_{mart.} = \sigma_{mart.} \cdot f_{transf.}$$

where  $\Delta\sigma_{mart.}$  is the strength of the martensite and  $f_{transf.}$  is the fraction of parent austenite that has transformed to martensite. In the **Martensitic Steel Strength** Property Model, the fraction transformed is evaluated using the **Martensite Fractions** Model.



[About the Martensite Fractions Property Model](#) and [Martensite Fractions Property Model Settings](#).

This means that the evaluation of the fraction of martensite is dependent on composition, austenitization temperature, quench temperature, and other parameters. These parameters are also available from within the Martensitic Steel Strength model.



Be aware of the absence of the contribution from grain size in [Eq. 1](#). It has been shown that parent austenite grain boundaries contribute very little to the overall strength, due to the numerous other defects and interfaces that form during a martensitic transformation. The grain size may, however, affect the martensitic transformation itself in terms of the residual austenite content, which is modeled separately.

[Eq. 1](#) is valid for the as-quenched condition, however, martensitic steels are most often tempered, which decomposes the martensite (and the residual austenite), to reduce brittleness. In the tempered condition, the martensite strength contribution is reduced by reduction of dislocation density and precipitation of carbides. The carbides may also contribute significantly to the overall strength (or hardness) and thus reduce the softening or even cause what is known as a secondary hardening peak. [Eq. 2](#) thus becomes:

$$[\text{Eq. 3}] \quad \Delta\sigma_{\text{mart.tempered}} = \sigma_{\text{mart.}} \cdot f_{\text{transf.}} \cdot \delta_{\text{temper}}$$

where  $\delta_{\text{temper}}$  is a number between 0 and 1 and depends on tempering temperature and composition. In addition to this, the first two terms of [Eq. 1](#) now also contain contributions from any tempering carbides that form.

## Tempering

In the **Martensitic Steel Strength** Model, tempering is assumed to be time-independent. While this may at first glance seem incorrect, it is in fact so that the main effect of tempering occurs very rapidly for most steels, the main changes in the structure being annihilation of dislocations and precipitation of carbides. The hardness and dislocation density as function of tempering time has been shown to level out very rapidly for many steels, see [Figure 9](#). Thus, tempering time is for most steels only of strong importance if it is very short. This assumption breaks down for extremely low tempering temperatures and for very high alloy

contents, especially if alloyed with elements such as W or Mo, due to sluggish kinetics. However, for both cases, the as-quenched strength serves as a very good approximation (unless tempering is performed for extremely long times).

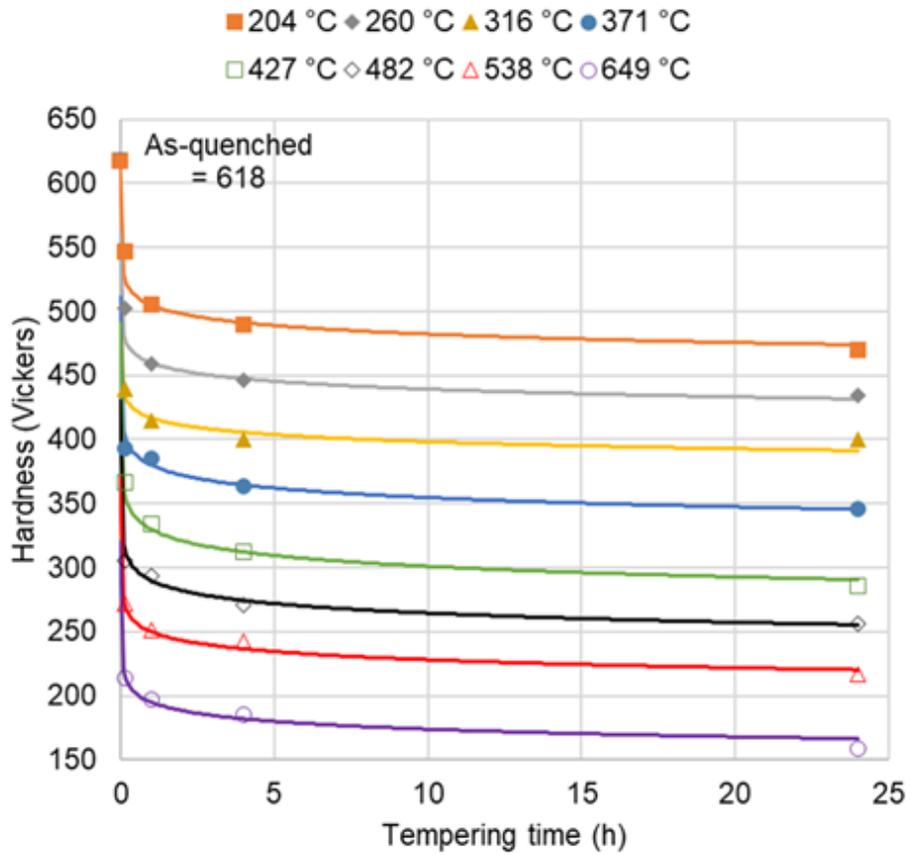


Figure 9: Hardness as function of tempering temperature and time for AISI 1030 carbon steel, data from Grange [1956Gra].

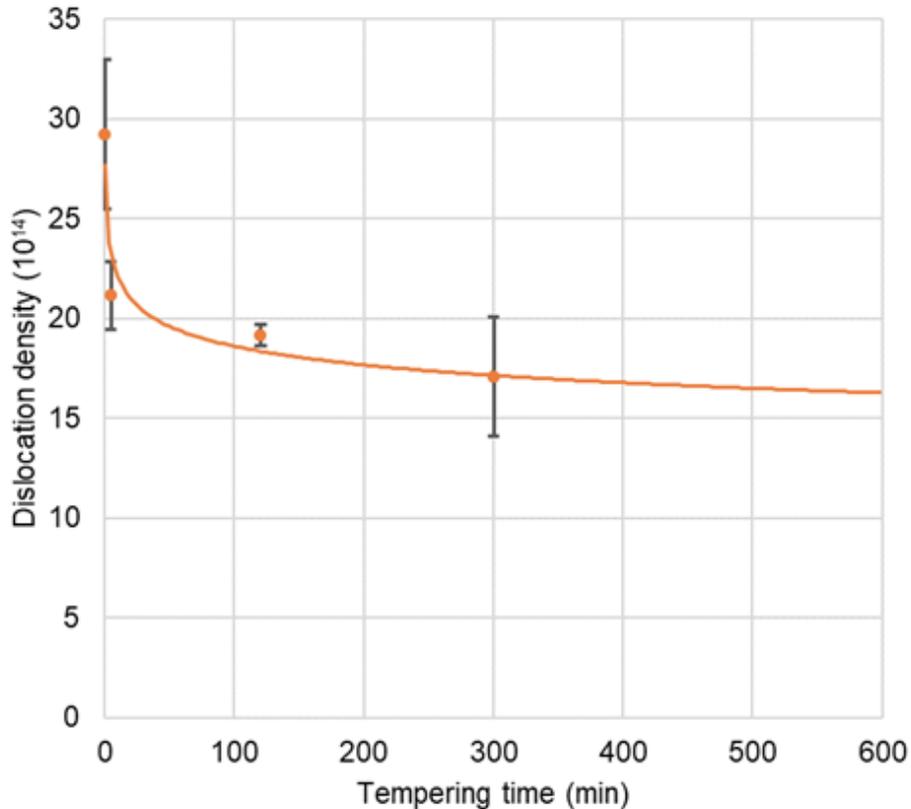


Figure 10: Dislocation density as a function of tempering time, data from Zhou et al [2020Zho] for a martensitic steel (1.4Cr-0.8Mo-0.14V-1.1Si-0.8Mn-0.7Ni) tempered at 450 °C.

The above is not to be interpreted as tempering time having no effect, it is simply a recognition of the limited effect of prolonged tempering time as compared to a short tempering time. It can also be shown that the equilibrium volume fraction of stable (or meta-stable) carbides precipitates very quickly. Initially, the precipitates are very small and the main effect of further tempering would be to coarsen said precipitates, which is a rather slow process at typical tempering temperatures, thus having limited impact on the overall strength.

Tempering carbides are obtained from an equilibrium calculation using the parent austenite composition and tempering temperature, where during set up you can select what phases are allowed on the **System Definer Configuration** window. This is justified by the abundant nucleation sites (due to very high dislocation density) and the fact that the carbides that form are very small, which implies very short diffusion distances.



FCC\_A1, BCC\_A2, and Cementite must be selected on the **System Definer Configuration** window but can be suspended with settings available on the **Property Model Configuration** window. If these phases are excluded on the System Definer, the calculation does not work.



The Model only accepts phases that have had the intrinsic strength assessed. For a list of available phases see the section in the theory for Yield Strength Property Model regarding the [Intrinsic Strength](#) model.



Next steps are to set up this Model. See [Martensitic Steel Strength Property Model Settings](#).



For an example see [PM\\_Fe\\_10: Martensitic Steel Strength](#).

## Note about Hardness Data

Hardness data is the most abundant quantity available for martensitic steels as compared to tensile properties and especially so for studies of pure martensite. Therefore, this Model is developed based on hardness data. However, the hardness of materials with a significant proportion of hard phases, such as martensitic steels, often correlates better with tensile strength rather than yield strength (see e.g., [2008Pav]). For the **Martensitic Steel Strength** Model, the hardness to strength relationship follows Tabors relation, i.e., hardness (MPa) = 3 \* strength (MPa). This means that the strength corresponds to tensile strength for the most part. In case of ambiguity, it is recommended to rely on hardness as calculated by the model.

## References

- [1956Gra] R. A. Grange, R. W. Baughman, Hardness of tempered martensite in carbon and low alloy steels. *Trans. ASM.* 48 (XLVIII), 165–197 (1956).
- [2008Pav] E. J. Pavlina, C. J. Van Tyne, Correlation of Yield Strength and Tensile Strength with Hardness for Steels. *J. Mater. Eng. Perform.* 17, 888–893 (2008).
- [2020Zho] T. Zhou, J. Lu, P. Hedström, Mechanical Behavior of Fresh and Tempered Martensite in a CrMoV-Alloyed Steel Explained by Microstructural Evolution and Strength Modeling. *Metall. Mater. Trans. A.* 51, 5077–5087 (2020).

## Martensitic Steel Strength Property Model Settings

The **Martensitic Steel Strength** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the strength and/or hardness of martensitic steels.



[About the Martensitic Steel Strength Property Model](#)



For an example see [PM\\_Fe\\_10: Martensitic Steel Strength](#).

### System Definer Configuration Settings

The following details relate to choosing the phases on the System Definer.

Then on the Property Model Calculator Configuration Settings window you can suspend as needed using the [Suspend FCC, BCC, or Cementite](#) settings described below.



FCC\_A1, BCC\_A2, and Cementite must be selected on the **System Definer Configuration** window but can be suspended with settings available on the **Property Model Configuration** window. If these phases are excluded on the System Definer, the calculation does not work.



The Model only accepts phases that have had the intrinsic strength assessed. For a list of available phases see the section in the theory for Yield Strength Property Model regarding the [Intrinsic Strength](#) model.

### Property Model Calculator Configuration Settings

The settings are found on the Property Model Calculator when **Martensitic Steel Strength** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

## Evaluation Temperature

The default **Evaluation temperature** is room temperature. Otherwise, select the **Evaluation temperature** checkbox to evaluate the strength at a temperature other than room temperature. The value is taken from the **Temperature** defined under *Condition Definitions*.

Also:

1. If the **Evaluation temperature** is *lower* than the tempering temperature, then the model evaluates the intrinsic and solid solution strengthening at the selected temperature.
2. Alternatively, if the **Evaluation temperature** is *higher* than the tempering temperature, then in addition to (1), the model treats the evaluation temperature as the tempering temperature.

## Annealing Temperature

Enter an **Annealing temperature**, which is the temperature at which austenization or intercritical annealing is performed before the steel is quenched. The default is 1273.15.

## Quench Temperature

Enter a **Quench temperature**, which is the end temperature for quench after austenization. The default is 298.15.

## Tempering and Tempering Temperature

Select the **Tempering** checkbox to estimate the effect of tempering at a specified temperature. It is selected by default.

Then when selected, also enter a **Tempering temperature**, which is the temperature at which tempering is performed. This is used to estimate martensite hardness after tempering. The default is 673.15.

## Suspend FCC, BCC, or Cementite

Select the **Suspend FCC, BCC, or cementite** checkbox to choose to suspend one or more of these phases. All other phases can be deselected on the System Definer to exclude from the calculation as required.



Austenite (FCC\_A1), ferrite (BCC\_A2), and Cementite phases all need to be selected on the System Definer. However, for some alloys it might be necessary to suspend one or more of these from the tempering calculation.

Then from one or more of the **FCC\_A1**, **BCC\_A2**, or **Cementite** lists choose **Entered** or **Suspended**.

## More Options

Select the **More options** checkbox to adjust settings for **Grain size** and the **Parent phase Gibbs energy addition**.

### Grain Size

This setting is available when **More options** is selected.

Enter the **Grain size** of the austenite in micrometers ( $\mu\text{m}$ ). The default is 100  $\mu\text{m}$ .

### Parent Phase Gibbs Energy Addition

This setting is available when **More options** is selected.

In the **Parent phase Gibbs energy addition [J/mol]** field enter a value in J/mol to add a Gibbs energy offset to the parent austenite phase. This can be used, for example, to correct database offsets. The default is 0 J/mol.

## Plot Renderer Settings



[Plot Renderer](#) and [Configuration Settings](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

These plot quantities are available:



For all strength quantities listed below, there is an identical hardness quantity.

- **Martensite strengthening as-quenched:** The contribution of martensite to the overall strength after quenching.
- **Martensite strengthening quenched and tempered:** The contribution of martensite to the overall strength after quenching and tempering.
- **Solid solution strengthening:** Total intrinsic + solid solution strengthening contribution as-quenched.
- **Solid solution strengthening after temper:** Total intrinsic + solid solution strengthening contribution after tempering.
- **Total martensite percentage:** The percentage of martensite after quenching.
- **Retained austenite percentage:** The percentage of retained austenite after quenching.
- **Total strength of the as-quenched steel:** Total strength, including all modeled contributions to the strength, after quenching.
- **Total strength of the tempered steel:** Total strength, including all modeled contributions, after quenching and tempering.
- **Precipitate fraction (Annealing):** The fraction of any given phase present at the annealing temperature before quenching.
- **Precipitate fraction (Tempering):** The fraction of any given phase present after quenching and tempering (precipitates after temper are added to those at annealing, if any).

## About the Pearlite Property Model

---

The **Pearlite** Property Model [2020Yan], available with the Property Model Calculator and the Steel Model Library, calculates the thermodynamics and kinetics of pearlite formation from austenite during isothermal heat treatment.

The **Pearlite** Model describes the thermodynamics and kinetics of pearlite formation from austenite. It is assumed that the overall composition of pearlite is the same as the austenite composition, and growth rate is constant over time. Growth rate and lamellar spacing of pearlite are determined by a criterion where either growth rate or Gibbs energy dissipation rate is maximized. The Model considers Gibbs energy dissipation due to formation of ferrite-cementite interface in pearlite, finite austenite-pearlite interfacial mobility, solute drag, and diffusion of elements within austenite and along austenite-pearlite interface.



Divergent pearlite, of which growth rate and lamellar spacing change with time, is not considered. This Model only considers the transformation from FCC\_A1 phase to pearlite consisting of BCC\_A2 and CEMENTITE\_D011 (for TCFE9 this is CEMENTITE). Pearlite with other phases is not implemented.

Pearlite is assumed to nucleate on austenite grain boundaries. Grain boundary nucleation rate is modeled as a function of composition and temperature. The isothermal transformation kinetics follows Cahn [1956Cah]. Start, half, and finish times are taken as 2%, 50%, and 98% transformation, respectively.

Austenite composition can be taken either as the nominal composition or from an equilibrium calculation at an austenitization temperature. The latter option can be useful when austenite coexists with other phase(s), e.g. primary carbide or ferrite, before pearlite starts to form in austenite. But the Model only considers the effect of other phase(s) on austenite composition.

Model parameters have been determined from published data of maximal growth rate, minimal lamellar spacing, partition coefficients, and isothermal transformation kinetics in ternary and multicomponent systems.



The included elements in the Pearlite Property Model are Fe (dependent element), C, Mn, Cr, Mo, W, Si, Al, Ni, Co. Other elements are neglected by weight percent.



For an example see [PM\\_Fe\\_03: Fe-C-Mn Pearlite](#).



The input parameters are entered on the Configuration window for the Property Model Calculator and described in [Pearlite Property Model Settings](#).



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

## References

[1956Cah] J. W. Cahn, The kinetics of grain boundary nucleated reactions. Acta Metall. 4, 449–459 (1956).

[2020Yan] J.-Y. Yan, J. Ågren, J. Jeppsson, Pearlite in Multicomponent Steels: Phenomenological Steady-State Modeling. Metall. Mater. Trans. A. 51, 1978–2001 (2020).

## Pearlite Property Model Settings

The **Pearlite** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the thermodynamics and kinetics of pearlite formation from austenite during isothermal heat treatment.



The included elements in the Pearlite Property Model are Fe (dependent element), C, Mn, Cr, Mo, W, Si, Al, Ni, Co. Other elements are neglected by weight percent.



For the **Ferrite**, **Bainite**, and **Pearlite** Property Models, it is also useful to review the [Recommended Composition Ranges for Steel Models](#).



[About the Pearlite Property Model](#)



For an example see [PM\\_Fe\\_03: Fe-C-Mn Pearlite](#).

### Configuration Settings

The settings are found on the Property Model Calculator when **Pearlite** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

### Austenite Composition From

Select an option from the **Austenite composition from** list:

- **Nominal composition** uses the system composition as austenite composition.
- **Equilibrium composition at austenizing temperature.** Select to enter an **Austenizing temperature**. The austenite composition is determined from an equilibrium calculation at the austenizing temperature (if austenite is stable). This option is useful when austenite coexists with other phase(s), e.g. primary carbide, before the transformation(s) being calculated starts. But the Model only considers the effect of other phase(s) on austenite composition.

## Austenizing Temperature

This setting is available when **Equilibrium composition at austenizing temperature** is selected from the **Austenite composition from** list.

Enter an **Austenizing temperature** in the field. The unit is the same as that selected under *Condition Definitions*.

## Grain Size [ $\mu\text{m}$ ]

Enter the **Grain size** of the austenite in micrometers ( $\mu\text{m}$ ). The default is 100  $\mu\text{m}$ .

## Criterion

This setting is available when the **Pearlite selected** checkbox is selected.

Select an option from the **Criterion** list to determine the pearlite growth condition. This list is called **Pearlite criterion** for the CCT Diagram and TTT Diagram Property Models.

- For **Maximize growth rate**, pearlite growth rate is maximized with respect to lamellar spacing (and for optimal pearlite, partition coefficients of substitutional alloying elements between ferrite and cementite). The calculation gives maximal growth rate and minimal lamellar spacing. This is the default criterion.
- For **Maximize Gibbs energy dissipation rate**, Gibbs energy dissipation rate is maximized with respect to lamellar spacing (and for optimal pearlite, partition coefficients of substitutional alloying elements between ferrite and cementite). The calculation gives a smaller growth rate than the maximal, and a larger lamellar spacing than the minimal, which may better represent the average growth rate and lamellar spacing of all pearlite colonies in the material.

---

## Pearlite Mode

This setting is available when the **Pearlite selected** checkbox is selected. It is used to determine how substitutional alloying elements partition between ferrite and cementite in pearlite. Select an option from the **Pearlite mode** list.

- For **Optimal pearlite**, the partitioning of substitutional alloying elements between ferrite and cementite is optimized according to the **Criterion** while keeping carbon equilibrium between the two phases. This is the default mode. This mode realizes a smooth transition between ortho-pearlite and para-pearlite.
- For **Para-pearlite**, ferrite and cementite are in para-equilibrium where substitutional alloying elements do not partition and carbon reaches equilibrium.
- For **Ortho-pearlite**, ferrite and cementite are in ortho-equilibrium.

### *Plot Renderer Settings*



[Plot Renderer](#), [Configuration Settings](#), and [Plot Type: TTT Mode](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

- **Temperature**
- **Start time (2% pearlite)**
- **Half time (50% pearlite)**
- **Finish time (98% pearlite)**
- **Growth rate [m/s]**
- **Lamellar spacing [m]**
- **Cementite molar fraction in pearlite**
- **Driving force of pearlite formation [J/mol]**

## About the TTT Diagram Property Model

The **TTT Diagram** Property Model, available with the Property Model Calculator and the Steel Model Library, can be used to calculate a Time-Temperature-Transformation (TTT) diagram, with a **One Axis** calculation on temperature. The Model calculates the isothermal kinetics of concurrent transformation of austenite into ferrite, pearlite, bainite, and martensite. The algorithms and parameters for ferrite, pearlite, bainite, and martensite are the same as in the individual Property Models (**Ferrite, Pearlite, Bainite, Martensite Temperatures**, and **Martensite Fractions**), but the competitive occupation of prior austenite grain boundaries and bulk volume is handled in the **TTT Diagram** Property Model.

If the reaction temperature is below martensite-start temperature ( $M_s$ ), it is assumed that a fraction of (athermal) martensite according to the calculation forms instantaneously at time zero, prior to the formation of ferrite, pearlite, and bainite. Otherwise austenite transforms to ferrite, pearlite, and/or bainite.



For an example see [PM\\_Fe\\_06: TTT Diagram for Fe-C-Mn-Si-Cr-V](#)



The input parameters are entered on the Configuration window for the Property Model Calculator. There are also additional axis variables available on the Plot Renderer.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

## TTT Diagram Property Model Settings

The **TTT Diagram** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the concurrent transformations to ferrite, pearlite, bainite, and martensite under isothermal conditions and is used to construct a time-temperature-transformation (TTT) diagram.



[About the TTT Diagram Property Model](#) and [Steel Library TTT Diagram Template](#)



For an example see [PM\\_Fe\\_06: TTT Diagram for Fe-C-Mn-Si-Cr-V](#)

## Configuration Settings

These settings are found on the Property Model Calculator when **TTT Diagram** is selected under **Steel Models**.



When working in the **Configuration** window, click the **Description** tab for more information about the Model.



For the details about the *Condition Definitions*, *Calculation Type*, *Timeout in minutes*, and calculation associated settings, see [Configuration Window Settings](#).

## Austenite Composition From

Select an option from the **Austenite composition from** list:

- **Nominal composition** uses the system composition as austenite composition.
- **Equilibrium composition at austenizing temperature**. Select to enter an **Austenizing temperature**. The austenite composition is determined from an equilibrium calculation at the austenizing temperature (if austenite is stable). This option is useful when austenite coexists with other phase(s), e.g. primary carbide, before the transformation(s) being calculated starts. But the Model only considers the effect of other phase(s) on austenite composition.

## Austenizing Temperature

This setting is available when **Equilibrium composition at austenizing temperature** is selected from the **Austenite composition from** list.

Enter an **Austenizing temperature** in the field. The unit is the same as that selected under *Condition Definitions*.

## Grain Size

Enter the **Grain size** of the austenite in micrometers ( $\mu\text{m}$ ). The default is 100  $\mu\text{m}$ .

## Calculation Setting

Select **Quick** (the default), **Accurate**, or **Custom** to balance accuracy and calculation time. Choose **Custom** to define the numerical settings yourself, which are otherwise set in the background when **Quick** or **Accurate** is selected. See the separate section below about the details for the **Custom** settings.

### Quick Calculation Setting

**Quick** is used for a relatively quick and rough calculation to reveal the general feature of a TTT diagram. Interpolation is used when necessary in order to avoid calculating the growth rates and nucleation rates of pearlite and bainite at every time step.

- For simplicity, **Ferrite mode** is **PE** (paraequilibrium), and **Pearlite mode** is **Para-pearlite**.
- Recommended temperature interval: 50-100 K.



Each temperature can take several minutes.

### Accurate Calculation Setting

**Accurate** evaluates the growth rates and nucleation rates at every time step accurately (without interpolation). The time step control is also tighter than when using the **Quick** setting.

- **Ferrite mode** is *faster start*, meaning the mode between OE and PE which gives a faster initial transformation.
- **Pearlite mode** is **Optimal pearlite**, but its rate calculation becomes slower if austenite contains many alloying elements.
- Recommended temperature interval: 20-50 K or even lower.



Calculating the whole TTT diagram may take several hours.

## Details about the Quick and Accurate Calculation Settings

When you choose **Quick** or **Accurate** as the calculation setting, the following is what is automatically set in the background for each of the sections. These sections are visible to you and also can be edited when **Custom** is chosen:

<i>Setting</i>	<i>Quick</i>	<i>Accurate</i>
<b>Ferrite selected</b>	checkbox selected (yes)	checkbox selected (yes)
<b>Pearlite selected</b>	checkbox selected (yes)	checkbox selected (yes)
<b>Bainite selected</b>	checkbox selected (yes)	checkbox selected (yes)
<b>Martensite selected</b>	checkbox selected (yes)	checkbox selected (yes)
<b>Ferrite mode</b>	PE only	Faster start (TTT); PE if possible otherwise OE (CCT)
<b>Pearlite criterion</b>	Maximize growth rate	Maximize growth rate
<b>Pearlite mode</b>	Para-pearlite	Optimal pearlite
<b>Use interpolation when necessary</b>	checkbox selected (yes)	checkbox NOT selected (no)
<b>Interpolation error tolerance</b>	0.1	(does not apply)
<b>Maximum phase fraction change (absolute) in a time step</b>	0.005	0.005
<b>Maximum phase fraction change (relative) in a time step</b>	0.05	0.05
<b>Error tolerance for austenite fraction</b>	1E-3	1E-4

## Custom Calculation Setting

The following settings are available when **Custom** is selected as the *Calculation Setting*. Otherwise these settings are handled in the background as described above for when **Quick** or **Accurate** is selected.

### Ferrite/Pearlite/Bainite/Martensite Selected Checkboxes

By default the **Ferrite selected**, **Pearlite selected**, **Bainite selected**, and **Martensite selected** checkboxes are selected. For ferrite and pearlite, additional settings are then available.

Click to deselect as required to exclude one or more from the calculation.

### Pearlite Criterion

This setting is available when the **Pearlite selected** checkbox is selected.

Select an option from the **Criterion** list to determine the pearlite growth condition. This list is called **Pearlite criterion** for the CCT Diagram and TTT Diagram Property Models.

- For **Maximize growth rate**, pearlite growth rate is maximized with respect to lamellar spacing (and for optimal pearlite, partition coefficients of substitutional alloying elements between ferrite and cementite). The calculation gives maximal growth rate and minimal lamellar spacing. This is the default criterion.
- For **Maximize Gibbs energy dissipation rate**, Gibbs energy dissipation rate is maximized with respect to lamellar spacing (and for optimal pearlite, partition coefficients of substitutional alloying elements between ferrite and cementite). The calculation gives a smaller growth rate than the maximal, and a larger lamellar spacing than the minimal, which may better represent the average growth rate and lamellar spacing of all pearlite colonies in the material.

### Carbide in Pearlite

From the **Carbide in pearlite** list, select the carbide to be present in pearlite: **CEMENTITE** (default), **M7C3**, or **M23C6**. For M23C6, this usually depends on Cr content.



When M23C6 is selected, parameters optimized for stainless steels are invoked.



## Recommended Composition Ranges for Steel Models

### Carbide in Bainite

From the **Carbide in bainite** list, select the carbide to be present in pearlite: **CEMENTITE** (default) or **M7C3**. For M7C3, this usually depends on Cr content.

### Pearlite Mode

This setting is available when the **Pearlite selected** checkbox is selected. It is used to determine how substitutional alloying elements partition between ferrite and cementite in pearlite. Select an option from the **Pearlite mode** list.

- For **Optimal pearlite**, the partitioning of substitutional alloying elements between ferrite and cementite is optimized according to the **Criterion** while keeping carbon equilibrium between the two phases. This is the default mode. This mode realizes a smooth transition between ortho-pearlite and para-pearlite.
- For **Para-pearlite**, ferrite and cementite are in para-equilibrium where substitutional alloying elements do not partition and carbon reaches equilibrium.
- For **Ortho-pearlite**, ferrite and cementite are in ortho-equilibrium.

### Use Interpolation When Necessary

If the **Use interpolation when necessary** checkbox is selected, and depending on alloy composition and temperature, if the code determines that an interpolation is necessary and possible to implement, then a composition grid (TTT diagram) or composition-temperature grid (CCT diagram) is generated and refined as needed.

The growth rates and nucleation rates of pearlite and bainite, as functions of composition (TTT diagram) or composition and temperature (CCT diagram), are determined by interpolation (if sufficiently accurate). This is intended to avoid direct calculations of pearlite and bainite rates for every time step, which is usually time-consuming.

### Interpolation Error Tolerance

This setting is available when the **Use interpolation when necessary** checkbox is selected.

The growth and nucleation rates of pearlite and bainite, if determined from interpolation, are accepted only when the relative interpolation error estimated is below the value entered here. A large number implies few grid points necessary (shorter calculation time) and lower accuracy, and a small number implies the opposite.

### Maximum Phase Fraction Change (Absolute/Relative) in a Time Step

The **Maximum phase fraction change (absolute) in a time step** and **Maximum phase fraction change (relative) in a time step** settings are used to control time step size during time integration.

The phase fraction ( $X$ ) of ferrite, pearlite, and bainite cannot change more than  $atol + X * rtol$  during a time step, where  $atol$  and  $rtol$  are the absolute and relative maximum change, respectively. If the change is too large, the new time step is rejected, and time integration is retried with a smaller time step.

### Error Tolerance for Austenite Fraction

The **Error tolerance for austenite fraction** setting is used to control the accuracy for austenite fraction during time integration. Finite time steps cause an error in austenite volume fraction. When the error is larger than the tolerance, time integration is backed up and retried with a smaller time step.

### Plot Renderer Settings



[Plot Renderer](#), [Configuration Settings](#), and [Plot Type: TTT Mode](#)

When setting up your calculation on the Plot Renderer and/or Table Renderer, the following axis variables are available for the conditions defined on the Property Model Calculator.

- **Ferrite start (2%)**
- **Pearlite start (2%)**
- **Bainite start (2%)**
- **Total ferrite start (2%)**: The start of the sum of polygonal ferrite and bainitic ferrite.
- **Total ferrite + cementite start (2%)**: The start of the sum of pearlite and cementite-containing bainite.
- **Total ferrite + cementite finish (98%)**: 98% austenite transformed to ferrite, pearlite, and bainite.
- **Austenite transformed 2%**
- **Austenite transformed 10%**

- **Austenite transformed 50%**
- **Austenite transformed 90%**
- **Austenite transformed 98%**
- **Martensite start**
- **Martensite 50%**
- **Martensite 98%**

Fractions of transformation products at the last time step are also available:

- **Terminal fraction of ferrite**
- **Terminal fraction of pearlite**
- **Terminal fraction of bainite**
- **Terminal fraction of martensite**
- **Terminal fraction of total ferrite**
- **Terminal fraction of total ferrite+cementite**



Martensite start, Martensite 50%, and Martensite 98% are temperature quantities. Terminal fractions are unit-free quantities. The others are time quantities.



In the **TTT Mode** of a Plot Renderer, the three temperature quantities for martensite are plotted as horizontal lines because these are independent of time. The other time quantities are plotted as functions of temperature.

# **Graphical Mode Examples Guide**

---

**Thermo-Calc Version 2024a**



---

# Thermo-Calc Examples Collection

---



These are the Graphical Mode examples. There are separate Console Mode examples also available.

## In this section:

---

Graphical Mode Examples List by Product .....	2
T_01: Calculating a Single-Point Equilibrium .....	3
T_02: Stepping in Temperature in the Fe-C System .....	5
T_03: Fe-C Phase Diagrams .....	8
T_04: Fe-Cr-C Ternary Phase Diagram at 1000 K .....	11
T_05: Stable and the Metastable Fe-C Phase Diagrams .....	14
T_06: Serially Coupled Equilibrium Calculators .....	16
T_07: User-Defined Functions .....	19
T_08: Scheil and Equilibrium Solidification .....	22
T_09: Carbide Driving Force Heat Map .....	26
T_10: Scheil Solidification with Back Diffusion .....	28
T_11: Surface Tension in Cu-Zr .....	30
T_12: Viscosity in Cr-Ni .....	33
T_13: Scheil Solidification with Solute Trapping .....	36
T_14: Fe-Cr-Ni Transition Comparison Using the Material to Material Calculator .....	38
T_15: Molar Volume and Thermal Expansion Coefficients for L12-type Al <sub>3</sub> Sc and Al <sub>3</sub> Zr Compounds .....	41
T_16: Electrical Resistivity and Thermal Conductivity for Pure Copper .....	44
T_17: Al <sub>2</sub> O <sub>3</sub> -MgO Phase Diagram .....	48
T_18: Scheil with Delta Ferrite to Austenite Transition .....	50

---

## Graphical Mode Examples List by Product

This guide includes descriptions of the following examples.



### [About the Examples](#)

Category	Example number
<a href="#">Thermo-Calc Examples Collection</a> includes binary and ternary system examples, Scheil, the Equilibrium Calculator, Material to Material Calculator, plus more	T_01 to T_18
<a href="#">Thermo-Calc General Property Models Examples Collection: General Models Library</a>	General Models: PM_G_01 to PM_G_17
<a href="#">Steel Model Library Examples Collection</a>	Steel Models: PM_Fe_01 to PM_Fe_10
<a href="#">Nickel Model Library Examples Collection</a>	Nickel Models: PM_Ni_01 to PM_Ni_04
<a href="#">Process Metallurgy Module Examples Collection</a>	PMET_01 to PMET_08
<a href="#">Diffusion Module (DICTRA) Examples Collection</a>	D_01, D_02, D_03 (the Quick Start Guide Examples) D_04 to D_10
<a href="#">Precipitation Module (TC-PRISMA) Examples Collection</a>	P_01 to P_16
<a href="#">Additive Manufacturing (AM) Module Examples Collection</a>	AM_01 to AM_06

---

## T\_01: Calculating a Single-Point Equilibrium

---

This example shows the result from a single-point equilibrium calculation in the Fe-C system. It demonstrates the use of the Equilibrium Calculator and generates a Table Renderer in the **Visualizations** window. The number of equilibrium conditions is  $C+2$  where  $C$  is the number of components, meaning that four conditions are needed in this example:

- **Temperature** is 1000 K
- **Pressure** is 100000 Pa
- **System size** is 1 mole
- **Mass percent carbon** is 0.1%



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

### ***Project File Information***

- Folder: **Thermo-Calc**
  - File name: *T\_01\_Single-point\_equilibrium.tcu*
-

Visualizations				
Table Renderer 1				
<b>System</b>				
Moles	1.00000			
Mass	55.64392	[g]		
Temperature	1000.00000	[K]		
Total Gibbs Energy	-42141.28527	[J]		
Enthalpy	24706.54116	[J]		
Volume	7.30429E-6	[m <sup>3</sup> ]		
<i>Component</i>	<i>Mole Fraction</i>	<i>Mass Fraction</i>	<i>Activity</i>	<i>Potential</i>
C	0.00463	0.00100	0.21816	-12658.89428
Fe	0.99537	0.99900	0.00619	-42278.50542
<b>Stable Phases</b>				
<b>BCC_A2#1</b>	<i>Moles</i>	<i>Mass</i>	<i>Volume Fraction</i>	<input type="text" value="Composition"/>
	0.99609	55.59692	0.99711	
<b>Composition</b>				
<i>Component</i>	<i>Mole Fraction</i>	<i>Mass Fraction</i>		
Fe	0.99928	0.99984		
C	0.00072	0.00016		
<b>GRAPHITE#1</b>	<i>Moles</i>	<i>Mass</i>	<i>Volume Fraction</i>	<input type="text" value="Composition"/>
	0.00391	0.04700	0.00289	
<b>Composition</b>				
<i>Component</i>	<i>Mole Fraction</i>	<i>Mass Fraction</i>		
C	1.00000	1.00000		
Fe	0.00000	0.00000		

Figure 1: The displayed results of the calculation show that the BCC\_A2 (ferrite) and GRAPHITE phases are stable for this set of equilibrium conditions.

---

## T\_02: Stepping in Temperature in the Fe-C System

---

This example shows how the fractions of stable phases vary for an Fe-0.1 mass-% C alloy when the temperature is varied between 500 and 2000 K. It demonstrates the use of the Equilibrium Calculator. To allow temperature to vary, the corresponding **Axis Definition** checkbox is selected.



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Thermo-Calc**
  - File name: *T\_02\_Step\_in\_temperature\_in\_Fe-C.tcu*
-

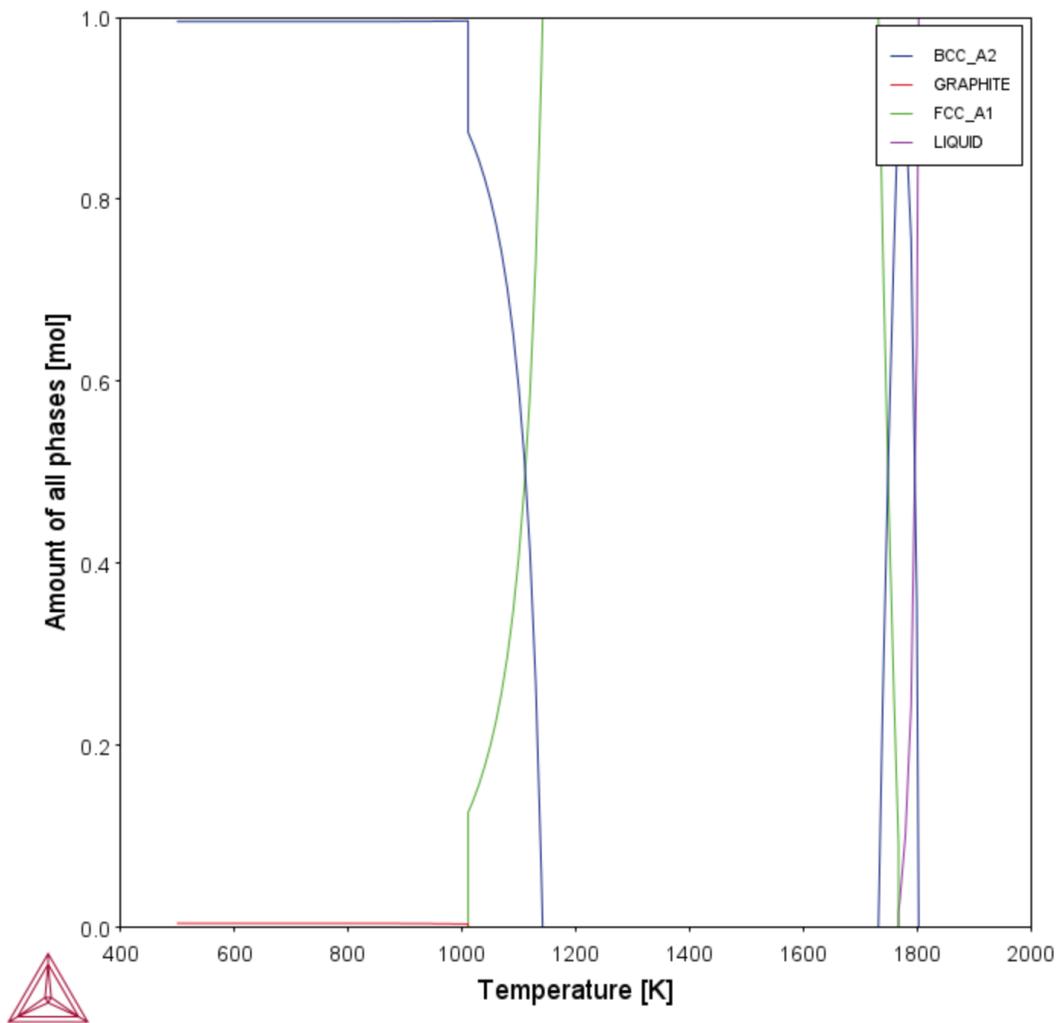


Figure 2: In this example, results are displayed graphically using a Plot Renderer activity.



For the Plot Renderer, there is an option to use the Table View  button to convert plots to table data for all calculation types and calculators. See [Plot Renderer: Convert a Plot to a Table and Export the Data](#).

If you want, you can also use a Table Renderer to generate text results instead, as shown in this example after adding a Table Renderer to the Equilibrium Calculator and clicking **Perform**.

Results				
Plot Renderer 1	Table Renderer 1			
Temperature [K]	Volume fraction of BCC_A2	Volume fraction of FCC_A1	Volume fraction of GRAPHITE	Volume fraction of LIQUID
1000.00000	0.99711		0.00289	
1010.00000	0.99718		0.00282	
1011.17630	0.99719		0.00281	
1011.17630	0.99719	0.00000	0.00281	
1011.17630	0.87704	0.12296	0.00000	
1011.17630	0.87704	0.12296		
1020.00000	0.86401	0.13599		
1030.00000	0.84702	0.15298		
1040.00000	0.82743	0.17257		
1050.00000	0.80486	0.19514		

## T\_03: Fe-C Phase Diagrams

---

This example shows the stable Fe-C phase diagram (stable meaning that the graphite phase is entered in the calculation). The same diagram is calculated using the Binary Calculator activity and then using a System Definer and Equilibrium Calculator.

In both cases, a Plot Renderer is used to display results. The purpose of the Binary Calculator is to simplify common calculations for binary systems. In the Equilibrium Calculator two axes are defined: *Mass percent C* and *Temperature*. An axis variable must also be an equilibrium condition.



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Thermo-Calc**
- File name: *T\_03\_Fe-C\_phase\_diagram.tcu*

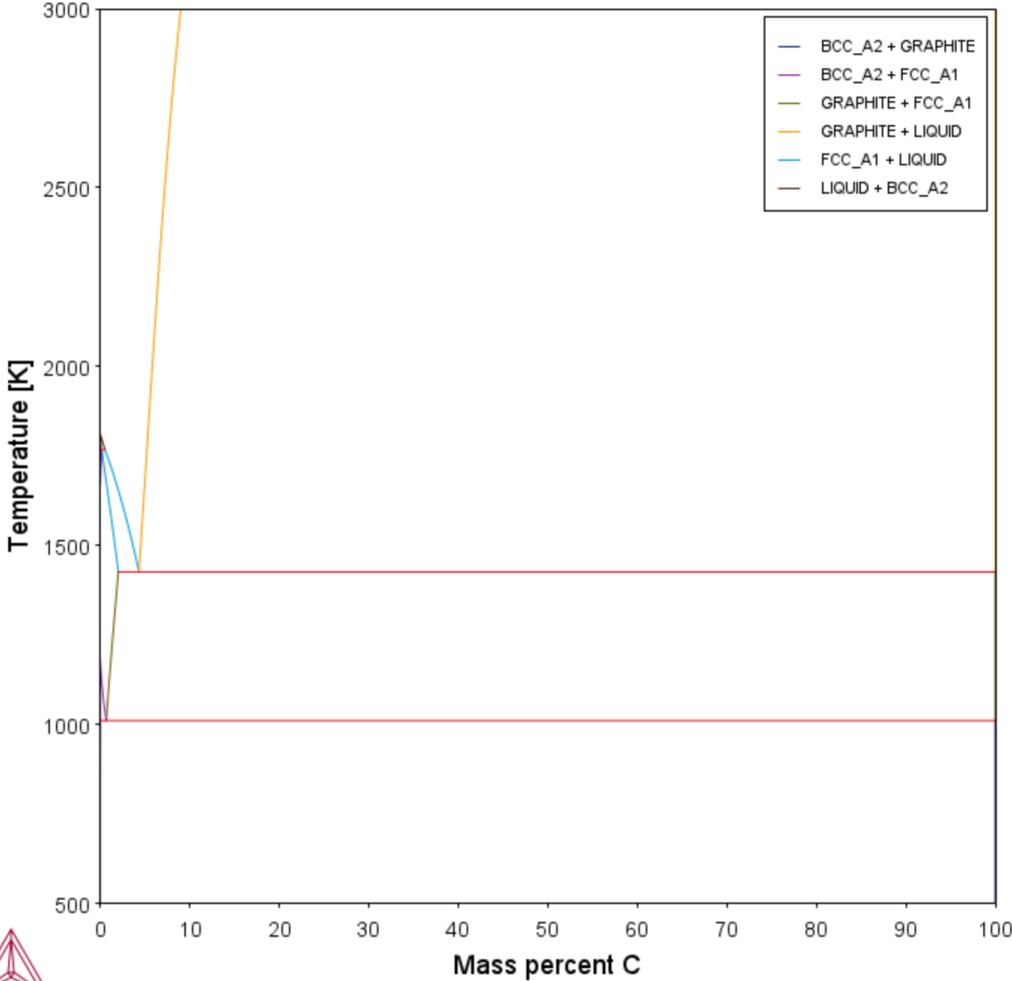


Figure 3: The plot result of the System Definer and Equilibrium Calculator.

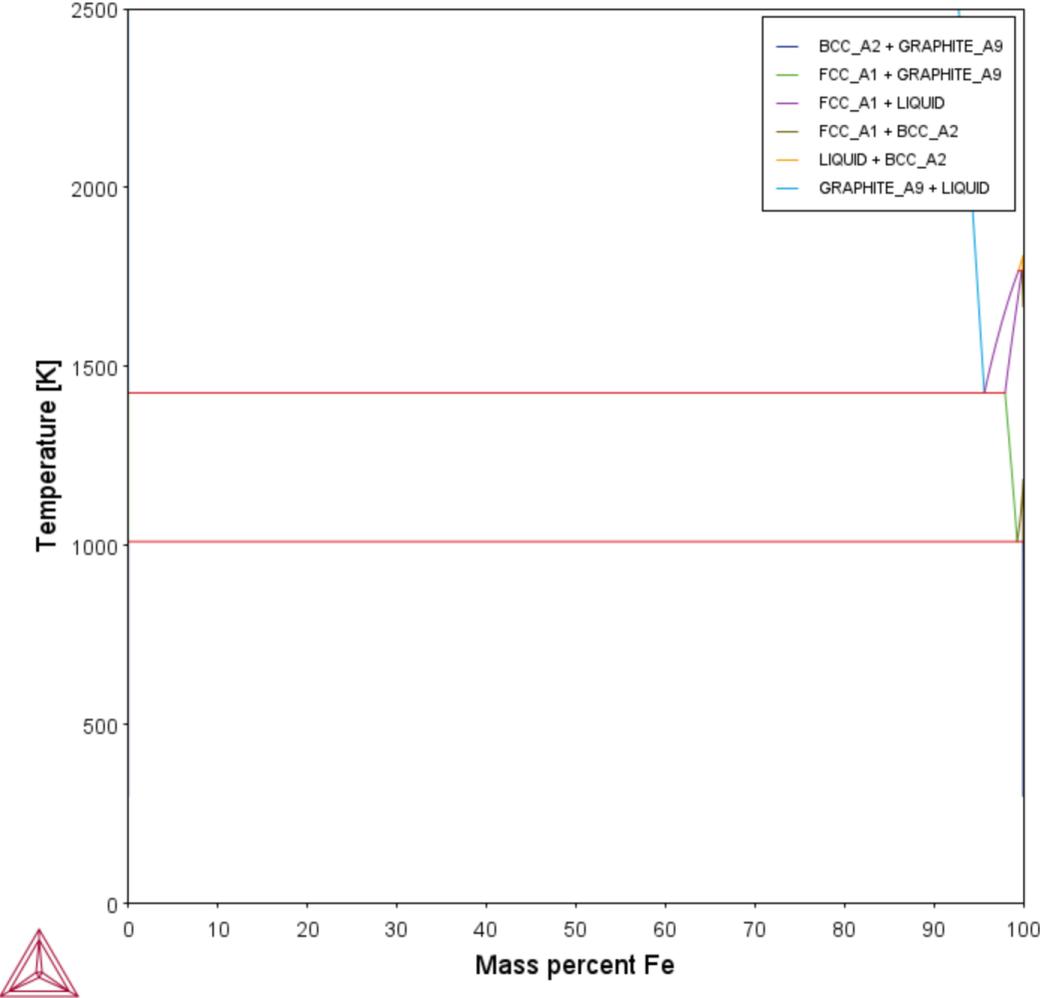


Figure 4: The plot result of the Binary Calculator simulation.

---

## T\_04: Fe-Cr-C Ternary Phase Diagram at 1000 K

---

This example shows a ternary phase diagram in the Fe-Cr-C system at 1000 K. Similar to example [T\\_03: Fe-C Phase Diagrams](#), the same diagram is calculated using a Ternary Calculator and then using a System Definer and Equilibrium Calculator. Two Plot Renderer activities are added to create two plots.



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Thermo-Calc**
- File name: *T\_04\_Fe-Cr-C\_ternary\_phase\_diagram.tcu*

Try practicing with two features on the Plot Renderer Configuration window. To toggle between a triangular and a rectangular diagram, click the **Show Triangular** button then click **Perform** to see what happens. To toggle the X and Y axis variables, click the **Switch Axes** button and then click **Perform**.

---

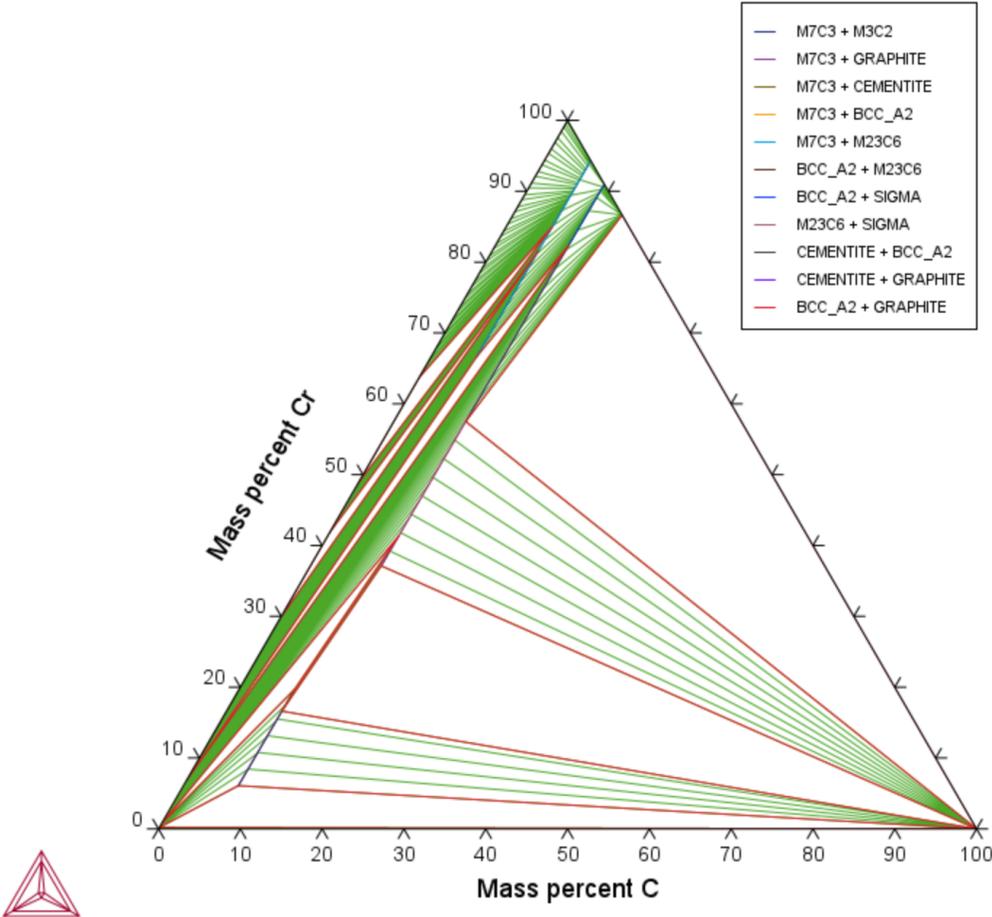


Figure 5: Ternary Calculator

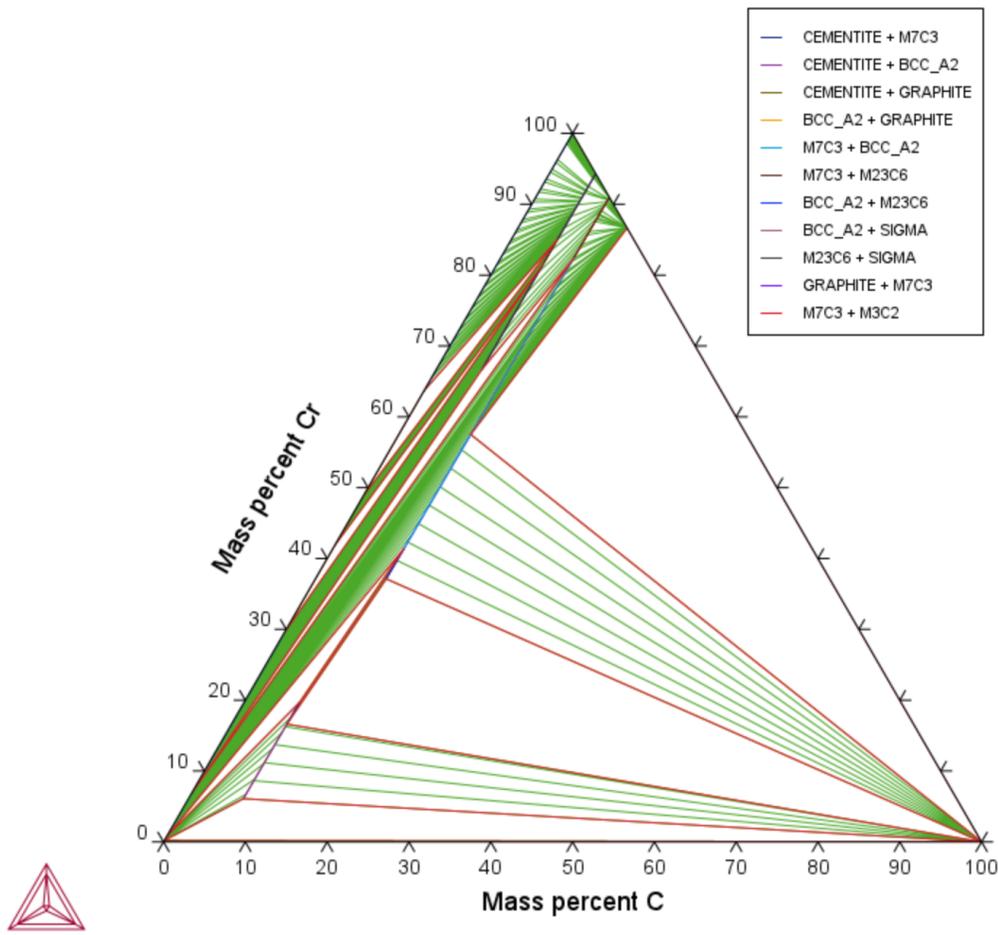


Figure 6: Equilibrium Calculator

## T\_05: Stable and the Metastable Fe-C Phase Diagrams

This example shows how to overlay results from two calculations in the same plot using the Equilibrium Calculator.



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

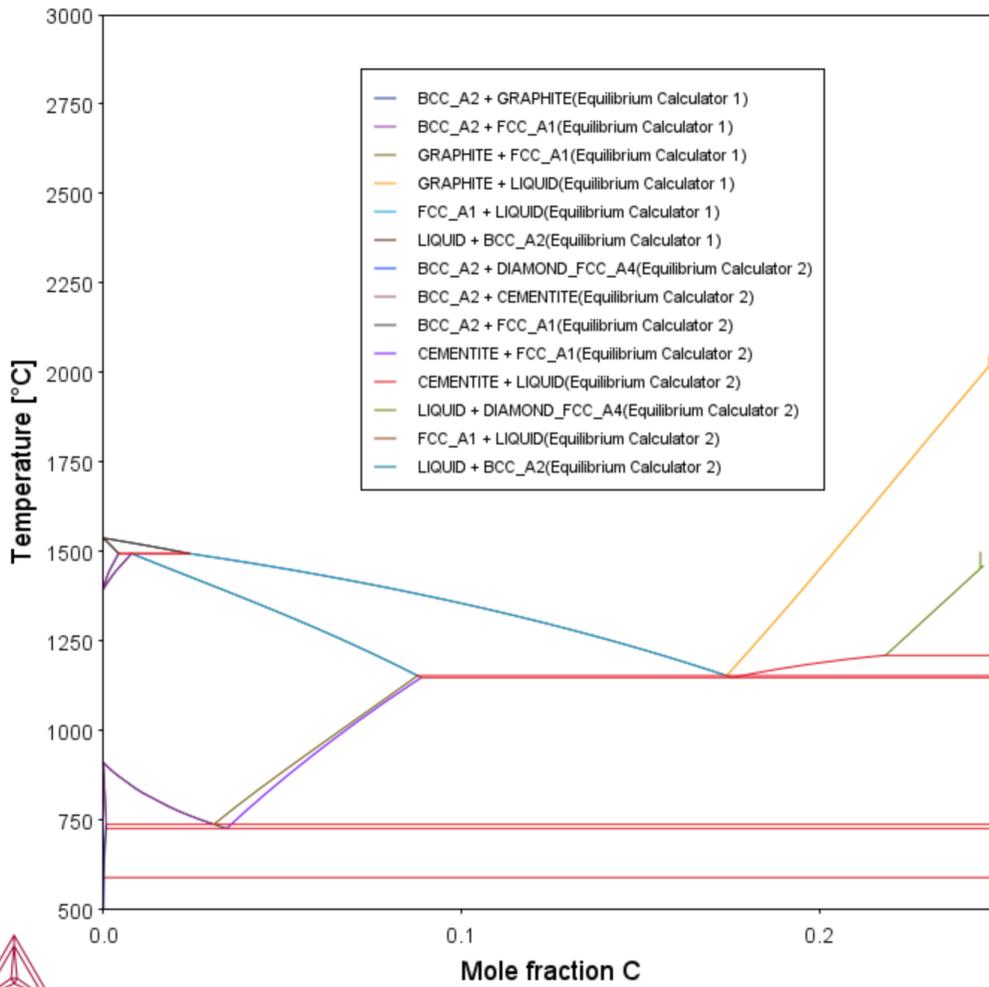
- Folder: **Thermo-Calc**
- File name: *T\_05\_Fe-C\_stable\_and\_metastable\_phase\_diagram.tcu*



[Plotting Several Calculations in One Diagram](#)

1. The Plot Renderer activity is first created as a successor to Equilibrium Calculator 1.
2. Then right-click the Plot Renderer node and select **Add predecessor** and the results from both equilibrium calculators are in the same plot. In **Equilibrium Calculator 1** the stable phase diagram is calculated and in **Equilibrium Calculator 2** the metastable phase.
3. The metastable diagram is obtained by deselecting the graphite phase on System Definer2 → **Phases and phase constitution** tab.

Elements	Species	Phases and Phase Constitution
<i>Phases</i>		
Status	Name ^	FEDEMO
Entered	BCC_A2	<input checked="" type="checkbox"/>
Entered	CBCC_A12	<input checked="" type="checkbox"/>
Entered	CEMENTITE	<input checked="" type="checkbox"/>
Entered	CUB_A13	<input checked="" type="checkbox"/>
Entered	DIAMOND_FCC_...	<input checked="" type="checkbox"/>
Entered	FCC_A1	<input checked="" type="checkbox"/>
Entered	GAS	<input checked="" type="checkbox"/>
Entered	GRAPHITE	<input type="checkbox"/>
Entered	HCP_A3	<input checked="" type="checkbox"/>



---

## T\_06: Serially Coupled Equilibrium Calculators

---

Sometimes there are multiple solutions for a given set of equilibrium conditions. In other cases the equilibrium calculation does not converge. You can then aid the final calculation by, in effect, telling the software where it should start the search for the equilibrium.

This example shows how to serially couple two Equilibrium Calculator nodes together for more complex equilibrium conditions. For each calculation, the output is to a Table Renderer.



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Thermo-Calc**
- File name: *T\_06\_Serial\_equilibrium\_calculators.tcu*

## **Table Renderer Results**

### **Equilibrium Calculator 1**

In the first example, the objective is to calculate the equilibrium at 850° C where the mole fractions of the FCC A1 and BCC A2 phases are 0.5 each for an Fe-Cr-C steel with 10% chromium. In Equilibrium Calculator 1 a simple set of equilibrium conditions (temperature, pressure, system size and composition) are used to find a carbon content where only the FCC A1 and BCC A2 phases are stable. The results from this preliminary calculation are displayed in Table Renderer 1.

---

Visualizations 🗄

Table Renderer 1 Table Renderer 2 Table Renderer 3 Table Renderer 4

System				
Moles	1.00000			
Mass	55.41634	[g]		
Temperature	1123.15000	[K]		
Total Gibbs Energy	-52361.90905	[J]		
Enthalpy	32589.15580	[J]		
Volume	7.34239E-6	[m <sup>3</sup> ]		
Component	Mole Fraction	Mass Fraction	Activity	Potential
C	0.00046	0.00010	0.00137	-61592.91681
Cr	0.10658	0.10000	0.00235	-56531.30746
Fe	0.89296	0.89990	0.00387	-51859.50665
Stable Phases				
BCC_A2#1	Moles	Mass	Volume Fraction	
	0.76777	42.55315	0.76989	Composition <input type="text" value="Composition"/>
Composition				
Component	Mole Fraction	Mass Fraction		
Fe	0.89183	0.89864		
Cr	0.10801	0.10133		
C	0.00016	0.00003		
FCC_A1#1	Moles	Mass	Volume Fraction	
	0.23223	12.86320	0.23011	Composition <input type="text" value="Composition"/>
Composition				
Component	Mole Fraction	Mass Fraction		
Fe	0.89669	0.90408		
Cr	0.10185	0.09560		
C	0.00147	0.00032		

### Equilibrium Calculator 2, 3, and 4

Click each tab along the top of the **Visualizations** window to view the results associated to **Table Renderer 2**, **Table Renderer 2**, and **Table Renderer 2**.

Visualizations			
Table Renderer 1	Table Renderer 2	Table Renderer 3	Table Renderer 4

- In Equilibrium Calculator 2 the carbon content equilibrium condition is replaced by the condition that the numbers of moles of the BCC A2 phase should be 0.5. The final result is displayed in Table Renderer 2. In this case, the final objective is to calculate the solidus temperature of an Fe-Cr-C steel with 10% Cr and 0.01% C.
- In Equilibrium Calculator 3 the state at 2000 K is calculated and the result is displayed in Table Renderer 3.
- In Equilibrium Calculator 4 the temperature equilibrium condition is replaced by the condition *Fix phase / liquid / 0.0*, meaning that liquid should be stable in an amount of zero moles, i.e. the solidus temperature. The final result is displayed in Table Renderer 4.

## T\_07: User-Defined Functions

This example shows how to implement user-defined functions. A series of equilibria for an Fe-Cr-C alloy are calculated by varying temperature between 500 and 3000 K. In the configuration window of the Equilibrium Calculator under the **Functions** tab, two identically meaning functions are defined, *fraction solid* and *f solid*, the values of which are plotted against temperature in two Plot Renderer activities. Functions can be entered in terms of *Quantities*  $Q_1$ ,  $Q_2$ ,  $Q_3$  and so forth, or by using the Thermo-Calc syntax.

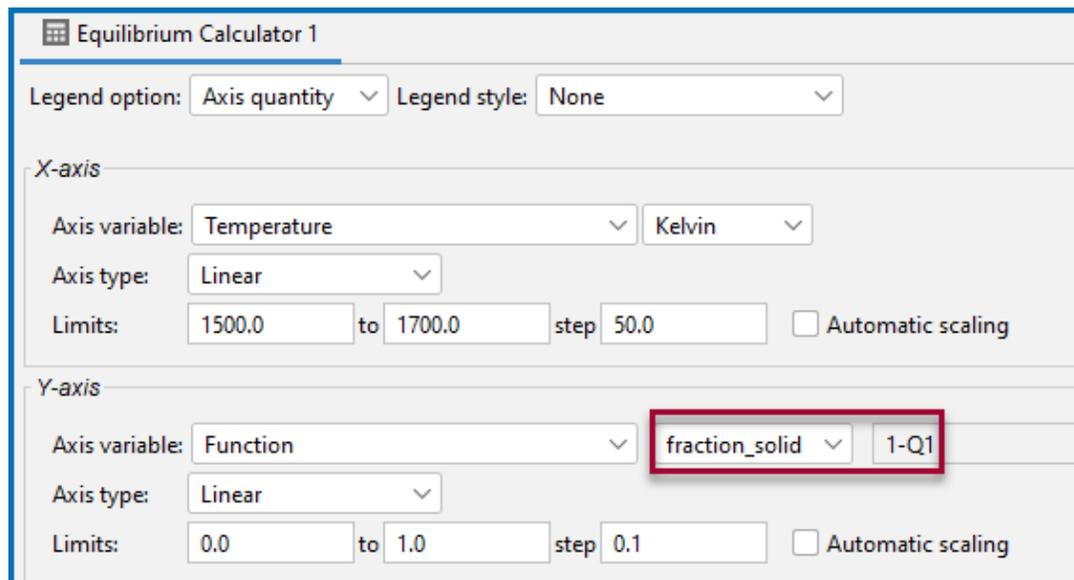


This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Thermo-Calc**
- File name: *T\_07\_User\_defined\_functions.tcu*



The screenshot shows the configuration window for the Equilibrium Calculator. The X-axis is set to Temperature in Kelvin, with a linear scale from 1500.0 to 1700.0 and a step of 50.0. The Y-axis is set to Function, with a linear scale from 0.0 to 1.0 and a step of 0.1. The function name 'fraction\_solid' and the quantity '1-Q1' are highlighted in a red box.

Axis	Variable	Type	Limits	Step	Automatic Scaling
X-axis	Temperature	Linear	1500.0 to 1700.0	50.0	<input type="checkbox"/>
Y-axis	Function	Linear	0.0 to 1.0	0.1	<input type="checkbox"/>

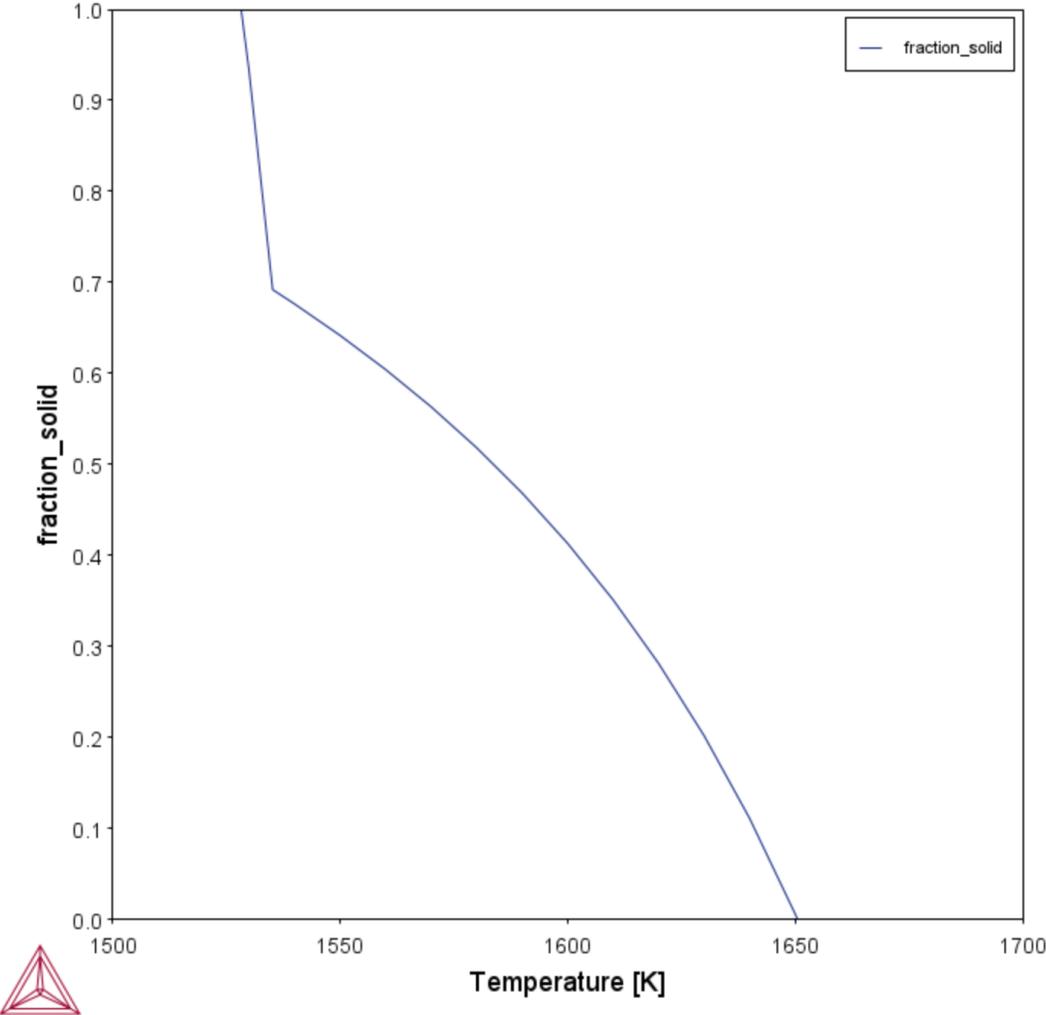


Figure 7: The result of the fraction\_solid function.

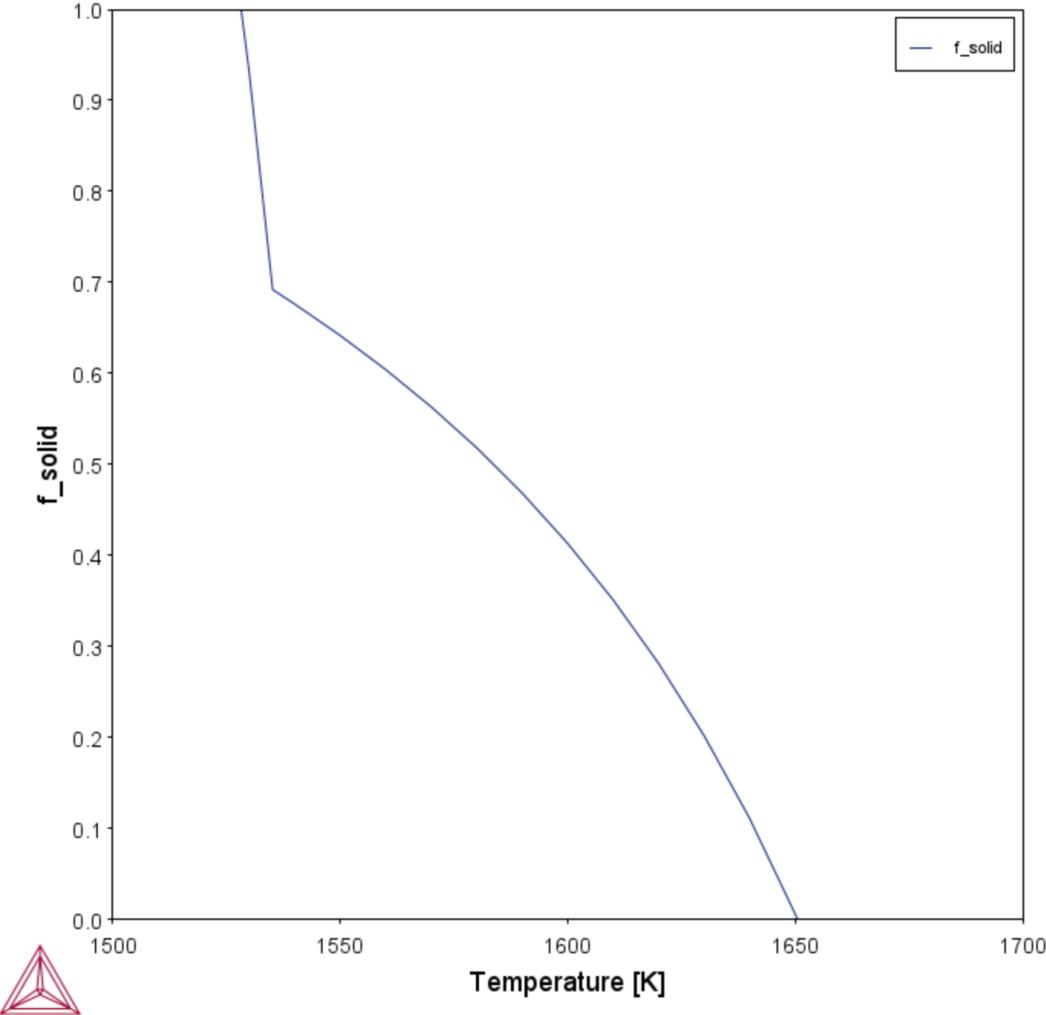


Figure 8: The result of the  $f_{solid}$  function.

---

## T\_08: Scheil and Equilibrium Solidification

---

This example shows a comparison for an Al-Si alloy solidified under full local equilibrium and under the Scheil assumptions, i.e. zero diffusion in the solidified material and infinitely fast diffusion in the liquid. The example uses the Scheil Calculator and adds a Plot and Table Renderer.



This example is included as a tutorial on our [website](#) and as part of the Scheil Solidification Simulations playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Thermo-Calc**
  - File name: *T\_08\_Scheil\_and\_equilibrium\_solidification.tcu*
-

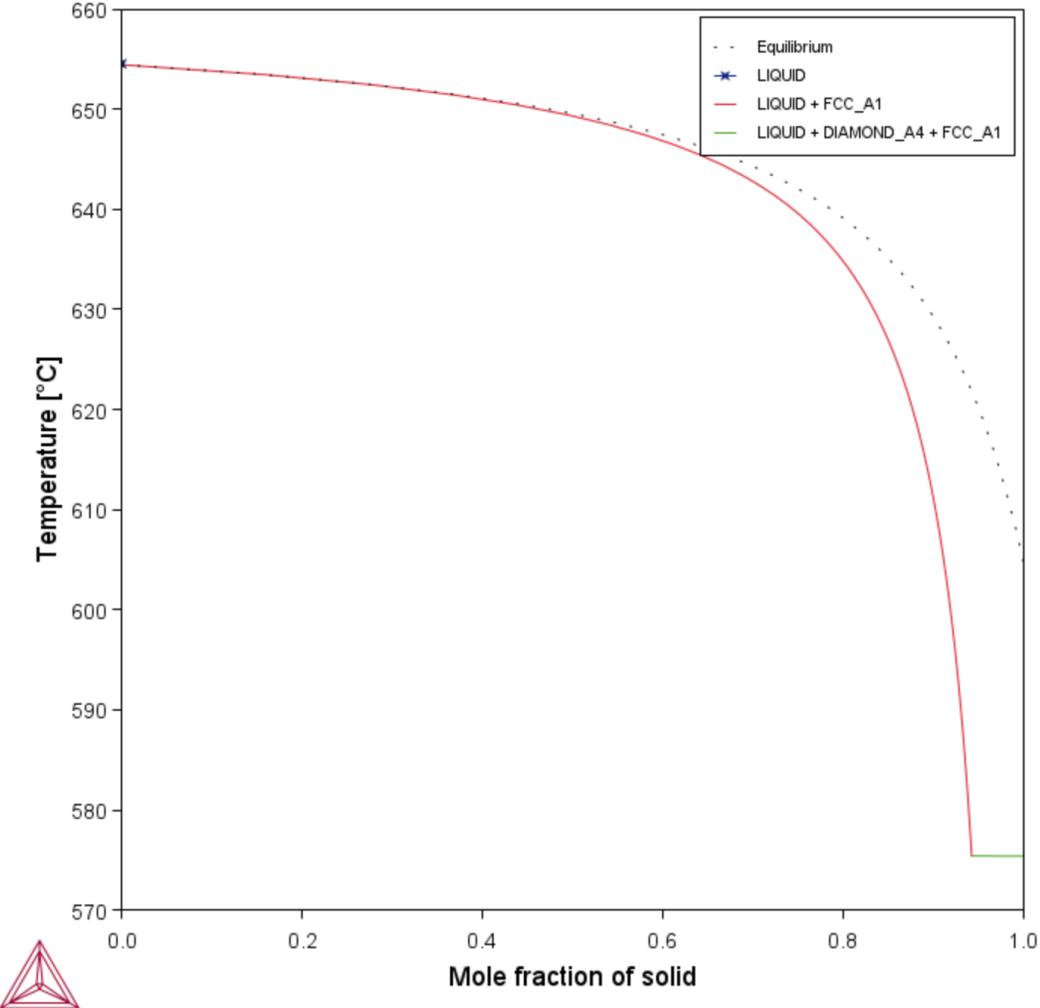
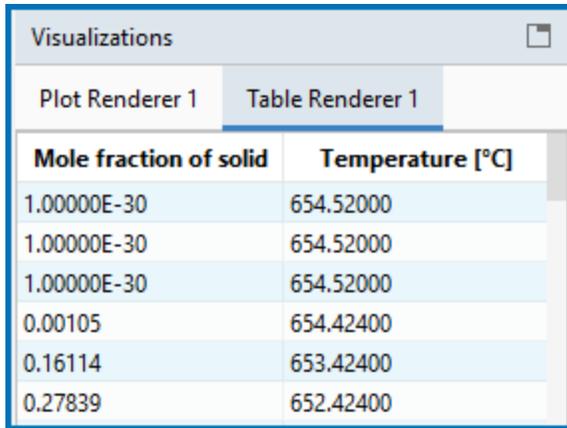


Figure 9: The plot compares the mole fraction of a solid vs temperature.



Mole fraction of solid	Temperature [°C]
1.00000E-30	654.52000
1.00000E-30	654.52000
1.00000E-30	654.52000
0.00105	654.42400
0.16114	653.42400
0.27839	652.42400

Figure 10: The table provides the data, which you can also export.

## To Save the Table (via Table Renderer)

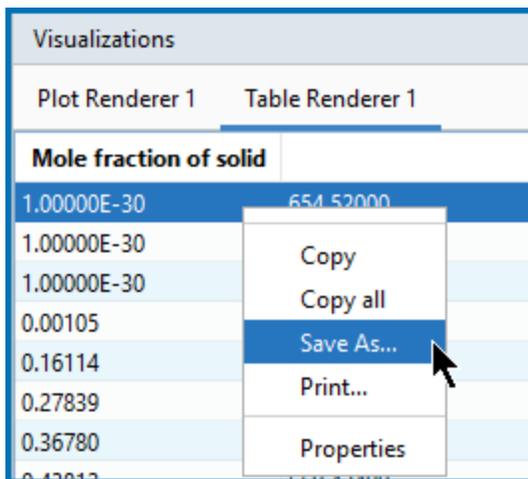
There are different ways to save the table from the Table Renderer **Visualizations** or **Configuration** windows.



For the Plot Renderer, there is an option to use the Table View  button to convert plots to table data for all calculation types and calculators. See [Plot Renderer: Convert a Plot to a Table and Export the Data](#).

### Method 1

In the **Visualizations** window, right click the table and select **Save As...**

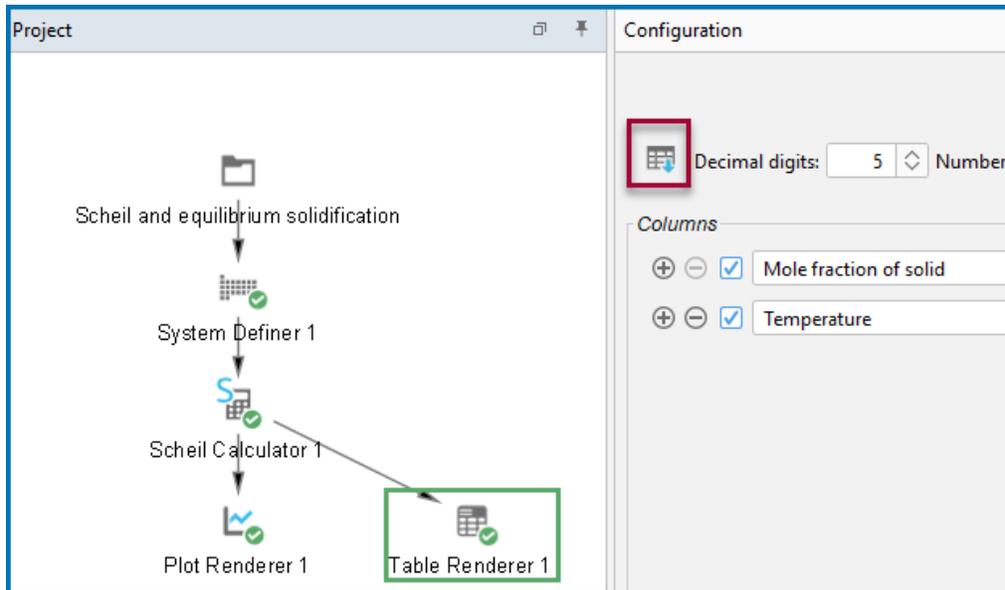


Mole fraction of solid	Temperature [°C]
1.00000E-30	654.52000
1.00000E-30	654.52000
1.00000E-30	654.52000
0.00105	654.42400
0.16114	653.42400
0.27839	652.42400
0.36780	651.42400
0.43812	650.42400

- Copy
- Copy all
- Save As...
- Print...
- Properties

## Method 2

1. In the **Project** window, click the **Table Renderer**.
2. In the **Configuration** window, click the **Save table** button .



---

## T\_09: Carbide Driving Force Heat Map

---

This is an example of using **Grid** calculations to plot the driving force for a carbide as a function of two composition variables. With the **Grid Calculation Type**, a 2D grid is generated from the two calculation axes. After the calculation is done, an equilibrium is calculated in each grid point. A Plot Renderer connected to a grid calculation plots the Z-axis property for each equilibrium as a function of the two calculation axes. You can choose different *Plot types* such as **Heat map** and **Contour** to display the results.



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Thermo-Calc**
  - File name: *T\_09\_Heat\_map\_of\_carbide\_driving\_force.tcu*
-

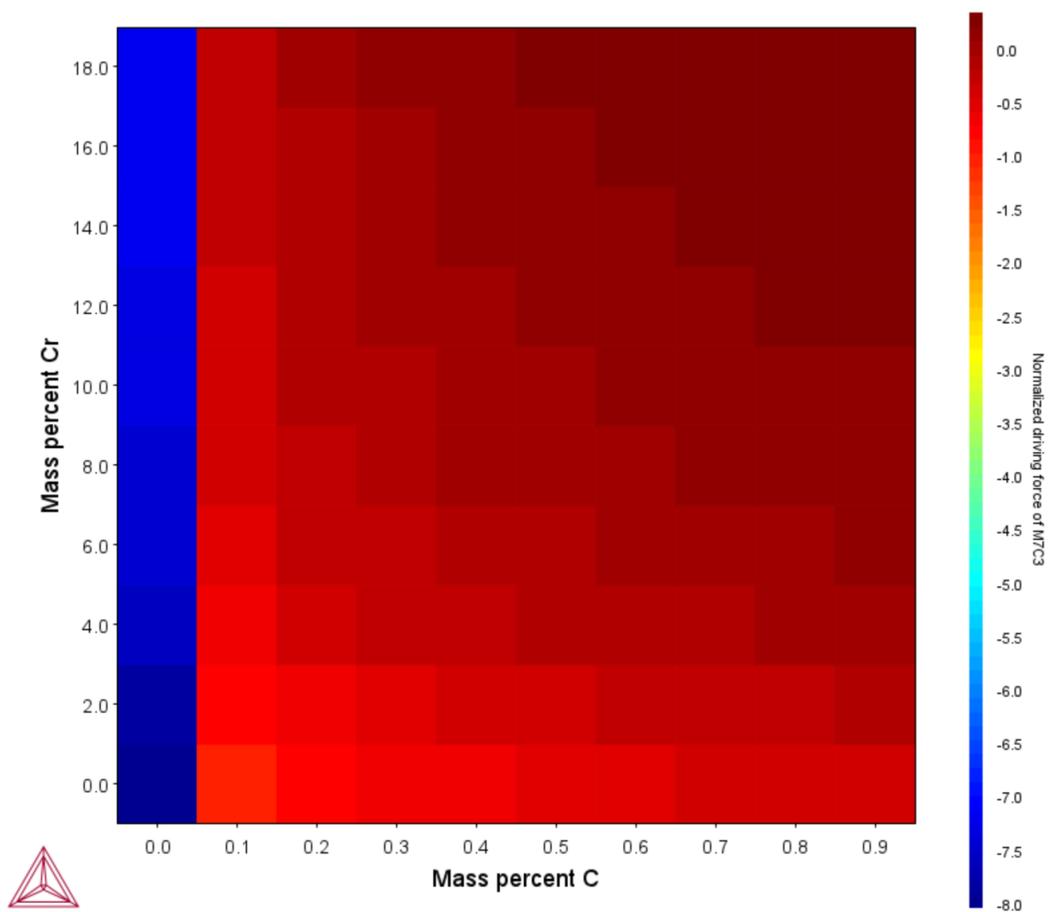


Figure 11: The results are plotted as a Heat map. Alternatively, a Contour Plot type can be selected on the Plot Renderer.

## T\_10: Scheil Solidification with Back Diffusion

---

This is an example of a Scheil solidification simulation of a Al-2.1Cu-1Si alloy including back diffusion in the primary phase.

The example uses two of the Scheil Calculator activity nodes with different cooling rates, 10 K/s and 0.005 K/s. The lower cooling rate produces a solidification curve that is closer to the equilibrium curve as shown in the plot result. Both a thermodynamic ALDEMO (aluminum demo) and mobility MALDEMO (Al-alloys mobility) database are used for this calculation.



The Scheil with back diffusion feature is only available for systems with diffusion data, i.e. this model requires the use of a mobility database.



See [Scheil Calculator](#) for an overview of the different Scheil simulation models available.



This example is included as a tutorial on our [website](#) and as part of the Scheil Solidification Simulations playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Thermo-Calc**
- File name: *T\_10\_Scheil\_with\_back\_diffusion.tcu*.

Running the example itself does not require an additional license.

---

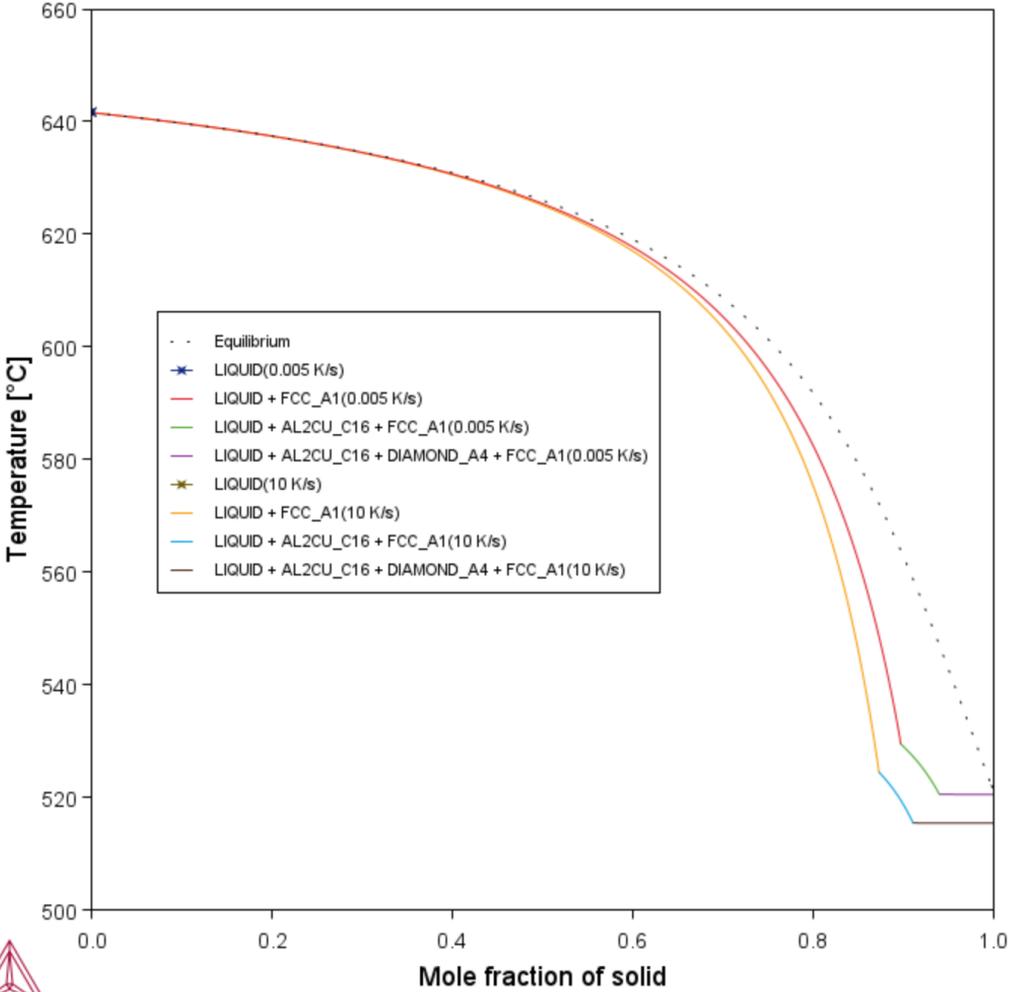


Figure 12: Scheil solidification. The lower cooling rate produces a solidification curve that is closer to the equilibrium curve .

## T\_11: Surface Tension in Cu-Zr

---

This is an example of including surface tension in the calculations.

Using the ALDEMO database, which is the free demonstration version of the TCS AI-based Alloy Database (TCAL), the surface tension of liquid metallic is plotted at 1373 K for Cu-Zr and compared to experimental data from [2005Kra].

### Reference

[2005Kra] V. P. Krasovskyy, Y. V. Naidich, N. A. Krasovskaya, "Surface tension and density of copper–Zirconium alloys in contact with fluoride refractories," J. Mater. Sci. 40, 2367–2369 (2005).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Thermo-Calc**
- File name: *T\_11\_Surface\_tension\_in\_Cu-Zr.tcu*



The same example is also provided in Console Mode as `tcex56`.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

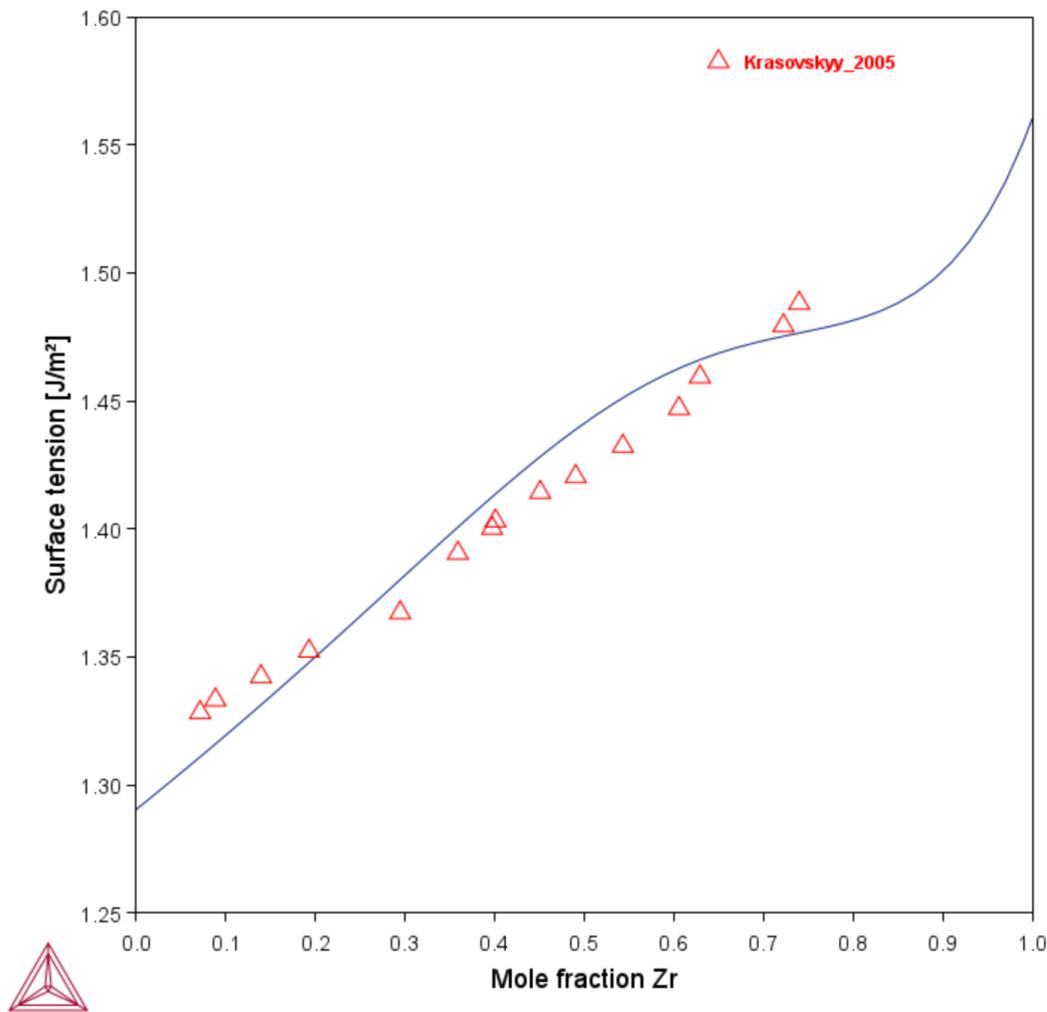


Figure 13: Surface tension of liquid metallic at 1373 K for Cu-Zr and compared to experimental data from [2005Kra].

### More Information

The thermophysical properties are progressively being added to the Thermo-Calc databases starting with Thermo-Calc software version 2020a.



You can learn more about the models by searching the help (press F1 when in Thermo-Calc).



You can find information on our website about the [properties that can be calculated](#) with Thermo-Calc and the Add-on Modules.

---

## T\_12: Viscosity in Cr-Ni

---

This is an example of including viscosity in the calculations.

Using the FEDEMO database, which is the free demonstration version of the TCS Steel and Fe-alloys Database (TCFE), the viscosity of metallic liquids is plotted at 1873 K for Cu-Ni and compared to experimental data from [2005Sat].

### **Reference**

[2005Sat] Y. Sato, K. Sugisawa, D. Aoki, T. Yamamura, "Viscosities of Fe–Ni, Fe–Co and Ni–Co binary melts," Meas. Sci. Technol. 16, 363–371 (2005).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Thermo-Calc**
- File name: *T\_12\_Viscosity\_in\_Cr-Ni.tcu*



The same example is also provided in Console Mode as `tcex55`.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

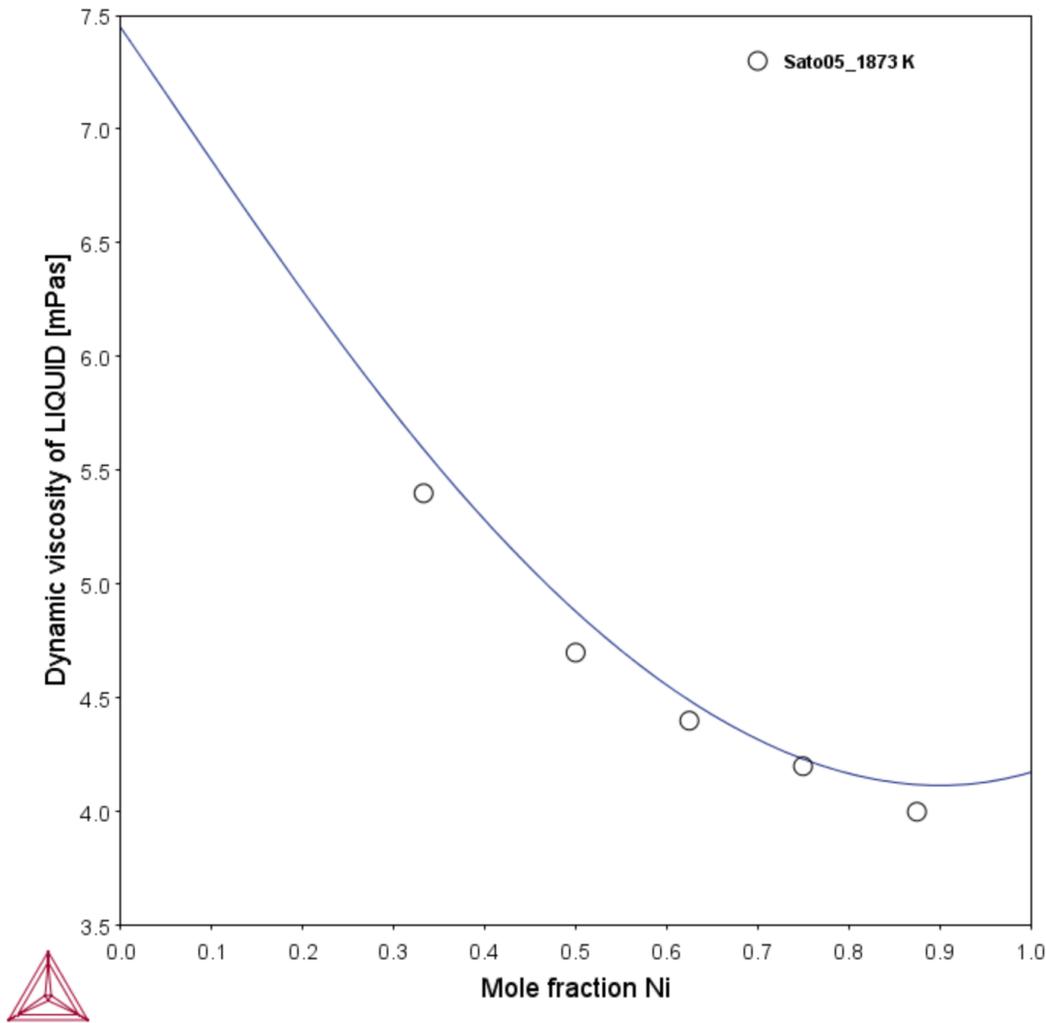


Figure 14: The viscosity of metallic liquids at 1873 K for Cu-Ni and compared to experimental data from [2005Sat].

### More Information

The thermophysical properties are progressively being added to the Thermo-Calc databases starting with Thermo-Calc software version 2020a.



You can learn more about the models by searching the help (press F1 when in Thermo-Calc).



You can find information on our website about the [properties that can be calculated](#) with Thermo-Calc and the Add-on Modules.

---

---

## T\_13: Scheil Solidification with Solute Trapping

---

In Graphical Mode, you use the **Scheil Calculator** to set up the various Scheil solidification simulations outlined in more detail in [Scheil-Gulliver Solidification Calculations](#).

This example shows the use of the *Scheil with solute trapping* option with an Al-7.5Si-0.2Cu alloy as compared to a Classic Scheil calculation. The thermodynamic ALDEMO (Aluminum demo) database, available to all users, is selected for this calculation to simulate the effect of solute trapping in the primary phase. This type of simulation is useful for additive manufacturing applications.

This example shows how the alloy starts to solidify at a lower temperature compared to a classic Scheil simulation, an effect which increases with increasing solidification speed. The solidification speed is calculated from user supplied scanning speed and angle between the solid/liquid interface and the scanning direction.



This example is included as a tutorial on our [website](#) and as part of the Scheil Solidification Simulations playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Thermo-Calc**
  - File name: *T\_13\_Scheil\_with\_Solute\_Trapping.tcu*
-

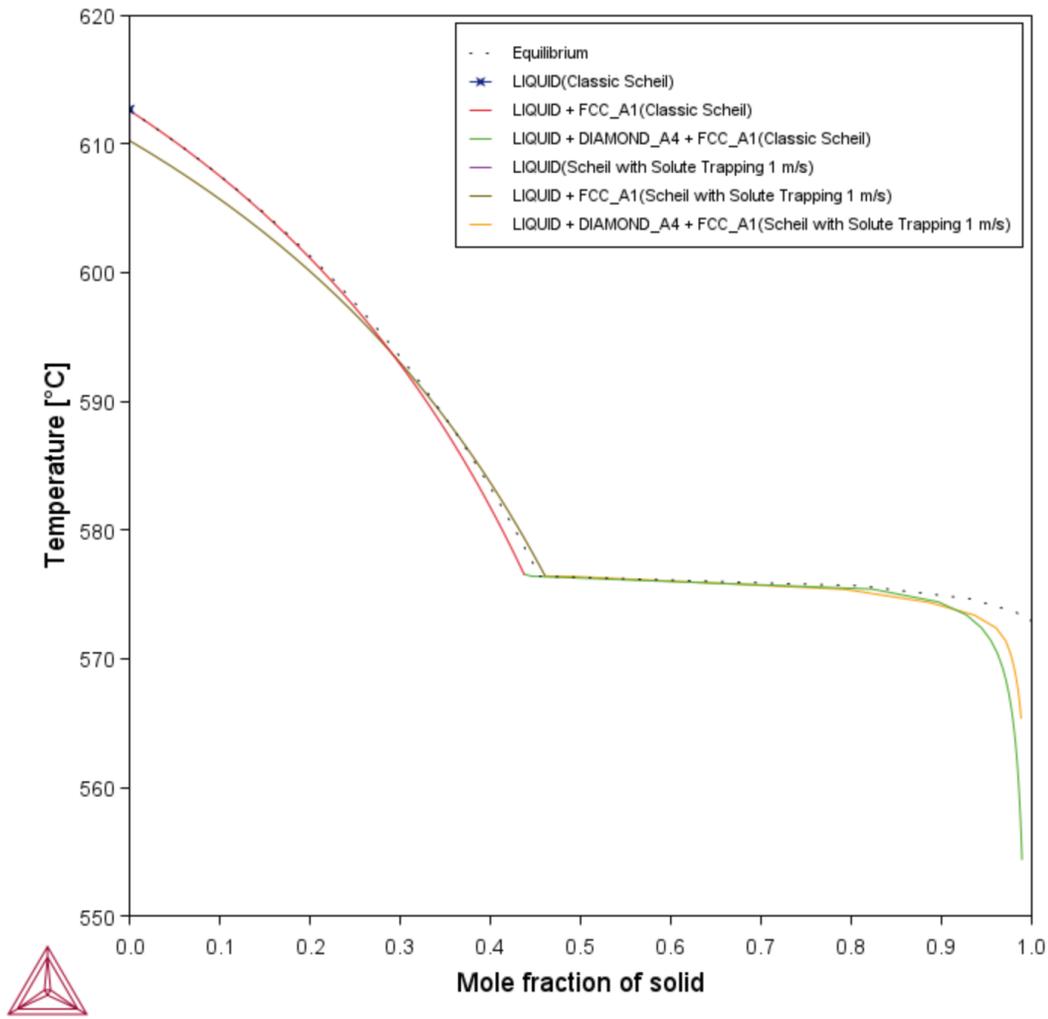


Figure 15: The effect of Scheil solidification with solute trapping of a Al-7.5Si-0.2Cu alloy compared to equilibrium and a Classic Scheil calculation.

## T\_14: Fe-Cr-Ni Transition Comparison Using the Material to Material Calculator

---

In Graphical Mode, you use the **Material to Material Calculator** to perform calculations and examine how two materials transition from one into the other. The calculations in this example search for potential deleterious phases during heat treatment of alloys joined together. In this example, the calculation steps between a model martensitic stainless steel (Fe-17Cr-2Ni) and a model Alloy 800 composition (Fe-35Ni-19Cr). Neither alloy is predicted to form the deleterious sigma phase at 600 °C. However sigma is predicted to form at a wide range of mixtures of these two materials, which could have an impact on mechanical properties if it forms.

The example, as defined when you open it, uses a **One Axis** calculation (also known as a property diagram), a temperature of 650 °C, to compare the volume fraction of all phases to the mass fraction of Alloy 800. The **Fraction of the second material** (Alloy 800) is set to 50% (0.5), which for a **One Axis** calculation is used as the start value.



As with all the examples, you can adjust the settings to plot a variety of combinations of output to see how the Material to Material Calculator works before defining your own project.

The plot for this calculation compares the volume fraction of all phases of Material 1 (martensitic stainless steel) to the mass fraction of Material 2 (Alloy 800). In the plots you can see where each phase (FCC\_A1, SIGMA\_D8B, and BCC\_A2) evolves. As you step from Material 1 to Material 2, the FCC\_A1 phase starts forming at about 5% (0.05), SIGMA-D8B phase starts to evolve at about 19% (0.19), and the BCC\_A2 transitions from Material 1 to Material 2 where it stops forming at approximately 24% (0.24). The purpose of the calculation is to determine at what fraction of Alloy 800 the deleterious sigma phase forms and this is shown for these compositions.



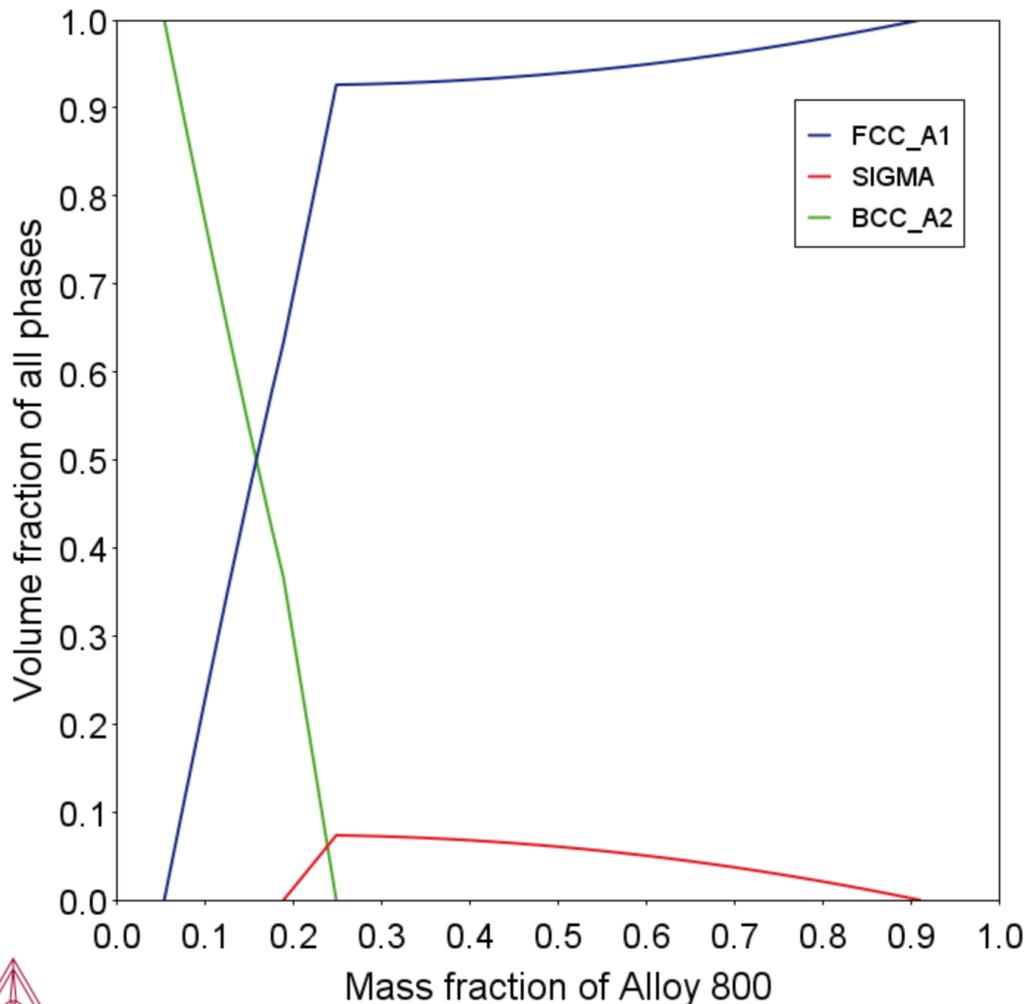
This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

### **Project File Information**

- Folder: **Thermo-Calc**
- File name: *T\_14\_Fe-Cr-Ni\_Material\_to\_Material.tcu*

The default plot shows the volume fraction of all phases as a function of the mass fraction of the second material, which in this case, is Alloy 800. You can see that the martensitic stainless composition is BCC at this temperature, and the highly alloyed stainless is FCC. Neither material is expected to form any deleterious phase on its own at 650 °C.

However, mixtures of these two materials, which could arise from a graded transition joint, weld, or potentially a diffusion couple, show some stability for the deleterious sigma phase at this temperature. You can try to make small adjustments to the initial chemistry to reduce the potential for sigma phase to form.



*Figure 16: This plot shows the volume fraction of phases when changing composition from the Martensitic Steel (First material) to the Alloy 800 (Second material). This calculation uses the Material to Material Calculator to examine the transition of phases for two compositions of Fe-Cr-Ni. The purpose is to determine at what fraction of Alloy 800 does the deleterious sigma phase start to form. See the text for details.*

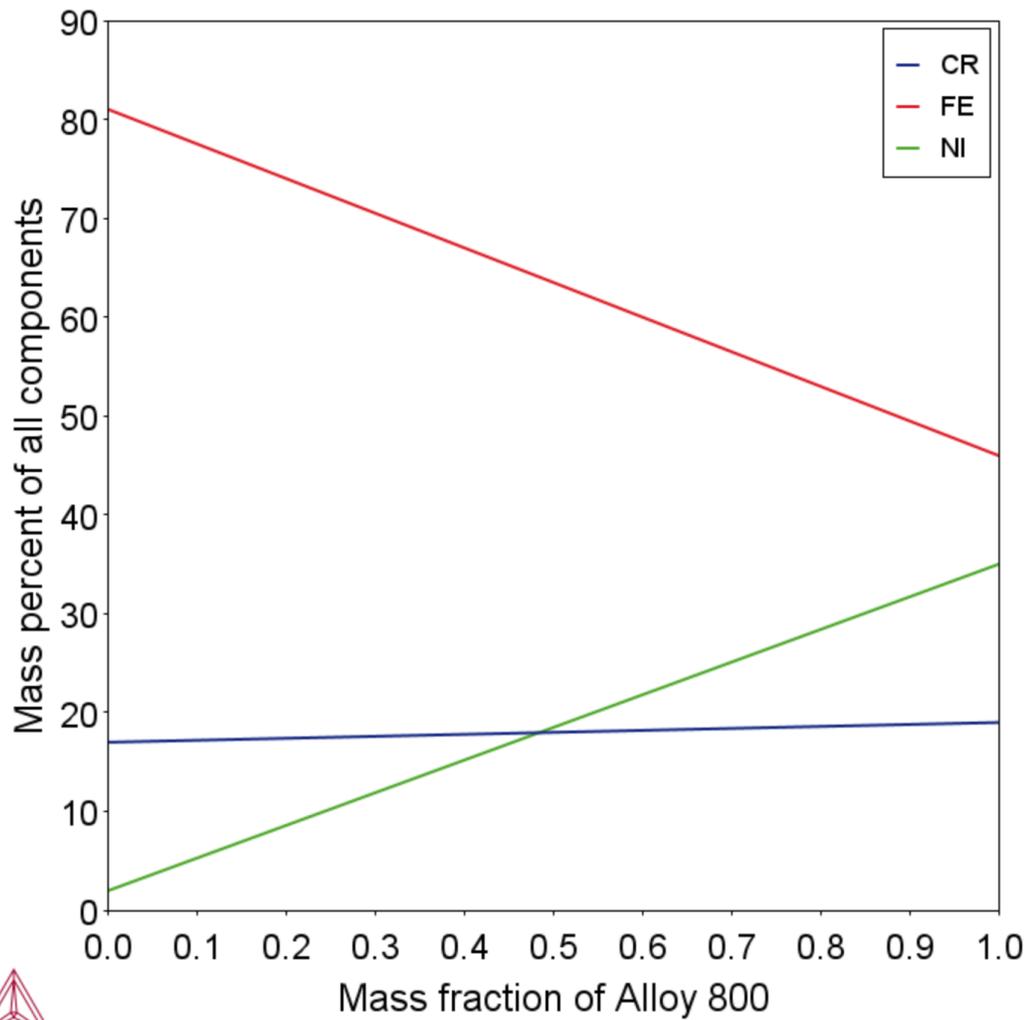


Figure 17: The composition varies linearly when the composition is changed from the Martensitic steel to the Alloy 800 using the Material to Material Calculator. See the text for details.

## T\_15: Molar Volume and Thermal Expansion Coefficients for L12-type Al<sub>3</sub>Sc and Al<sub>3</sub>Zr Compounds

---

This example calculates volumetric thermal expansion coefficients of the L12-type Al<sub>3</sub>Sc and the L12-type Al<sub>3</sub>Zr, respectively. Both Al<sub>3</sub>Zr and Al<sub>3</sub>Sc are modeled as the same phase (named as AL3SC, since Al<sub>3</sub>Sc is stable while Al<sub>3</sub>Zr is metastable).

The example set up uses two System Definers and Equilibrium Calculators to plot and compare the molar volume and thermal expansion coefficients of the Al<sub>3</sub>Sc and Al<sub>3</sub>Zr binary compounds. It uses the ALDEMO database, which is the free demonstration version of the TCS Al-based Alloy Database (TCAL) and an Experimental File Reader node to read the data from [2015Sah].

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Thermo-Calc**
- File name: *T\_15\_Molar\_Volume\_and\_Thermal\_Expansion\_Coefficient.tcu*

The example includes an experimental data (\*.exp) file called `Al3X.exp`.



The same example is also provided in Console Mode as two separate examples: `tcex58` and `tcex59`.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

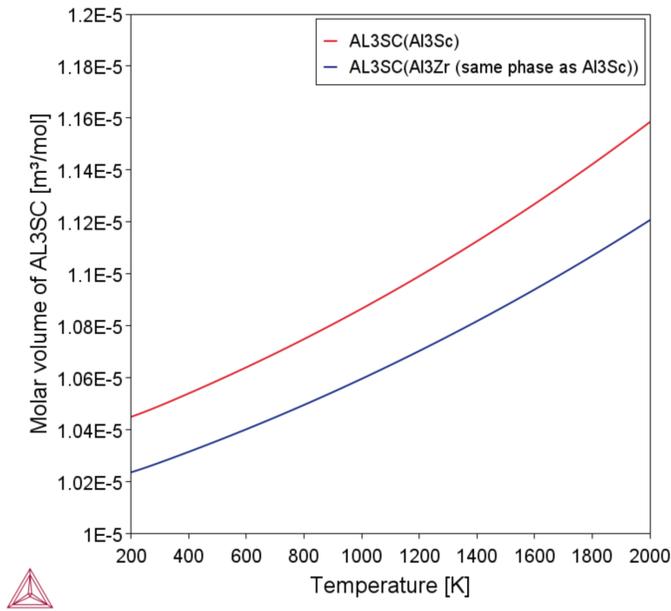


Figure 18: Compares the molar volume of both Al3Zr and Al3Sc, which are modeled as the same phase (named as AL3SC, since Al3Sc is stable while Al3Zr is metastable).

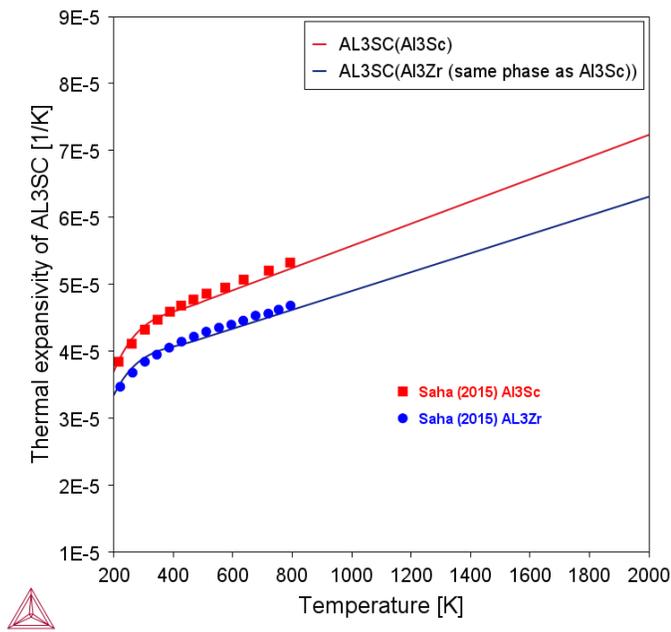


Figure 19: Compares volumetric thermal expansion coefficients of the L12-type Al3Sc and the L12-type Al3Zr using theoretical data with quasiharmonic approximation from [2015Sah].

## Reference

[2015Sah] S. Saha, T. Z. Todorova, J. W. Zwanziger, Temperature dependent lattice misfit and coherency of Al<sub>3</sub>X (X=Sc, Zr, Ti and Nb) particles in an Al matrix. Acta Mater. 89, 109–115 (2015).

## More Information

The thermophysical properties are progressively being added to the Thermo-Calc databases starting with Thermo-Calc software version 2020a.



You can learn more about the models by searching the help (press F1 when in Thermo-Calc).



You can find information on our website about the [properties that can be calculated](#) with Thermo-Calc and the Add-on Modules.

## T\_16: Electrical Resistivity and Thermal Conductivity for Pure Copper

---

This example calculates the electrical resistivity (ELRS) and thermal conductivity (THCD) of pure Cu. It makes a *One Axis* (step) calculation over a wide temperature range, covering both the FCC\_A1 state and the liquid state. It plots thermal conductivity of the system (both FCC\_A1 and liquid) and that of a single phase (taking FCC\_A1 as an example), respectively. Also it plots electrical resistivity of the system and that of FCC\_A1.

The example set up uses an Equilibrium Calculator combined with four Experimental File Reader nodes that take data from the included 1972Ho\_cu\_thcd.exp and 1981Ho\_cu\_elrs.exp files.

The example is available to all users as it uses the ALDEMO database, which is the free demonstration version of the TCS AI-based Alloy Database (TCAL).



For more advanced examples using the TCAL database, see [PM\\_G\\_10: Freeze-in Thermal Conductivity](#) and [PM\\_G\\_11: Freeze-in Electrical Resistivity](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Thermo-Calc**
- File name: *T\_16\_Electrical\_Resistivity\_Thermal\_Conductivity\_Cu.tcu*
- Experimental data files included 1972Ho\_cu\_thcd.exp and 1981Ho\_cu\_elrs.exp



The same example is also provided in Console Mode as tcex60.



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

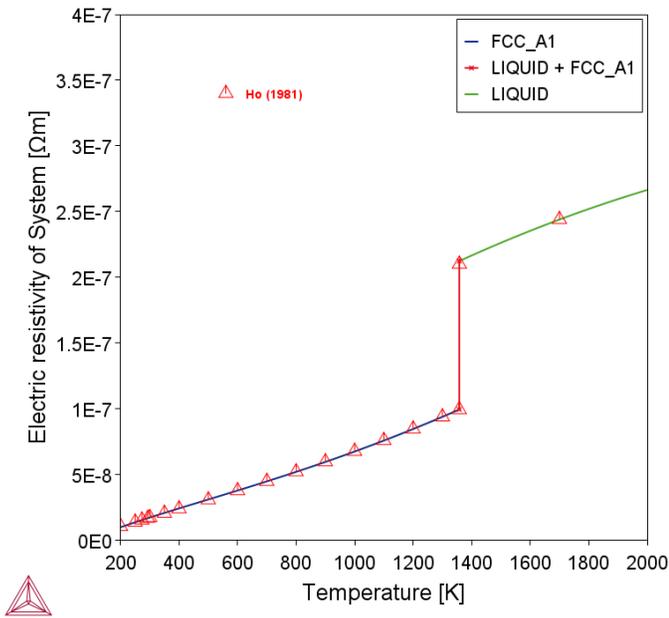


Figure 20: The electrical resistivity (ELRS) of pure Cu over a wide temperature range, covering both the FCC\_A1 state and the liquid state. The data was recommended by [1981Ho].

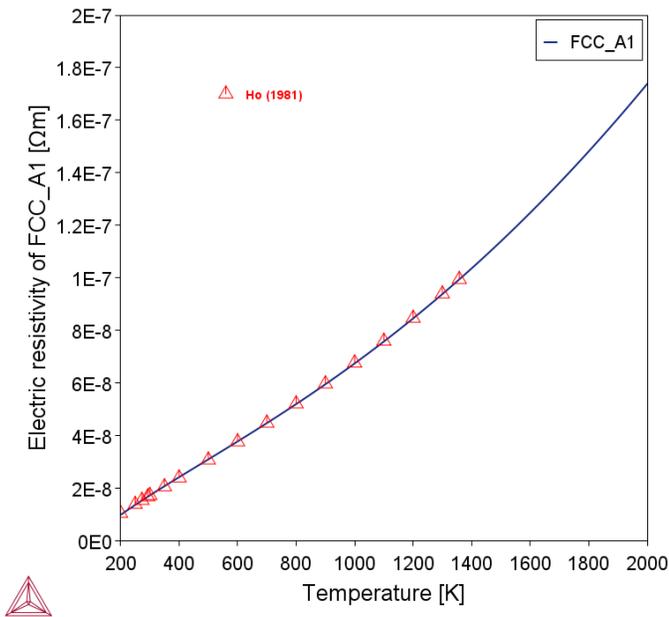


Figure 21: The electrical resistivity (ELRS) of a single phase (FCC\_A1) vs. temperature. The data was recommended by [1981Ho].

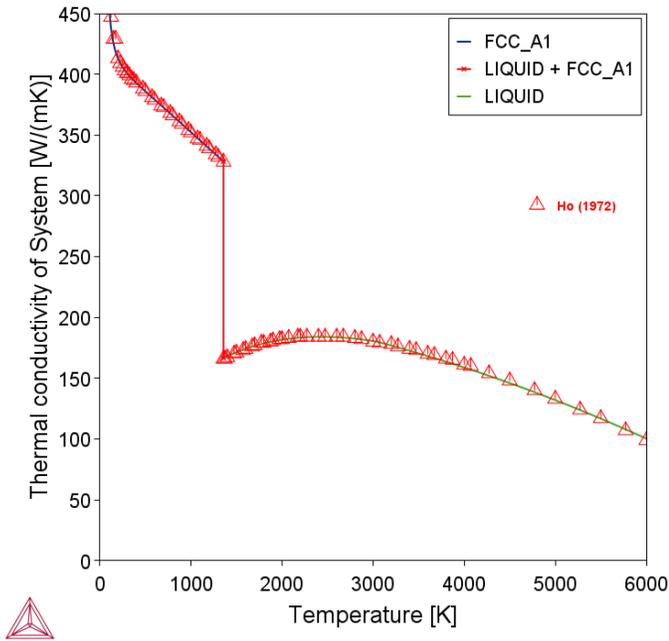


Figure 22: The thermal conductivity of pure Cu over a wide temperature range, covering both the FCC\_A1 state and the liquid state. The data was recommended by [1972Ho].

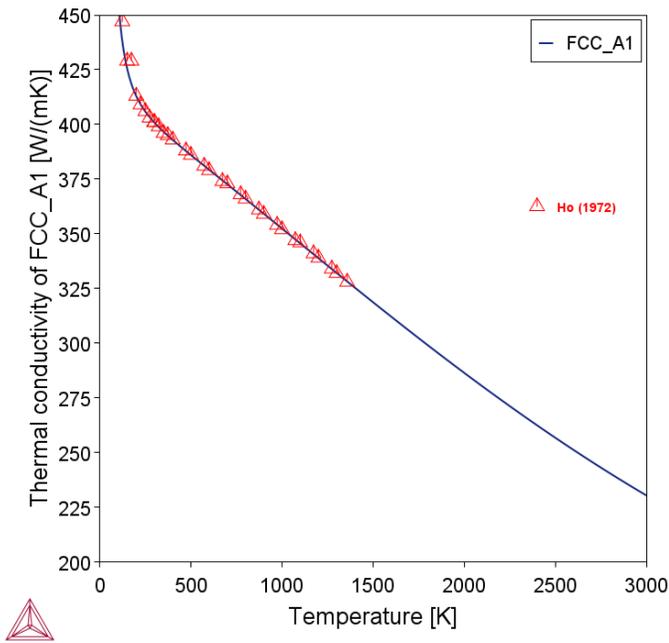


Figure 23: The thermal conductivity (THCD) of a single phase (FCC\_A1) vs. temperature. The data was recommended by [1972Ho].

## References

[1972Ho] C. Y. Ho, R. W. Powell, P. E. Liley, Thermal Conductivity of the Elements. J. Phys. Chem. Ref. Data. 1, 279–421 (1972).

[1981Ho] C. Y. Ho, M. W. Ackerman, K. Y. Wu, T. N. Havill, R. H. Bogaard, R. A. Matula, S. G. Oh, and H. M. James. Electrical resistivity of ten selected binary alloy systems. CINDAS report 59, for Office of standard reference data National Bureau of Standards, Department of Commerce (1981).

## More Information

The thermophysical properties are progressively being added to the Thermo-Calc databases starting with Thermo-Calc software version 2020a.



You can learn more about the models by searching the help (press F1 when in Thermo-Calc).



You can find information on our website about the [properties that can be calculated](#) with Thermo-Calc and the Add-on Modules.

## T\_17: Al<sub>2</sub>O<sub>3</sub>-MgO Phase Diagram

---

Al<sub>2</sub>O<sub>3</sub>-MgO is one of the basic systems to understand metallurgical slags and refractories. This example uses an Equilibrium Calculator to calculate a phase diagram of the pseudo-binary system Al<sub>2</sub>O<sub>3</sub>-MgO. It also demonstrates how to change components in the System Definer. In this case, the components Al<sub>2</sub>O<sub>3</sub> and MgO are defined instead of the elements Al, Mg, and O.



The TCS Metal Oxide Solutions Database (TCOX) is used in this example. A valid license for version 11 (TCOX11) or newer is required to run the example.

This example is also included in the *TCS Metal Oxide Solutions Database (TCOX) Examples Collection*.



There is a similar example provided in Console Mode that uses the OXDEMO database, which is available to all users. See `tcex17`.



Go to the [Metal Slag and Oxides Database](#) page on our website where you can access an examples collection and the technical information plus learn about its many applications with the [Process Metallurgy Module](#). Also explore further applications of Thermo-Calc to [Refractory Oxides](#) and [Slags](#) including links to resources such as examples, publications, and more.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Thermo-Calc**
- File name: `T_17_Al2O3-MgO_phase_diagram.tcu`



This example is included as a tutorial on our [website](#) and as part of the Graphical Mode Examples playlist on our [YouTube channel](#).

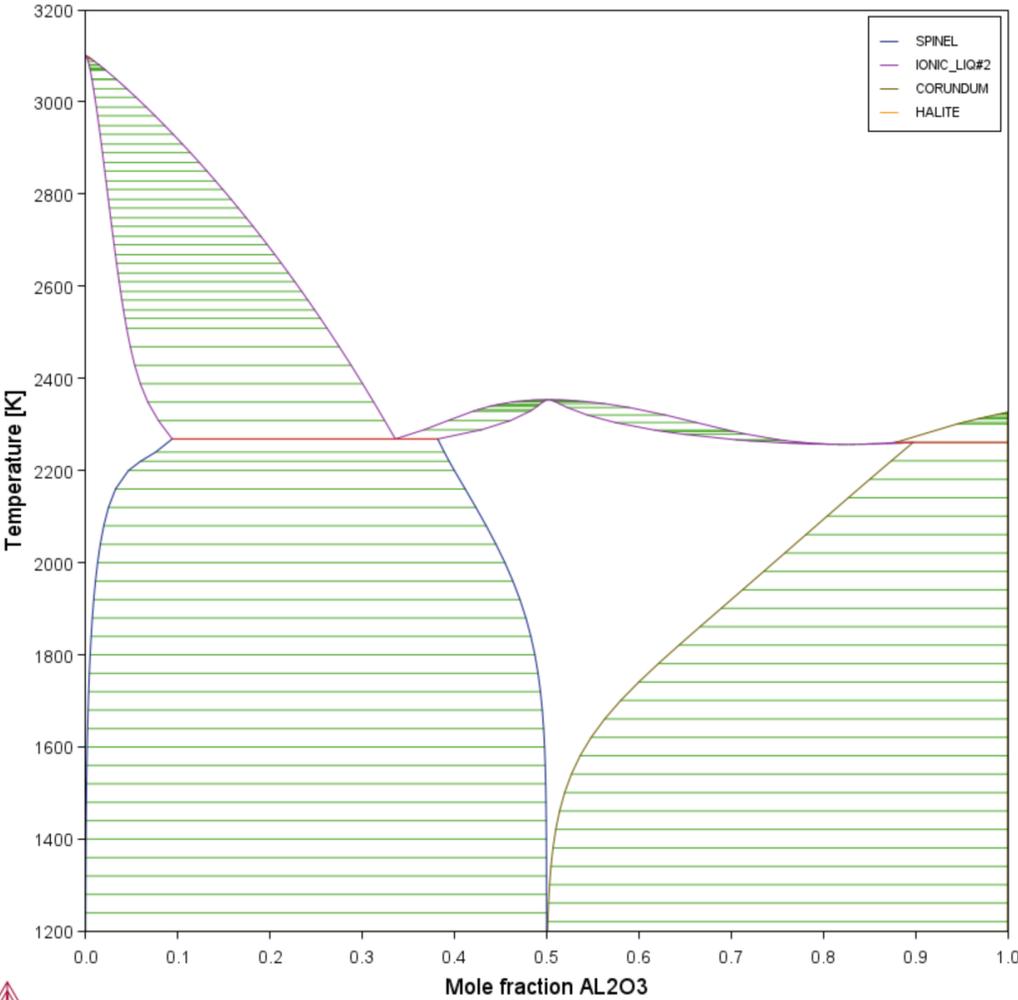


Figure 24: Phase diagram of the pseudo-binary system  $Al_2O_3$ - $MgO$ .

## T\_18: Scheil with Delta Ferrite to Austenite Transition

---

This is an example of a Scheil solidification simulation using an Fe-0.23C-0.19Si-0.47Mn-0.92Cr-0.09Ni-0.3Mo (wt%) alloy (Steel P) [2015Sch], which is a steel that undergoes the peritectic transformation.

The result of a Scheil Calculator with the default **Classic Scheil** calculation type is compared to the result of a calculator where the setting **Allow delta ferrite to austenite transition in steel** is selected. Carbon ( C ) is set as a **Fast diffuser** in both. A comparison to experimental solidus temperature from Schaffnit et al. [2015Sch] shows that for this steel, using the **Allow delta ferrite to austenite transition in steel** option gives a much better agreement with experiment than the standard **Classic Scheil**.



This uses the TCFE database. A license is required to run the example.



See [Scheil Calculator](#) for an overview of the different Scheil simulation models available.

### Project File Information

- Folder: **Thermo-Calc**
- File name: *T\_18\_Scheil\_with\_Delta\_Ferrite\_to\_Austenite\_Transition.tcu*.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

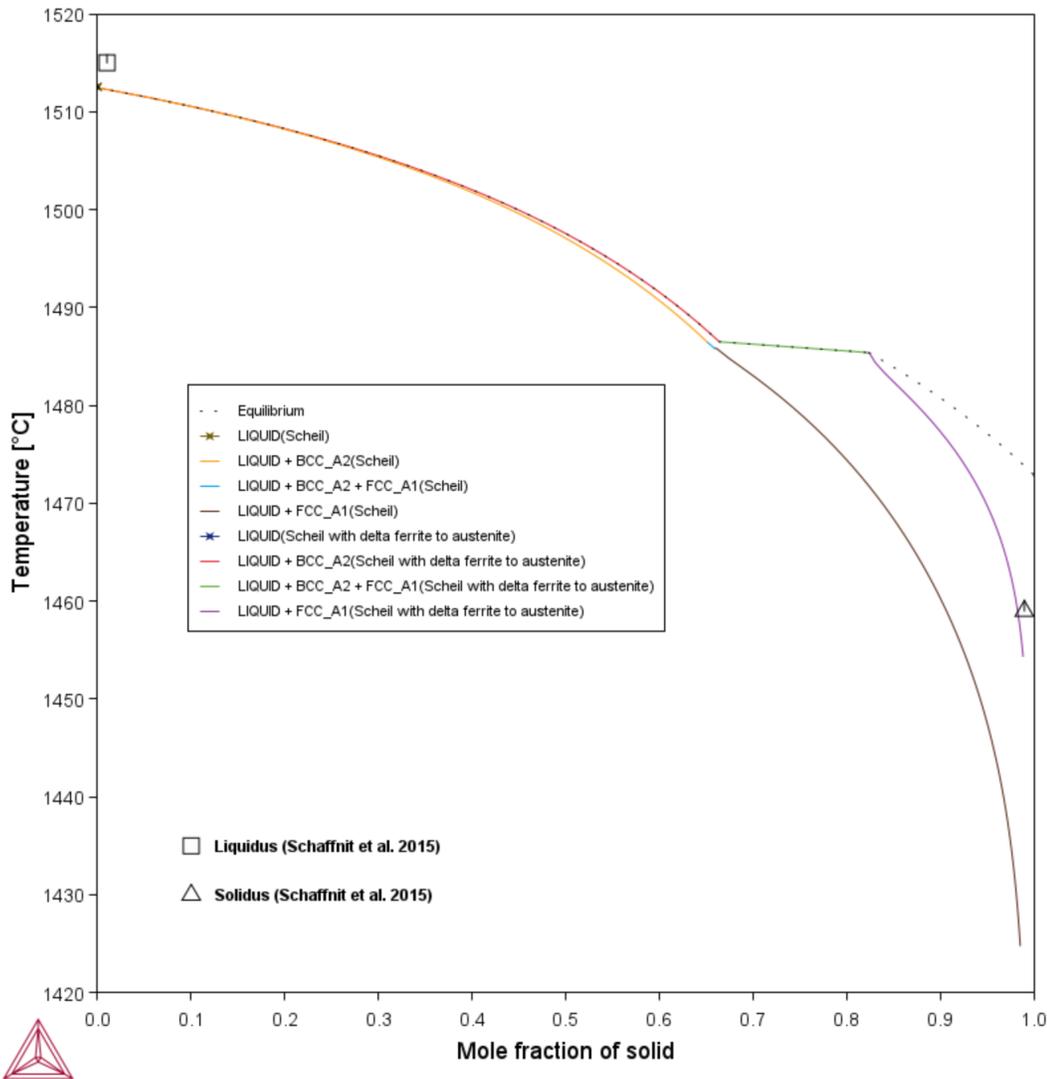


Figure 25: Mole fraction of solid as a function of temperature comparing the liquidus and solidus from experimental data [2015Sch].

## Reference

[2015Sch] P. Schaffnit, C. Stallybrass, J. Konrad, F. Stein, M. Weinberg, A Scheil–Gulliver model dedicated to the solidification of steel. *Calphad*. 48, 184–188 (2015).

# Thermo-Calc General Property Models Examples Collection

---

The **General Models** are available to all users.



To run calculations with the **Steel Models** (as part of the Steel Model Library) requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. To run calculations with the **Nickel Models** (as part of the Nickel Model Library) requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases.

These examples use the **Property Model Calculator**, an activity available with Thermo-Calc.



[About the Examples](#)

## In this section:

---

PM_G_01: Phase Transition .....	54
PM_G_02: Coarsening and Interfacial Energy .....	58
PM_G_03: Driving Force and Interfacial Energy .....	64
PM_G_04: Yield Strength .....	67
PM_G_05: Yield Strength NiAlCr .....	72
PM_G_06: Yield Strength HEA .....	74
PM_G_07: Hot Crack Susceptibility .....	76
PM_G_08: Spinodal .....	78
PM_G_09: T-Zero Temperature .....	80
PM_G_10: Freeze-in Thermal Conductivity .....	82
PM_G_11: Freeze-in Electrical Resistivity .....	84
PM_G_12: Solidus and Liquidus Batch Calculation .....	86
PM_G_13: Ti Alloy Design for Additive Manufacturing .....	89
PM_G_14: Ti_Fe_T-Zero_Martensite .....	92

---

---

PM_G_15: Columnar to Equiaxed Transition (CET) of a NiAlCr Alloy .....	94
PM_G_16: Columnar to Equiaxed Transition of a CMSX-4 Alloy .....	97
PM_G_17: Columnar to Equiaxed Transition of an IN718 Alloy .....	101

---

## PM\_G\_01: Phase Transition

---

The example uses the **Property Model Calculator** to predict the transition temperature to the unwanted brittle sigma phase. The example shows how the temperature is influenced by changes to a steel alloy's composition using the **Uncertainty Calculation Type** and then you can choose to use either a **Histogram** (frequency diagram) or **Probability** plot after choosing the **Statistical Plot type** on the **Plot Renderer**.



This example is included as a Property Model tutorial on our [website](#) and as part of the Property Model Calculator playlist on our [YouTube channel](#).



[Phase Transition Property Model Settings](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Property Models** → **General**
- File name: *PM\_G\_01\_Phase\_Transition.tcu*

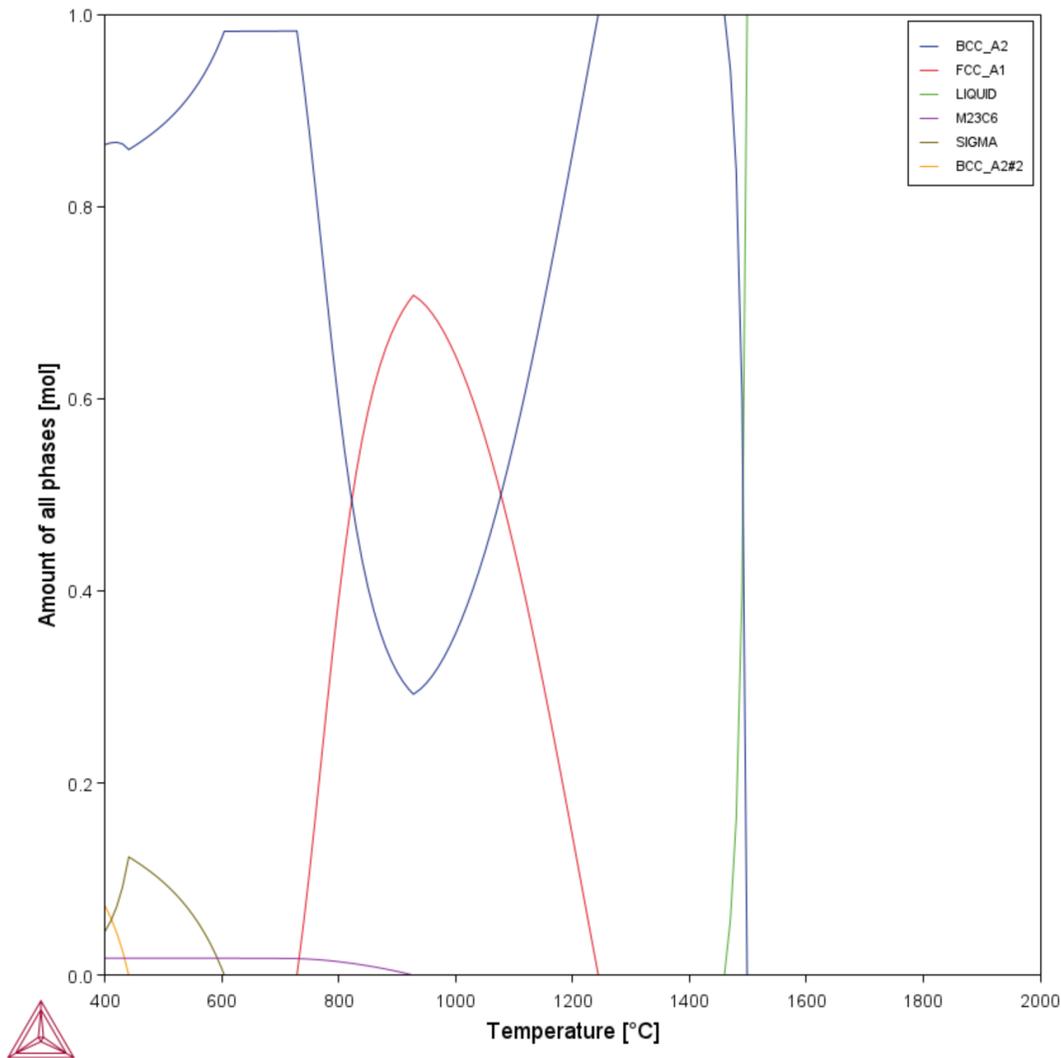


Figure 1: A property diagram as a result of using the Equilibrium Calculator with a One Axis calculation to determine a good starting temperature of the SIGMA phase.

Once you set up the Property Model Calculator, but before running a **Grid** calculation, it is recommended you run a **Single** calculation to make sure the calculation is valid.

- If the subprocess worked, it displays in the **Event Log**. The **Event Log** also has the result `Relaxed condition`, referring to the variable, which in this case is temperature.
- If the calculation did not work, `NaN` (not a number) displays instead. Check your configuration set up and run the calculation again before moving on to the more complicated calculation.



## About the Uncertainty Calculations

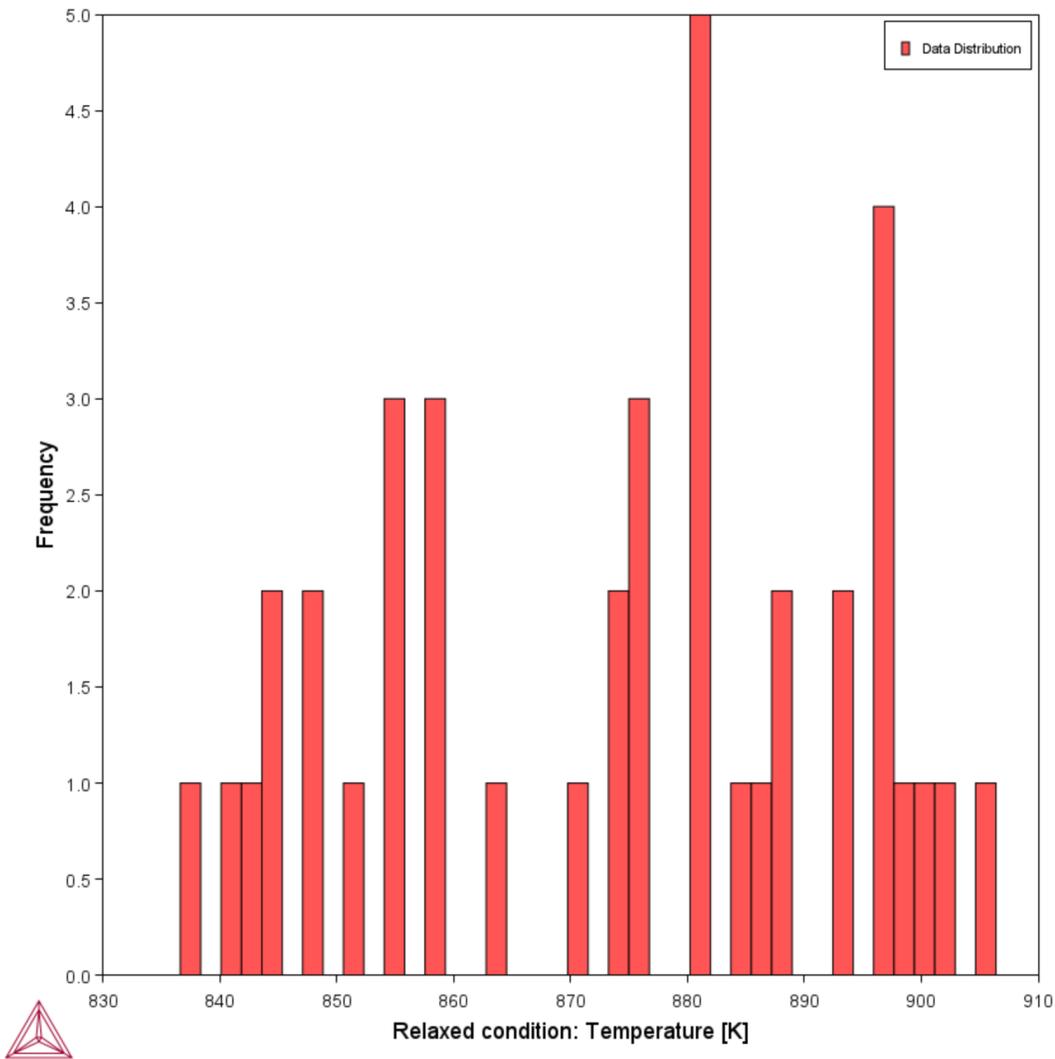


Figure 2: Using the Property Model Calculator with the Phase Transition Model to plot a histogram showing how the transition temperature of sigma varies when you vary the composition. The plot changes each time you run the calculation because it is taking a random sampling from the composition range set.



The X-axis is labeled the **Relaxed Condition**, which you can see on the **Configuration** window it is **Temperature** for the **Condition to Vary** setting. The Y-axis shows the **Frequency**, which is the number of samples out of 40 where the composition transitions at each temperature. You can see that the transitions occur more frequently near the 600 degree starting point and less frequently at the edges.



Since only one variable can serve as a **Relaxed condition** per calculation, the quantity that is not relaxed has a NaN result in its corresponding result quantity. The **Temperature** result quantity inherits the unit set in the conditions while the **Composition** result quantity inherits the normalization (mole or mass) but always returns a fraction.

## PM\_G\_02: Coarsening and Interfacial Energy

---

The example uses the **Property Model Calculator** and both thermodynamic (FEDEMO) and kinetic (MFEDEMO) demonstration steel databases. Using a **Grid Calculation Type** it produces these plot types: a **Heat map**, a **Contour plot** and an overlaid plot combining a cross plot with a contour plot (where both the interfacial energy and coarsening rate is shown). It also creates a diagram to show the phase fractions vs time and a **3D Plot type** comparing the coarsening rate coefficient.



This example is included as a Property Model tutorial on our [website](#) and as part of the Property Model Calculator playlist on our [YouTube channel](#).



[Coarsening Property Model Settings](#) and [Interfacial Energy Property Model Settings](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Property Models** → **General**
- File name: *PM\_G\_02\_Coarsening\_and\_Interfacial\_energy.tcu*

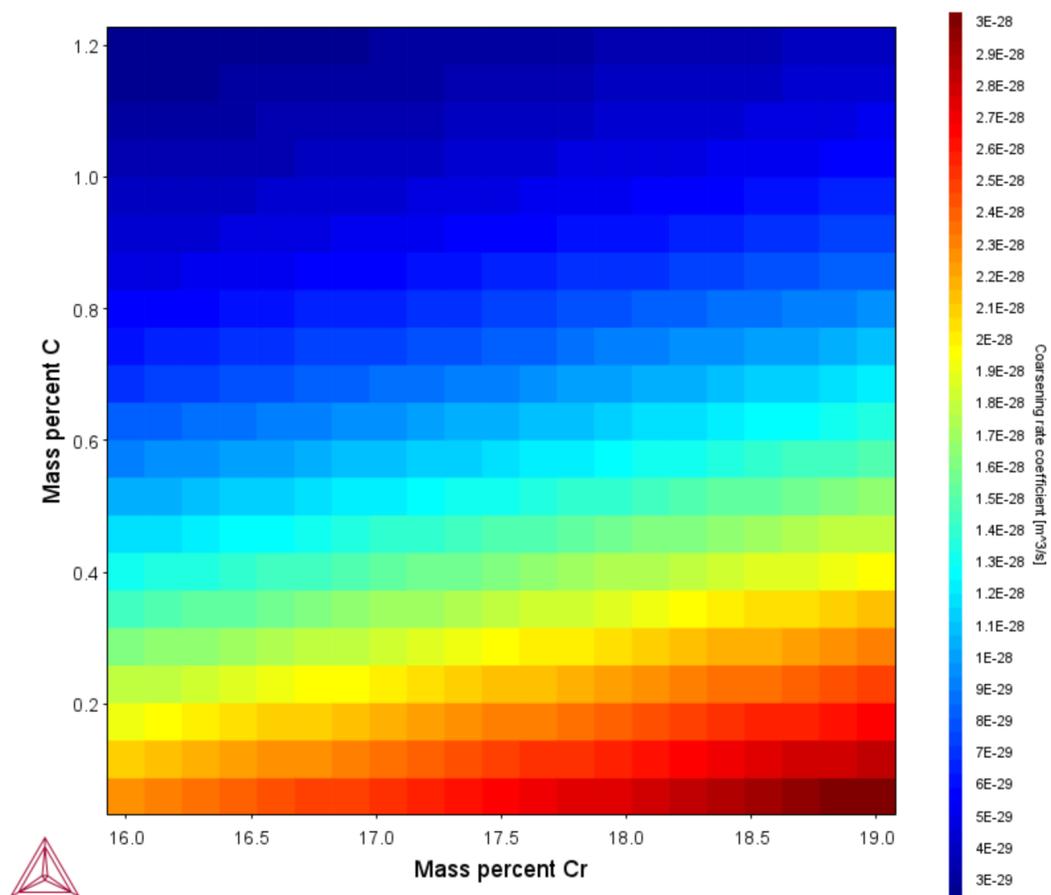


Figure 3: Uses a Property Model Calculator with a Grid Calculation type to plot the coarsening rate coefficient as a Heat map.

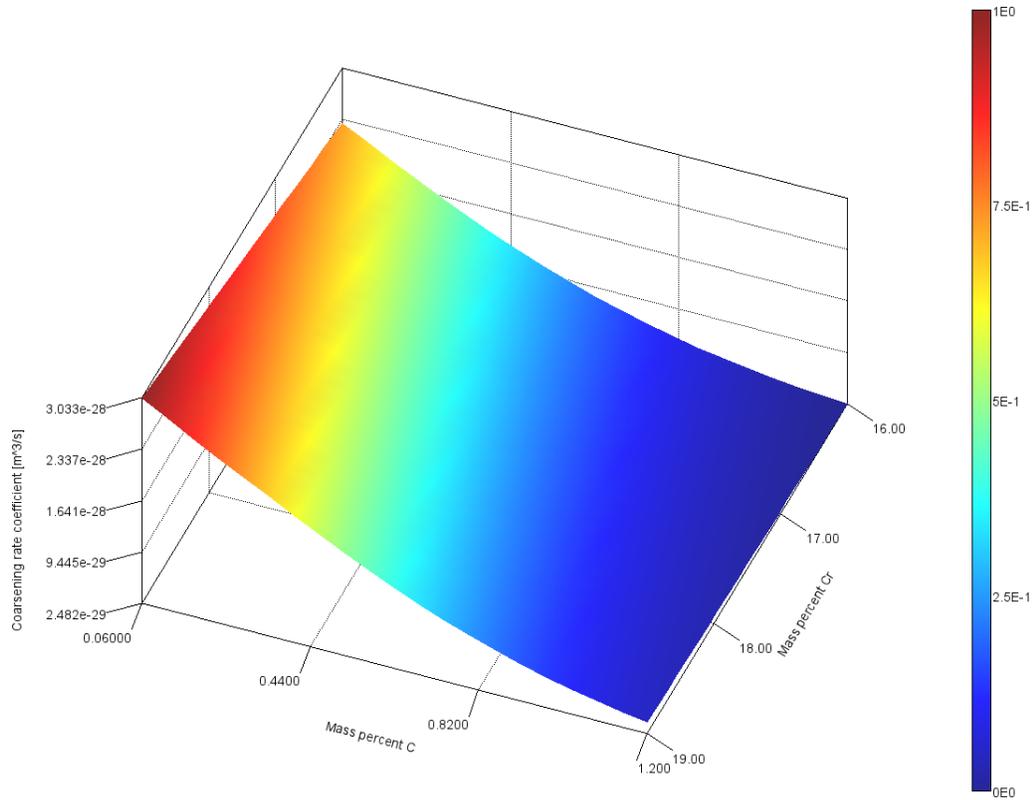


Figure 4: Uses a Property Model Calculator with a Grid Calculation type to plot the coarsening rate coefficient as a 3D plot.



When in the Thermo-Calc help (press F1), go to [Rotating 3D Plots](#) to watch a short video showing you how to rotate this plot in the Visualizations window.

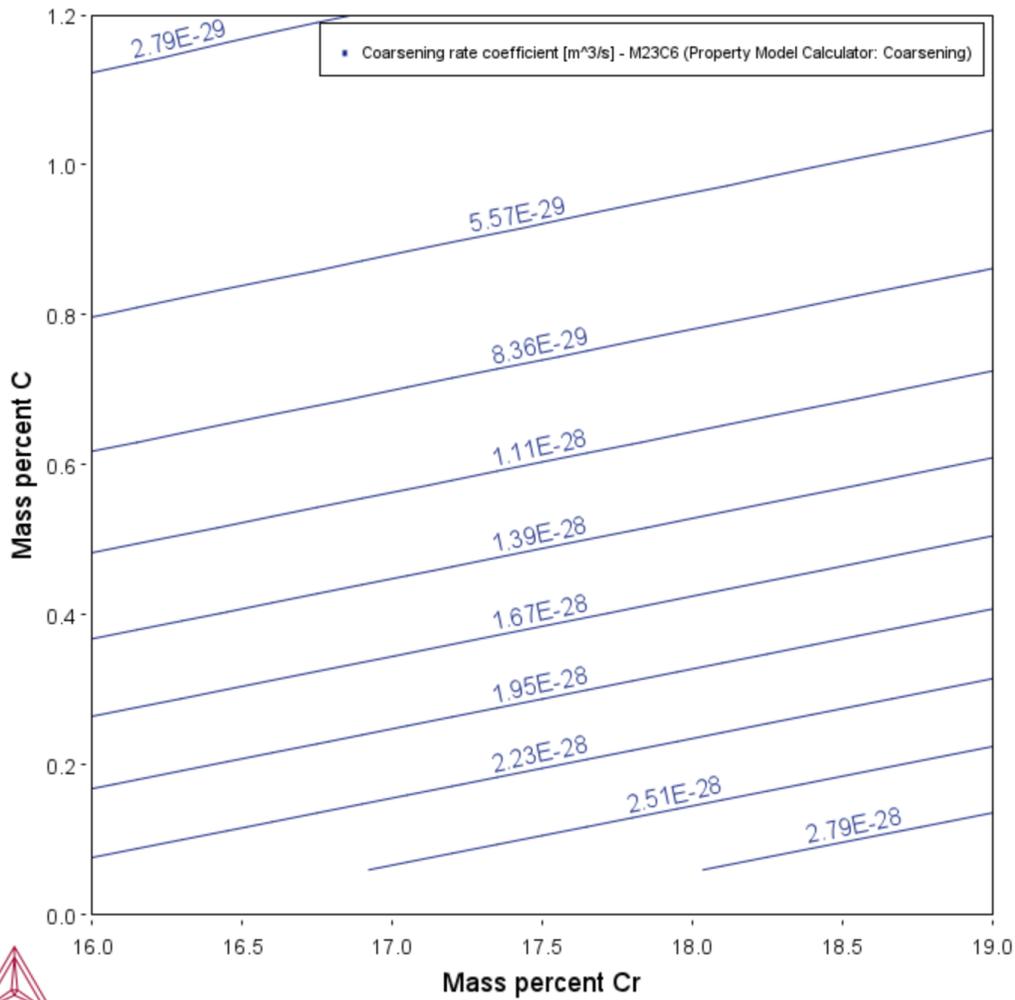


Figure 5: Uses a Property Model Calculator with a Grid Calculation type to plot the coarsening rate coefficient as a Contour plot.

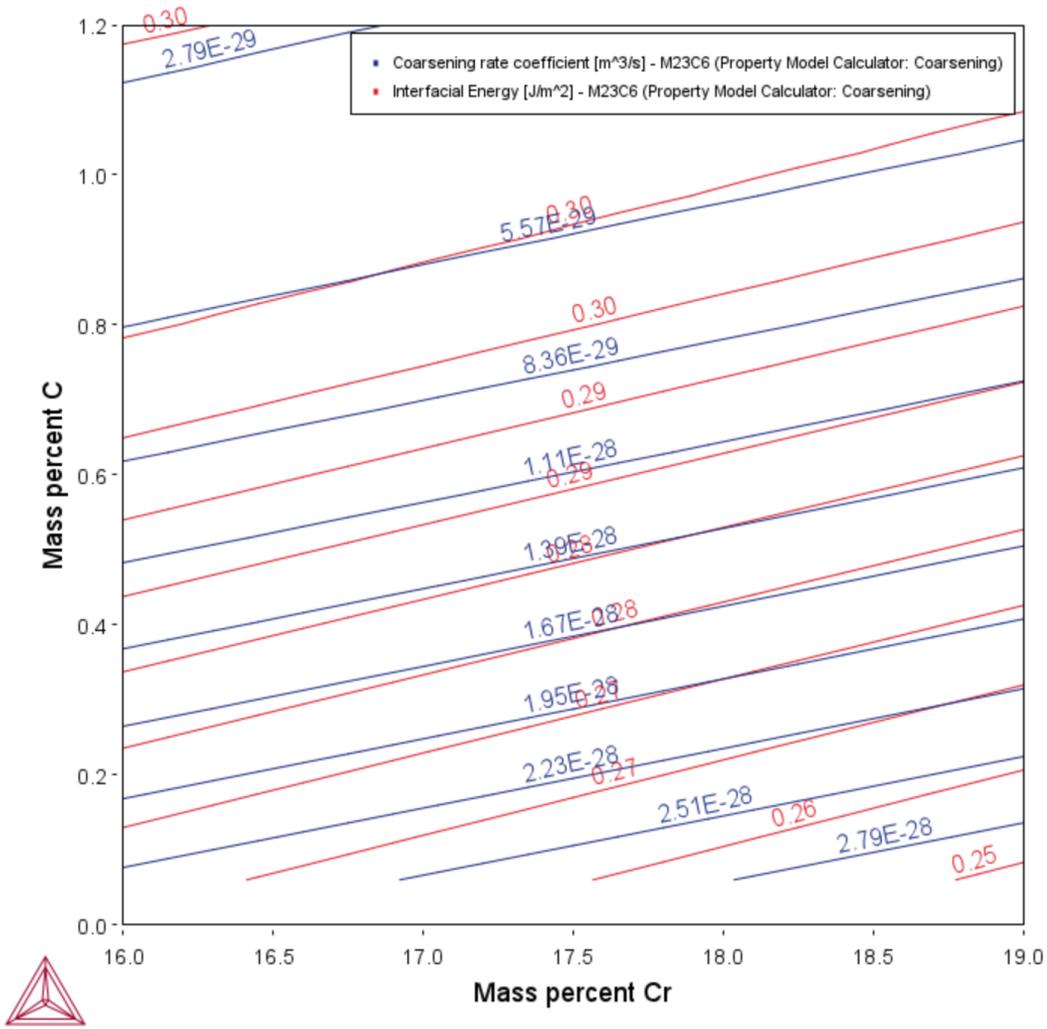


Figure 6: Comparing the coarsening rate coefficient and interfacial energy in overlaid plots.

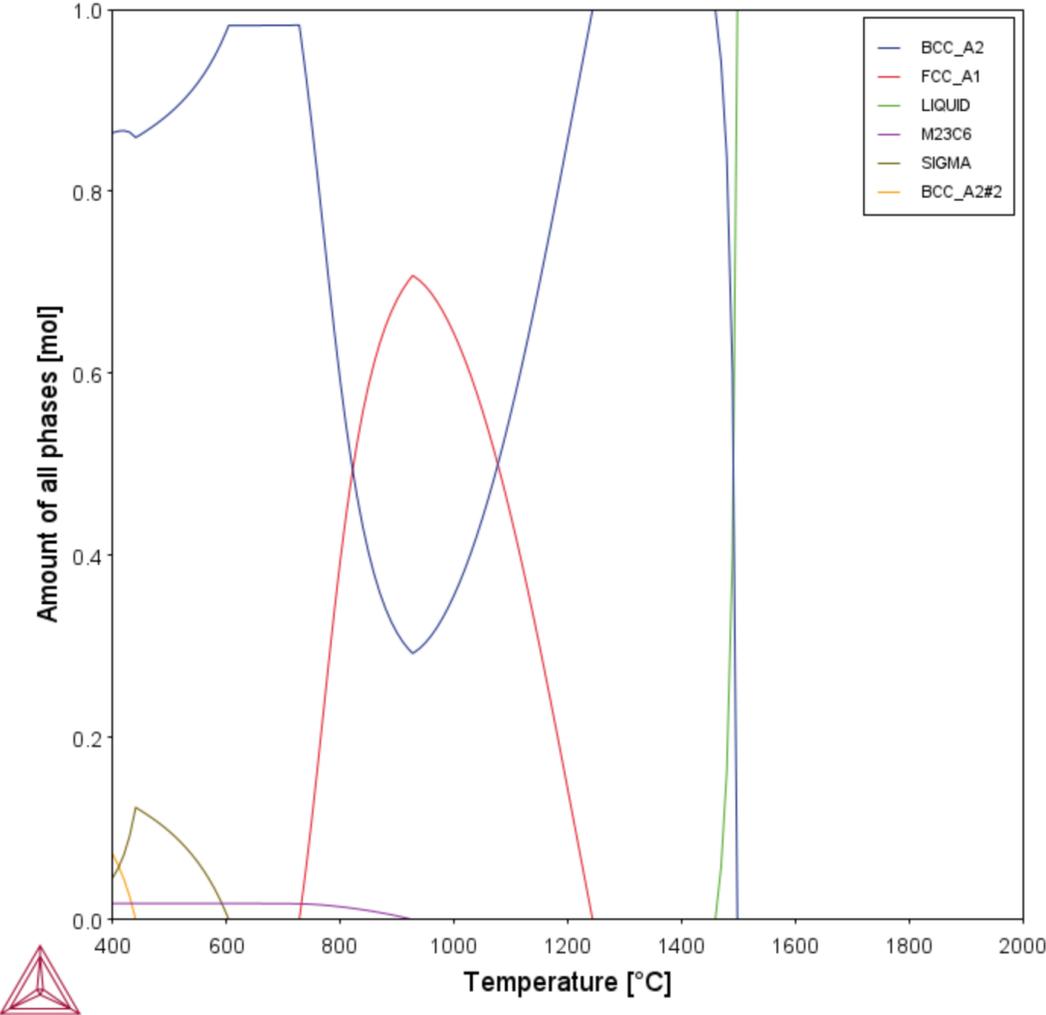


Figure 7: A One Axis Equilibrium calculation showing the phase fractions vs T (with a Flexible Mode plot type).

## PM\_G\_03: Driving Force and Interfacial Energy

---

The example uses the **Property Model Calculator** and a thermodynamic demonstration steel database (FEDEMO). Using a **Grid Calculation type** it produces a **Contour** plot comparing the driving force and interfacial energy.



This example is included as a Property Model tutorial on our [website](#) and as part of the Property Model Calculator playlist on our [YouTube channel](#).



[Driving Force Property Model Settings](#) and [Interfacial Energy Property Model Settings](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Property Models** → **General**
- File name: *PM\_G\_03\_Driving\_force\_and\_Interfacial\_energy.tcu*

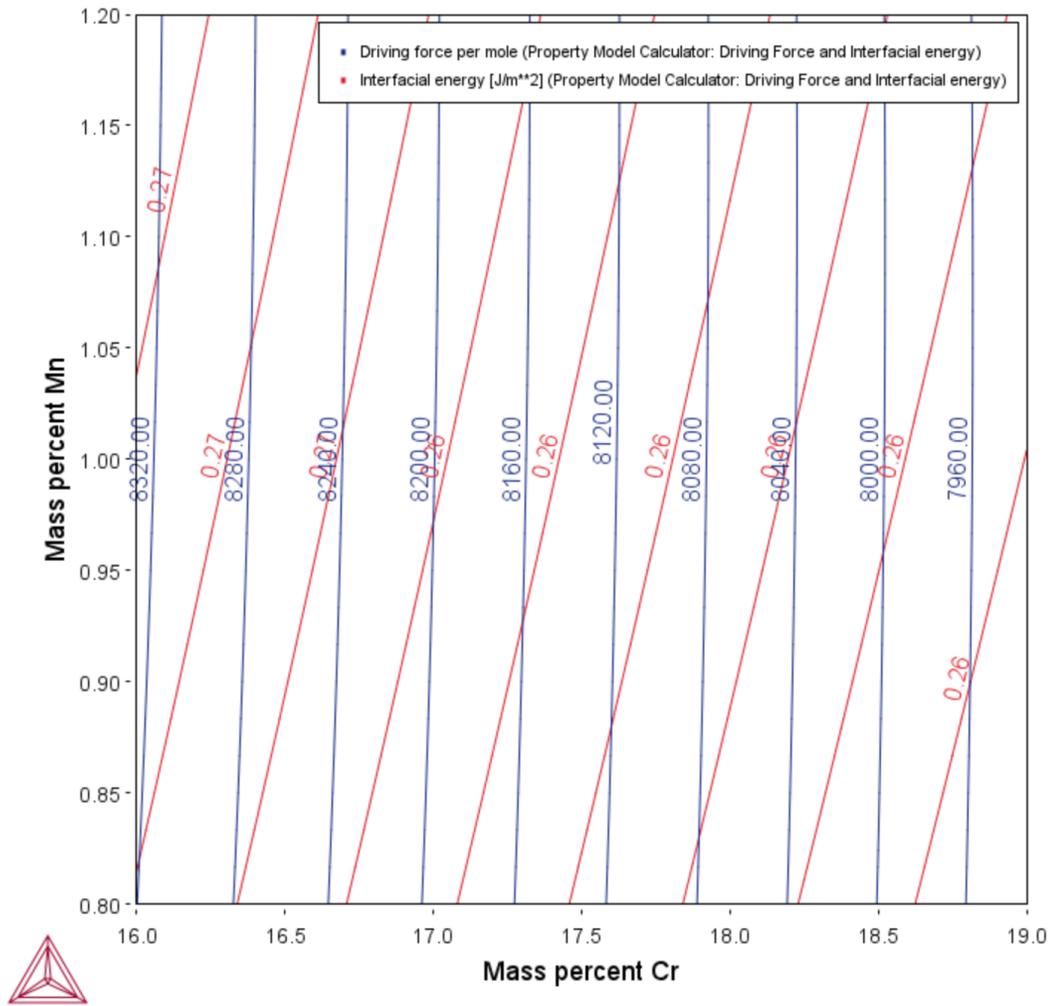


Figure 8: An overlaid Contour plot comparing the driving force and interfacial energy.

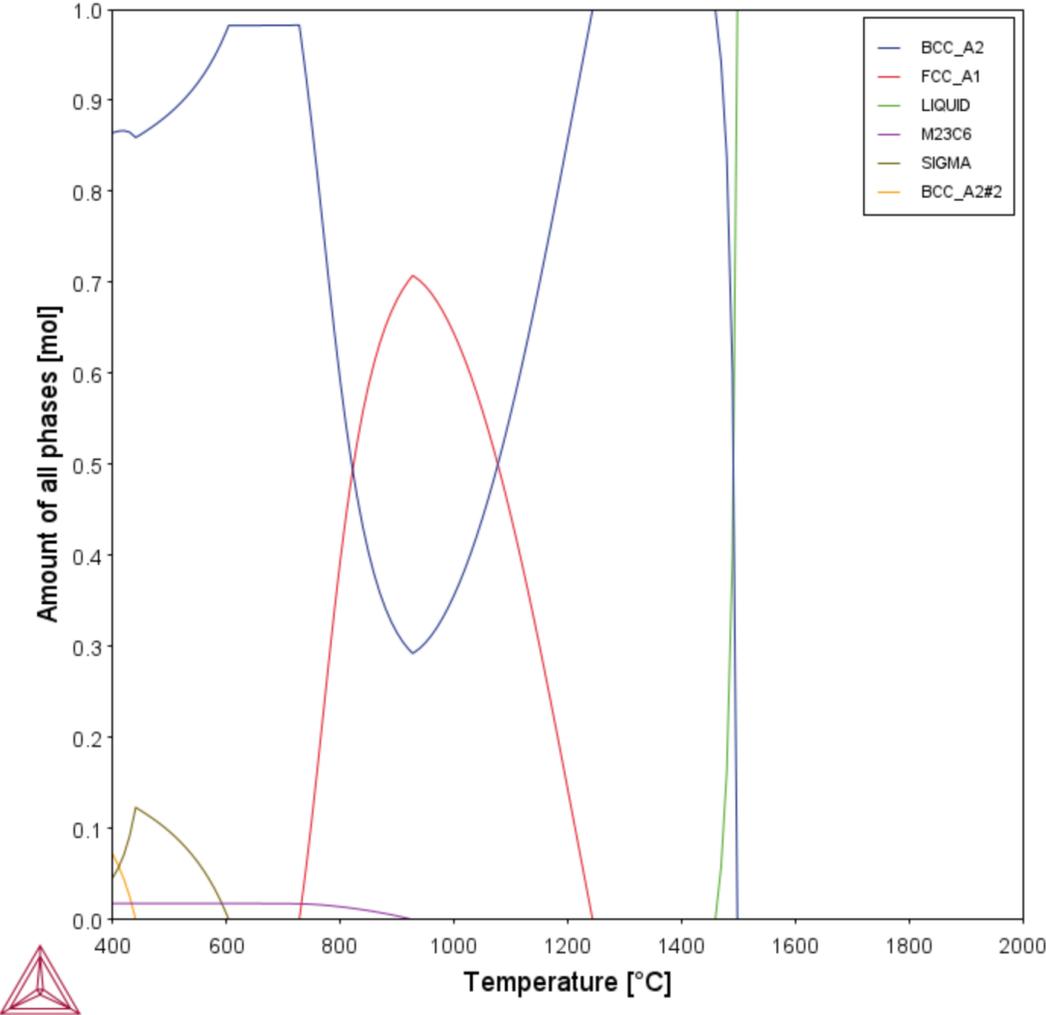


Figure 9: A One Axis Equilibrium calculation showing the Phase fractions vs T.

## PM\_G\_04: Yield Strength

---

The example uses the **Property Model Calculator** and the **Yield strength** Property Model with a thermodynamic demonstration aluminum database (ALDEMO).

Using default options for the precipitation strengthening models, this example compares the **Simplified model (general)**, **Seidman model (Al-base)** and **Deschamps model (Al-base)** yield strength versus precipitate radius to experimental data for an Al-0.3wt%Sc alloy homogenized at 648 °C for 24 hours and subsequently aged at 350 °C.



For more settings information, see [Yield Strength Property Model: Simplified Mode Settings](#) or [Yield Strength Property Model: Advanced Mode Settings](#). For additional theoretical information about the models, see [About the Yield Strength Property Model](#).

The Seidman model [2002Sei] considers a combination of a shearing mechanism and Orowan dislocation bypass mechanism. For the shearing mechanism, the increase in yield strength results from the contributions of ordering strengthening, coherency strengthening, and modulus mismatch strengthening.

When **Seidman model (Al-base)** is selected as the *Precipitation strengthening model*, the setting for  $\alpha$  is a constant related to the coherency strengthening of precipitates, which usually varies from 2 to 3. In this example, the default  $\alpha$  is 2.6 [1985Ard]. The  $m$  setting is the parameter that relates to the modulus mismatch strengthening and the default is set as 0.85 [1985Ard]. This model is applied in the study of precipitation hardening in dilute Al(Sc) alloys containing coherent Al<sub>3</sub>Sc precipitates. The model parameters described above can be adjusted to work for hardenable aluminum alloy calculations from the 2000 (Al–Cu and Al–Cu–Mg), 6000 (Al–Mg–Si), and 7000 (Al–Zn–Mg) series.

As a comparison, the **Simplified model** [2008Zan] only considers the general mechanisms of cutting and looping (Orowan mechanism) without regard to any detailed dislocation mechanisms.

When **Simplified model (general)** is selected as the *Precipitation strengthening model*, the default *Critical radius* (the critical size  $r_c$ ) is optimized against experimental results for dilute Al-Sc. If the precipitates are smaller than a critical size (approximately 5–10 nm) the dislocations cut the precipitates. The strength of the obstacle is then proportional to the square root of the average radius. If the particles are larger than the critical size,  $r_c$ , it is less costly for the dislocations to bow round the particles and the strengthening contribution is inversely proportional to  $r$ . Since the **Simplified model (general)** only depends on generic material constants ( $k_p$  and  $r_c$ ) it may be applied to any precipitation strengthened material.

The **Deschamps model (Al-base)** model [1998Des], is for precipitation strengthening, mainly intended for use with Al-base alloys. The Deschamps model includes use of Friedel or Kock's statistics for particle size distribution. Default parameters are used to match a dilute Al-Sc alloy with  $\text{Al}_3\text{Sc}$  precipitates. As is done for the other precipitation strengthening models, a critical radius, describing the transition from cutting to looping, must be specified when you are setting up the configuration.

### ***Project File Information***

- Folder: **Property Models** → **General**
- File name: *PM\_G\_04\_Yield\_Strength.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

---

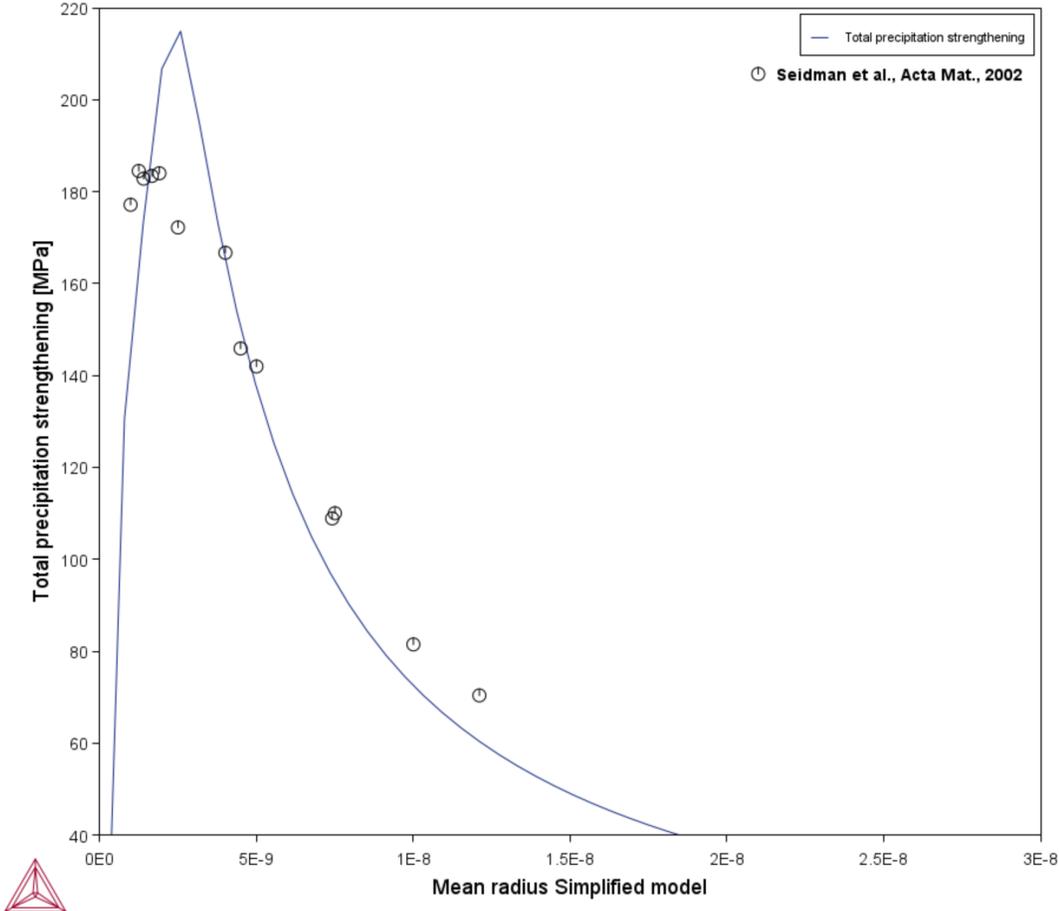


Figure 10: Strength vs Radius - Simplified compared to experimental data from [2002Sei].

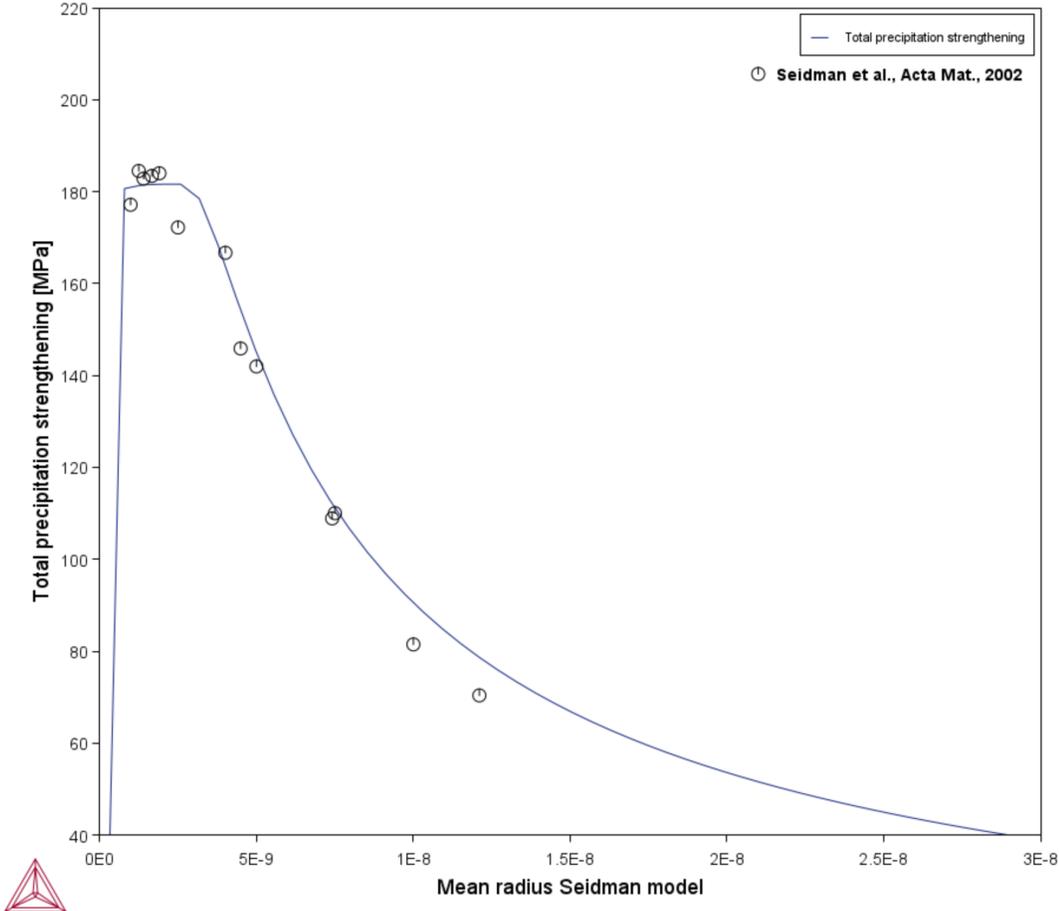


Figure 11: Strength vs Radius - Seidman compared to experimental data from [2002Sei].

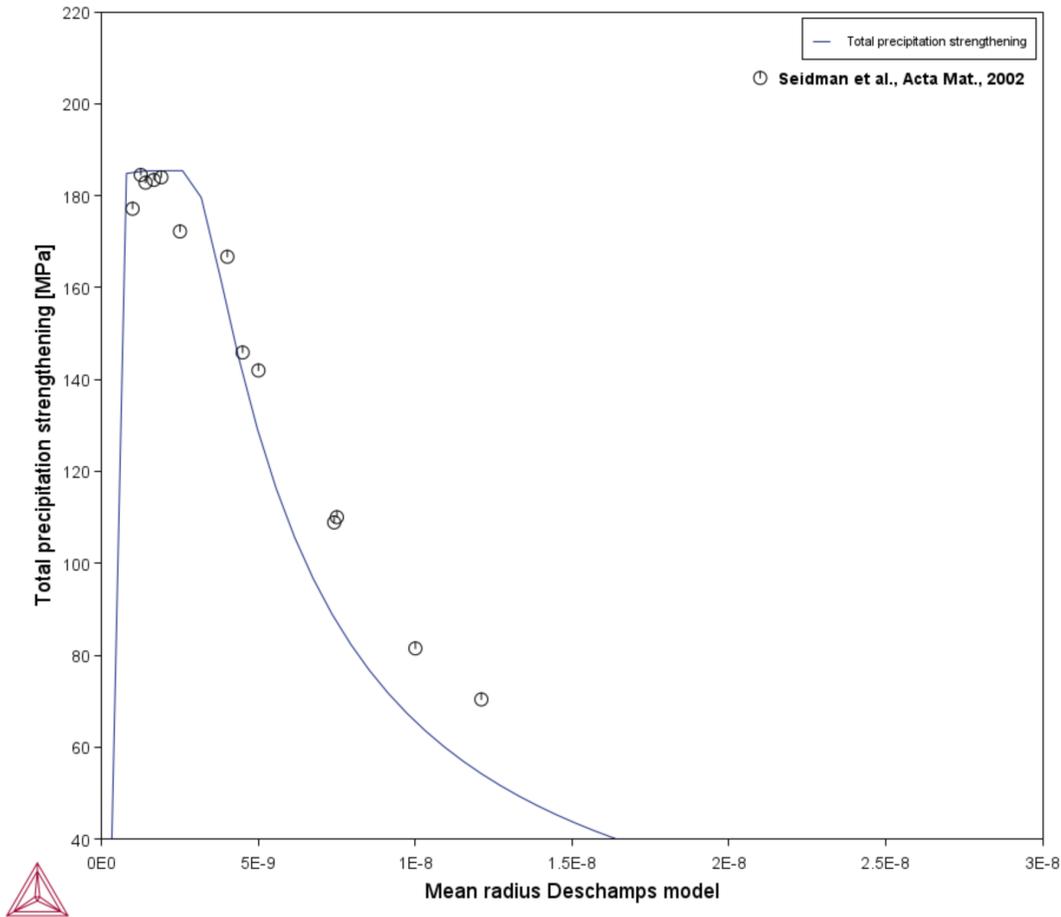


Figure 12: Strength vs Radius - Deschamps compared to experimental data from [2002Sei].

## References

- [1985Ard] A. J. Ardell, Precipitation hardening. Metall. Trans. A. 16, 2131–2165 (1985).
- [1998Des] A. Deschamps, Y. Brechet, Influence of predeformation and ageing of an Al–Zn–Mg alloy—II. Modeling of precipitation kinetics and yield stress. Acta Mater. 47, 293–305 (1998).
- [2002Sei] D. N. Seidman, E. A. Marquis, D. C. Dunand, Precipitation strengthening at ambient and elevated temperatures of heat-treatable Al(Sc) alloys. Acta Mater. 50, 4021–4035 (2002).
- [2008Zan] J. Zander, R. Sandström, One parameter model for strength properties of hardenable aluminium alloys. Mater. Des. 29, 1540–1548 (2008).

## PM\_G\_05: Yield Strength NiAlCr

---

The example uses the **Property Model Calculator** and the **Yield strength** Property Model with a thermodynamic demonstration nickel database (NIDEMO).

The Reppich model is developed for calculation of precipitation strengthening in Ni-based super alloys consisting mainly of gamma prime ( $\gamma'$ ) precipitates in a gamma matrix. The example shows a calculation of the precipitation strengthening vs precipitate radius in a Ni-10at%Al-10at%Cr alloy.

This example uses the Reppich model [1982Rep] which is developed and rests on existing hardening models estimating the increase of the yield stress due to pairwise particle cutting and antiphase domain boundary (APB) formation. The effect of the anisotropy on the dislocation line tension of two-phase material and on the Orowan stress is taken into account. Compared to conventional APB hardening theories, this model can be applied to large volume fraction of precipitates in the anisotropic systems.



For more settings information, see [Yield Strength Property Model: Simplified Mode Settings](#) and for additional theoretical information about the models, see [About the Yield Strength Property Model](#).

For the settings input of the **Reppich model (Ni-base)**:

- $A$  is a numerical factor depending on the morphology of the precipitates. For spherical particles,  $A$  equals to 0.72.
- $w$  represents the elastic repulsion between the strongly paired dislocations, which can be determined by the experimental data [1979Hüt].

In this example,  $A$  and  $w$  are set by default to 0.7639 and 0.2919, respectively.

### **Project File Information**

- Folder: **Property Models** → **General**
- File name: *PM\_G\_05\_Yield\_Strength\_NiAlCr.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

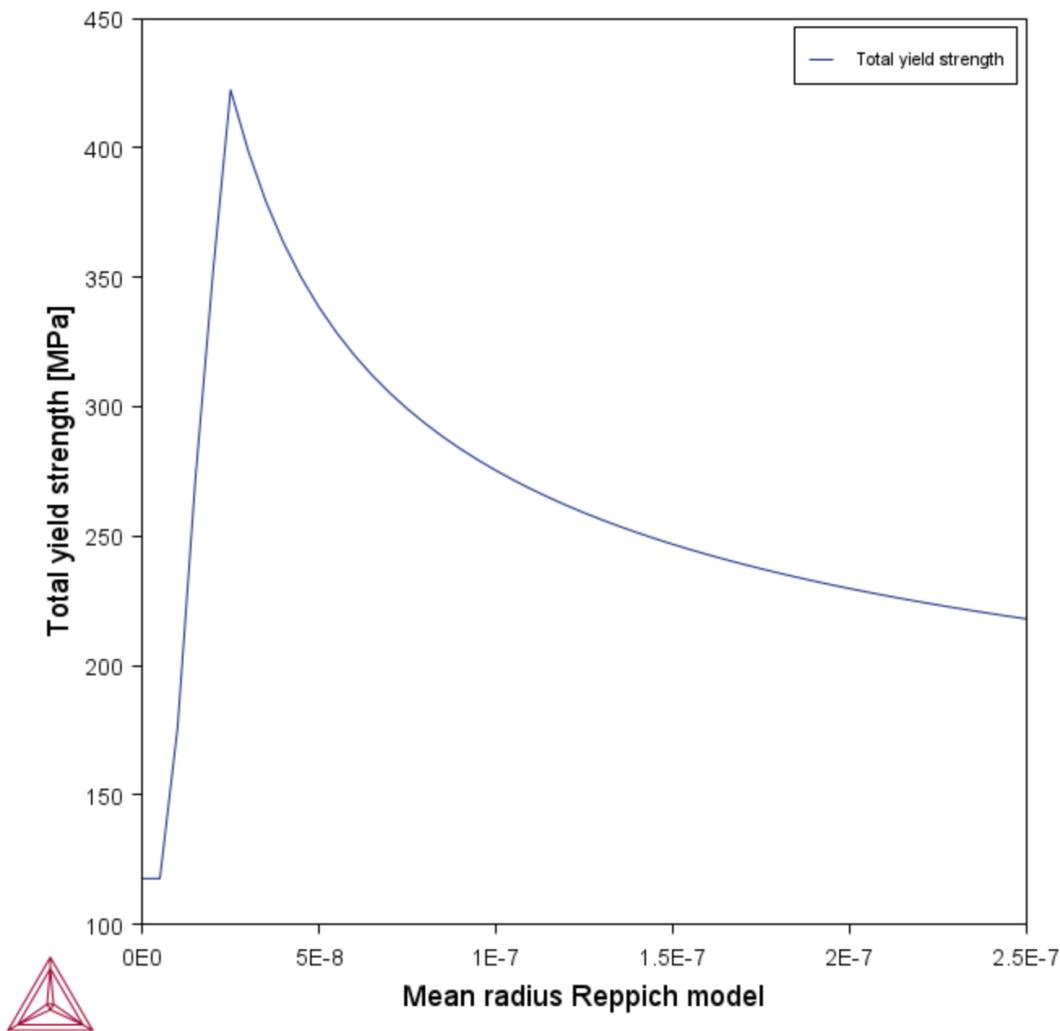


Figure 13: Reppich model plotted with results and experimental data for PE16, all data is normalized with the square root of the volume fraction of precipitate.

## References

- [1979Hüt] Hüther, W. and Reppich, B. "Order hardening of MgO by large precipitated volume fractions of spinel particles", Mater. Sci. Eng. 39, 247–259 (1979).
- [1982Rep] Reppich, B. "Some new aspects concerning particle hardening mechanisms in  $\gamma'$  precipitating Ni-base alloys—I. Theoretical concept," Acta Metall. 30, 87–94 (1982).

## PM\_G\_06: Yield Strength HEA

---

The example uses the **Property Model Calculator** and the **Yield strength** Property Model with the thermodynamic TCS High Entropy Alloys Database (TCHEA). It is an example of solid solution strengthening, which is the contribution to total strength due to the elastic strains in the crystal lattice caused by alloying elements of a lattice parameter differing from the main constituent. The example shows the solid solution strengthening at 300 K and 411 K over the full solubility range for the Mo-Ta system as compared to experimental data [1972Ste].



This example requires the use of the TCS High Entropy Alloys Database (TCHEA). A license is required to run the example.



For more settings information, see [Yield Strength Property Model: Simplified Mode Settings](#) and for additional theoretical information about the models, see [About the Yield Strength Property Model](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Property Models** → **General**
- File name: *PM\_G\_06\_Yield\_Strength\_HEA.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

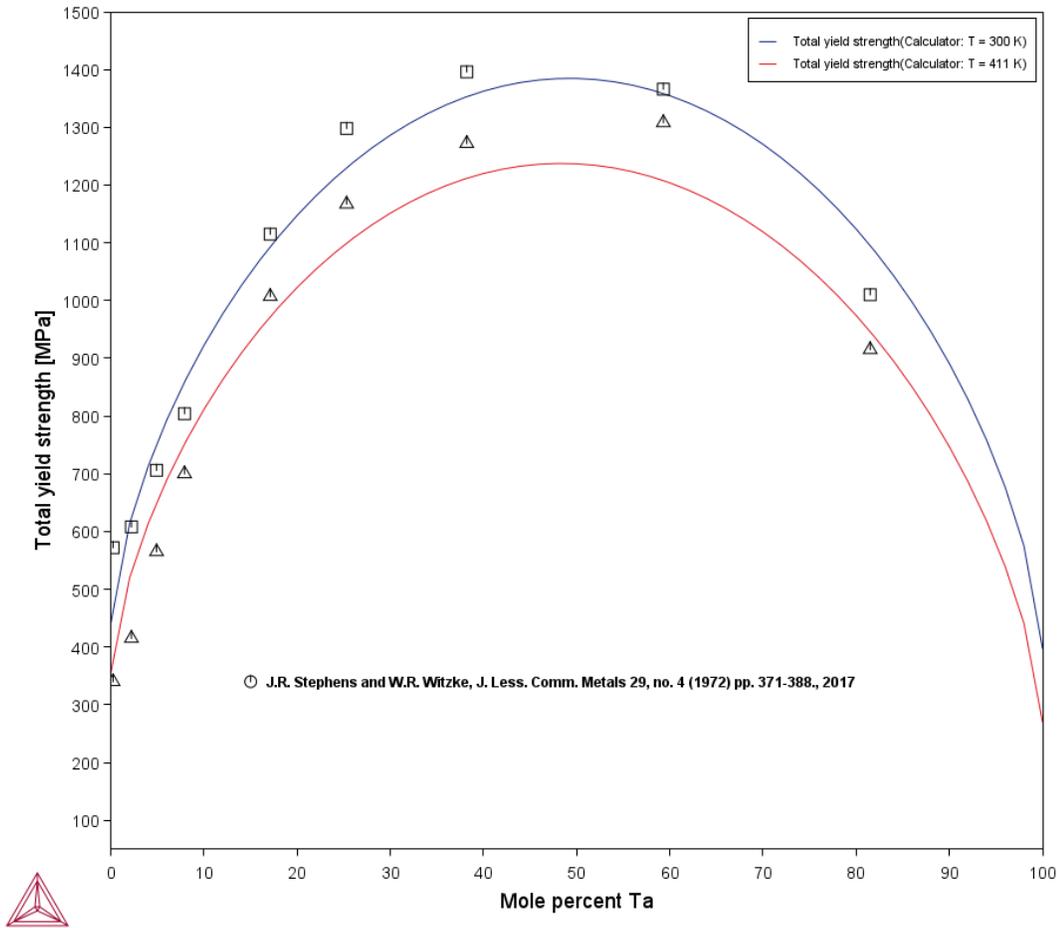


Figure 14: The solid solution strengthening over the full solubility range for the Mo-Ta system as compared to experimental data [1972Ste].

## Reference

[1972Ste] J. R. Stephens, W. R. Witzke, Alloy hardening and softening in binary molybdenum alloys as related to electron concentration. J. Less Common Met. 29, 371–388 (1972).

## PM\_G\_07: Hot Crack Susceptibility

---

The example uses the **Property Model Calculator** and the **Crack Susceptibility Coefficient** Property Model to calculate the hot tearing tendency during solidification for an Al-Si alloy. The **ALDEMO: Aluminum Demo Database** is used and this example is available to all users.

Hot tearing is one of the most common and serious defects encountered during the casting of, for example, aluminum alloys. In general, it is defined by the formation of a macroscopic fissure in a casting as a result of stress and the associated strain, generated during cooling, at a temperature above the non-equilibrium solidus.

The Model is based on the publication by Yan and Lin [2006Yan] and uses experimental data [1955Pum; 1976Feu; 2004Bar] from this paper.

The experimental hot cracking susceptibility of an alloy is defined as the ratio between the cracking length for that alloy and the maximum cracking length in the alloy system studied. Considering the uncertainties and difficulties in hot tearing measurements, the calculated hot tearing tendencies are in excellent agreement with the experimental data. The typical L-shaped curve is well reproduced in the current prediction. It rapidly increases at a low solute content and has a maximum at a composition of around 0.5 wt pct Si.



See [About the Crack Susceptibility Coefficient Property Model](#) for background information about this Property Model.



This example is included as a Property Model tutorial on our [website](#) and as part of the Property Model Calculator playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File Information***

- Folder: **Property Models** → **General**
- File name: *PM\_G\_07\_Hot\_Crack\_Susceptibility.tcu*

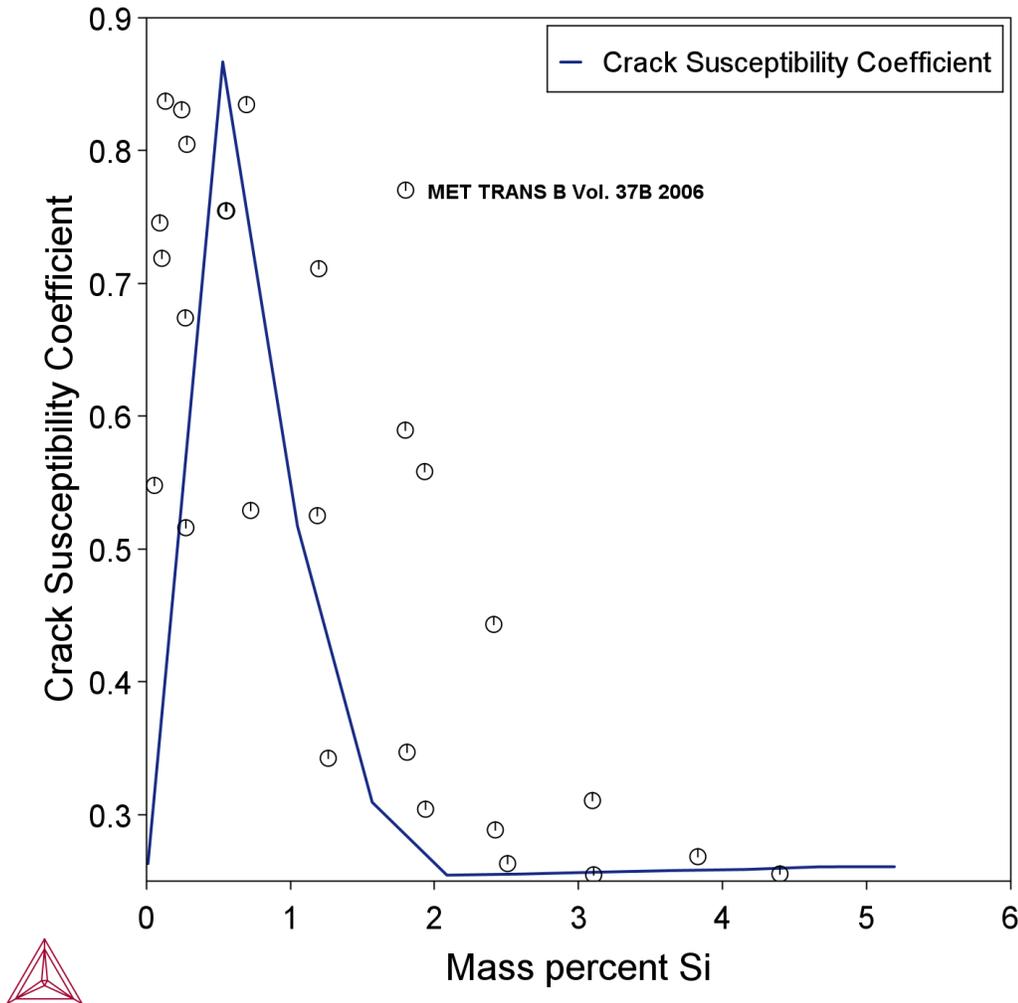


Figure 15: The plot compares the predicted cracking susceptibility/composition curve for the Al-Si system with the experimental hot tearing tendencies [1955Pum; 1976Feu; 2004Bar].

## References

- [1955Pum] W.I. Pumphrey: "The Aluminum Development Association Report No. 27," Aluminum Development Association, London, 1955.
- [1976Feu] V. Feurer: *Giessereiforschung*, 1976, vol. 28, pp. 75-80.
- [2004Bar] S. R. Barnett, J. A. Taylor, and D. H. St. John: *Solidification of Aluminum Alloys*, M.G. Chu, D.A. Granger, and Q. Han, eds., TMS, Warrendale, PA, 2004, pp. 201-09.
- [2006Yan] X. Yan and J. C. Lin, "Prediction of hot tearing tendency for multicomponent aluminum alloys," *Metall. Mater. Trans. B*, vol. 37, no. 6, pp. 913-918, Dec. 2006.

## PM\_G\_08: Spinodal

---

The example uses the **Property Model Calculator** and the **Spinodal** Property Model to calculate the so-called spinodal curve for the BCC\_A2 miscibility gap in the system Fe-Cr. The **FEDEMO: Iron Demo Database** is used and this example is available to all users.

Two Property Model Calculators are used in this example. The spinodal curve (a **One Axis** calculation), is overlaid on top of a the Fe-Cr **Phase diagram** calculation. As can be seen in the plot, the spinodal curve for BCC goes through the one-phase region for Sigma. This is because the spinodal curve for a phase is calculated when all other phases are suspended.



[About the Spinodal Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Property Models** → **General**
- File name: *PM\_G\_08\_Spinodal.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

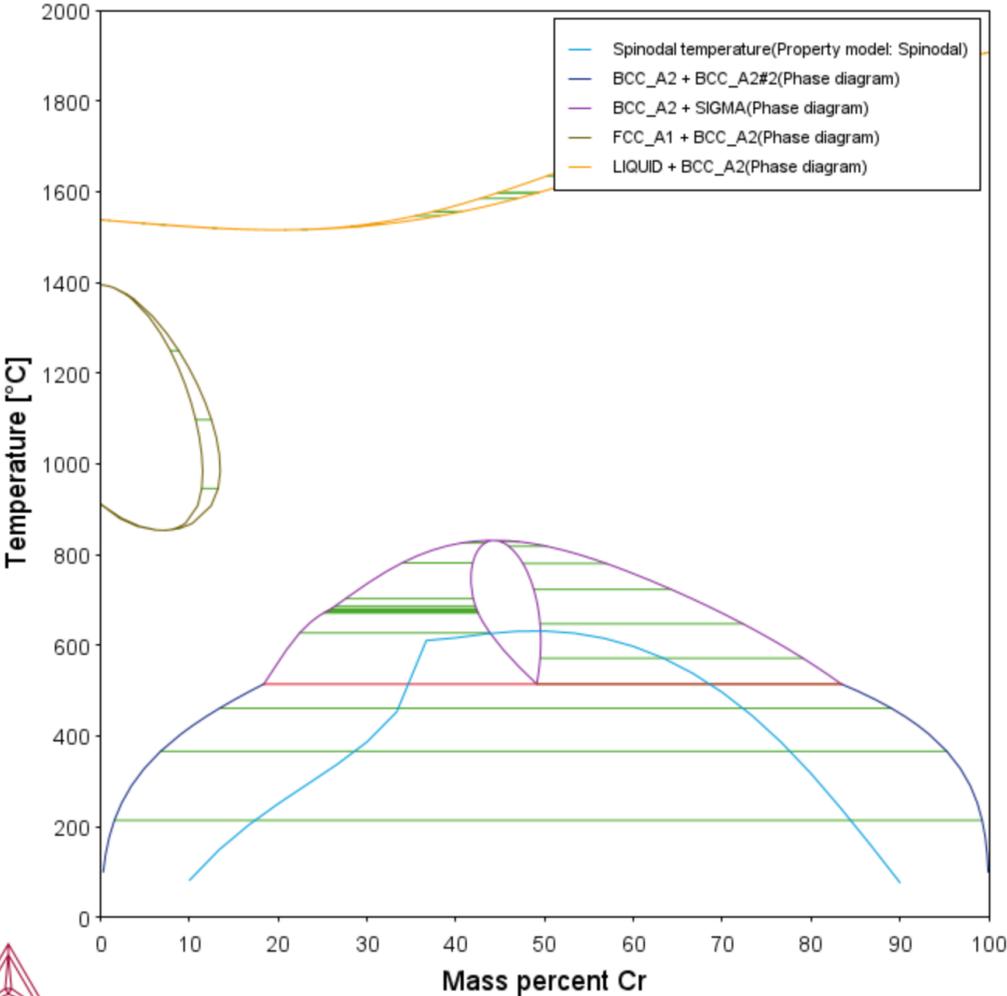


Figure 16: An Fe-Cr spinodal curve for BCC goes through the one-phase region for Sigma.

## PM\_G\_09: T-Zero Temperature

---

The example uses the **Property Model Calculator** and the **T-Zero Temperature** Property Model to calculate the so-called  $T_0$  line for the two-phase field FCC\_A1 and BCC\_A2 in the Fe-Ni system. The line is plotted together with the phase diagram for the same Fe-Ni system. The **FEDEMO: Iron Demo Database** is used and this example is available to all users.

The  $T_0$  temperature is defined as the temperature where two phases of identical chemical composition have the same molar Gibbs free energy. This temperature is an important quantity in the field of diffusionless phase transformations, e.g. martensitic transformation, since it is the upper limit where diffusionless phase transformations can occur.

Two Property Model Calculators are used in this example. The  $T_0$  line (a **One Axis** calculation) for the two-phase field FCC\_A1 and BCC\_A2 is overlaid on top of a the Fe-Ni **Phase diagram** calculation.



[About the T0 Temperature Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Property Models** → **General**
- File name: *PM\_G\_09\_T-Zero\_temperature.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

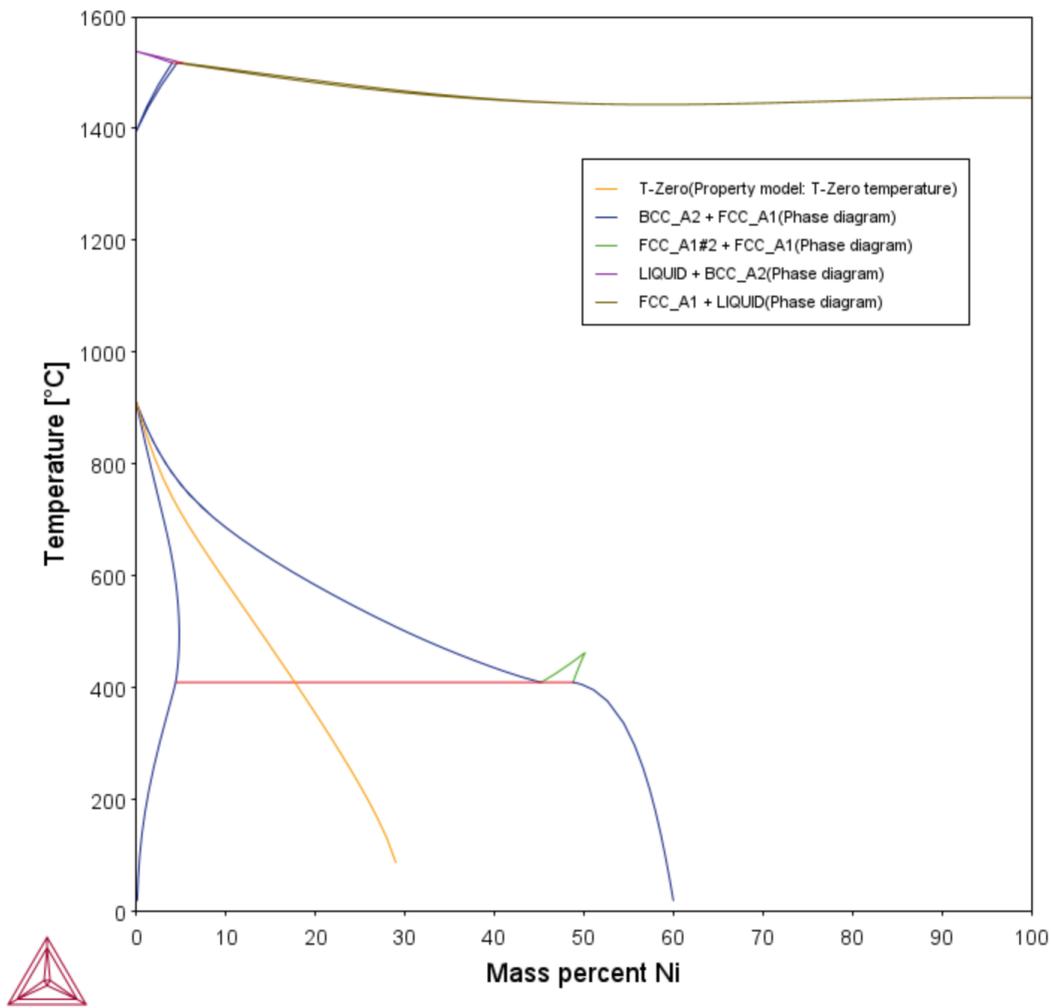


Figure 17: In this plot for the Fe-Ni system, the  $T_0$  line for the FCC\_A1 and BCC\_A2 phases is located in the middle of the two-phase region. No solution for the  $T_0$  temperature exists above about 30 mass% Ni.

## PM\_G\_10: Freeze-in Thermal Conductivity

---

The example uses the **Property Model Calculator** and the **Equilibrium with Freeze-in Temperature** Property Model to compare the use of the model with thermal conductivity with and without grain boundary (GB) phase scattering being included in the calculation.

The TCS Al-based Alloy Database (TCAL) is used to evaluate a 7075\_T6 alloy (Al-1.6Cu-2.5Mg-5.6Zn-0.25Fe-0.15Mn-0.23Cr-0.2Ti-0.2Si) mass%. The "T6" in the alloy name means that the alloy has been artificially peak-aged, i.e. the matrix is almost depleted. The heating is assumed to be at a typical artificial aging temperature of 150 °C. The temperature has then been increased for measuring of the reported values thermal conductivity at 400, 500, and 532 °C.

The thermal conductivity due to grain boundary phase scattering is approximated as a scattering constant times the total volume fraction of the grain boundary phases. The contribution to thermal conductivity is assumed to be related to that to electrical resistivity, following the Wiedemann-Franz law.

The calculation set up assumes that the alloy reaches equilibrium at the peak-age temperature, i.e. that the amounts of phases and their composition freeze-in at 150 °C and that these do not change for the evaluation of thermal conductivity at the temperatures 400, 500, and 532 °C. Calculations are compared between accounting and ignoring contributions from grain boundary (GB) phase scattering.

As can be seen in [Figure 18](#), the calculation including the approximate effect of grain boundary phase scattering is close to the experimentally measured values. Excluding the effect of grain boundary phase scattering slightly overestimates the thermal conductivity. Experimental data is from [2008ASM].



The TCS Al-based Alloy Database (TCAL) is used in this example. A valid license for version 7 (TCAL7) or newer is required to run the example.



[About the Equilibrium with Freeze-in Temperature Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Property Models** → **General**
- File name: *PM\_G\_10\_Freeze\_In\_Thermal\_Conductivity.tcu*

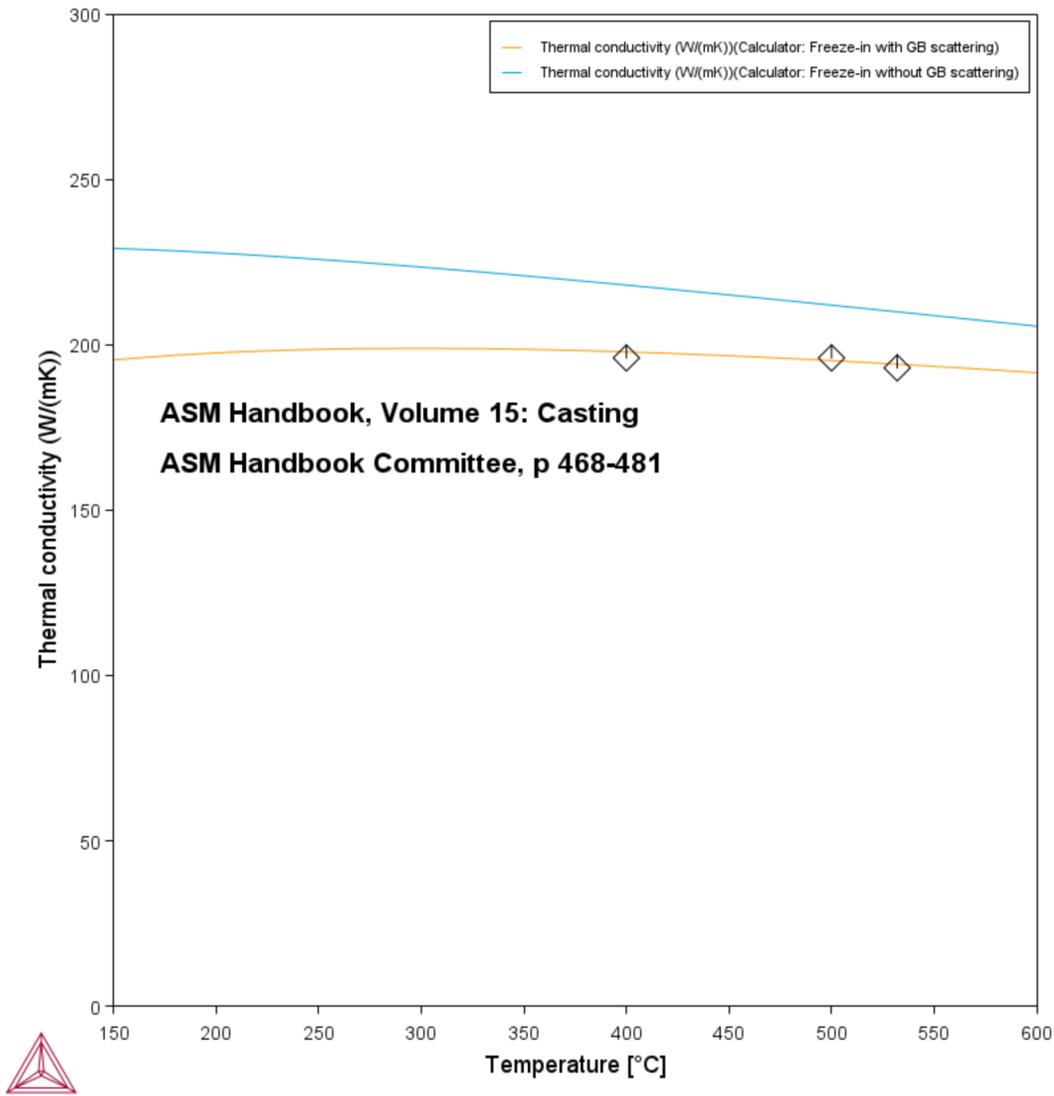


Figure 18: Comparing the freeze-in temperature for the 7075\_T6 alloy with and without grain boundary (GB) scattering included in the calculations.

## Reference

[2008ASM] ASM Handbook Committee, ASM Handbook Volume 15: Casting. ASM International, p. 468-481, 2008, ISBN 978-0-87170-711-6.

## PM\_G\_11: Freeze-in Electrical Resistivity

---

The example uses the **Property Model Calculator** and the **Equilibrium with Freeze-in Temperature** Property Model to compare the use of the model for electrical resistivity with and without grain boundary (GB) scattering being included in the calculation.

The TCS Al-based Alloy Database (TCAL) is used to evaluate a 4032-O alloy (Al-0.9Cu-1.0Mg-0.9Ni-12.2Si) mass%. The "O" in the alloy name means that the Al alloy has been heat-treated at a typical temperature of 350 °C. The measurement of electric resistivity is usually performed at room temperature.

The electrical resistivity due to grain boundary phase scattering is approximated as a scattering constant times the total volume fraction of the grain boundary phases.

The calculation set up assumes that the alloy reaches equilibrium at the "O" heat-treated temperature, i.e. that the amounts of phases and their composition freeze-in at 350 °C and that these do not change for the evaluation of electrical resistivity at the room temperature. Calculations are compared between accounting and ignoring contributions from grain boundary (GB) scattering.

As can be seen from [Figure 19](#), the calculated electrical resistivity at room temperature, including grain boundary phase scattering, is very close to the experimentally measured value (4.3E-8) for the alloy. Excluding the grain boundary phase scattering somewhat underestimates the electrical resistivity for the alloy. Experimental data is from [1993Dav].



The TCS Al-based Alloy Database (TCAL) is used in this example. A valid license for version 7 (TCAL7) or newer is required to run the example.



[About the Equilibrium with Freeze-in Temperature Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Property Models** → **General**
- File name: *PM\_G\_11\_Freeze\_In\_Electric\_Conductivity.tcu*

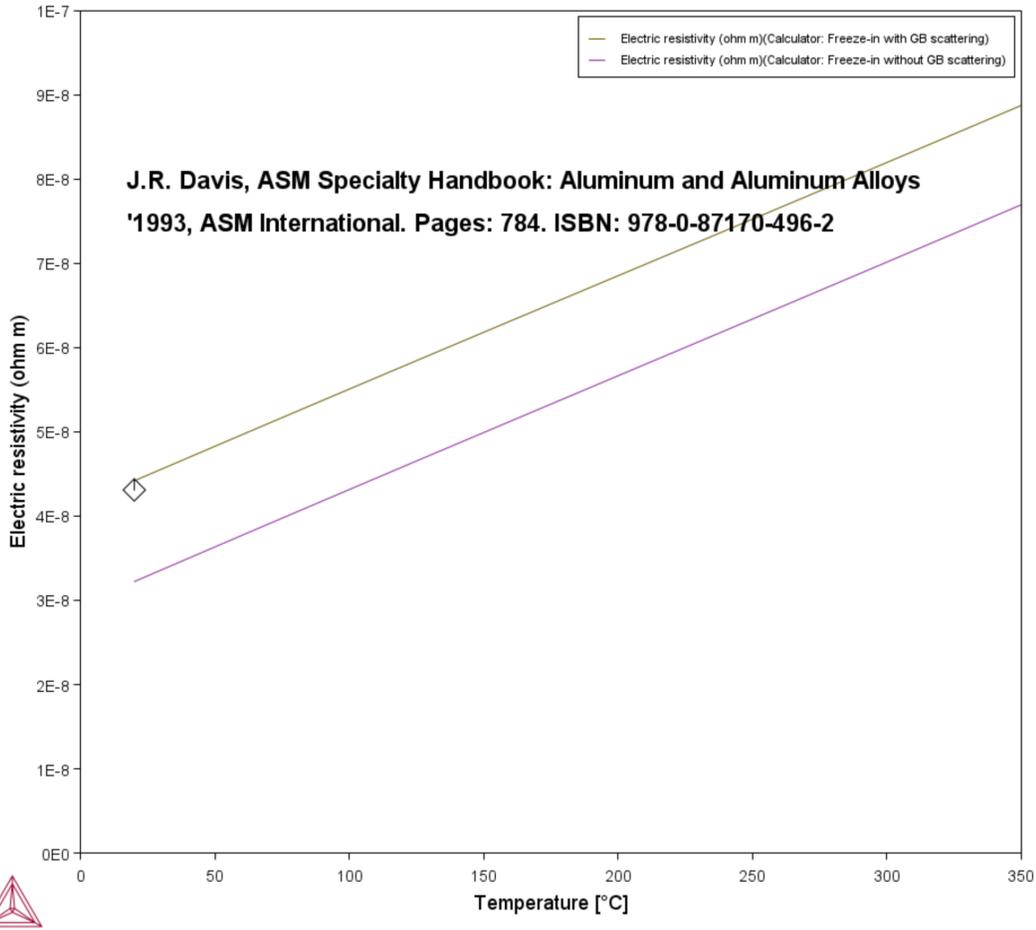


Figure 19: Comparing the freeze-in temperature for the 4032-O alloy with and without grain boundary (GB) scattering included in the calculations.

## Reference

[1993Dav] J.R. Davis, ASM Specialty Handbook: Aluminum and Aluminum Alloys, 1993, ASM International. Pages: 784. ISBN: 978-0-87170-496-2.

## PM\_G\_12: Solidus and Liquidus Batch Calculation

---

The example uses the **Property Model Calculator** and the **Liquidus and Solidus Temperature** Property Model to demonstrate the use of the **Batch Calculation Type**.

The thermodynamic demonstration steel database (FEDEMO) is used along with a randomized set of experimental data points that are shown on the **Configuration** window in a table. The data showing in this example is imported from a data file prepared with specific data entry requirements.

A variety of compositions for an Fe-Cr-Mn-Ni-C alloy is used for the calculations; these compositions are also taken from the same data file. When the heading includes `Exp`, then these columns are considered experimental data by Thermo-Calc, where in this example it is the columns titled `Exp Liq temp C` and `Exp Sol temp C`. This experimental data can then be plotted as a function of the calculated solidus/liquidus temperature and compared on the plot using a **Cross plot**, which is selected on the **Plot Renderer**. The limits are shown using the setting RMS (root mean square).



The experimental data entered into the batch data file is unitless even though the actual experimental data unit is Celsius (as written in the header text). In order for the plot and calculations to match, Celsius is chosen as the temperature unit when defining the **Solidus/Liquidus** Property Model and all the axes manually scaled to the same limits (1360 → 1520).



[Working with Batch Calculations](#)



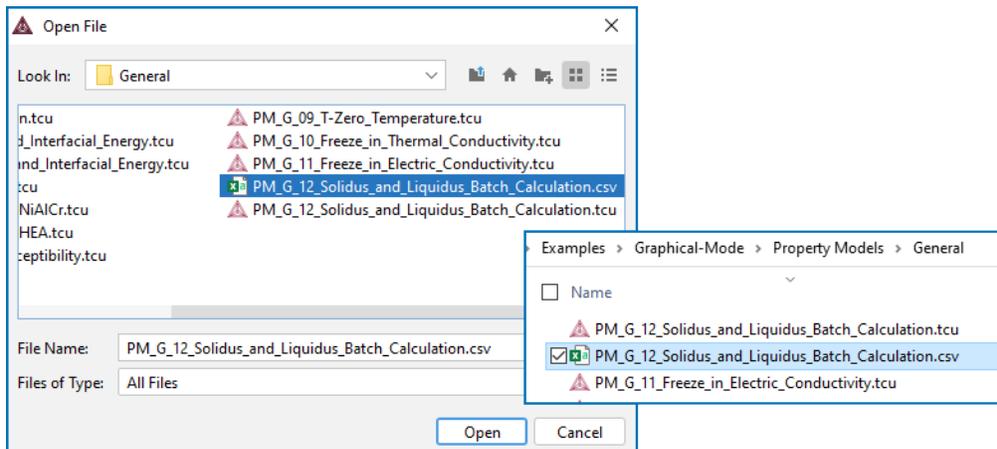
This example is included as a Property Model tutorial on our [website](#) and as part of the Property Model Calculator playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

---

## Project File Information

- Folder: **Property Models** → **General**
- File name: *PM\_G\_12\_Solidus\_and\_Liquidus\_Batch\_Calculation.tcu*
- There is also a *.csv* file with the same name available in this folder that is used with this example. You can navigate to use it as a template for other data files or just to see its format.



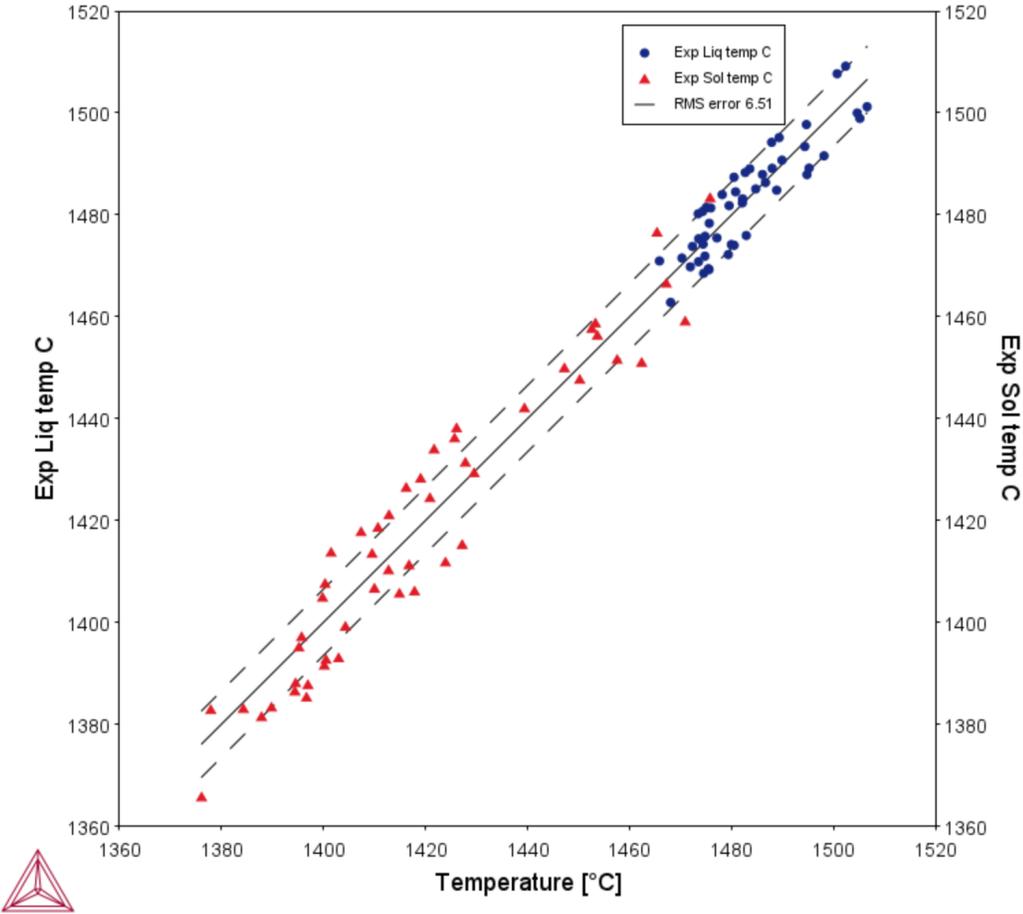


Figure 20: A comparison of experimental liquidus and solidus data using a Cross plot with a Batch calculation. A root mean square (RMS) setting shows the distribution of the data points.

---

## PM\_G\_13: Ti Alloy Design for Additive Manufacturing

---

This in-depth example, which is [showcased on our website](#), uses the **Property Model Calculator** to find the optimal composition for a titanium-based alloy to be used for additive manufacturing applications.

Additive manufacturing provides new opportunities to fabricate complex parts. However, traditional alloys may not be readily printable using techniques such as laser powder bed fusion. As such, the development of new alloys may be necessary to fully realize the benefits of additive manufacturing technologies.

In this example, inspired by the work of Ackers et al. [2021Ack], **Batch** calculations within the Property Model Calculator are used to identify candidate titanium alloy chemistries for biomedical implant applications using the thermodynamic TCS Ti/TiAl-based Alloys Database (TCTI). To optimize printability and material weight, the liquidus temperature, solidus temperature, and density of 2000 randomized titanium alloy chemistries from the Ti-Nb-Zr-Sn-Ta-Fe-Mo system are calculated. An estimate of alloy cost is also made for each alloy. Taken together, this methodology can be used to identify compositions of interest and accelerate material development activities.

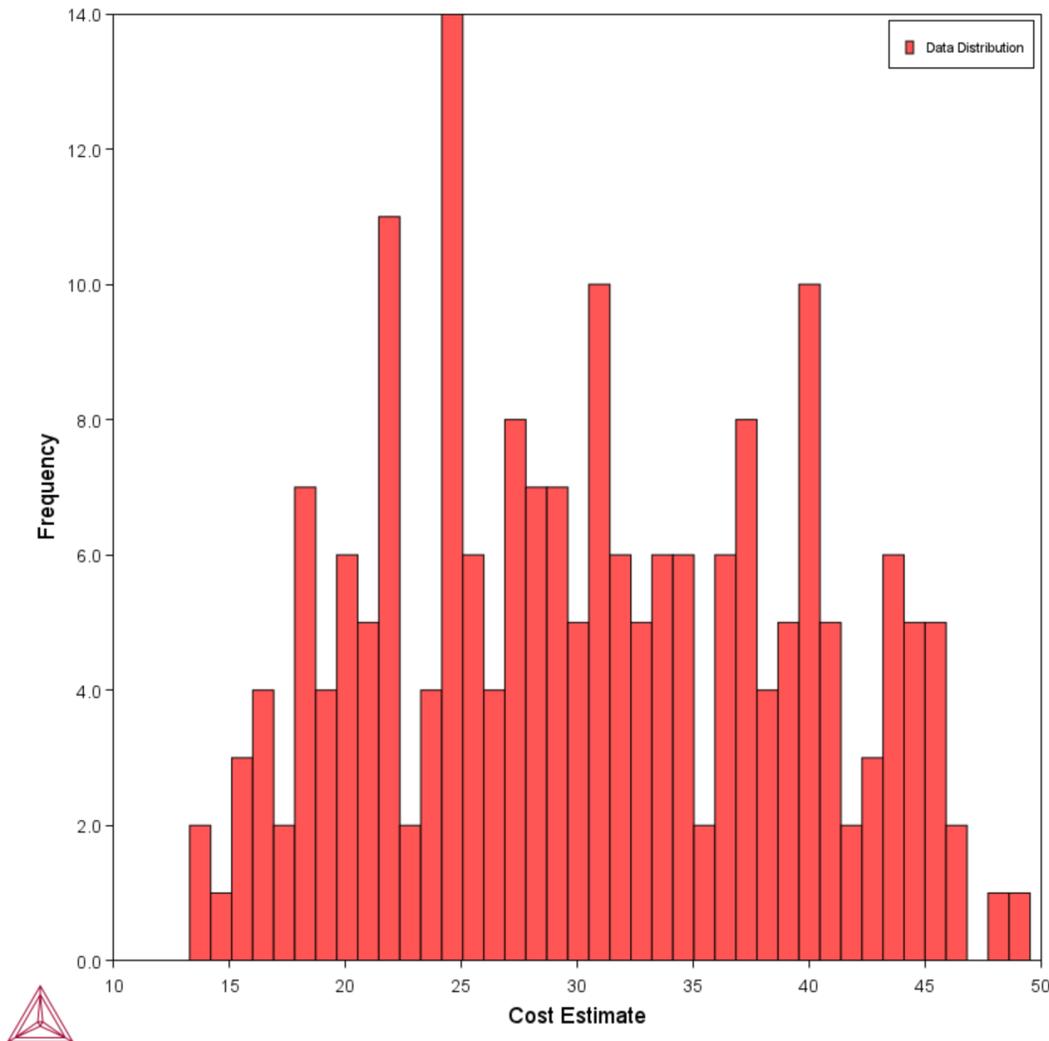


Figure 21: Cost estimate histogram for 2000 randomized chemistries from the Ti-Nb-Zr-Sn-Ta-Fe-Mo system. The cost estimate was calculated using a user-defined function in the Property Model Calculator.

### Project File Information

- Folder: **Property Models** → **General**
- File name: `PM_G_13_Ti_Alloy_Design_for_AM.tcu`



The TCS Ti/TiAl-based Alloys Database (TCTI) is used in this example. A valid license for version 4 (TCTI4) or newer is required to run the example.



The resulting plots and details related to setting up this example are available either via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

## Reference

[2021Ack] M. A. Ackers, O. M. D. M. Messé, U. Hecht, Novel approach of alloy design and selection for additive manufacturing towards targeted applications. *J. Alloys Compd.* 866, 158965 (2021).

## PM\_G\_14: Ti\_Fe\_T-Zero\_Martensite

---

The example uses the **Property Model Calculator** and the **T-Zero Temperature** Property Model to demonstrate the martensitic BCC  $\rightarrow$  HCP transformation for a binary Ti-Fe system. A setting is used to include a Gibbs energy addition to the second phase, which allows for the inclusion of an energy barrier for martensitic phase transformations.



[About the T0 Temperature Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Property Models**  $\rightarrow$  **General**
- File name: *PM\_G\_14\_Ti\_Fe\_T-Zero\_Martensite.tcu*



The TCS Ti/TiAl-based Alloys Database (TCTI) is used in this example. A valid license is required to run the example.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help**  $\rightarrow$  **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

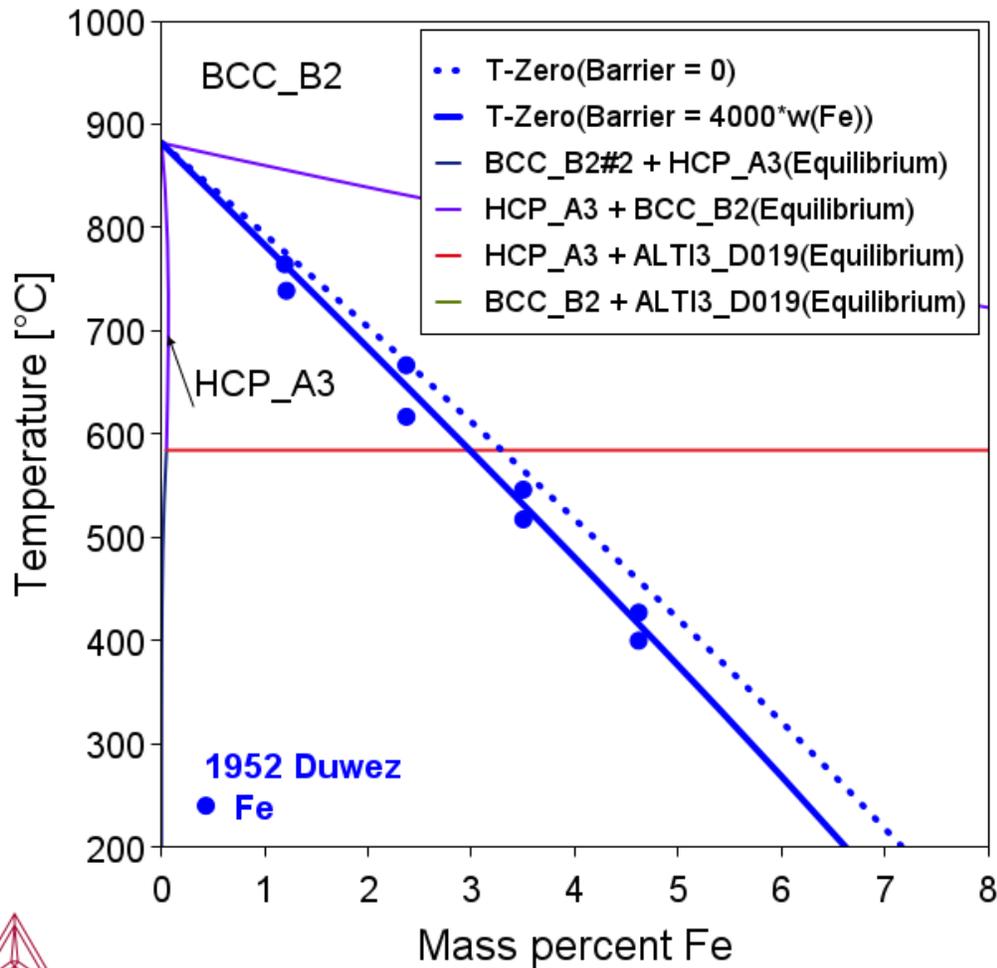


Figure 22: The T-Zero (barrier = 0) line is the temperature without energy addition, the solid blue line is the T-Zero temperature with an energy addition of  $4000 \cdot w(\text{Fe})$  [J/mol], that well reproduces the experimental data from Duwez (1952).

## Reference

[1952Duw] P. Duwez, "The Martensite Transformation Temperature in Titanium Binary Alloys, Third Technical Report" (U.S. Atomic Energy Commission, 1952) <https://doi.org/10.2172/4400681>.

## PM\_G\_15: Columnar to Equiaxed Transition (CET) of a NiAlCr Alloy

---

This example uses the **Property Model Calculator** and the **Columnar to Equiaxed Transition** Property Model with nickel demo databases to mimic the result of CMSX-4 alloy, which compares the tip radius and undercooling with growth velocity.

A Ni-26Al-9Cr (at.%) alloy is used so that its calculated undercooling is close to that of a 10-component CMSX-4 alloy. A **One Axis** calculation for the FCC\_L12 primary phase is done using the CET parameters entered on the **Property Model Calculator Configuration** window.

The settings used are the same as those from the literature [2001Gäu], i.e.:

- **Number of nucleation sites:**  $2.0E15/m^3$
- **Equiaxed exponent:** 3.4
- **Nucleation undercooling:** 2.5 K
- **Solve for:** Thermal gradient
- **Equiaxed fractions:** 0.0066 0.1



Also see these examples that use the same Property Model with the TCS Ni-based Superalloys Database (TCNI) and TCS Ni-alloys Mobility Database (MOBNI): [PM\\_G\\_16: Columnar to Equiaxed Transition of a CMSX-4 Alloy](#) and [PM\\_G\\_17: Columnar to Equiaxed Transition of an IN718 Alloy](#).



[About the Columnar to Equiaxed Transition \(CET\) Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Property Models** → **General**
- File name: *PM\_G\_15\_Ni-Al-Cr\_Columnar\_Equiaxed\_Transition.tcu*



The NIDEMO and MNIDEMO databases are used in this example and it is available to all users.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

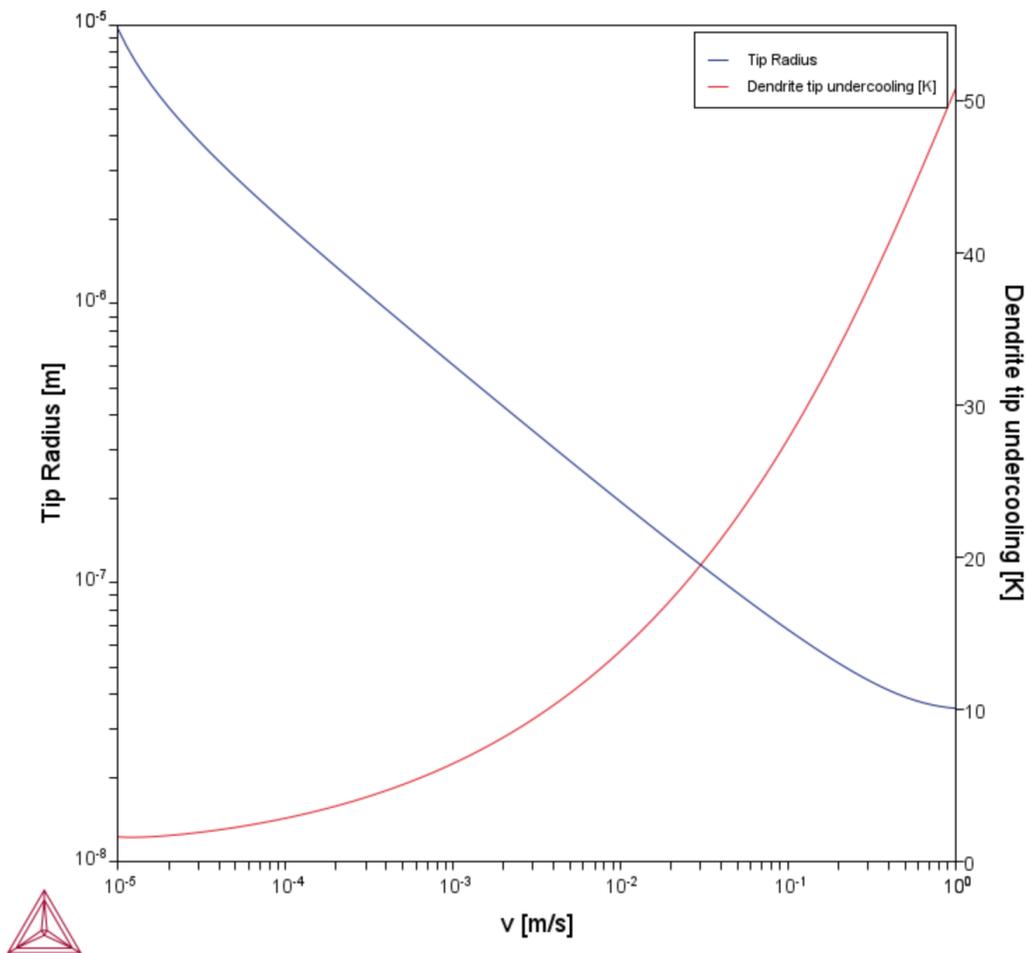


Figure 23: Tip radius and undercooling vs growth velocity for a Ni-26Al-9Cr alloy.

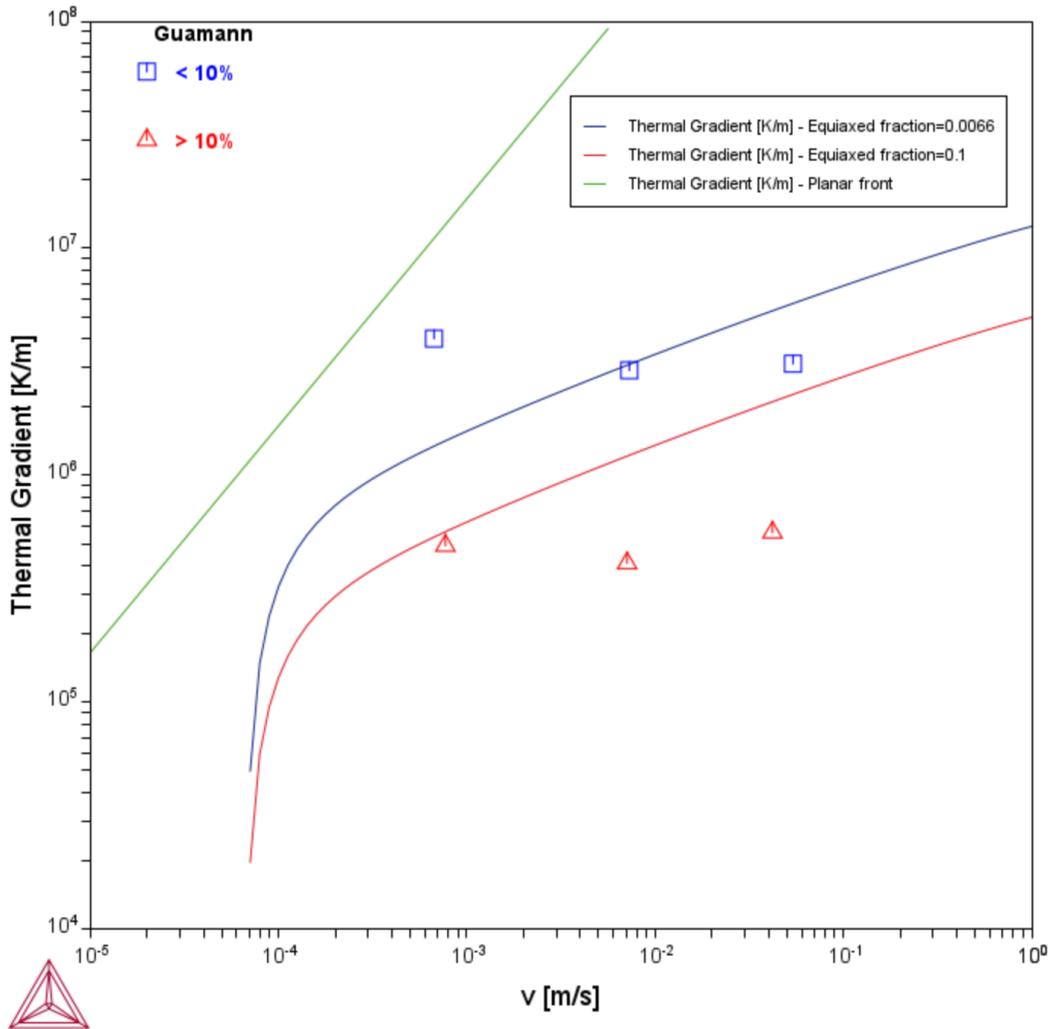


Figure 24: CET curves for a Ni-26Al-9Cr alloy using experimental data from [2001Gäu]. Also see PM\_G\_16 to compare the results to the CMSX-4 alloy results.

## Reference

[2001Gäu] M. Gäumann, C. Bezençon, P. Canalis, W. Kurz, Single-crystal laser deposition of superalloys: processing–microstructure maps. *Acta Mater.* 49, 1051–1062 (2001).

## PM\_G\_16: Columnar to Equiaxed Transition of a CMSX-4 Alloy

---

This example uses the **Property Model Calculator** and the **Columnar to Equiaxed Transition** Property Model with a CMSX-4 alloy to compare calculated CET curves to experimental data.



Also see these examples that use the same Property Model: [PM\\_G\\_15: Columnar to Equiaxed Transition \(CET\) of a NiAlCr Alloy](#) and [PM\\_G\\_17: Columnar to Equiaxed Transition of an IN718 Alloy](#).



[About the Columnar to Equiaxed Transition \(CET\) Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Property Models** → **General**
- File name: *PM\_G\_16\_CMSX-4\_Columnar\_Equiaxed\_Transition.tcu*



To run this example requires a license for the TCS Ni-based Superalloys Database (TCNI) version 12 (TCNI12) or newer and the TCS Ni-alloys Mobility Database (MOBNI) version 6 (MOBNI6) or newer.



Example [PM\\_G\\_15: Columnar to Equiaxed Transition \(CET\) of a NiAlCr Alloy](#) uses demo databases available to everyone and mimics this alloy in its results.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

A 10-component system, with a composition from Gäumann et al. [2001Gäu], is used for this simulation of a CMSX-4 alloy.

*Alloy composition of CMSX-4 [2001Gäu].*

	<i>Co</i>	<i>Cr</i>	<i>Al</i>	<i>Ti</i>	<i>W</i>	<i>Ta</i>	<i>Re</i>	<i>Hf</i>	<i>Mo</i>	<i>Ni</i>
wt.%	9.0	6.5	5.6	1.0	6.0	6.5	3.0	0.1	0.6	Bal
at.%	9.262	7.581	12.587	1.267	1.979	2.178	0.977	0.034	0.379	Bal

The results are compared with literature data (epitaxial laser metal forming) and the original model [1999Gäu; 2001Gäu].

A **One Axis** calculation for the FCC\_L12 primary phase is done using the CET parameters entered on the **Property Model Calculator Configuration** window.

The settings entered include:

- **Interfacial energy:** 0.25 J/m<sup>2</sup>, which gives a Gibbs-Thomson coefficient of 1.87 x 10<sup>-7</sup> mK, in agreement with the literature [2001Gäu].
- **Number of nucleation sites:** 2.0E15/m<sup>3</sup>
- **Nucleation undercooling :** 3.0 K
- **Equiaxed exponent:** 3.4
- **Solve for: Thermal gradient**
- **Equiaxed fractions:** 0.0066 0.1

In the calculation, the nucleation undercooling is slightly larger than originally used (2.5 K) [2001Gäu], since it has been found that at a low speed limit, the Property Model gives an undercooling close to 3 K.



The original value is arbitrary and does not affect the results at a high speed range.

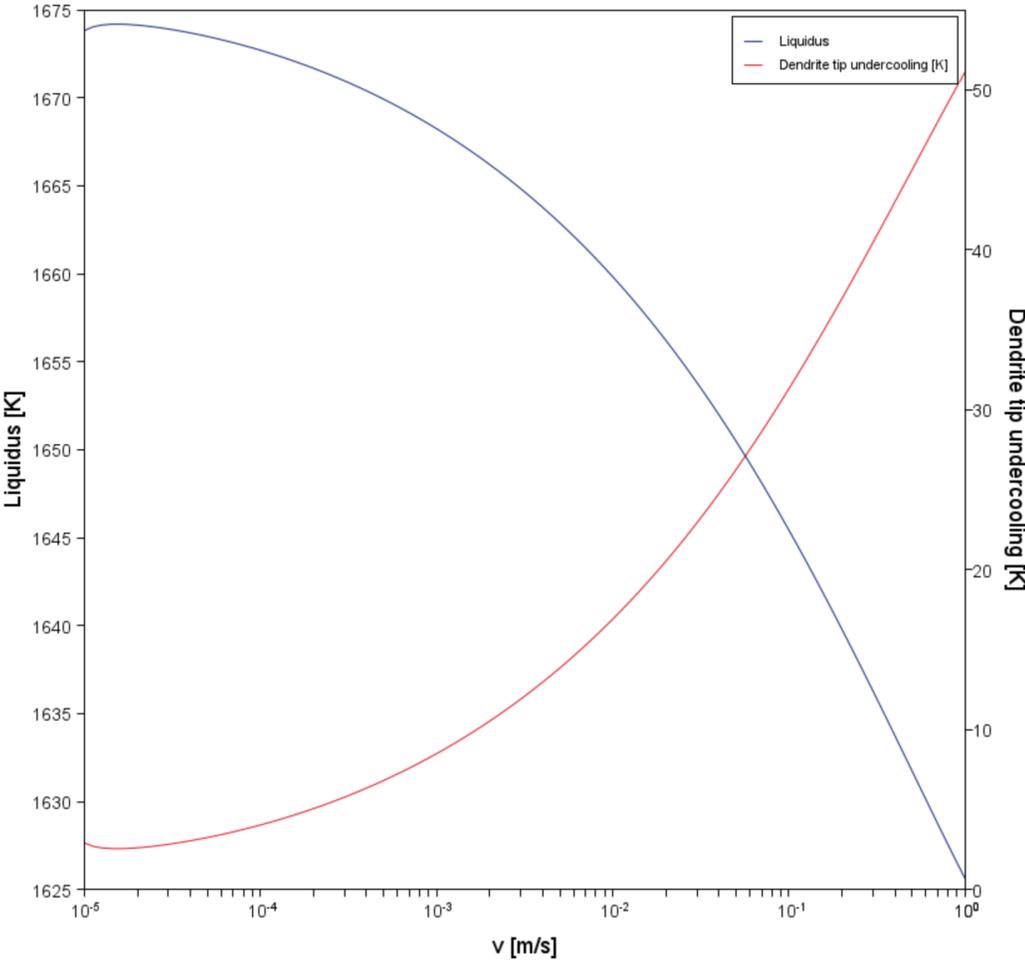


Figure 25: Tip radius and undercooling vs growth velocity for a CMSX-4 alloy.

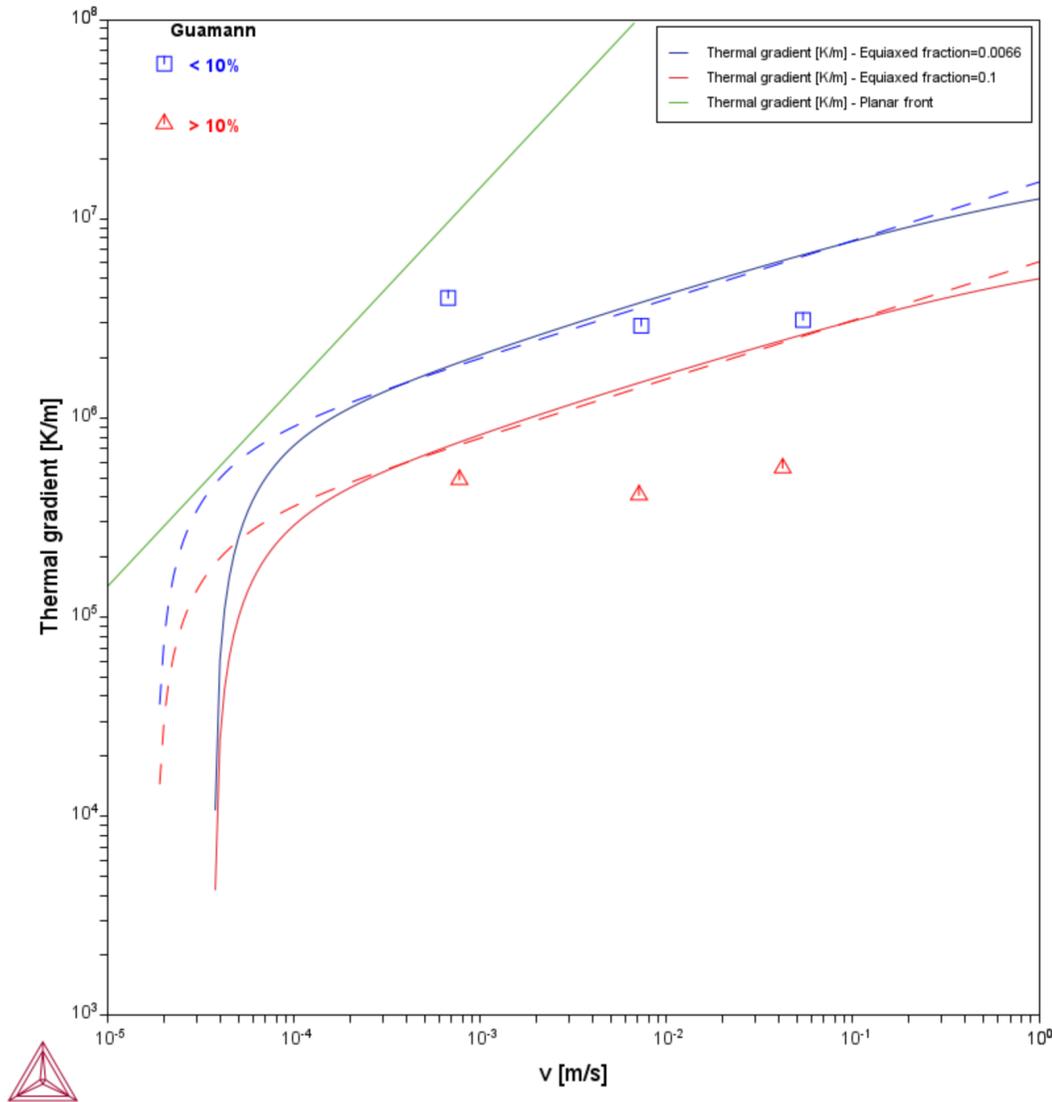


Figure 26: Calculated CET curves (solid lines) in comparison to experimental data (data points) and calculations (dashed lines) from the literature [2001Gäu].

## References

- [1999Gäu] M. Gäumann, PhD thesis, Epitaxial laser metal forming of a single crystal superalloy, École polytechnique fédérale de Lausanne (EPFL) (1999).
- [2001Gäu] M. Gäumann, C. Bezençon, P. Canalis, W. Kurz, Single-crystal laser deposition of superalloys: processing–microstructure maps. *Acta Mater.* 49, 1051–1062 (2001).

## PM\_G\_17: Columnar to Equiaxed Transition of an IN718 Alloy

---

The example uses the **Property Model Calculator** and the **Columnar to Equiaxed Transition** Property Model to compare calculated CET curves for an IN718 alloy to experimental data.



Also see these examples that use the same Property Model: [PM\\_G\\_15: Columnar to Equiaxed Transition \(CET\) of a NiAlCr Alloy](#) and [PM\\_G\\_16: Columnar to Equiaxed Transition of a CMSX-4 Alloy](#).



[About the Columnar to Equiaxed Transition \(CET\) Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Property Models** → **General**
- File name: *PM\_G\_17\_IN718\_Columnar\_Equiaxed\_Transition.tcu*



To run this example requires a license for the TCS Ni-based Superalloys Database (TCNI) version 12 (TCNI12) or newer and the TCS Ni-alloys Mobility Database (MOBNI) version 6 (MOBNI6) or newer.



Example [PM\\_G\\_15: Columnar to Equiaxed Transition \(CET\) of a NiAlCr Alloy](#) uses demo databases available to everyone.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

A **One Axis** calculation for the FCC\_L12 primary phase is done using the CET parameters entered on the **Property Model Calculator Configuration** window.

The settings entered include:

- **Interfacial energy:** 0.5 J/m<sup>2</sup>.
- **Number of nucleation sites:** 4.0E11/m<sup>3</sup>
- **Nucleation undercooling :** 4.0 K
- **Equiaxed exponent:** 3.13
- **Solve for: Thermal gradient**
- **Equiaxed fractions:** 0.01 0.49 0.99

The literature data from Polonsky et al. [2020Pol] are not purely experimental data, but rather combined with experimental equiaxed evidence and model calculations with numerical values of thermal gradients, growth velocity as well as tip undercooling. Polonsky et al. estimated tip undercooling based on a Scheil calculation, which smeared out the composition inhomogeneity at the dendrite front, hence underestimating the undercooling. Since both tip undercooling and nucleation site density promotes the formation of equiaxed crystals, an increase in undercooling in this example calculation leads to a decrease in nucleation site density compared to the results in [2020Pol] if one were fitting the same set of data.

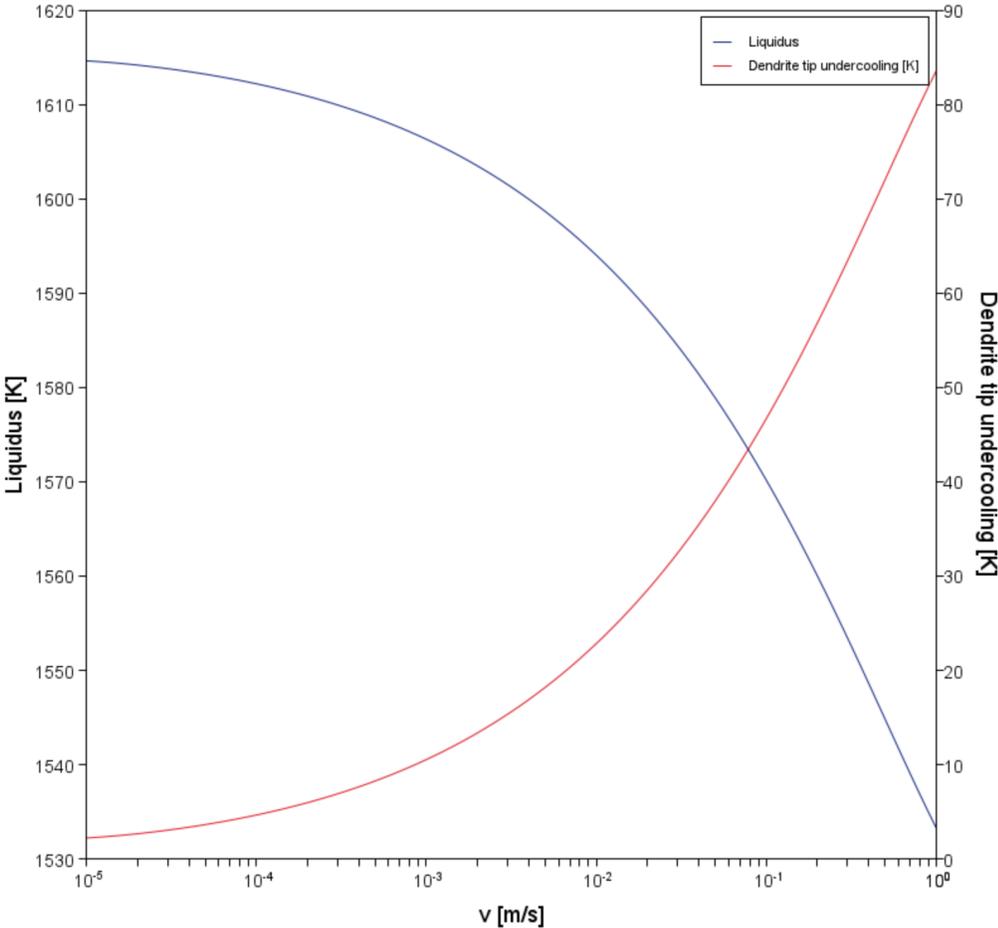


Figure 27: Tip radius and undercooling vs growth velocity for an IN718 alloy.

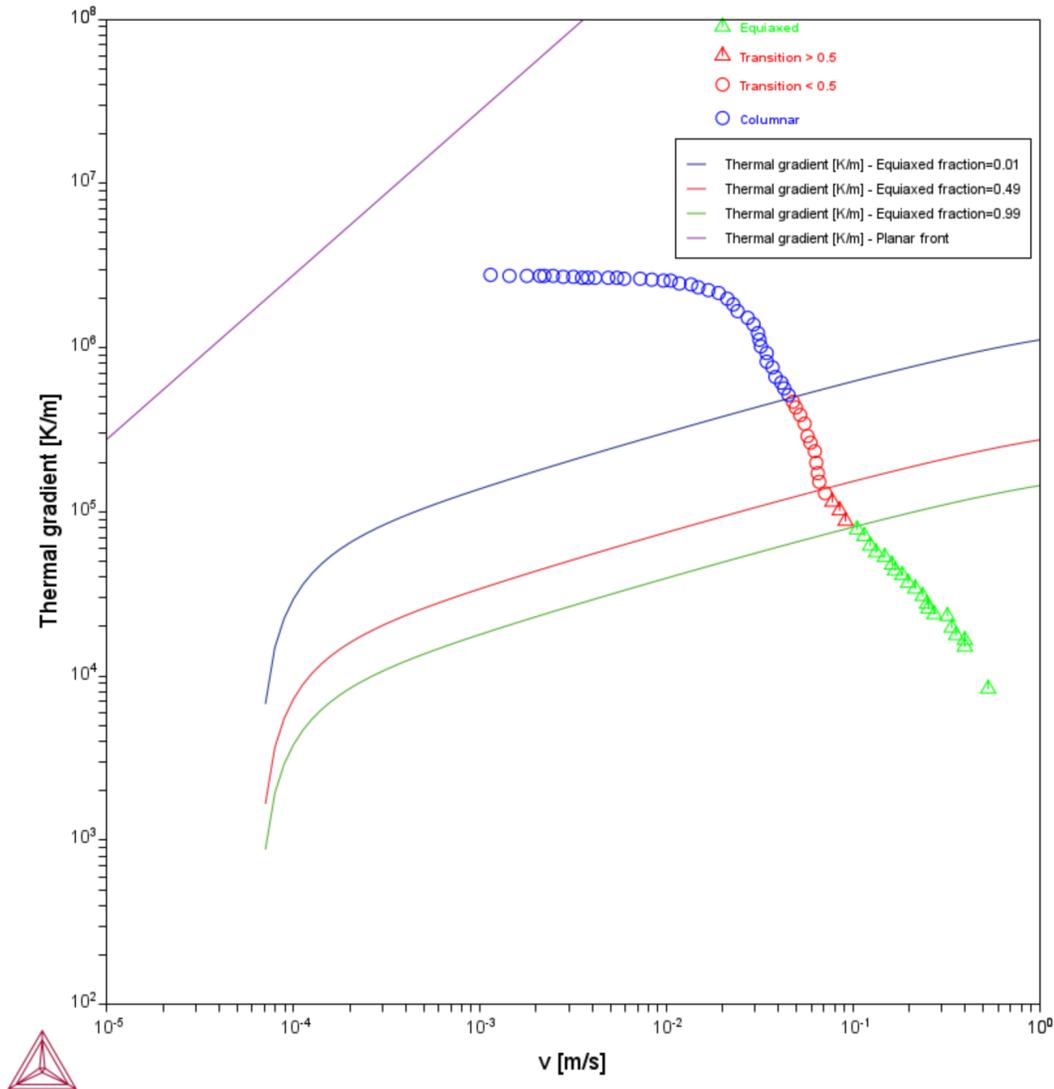


Figure 28: Calculated CET curves for IN718 in comparison with literature data [2020Pol].

## Reference

[2020Pol] A. T. Polonsky, N. Raghavan, M. P. Echlin, M. M. Kirka, R. R. Dehoff, T. M. Pollock, 3D Characterization of the Columnar-to-Equiaxed Transition in Additively Manufactured Inconel 718, in *Superalloys 2020* (2020), pp. 990–1002.

## Steel Model Library Examples Collection

---



The **General Models** are available to all users. To run calculations with the **Steel Models** (as part of the Steel Model Library) requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases.

These examples use the **Property Model Calculator**, an activity available with Thermo-Calc plus Property Models in the Steel Model Library.



[About the Examples](#)

### In this section:

---

PM_Fe_01: Fe-Cr-C Martensite with Intercritical Annealing .....	106
PM_Fe_02: Fe-Mn Martensite Morphologies .....	109
PM_Fe_03: Fe-C-Mn Pearlite .....	111
PM_Fe_04: Critical Temperatures .....	115
PM_Fe_05: Fe-C-Mn-Si-Ni-Cr-Mo Bainite .....	118
PM_Fe_06: TTT Diagram for Fe-C-Mn-Si-Cr-V .....	120
PM_Fe_07: Hardenability Design of Steel .....	123
PM_Fe_08: CCT Diagram for Fe-C-Mn-Si-Cr-V .....	125
PM_Fe_09: Fe-C-Ni Ferrite .....	128
PM_Fe_10: Martensitic Steel Strength .....	132

---

## PM\_Fe\_01: Fe-Cr-C Martensite with Intercritical Annealing

The example uses the **Property Model Calculator** with the **Martensite Fractions** and **Martensite Temperatures** Models to calculate martensite fractions and martensite temperatures [martensite start (Ms) and 90% transformation temperature (M90)].

The alloy is first intercritically annealed and then quenched. Austenite composition is determined by an equilibrium calculation at the annealing temperature. Martensite fraction is calculated as a function of temperature which the alloy is quenched to. The example also shows how Cr content in the alloy influences Ms and M90 after intercritical annealing.



[Property Model Calculator](#), [About the Martensite Temperatures Property Model](#), and [About the Martensite Fractions Property Model](#)



This example is included as a Property Model tutorial on our [website](#) and as part of the Property Model Calculator playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

### Project File Information

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_01\_Fe-Cr-C\_martensite\_intercritical\_annealing.tcu*

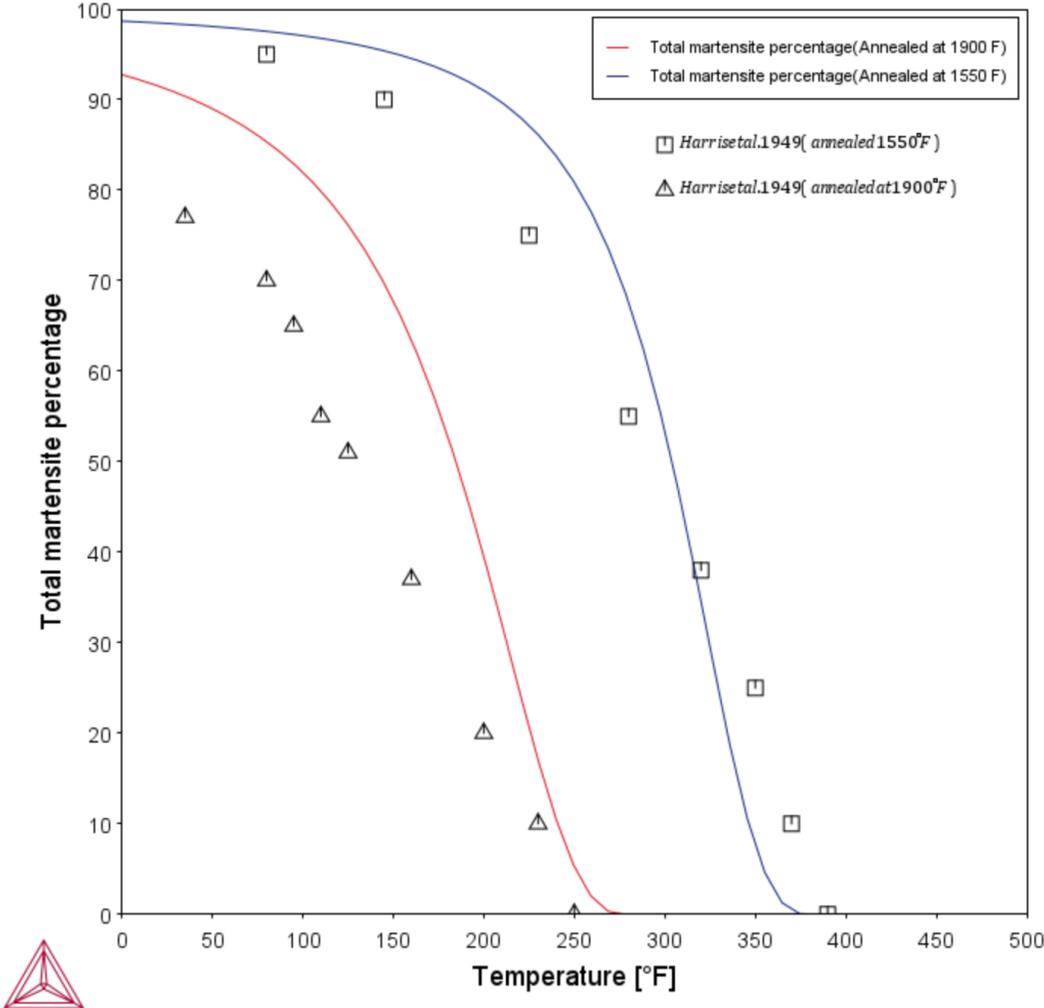


Figure 1: The transformation curves plot showing Fe-Cr-C martensite with intercritical annealing.

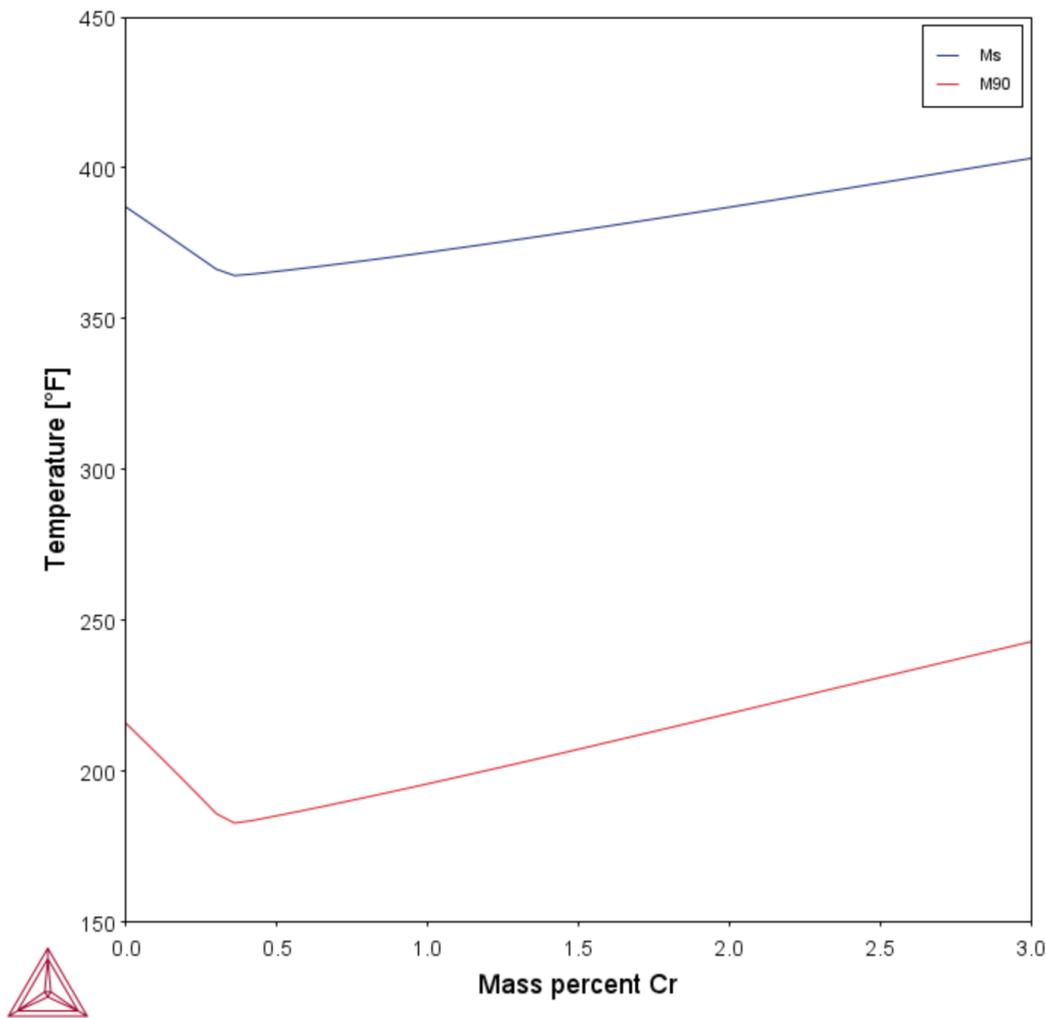


Figure 2: Comparing Martensite start ( $M_s$ ) to Martensite finish ( $M_f$ ).

## Reference

[1949Har] Harris, William J, and Morris Cohen. 1949. "Stabilization Of The Austenite-Martensite Transformation." Trans. AIME 180: 447–70.

## PM\_Fe\_02: Fe-Mn Martensite Morphologies

---

The example uses the **Property Model Calculator** with the **Martensite Temperatures** Property Model to calculate the  $M_s$  temperatures of different types of martensites: lath, plate, and  $\epsilon$  (HCP), compared with experimental  $\epsilon$   $M_s$  values taken from several sources.



[Property Model Calculator](#) and [About the Martensite Temperatures Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

### ***Project File Information***

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_02\_Fe-Mn\_Martensite\_Morphologies.tcu*

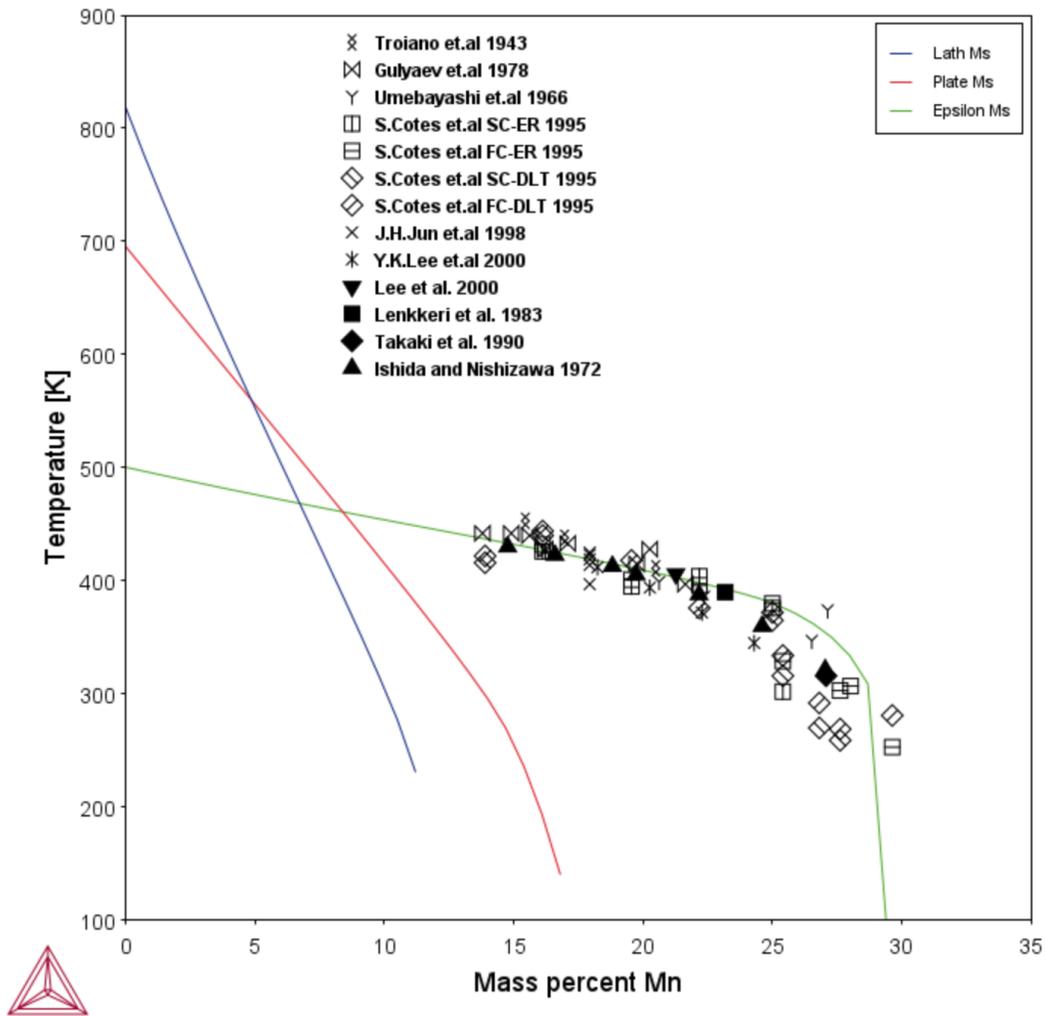


Figure 3: A diagram showing all the Ms temperatures of different types of martensite morphologies (lath, plate and epsilon (HCP) compared with experimental epsilon Ms values.

## PM\_Fe\_03: Fe-C-Mn Pearlite

---

The example uses the **Property Model Calculator** and with the **Pearlite** Property Model and shows how to calculate pearlite growth rate, lamellar spacing, and times of start (2% transformation) and finish (98% transformation) as functions of isothermal heat treating temperature in an Fe-0.69C-1.80Mn alloy (mass %). With *maximize growth rate* set as the *criterion*, the model gives maximal growth rate and minimal lamellar spacing. With *optimal pearlite* as the *pearlite mode*, the model optimizes partitioning of substitutional alloying element(s) (Mn in this example) according to the criterion, which realizes a smooth transition between ortho-pearlite at high temperature and para-pearlite at low temperature.



[Property Model Calculator](#) and [About the Pearlite Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

### **Project File Information**

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_03\_Fe-C-Mn\_Pearlite.tcu*

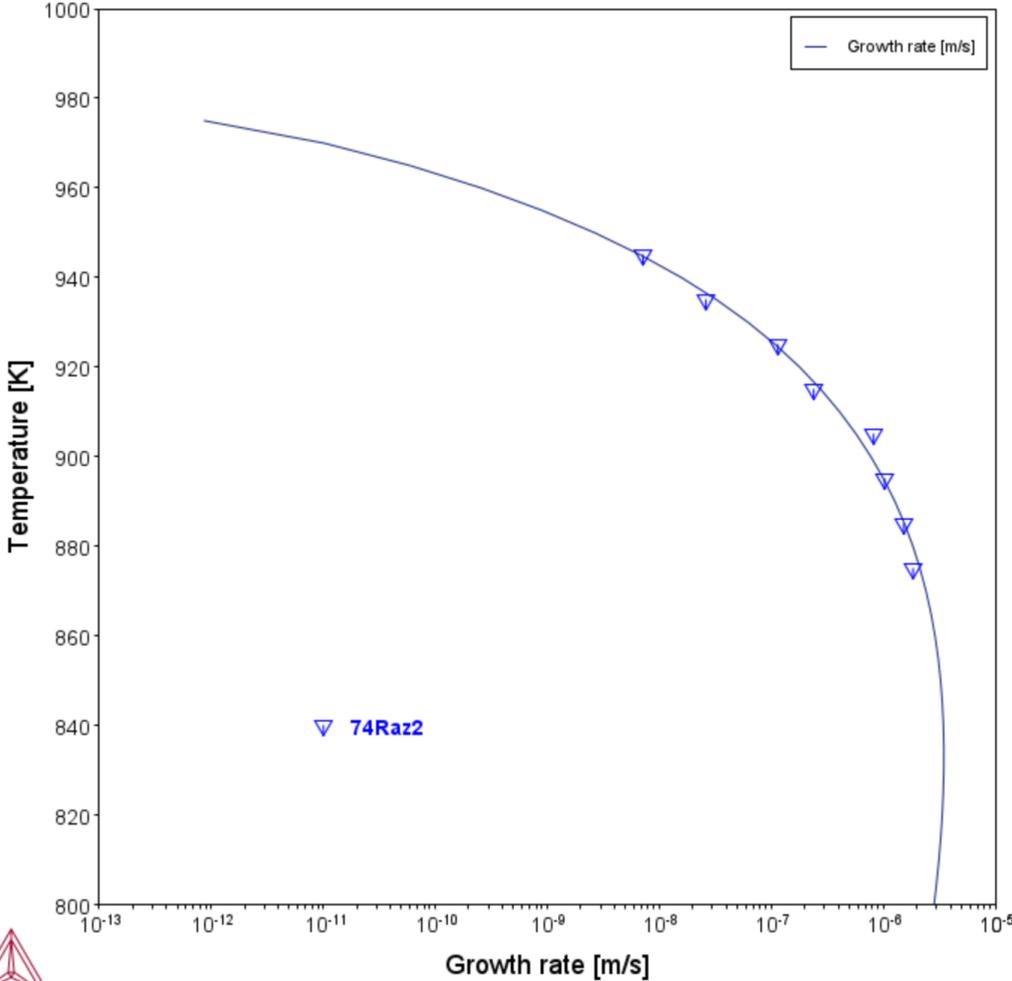


Figure 4: Growth rate as a function of temperature.

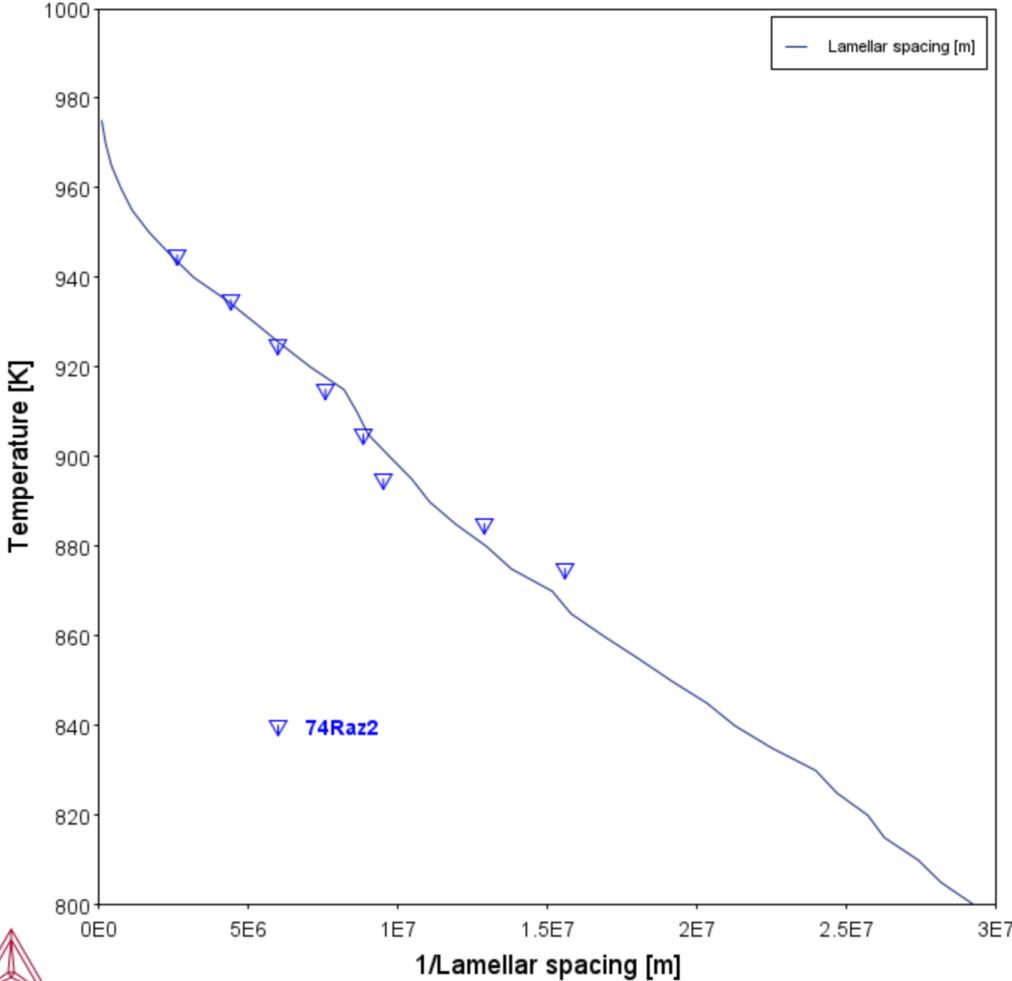


Figure 5: Lamellar spacing as a function of temperature.

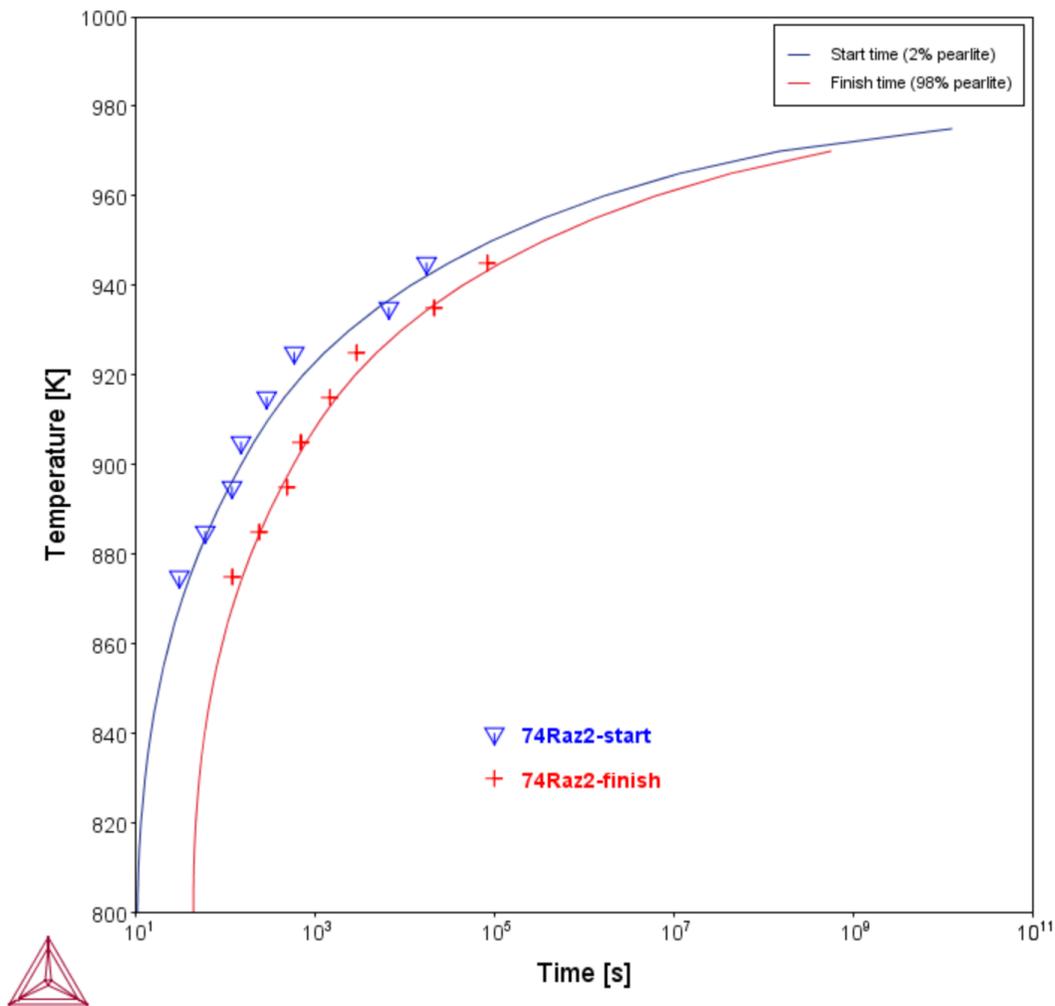


Figure 6: The TTT (time-temperature-transformation) diagram showing times of start (2% transformation) and finish (98% transformation) as functions of isothermal heat treating temperature in an Fe-0.69C-1.80Mn alloy (mass %).

## Reference

[1974Raz] Razik, N.A, G.W. Lorimer, and N Ridley. 1974. "An Investigation of Manganese Partitioning during the Austenite-Pearlite Transformation Using Analytical Electron Microscopy." *Acta Metallurgica* 22 (10): 1249–58.

## PM\_Fe\_04: Critical Temperatures

---

The example uses the **Property Model Calculator** and the **Critical Transformation Temperatures Model** to calculate the distribution of the typical phase transition temperatures for a low alloy steel (Fe-0.3Cr-1.0Mn-0.3Mn-0.18C) when the composition is varied within the specification for the alloy.

In the set up of the example, the **Critical Transformation Temperatures** model is used with a Property Model Calculator **Uncertainty** calculation to plot a histogram showing the distribution of A1- and A3-temperatures ([Figure 7](#)). As a comparison, an Equilibrium Calculator, **One Axis** calculation results in a property diagram showing the phase transitions for the nominal composition ([Figure 8](#)).

The phase transition temperatures are defined as:

- Liquidus: First austenite or ferrite transformation from the liquid
- Solidus: Liquid fully transformed to solid
- A0: Magnetic transition temperature (Curie temperature) of cementite. The cementite is paramagnetic above A0 and ferromagnetic below
- A1: Austenite (FCC\_A1) transforms to ferrite (BCC\_A2) + carbide (cementite or graphite or M<sub>23</sub>C<sub>6</sub>)
- A2: Magnetic transition temperature (Curie temperature) of ferrite (BCC\_A2). Paramagnetic above A2 and ferromagnetic below
- A3: Austenite (FCC\_A1) transforms to ferrite (BCC\_A2)



[Property Model Calculator](#) and [About the Critical Transformation Temperatures Property Model](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.



The **FEDEMO: Iron Demo Database** is used and this example is available to all users. However, in general a license is required to run the Steel Models.

### Project File Information

- Folder: **Property Models** → **Steel**
  - File name: *PM\_Fe\_04\_Critical\_Temperatures.tcu*
-

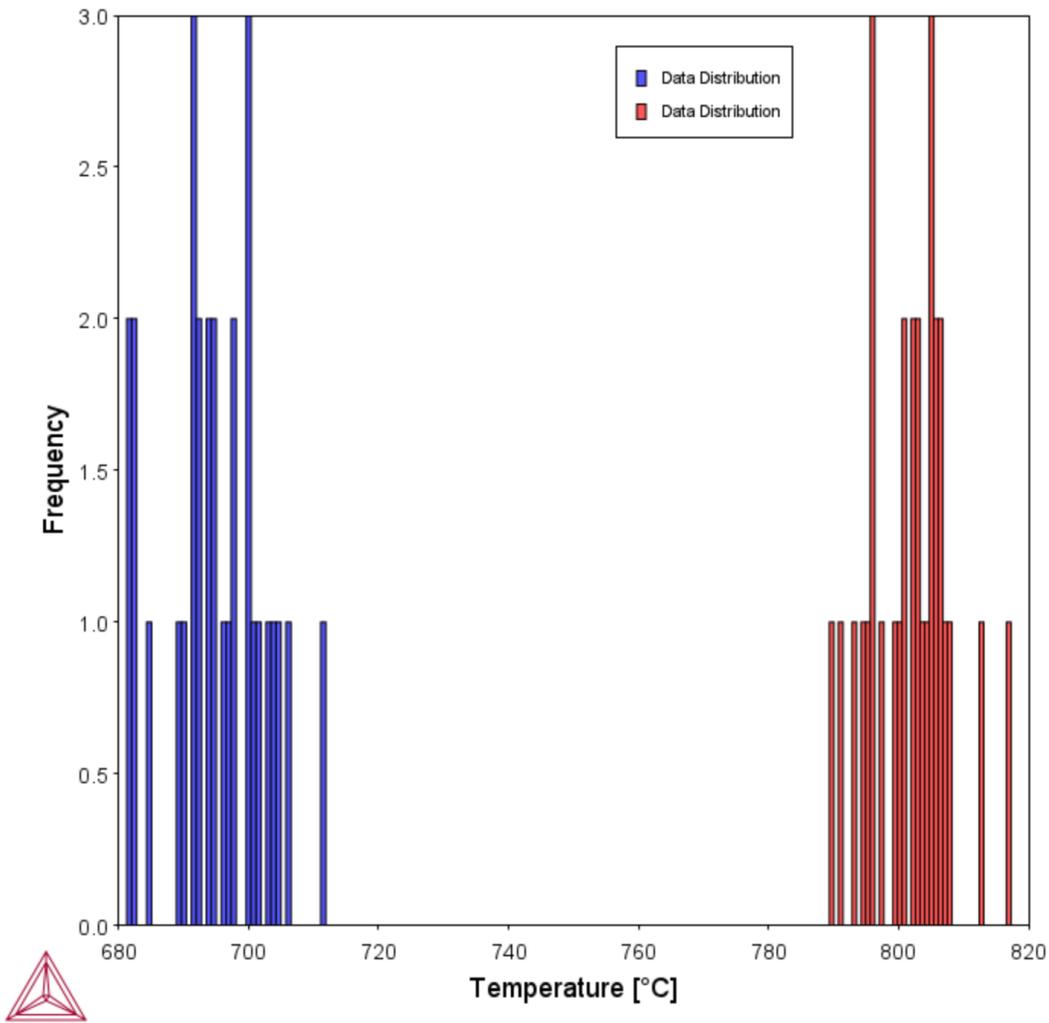


Figure 7: The distribution of the A1 and A3 phase transition temperatures for a low alloyed steel (Fe-0.3Cr-1.0Mn-0.3Mn-0.18C) when the composition is varied within the specification. This plot uses the Critical Transformation Temperatures model with the Property Model Calculator.

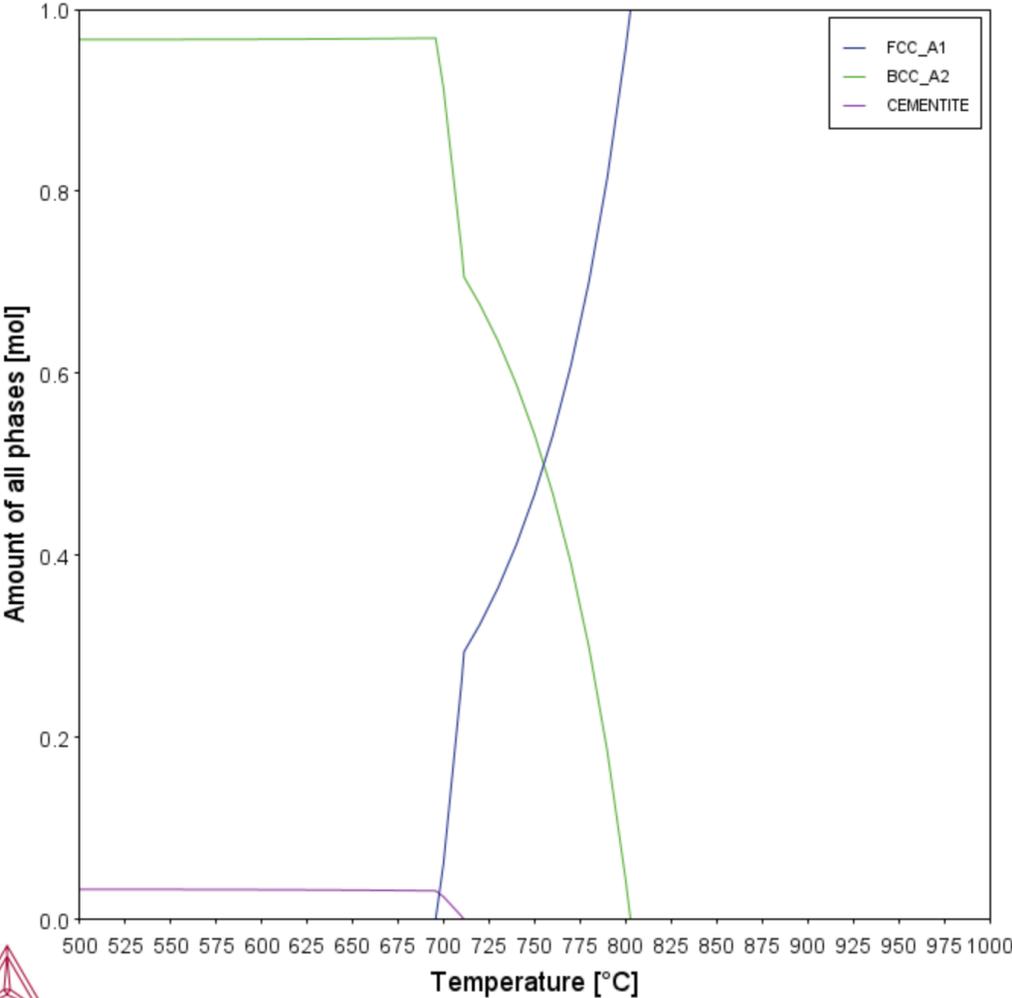


Figure 8: This plot uses an Equilibrium Calculator to show an alternate visualization of the phase transitions in a property diagram for the nominal composition.

## PM\_Fe\_05: Fe-C-Mn-Si-Ni-Cr-Mo Bainite

---

The example uses the **Property Model Calculator** and the **Bainite Steel Model** to calculate a Time-Temperature-Transformation (TTT) diagram for an Fe-0.97C-0.72Mn-0.32Si-1.54Ni-0.8Cr-0.26Mo alloy. The result is compared to experimental results from [1948Jaf]. Other elements with low amount in the experimental alloy are omitted for the calculation.



Considered elements: Fe, C, Mn, Si, Cr, Ni, and Mo. Other elements in the system are neglected for bainite by mass percent.

A Property Model Calculator is used with a *One Axis* calculation and the *TTT mode* selected on the Plot Renderer to plot the TTT diagram.



For more information see [About the Bainite Property Model](#), [Plot Type: TTT Mode](#), and [Plot Types](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_05\_Fe-C-Mn-Si-Ni-Cr-Mo\_Bainite.tcu*

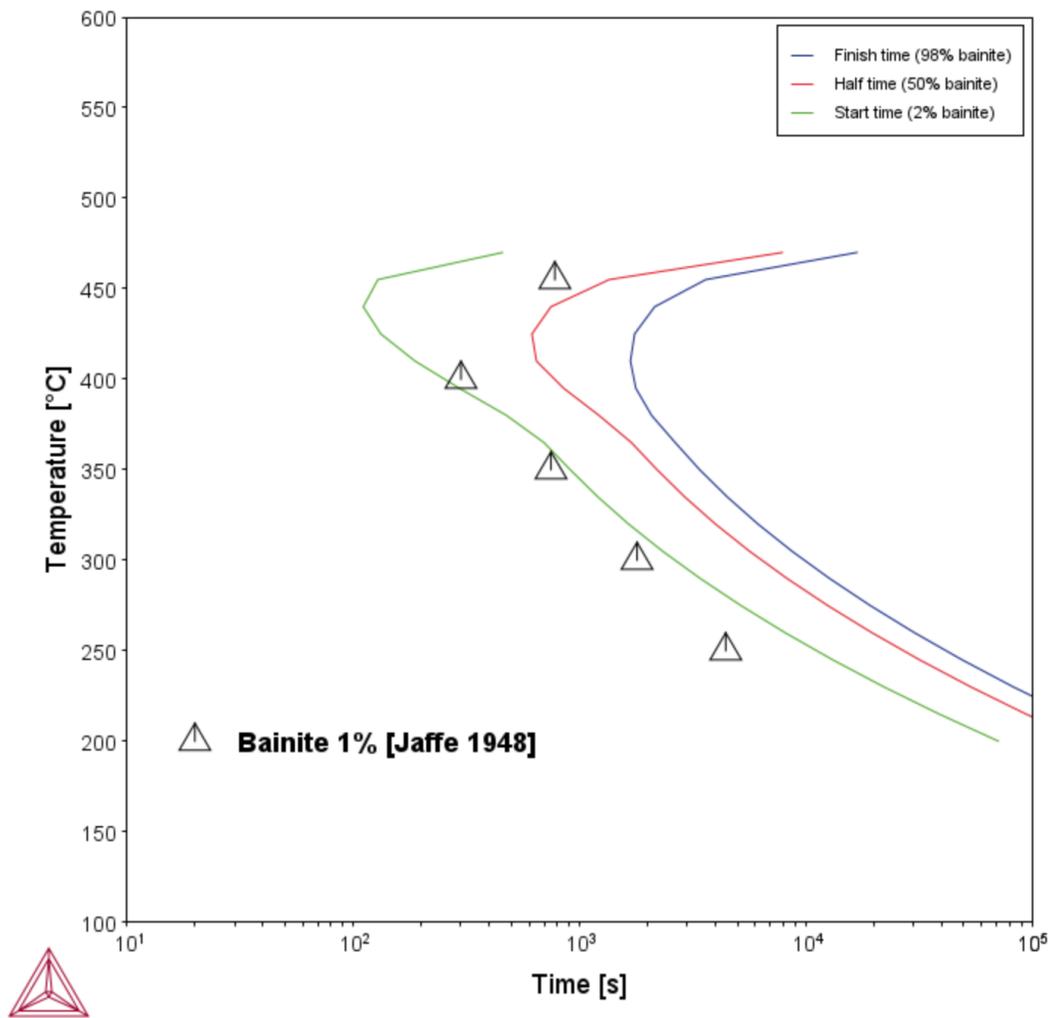


Figure 9: The bainite TTT diagram for an Fe-0.97C-0.72Mn-0.32Si-1.54Ni-0.8Cr-0.26Mo alloy.

## Reference

[1948Jaf] L. D. Jaffe, Anisothermal formation of bainite and proeutectoid constituents in steels, Trans. AIME, vol. 176, pp. 343–383 (1948).

## PM\_Fe\_06: TTT Diagram for Fe-C-Mn-Si-Cr-V

---

The example uses the **Property Model Calculator** and the **TTT Diagram** Property Model to calculate a Time-Temperature-Transformation (TTT) diagram for an Fe-C-Mn-Si-Cr-V alloy (Steel 42CrV6). The calculation is compared to an experimental TTT diagram for the same steel (with some other impurities) from data in [1958Wev].

A **One Axis Calculation Type** is selected on the Property Model Calculator and then the **TTT Mode** selected on the Plot Renderer as the *Plot type* in order to plot the final TTT diagram.

A second plot also shows the calculated terminal fractions of ferrite, pearlite, and bainite.



For more information see [About the TTT Diagram Property Model](#), [TTT Diagram Property Model Settings](#), and [Plot Type: TTT Mode](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.



When you run (Perform) this example, it takes several hours to complete the calculations.

### Project File Information

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_06\_Fe-C-Mn-Si-Cr-V\_TTT.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

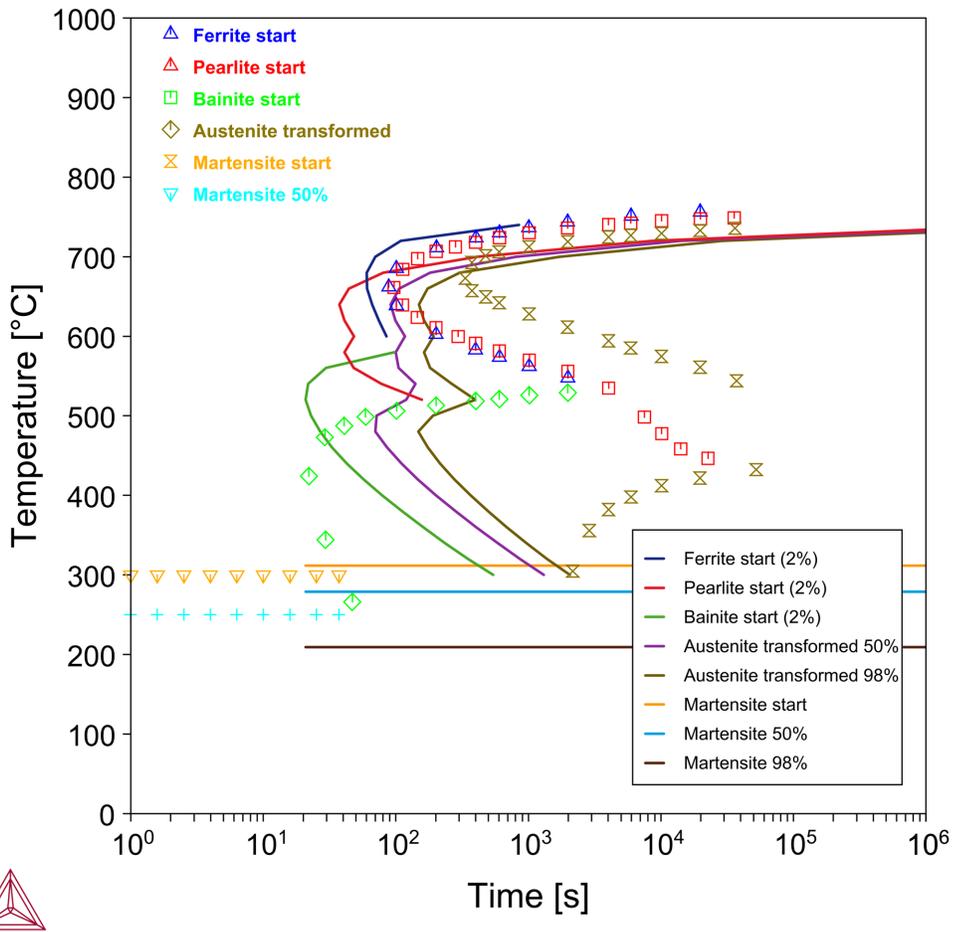


Figure 10: The TTT diagram for an Fe-C-Mn-Si-Cr-V alloy comparing the calculation with experimental data from [1958Wev].

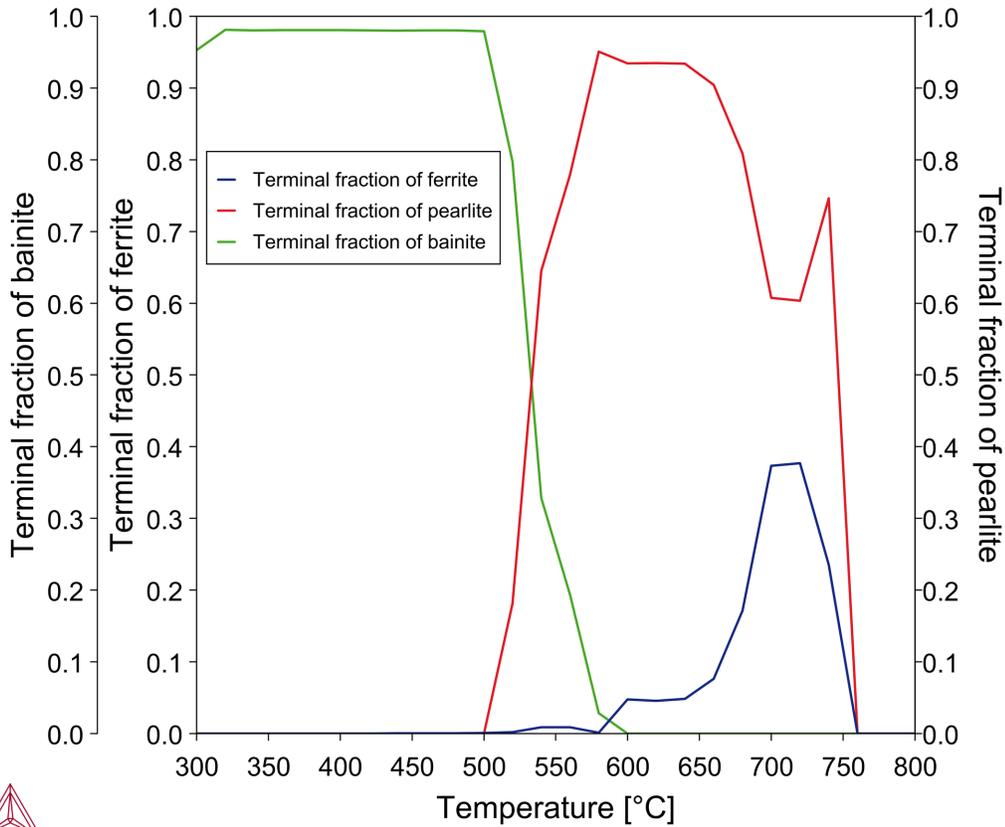


Figure 11: The terminal fractions quantities for bainite, ferrite, and pearlite are plotted against the cooling rate for the Fe-C-Mn-Si-Cr-V alloy.

## Reference

[1958Wev] F. Wever, A. Rose, Atlas zur Wärmebehandlung der Stähle: 1954/56/58 (Verlag Stahleisen, Düsseldorf, 1958), p. II-112.

## PM\_Fe\_07: Hardenability Design of Steel

---



When you run (Perform) this example, it takes several hours to complete the calculations.



It takes several minutes to open this advanced example file.

This application example, which is [showcased on our website](#), shows how the Steel Model Library in Thermo-Calc can be used to find the optimal compositions for an Fe–Mn–C steel to achieve high hardenability for the purpose of strength.

This example focuses on the hardenability design of steel using the Steel Model Library in Thermo-Calc. It shows how to find the optimal compositions for Fe–Mn–C steel to achieve high hardenability for the purpose of strength. Hardenability refers to the ability of steel to form martensite on quenching. It is a measure of the capacity of a steel to be hardened in depth when quenched from its austenitizing temperature. Among various factors, composition is one of the most important factors which has great influence on the hardenability of the steel. In this example, we use the Steel Model Library to investigate the possible composition ranges of Fe–C–Mn alloys to reach a fully martensitic microstructure.

To fulfil the requirement, room temperature martensite fraction should be large and the amount of retained austenite should be small. Meanwhile, other products of austenite decomposition should be avoided during continuous cooling. Such transformation products can be suppressed by fast quenching, yet the cooling rate is usually limited by the capacity of facilities or other problems such as cracking. Therefore, for hardenability purposes, it is desirable to achieve a fully martensitic structure with a relatively low cooling rate. This is done by promoting martensite formation while retarding other transformations through adjusting the steel composition.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



Although this example uses the FEDEMO and MFEDEMO databases, running the calculation requires a license for Thermo-Calc 2022b or newer and for the Steel Model Library.

## ***Project File Information***

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_07\_Hardenability\_Design\_of\_Steel.tcu*



The resulting plots and details related to setting up this example are available either via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

---

## PM\_Fe\_08: CCT Diagram for Fe-C-Mn-Si-Cr-V

---

The example uses the **Property Model Calculator** and the **CCT Diagram** Property Model to calculate a Continuous-Cooling-Transformation (CCT) diagram for an Fe-C-Mn-Si-Cr-V alloy (Steel 42CrV6). The calculation is compared to an experimental CCT diagram for the same steel (with some other impurities) from data in [1958Wev].

A **One Axis Calculation Type** is selected on the Property Model Calculator and then the **CCT Mode** selected as the *Plot type* on the Plot Renderer in order to plot the final CCT diagram.

A second plot also shows the calculated terminal fractions of ferrite, pearlite, bainite, and martensite.



For more information see [About the CCT Diagram Property Model](#), [CCT Diagram Property Model Settings](#), and [Plot Type: CCT Mode](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.



When you run (Perform) this example, it takes several hours to complete the calculations.

### Project File Information

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_08\_Fe-C-Mn-Si-Cr-V\_CCT.tcu*

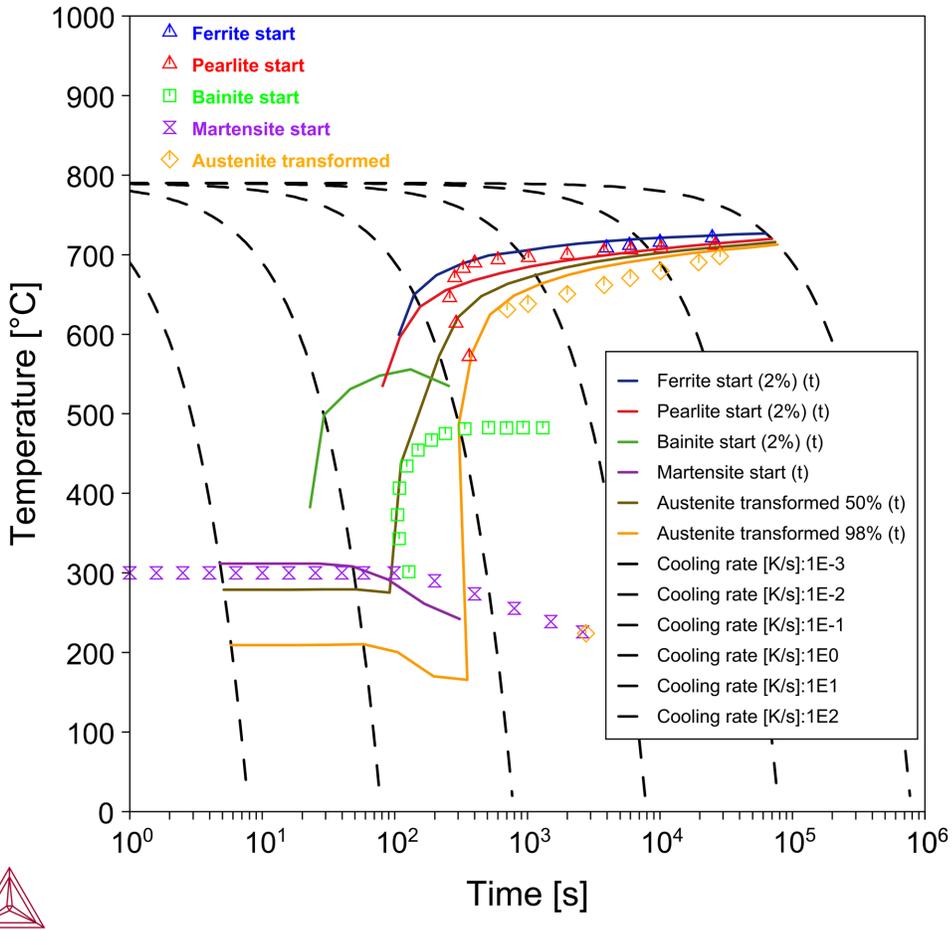


Figure 12: The CCT diagram for an Fe-C-Mn-Si-Cr-V alloy comparing the calculation with experimental data from [1958Wev].

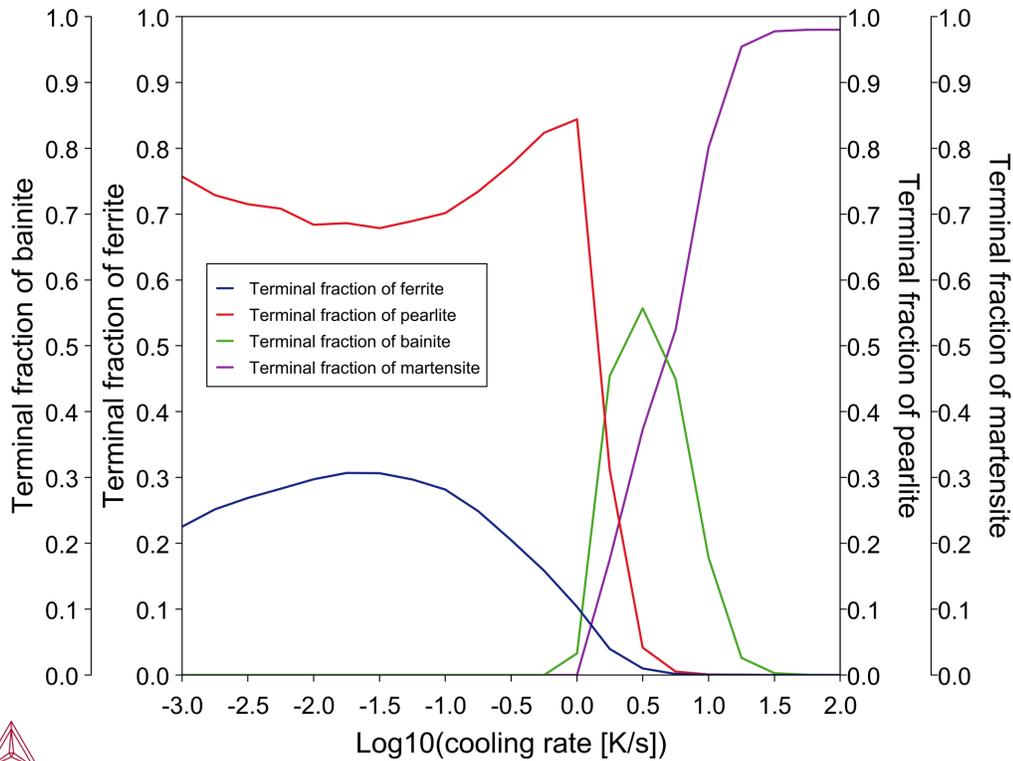


Figure 13: The terminal fractions quantities for bainite, ferrite, martensite, and pearlite are plotted against the cooling rate for the Fe-C-Mn-Si-Cr-V alloy.

## Reference

[1958Wev] F. Wever, A. Rose, Atlas zur Wärmebehandlung der Stähle: 1954/56/58 (Verlag Stahleisen, Düsseldorf, 1958), p. II-112.

## PM\_Fe\_09: Fe-C-Ni Ferrite

---

The example uses the **Property Model Calculator** and the **Ferrite** Property Model to calculate the ferrite fraction parabolic rate constant and ferrite start time for an Fe-0.12C-3.28Ni alloy.

Two growth modes—Orthoequilibrium (OE) and Paraequilibrium (PE)—are used with a *One Axis* calculation type to compare the ferrite molar fraction and to determine the ferrite start time for each growth mode.

The experimental data for the ferrite start is taken from [1966Aar]. For the parabolic rate constant experimental data is from [1981Bra].



For more information about the Property Model, see [About the Ferrite Property Model](#) and [Ferrite Property Model Settings](#).



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### **Project File Information**

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_09\_Fe-C-Ni\_Ferrite.tcu*

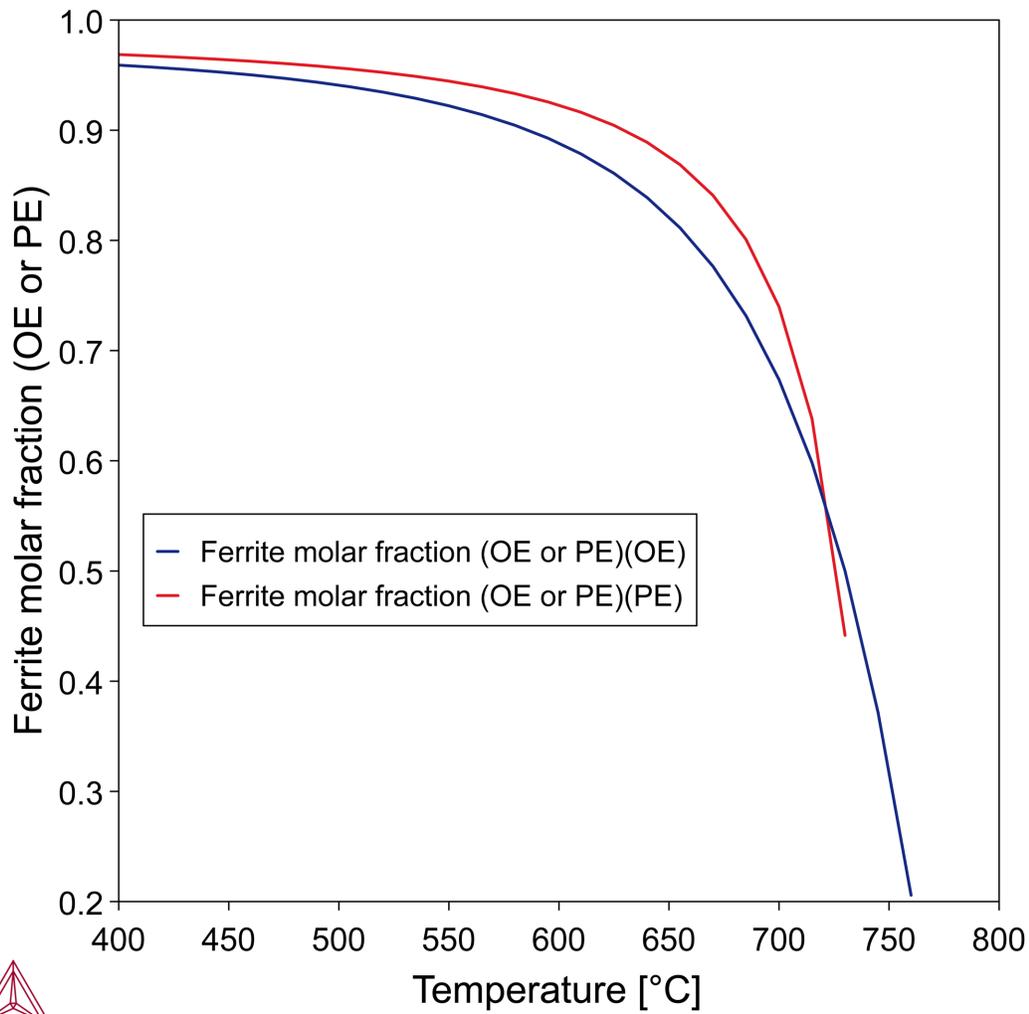


Figure 14: Ferrite fraction at the end of the transformation, under orthoequilibrium (OE) or paraequilibrium (PE).

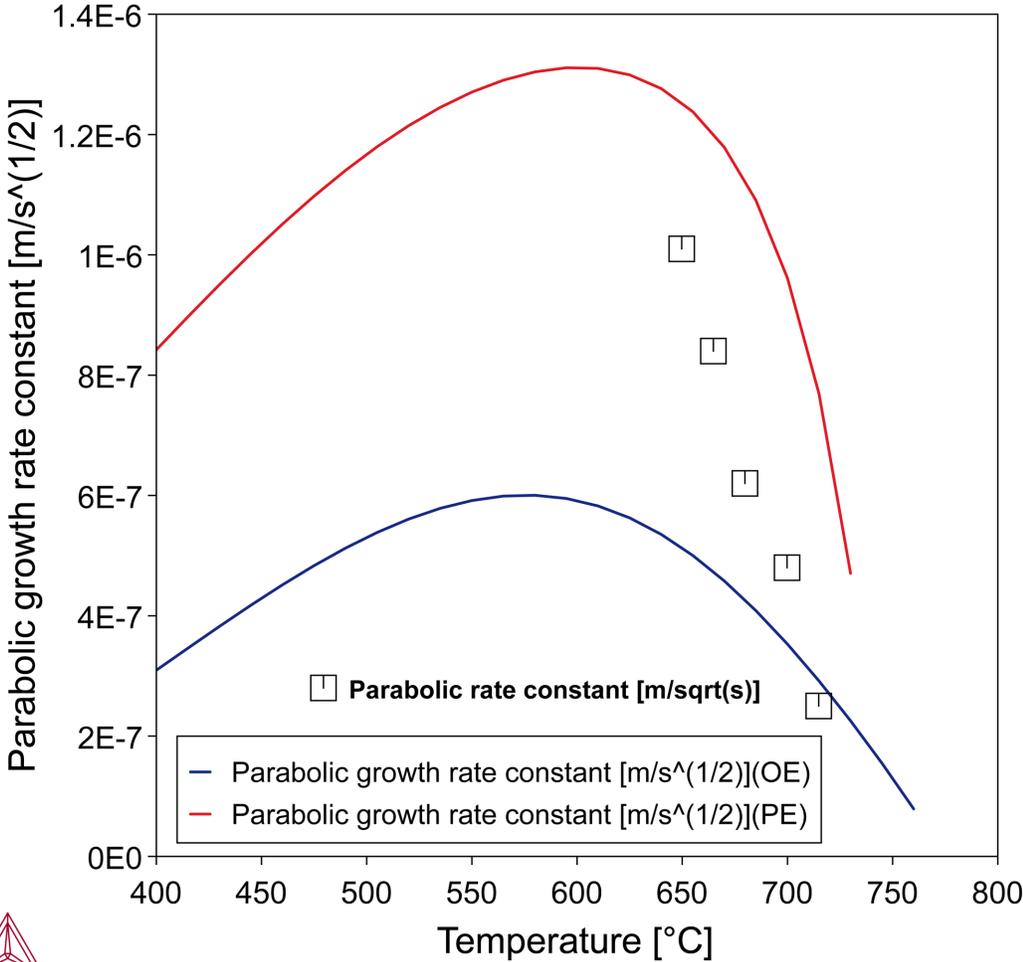


Figure 15: Parabolic rate constant. Experimental data is from [1981Bra].

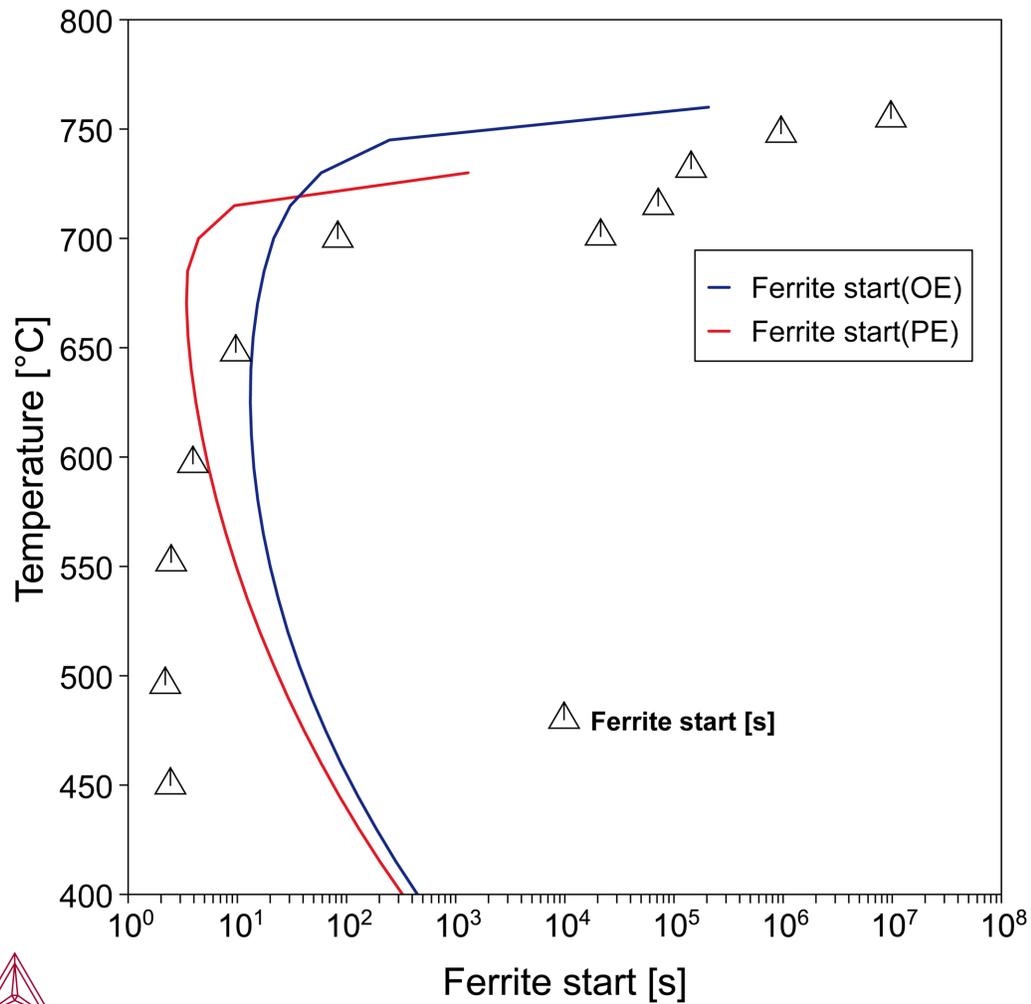


Figure 16: Ferrite start time. This plot also uses the TTT mode. Experimental data from [1966Aar].

## References

- [1966Aar] H. I. Aaronson, H. A. Domian, Partition of alloying elements between austenite and proeutectoid ferrite or bainite. AIME MET SOC TRANS. 236, 781–796 (1966).
- [1981Bra] J. R. Bradley, H. I. Aaronson, Growth kinetics of grain boundary ferrite allotriomorphs in Fe-C-X alloys. Metall. Trans. A. 12, 1729–1741 (1981).

## PM\_Fe\_10: Martensitic Steel Strength

---

The **Martensitic Steel Strength** Property Model, available with the Property Model Calculator and the Steel Model Library, calculates the strength and/or hardness of martensitic steels.



For more information about the Property Model, see [About the Martensitic Steel Strength Property Model](#) and [Martensitic Steel Strength Property Model Settings](#).



To run calculations with the **Steel Models** requires a valid maintenance license plus licenses for both the TCFE (version 9 and newer) and MOBFE (version 4 and newer) databases. Also see our website to learn more about the [Steel Model Library](#) and other related examples.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Property Models** → **Steel**
- File name: *PM\_Fe\_10\_Martensitic\_Steel\_Strength.tcu*



This example uses TCFE12. Earlier versions of this database can be used although there may be slight differences in the results.

In this example, a variety of handbook data for different AISI steels (1030, 1095, 1144, and 4130) are compared to calculations using the Martensitic Steel Strength model. In the set up, two System Definers and two Experimental File Reader activities are used along with four Property Model Calculators. The **Martensitic Steel Strength** Property Model is used in all cases with **One Axis** calculations to produce two plots that compare the tempering temperature to the total hardness of tempered steels. Data from the ASM Handbook [2013Pen] is included.



For some Windows-based systems, performing the full calculation from the top **My Project** node can result in out of memory issues. If this happens, perform individual calculations for each Property Model Calculator (e.g. right-click the node and select **Perform Now**). This produces the full plots that are output to the Visualizations window.

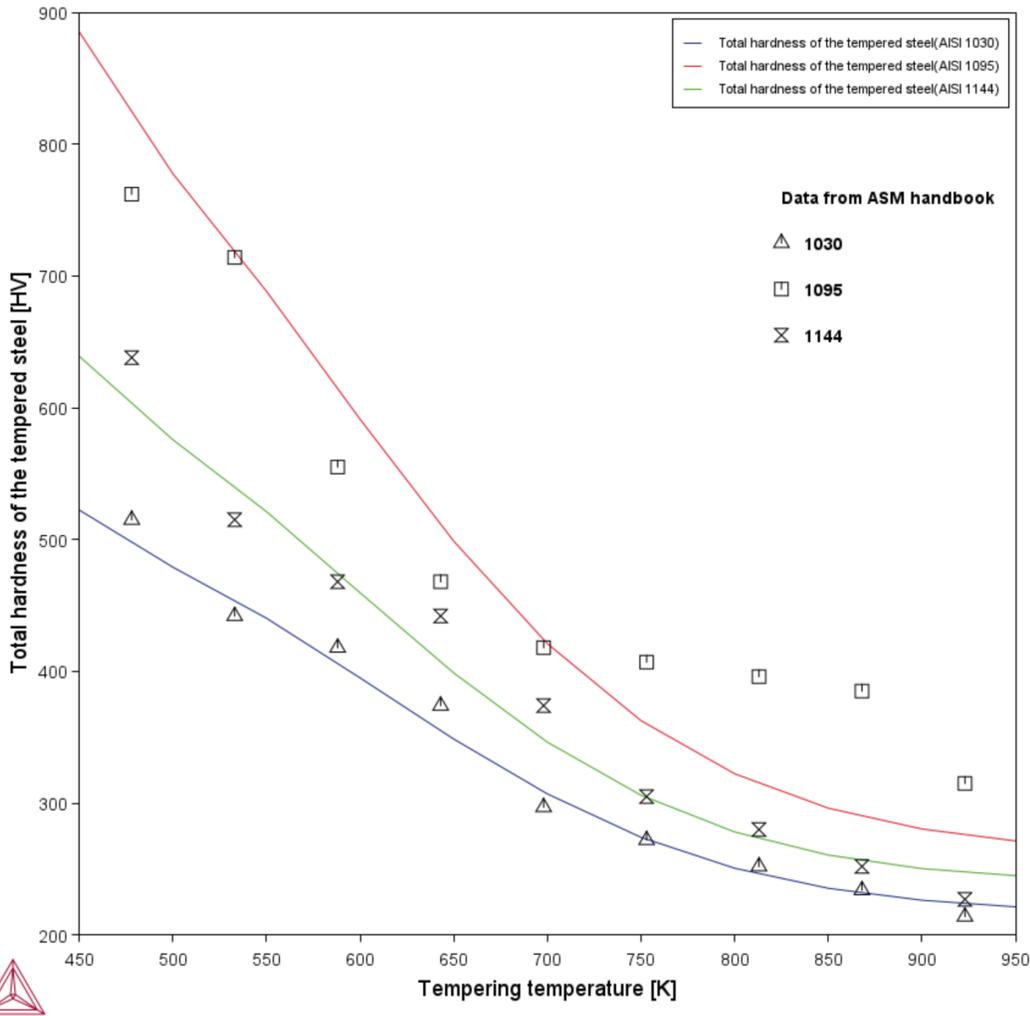


Figure 17: Tempering temperature compared to Total hardness of the tempered steel compared to handbook data for 1030, 1095, and 1144 steels [2013Pen].

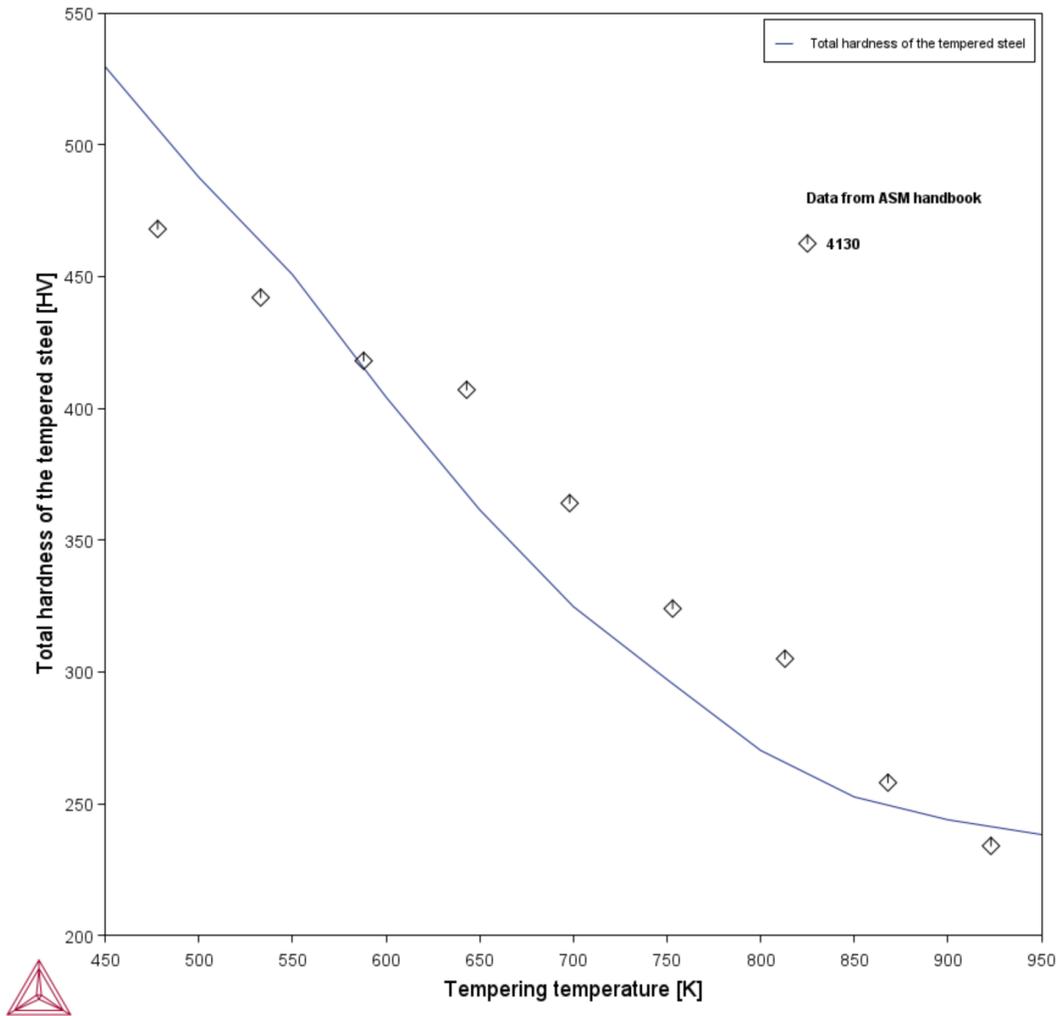


Figure 18: Tempering temperature compared to Total hardness of the tempered steel compared to handbook data for a 4130 steel [2013Pen].

## Reference

[2013Pen] R. N. Penha, L. C. F. Canale, J. Vatauvuk, S. Lampman, Tempering of Steels, in Steel Heat Treating Fundamentals and Processes (ASM International, 2013), pp. 327–351.

## Nickel Model Library Examples Collection

---



The **General Models** are available to all users. To run calculations with the **Nickel Models** (as part of the Nickel Model Library) requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases.

These examples use the **Property Model Calculator**, an activity available with Thermo-Calc plus the available Property Models in the Nickel Model Library.



[About the Examples](#)

### In this section:

---

PM_Ni_01: Lattice Parameter of $\gamma/\gamma'$ .....	136
PM_Ni_02: Antiphase Boundary Energy of $\gamma'$ .....	138
PM_Ni_03: Critical Temperatures of Alloy 718 .....	140
PM_Ni_04: Strain Age Cracking (SAC) .....	142

---

## PM\_Ni\_01: Lattice Parameter of $\gamma/\gamma'$

The example uses the **Property Model Calculator** with the **Equilibrium with Freeze-in Temperature - Ni Model**.



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).

### Project File Information

- Folder: **Property Models** → **Nickel**
- File name: *PM\_Ni\_01\_Lattice\_Parameter\_of\_Gamma\_Gamma\_Prime.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



[Property Model Calculator](#) and [About the Equilibrium with Freeze-in Temperature Nickel Property Model](#)

The lattice parameters are measured experimentally for  $\gamma/\gamma'$  in a Ni0.6Mo0.92Ta12.5Al1.83Ti10.5Cr3.3W alloy.

The thermodynamically stable equilibrium will not be reached for this alloy during the experimental heat treatment; only the phases  $\gamma$  and  $\gamma'$  are noticed in the experiment. The  $\gamma/\gamma'$  microstructure is in this calculation example assumed to freeze-in at 1000 °C where the phase compositions do not change during cooling to room temperature.

Some of the key settings to note in this example:

- The **Subset of phases** is selected as **Gamma and gamma-prime only** to match the experimental observations.
- The **Freeze-in temperature** is set to 1000 °C.
- A **One Axis** calculation is used where the evaluation temperatures are within a range of 20 °C to 1000 °C.

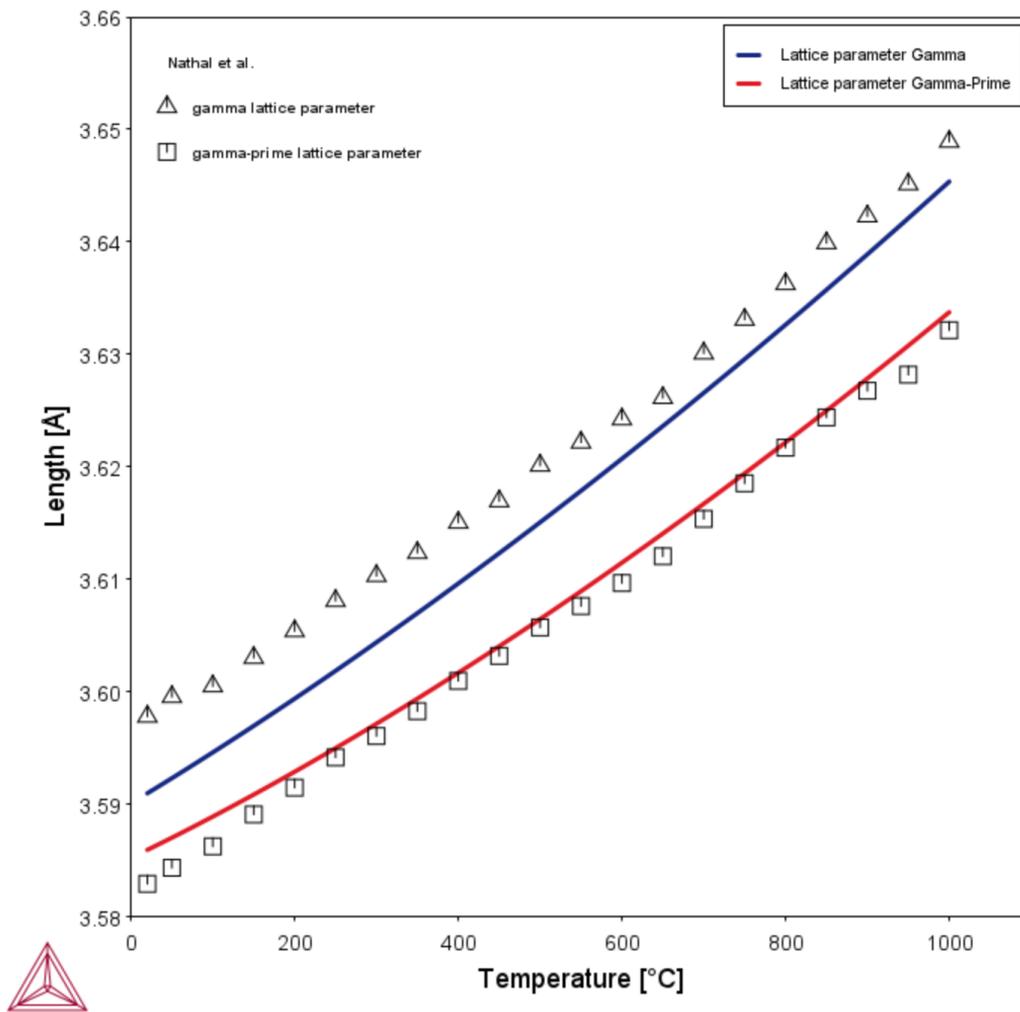


Figure 1: The lattice parameters for  $\gamma$  and  $\gamma'$  are plotted against experimental data from [1985Nat].

## Reference

[1985Nat] M. V. Nathal, R. A. Mackay, R. G. Garlick, Temperature dependence of  $\gamma$ - $\gamma'$  lattice mismatch in Nickel-base superalloys. Mater. Sci. Eng. 75, 195–205 (1985).

## PM\_Ni\_02: Antiphase Boundary Energy of $\gamma'$

The example uses the **Property Model Calculator** with the **Antiphase Boundary Energy - Ni** Model.



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).

### Project File Information

- Folder: **Property Models** → **Nickel**
- File name: *PM\_Ni\_02\_Antiphase\_Boundary\_Energy\_of\_Gamma\_Prime.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



[Property Model Calculator](#) and [About the Antiphase Boundary Energy Property Model](#)

This example shows a calculation of the antiphase boundary energy for an Al<sub>75</sub>Ni alloy at room temperature with increasing amounts of Ti. The plot shows the (111) APB energy in Ni<sub>3</sub>Al<sub>1-x</sub>Ti<sub>x</sub> over the whole compositional range together with first-principle calculations from Chandran and Sondhi [2011Cha] and Vamsi and Karthikeyani [2012Vam] evaluated at 0 K. The trend with increasing 111 APBE with an increasing amount of Ti is predicted by all calculations but the absolute values between the first principle calculations are quite different.

Some of the key settings to note in this example:

- $\gamma'$  is the only stable phase for these compositions. FCC\_L12 is therefore selected as the only phase in the System Definer.
- The **Freeze-in temperature** and evaluation temperature are both set at 20 °C with the **Subset of phases** for **All phases**.
- A **One Axis** calculation is used with **Mole percent Ti** between 0 and 25 in 20 steps.
- The plotting quantity **APBE for 111 plane (1/2)** is then compared with the first principle calculations.

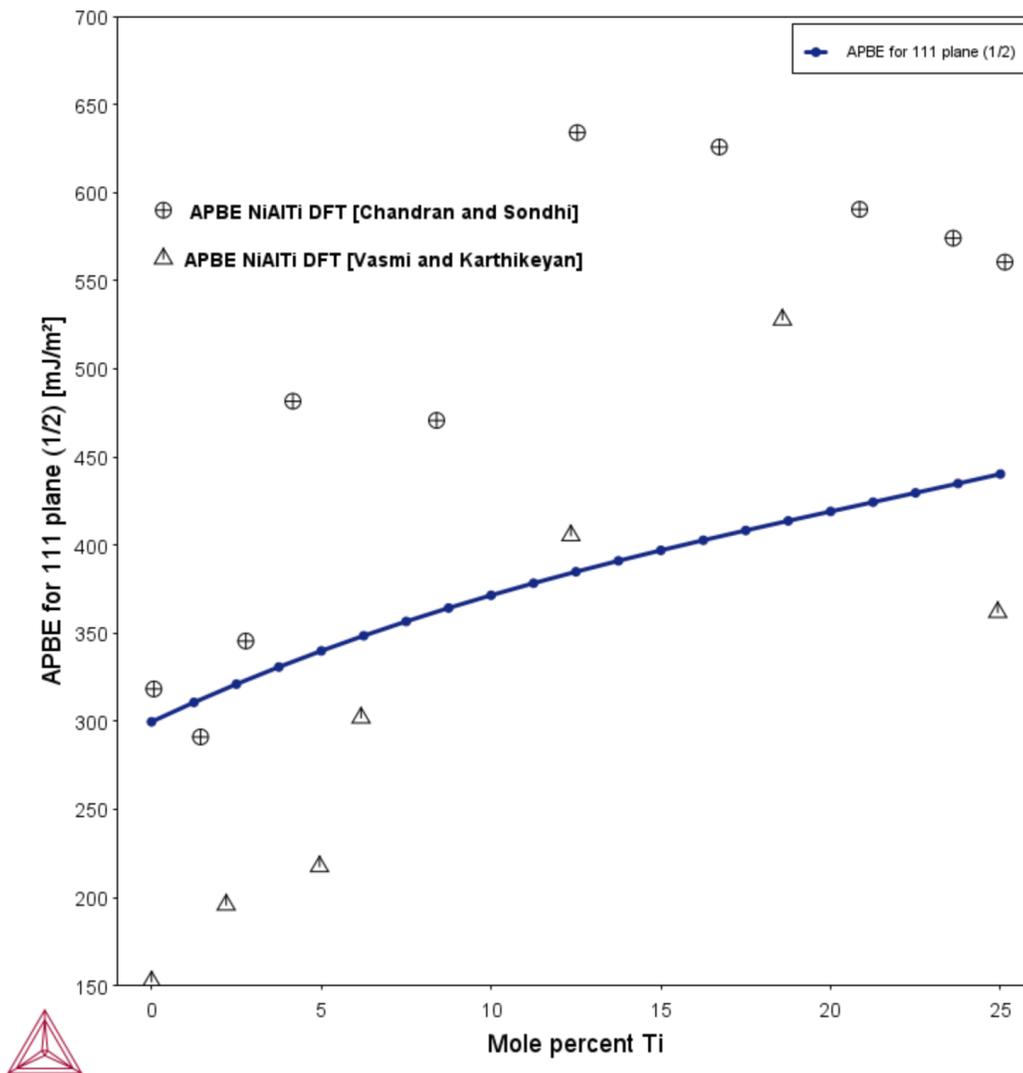


Figure 2: Composition of Ti (mole %) compared to antiphase boundary surface energy for 111 plane and experimental data from [2011Cha; 2012Vam].

## References

- [2011Cha] M. Chandran, S. K. Sondhi, First-principle calculation of APB energy in Ni-based binary and ternary alloys. *Model. Simul. Mater. Sci. Eng.* 19, 025008 (2011).
- [2012Vam] K. V. Vamsi, S. Karthikeyan, Effect of Off-Stoichiometry and Ternary Additions on Planar Fault Energies in Ni3Al, in *Superalloys 2012* (John Wiley & Sons, Inc., Hoboken, NJ, USA, 2012; pp. 521–530).

## PM\_Ni\_03: Critical Temperatures of Alloy 718

---

The example uses the **Property Model Calculator** with the **Solvus for Ordered Phase-Ni** Model.



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).

### Project File Information

- Folder: **Property Models** → **Nickel**
- File name: *PM\_Ni\_03\_Critical\_Temperatures\_Alloy\_718.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



[Property Model Calculator](#) and [About the Solvus for Ordered Phase Property Model](#)

In this example, the respective solvus temperatures of the Delta and Gamma double-prime phases are calculated.

For the Delta phase, the **Subset of phases** is set to **All phases** but the result varies very little with the selected subset (provided that the Delta phase is included). For the Gamma double-prime phase, which is a metastable phase, it is necessary to use the **Typical Ni-base superalloy** phase subset.

The resulting solvus temperatures are compared with experimental data from [2010Fra].

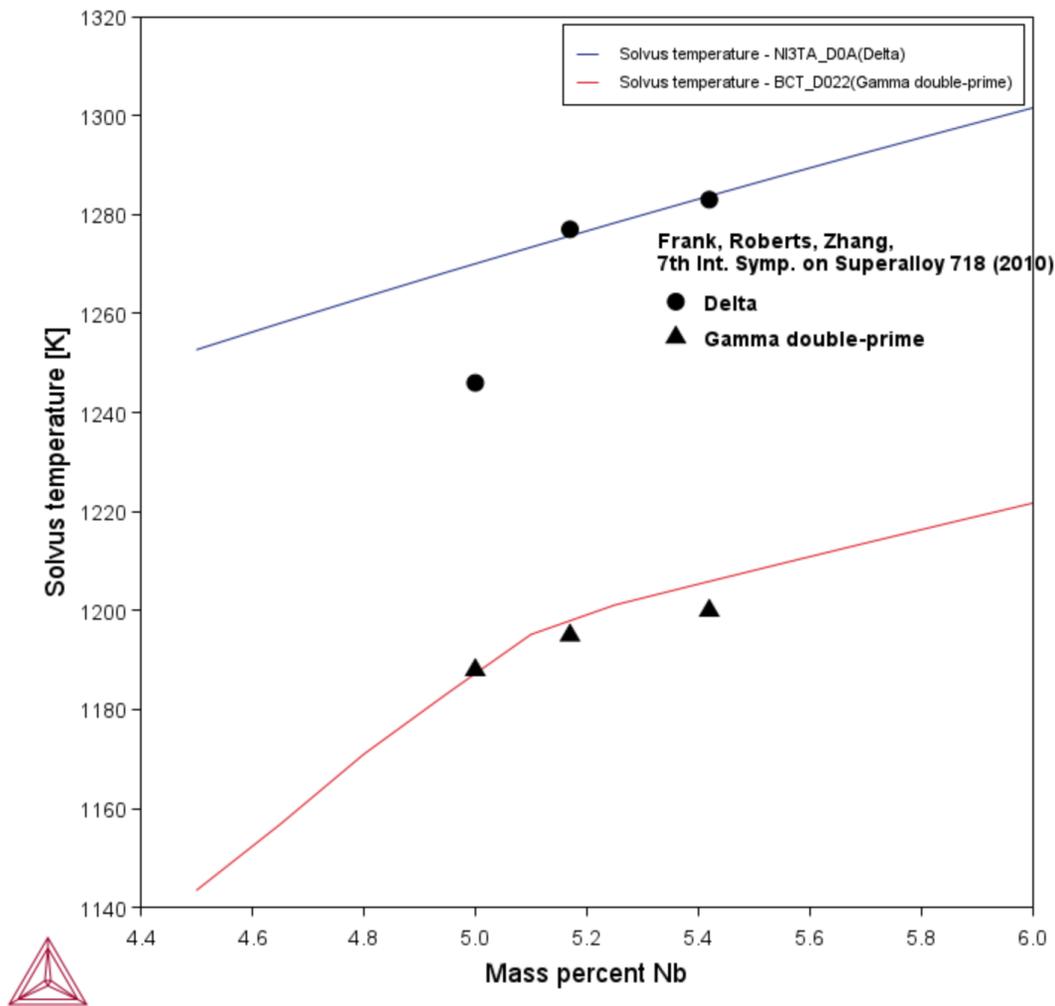


Figure 3: Solvus temperatures for NI3TA\_D0A (Delta) and BCT\_D022 (Gamma double-prime) compared to experimental data [2010Fra].

## Reference

[2010Fra] R. B. Frank, C. G. Roberts, J. Zhang, Effect of Nickel Content on Delta Solvus Temperature and Mechanical Properties of Alloy 718, in 7th International Symposium on Superalloy 718 and Derivatives (2010) (TMS (The Minerals, Metals & Materials Society), 2010, pp. 725–736.

## PM\_Ni\_04: Strain Age Cracking (SAC)

---

The example uses the **Property Model Calculator** with the **Strain-Age Cracking - Ni** Model.



To run calculations with the **Nickel Models** requires a valid maintenance license plus licenses for both the TCNI (version 11 and newer) and MOBNI (version 5 and newer) databases. For some Property Models, additional recommendations for the database version to use is indicated in its description. Also see our website to learn more about the [Nickel Model Library](#).

### Project File Information

- Folder: **Property Models** → **Nickel**
- File name: *PM\_Ni\_04\_Strain\_Age\_Cracking.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



This example uses the TCNI version 12 database. It can also be run with the TCNI version 11 database.



For additional information, see [About the Strain-Age Cracking Property Model](#).

Welding or additive manufacturing (AM) of Ni-base alloys has so far been strongly limited by various cracking issues during processing or post-processing. Strain-age cracking (SAC) occurs when the combination of thermal stresses and coherency stresses between the  $\gamma$  matrix and  $\gamma'$  precipitates exceed the local strength of the alloy structure. Thermal residual stresses originate from the initial processing (welding or AM), while coherency stresses form during post-processing heat treatments, when  $\gamma'$  re-precipitates in the matrix. Coherency stresses may be tensile or compressive depending on the sign of the lattice misfit.

Furthermore, coherency stresses scale with the fraction of  $\gamma'$ , making conventional high- $\gamma'$  alloys virtually impossible to weld or shape via AM. At the same time, high- $\gamma'$  fraction alloys offer attractive properties, such as high strength at high service temperatures. In order for such alloys to be processable, it is necessary to reduce the cracking tendency.

Thermal stresses are very difficult to alleviate, as this requires heating, which in turn triggers re-precipitation of  $\gamma'$ . Therefore, an alternative strategy may be to target the coherency stresses, which are determined by the respective lattice parameters of the matrix and precipitate. Such a strategy was adopted by Zhou et al. [2020Zho], where they optimized the composition of the Ni-base alloy to fulfill the above criteria, i.e., high fraction of  $\gamma'$  and no strain-age cracking tendency.

In this example, the experimental alloys from [2020Zho] are analyzed based on the **Strain-Age Cracking - Ni** Property Model to compare the tendency for cracking using an SAC risk factor.

Performing the calculation for the two experimental alloys, along with the CM247LC benchmark alloy and using the temperature interval setting produces a set of relevant output results as shown in a table (Figure 4) and **Parallel coordinates** plot (Figure 5). CM247LC is considered to be very difficult to weld, which is evidenced by the very high value of the SAC risk factor. Comparatively, the two experimental alloys, ExpAM and ExpAMmod, show values very close to zero. Low or negative values of the SAC risk factor can be interpreted as low or no risk for strain-age cracking.

Conventionally, the *weldability* of a given Ni-base alloy has been assessed based on the Ti and Al contents, which correlate with the fraction of  $\gamma'$ . However, such an approach neglects the fact that it is the coherency stresses that cause cracking, and these are also dependent on the lattice misfit. By tailoring an alloy to have low misfit, in relation to the  $\gamma'$  fraction and in the relevant temperature range, it is possible to design a high- $\gamma'$  fraction alloy with very low risk for SAC, as shown in [2020Zho]. The SAC model used here captures all of these effects and can be used as a design tool for the purpose of eliminating this cracking tendency.

Visualizations				
Plot Renderer 1	Table Renderer 1			
SAC risk factor	Temperature [K]	Mass percent Ni	Mass percent Al	Mass percent C
0.45355	1000.00000	64.69498	4.00000	5.20000
0.84060	1000.00000	65.39995	4.10000	5.00000
345.11631	1000.00000	61.69999	5.60000	9.20000

Figure 4: There are a two ways to view the data in table format. In this example, there is a Table Renderer node added to the Project window that lists the data points in the Visualizations window.



You can also access the same information via the Plot Renderer Configuration window by clicking the **Table View** button followed by clicking **Perform**. This converts the **Parallel coordinates** plot type to the same table of information.

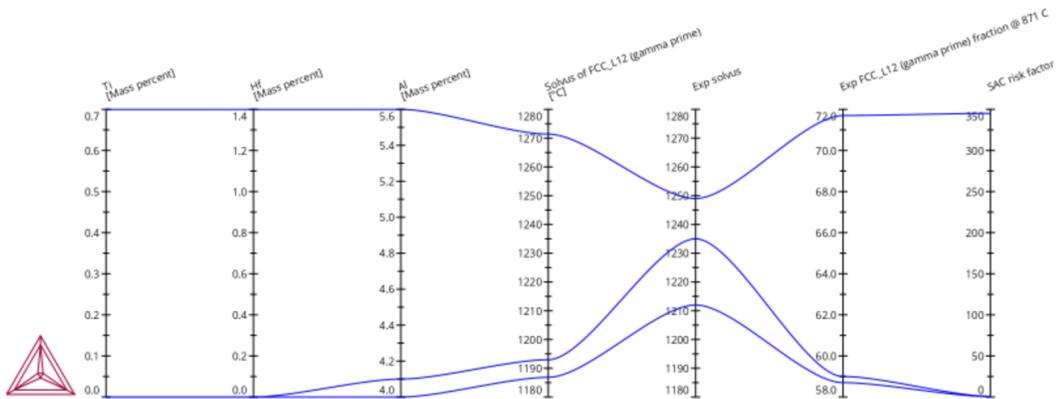


Figure 5: Parallel plot of the resulting SAC risk factor along with experimental and calculated solvus of  $\gamma'$  for the three alloys compared in [2020Zho].



[Plot Type: Parallel Coordinates](#)

## Reference

[2020Zho] N. Zhou, A. D. Dicus, S. A. J. Forsik, T. Wang, G. A. Colombo, M. E. Epler, Development of a New Alumina-Forming Crack-Resistant High- $\gamma'$  Fraction Ni-Base Superalloy for Additive Manufacturing, in *Superalloys 2020*, pp. 1046–1054.

---

## Process Metallurgy Module Examples Collection

---



The Process Metallurgy Module requires both a valid Maintenance and Support Subscription (M&SS) and a license for the TCS Metal Oxide Solutions Database (TCOX8 or newer).



All users can test the Process Metallurgy Module with the included OXDEMO database, which is limited to these elements: Al, C, Ca, Fe, O, S, and Si. For more information about this and other products [visit our website](#).



[The Process Metallurgy Module](#)



[About the Examples](#)

### In this section:

---

PMET_01: Basic Oxygen Furnace (BOF) .....	146
PMET_02: Desulphurization in a Ladle Furnace (LF) .....	148
PMET_03: Argon Oxygen Decarburization (AOD) .....	150
PMET_04: Basic Oxygen Furnace (BOF) Kinetics .....	157
PMET_05: Lab Scale Ladle Furnace (LF) Kinetics .....	158
PMET_06: Ladle Furnace (LF) Kinetics .....	164
PMET_07: Vacuum Oxygen Decarburization Kinetics .....	167
PMET_08: Steel Deoxidation on Tapping .....	168

---

## PMET\_01: Basic Oxygen Furnace (BOF)

---

The example uses the **Process Metallurgy Calculator** to demonstrate a simplified steelmaking process in a Basic Oxygen Furnace (BOF).

This application example in its various forms is [showcased on our website](#). There are two application examples (one equilibrium, one kinetic) showing how the Process Metallurgy Module can be used to calculate the BOF process.

This basic example uses equilibrium calculations to gain a general understanding of your BOF process and help you determine optimal operation conditions and predict and optimize costs of raw materials and recycling. The other set of examples gives detailed instructions on how to simulate the kinetics of the BOF process using the kinetic process simulation (see [PMET\\_04: Basic Oxygen Furnace \(BOF\) Kinetics](#)).



One way to learn how to work with the Process Metallurgy Calculator is to use the project files as templates to help you set up your own simulation. For example, you can examine the settings available on the **Configuration** window and that are described in [Process Simulation: Conditions Tab](#) and the [Options Tab](#).

### Project File Information

Open the example project file(s) from Thermo-Calc **Help** → **Example Files** → **Process Metallurgy** → **PMET\_01\_Basic\_Oxygen\_Furnace.tcu**.



Choose a, b, or c versions of the example based on your license.

- **PMET\_01a\_Basic\_Oxygen\_Furnace** is included in a regular Thermo-Calc installation as well as the free Educational version of Thermo-Calc. This is a highly simplified example and uses the OXDEMO database, and only considers the elements Fe, C, and O.
- **PMET\_01b\_Basic\_Oxygen\_Furnace** is also a simplified calculation but uses the OXDEMO database. It requires a full license of Thermo-Calc 2021b and newer.
- **PMET\_01c\_Basic\_Oxygen\_Furnace** requires the TCOX11 database and newer and a full license of Thermo-Calc 2021b and newer.



The resulting plots and details related to setting up this example are available either via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

## PMET\_02: Desulphurization in a Ladle Furnace (LF)

---

The example, which is also [showcased on our website](#), uses the **Process Metallurgy Calculator** to demonstrate the use of a Ladle Furnace (LF) with the Process Metallurgy Module.

The ladle furnace fulfils many purposes in the steelmaking process. Desulphurization, which we will focus on in this example, is merely one of them.

Desulphurization is usually performed by transferring S that is dissolved in the liquid metal to a CaO-rich slag phase. For this process to be successful, two conditions need to be fulfilled:

1. The slag must be fully liquid (liquid fraction > 0.9). This is required for kinetic reasons. The slag phase must be fluid so that it can emulsify with the liquid steel and form a large surface area where the reaction between steel and slag can take place.
2. The slag must take up a large amount of S from the liquid steel (have a “high sulphur capacity”) so that a significant amount of sulphur will move from the liquid steel to the slag phase.

It is difficult to decrease the S content in the liquid steel during steelmaking in a basic oxygen converter (BOF) or an electric arc furnace (EAF) because conditions are predominantly very oxidizing. Desulphurization is therefore usually performed in a subsequent step, during steel refining in a ladle furnace or a vacuum degasser after the steel has been fully killed by a suitable deoxidizing agent and the oxygen activity is low.

The following example explores these two conditions for an equilibrium between liquid steel and slag in a ladle furnace, trying to find good slag compositions.



One way to learn how to work with the Process Metallurgy Calculator is to use the project files as templates to help you set up your own simulation. For example, you can examine the settings available on the **Configuration** window and that are described in [Process Simulation: Conditions Tab](#) and the [Options Tab](#).

### **Project Files**

Open the example project files from Thermo-Calc **Help** → **Example Files** → **Process Metallurgy** → **PMET\_02a\_Ladle\_Furnace.tcu** or **PMET\_02b\_Desulphurization\_in\_Ladle\_Furnace.tcu**.



Choose a or b versions of the example based on your license

- **PMET\_02a\_Ladle\_Furnace.tcu** can be run with OXDEMO database but requires the full license for Thermo-Calc 2021b and newer.
- **PMET\_02b\_Desulphurization\_in\_Ladle\_Furnace.tcu** requires the TCOX11 database and a full license for Thermo-Calc 2021b and newer.



The resulting plots and details related to setting up this example are available either via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

## PMET\_03: Argon Oxygen Decarburization (AOD)

---

The Argon Oxygen Decarburization (AOD) process is a converter process applied in stainless steel making. In this process the oxidation of chromium in the steel melt needs to be prevented while carbon is oxidized. This is achieved by reducing the oxygen partial pressure by having a high Ar-content in the blowing gas. It is a highly exothermic process that requires adiabatic modeling. In this example the conversion of carbon-rich steel scrap is modeled together with additions of Ferronickel and Ferrochrome to a stainless steel. The example uses the **Process Metallurgy Calculator**.



Visit the website [Application Examples → Process Metallurgy](#) page for more background information as well as more in depth analyses of this and other examples. Also visit the [Process Metallurgy Module](#) page to access resources such as training videos, presentations, publications, webinars, and much more.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### **Project File Information**

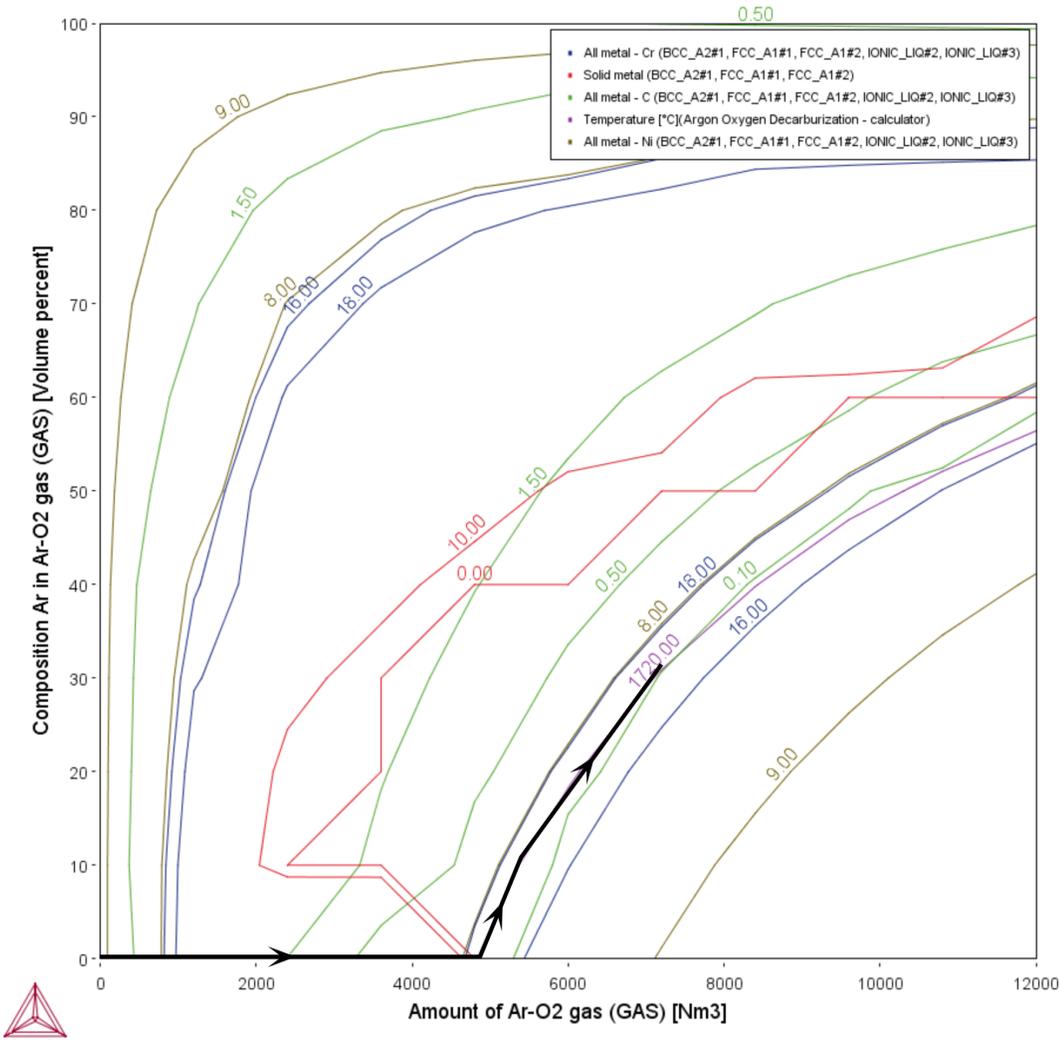
Open the example project file from Thermo-Calc **Help** → **Example Files** → **Process Metallurgy** → **PMET\_03\_Argon\_Oxygen\_Decarburization.tcu**.



This example requires a license to run the simulation. It works with TCS Metal Oxide Solutions Database (TCOX) versions 8 and newer.

## **Background Description**

The AOD-process is characterized by an initially high O<sub>2</sub>-content of the blowing gas, which is gradually replaced by Ar. The maximum temperature of the process is typically around 1720 °C. A grid plot of an adiabatic calculation is well suited to model the required simultaneous change of amount of gas and its composition during this process. The black line in the plot above indicates a possible process control path that finally leads to suitable alloy composition close to a X5CrNi 18-10 stainless steel.



In principle, this plot could be used to control the blowing gas flow in a plant process. However, note that the simulation only considers a global gas phase, i.e. additional gas is added but never removed. This means that the given gas phase composition is an average one and not identical to the composition of the gas flowing in at each point in time. Additionally there is a certain impact on the resulting equilibria from the fact that all gas remains in the system and is not removed.

## Slag Basicity

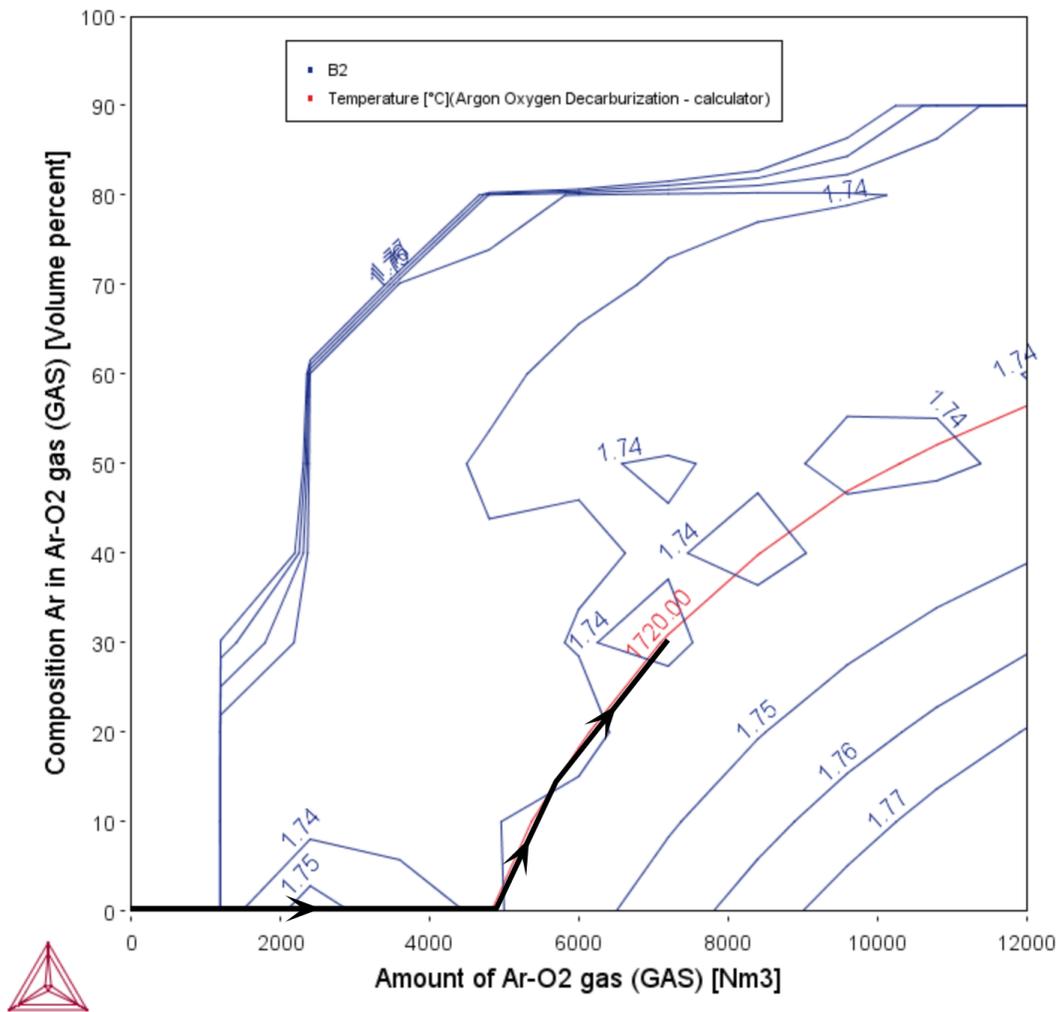


Figure 1: The Slag Basicity plot shows the basicity of the slag (which is a measure for the sulphur capacity of the slag). The process achieves a final basicity of 1.74 which is typical for an AOD-process. The black line in the plot indicates a possible process control path that finally leads to suitable alloy composition close to a X5CrNi 18-10 stainless steel.

## S Content in Steel

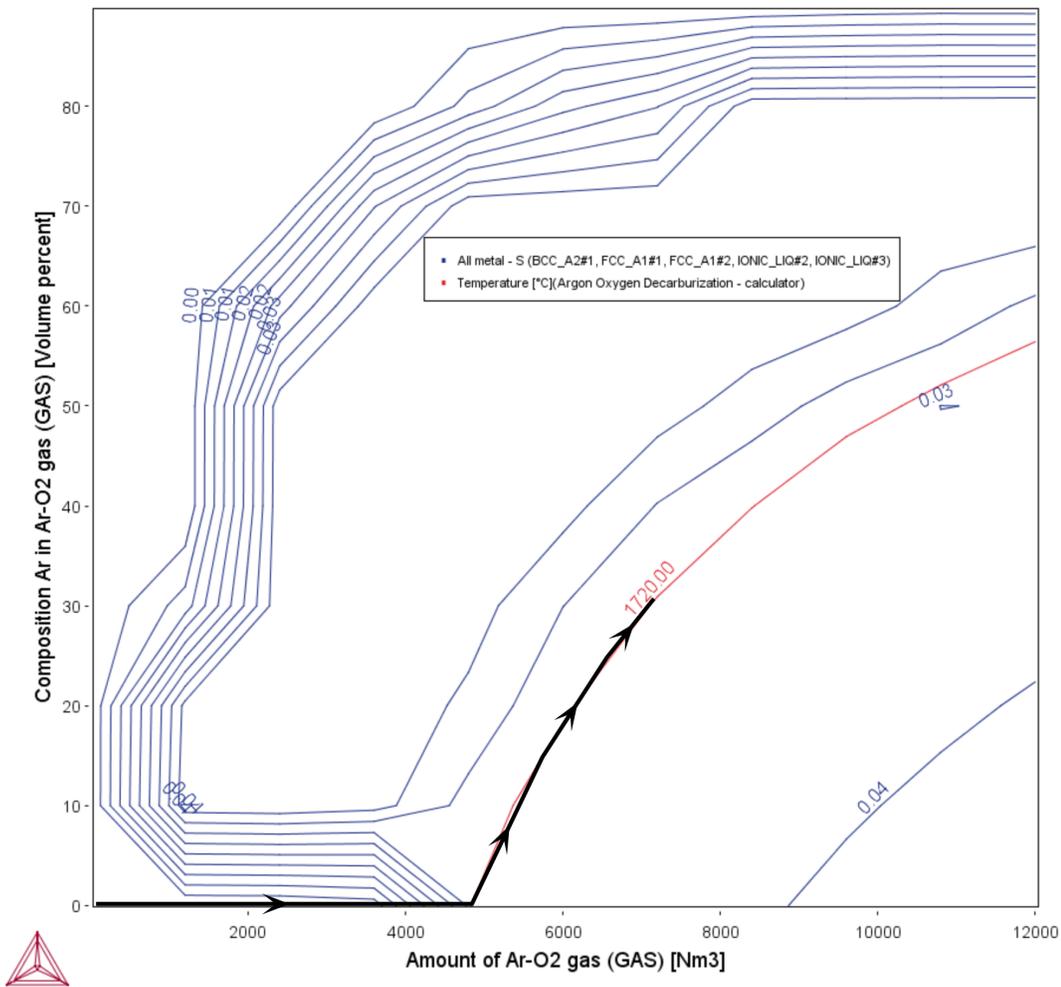


Figure 2: In the S Content in Steel plot, the sulphur content in the steel melt is slightly reduced during the process from 0.056% in the initial steel scrap to finally ca. 0.035%. The black line in the plot indicates a possible process control path that finally leads to suitable alloy composition close to a X5CrNi 18-10 stainless steel.



## Amount of Slag

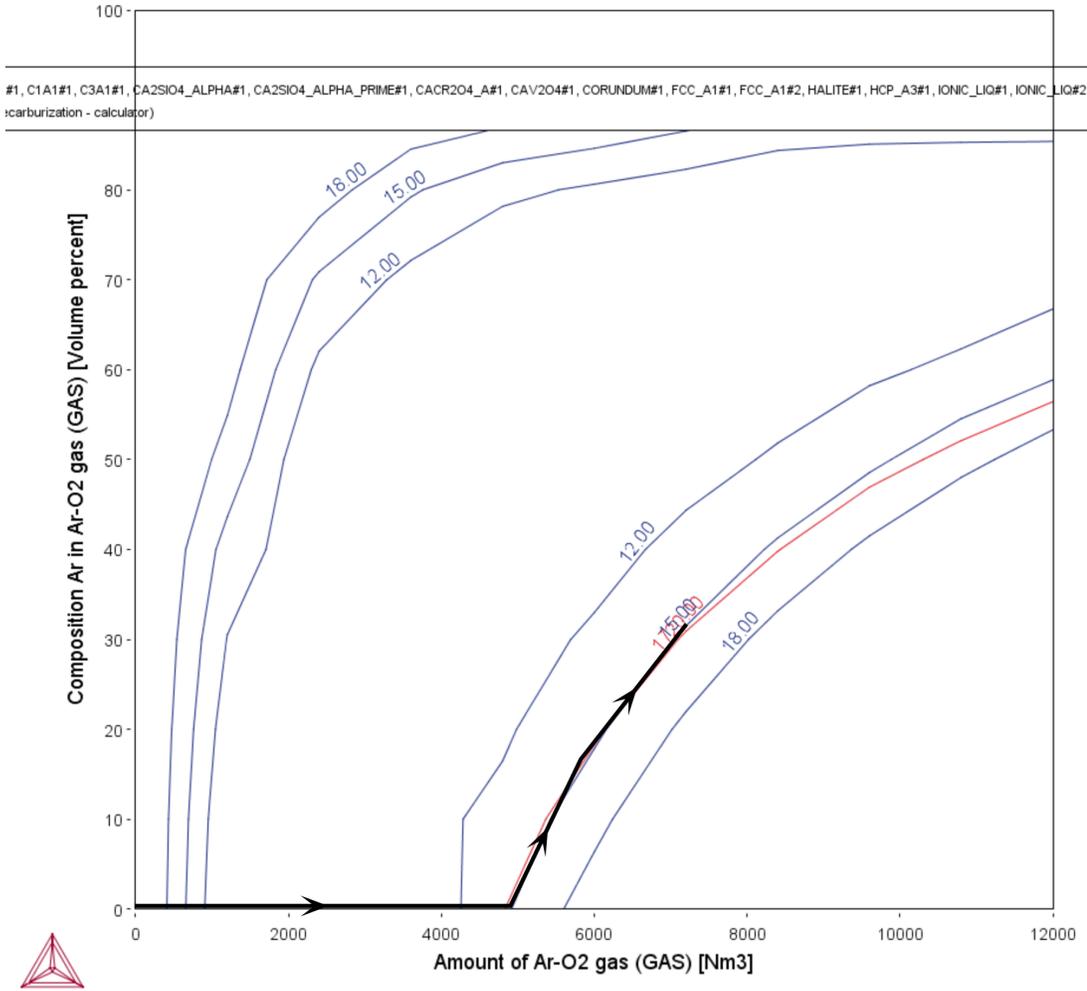


Figure 4: In the Amount of Slag plot, you can see that the process generates about 15 tons of slag in the total. The black line in the plot indicates a possible process control path that finally leads to suitable alloy composition close to a X5CrNi 18-10 stainless steel.

## Liquid Slag Fraction

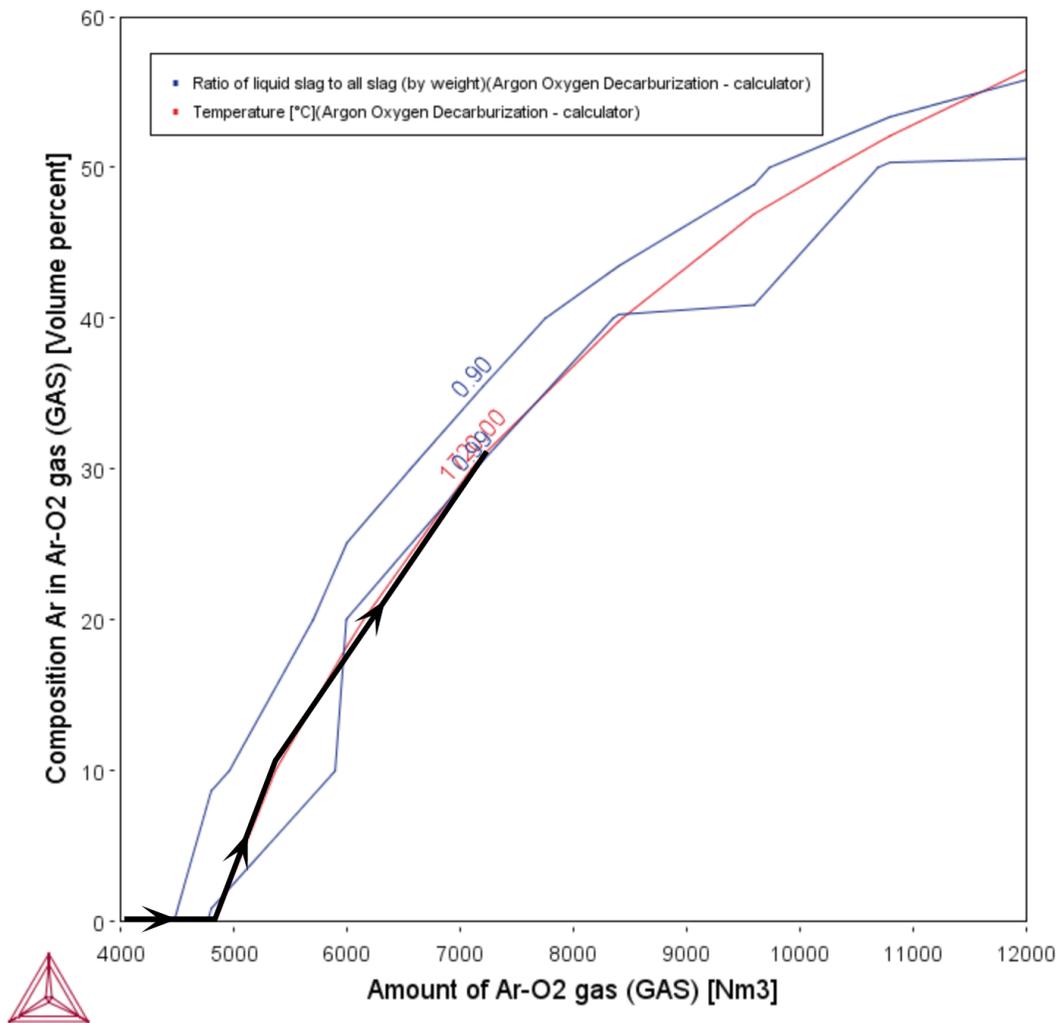


Figure 5: In the Liquid Slag Fraction plot, you can see that towards the end of the process the slag is calculated to be mostly liquid. The black line in the plot indicates a possible process control path that finally leads to suitable alloy composition close to a X5CrNi 18-10 stainless steel.

## PMET\_04: Basic Oxygen Furnace (BOF) Kinetics

---

The example uses the **Process Metallurgy Calculator** to demonstrate the steelmaking process in a Basic Oxygen Furnace (BOF).

This application example in its various forms is [showcased on our website](#). There are two application examples (one equilibrium, one kinetic) showing how the Process Metallurgy Module can be used to calculate the BOF process.

This example simulates the kinetics of the BOF process, whereas the other only considers equilibrium. For information about the equilibrium examples, see [PMET\\_01: Basic Oxygen Furnace \(BOF\)](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

Open the example project file(s) from Thermo-Calc **Help** → **Example Files** → **Process Metallurgy** →.



Choose a, b, or c versions of the example based on your license.

- **PMET\_04a\_Basic\_Oxygen\_Furnace\_Kinetics** is included in a regular Thermo-Calc installation as well as the free Educational version of Thermo-Calc. This only considers the elements Fe, C, and O.
- **PMET\_04b\_Basic\_Oxygen\_Furnace\_Kinetics** needs a full license of Thermo-Calc 2021b or newer, but uses the free OXDEMO database. It considers the elements Fe, C, O, Ca, Al, Si, and S.
- **PMET\_04c\_Basic\_Oxygen\_Furnace\_Kinetics** requires both a full license of Thermo-Calc 2021b or newer and a license for the TCOX11 database and newer. This simulation corresponds pretty much to a real BOF process with the elements Fe, Mn, C, O, Ca, Al, Mg, Si, S, and P considered.



The resulting plots and details related to setting up this example are available either via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

## PMET\_05: Lab Scale Ladle Furnace (LF) Kinetics

---

Using the **Process Metallurgy Calculator** and the **Process simulation** branch, this example is based on the publication by Piva et al [2017Piv] where a lab scale sample of pure iron is first deoxidized with Si-Mn and then a synthetic top slag is added to the steel. Generally, by using the project file and the information below, you can get an idea about how to set up a ladle furnace simulation using the Process Metallurgy Module.

 [Setting Up a Process Metallurgy Simulation](#)

 [Defining the Process Simulation](#)



Visit the website [Application Examples → Process Metallurgy](#) page for more background information as well as more in depth analyses of this and other examples. Also visit the [Process Metallurgy Module](#) page to access resources such as training videos, presentations, publications, webinars, and much more.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### **Project File Information**

To open this project file, from the Thermo-Calc menu, go to **Help** → **Example Files** → **Process Metallurgy** → **PMET\_05\_Lab\_Scale\_Ladle\_Furnace\_Kinetics.tcu**.



This example is best run with the TCS Metal Oxide Solutions Database (TCOX) version 13 or newer and a full license of Thermo-Calc 2024a or newer. Running the example with earlier versions of the database is possible but the results are not consistent with the referenced paper.

## About the Plot Results

Many different aspects of the reactions taking place in the LF can be plotted and analyzed. After you run the project file and obtain the plots, you can experiment by adjusting the settings on each Plot Renderer to see what happens in each case. There is more analysis about this and other examples available at the links to our website.

 For an example of a more realistic industry scale, see [PMET\\_06: Ladle Furnace \(LF\) Kinetics](#).

## Setting Up the LF Process Simulation

In the paper by Piva et al. [2017], it is experimentally investigated how the steel and inclusion composition changes in function of time as the Si-Mn killed steel reacts with the slag. This experiment can serve as a model for the production of SiMn killed steel with subsequent top-slag deoxidation.

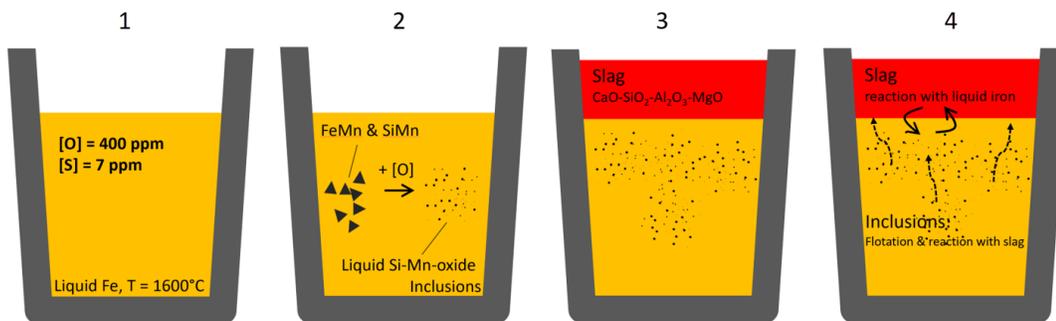


Figure 6: Diagram of the experiment performed by Piva et al. [2017]. The sequence as described in the text is: (1) liquid Fe with dissolved oxygen; (2) FeMn and SiMn added to deoxidize Fe; (3) top slag added after 360s; and (4) gradual reaction between steel, slag and inclusions.

According to the publication the following reaction sequence is expected as shown in the diagram:

1. In the initial state the steel contains 400 ppm of dissolved oxygen.
2. The deoxidizing agents are added. They dissolve and react with the oxygen in the steel forming oxide inclusions, thereby reducing the amount of dissolved oxygen ("killing" the steel). The kinetics are fast, and it can be assumed that the reaction proceeds to thermodynamic equilibrium.
3. On adding the slag (360 s after step 2, killing the steel) the liquid steel starts reacting with the slag.
4. According to the authors the most important reaction that takes place is the dissolution of Al out of the slag and its transferal into the liquid steel, where it reacts with the inclusions and changes their chemistry. Flotation of the inclusions is assumed to be negligible, due to the small size of the crucible and lack of stirring.



In this example only the last reaction is simulated after adding the slag. The kinetic parameters, compositions of all materials and process schedule are all taken from the publication.

## Results and Experimental Analysis

The plots below compare the Al content in the liquid steel with the experimentally determined amount. The bottom plot shows how the  $\text{Al}_2\text{O}_3$  from the slag phase is gradually reduced to metallic Al that dissolves in the liquid steel.



In this example, only part of the Al in the liquid steel is dissolved as metallic Al. With the Process Metallurgy Module, it is possible to plot both dissolved Al and also total Al.

In the steel industry this fraction of Al in the liquid steel is often termed *dissolved Al*. The rest of the Al reacts with the oxygen in the liquid steel and forms oxide inclusions. The amount of dissolved Al plus the Al bound up in oxide inclusions is often termed *total Al*. The difference between total Al minus dissolved Al is an important measure for the steel cleanliness.

Total Al is what was measured by Piva et al. [2017] and their experimental data are compared to the calculated values below.

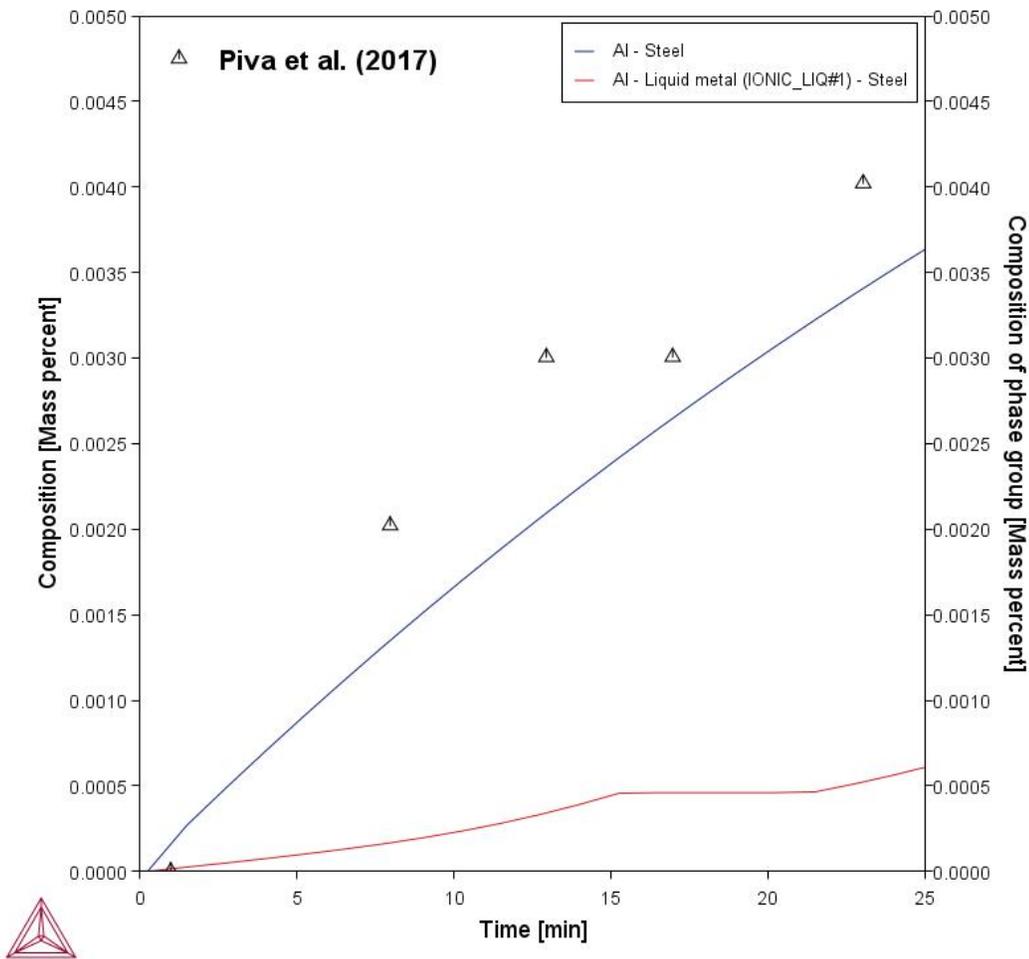


Figure 7: Total Al in the steel compared with experimental data (blue/top curve). Additionally the amount of dissolved Al is plotted (red/bottom curve), this is done by selecting “Composition of phase group” and “Liquid metal”. Note that the red/bottom curve is not available with the example.

Immediately after killing the steel with SiMn and FeMn, the liquid steel contains liquid Si-Mn-oxide inclusions only. As the liquid steel picks up Al from the slag phase in function of time, the inclusions get richer and richer in Al. This experimentally verified change in inclusion chemistry is well reproduced by this simulation.



In the experiment, only the average composition of all inclusions could be measured; the type of inclusion (liquid / solid) was not determined.

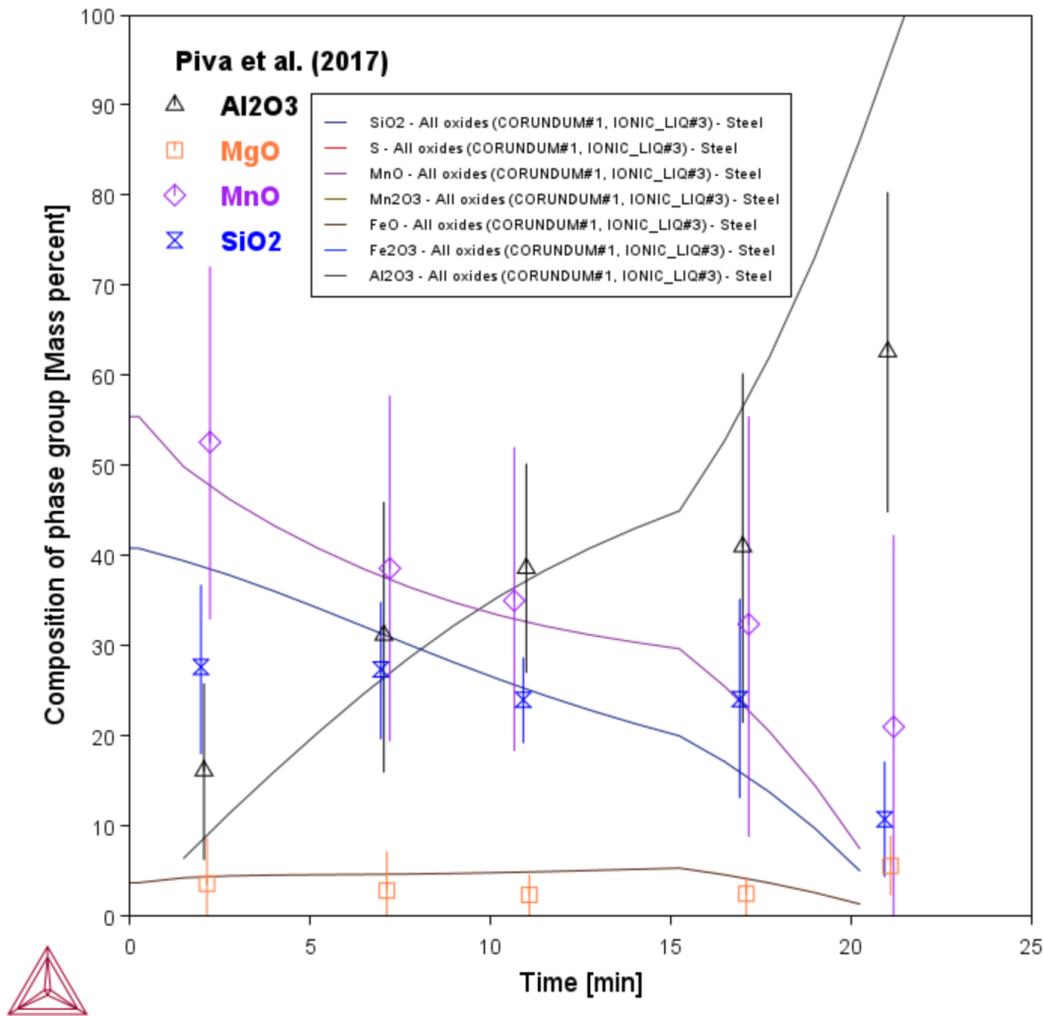


Figure 8: Calculated composition of non-metallic inclusions in function of processing time compared to experimental data [2017Piv].

With this simulation it is also possible to calculate the amount and type of inclusions present in the liquid steel. At the beginning of the process all the inclusions are liquid oxides, that gradually get richer in Al<sub>2</sub>O<sub>3</sub> as the steel picks up Al from the slag. After 20 minutes processing time, the first solid corundum (Al<sub>2</sub>O<sub>3</sub>) inclusions start appearing. After 25 minutes almost no liquid inclusions remain.

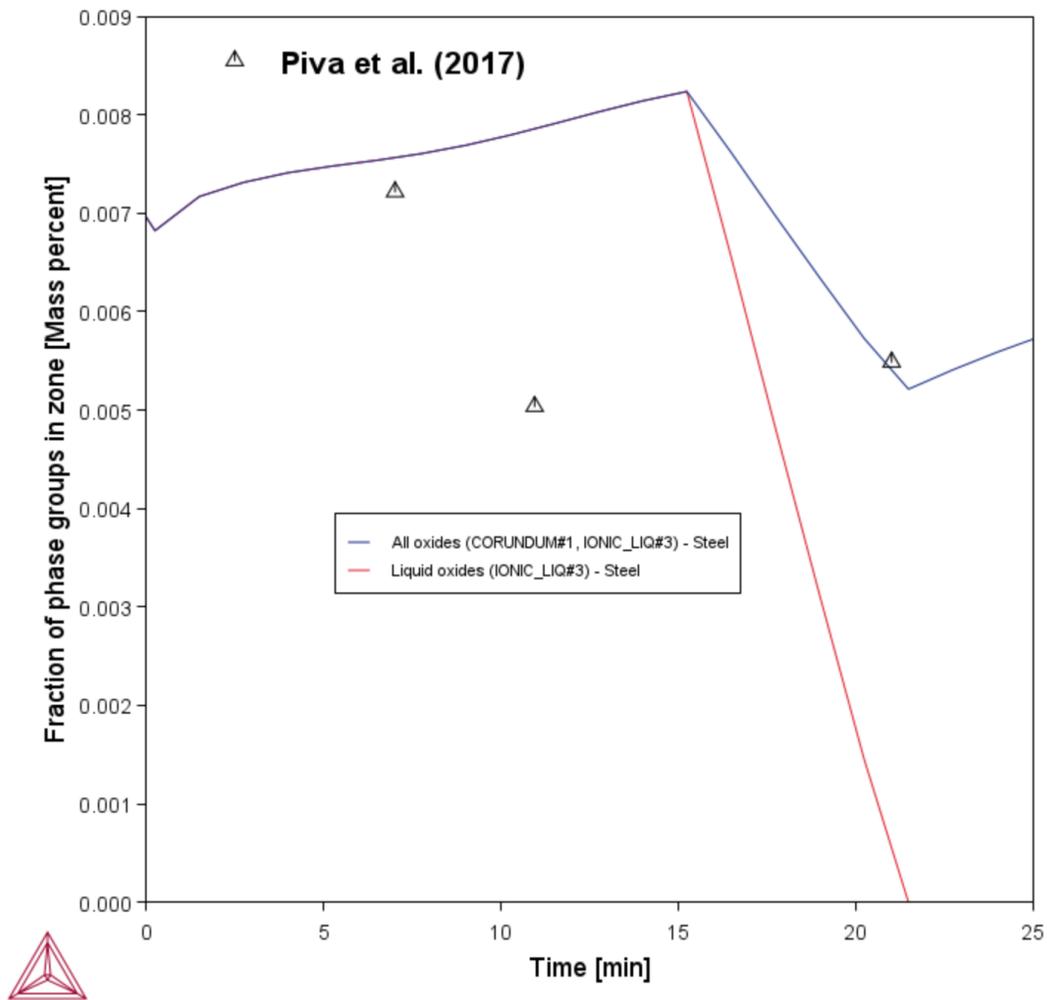


Figure 9: Amount of inclusions in the liquid steel in function of processing time. The total amount of inclusions (red line) and liquid oxide inclusions (blue line) are plotted. Up to about 20 min all inclusions are liquid oxide type, then they are replaced by solid corundum ( $Al_2O_3$ ) inclusions. Experimental data from [2017Piv].

## Reference

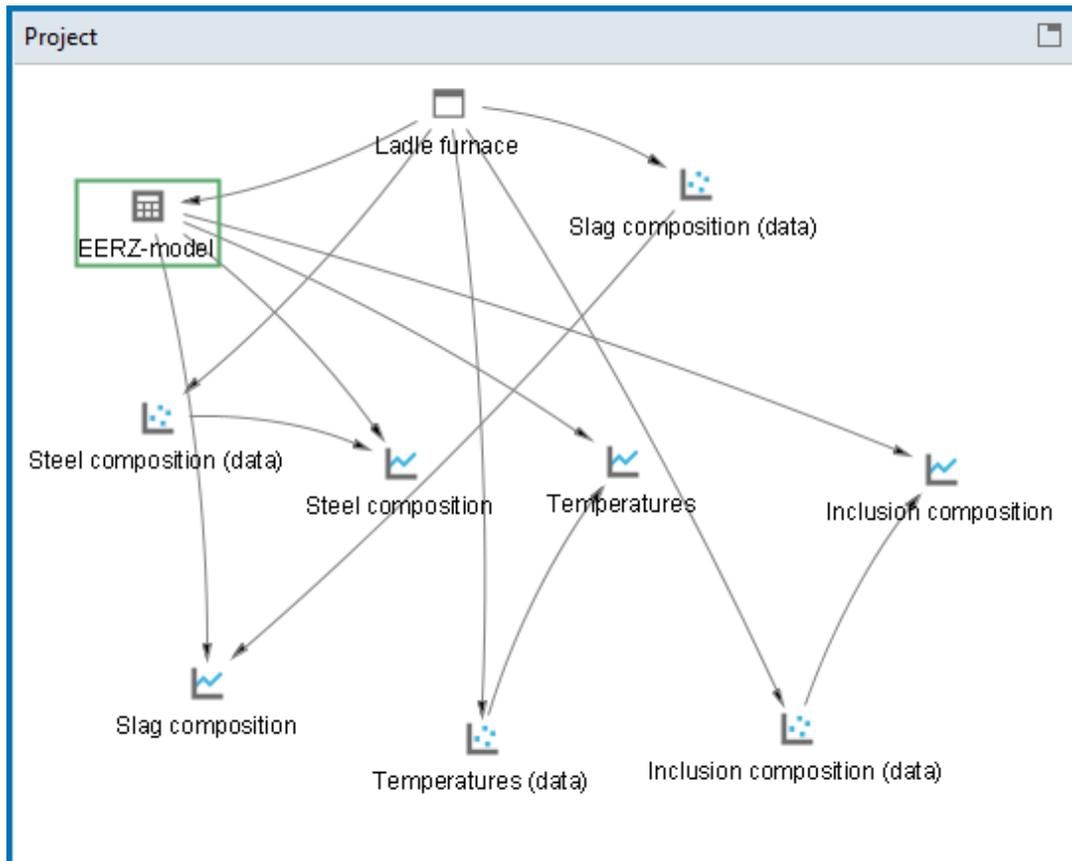
[2017Piv] Piva, S. P. T., Kumar, D. & Pistorius, P. C. "Modeling Manganese Silicate Inclusion Composition Changes during Ladle Treatment Using FactSage Macros," Metall. Mater. Trans. B 48, 37–45 (2017).

## PMET\_06: Ladle Furnace (LF) Kinetics

This in-depth example, which is also [showcased on our website](#), uses the **Process Metallurgy Calculator** to show how to set up a full kinetic simulation of steel refining in a ladle furnace using the Process Metallurgy Module in Thermo-Calc. The Process Metallurgy Module uses the Effective Equilibrium Reaction Zone model (EERZ) to simulate the kinetics of the process. In this example, a full kinetic simulation of the LF refining process is set up using the Process Metallurgy Module.



For a simpler process that describes a lab-scale ladle furnace process, see [PMET\\_05: Lab Scale Ladle Furnace \(LF\) Kinetics](#).



The nodes in the Project window for this example. The Process Metallurgy Calculator is renamed to EERZ model. If you click this node the Configuration window opens, which is where all the settings for the process simulation are located. This advanced example also uses three Experimental Data Reader nodes to read the experimental data that is included in the simulation results. After setting up the system and accessing the experimental data, you then add Plot Renderers to the calculator to generate the output in the Visualizations window.

## Project File Information

To open this project file, from the Thermo-Calc menu, go to **Help** → **Example Files** → **Process Metallurgy** → **PMET\_06\_Ladle\_Furnace\_Kinetics.tcu**.



This example requires the database TCOX10 or newer, a full license of Thermo-Calc 2020b or newer, plus a license for the Process Metallurgy Module.



The resulting plots and details related to setting up this example are available either via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

## Background Overview

After steelmaking, which is mostly performed in a basic oxygen furnace (BOF) or electric arc furnace (EAF), the steel is usually tapped into a ladle where certain additions are made (deoxidation agents, slag formers, certain alloying elements) and then transferred to the ladle furnace (LF). The LF fulfils many purposes in the steel refining process, the most important being:

- Temperature control / heating by an electrical arc.
- Mixing by Ar or N<sub>2</sub> bubbling through porous plugs in the bottom of the ladle to achieve homogeneous temperature and composition throughout the ladle.
- Removal of unwanted non-metallic phases / inclusions such as corundum (Al<sub>2</sub>O<sub>3</sub>), liquid oxide inclusions, spinel, on so on, by flotation, aided by Ar or N<sub>2</sub> bubbling.
- Modification / engineering of non-metallic inclusions so that they are not detrimental for the downstream processing and/or the final product.
- Removal of unwanted volatile elements such as Pb, Zn, Sn, and so on. Due to their high vapor pressure, these elements are enriched in the rising Ar or N<sub>2</sub> bubbles. After the gas escapes out of the ladle and cools, they condense, forming copious amounts of dust.
- Removal of unwanted elements such as sulphur by liquid steel / slag reactions.
- Lowering of the dissolved gas content. In the LF this is mainly achieved through chemical reactions. For direct removal of dissolved gas, vacuum degassing (VD) is usually required.
- Alloying and trimming of the steel to achieve the exact alloy composition required by the specification of the steel that is to be produced.

The reactions taking place in a LF are a complex interplay between equilibrium thermodynamics that define the direction of chemical reactions, and kinetics that define how fast the equilibrium state is approached.

---

## PMET\_07: Vacuum Oxygen Decarburization Kinetics

---

This application example, which is [showcased on our website](#) and uses the **Process Metallurgy Calculator**, shows how the Process Metallurgy Module in Thermo-Calc can be used to simulate the Vacuum Oxygen Decarburization (VOD) process for a stainless steel.

This example, based on data of a real VOD process published by Ding et al. [2000Din], shows how to set up the VOD process in Thermo-Calc's Process Metallurgy Module. Use of the following features in the Process Metallurgy Module are highlighted in this example:

- Change of pressure as a function of time during the process
- Change of reaction kinetics as a function of time during the process
- Selection of zone where degassing is allowed

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

Open this project file, from the Thermo-Calc menu, go to **Help** → **Example Files** → **Process Metallurgy** → **PMET\_07\_Vacuum\_Oxygen\_Decarburization\_Kinetics.tcu**.



This example requires a license for Thermo-Calc 2021a or newer, the database TCOX10 or newer, and a license for the Process Metallurgy Module.



The resulting plots and details related to setting up this example are available either via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

## Reference

[2000Din] R. Ding, B. Blanpain, P. T. Jones, P. Wollants, Modeling of the vacuum oxygen decarburization refining process. Metall. Mater. Trans. B. 31, 197–206 (2000).

## PMET\_08: Steel Deoxidation on Tapping

---

This application example, which is [showcased on our website](#) and uses the **Process Metallurgy Calculator**, shows a few of the ways the Process Metallurgy Module in Thermo-Calc can be used to investigate steel deoxidation (killing) on tapping.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

Open the project files, from the Thermo-Calc menu, go to **Help** → **Example Files** → **Process Metallurgy** → **PMET\_08\_Steel\_Deoxidation\_on\_Tapping.tcu**.



Choose a or b versions of the example based on your license.

- **PMET\_08a\_Steel\_Deoxidation\_on\_Tapping.tcu** is included in a regular Thermo-Calc installation as well as the free Educational version of Thermo-Calc. It is a simplified version that uses the OXDEMO database.
- **PMET\_08b\_Steel\_Deoxidation\_on\_Tapping.tcu** requires a full license of Thermo-Calc version 2021b or newer and database TCOX11 or newer.



The resulting plots and details related to setting up this example are available either via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

## About Deoxidation or “Killing” of Steel

Deoxidation of steel, also known as killing of steel, occurs toward the end of the steelmaking process, but before desulphurization. After primary steelmaking, for example in a basic oxygen furnace (BOF) or electric arc furnace (EAF), the liquid steel contains a large amount of dissolved oxygen [O] (400 to over 1000 ppm, see [PMET\\_04: Basic Oxygen Furnace \(BOF\) Kinetics](#)). This oxygen must be removed from the liquid steel for several reasons, including:

- Desulphurization is not effective with high oxygen content in the steel (see example on desulphurization)
- Continuous casting of un-killed steel is not possible and will result in “boiling” of the steel in the mould and massive porosity, as shown in the figure.

The two most widely applied methods of deoxidizing or killing the steel are adding Al (Al-killed steel) to the steel and adding Si (Si-killed steel) to the steel. These elements readily react with oxygen to form oxides that precipitate within the liquid steel. This does not reduce the total amount of oxygen in the liquid steel; it simply transforms the dissolved oxygen into an oxide precipitate. This oxide precipitate then needs to be removed from the steel by flotation during secondary metallurgy in the ladle furnace (LF).

In this example we look at how the Process Metallurgy Module can be used to investigate these two widely applied methods of deoxidizing or killing the steel: adding Al (Al-killed steel) or adding Si (Si-killed steel). This, however, can result in the formation of damaging inclusions that cause problems during further processing, so we go on to look at two simple and very common processes that are used to transform the solid  $\text{SiO}_2$  and  $\text{Al}_2\text{O}_3$  inclusions into liquid oxides so that they are less damaging.

---

## Diffusion Module (DICTRA) Examples Collection

---



These are the Graphical Mode examples. There are separate Console Mode examples also available.



Examples using up to three elements are available to all users. The other examples require a Diffusion Module (DICTRA) license to calculate and plot results.



[The Role of Diffusion in Materials: A Tutorial](#) is available for download on our website. It is intended for engineers interested in using the Diffusion Module (DICTRA), as well as students learning about the role of diffusion in materials. It is designed to be useful at many levels, from undergraduate studies to someone with a PhD and experience in a related field.

These examples use the **Diffusion Calculator**. All examples use demonstration database packages included with your installation no matter what licenses you have.



[About the Examples](#)

### In this section:

D_01: Homogenization of a Binary Fe-Ni Alloy .....	171
D_02: Ferrite(bcc)/Austenite(fcc) Transformation in a Binary Fe-C Alloy .....	174
D_03: Evolution of an Fe-Cr-Ni Diffusion Couple .....	178
D_04: Fe-C Moving Boundary: Austenite to Ferrite .....	181
D_05: $\gamma/\alpha/\gamma$ Diffusion Couple of Fe-Ni-Cr alloys .....	186
D_06: Diffusion Through a Tube Wall .....	195
D_07: Multiphase Carburization of an Alloy .....	200
D_08: Microsegregation During Solidification .....	205
D_09: Ni Post Weld Heat Treatment .....	208
D_10: Iron (Fe) Homogenization in Scheil .....	213

---

## D\_01: Homogenization of a Binary Fe-Ni Alloy

---

The single phase example simulates the diffusion of Fe and Ni at a temperature of 1400 K in a planar domain. At this temperature the material is fully austenitic, i.e. the only phase present is the so-called fcc (face centered cubic) phase. Initially, there is a linear variation in Ni going from 10 mass-% on the left-hand side to 50 mass-% on the right-hand side.



This example is included as a Diffusion Module (DICTRA) tutorial on our [website](#) and as part of the playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_01\_Diffusion\_Single\_Phase.tcu*



This example is part of the *Diffusion Module (DICTRA) Quick Start Guide* available to all users. A version of the example is also available for Console Mode.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.
- **Plot** or **Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

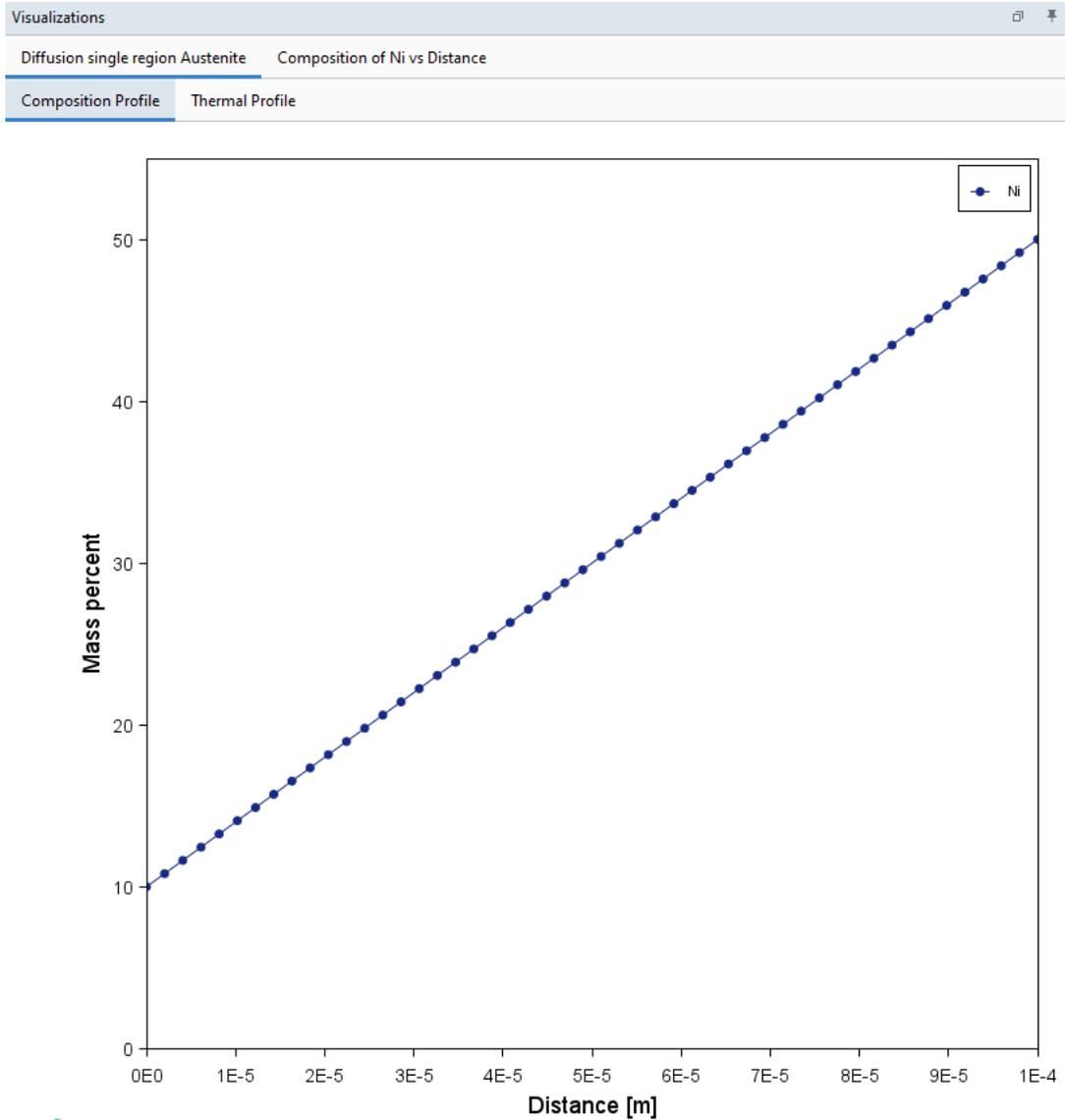


Figure 1: During set up of the calculation, you can preview the Composition Profile (shown) or Thermal Profile. Click the tab(s) in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window. For an Isothermal Thermal Profile this shows the constant temperature as entered.

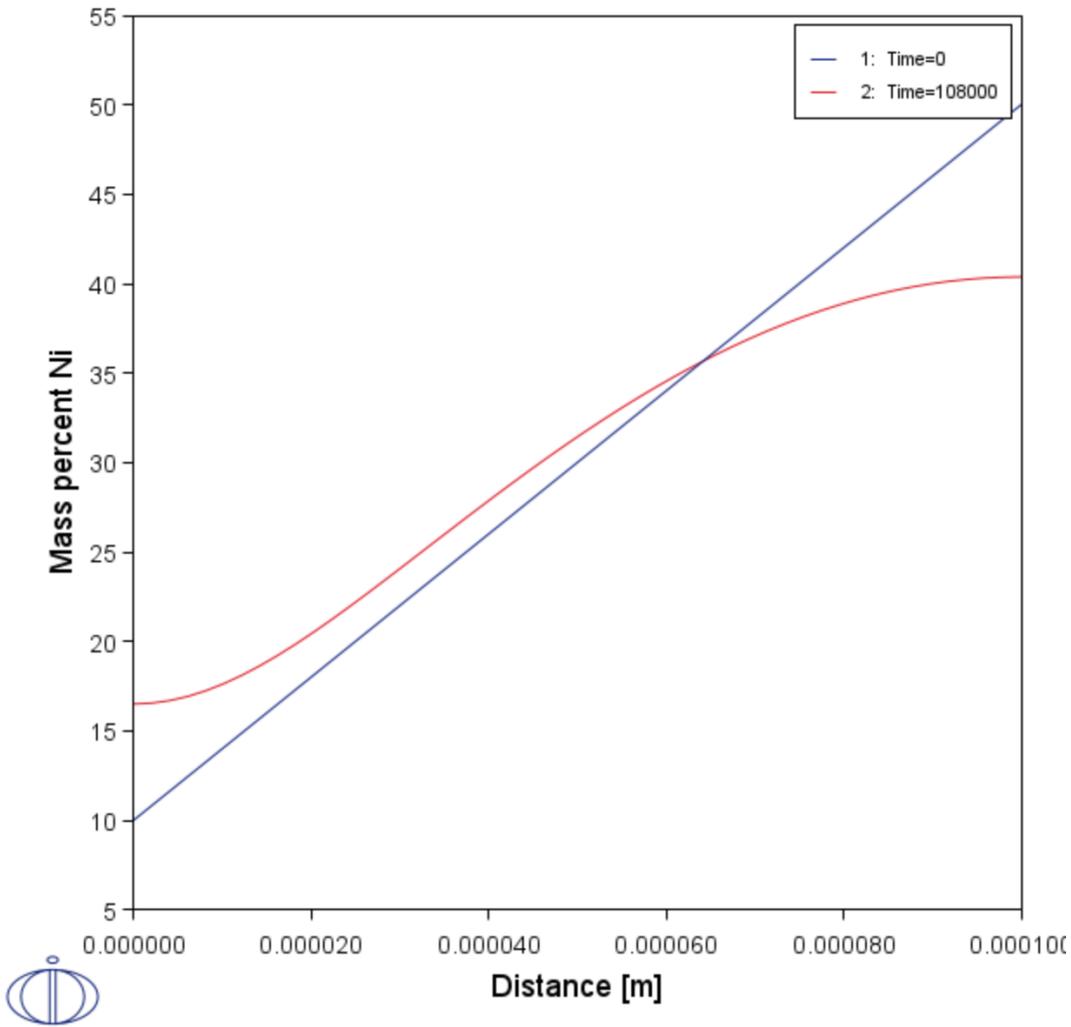


Figure 2: After performing the calculation, you can view the final Plot Renderer result on the Composition of Ni vs Distance tab, which shows the initial and final Ni profile. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

## D\_02: Ferrite(bcc)/Austenite(fcc) Transformation in a Binary Fe-C Alloy

---

The moving phase boundary example simulates the growth of ferrite (bcc) into austenite (fcc). The austenite is assumed to be initially homogeneous with the composition Fe-0.15 mass-% C. The transformation temperature is 1050 K. The initial thickness of the austenite is 2 mm and an initially very thin ferrite (1 nm) is also present at the start of the simulation.



This example is included as a Diffusion Module (DICTRA) tutorial on our [website](#) and as part of the playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_02\_Diffusion\_Moving\_Boundary.tcu*



This example is part of the *Diffusion Module (DICTRA) Quick Start Guide* available to all users. A version of the example is also available for Console Mode.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.
- **Plot** or **Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

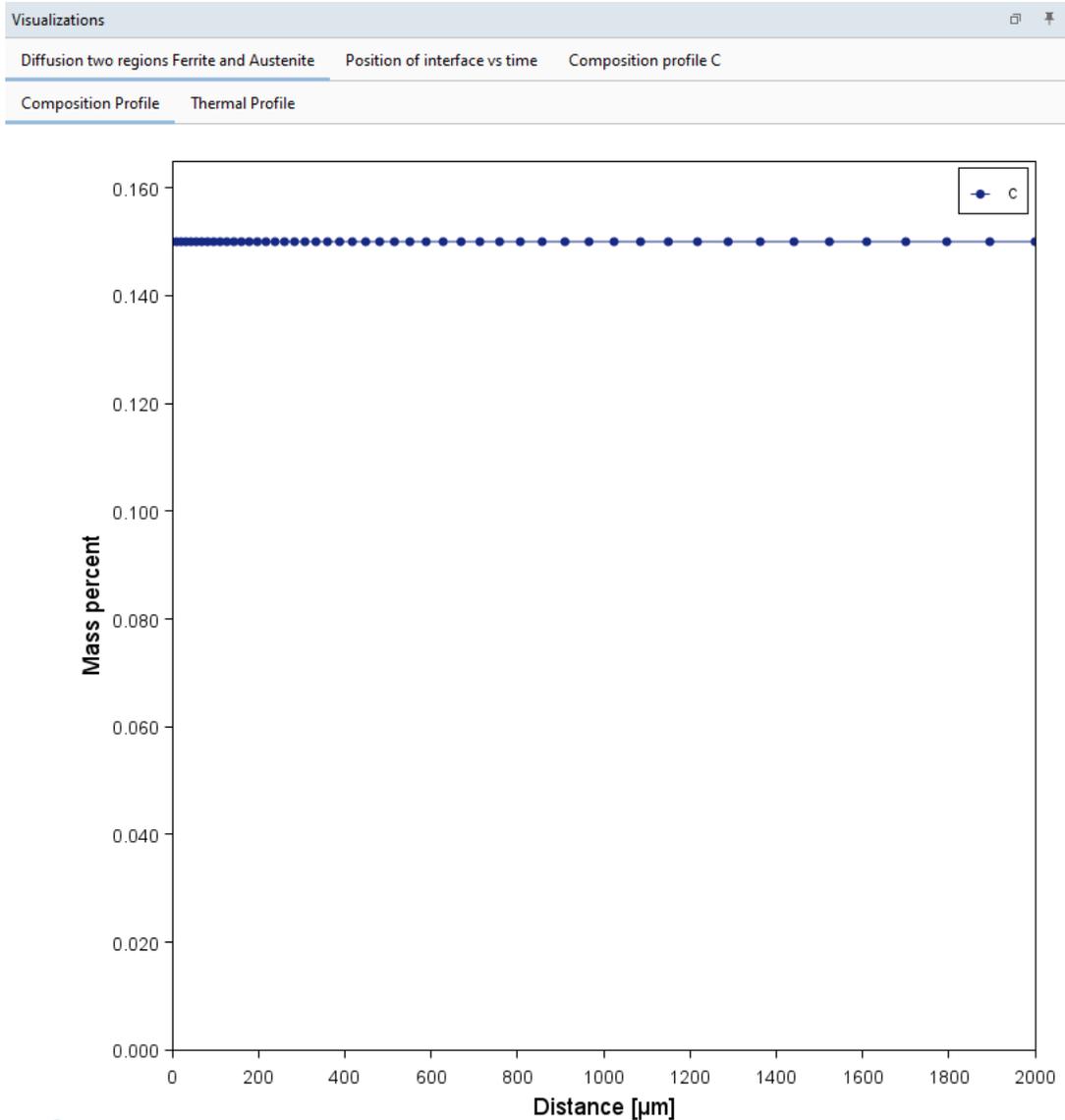


Figure 3: During set up of the calculation, you can preview the Composition Profile (shown) or Thermal Profile. Click the tab(s) in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window. For an Isothermal Thermal Profile this shows the constant temperature as entered.

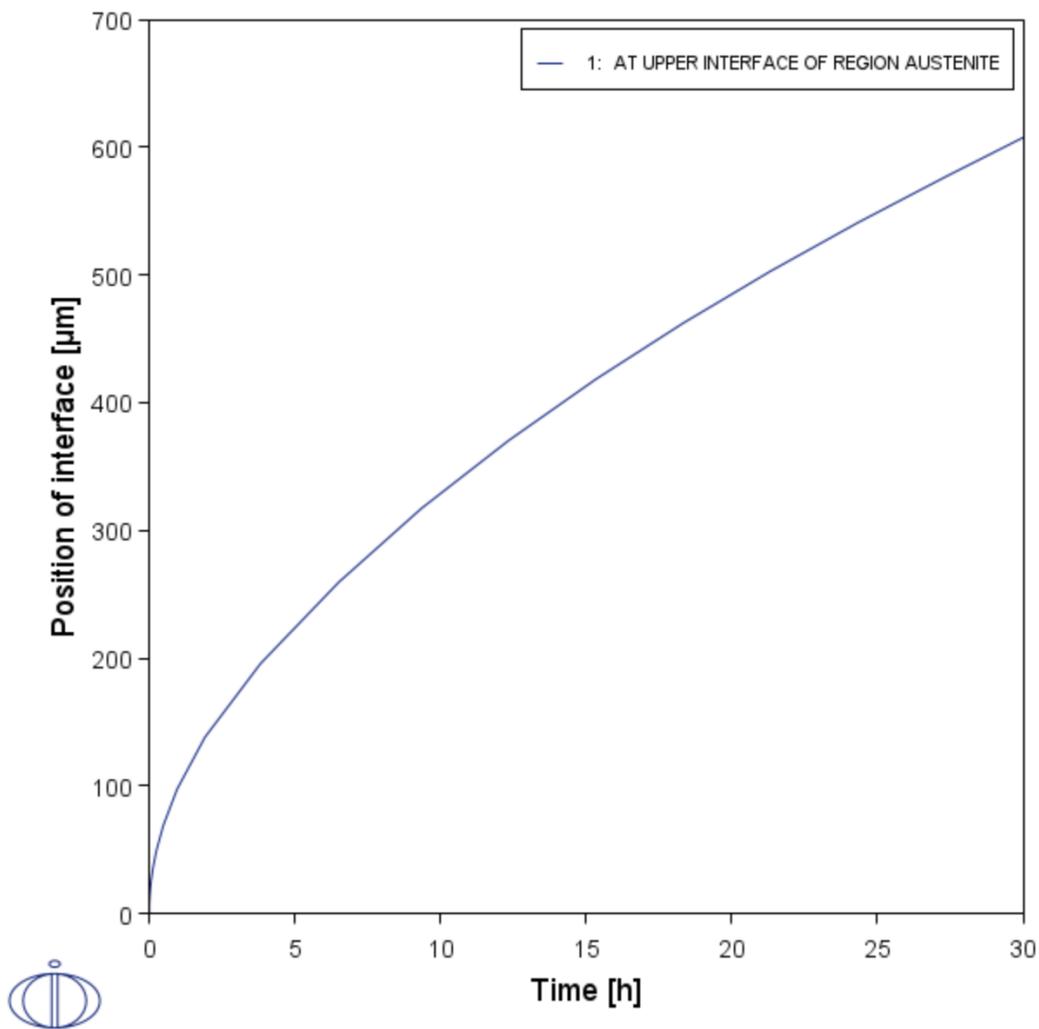


Figure 4: After performing the calculation, you can view the final Plot Renderer result on the Position of interface vs time tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

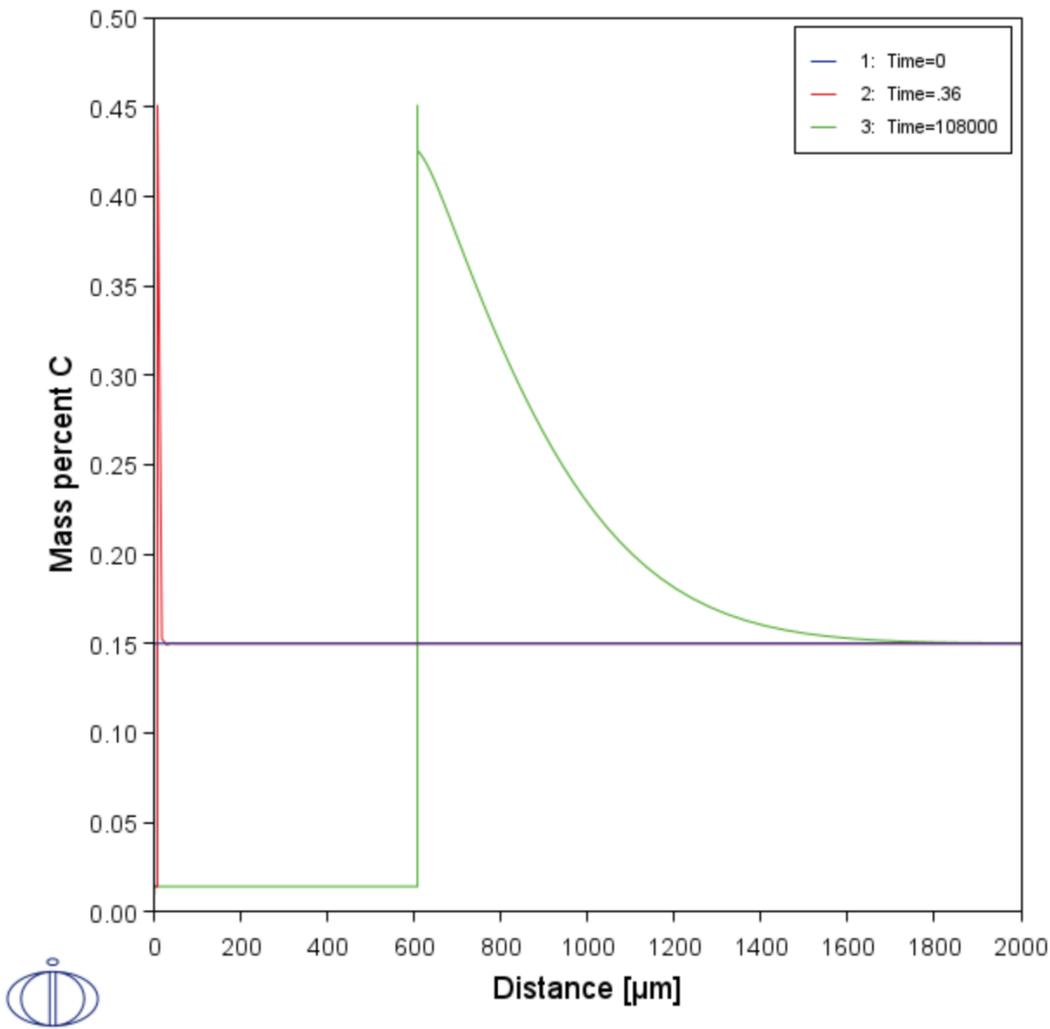


Figure 5: After performing the calculation, you can view the final Plot Renderer result on the Composition profile C tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

## D\_03: Evolution of an Fe-Cr-Ni Diffusion Couple

---

The multiphase example simulates the evolution of an Fe-Cr-Ni diffusion couple during a 100 hour heat treatment at 1100° C (1373.15 K). Both end members of the diffusion couple are duplex ferrite plus austenite, but the majority phase is ferrite on the left-hand side and austenite on the right.

With this type of simulation it is assumed that the material is fully equilibrated at each grid point, i.e. the local phase fractions, phase compositions and so forth are obtained from an equilibrium calculation with the local overall composition as a condition.

More details about the homogenization model for multiphase simulations can be found in Larsson and Engström [2006Lar] and Larsson and Höglund [2009Lar]. Experimental data is from Engström [1995Eng].

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_03\_Diffusion\_Multiphase.tcu*



This example is part of the *Diffusion Module (DICTRA) Quick Start Guide* available to all users. A version of the example is also available for Console Mode.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.
- **Plot** or **Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer**

or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

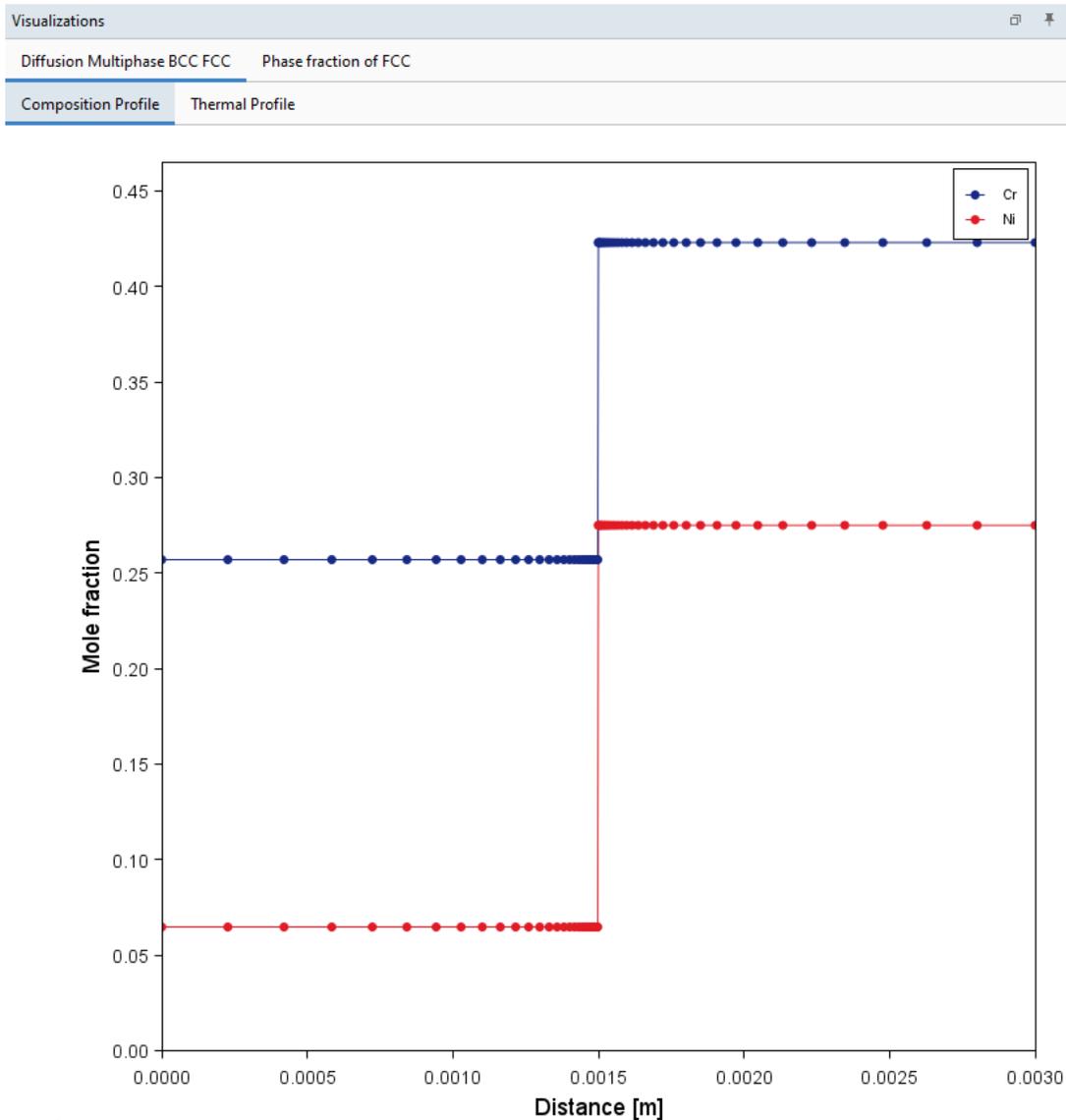


Figure 6: During set up of the calculation, you can preview the Composition Profile (shown) or Thermal Profile. Click the tab(s) in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window. For an Isothermal Thermal Profile this shows the constant temperature as entered.

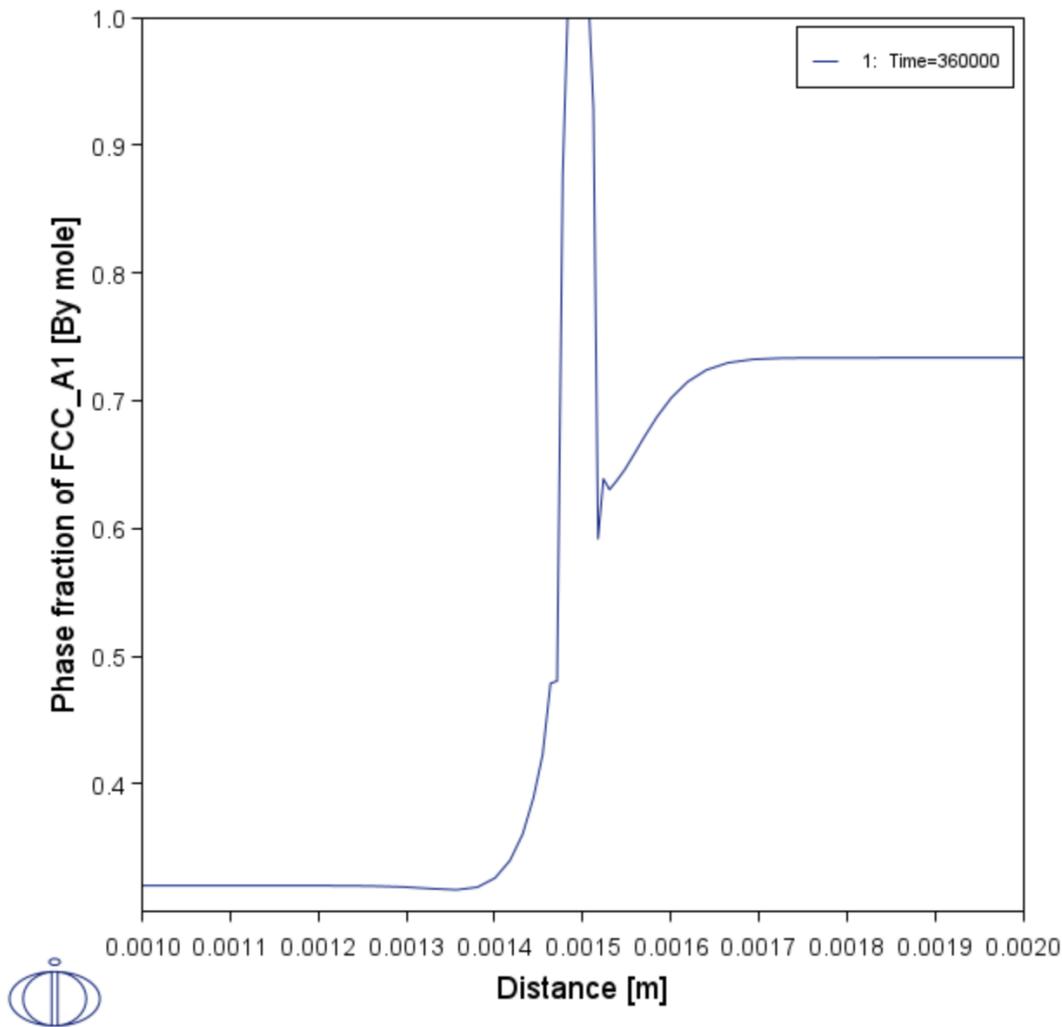


Figure 7: After performing the calculation, you can view the final Plot Renderer result on the Phase fraction of FCC tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

## References

- [1995Eng] A. Engström, Interdiffusion in multiphase, Fe-Cr-Ni diffusion couples. Scand. J. Metall. 24, 12–20 (1995).
- [2006Lar] H. Larsson, A. Engström, A homogenization approach to diffusion simulations applied to  $\alpha+\gamma$  Fe-Cr-Ni diffusion couples. Acta Mater. 54, 2431–2439 (2006).
- [2009Lar] H. Larsson, L. Höglund, Multiphase diffusion simulations in 1D using the DICTRA homogenization model. Calphad. 33, 495–501 (2009).

## D\_04: Fe-C Moving Boundary: Austenite to Ferrite

---

This example simulates the austenite to ferrite transformation in a Fe-0.01 mass% C steel during continuous cooling. The simulation starts at a temperature where only austenite is stable, ferrite nucleates and grows into the austenite during cooling in the two phase region. Plots of thermal and carbon composition profiles are generated.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_04\_Diffusion\_Fe-C\_Moving\_Boundary\_Austenite\_to\_Ferrite.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## **Visualizations**

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.
- **Plot** or **Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

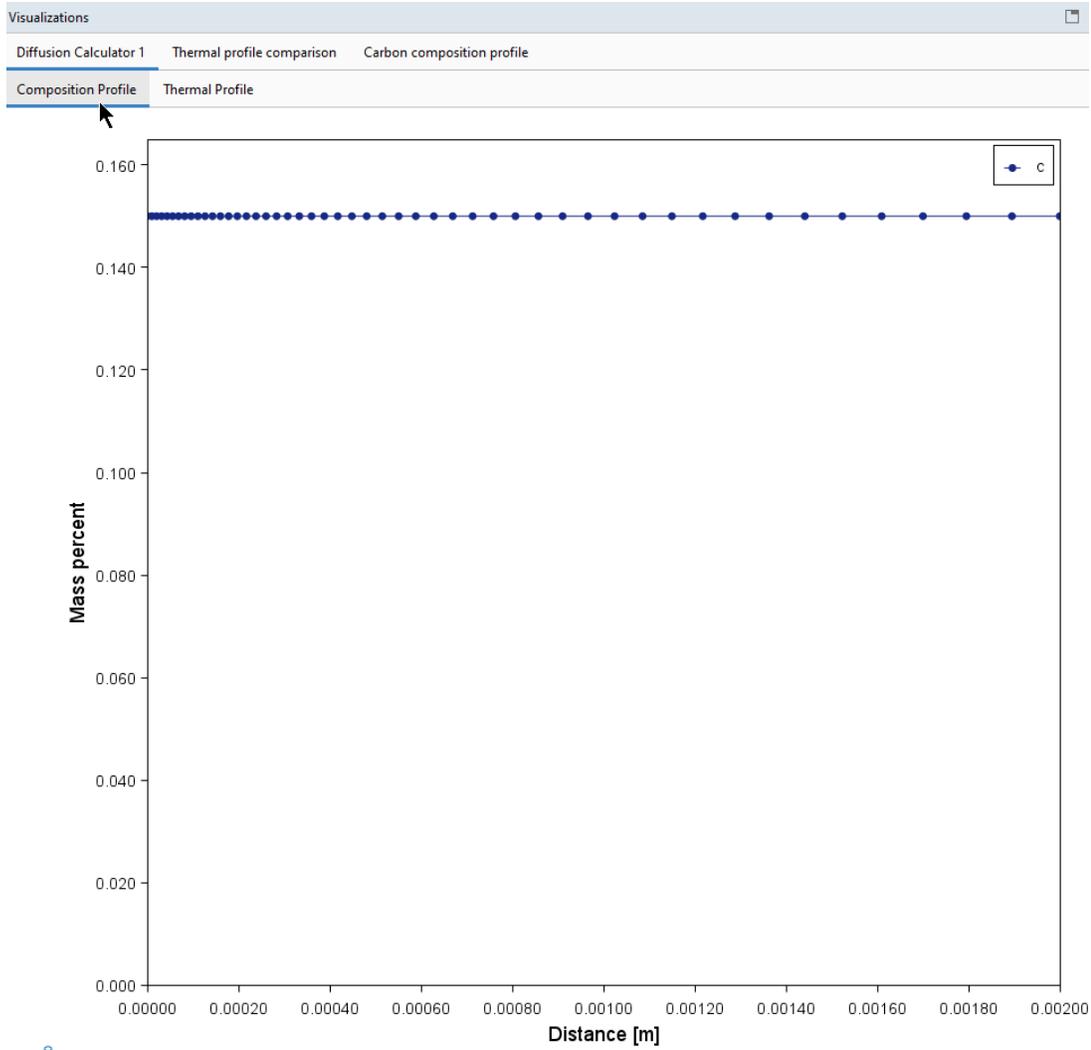


Figure 8: During set up of the calculation, you can preview the Composition Profile. Click the tab in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window.

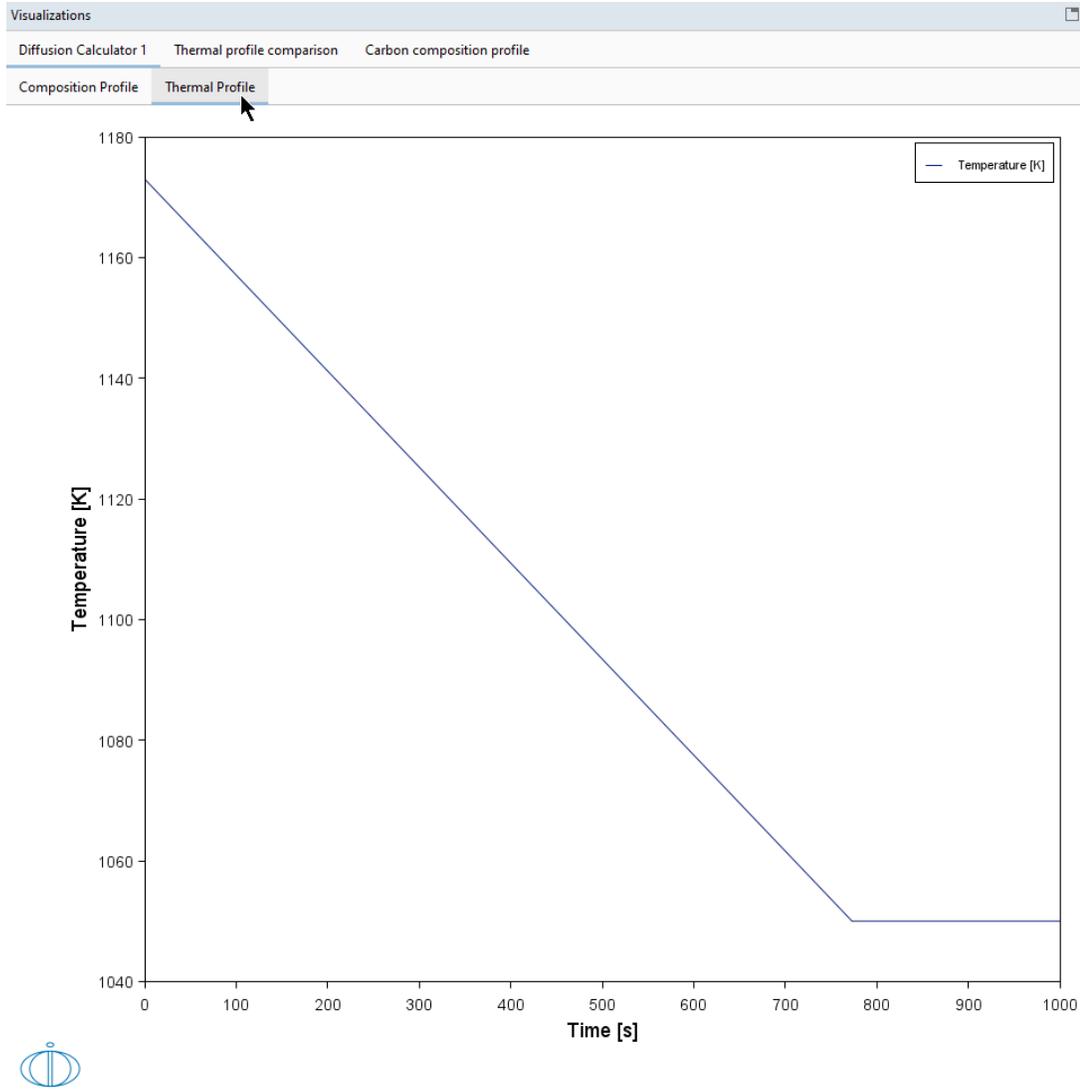


Figure 9: During set up of the calculation, you can preview the non-isothermal Thermal Profile. Click the tab in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window. Also see the plot result comparing the thermal profile for the two regions.

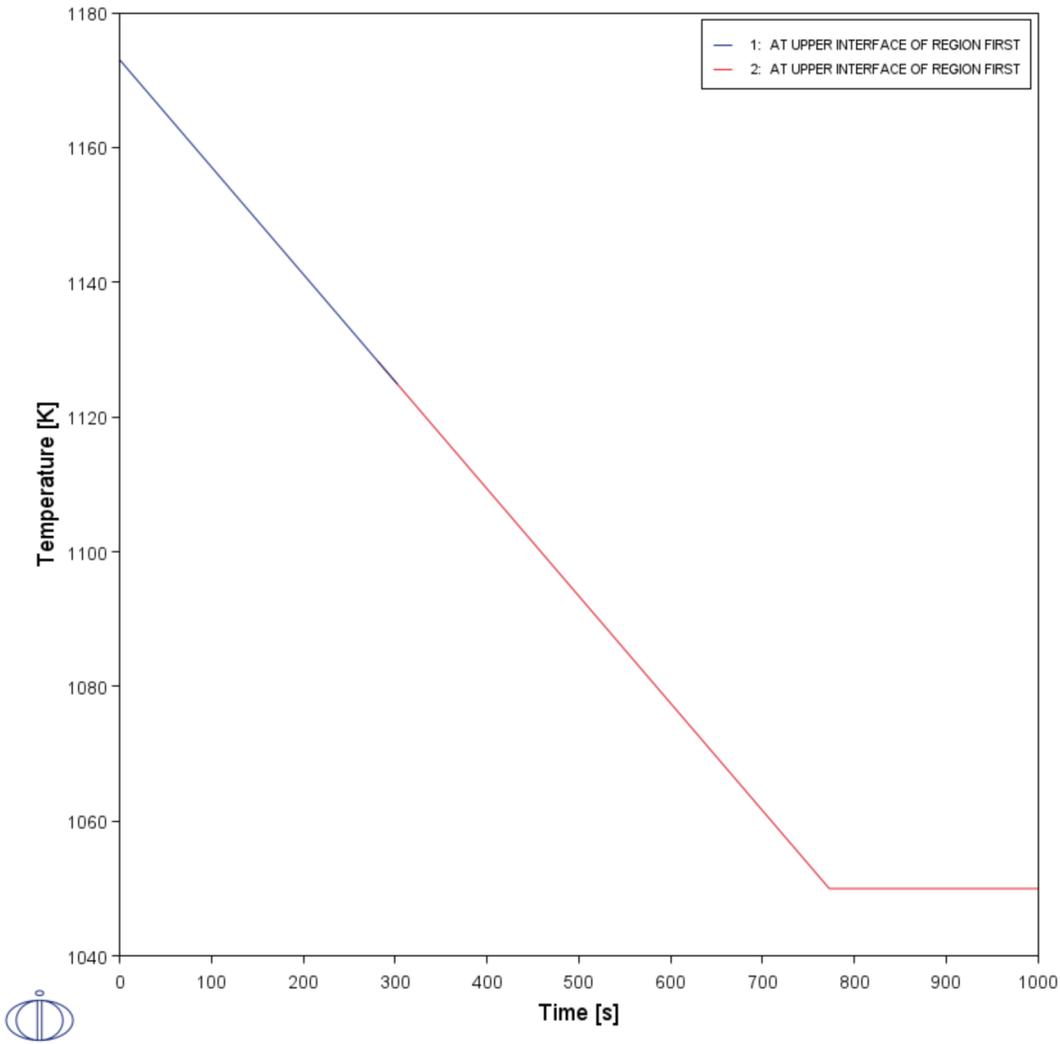


Figure 10: After performing the calculation, you can view the final Plot Renderer result on the Thermal profile tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window. Note that this Thermal Profile is slightly different than what you can preview during set up.

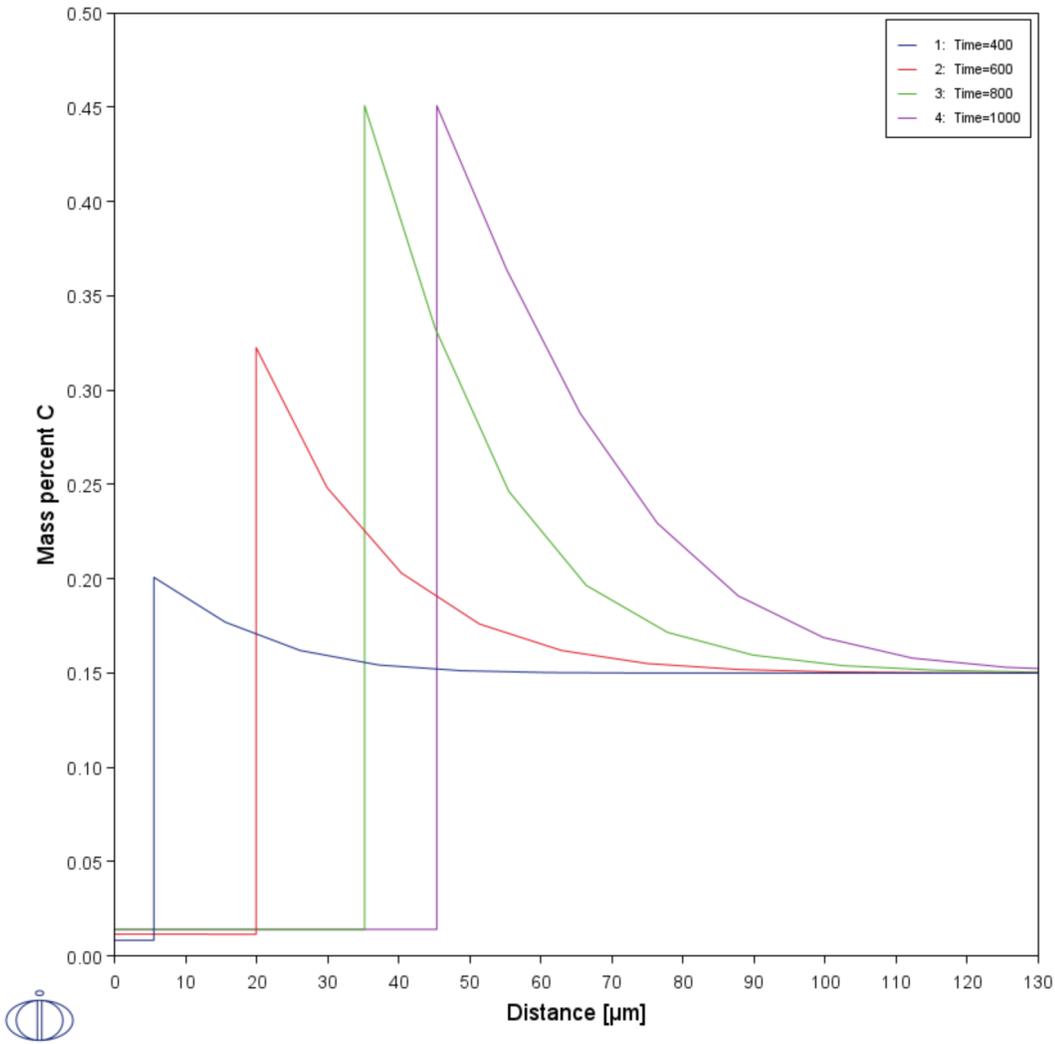
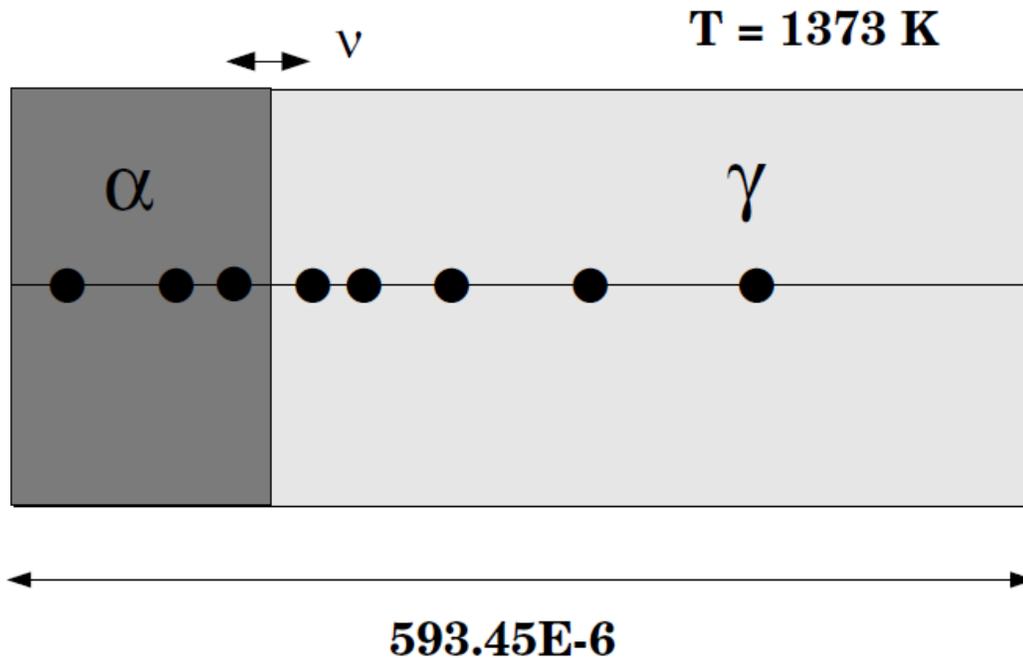


Figure 11: After performing the calculation, you can view the final Plot Renderer result on the Carbon composition profile tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

## D\_05: $\gamma/\alpha/\gamma$ Diffusion Couple of Fe-Ni-Cr alloys

This example demonstrates the evolution of a ternary Fe-Cr-Ni diffusion couple. A thin slice of ferrite ( $\alpha$  phase) (38%Cr,0%Ni) is clamped between two thicker slices of austenite ( $\gamma$  phase) (27%Cr, 20%Ni). The assembly is subsequently heat treated at 1373 K.

This set up corresponds to diffusion couple in Kajihara et al. [1993aKaj] and Kajihara and Kikuchi [1993bKaj].



Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_05\_Diffusion\_Fe\_Ni\_Cr\_Moving\_Boundary\_Diffusion\_Couple.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.
- **Plot** or **Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

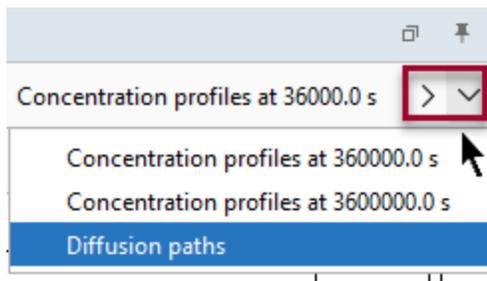


Figure 12: In this example, all Plot Renderer nodes are renamed and each match a tab name in the Visualizations window. Use the left and right arrows in the Visualizations window to move through the tabs and the down arrow to choose by name when a tab is not visible.

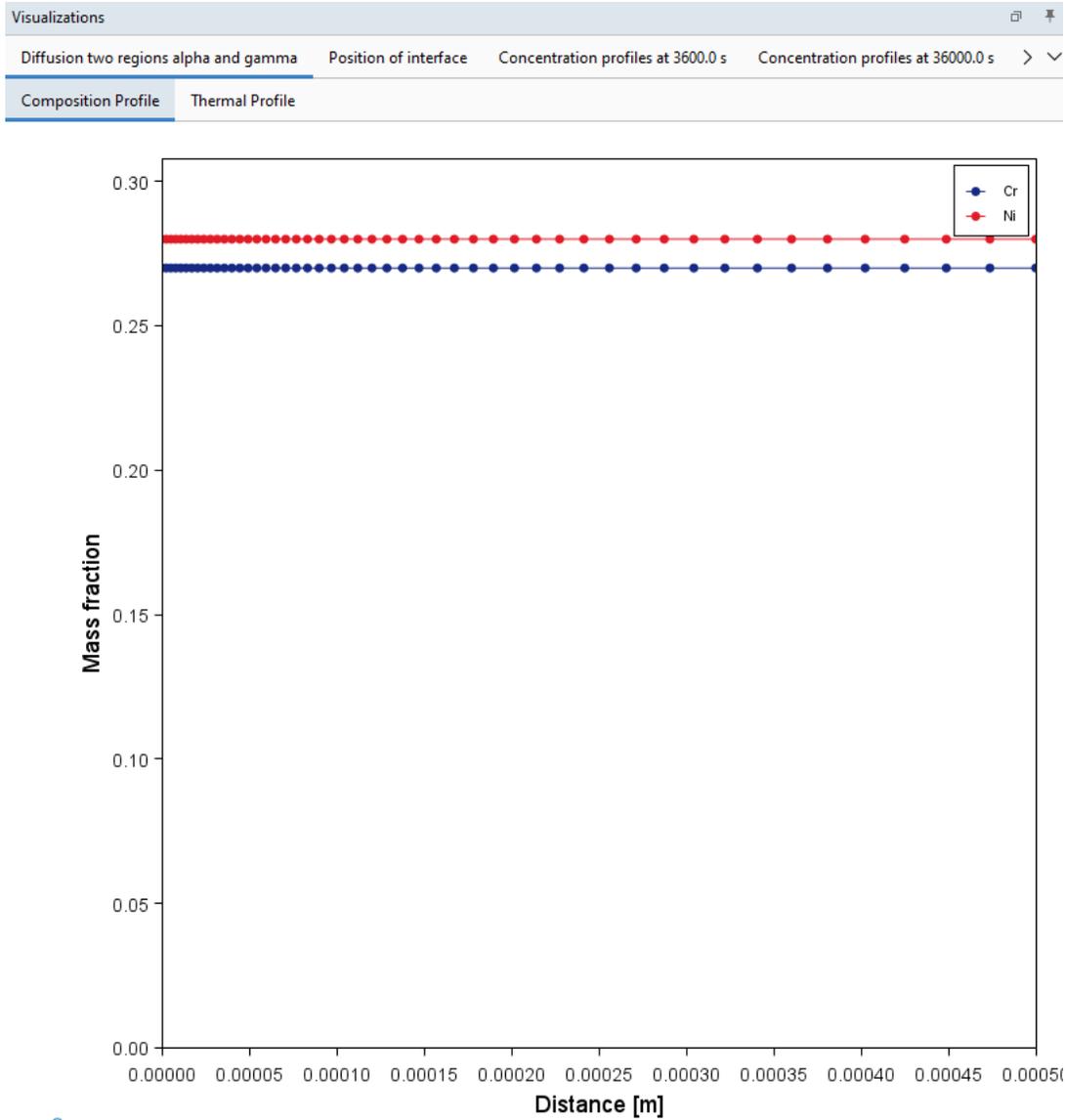


Figure 13: During set up of the calculation, you can preview the Composition Profile (shown) or Thermal Profile. Click the tab(s) in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window. For an Isothermal Thermal Profile this shows the constant temperature as entered.

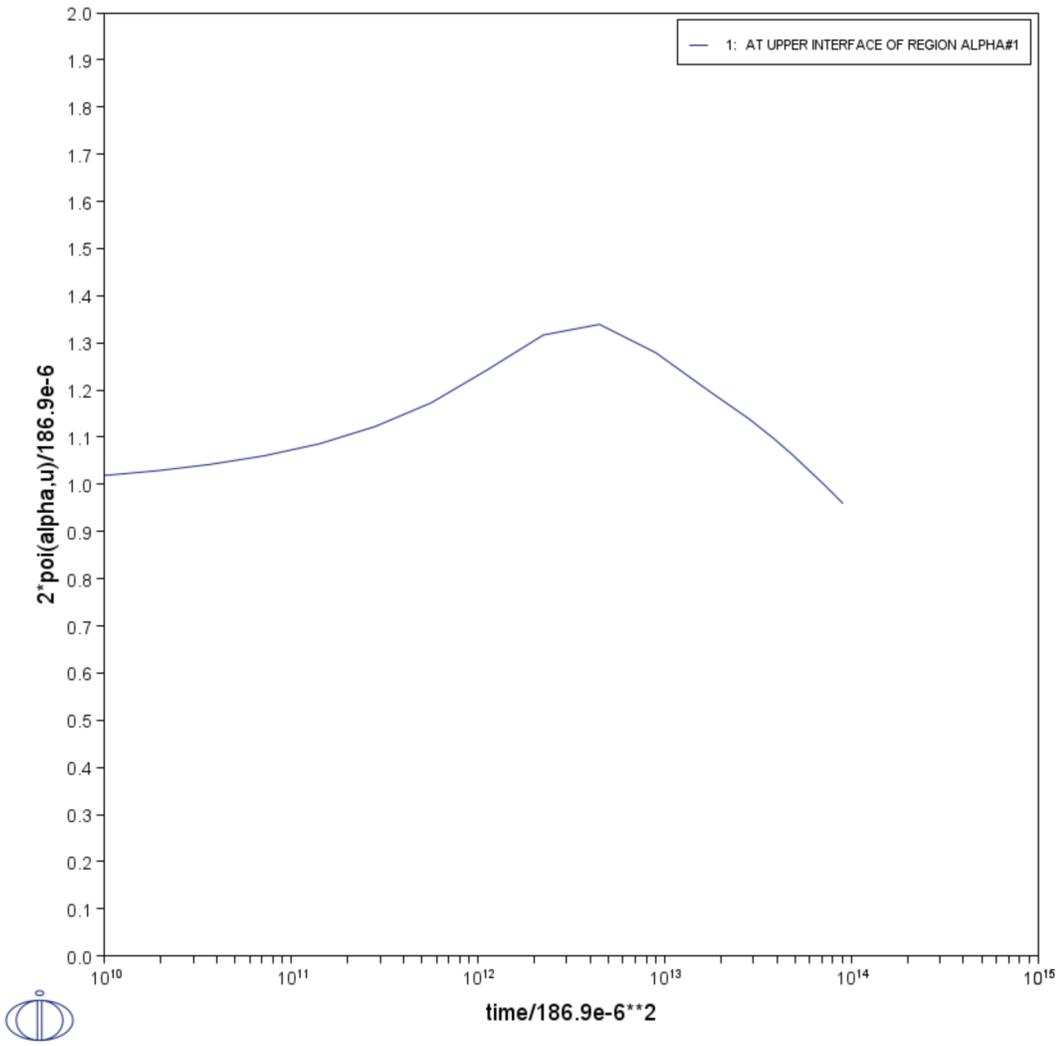


Figure 14: After performing the calculation, you can view the final Plot Renderer result on the Interface position tab.

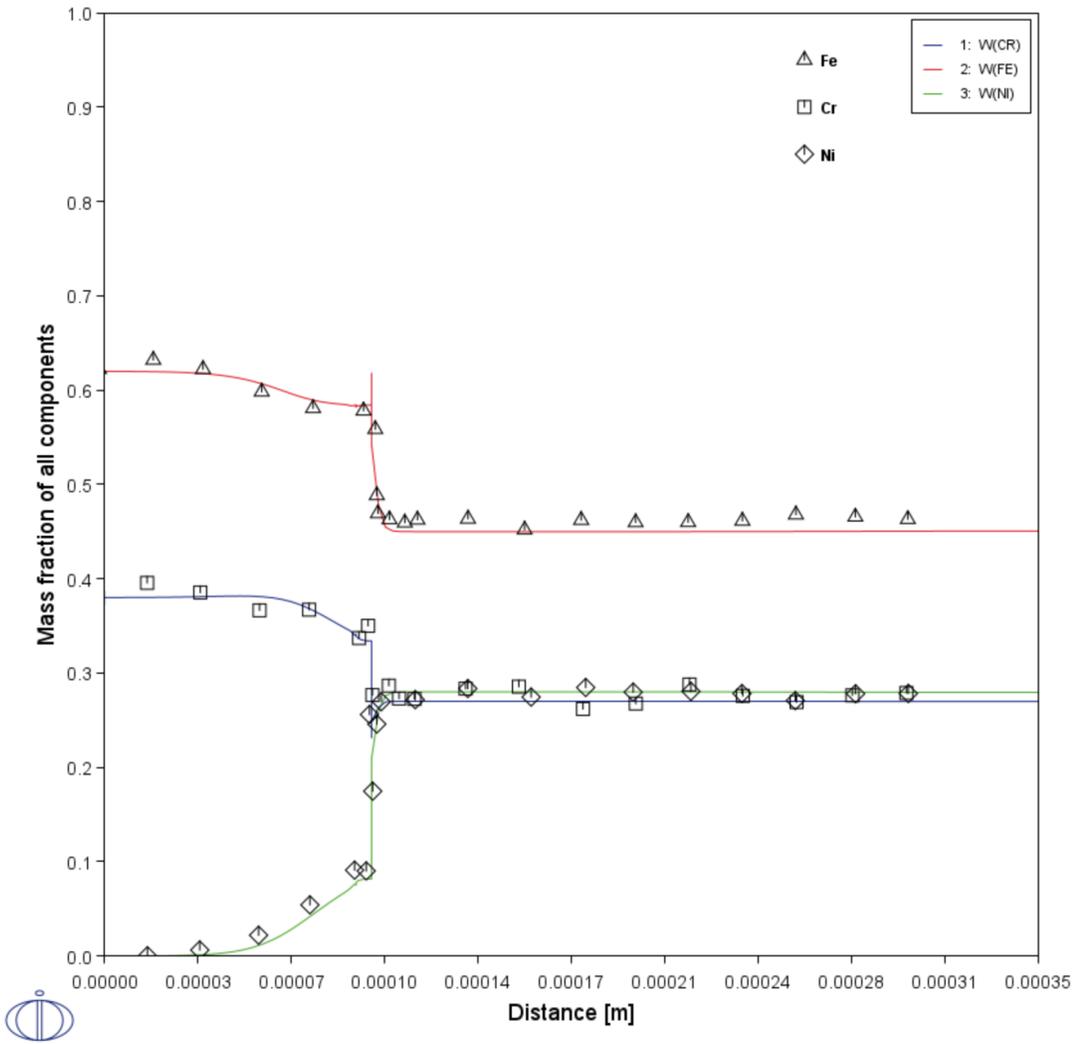


Figure 15: After performing the calculation, you can view the final Plot Renderer result on the Concentration profiles at 3600 s tab.

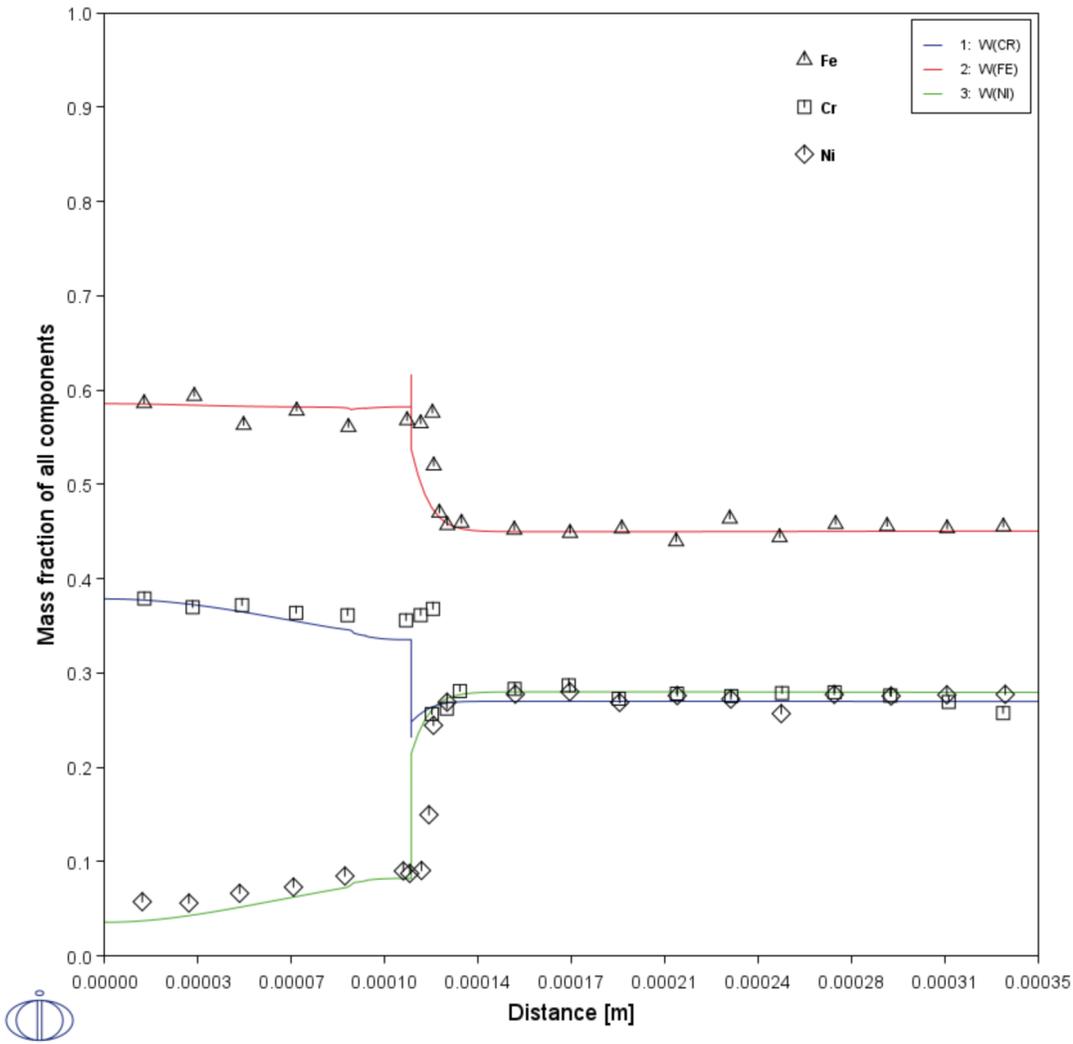


Figure 16: After performing the calculation, you can view the final Plot Renderer result on the Concentration profiles at 36 000 s tab.

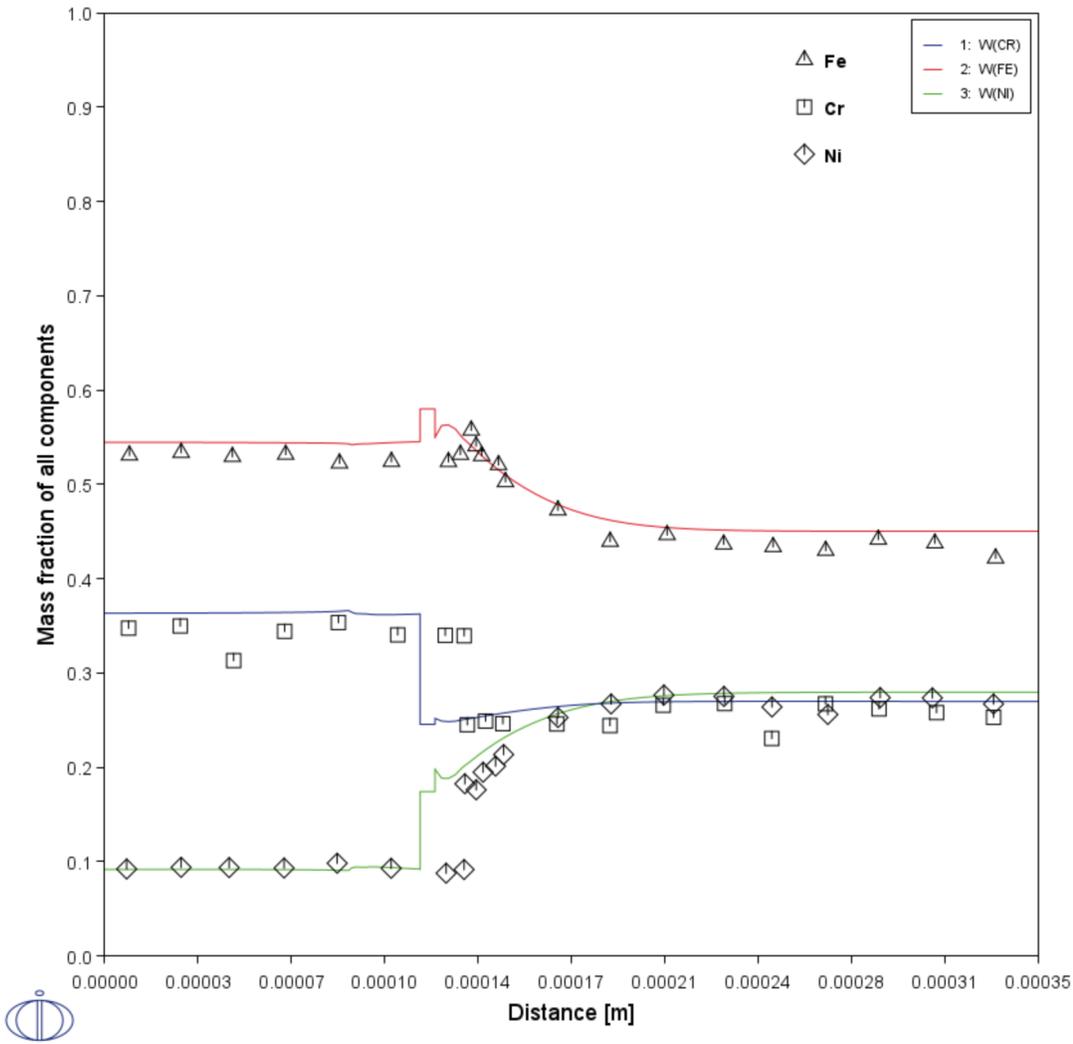


Figure 17: After performing the calculation, you can view the final Plot Renderer result on the Concentration profiles at 360 000 s tab.

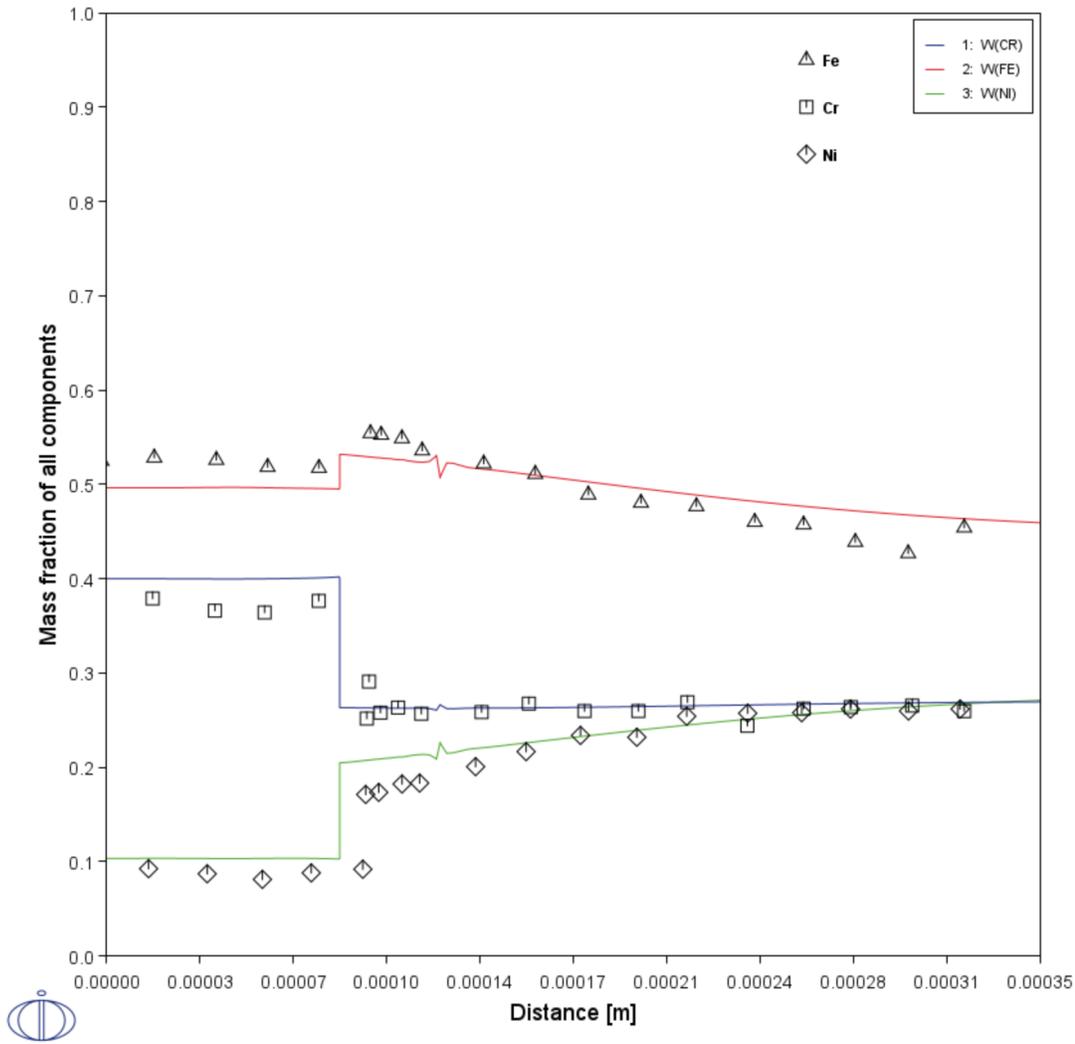


Figure 18: After performing the calculation, you can view the final Plot Renderer result on the Concentration profiles at 3 600 000 s tab.

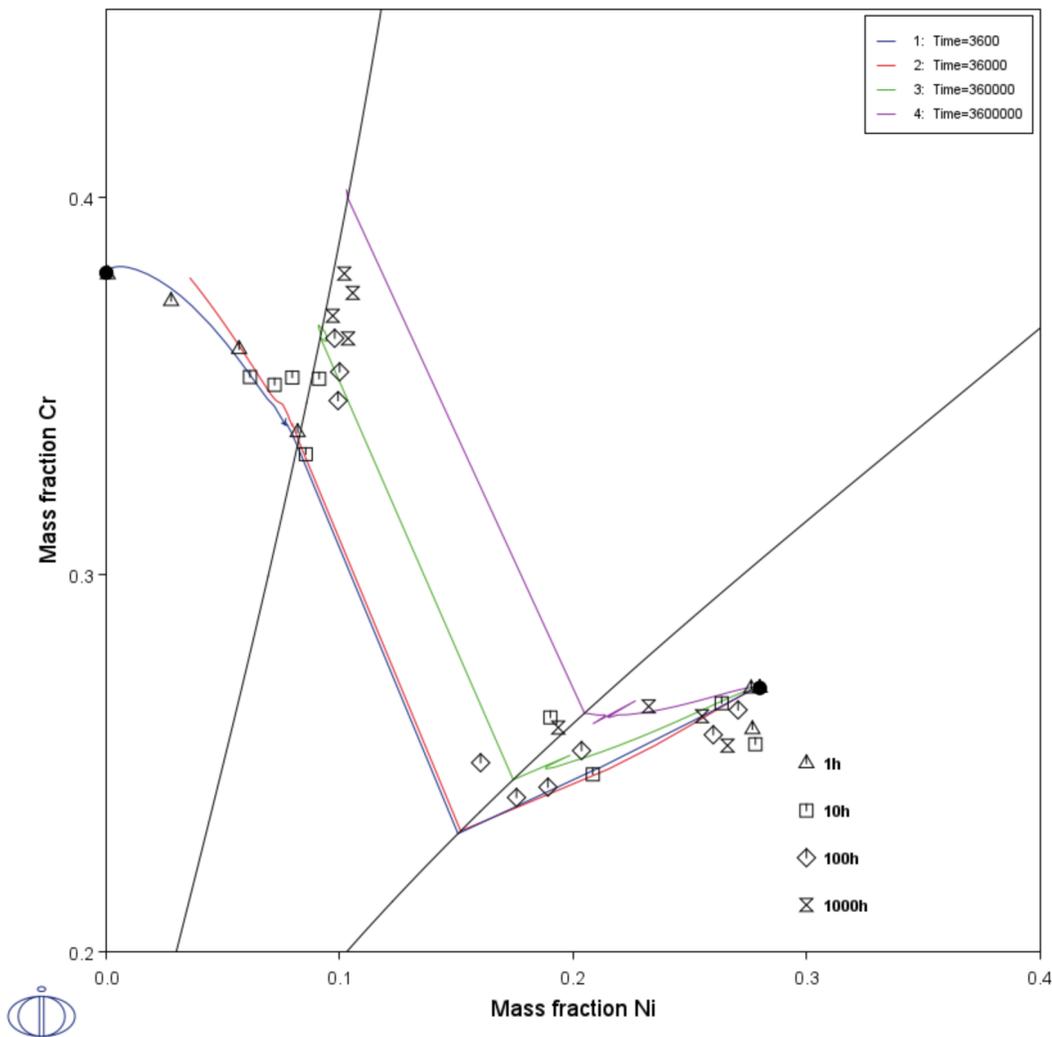


Figure 19: After performing the calculation, you can view the final Plot Renderer result on the Diffusion paths tab.

## References

- [1993aKaj] M. Kajihara, C.-B. Lim, M. Kikuchi, Experimental Study on Dissolution of ALPHA Phase in GAMMA/ALPHA/GAMMA Diffusion Couples of the Fe-Cr-Ni System. *ISIJ Int.* 33, 498–507 (1993).
- [1993bKaj] M. Kajihara, M. Kikuchi, Numerical analysis of dissolution of  $\alpha$  phase in  $\gamma/\alpha/\gamma$  diffusion couples of the Fe-Cr-Ni system. *Acta Metall. Mater.* 41, 2045–2059 (1993).

## D\_06: Diffusion Through a Tube Wall

---

This is a simple example of diffusion through a tube wall. The tube material is an Fe-0.06Mn-0.05C alloy. Two plots comparing distance to the U-fraction of manganese and composition of carbon are generated to visualize the austenite region. A *cylindrical* geometry is used with *mixed zero flux and activity* boundary conditions.

On the inside wall a carbon activity of 0.9 is maintained whereas on the outside the carbon activity is very low. This example demonstrates the use of boundary conditions, advanced plotting and tables.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_06\_Diffusion\_Carburization\_Tube.tcu*



This example is based on Console Mode exab6. When in Console Mode, you can open the example from Thermo-Calc (**File** → **Examples Files** → **Diffusion Module**).



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.
  - **Plot** or **Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.
-

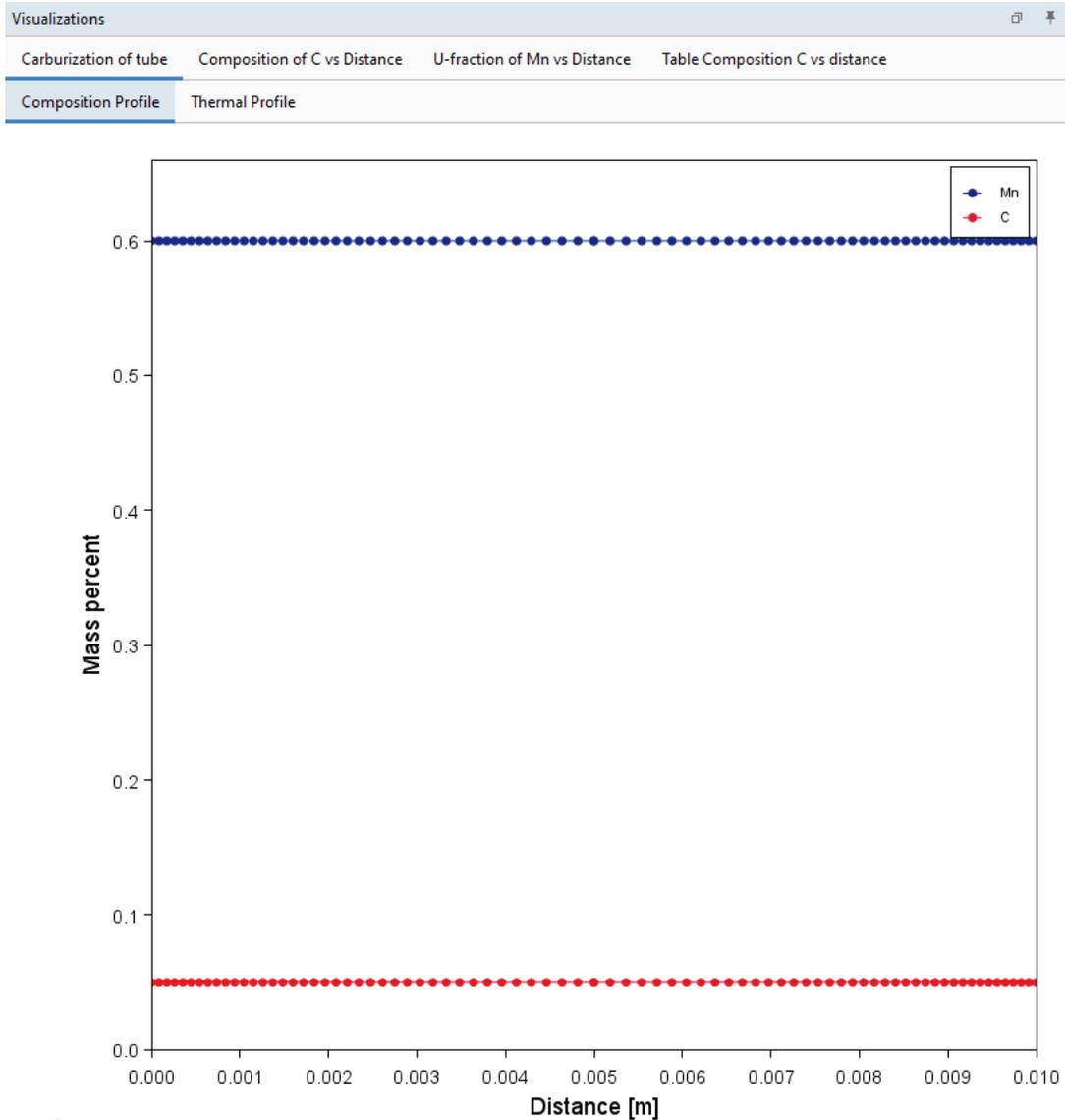


Figure 20: During set up of the calculation, you can preview the Composition Profile (shown) or Thermal Profile. Click the tab(s) in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window. For an Isothermal Thermal Profile this shows the constant temperature as entered.

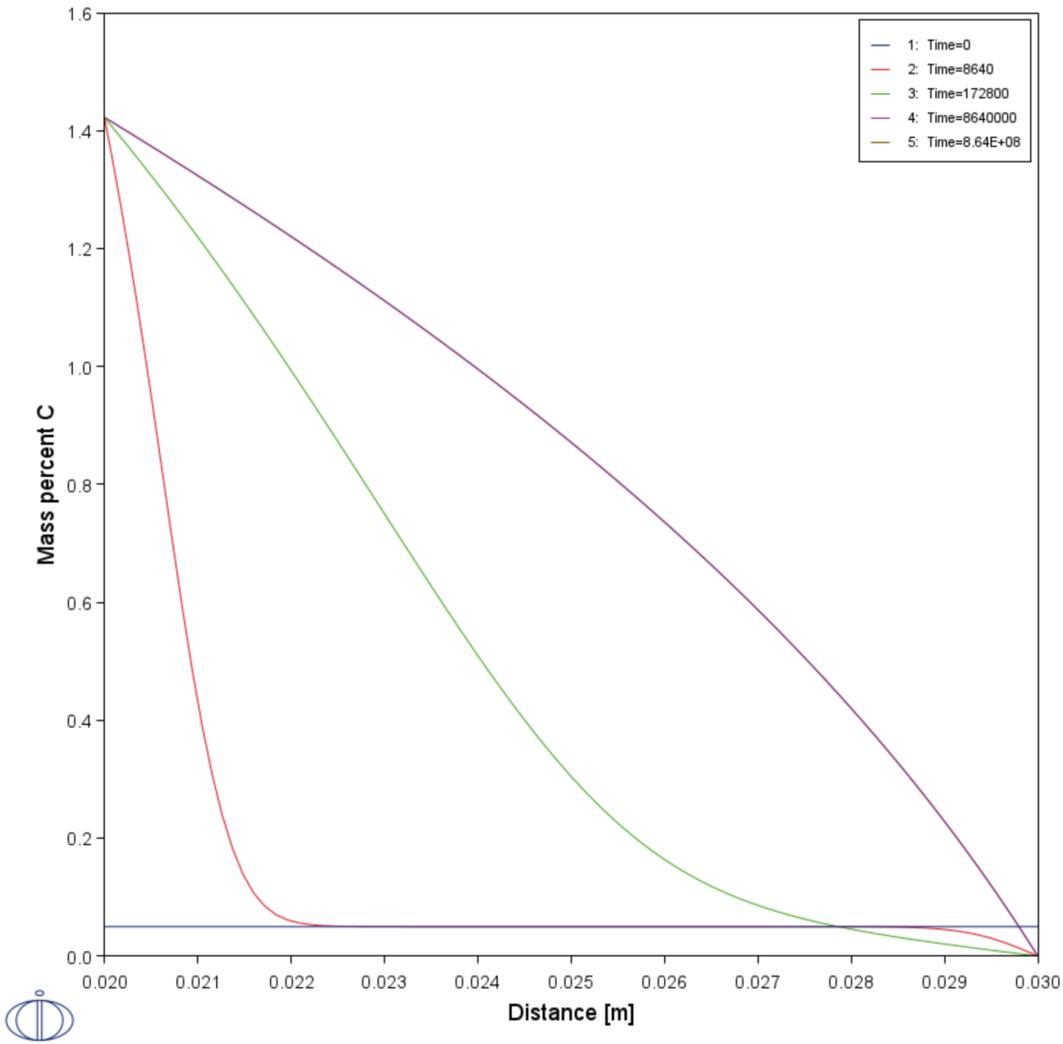


Figure 21: After performing the calculation, you can view the final Plot Renderer result on the Composition of C vs Distance tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

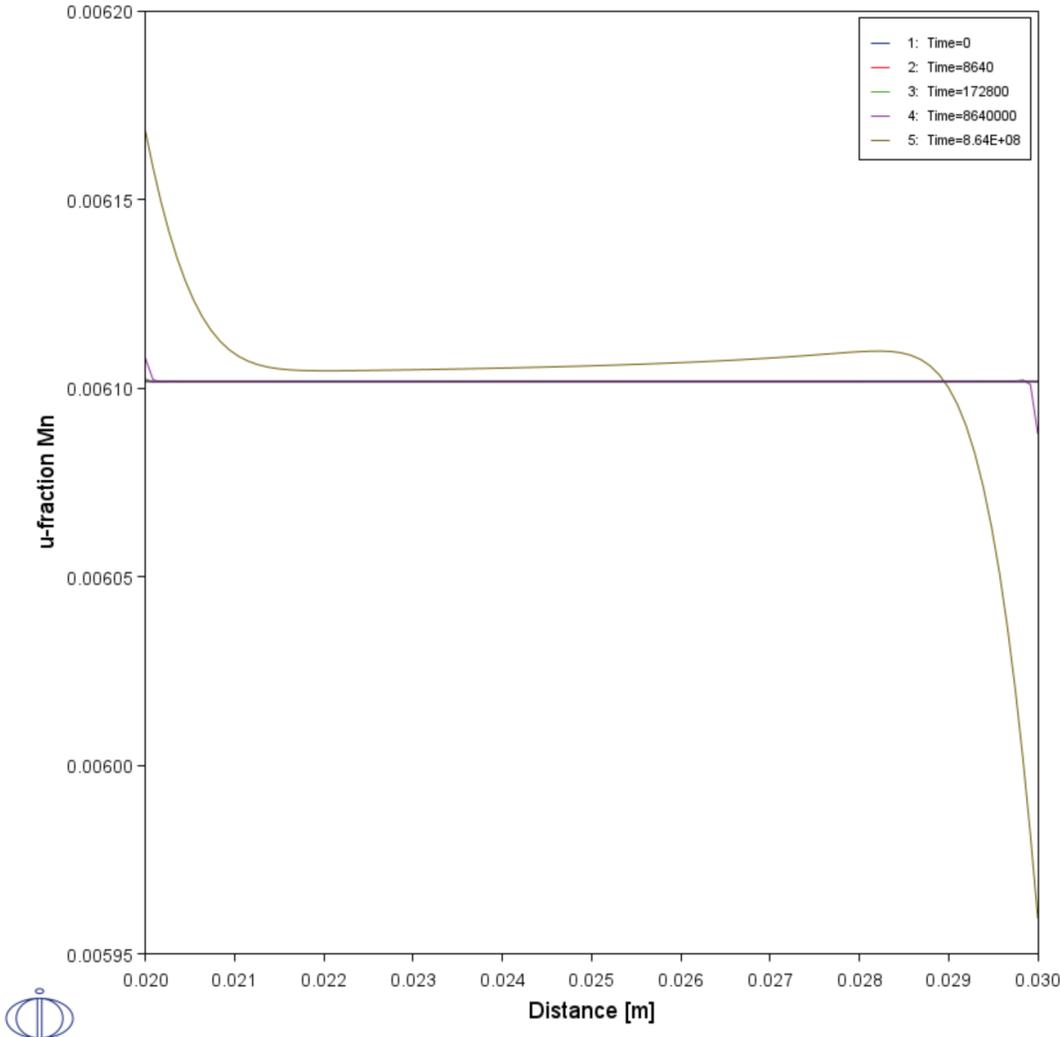


Figure 22: After performing the calculation, you can view the final Plot Renderer result on the U-fraction of Mn vs Distance tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

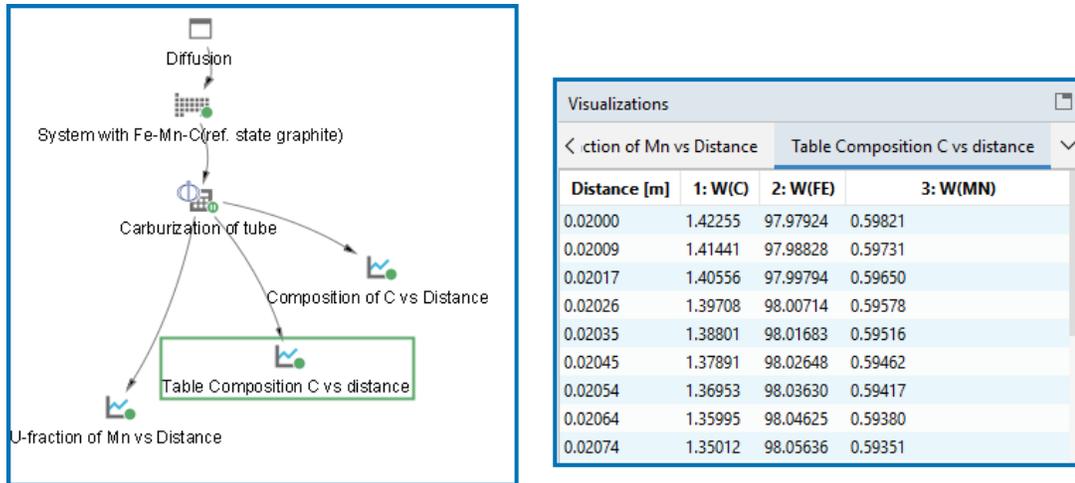


Figure 23: After performing the calculation, you can view the final Table Renderer result on the Table Composition C vs Distance tab. In this example, the Table Renderer is renamed and this matches the tab name in the Visualizations window.

## D\_07: Multiphase Carburization of an Alloy

---

This example is based on Engström et al. [1994Eng] and is about carburization of a Ni-25Cr-0.0001C alloy. In this case the M7C3 and M3C2 carbides are entered as spheroid phases in an FCC\_A1 matrix. It is similar to Graphical Mode example D\_06 except the default simulation condition is automatically set to use the homogenization model.

The isothermal calculation is run for 1000 hours at a temperature of 1123 K using the mixed zero flux and activity left boundary condition. Results are plotted using two Plot Renderers and a Table Renderer.



This example is included as a Diffusion Module (DICTRA) tutorial on our [website](#) and as part of the playlist on our [YouTube channel](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_07\_Diffusion\_Carburization\_Multiphase.tcu*



This example is based on Console Mode exd1b. When in Console Mode, you can open the example from Thermo-Calc (**File** → **Examples Files** → **Diffusion Module**).

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

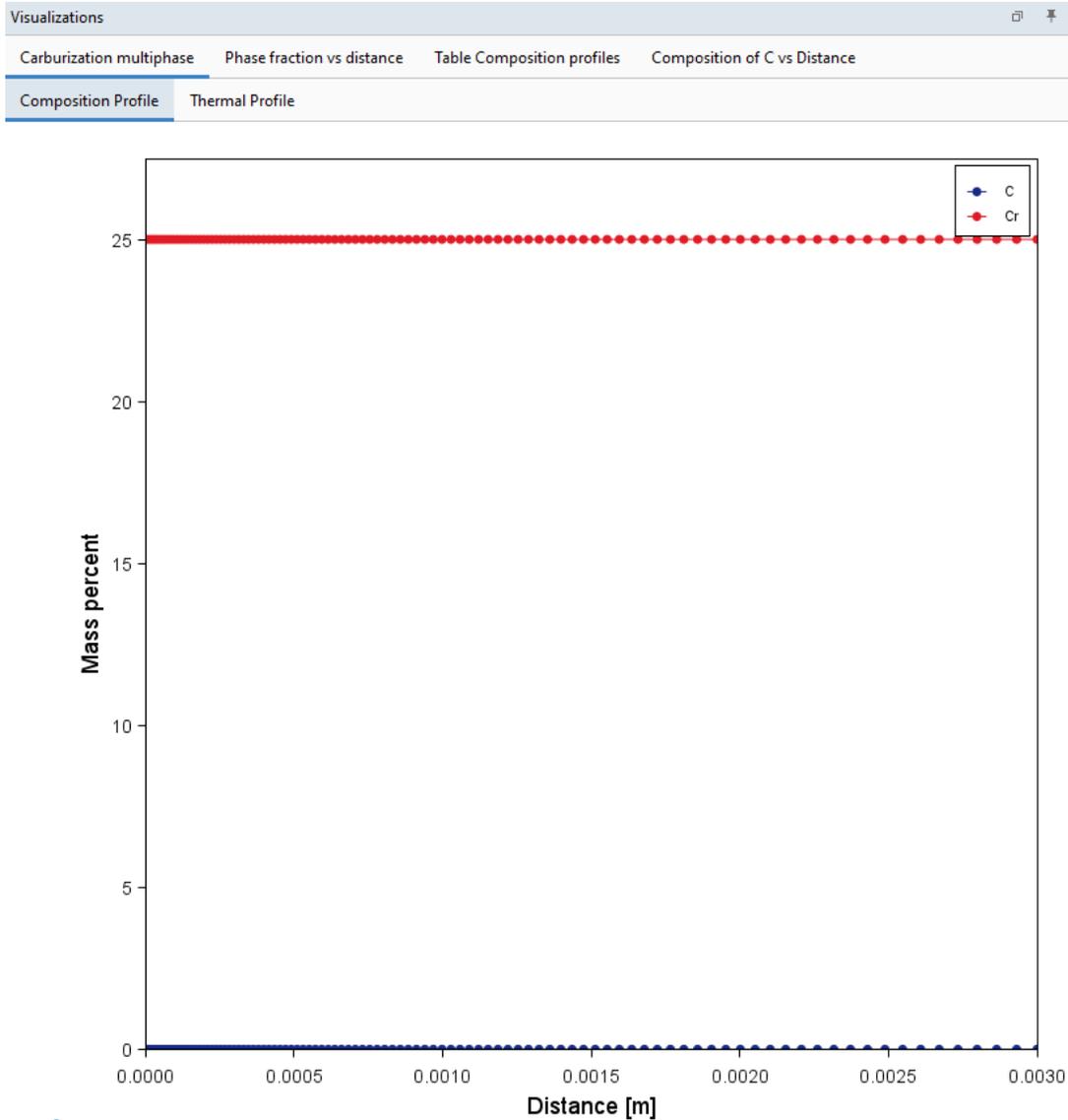


Figure 24: During set up of the calculation, you can preview the Composition Profile (shown) or Thermal Profile. Click the tab(s) in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window. For an Isothermal Thermal Profile this shows the constant temperature as entered.

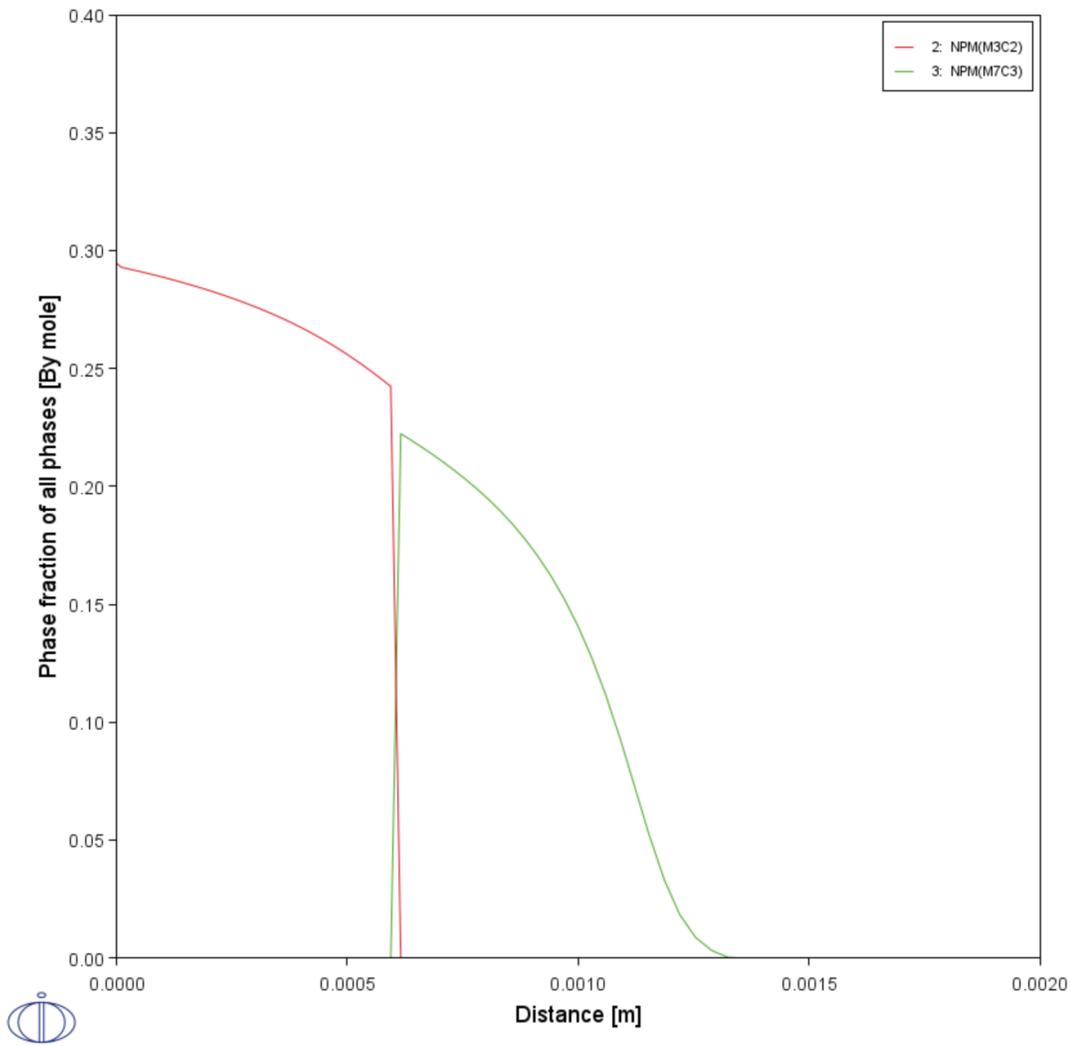


Figure 25: After performing the calculation, you can view the final Plot Renderer result on the Phase fraction vs distance tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

Visualizations			
Carburization multiphase	Phase fraction vs distance	Table Composition profiles	Composition of C vs Distance
Distance [m]	1: W(C)	2: W(CR)	3: W(NI)
0.00000	2.94652	24.36016	72.69332
9.88104E-6	2.92735	24.27154	72.80111
0.00002	2.92100	24.27340	72.80561

Figure 26: After performing the calculation, you can view the final Table Renderer result on the Table of Composition profiles tab. In this example, the Table Renderer is renamed and this matches the tab name in the Visualizations window.

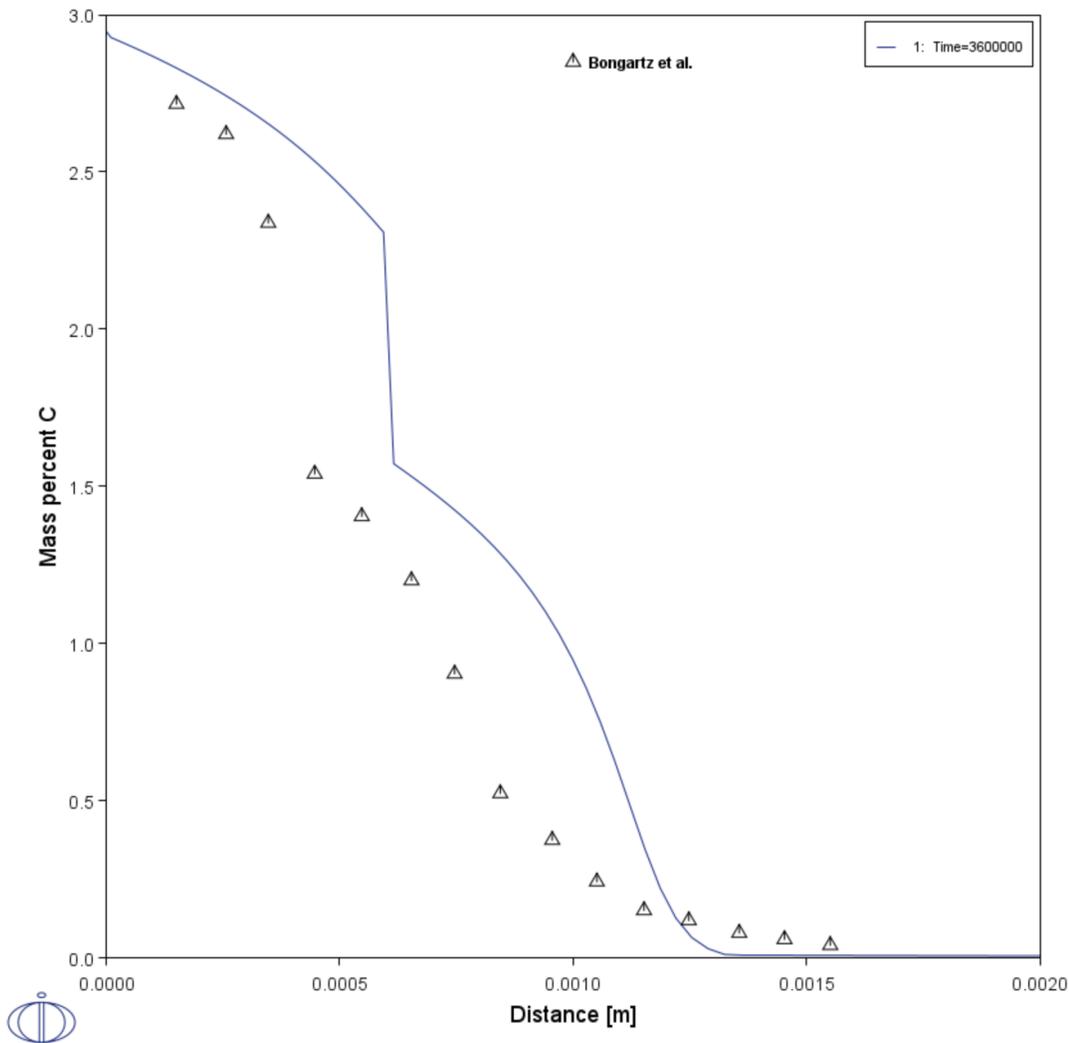


Figure 27: After performing the calculation, you can view the final Plot Renderer result on the Composition of C vs distance tab. In this example, the Plot Renderer is renamed and this matches the tab name in the Visualizations window.

## Reference

[1994Eng] A. Engström, L. Höglund, J. Ågren, Computer simulation of diffusion in multiphase systems. Metall. Mater. Trans. A. 25, 1127–1134 (1994).

## D\_08: Microsegregation During Solidification

---

This application example, which is [showcased on our website](#), shows how Thermo-Calc together with the Diffusion Module (DICTRA) can be used to investigate and predict microsegregation during solidification, also known as solute redistribution during solidification. The example uses a steel with composition Fe – 0.8% Mn – 0.7%Si – 0.03%P – 0.4%C.

The example is based on a real case where the elemental distribution of a continuously cast steel was measured revealing the concentration of the elements Si, Mn, and P. The profile results from the well-known segregation across secondary dendrite arms. The interesting point is that the elements Si and Mn show the expected positive segregation (higher concentrations) in the interdendritic regions. The peak of the P content, on the other hand, is shifted compared to Si and Mn, and, in fact, shows negative segregation in the interdendritic region. This is unexpected and counter-intuitive.

The example also touches on some very basic concepts for metallurgy and using Thermo-Calc, such as stable and meta-stable phase diagrams, Scheil and para- or partial-equilibrium Scheil solidification simulations.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_08\_Diffusion\_Microsegregation\_During\_Solidification.tcu*



Running this calculation requires licenses for the Add-on Diffusion Module (DICTRA) plus the steels thermodynamic database (TCFE9 or newer) and mobility database (MOBFE4 or newer).

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

---

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.
- **Plot** or **Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

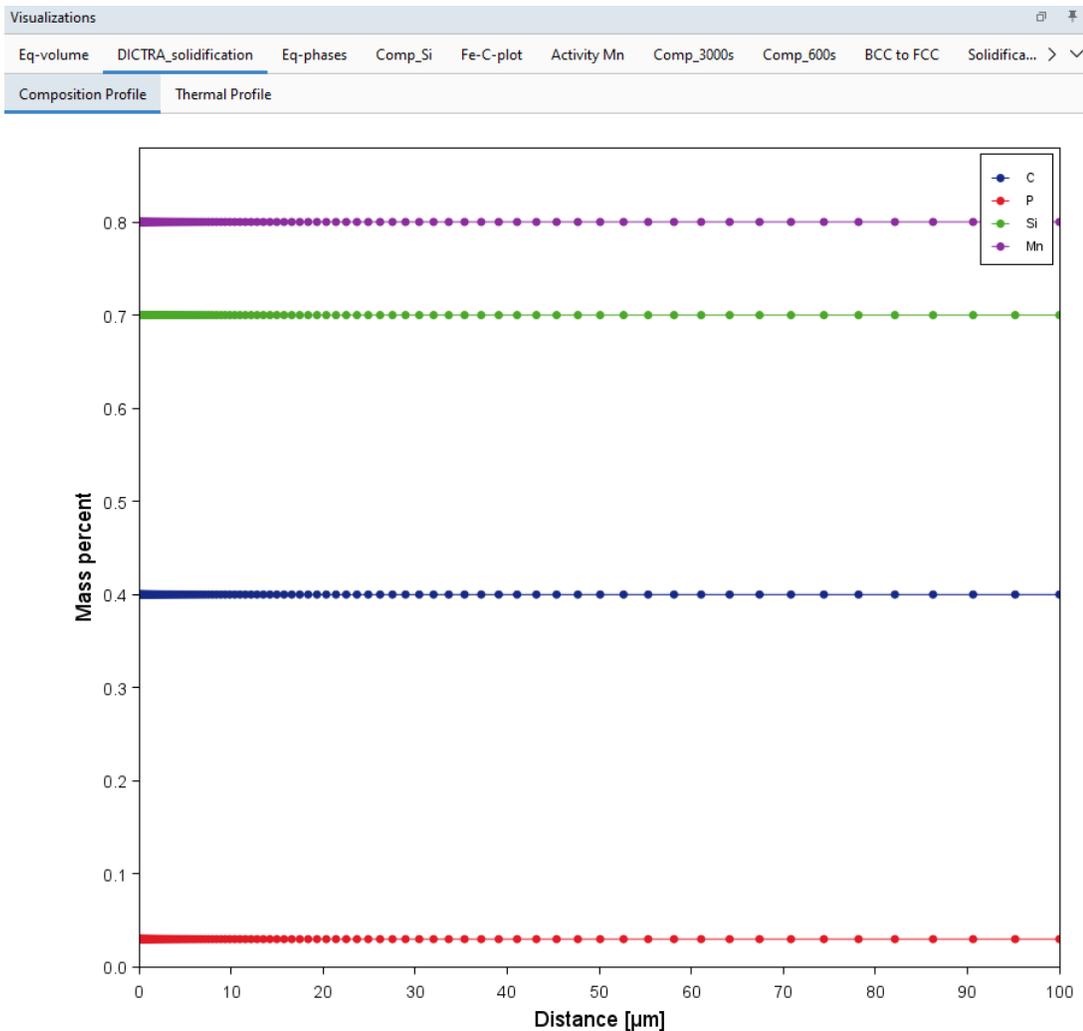


Figure 28: During set up of the calculation, you can preview the Composition Profile. Click the tab in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window.

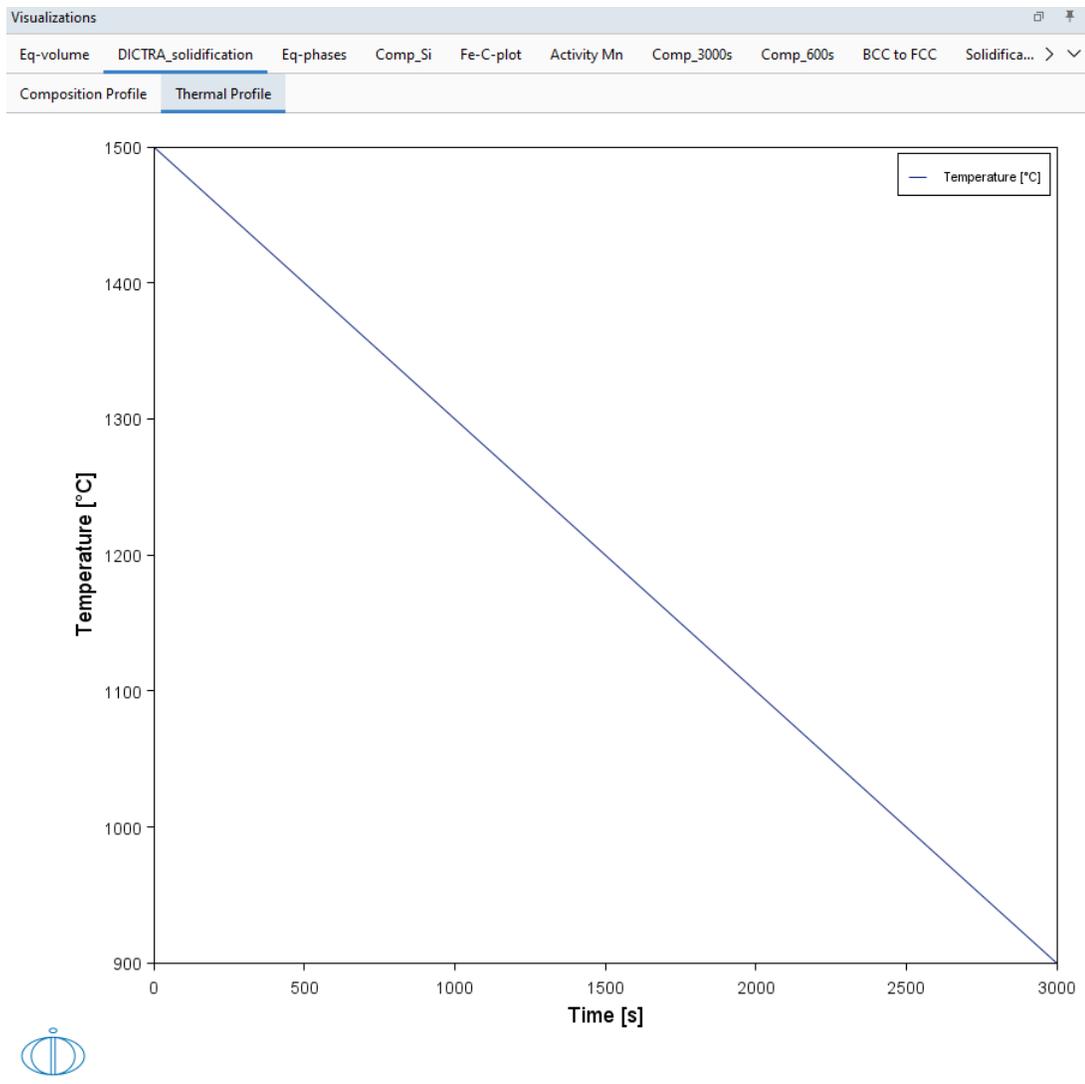


Figure 29: During set up of the calculation, you can preview the Thermal Profile. Click the tab in the Visualizations window to switch between these previews and adjust settings on the Diffusion Calculator Configuration window.



The other resulting plots and details related to setting up this example are available via the [dedicated web page](#), or you can read the more in depth step-by-step documentation [as a PDF](#), also available via our website.

## D\_09: Ni Post Weld Heat Treatment

---

In this example, a Ni-based alloy (Ni-Al-Co-Cr-Mo-Ti) post weld heat treatment analysis is completed. It shows how to import an initial composition profile from a file into the Diffusion Calculator. Then the calculation is run and plotted to show the composition of Al vs distance compared to experimental data from [2017She]. Such simulations can be used to design the heat treatment such that the desired homogenization of the weld is achieved. The initial composition profile needs to be determined by suitable experimental techniques, for example microprobe.

This example highlights the dual purpose of the **Visualizations** window where you can enter or import *Composition profiles* via the Diffusion Calculator **Configuration** window as well as being able to view and analyze results via the Plot Renderer after a calculation runs.

### **Project File Information**

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_09 Ni\_Post\_Weld\_Heat\_Treatment\_From\_File.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

On the Diffusion Calculator, when defining its setup, the **Composition profiles** are imported using the **Table input** option. A prepared file, in this case a csv file, is set up with the correct data columns, which then are included on the **Configuration** window. After import the **Composition Profiles** are further able to be seen via the **Visualizations** window, at which point adjustments can be made as needed. In this isothermal example, you can also see that the constant temperature of the **Thermal Profile** is previewed.

**Configuration**

 **Post-weld heat treatment**

---

**Region**

Name:

Width:   $\mu\text{m}$   points and type

Phase:

Phase:

Phase:

Composition profiles:  Given by function  **Table input**

Data file:

Dependent component:

Composition u... Mole fraction

Length up... Micromet

Distance	Al	Co	Cr	Mo	Ti
0.000000	0.032736	0.098891	0.214996	0.055793	0.033010
1.270410	0.032855	0.097936	0.215575	0.055723	0.032531
2.520830	0.032636	0.098414	0.216068	0.054484	0.030978

Figure 30: The Diffusion Calculator Configuration window showing the composition profile data, which is then compiled and displayed on the Visualizations window.

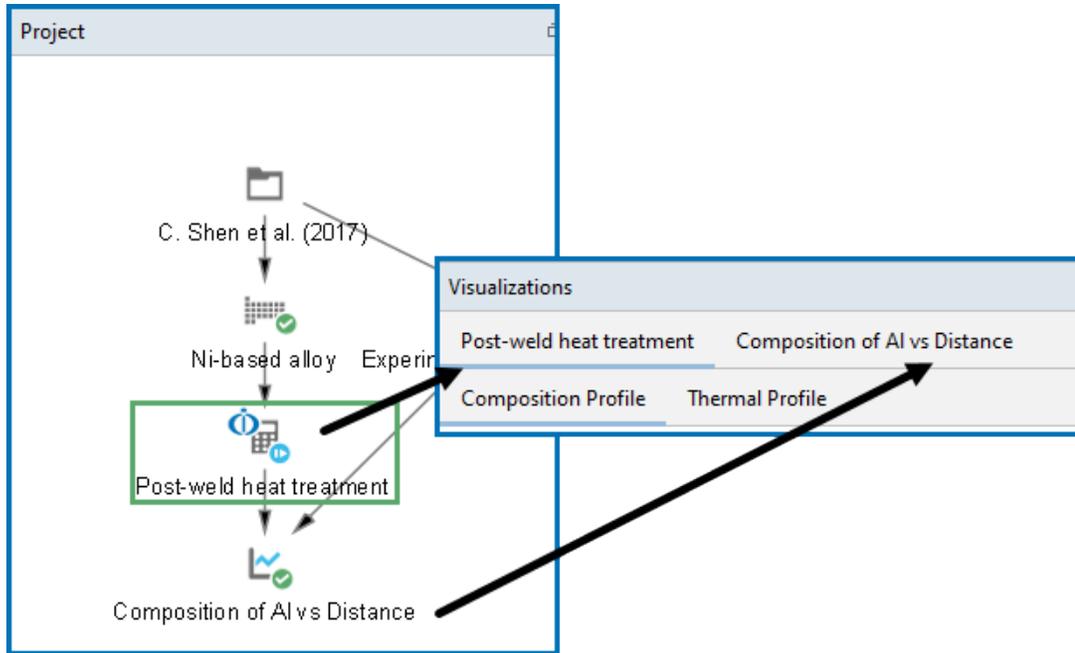


Figure 31: The Visualizations window displays details from both the Diffusion Calculator node (renamed to Post-weld heat treatment) via the Composition Profile and Thermal Profile tabs as well as the Plot Renderer node (renamed to Composition of Al vs Distance).

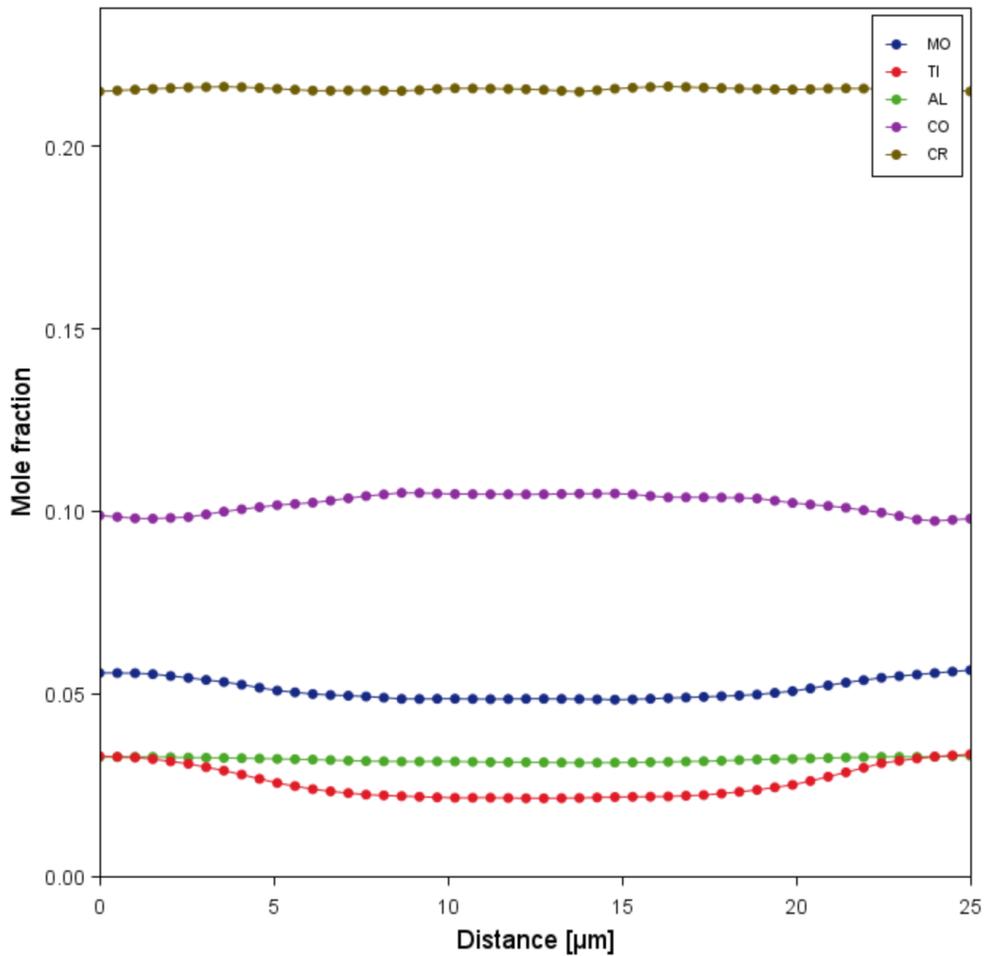


Figure 32: The Composition Profiles as shown in the Visualizations window after data is prepared and then imported into the Diffusion Calculator Configuration window.

An **Experimental File Reader** node is used to supply experimental data from [2017She] to the Plot Renderer to validate the composition of, for example, Al vs distance on the Plot Renderer. The plot shows the content of Al along the cross-section of the weld, both before and after the post weld heat treatment. There is good agreement between the diffusion simulation and the data measured in the literature.

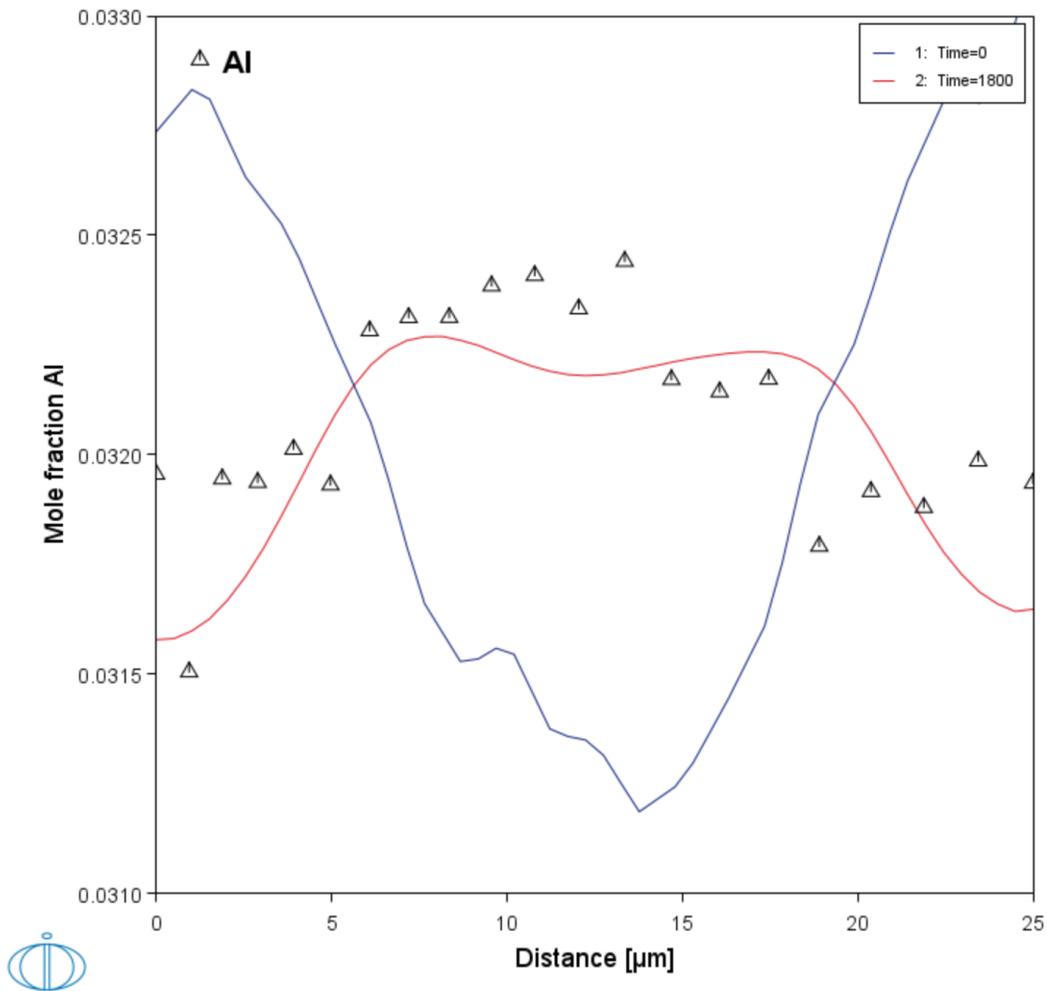


Figure 33: Plot results showing the mole fraction of Al before and after the heat treatment as a function of distance for a Ni-based alloy compared to experimental data [2017She].

Other plots could be added to investigate further details of the simulations, such as phase fractions or compositions.

## Reference

[2017She] C. Shen, V. Gupta, S. Huang, M. Soare, P. Zhao, Y. Wang, "Modeling Long-term Creep Performance for Welded Nickel-base Superalloy Structures for Power Generation Systems" (Pittsburgh, PA, and Morgantown, WV (United States), 2017), <http://www.osti.gov/servlets/purl/1345084/>.

## D\_10: Iron (Fe) Homogenization in Scheil

---

This example models the homogenization of a cast Ni-Cr steel and compares predictions with the experimental data collated by Fuchs and Roósz [1975Fuc]. The conditions studied have a dendritic half spacing of 200  $\mu\text{m}$  and are heat treated to temperatures of 1120 °C and 1235 °C. The data has been un-normalized to allow comparison using wt.% composition values.

The composition of the steel is 0.4C, 0.65Mn, 0.35Si, 0.015S, 0.01P, 1.9Ni, 0.95Cr, and 0.3 Mo (wt.%). The diffusion of Cr and Ni are modeled using a simplified chemistry of 0.4C, 0.65Mn, 1.9Ni, and 0.95Cr.



The FEDEMO thermodynamic and MFEDEMO mobility databases are used with this example. Users with a Diffusion Module (DICTRA) license can run the example. Other users can read the documentation as a DEMO license only allows the use of up to three components.

### Project File Information

- Folder: **Diffusion Module - DICTRA**
- File name: *D\_10\_Fe\_Homogenization\_from\_Scheil.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).



The Diffusion Module (DICTRA) Console Mode example `exa7` is available to show the use of Scheil segregation profiles in Console Mode.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Composition Profiles** and **Thermal Profile**: When setting up a calculator on a **Configuration** window you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the

**Visualizations window.**

- **Plot** or **Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

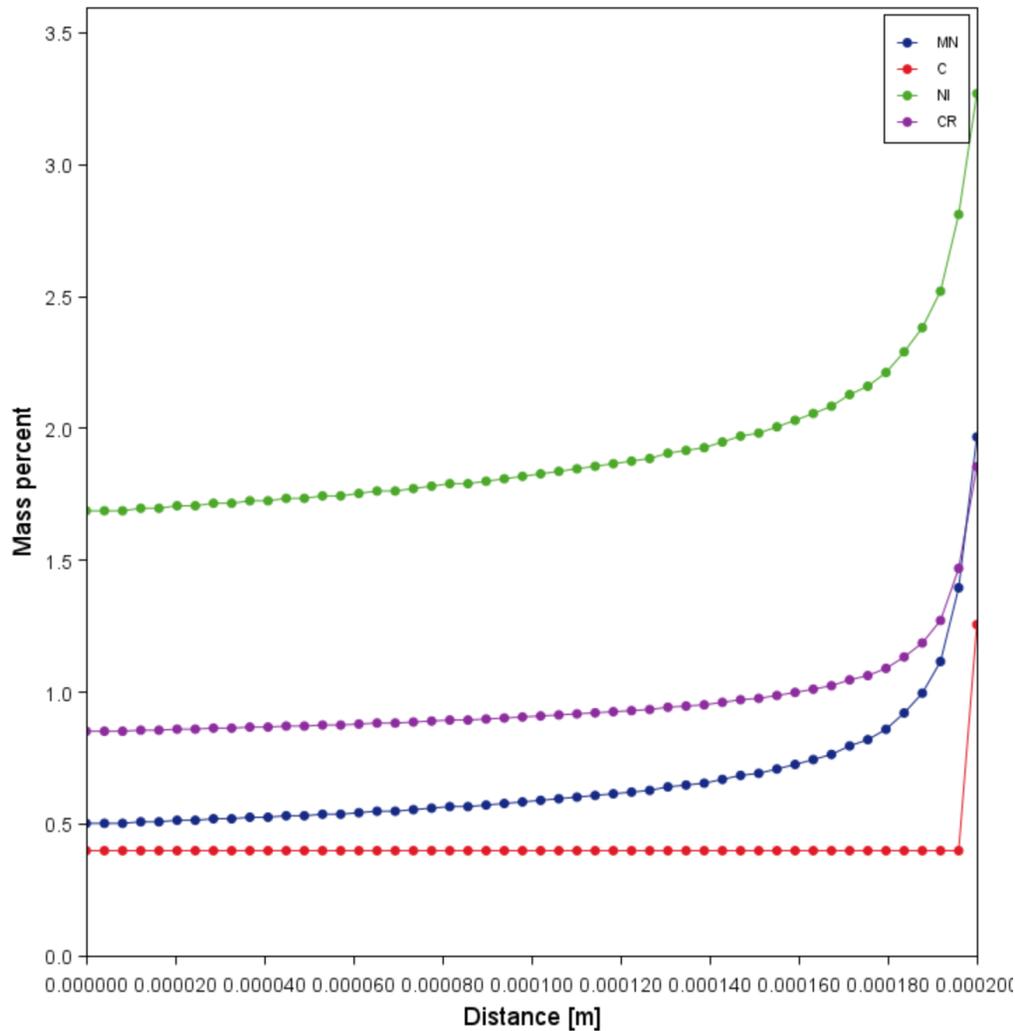
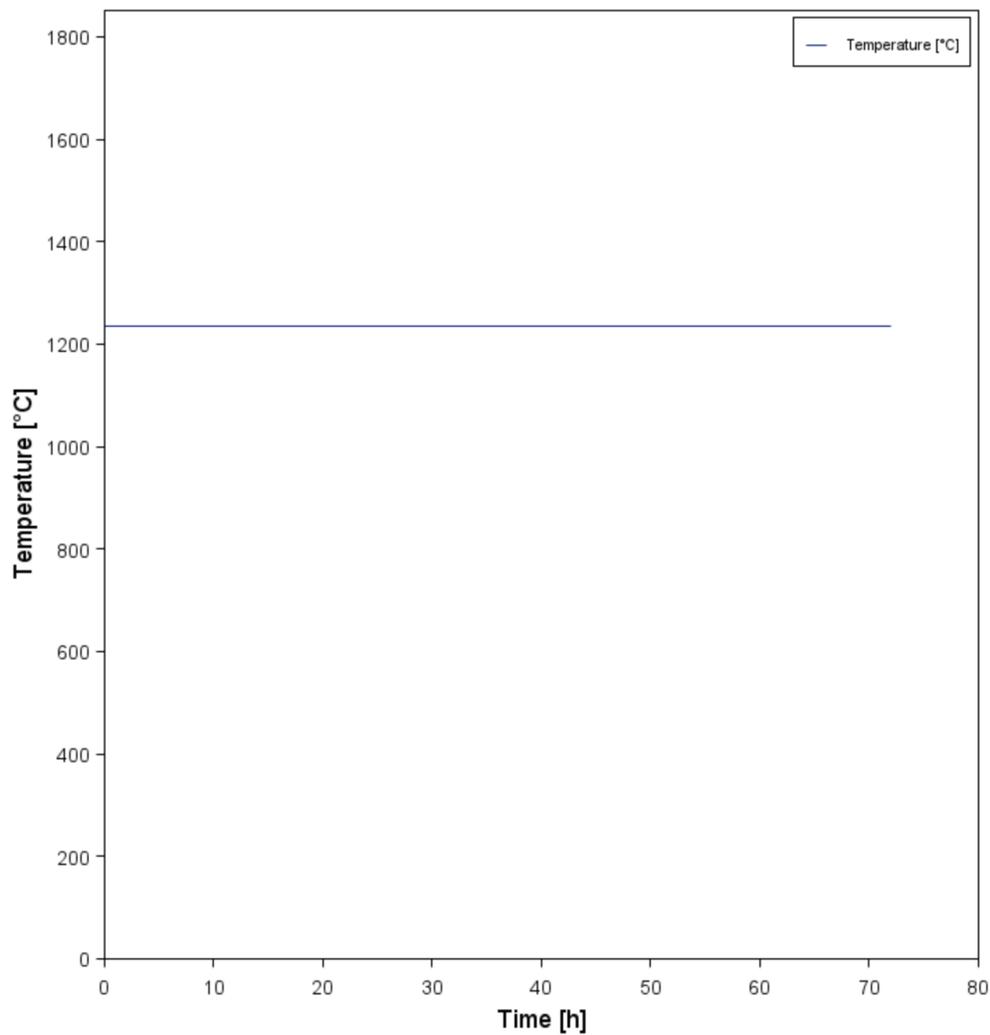


Figure 34: The predicted chemical segregation using the Scheil calculation, with a dendritic half spacing of 200  $\mu\text{m}$ . This is the Composition Profile that is set up on the Diffusion Calculator and then previewed in the Visualizations window.



*Figure 35: An example of the modeled isothermal heat treatments. The homogenization kinetics at 1235 °C and 1120 °C have been modeled for 72 hours. This is the Thermal Profile that is set up on the Diffusion Calculator and then previewed in the Visualizations window.*

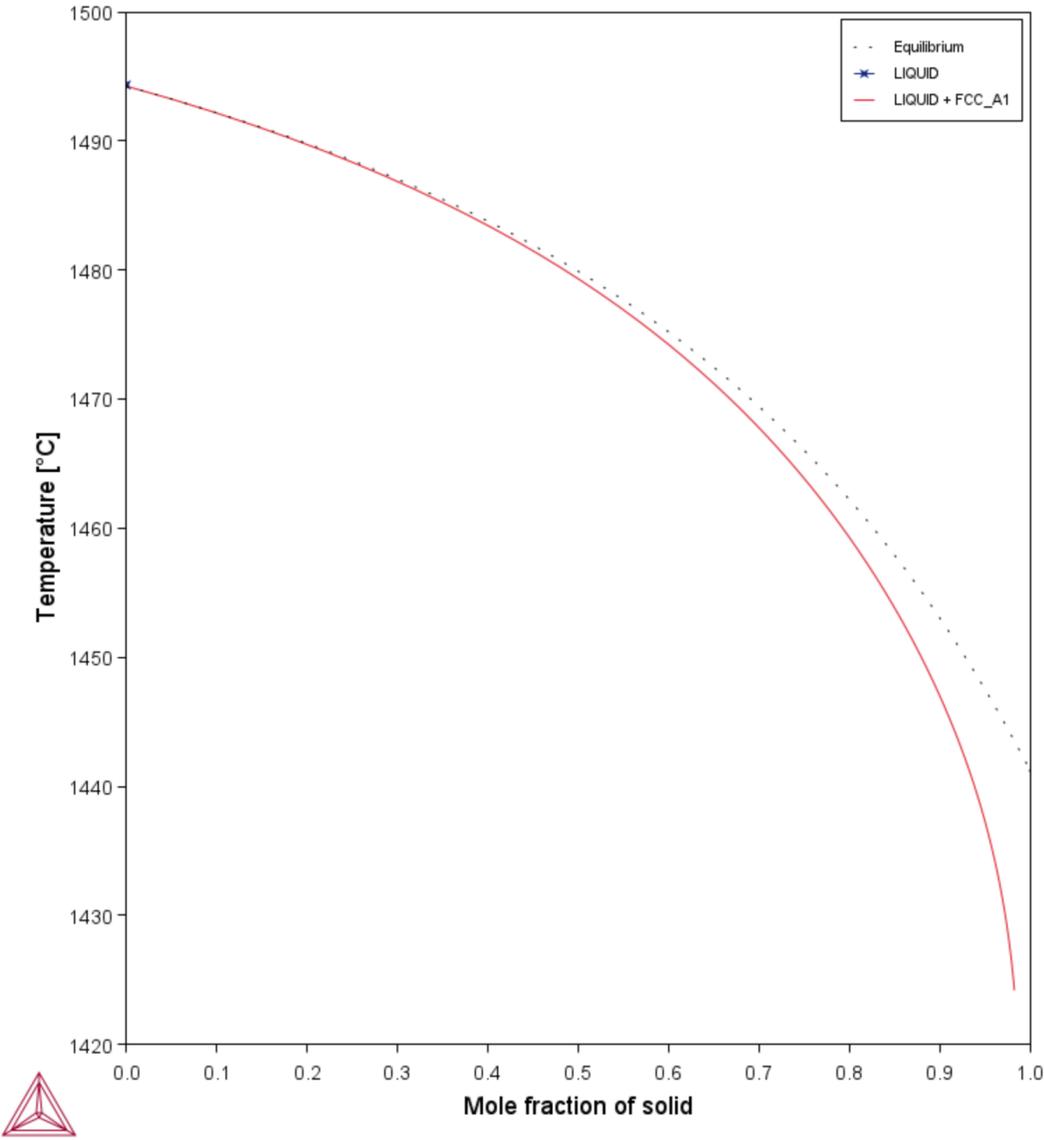


Figure 36: The solidification behavior predicted from the Scheil calculation compared with the equilibrium solid fraction.

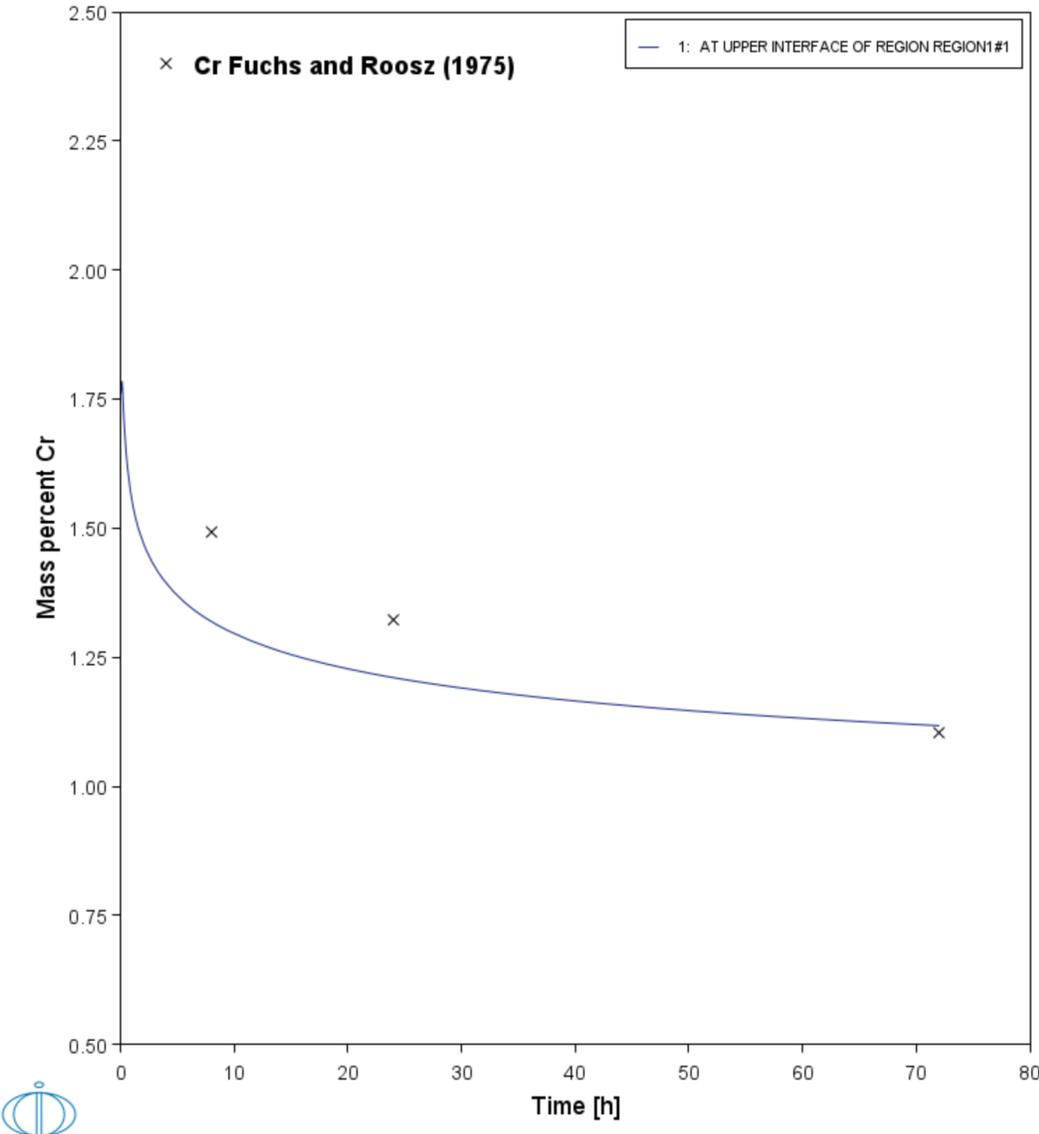


Figure 37: A comparison of the predicted and measured maximum Cr across the dendrites during homogenization at 1120 °C.

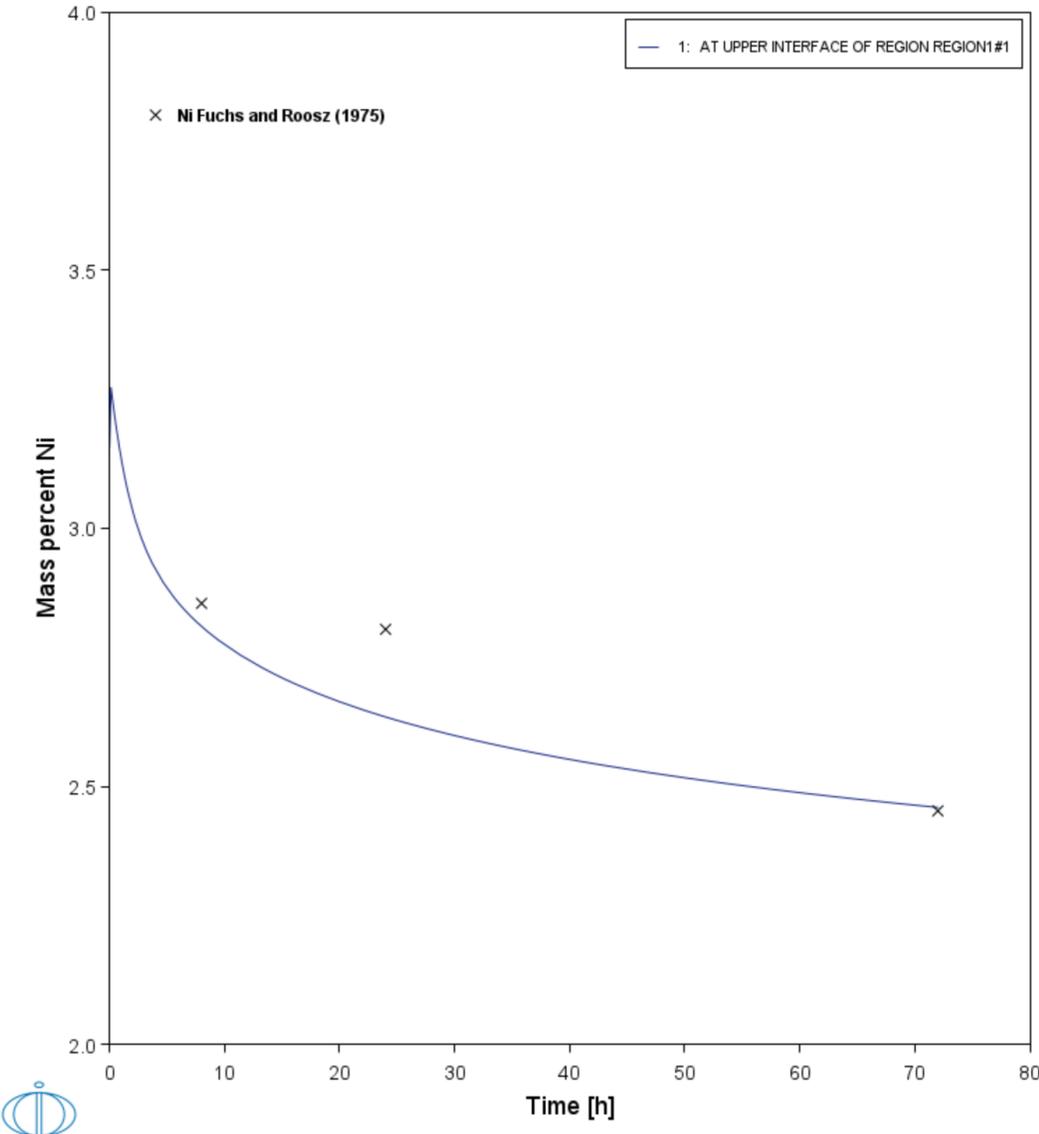


Figure 38: A comparison of the predicted and measured maximum Ni across the dendrites during homogenization at 1120 °C.

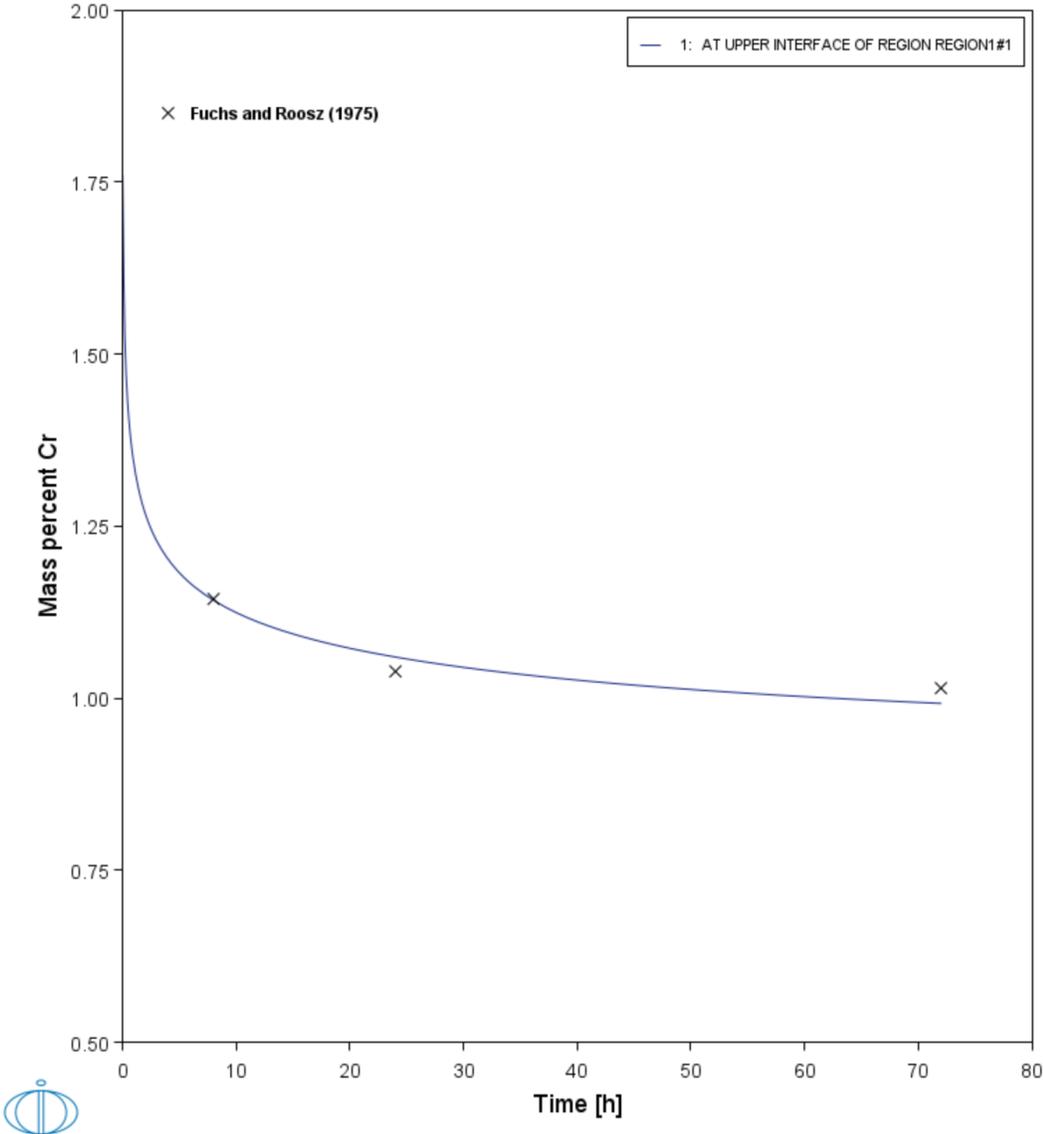


Figure 39: A comparison of the predicted and measured maximum Cr across the dendrites during homogenization at 1235 °C.

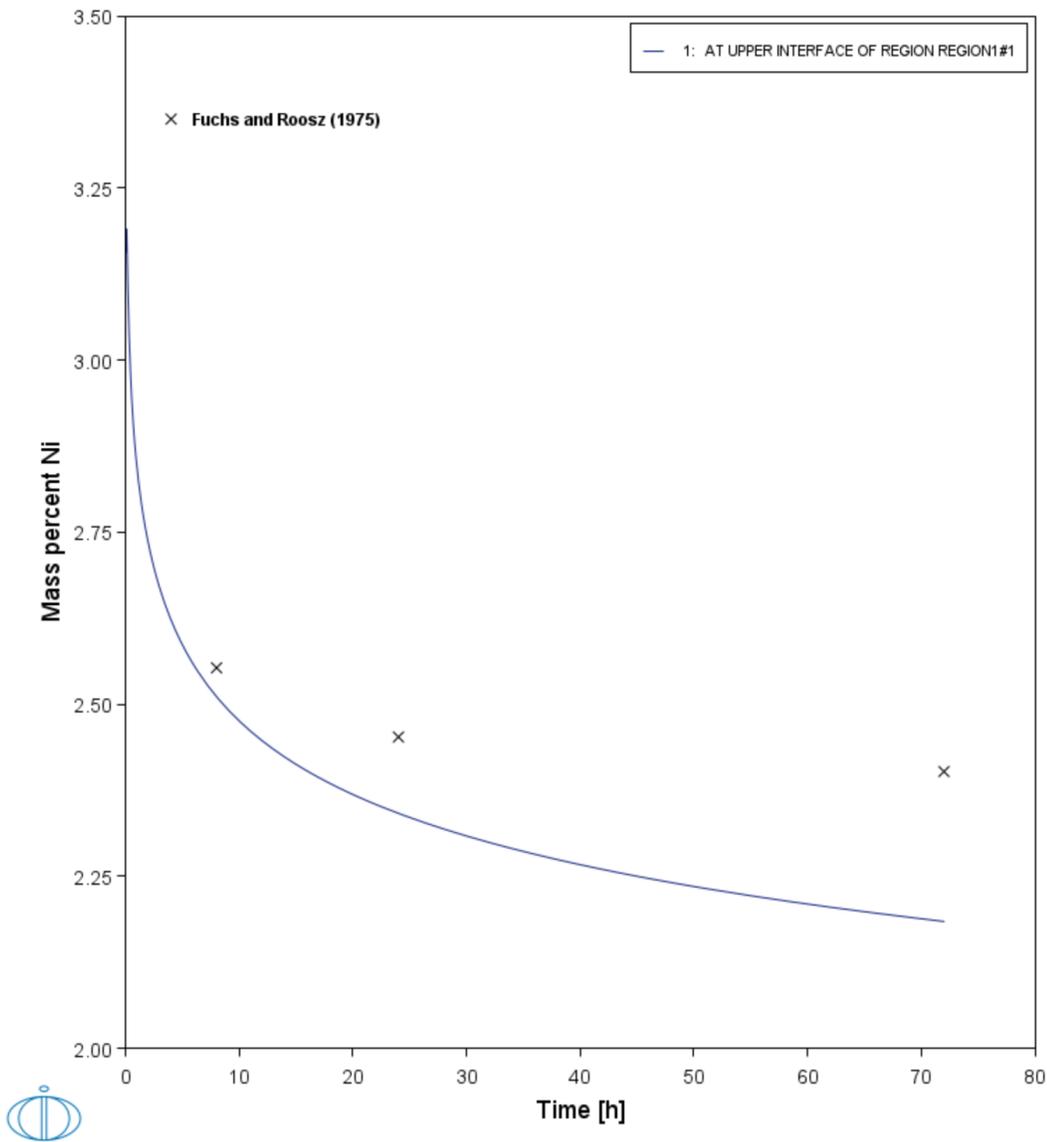


Figure 40: A comparison of the predicted and measured maximum Ni across the dendrites during homogenization at 1235 °C.

## Reference

[1975Fuc] E. G. Fuchs, A. Roósz, Homogenization of Iron-Base Cast Alloys. *Met. Sci.* 9, 111–118 (1975).

# Precipitation Module (TC-PRISMA) Examples Collection

---



Examples that use up to three elements are available to all users. Other examples require a Precipitation Module (TC-PRISMA) license to calculate and plot results.



When an example uses a demonstration (demo) database package, this is included with your installation.



Unless specified in tables for each example, all the numerical parameters are assumed default values.

## In this section:

P_01: Isothermal Precipitation of Al3Sc .....	223
P_02: Stable and Metastable Carbides - Isothermal .....	237
P_03: Stable and Metastable Carbides - TTT Diagram .....	241
P_04: Precipitation of Iron Carbon Cementite .....	245
P_05: Precipitation of $\gamma'$ in Ni Superalloys - Isothermal .....	248
P_06: Precipitation of $\gamma'$ in Ni Superalloys - Non-isothermal .....	253
P_07: Continuous Cooling Transformation (CCT) Diagram of Ni-Al-Cr $\gamma$ - $\gamma'$ .....	261
P_08: Precipitation of Cu-Ti CU4TI with Assumptions of Sphere and Needle Morphologies .....	270
P_09: Precipitation of Al-Sc AL3SC with Assumption of Sphere and Cuboid Morphologies .....	278
P_10: Initial Particle Size Distribution of Fe-Cr-C .....	294
P_11: Interfacial Energy Function .....	299
P_12: Comparing Growth Rate Models for an Al-Zr System .....	303
P_13: Paraequilibrium Precipitation of Cementite Fe-C-Cr .....	307

---

---

P_14: Grain Growth and the Zener Pinning Effect .....	311
P_15: Smooth Transition from Paraequilibrium to Ortho-equilibrium .....	316
P_16 Isothermal Coarsening and a 3D to 2D Stereological Conversion .....	321

---

## P\_01: Isothermal Precipitation of Al<sub>3</sub>Sc

This example simulates the kinetics of precipitation of Al<sub>3</sub>Sc from an FCC\_A1 solution phase. The simulation results can be compared with experimental data collected from Marquis and Seidman [2001Mar] and Novotny and Ardell [2001Nov].

This example also includes a plot using the **Yield strength** Property Model. This demonstrates how you can use the results from a Precipitation Module (TC-PRISMA) simulation as input to the Yield Strength Model, i.e. the calculated precipitate radius/radii for each time step is used to calculate the precipitation strengthening, and similarly, the matrix composition for each time step is used to calculate the solid solution strengthening when this is selected in the **Configuration** on the Plot Renderer. The experimental data for the Yield Strength Model is from Seidman et al. [2002Sei]. In this example, the *Precipitation strengthening model* used is **Seidman model (Al-base)**. This is selected on the Plot Renderer configuration panel that is connected to the Property Model.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File, Step-By Step Instructions, and Video Tutorial Information**

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_01\_Precipitation\_Al-Sc\_AL3SC.tcu*



This example is included as a Precipitation Module (TC-PRISMA) tutorial on our [website](#) and as part of the playlist on our [YouTube channel](#).



You can also use [the step-by-step instructions](#) included in a PDF to follow the video or compare to the project file in Thermo-Calc.

### **Example Settings**

System (System Definer)	
Database package	Demo: Aluminum-based alloys (ALDEMO, MALDEMO)
Elements	Al, Sc
Conditions (Precipitation Calculator)	
Composition	Al-0.18Sc Mole percent
Matrix phase	FCC_A1

Precipitate phase	AL3SC
<b>Precipitate Phase Data Parameters (Precipitation Calculator)</b>	
Nucleation sites	Bulk
Interfacial energy	Calculated
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal
Temperature	350° C
Simulation time	1.0E7 seconds
<b>Experimental File Reader 1 and 2</b>	
There are two Experimental File Reader nodes used. One for the mean radius plot and one to demonstrate the <i>Yield Strength Property Model</i> .	

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

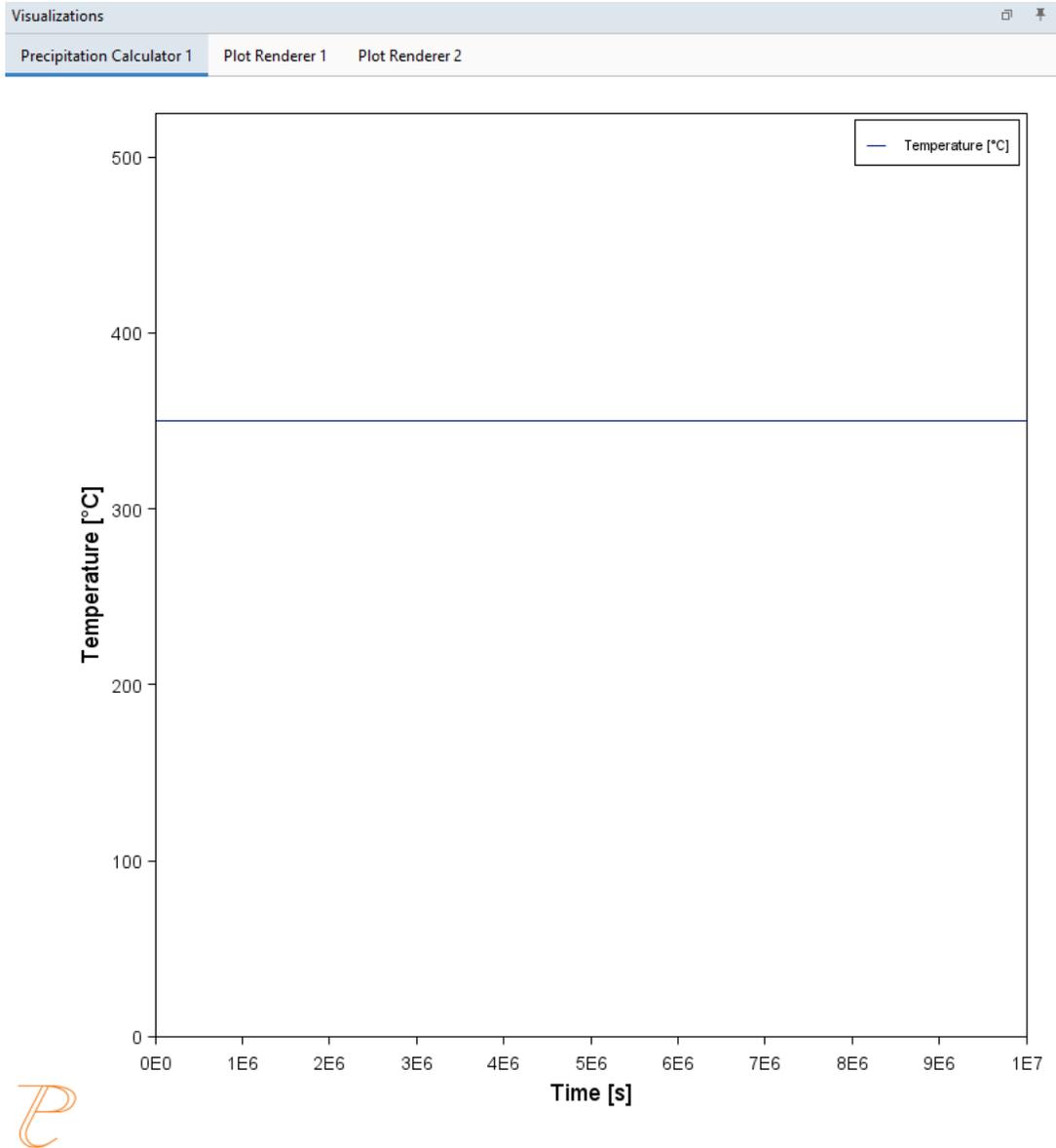
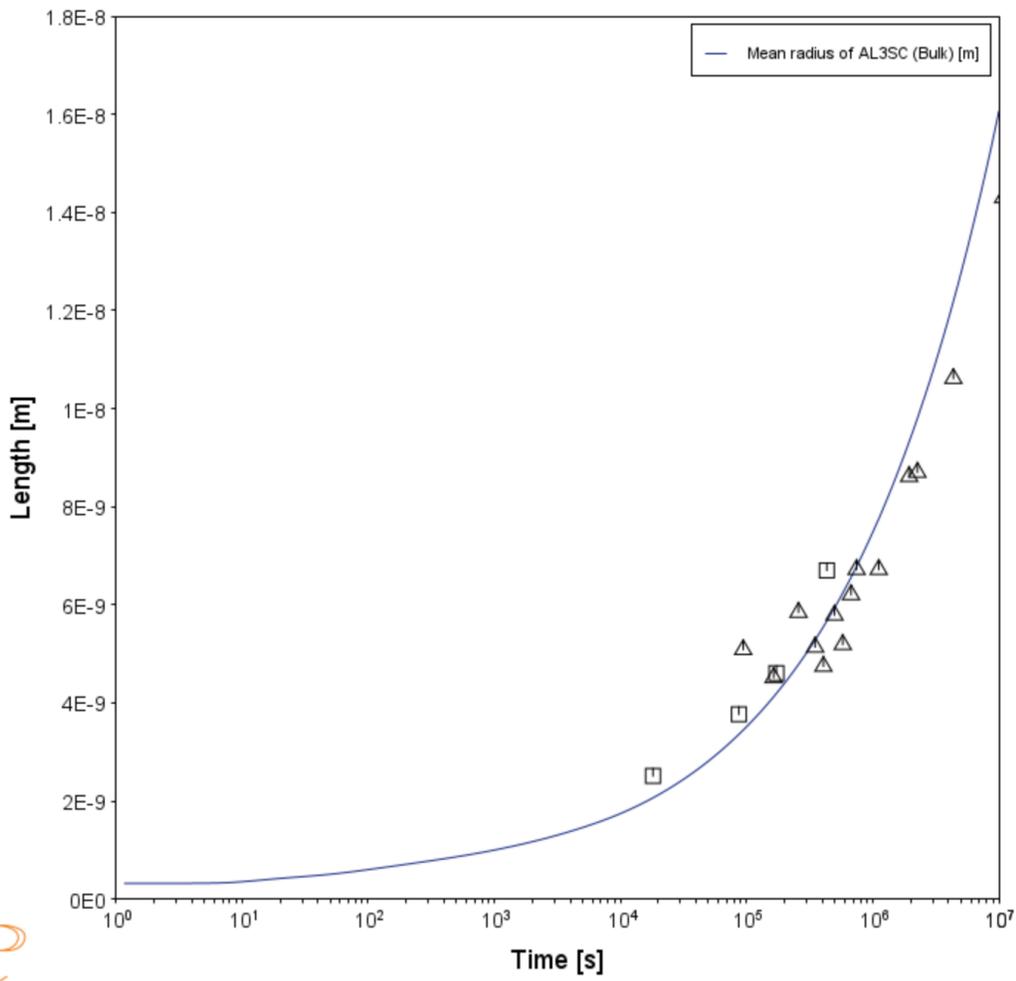


Figure 1: During set up of the calculation, you can preview the Thermal Profile. Click the tab in the Visualizations window to preview it and adjust settings on the Precipitation Calculator Configuration window. For an Isothermal Thermal Profile this shows the constant temperature as entered.



P

Figure 2: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. This plot shows the mean radius of the AL3SC precipitate as a function of time.

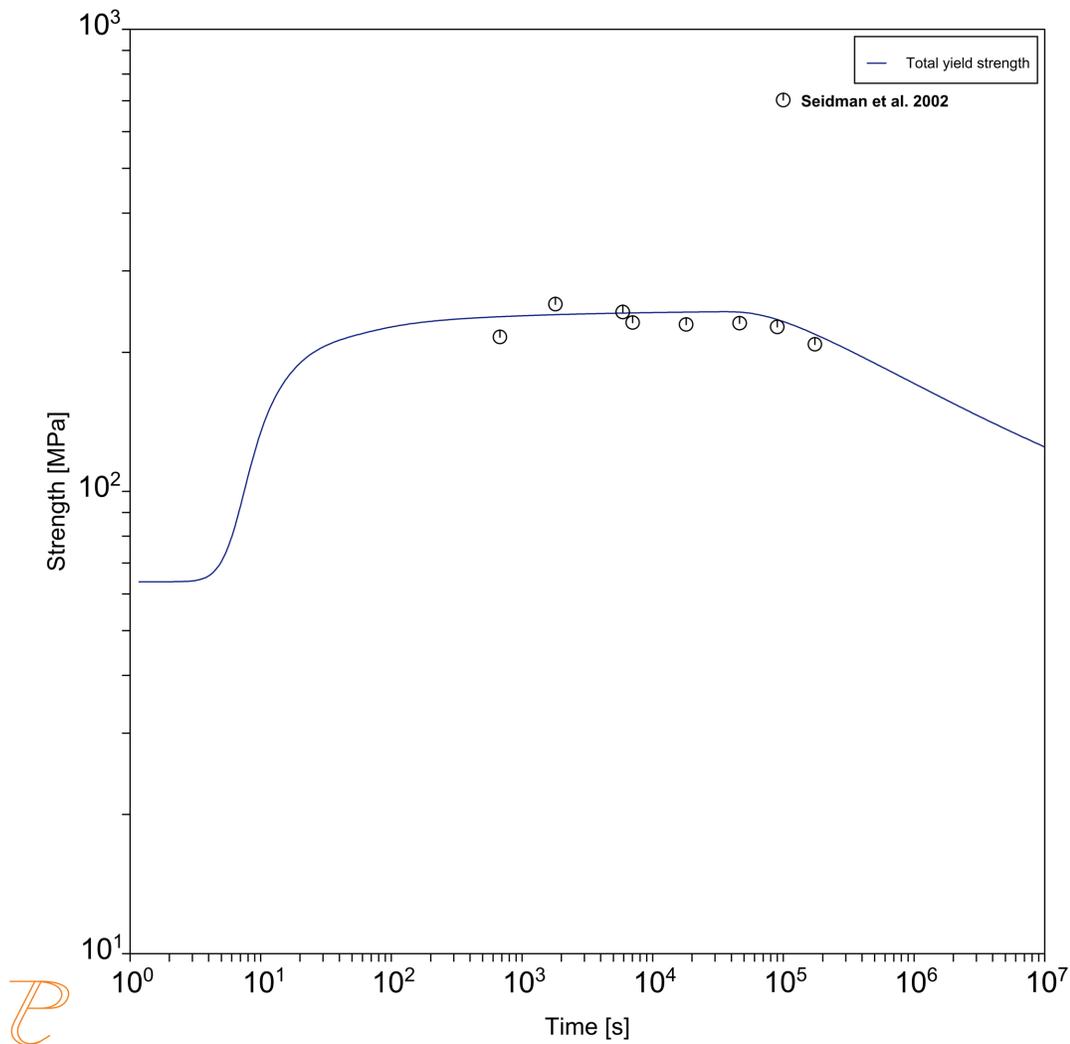


Figure 3: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. This plot shows the total yield strength of the  $Al_3Sc$  precipitate as a function of time compared to experimental data from Seidman et al. [2002Sei]. The model captures the over-tempering behavior from excessive coarsening of the strengthening precipitates.

## References

- [2001Mar] E. A. Marquis, D. N. Seidman, Nanoscale structural evolution of  $Al_3Sc$  precipitates in  $Al(Sc)$  alloys. *Acta Mater.* 49, 1909–1919 (2001).
- [2001Nov] G. M. Novotny, A. J. Ardell, Precipitation of  $Al_3Sc$  in binary  $Al-Sc$  alloys. *Mater. Sci. Eng. A Struct. Mater. Prop. Microstruct. Process.* 318, 144–154 (2001).
- [2002Sei] D. N. Seidman, E. A. Marquis, D. C. Dunand, Precipitation strengthening at ambient and elevated temperatures of heat-treatable  $Al(Sc)$  alloys. *Acta Mater.* 50, 4021–4035 (2002).

# Isothermal Precipitation Calculation: Example P\_01 – Precipitation Al-Sc AL3SC

This example shows you how to set up an isothermal precipitation calculation for the formation of AL<sub>3</sub>SC in an aluminium-scandium alloy at 350° using the Precipitation Module (TC-PRISMA).

The end of the example also shows how to use the results of the calculation to model Yield Strength.

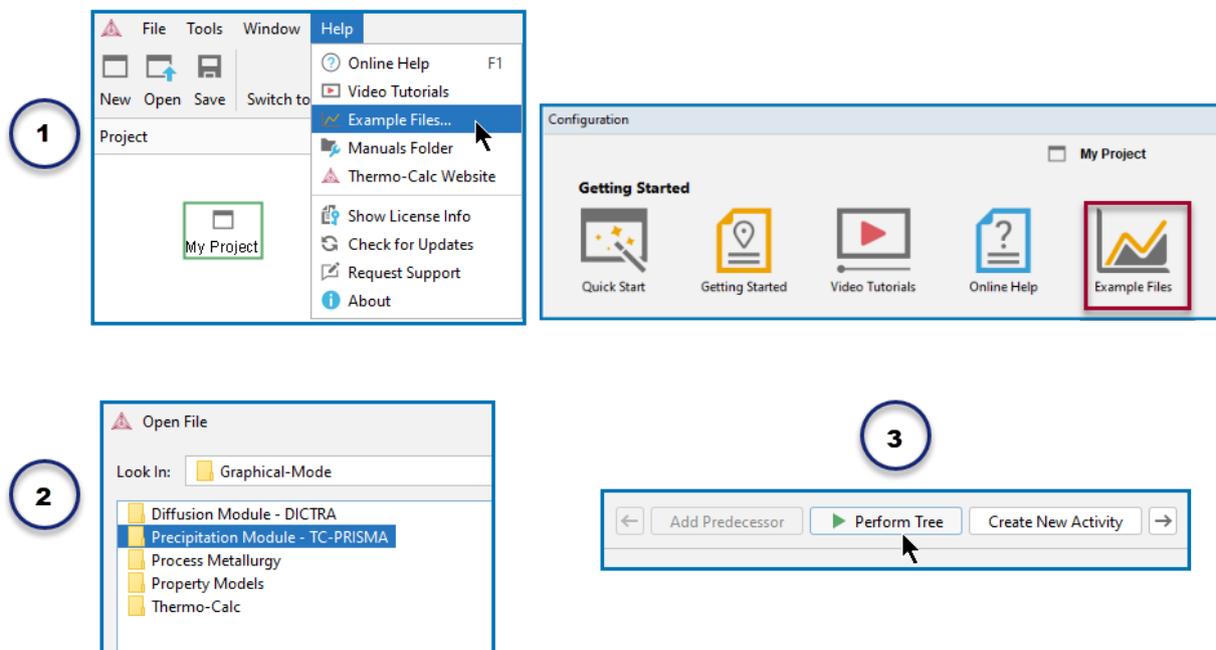
This is one of the most basic calculations using the Precipitation Module (TC-PRISMA), so it is a good place to start if you are new to this simulation type.

## HELPFUL INFORMATION

- **All users can run this calculation**, even those who do not have a license for the Precipitation Module (TC-PRISMA).
- **A companion video** is available for this example , which can be watched here:  
[https://www.youtube.com/playlist?list=PLfv6McToaTGSpqvUoY3b\\_UV-8xpgLUKj](https://www.youtube.com/playlist?list=PLfv6McToaTGSpqvUoY3b_UV-8xpgLUKj)

This calculation is based on Precipitation Module example *P\_01 – Precipitation Al-Sc AL3SC*, which is included in your installation.

1. To run the example file, open Thermo-Calc and select **Help > Examples Files** or click **Example Files** on the **My Project Configuration** window.
2. Open the *Precipitation Module (TC-PRISMA)* folder. Double-click the example file to open it.
3. In Thermo-Calc at the bottom center of the **Configuration** window click **Perform Tree**.

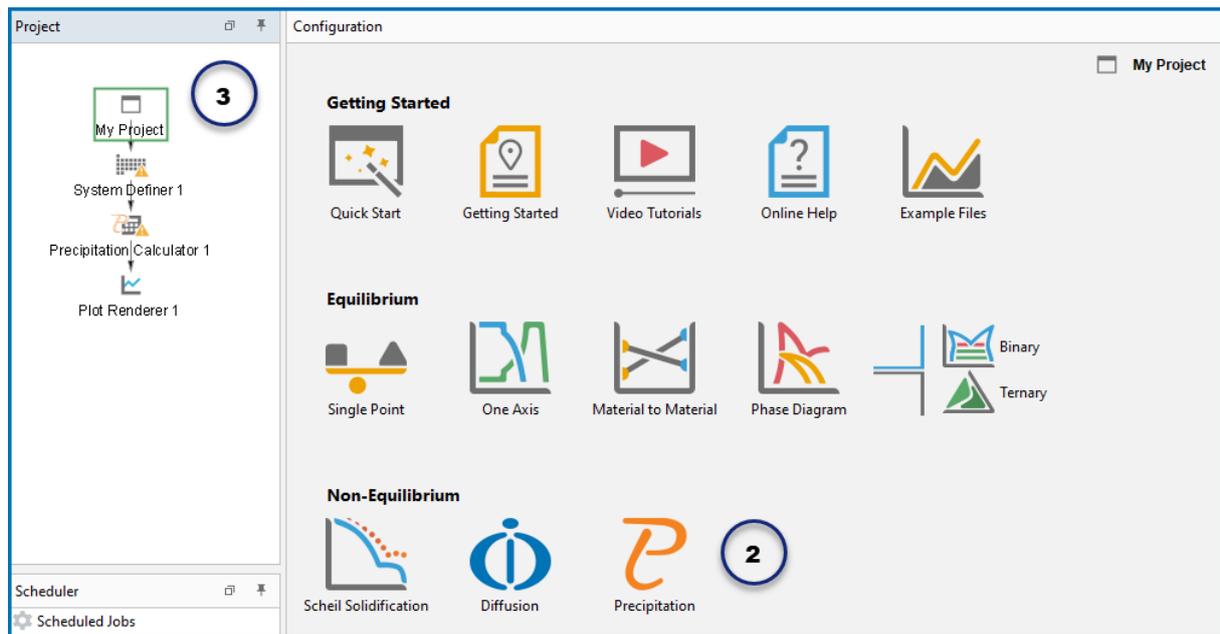


## ABOUT THE EXAMPLE

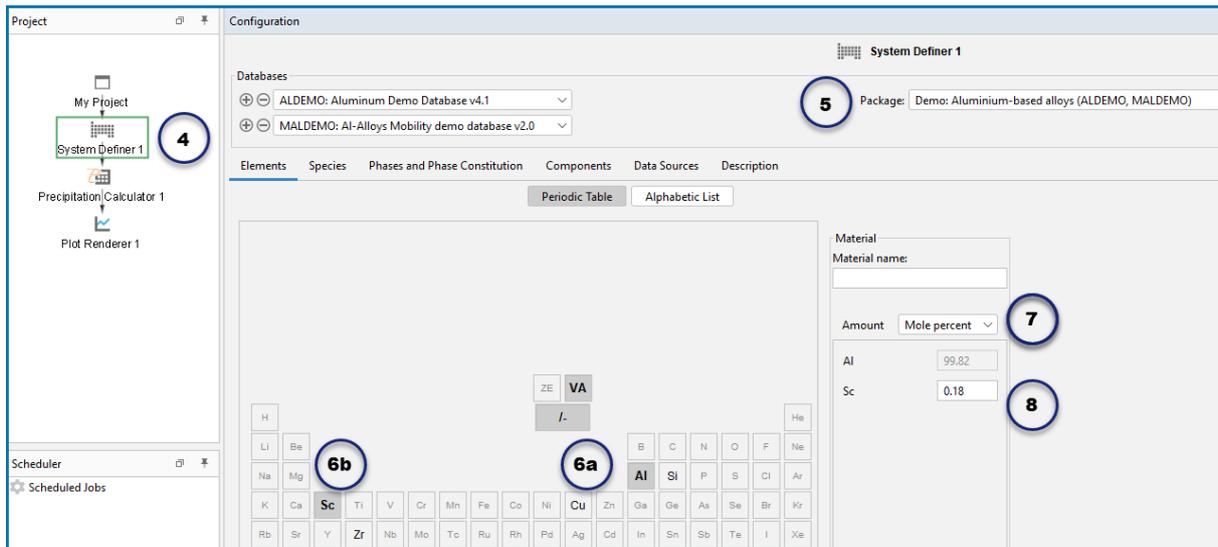
This example simulates the kinetics of precipitation of  $\text{Al}_3\text{Sc}$  from an FCC\_A1 solution phase. The results of the simulation are also used to model yield strength.

## SETTING UP THE SYSTEM

1. Open Thermo-Calc in Graphical Mode.
2. On the **My Project Configuration** window under **Non-Equilibrium**, click **Precipitation**.
3. All the nodes for a precipitation calculation are added to the **Project** window:



4. In the **Project** window, click the **System Definer 1** node.
5. On the **Configuration** window from the *Package* list, select **Demo: Aluminium-based alloys (ALDEMO, MALDEMO)**. This loads both thermodynamic and kinetic demonstration aluminium databases.
6. In the **Periodic Table**, select the elements in the following order.  
Select Al first so that it is the dependant element:
  - a. **Al** (aluminium)
  - b. **Sc** (scandium).
7. From the *Amount* list (to the right of the Periodic Table), select **Mole percent**.
8. Enter 0.18 for **Sc**. This automatically sets **Al** to 99.82.



The system is now defined. However, before starting the precipitation calculation, it is recommended to run a one axis calculation to find the phases present around 350° C.

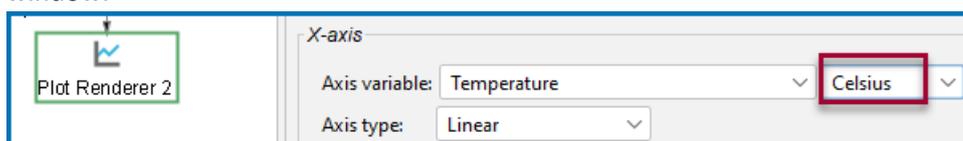
## ONE AXIS CALCULATION

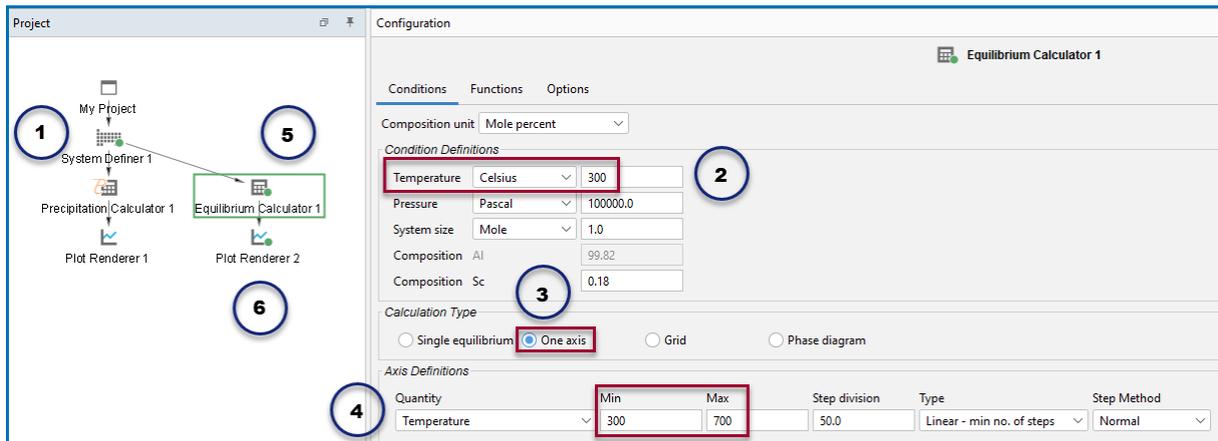
### Why We Do This

In order to set up an isothermal precipitation calculation, you need to know which phases are present at 350 °C. A *one axis* calculation provides this information. Note that this step is not included with the P\_01 example. This is an additional exercise.

### Setting up a One Axis Calculation

1. In the **Project** window, right-click the **System Definer 1** node, and select **Create New Successor > Equilibrium Calculator**.
2. In the **Configuration** window, set *Temperature* to **Celsius** and enter 300.
3. Under *Calculation Type*, click **One axis**.  
You now want to find the phases present around 350 °C.
4. Under *Axis Definitions*, enter the **Temperature** range from **Min** 300 to **Max** 700.  
Keep the default values for everything else. The system is now defined.
5. Right-click the **Equilibrium Calculator** node and select **Create New Successor > Plot Renderer**.
6. In the **Project** window, click the **Plot Renderer 2** node you just created. In the **Configuration** window, set the *X-axis variable* for **Temperature** to **Celsius**. Then right-click the **Plot Renderer 2** node select **Perform Now**. Or click **Perform** at the bottom of the **Configuration** window.





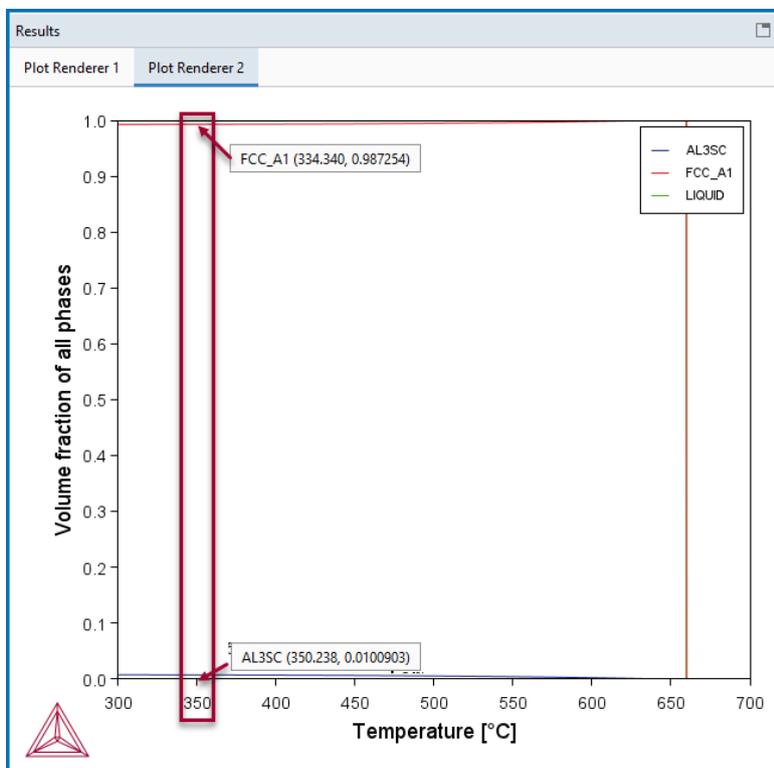
### Interpreting the Results of the One Axis Calculation

Once the calculation is complete, a plot is displayed in the **Results** window. This plot shows you which phases are present at each temperature between 300° and 700° and the amount of that phase at each temperature.

If you hover your cursor over any of the lines on the plot, a label gives you the name of the phase, the temperature and the amount of the phase at that temperature.

We can see that there are two phases present at 350°. **FCC\_A1** is at the very top of the plot and makes up more than 99% of the system. The remainder of the system is made up of **AL<sub>3</sub>SC**, which is near the very bottom of the plot and is the precipitate phase in the alloy we are considering.

Remember these two phases because these are used in the precipitation calculation.



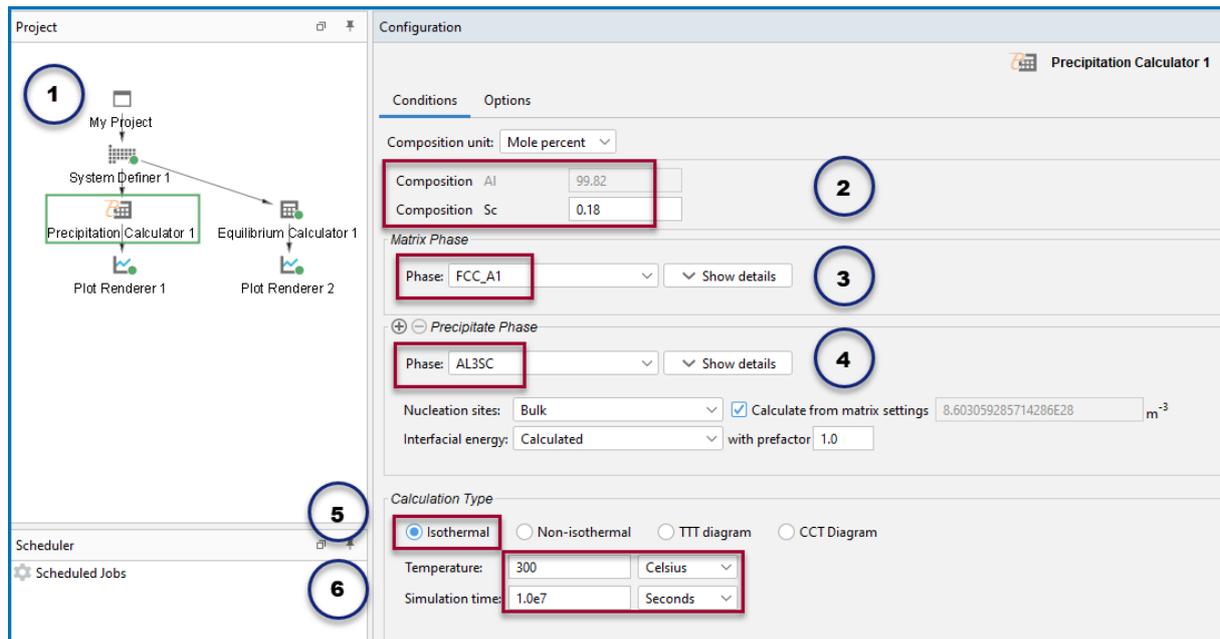
## PRECIPITATION CALCULATION

### Setting up the Precipitation Calculation

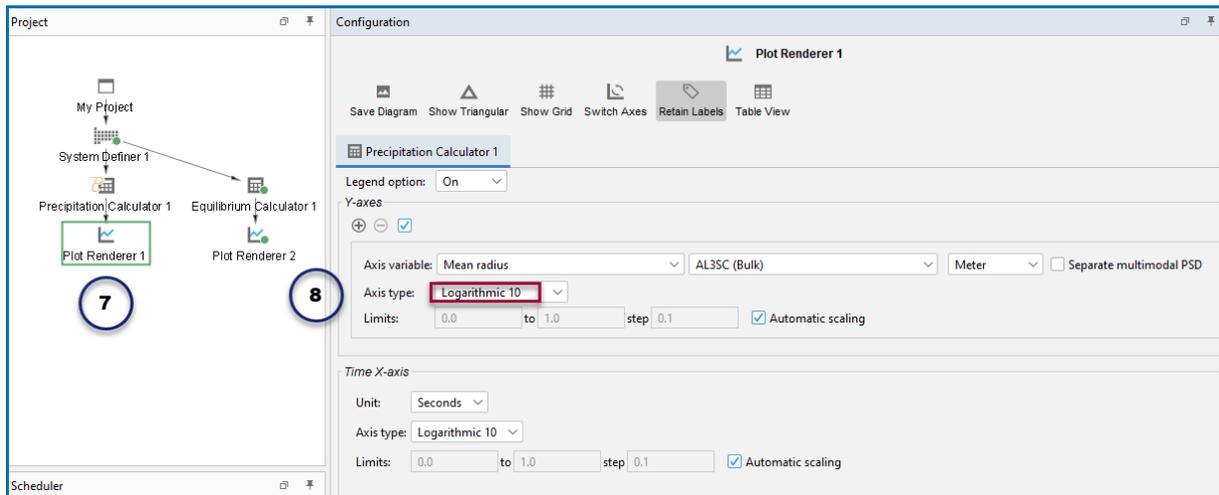
1. In the **Project** window, click the **Precipitation Calculator 1** node.
2. Notice that the *Composition* set in the *System Definer* auto-populated here.
3. Under *Matrix Phase*, the *Phase* defaults to **FCC\_A1**, which is the primary phase present in the one axis calculation.

NOTE: If you do not see any phases listed here it is probably because you do not have a kinetic database selected. If so, go back to the *System Definer* and add a kinetic (mobility) database. In this example it should be the **MALDEMO** database.

4. Under *Precipitate Phase*, select **AL3SC** from the *Phase* list because that is the other phase present around 350 °C, as shown in the one axis calculation.
5. Accept the rest of the default settings in the *Precipitate Phase* section. You could choose to define your own interfacial energy settings if you have them.
6. Under *Calculation Type*, make sure **Isothermal** is selected.
7. Enter:
  - a. 350 as the *Temperature* and select **Celsius**.
  - b.  $1.0 \times 10^7$  as the *Simulation time* and select **Seconds**, which is ten million seconds.

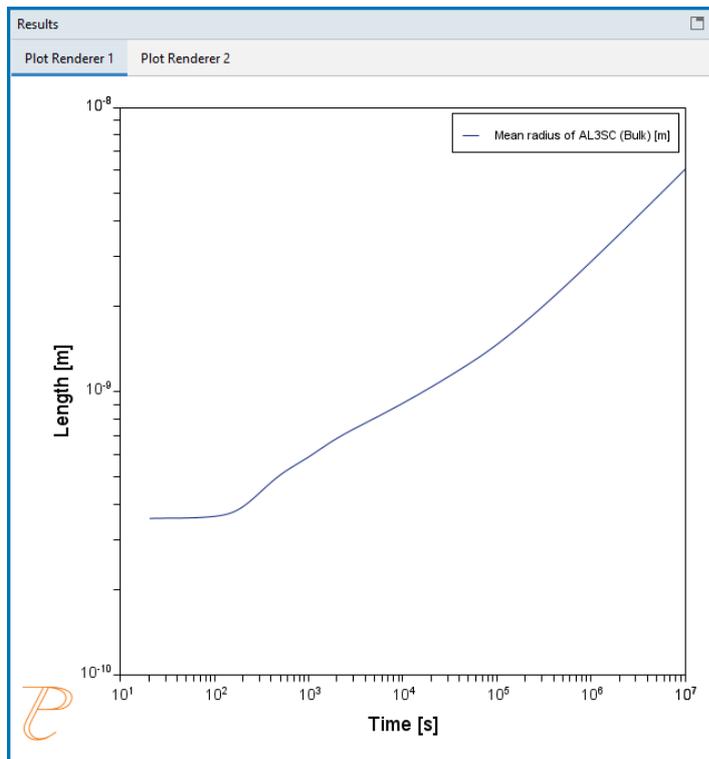


8. In the **Project** window, click the **Plot Renderer 1** node.
9. Under *Y-axes*, from the *Axis type* menu select **Logarithmic 10** for a better view of the plot.
10. Click **Perform** at the bottom, center of the **Configuration** window.



### Interpreting the Results of the Precipitation Calculation

Once the calculation is complete, your plot is shown in the **Results** window.



*This plot shows the Mean radius of the AL3SC precipitate as a function of time.*

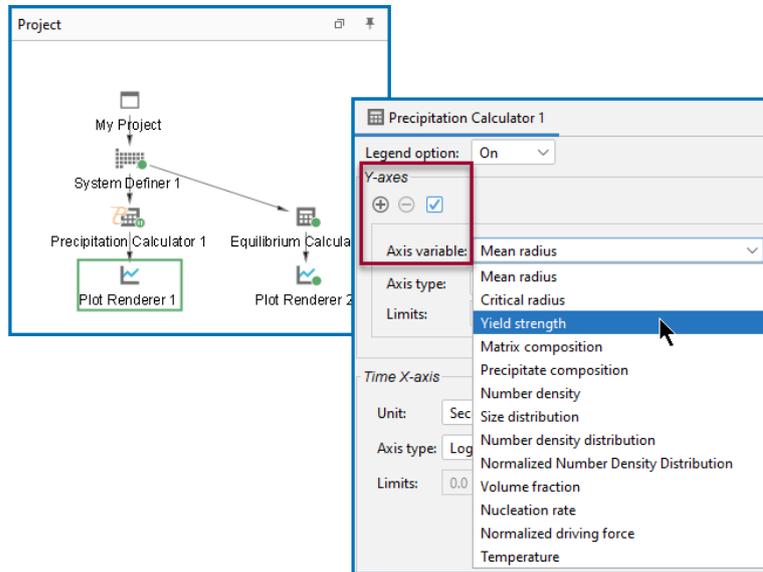
At the beginning, which is the left-most side of the plot, the mean radius is quite small and then grows rapidly over time, which is represented by moving rightward on the plot.

### Setting Other Variables in the Precipitation Module (TC-PRISMA)

The Precipitation Module (TC-PRISMA) offers many variables that you can plot for the same calculation.

1. Click the **Plot Renderer 1** node.
2. Under *Y-axes*, click the *Axis variable* menu to see the other available options in the list.

- Once you have made your selection, click **Perform** at the bottom, center of the **Configuration** window to create a new plot.

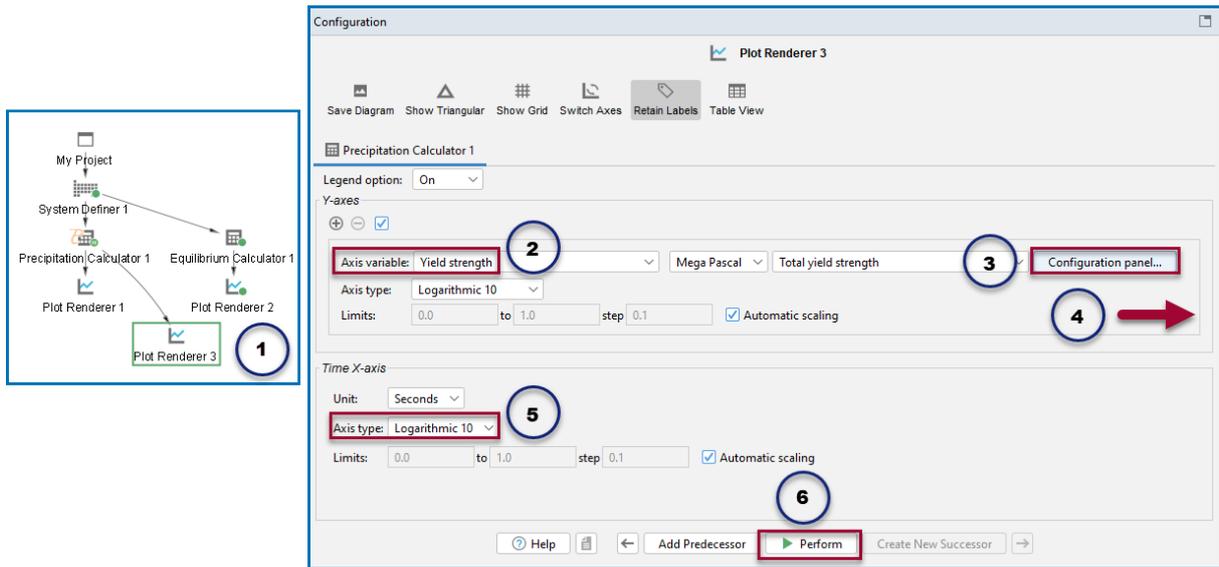


### **Model Yield Strength**

Beginning with Thermo-Calc 2020b, results of a precipitation simulation can be used as input to model yield strength using the Yield Strength Property Model. The following simulation requires a license for Thermo-Calc 2020b or newer.

- Right click the **Precipitation Calculator 1** node and select **Create New Successor > Plot Renderer**.
- Under **Y-axis**, from the **Axis variable** menu, select **Yield strength** (as in the figure above but for **Plot Renderer 3**).
- Click the *Configuration panel* button.
- If required, expand the **Plot Renderer Configuration** window. Then from the *Precipitation strengthening model* menu select **Seidman model (Al-base)**.
- Under **Y-axis**, from the **Axis type** menu, select **Logarithmic 10**.
- Click **Perform** at the bottom, center of the program.

The program uses the results from the precipitation simulation as the input for the Yield Strength model.



Configuration

Plot Renderer 3

Save Diagram Show Triangular Show Grid Switch Axes Retain Labels Table View

Precipitation Calculator 1

Legend option: On

Y-axes

Axis variable: Yield strength 2 Mega Pascal Total yield strength 3 Configuration panel... 4

Axis type: Logarithmic 10

Limits: 0.0 to 1.0 step 0.1 Automatic scaling

Time X-axis

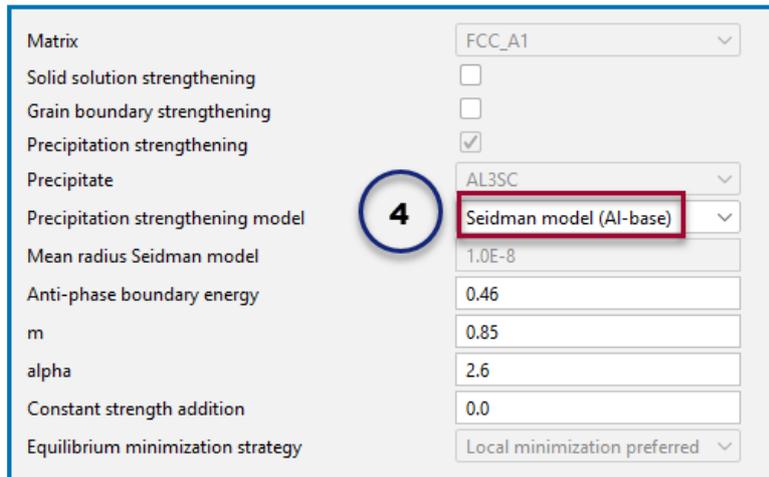
Unit: Seconds 5

Axis type: Logarithmic 10

Limits: 0.0 to 1.0 step 0.1 Automatic scaling 6

Help Add Predecessor Perform Create New Successor

3 Configuration panel...



Matrix FCC\_A1

Solid solution strengthening

Grain boundary strengthening

Precipitation strengthening

Precipitate AL3SC

Precipitation strengthening model 4 Seidman model (Al-base)

Mean radius Seidman model 1.0E-8

Anti-phase boundary energy 0.46

m 0.85

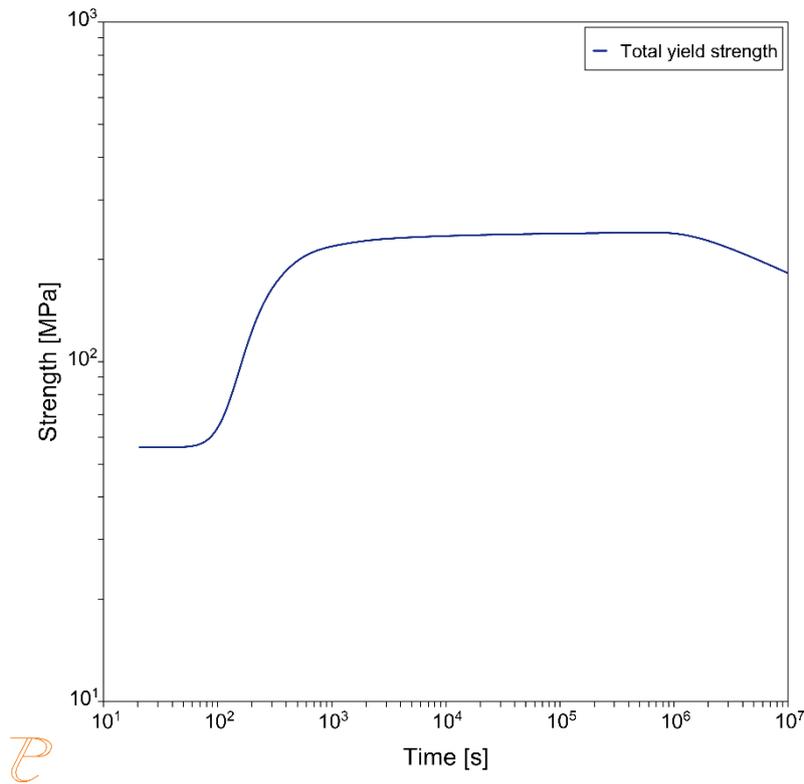
alpha 2.6

Constant strength addition 0.0

Equilibrium minimization strategy Local minimization preferred

### ***Interpreting the Results of the Yield Strength Model***

Once the calculation is complete, your plot is shown in the **Results** window.



*This plot shows the yield strength of the AL3SC precipitate as a function of time.*

## P\_02: Stable and Metastable Carbides - Isothermal

This example simulates the kinetics of precipitation of both stable and metastable carbides from ferrite phase. It demonstrates that metastable carbides (cementite, M7C3) may first emerge and then disappear and the stable phase (M23C6) prevails.

This example uses the Equilibrium Calculator and a one axis calculation to determine how the phases change with temperature. We are interested in the carbide precipitation at 1053 K where only the carbide M23C6 is stable according to the equilibrium calculation. The Precipitation Calculator is used to do an isothermal calculation of the three phases (cementite, M23C6, and M7C3) where cementite and M7C3 are metastable phases.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_02\_Precipitation\_Fe-C-Cr\_Cementite-M7C3-M23C6.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Example Settings

System (System Definer)	
Database package	Demo: Steels and Fe-alloys (FEDEMO,MFEDEMO)
Elements	Fe, C, Cr
Conditions (Precipitation Calculator)	
Composition	Fe-0.1C-12Cr Mass percent
Matrix phase	BCC_A2

Precipitate phases	Cementite, M23C6 and M7C3
<b>Matrix Phase Data Parameters (Precipitation Calculator)</b>	
Grain size (click <b>Show Details</b> to display this setting)	1.0E-4 m
<b>Precipitate Phase Data Parameters (Precipitation Calculator)</b>	
Nucleation sites	Grain boundaries
Interfacial energy	Cementite 0.167 J/m <sup>2</sup> , M23C6 0.252 J/m <sup>2</sup> , M7C3 0.282 J/m <sup>2</sup>
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal
Temperature	1053 K
Simulation time	400 000 seconds

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

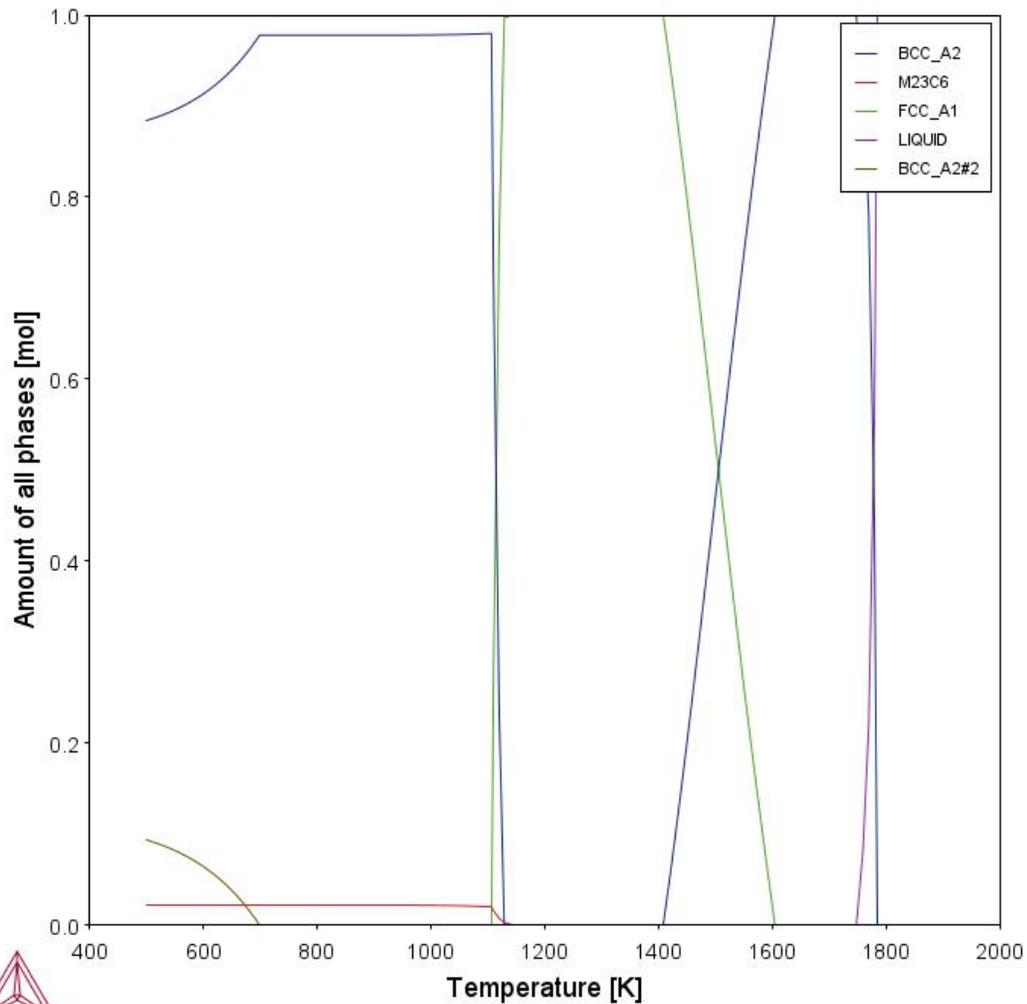
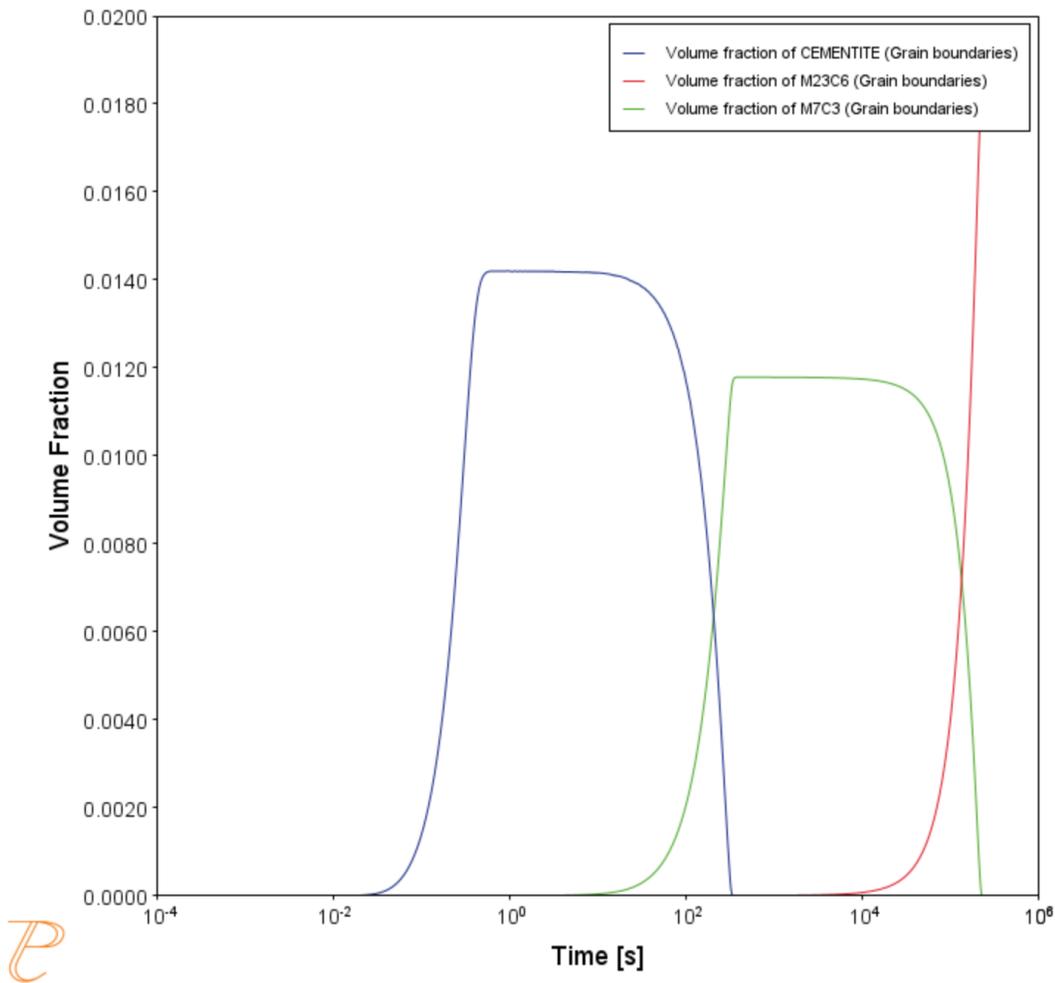


Figure 4: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. This shows the equilibrium property diagram for the Fe-C-Cr alloy, highlighting what phases are predicted to be at equilibrium for a range of temperatures. The calculation shows that at 1053 K, M23C6 is the stable carbide.



*Figure 5: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. This shows the evolution of the volume fraction of cementite, M7C3, and M23C6 during an isothermal heat treatment of an Fe-C-Cr alloy at 1073 K. The cementite and M7C3 are metastable at this temperature, however their nucleation kinetics are faster than the stable phase, allowing them to form and then dissolve when the next more thermodynamically stable phase nucleates.*

## P\_03: Stable and Metastable Carbides - TTT Diagram

In this example, the kinetics of precipitation of both stable and metastable carbides is calculated from the ferrite phase. It demonstrates that metastable carbides may first emerge and then disappear and the stable phase prevails.

This example uses the Equilibrium Calculator and a one axis calculation type to determine how the phases change with temperature. Using this result, the Precipitation Calculator is used to do a TTT (Time-Temperature-Transformation) diagram calculation of the three phases (cementite, M23C6 and M7C3) at the grain boundaries.

For a TTT diagram calculation, select **TTT diagram** in **Calculation Type**, then enter **Min**, **Max**, and **Step of Temperature**, as well as **Max annealing time**. In **Stop criterion**, choose **Volume fraction of phase** and enter the value.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_03\_Precipitation\_Fe-C-Cr\_TTT\_Cementite-M7C3-M23C6.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

### Example Settings

System (System Definer)	
Database package	Demo: Steels and Fe-alloys (FEDEMO and MFEDEMO)
Elements	Fe, C, Cr
Conditions (Precipitation Calculator)	
Composition	Fe-0.1C-12Cr Mass percent
Matrix phase	BCC_A2

Precipitate phases	Cementite, M23C6 and M7C3
<b>Matrix Phase Data Parameters (Precipitation Calculator)</b>	
Grain size (click <b>Show Details</b> to display this setting)	1.0E-4 m
<b>Precipitate Phase Data Parameters</b>	
Nucleation sites	Grain boundaries
Interfacial energy	Cementite 0.167 J/m <sup>2</sup> , M23C6 0.252 J/m <sup>2</sup> , M7C3 0.282 J/m <sup>2</sup>
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	TTT diagram
Temperature	500° to 800° C with 25° C steps
Max. annealing time	1.0E8 seconds
Stop criteria	Volume fraction of phase is set to 0.0001
<b>Options &gt; Numerical Parameters</b>	
No. of grid points over one order of magnitude in radius	150
Max no. of grid points over one order of magnitude in radius	200
Min no. of grid points over one order of magnitude in radius	100

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

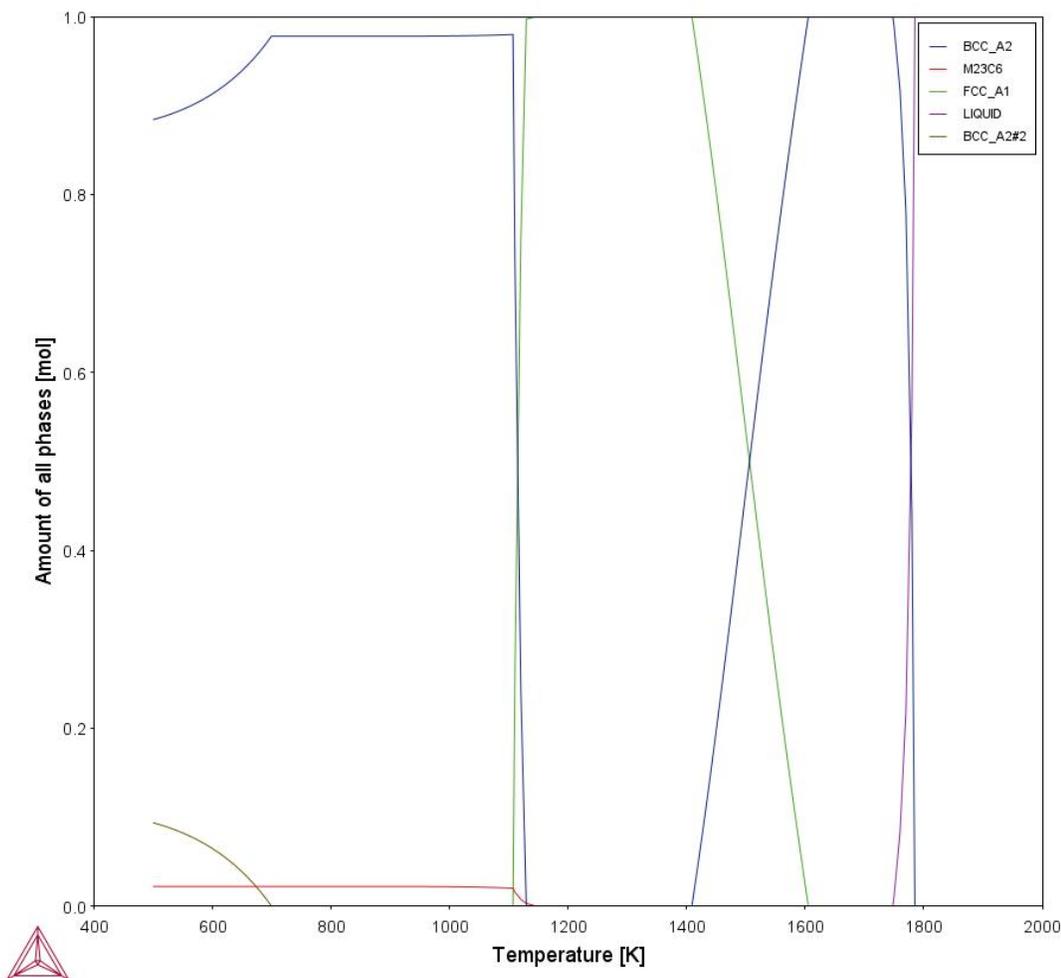


Figure 6: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. Here the results from the equilibrium calculator are shown, which assesses the stability of phases across the temperature range of interest.

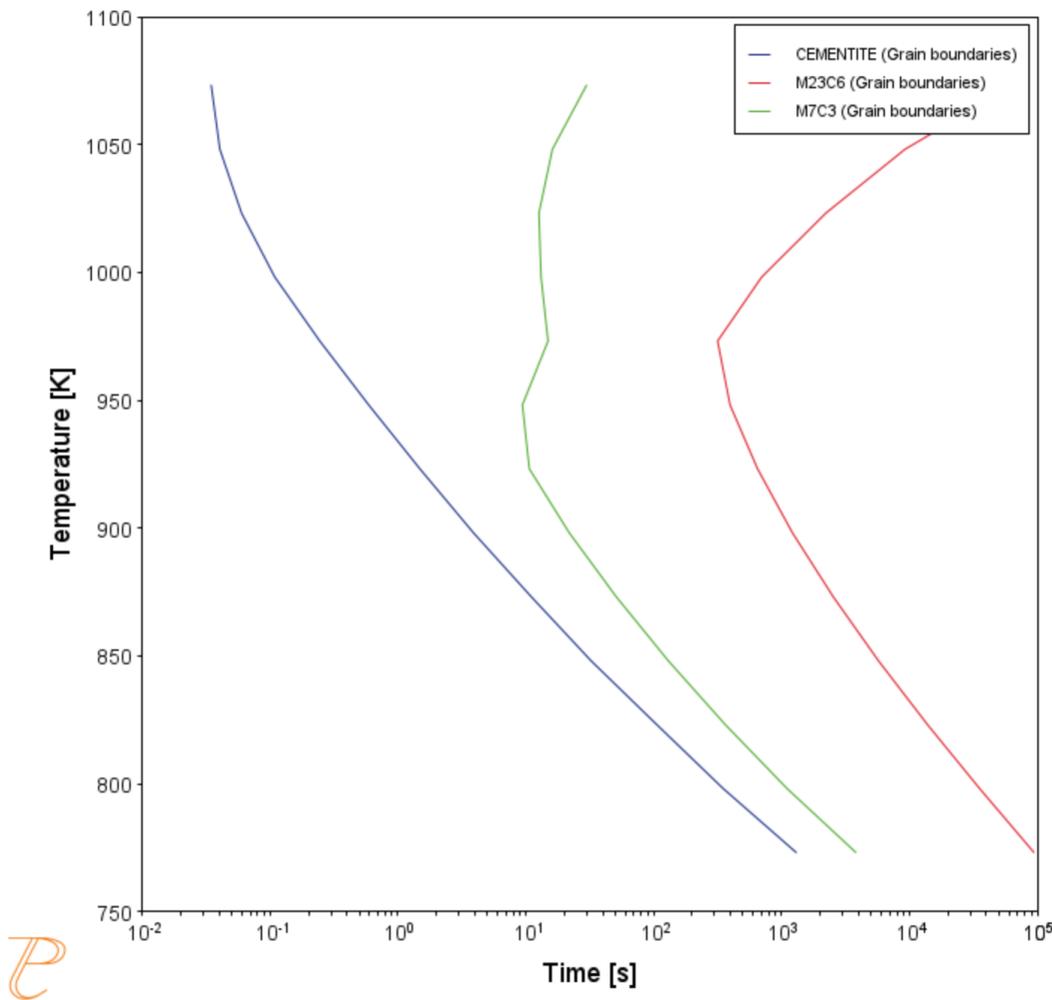


Figure 7: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. This shows results from a TTT (Temperature-Transformation) simulation using the Precipitation Calculator to examine the sequence of phase transitions considering the formation of cementite, M7C3, and M23C6 carbide phases. The previous Equilibrium Calculator reveals the temperatures where the cementite and M7C3 are metastable.

## P\_04: Precipitation of Iron Carbon Cementite

This example is based on [1949Wer] and simulates the kinetics of precipitation of carbides from a BCC Fe solution phase. This isothermal calculation example uses the Precipitation Calculator plus two Experimental File Reader activities to plot the volume fraction of the cementite phase.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_04\_Precipitation\_Fe-C\_Cemetite.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

### Example Settings

System (System Definer)	
Database package	Demo: Steels and Fe-alloys (FEDEMO and MFEDEMO)
Elements	Fe, C
Conditions (Precipitation Calculator)	
Composition	Fe-0.016C mass percent
Matrix phase	BCC_A2
Precipitate phase	Cementite
Matrix Phase Data Parameters (Precipitation Calculator)	
Grain aspect ratio (click <b>Show Details</b> to display this setting)	1.0

Dislocation density (click <b>Show Details</b> to display this setting)	1.5e11m <sup>-3</sup>
<b>Precipitate Phase Parameters (Precipitation Calculator)</b>	
Nucleation sites	Dislocations
Interfacial energy	0.24 J/m <sup>2</sup>
Growth rate model (click <b>Show Details</b> )	Advanced
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal
Temperature	102° C
Simulation time	600 000 seconds

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

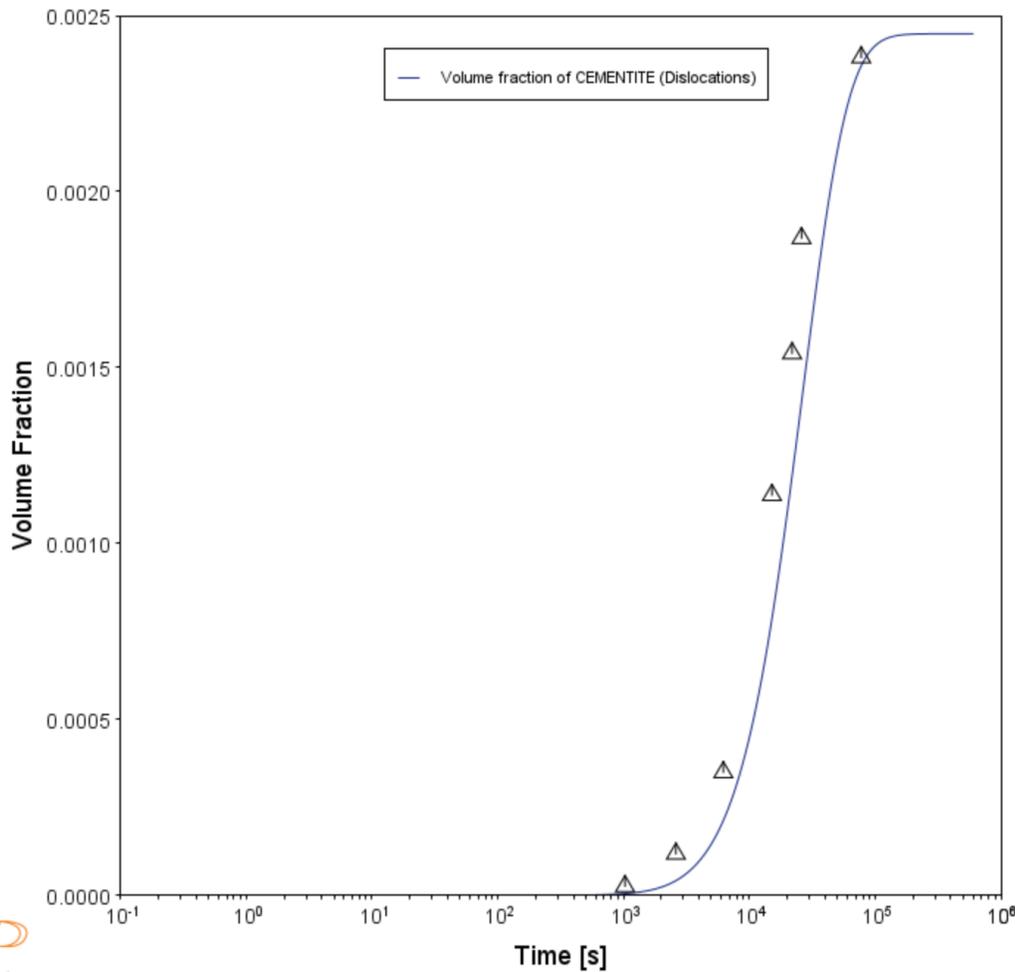


Figure 8: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. The result compares the predicted evolution of the volume fraction of cementite with the measurements of [1949Wer].

## Reference

[1949Wer] C. A. Wert, Precipitation from Solid Solutions of C and N in  $\alpha$ -Iron. J. Appl. Phys. 20, 943 (1949).

## P\_05: Precipitation of $\gamma'$ in Ni Superalloys - Isothermal

This example simulates the kinetics of precipitation of gamma prime ( $\gamma'$ ) phase from gamma ( $\gamma$ ) phase. The simulation results can be compared with experimental data collected from Sudbrack et al. [2008Sud].

This example uses three Experimental File Reader activities with the Precipitation Calculator. It does an isothermal calculation to plot the volume fraction, mean radius, and number density of the cementite phase.



DIS\_FCC\_A1 needs to be selected on the System Definer. See [Selecting the Disordered Phase as a Matrix Phase](#) for details.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_05\_Precipitation\_Ni-Al-Cr\_Isothermal\_Gamma-Gamma\_prime.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Example Settings

System (System Definer)	
Database package	Demo: Nickel-based Super Alloys (NIDEMO and MNIDEMO)
Elements	Ni, Al Cr
Conditions (Precipitation Calculator)	
Composition	Ni-9.8Al-8.3Cr Mole percent
Matrix phase	DIS-FCC_A1 See <a href="#">Selecting the Disordered Phase as a Matrix Phase</a>

Precipitate phase	FCC_L12#2
<b>Precipitate Phase Data Parameters (Precipitation Calculator)</b>	
Nucleation sites	Bulk
Interfacial energy	0.012 J/m <sup>2</sup>
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal
Temperature	800° C
Simulation time	1 000 000 seconds

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

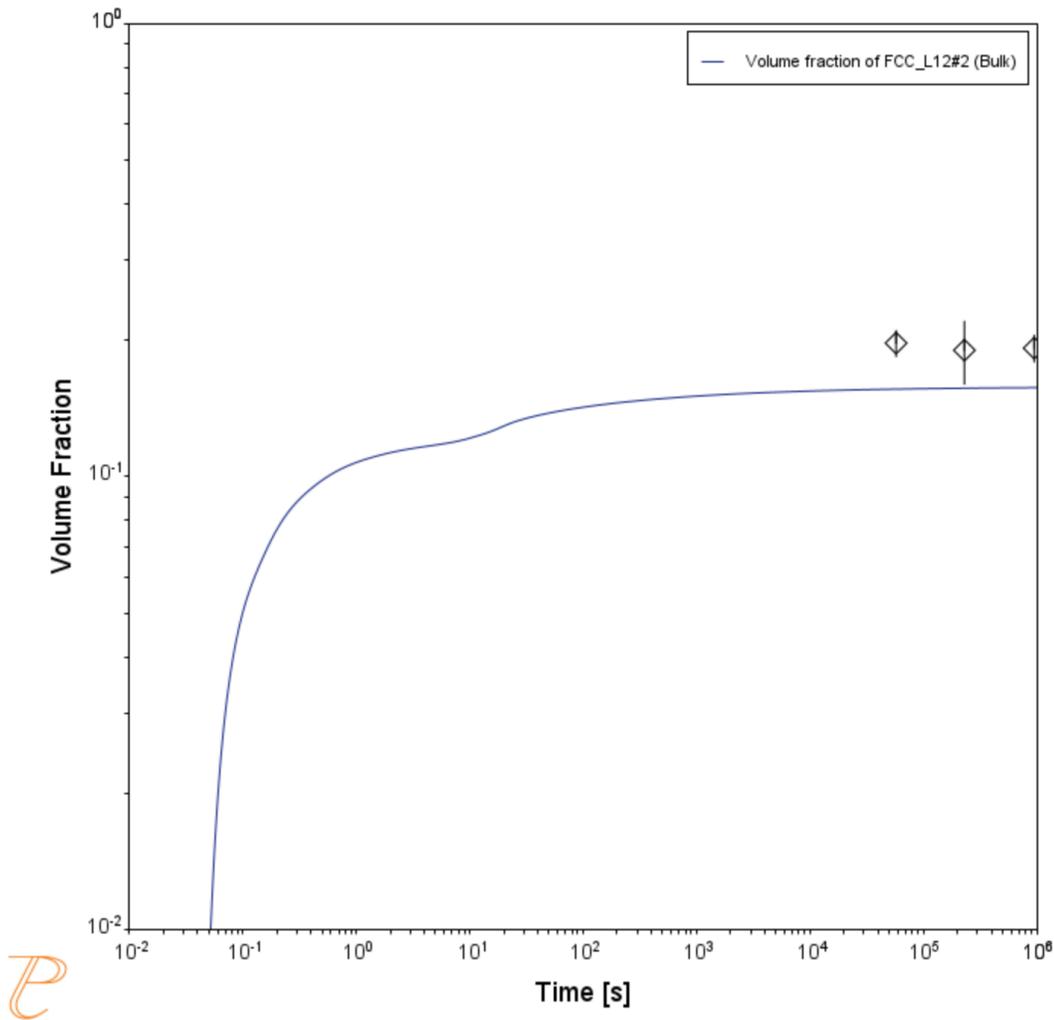


Figure 9: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. This plot compares the predicted evolution of the volume fraction of gamma prime with the measurements of [2008Sud].

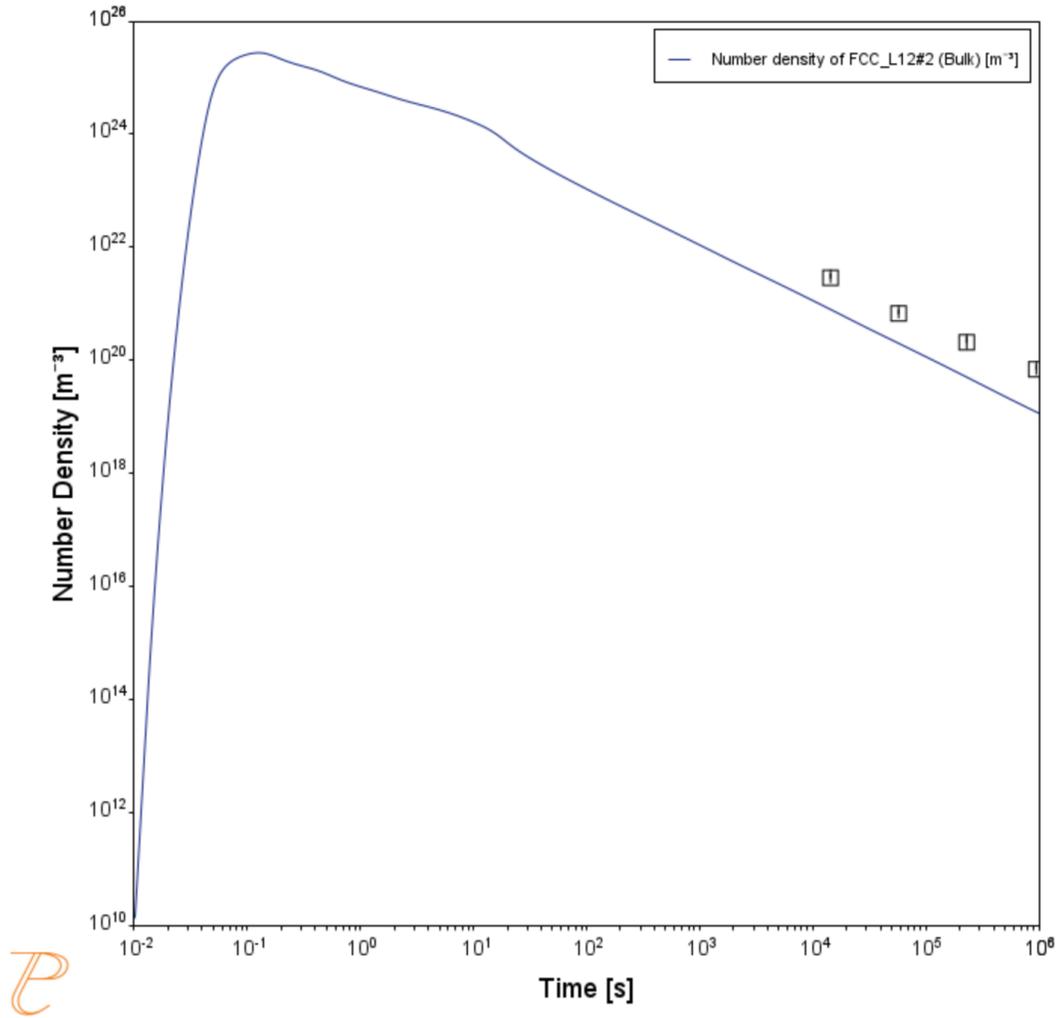


Figure 10: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. This plot compares the predicted evolution of the number density of gamma prime with the measurements of [2008Sud].

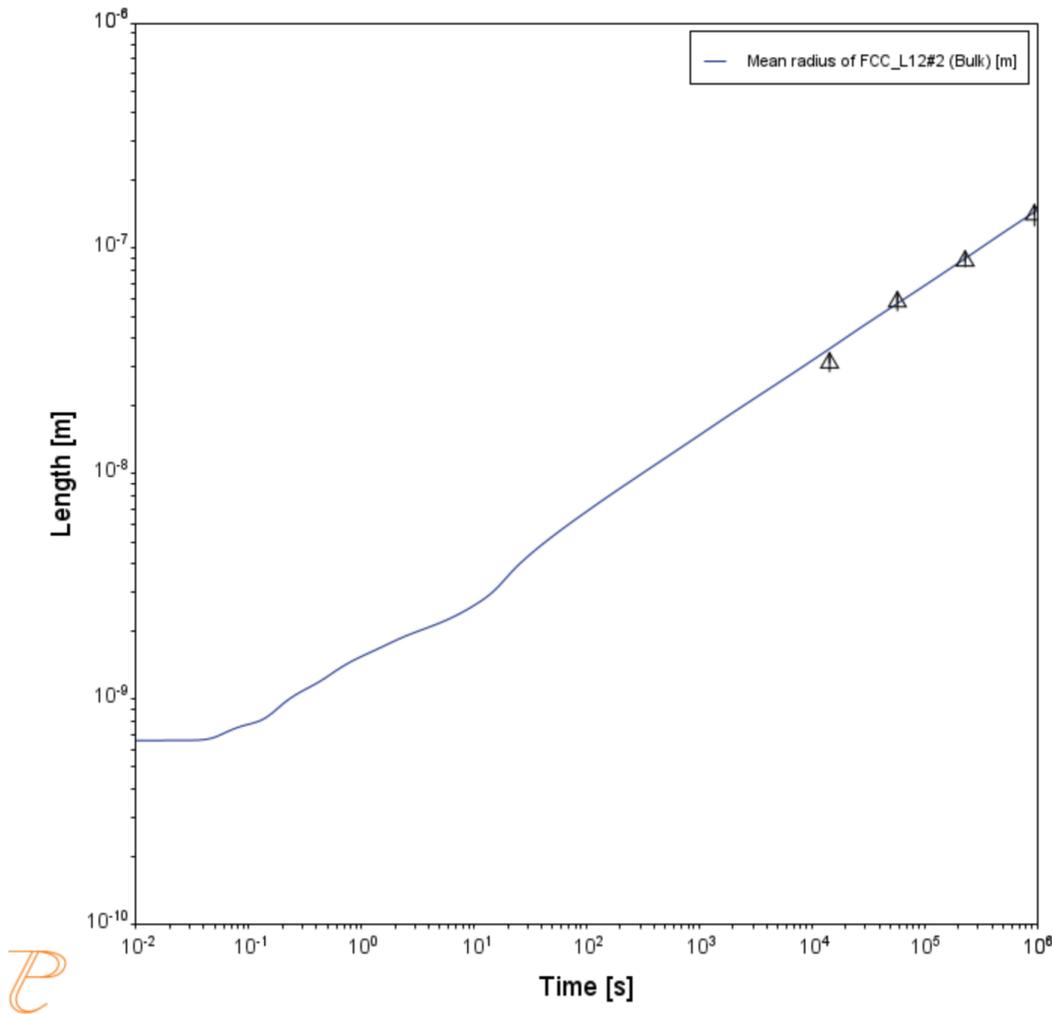


Figure 11: After performing the calculation, you can view the final Plot Renderer result on the applicable tab. This plot compares the predicted evolution of the mean particle radius of gamma prime with the measurements of [2008Sud].

## Reference

[2008Sud] C. K. Sudbrack, T. D. Ziebell, R. D. Noebe, D. N. Seidman, Effects of a tungsten addition on the morphological evolution, spatial correlations and temporal evolution of a model Ni–Al–Cr superalloy. *Acta Mater.* 56, 448–463 (2008).

## P\_06: Precipitation of $\gamma'$ in Ni Superalloys - Non-isothermal

This example simulates the kinetics of precipitation of gamma prime ( $\gamma'$ ) phase from gamma ( $\gamma$ ) phase in Ni-8Al-8Cr and Ni-10Al-10Cr at.% alloys during continuous cooling. The simulation results can be compared with experimental results from Rojhirunsakool et al. [2013Roj].



DIS\_FCC\_A1 needs to be selected on the System Definer. See [Selecting the Disordered Phase as a Matrix Phase](#) for details.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_06\_Precipitation\_Ni-Al-Cr\_Non-isothermal\_Gamma-Gamma\_prime.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Example Settings

System (System Definer)	
Database package	Demo: Nickel-based Super Alloys (NIDEMO and MNIDEMO)
Elements	Ni, Al, Cr
Conditions (Precipitation Calculator)	
Composition (Ni-8Al-8Cr)	Ni-8Al-8Cr Mole percent
Composition (Ni-10Al-10Cr)	Ni-10Al-10Cr Mole percent

Matrix phase	DIS_FCC_A1 See <a href="#">Selecting the Disordered Phase as a Matrix Phase</a>						
Precipitate phase	FCC_L12#2						
<b>Matrix Phase Data Parameters (Precipitation Calculator)</b>							
Mobility adjustment > Prefactor (click <b>Show Details</b> to display this setting)	Keep the default, Same for all elements, then enter 5.0 for the Prefactor.						
<b>Precipitate Phase Data Parameters (Precipitation Calculator)</b>							
Nucleation sites	Bulk						
Interfacial energy	0.023 J/m <sup>2</sup>						
<b>Calculation Type (Precipitation Calculator)</b>							
Calculation type	Non-isothermal						
Temperature unit	Celsius						
Time unit	Seconds						
Temperature	<p>1150 - 380 °C</p> <p><a href="#">Edit Thermal Profile</a></p> <div style="border: 1px solid #ccc; padding: 5px; margin-bottom: 5px;"> <input type="button" value="Import..."/> </div> <table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th style="width: 50%;">Time [s]</th> <th style="width: 50%;">Temperature [°C]</th> </tr> </thead> <tbody> <tr> <td>0.0</td> <td>1150.0</td> </tr> <tr> <td>3300.0</td> <td>380.0</td> </tr> </tbody> </table>	Time [s]	Temperature [°C]	0.0	1150.0	3300.0	380.0
Time [s]	Temperature [°C]						
0.0	1150.0						
3300.0	380.0						
Simulation time (Ni-8Al-8Cr)	3300 s						
Simulation time (Ni-10Al-10Cr)	3300 s						
<b>Multimodal PSD (Plot Renderer)</b>							
Separate multimodal PSD for 8Al-8Cr	The <b>Valley depth ratio</b> is set to 0.05 for both plots (mean radius and PSD) for this alloy. The number of <b>Points</b> is increased to 200 for an average radius plot.						
Separate multimodal PSD for 10Al-10Cr	The <b>Valley depth ratio</b> is set to 0.18 for both plots (mean radius and PSD) for this alloy.						

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the **Precipitation Calculator** tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



In this example, all nodes are renamed in the **Project** window, and this matches each of the tab names in the **Visualizations** window.

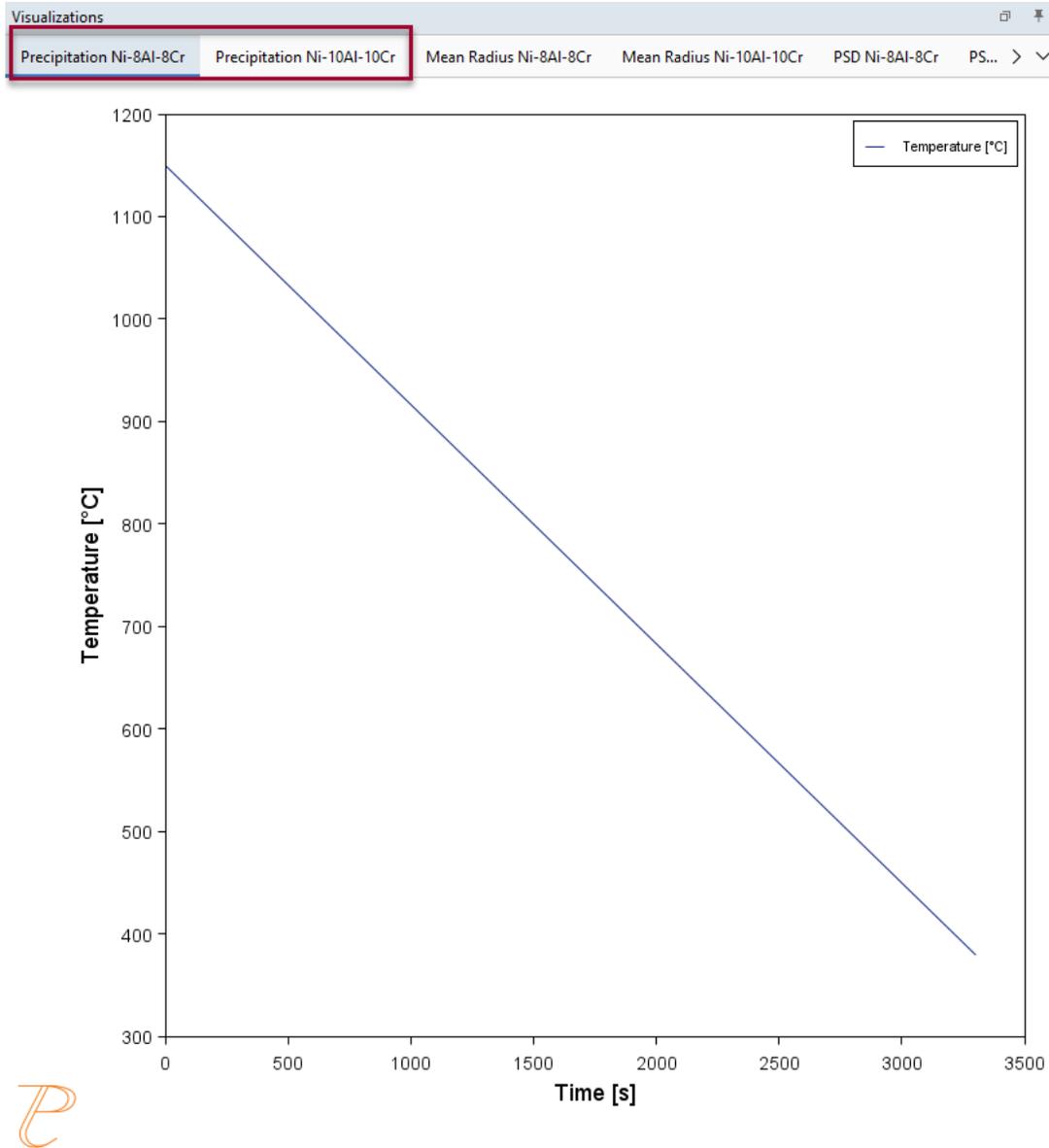


Figure 12: During set up of the calculation, you can preview the non-isothermal Thermal Profile(s). Click the tab in the Visualizations window to adjust settings on the Precipitation Calculator Configuration window. In this case the heat treatment is a continuous cooling curve, however any thermal profile can be modeled using the Non-isothermal Calculation Type.

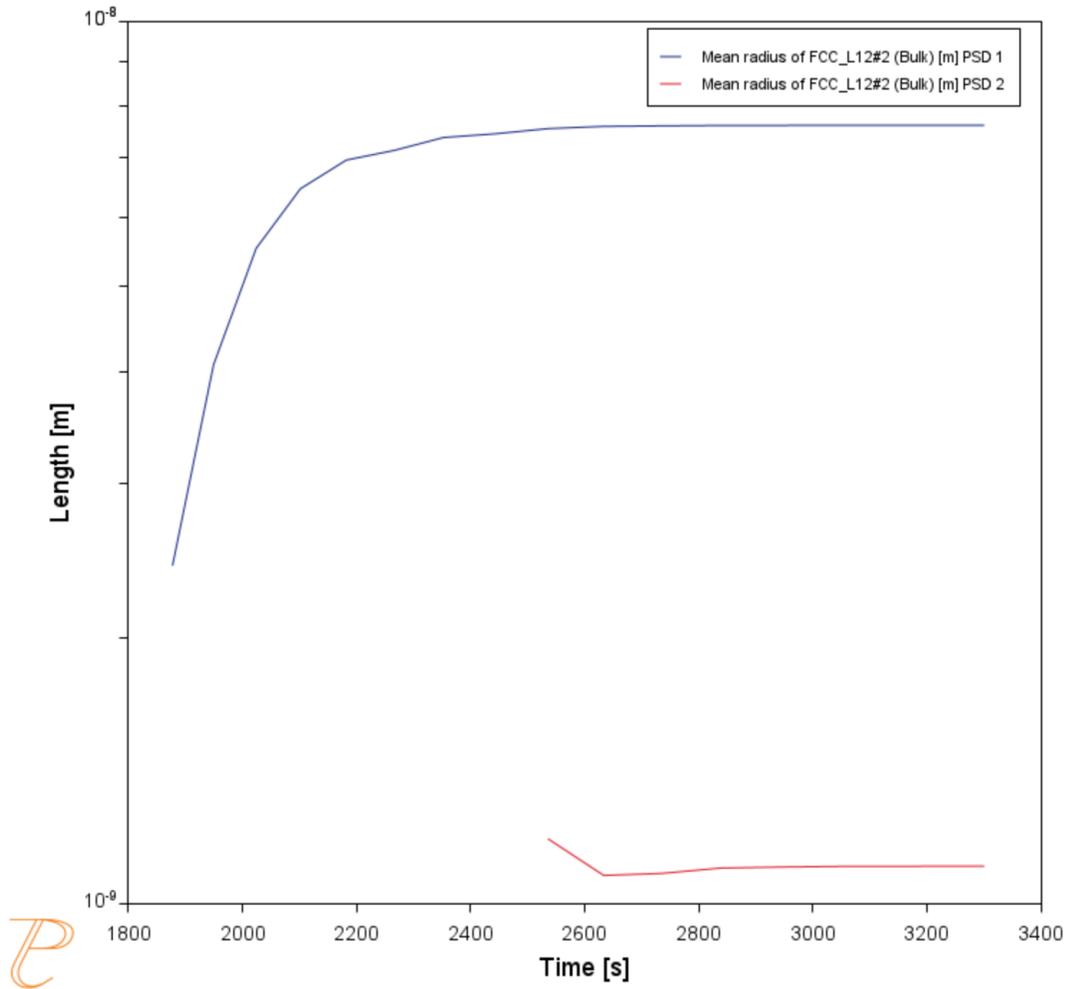


Figure 13: These results show the predicted evolution of the mean radius of gamma prime populations nucleating during the quench for the Ni-8Al-8Cr alloy. Note that the tertiary particles (second particle population) are very small, and this dispersion would appear monomodal without TEM characterization.

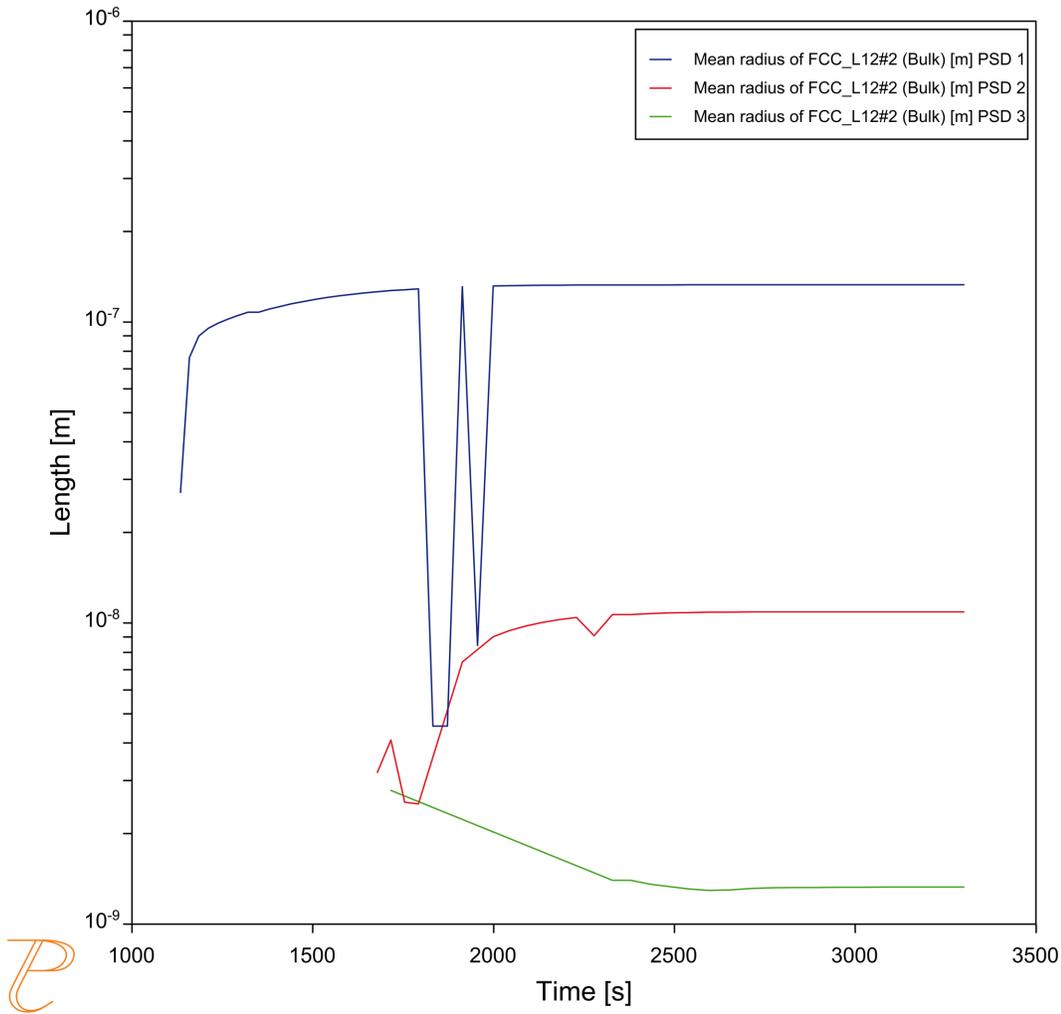


Figure 14: These results show the predicted evolution of the mean radius of gamma prime populations nucleating during the quench for the Ni-10Al-10Cr alloy. In this case three distinct particle populations are simulated to form, agreeing with [2013Roj].

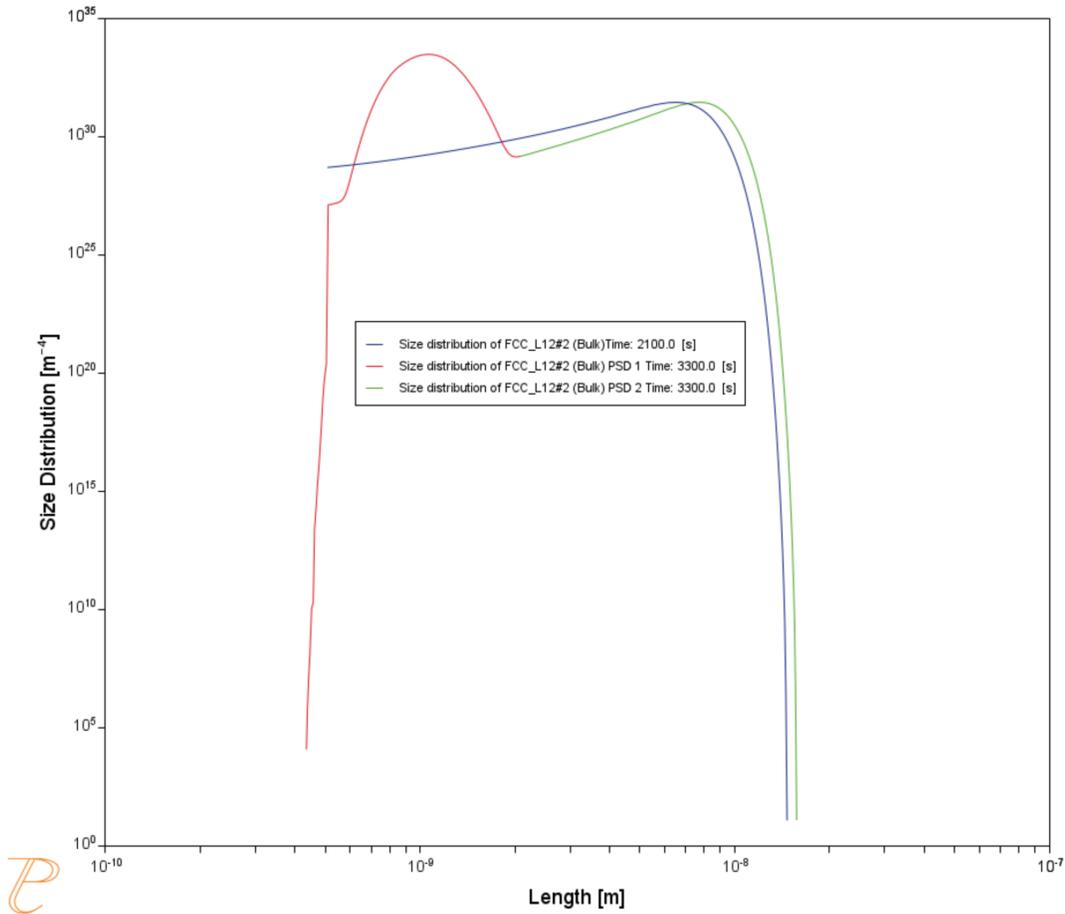


Figure 15: These results show the predicted size distributions of precipitates at different times during the quench of the for the Ni-8Al-8Cr alloy. The individual particle populations are distinguished using the “Separate multimodal PSD” option showing a bimodal distribution with very small nano-meter sized tertiary particles.

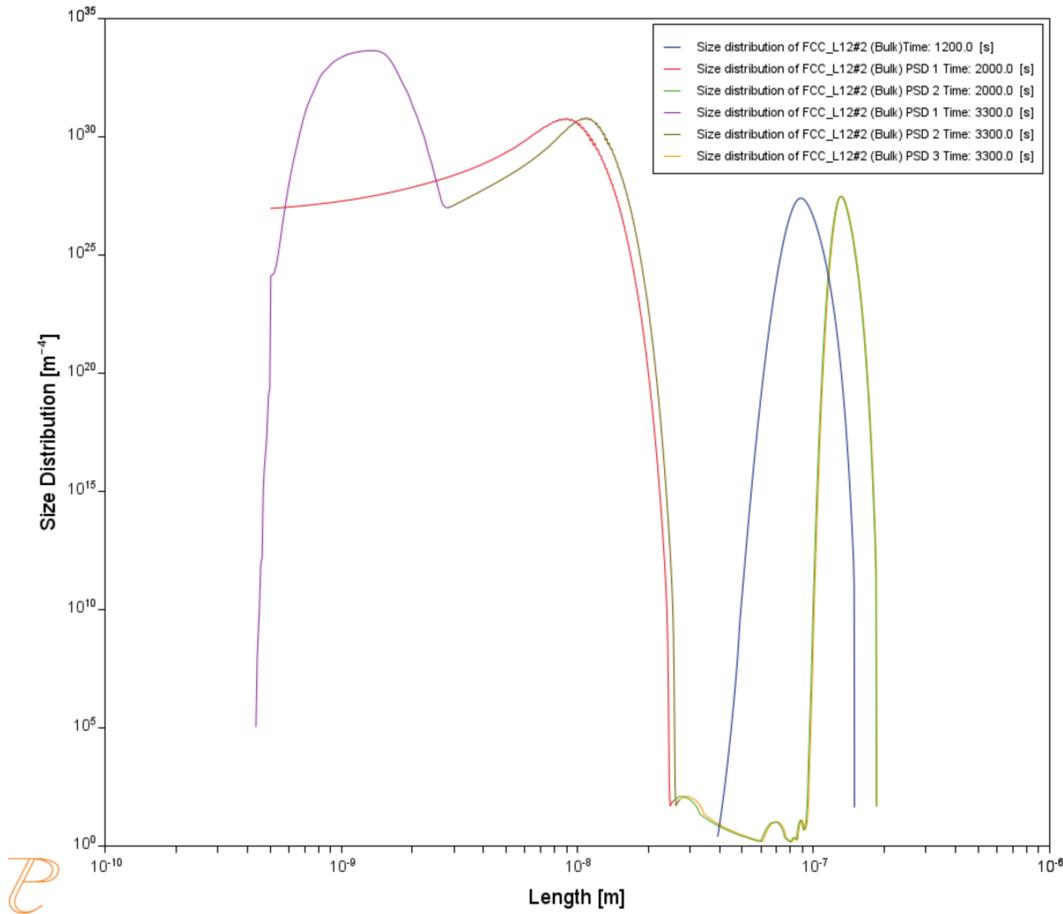


Figure 16: These results show the predicted size distributions of precipitates at different times during the quench of the for the Ni-10Al-10Cr alloy. The individual particle populations are distinguished using the “Separate multimodal PSD” option showing three large notable particle populations.

## Reference

[2013Roj] T. Rojhirunsakool, S. Meher, J. Y. Hwang, S. Nag, J. Tiley, R. Banerjee, Influence of composition on monomodal versus multimodal  $\gamma'$  precipitation in Ni–Al–Cr alloys. *J. Mater. Sci.* 48, 825–831 (2013).

## P\_07: Continuous Cooling Transformation (CCT) Diagram of Ni-Al-Cr $\gamma$ - $\gamma'$

This example shows you how to simulate a CCT (Continuous Cooling Transformation) diagram for gamma prime ( $\gamma'$ ) precipitation in a Ni-Cr-Al alloy using the Precipitation Calculator. A CCT calculation maintains the same cooling rate the entire time.

The system is a Ni-10Al-10Cr  $\gamma$  -  $\gamma'$  alloy and it is calculated and plotted with superimposition of the cooling rate values.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### ***Project File, Step-By Step Instructions, and Video Tutorial Information***

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_07\_Precipitation\_Ni-Al-Cr\_CCT\_Gamma-Gamma\_prime.tcu*



This example is included as a Precipitation Module (TC-PRISMA) tutorial on our [website](#) and as part of the playlist on our [YouTube channel](#).



You can also use [the step-by-step instructions](#) included in a PDF to follow the video or compare to the project file in Thermo-Calc.

### **Example Settings**

System (System Definer)	
Database package	Demo: Nickel-based Super Alloys (NIDEMO and MNIDEMO)
Elements	Ni, Al, Cr
Conditions (Precipitation Calculator)	
Composition	Ni-10Al-10Cr Mole percent
Matrix phase	DIS_FCC_A1

Precipitate phase	FCC_L12#2
<b>Precipitate Phase Data Parameters (Precipitation Calculator)</b>	
Nucleation sites	Bulk
Interfacial energy	0.023 J/m <sup>2</sup>
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	CCT Diagram
Temperature Min to Max	500 to 1200 Kelvin
Cooling rate(s)	.01 .1 1 10 100 K/s
Stop criteria	Volume fraction of phase 1.0E-4

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

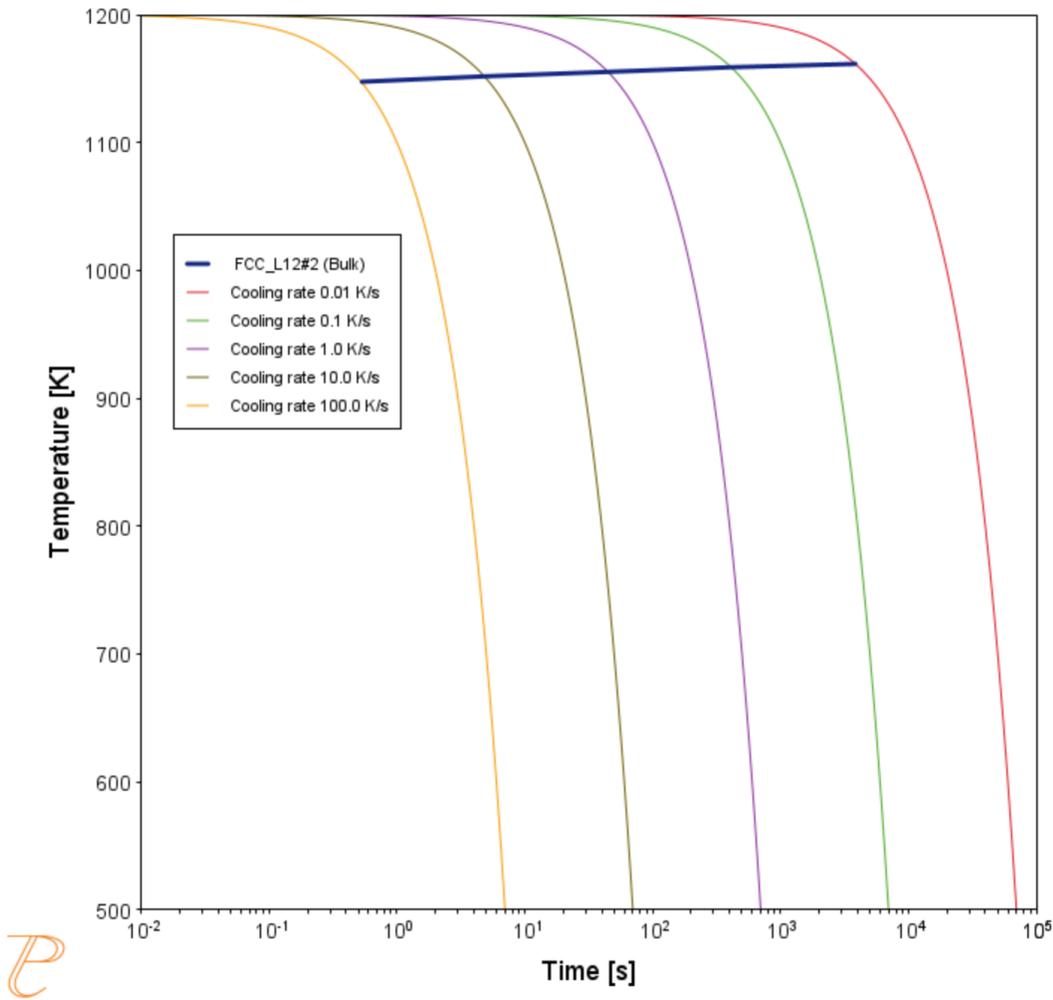
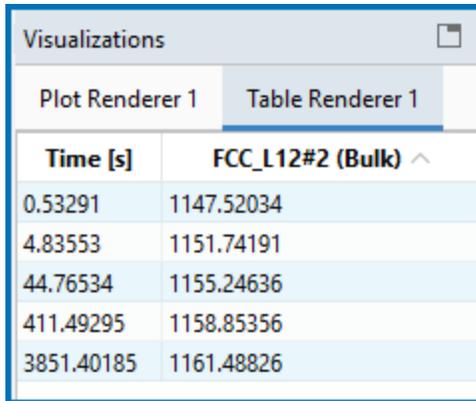


Figure 17: In this plot, the blue horizontal line shows the time it takes for  $\gamma'$  to transform at each of the cooling rates according to the stop criteria, which is set as  $1e-4$  volume fraction. The cooling rates are represented by the multicoloured curved lines.

If you hover your mouse over the intersection of the blue line and any of the vertical lines, a yellow box shows the approximate time it takes for  $\gamma'$  to transform according to the stop criteria, which is a volume fraction of  $1e-4$ , followed by the approximate temperature.



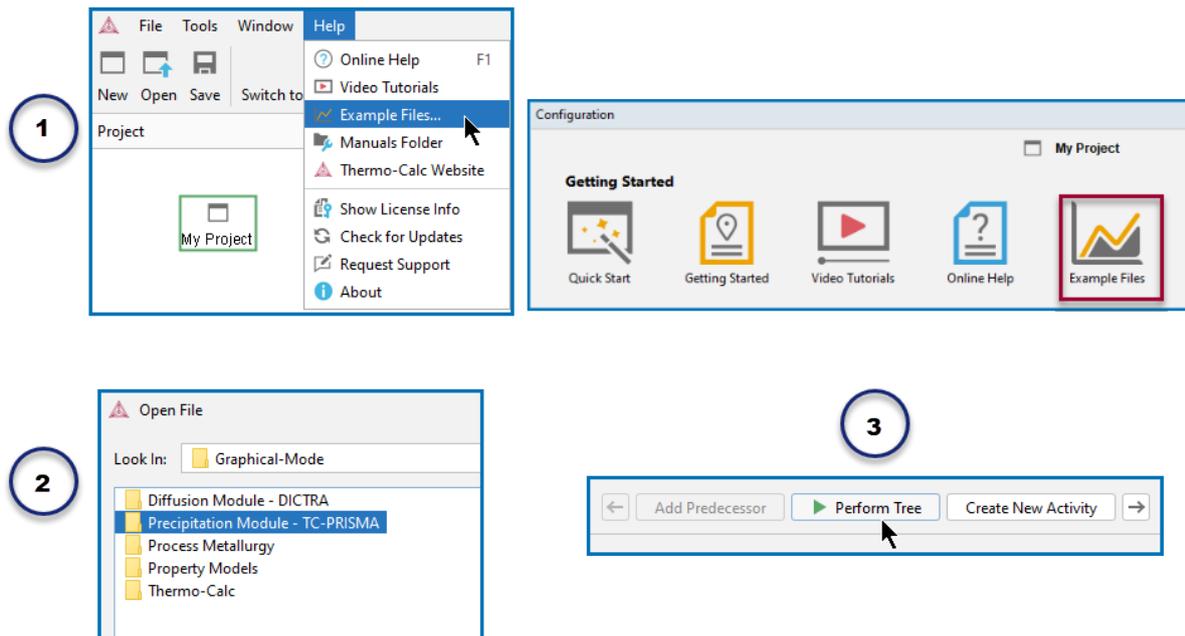
Visualizations	
Plot Renderer 1	Table Renderer 1
Time [s]	FCC_L12#2 (Bulk) ^
0.53291	1147.52034
4.83553	1151.74191
44.76534	1155.24636
411.49295	1158.85356
3851.40185	1161.48826

Figure 18: An example of the table shown in the Visualizations window, which shows the same information as in the plot - for each cooling rate the temperature and the time it takes for  $\gamma'$  to transform according to the stop criteria, which is a volume fraction of  $1e-4$ .

# Continuous Cooling Transformation (CCT) Diagram of Ni-Al-Cr $\gamma$ - $\gamma'$ : Precipitation Example P\_07

## HELPFUL INFORMATION

- **All users can run this calculation**, even those who do not have a license for the Precipitation Module (TC-PRISMA).
  - **A companion video** is available for this example , which can be watched here:  
[https://www.youtube.com/playlist?list=PLfv6McToaTGSpqvuLoY3b\\_UV-8xpgLUKJ](https://www.youtube.com/playlist?list=PLfv6McToaTGSpqvuLoY3b_UV-8xpgLUKJ)
  - This calculation is based on Precipitation Module example *P\_07 – Precipitation Ni-Al-Cr CCT\_Gamma-Gamma\_prime*, which is included in your installation.
1. To run the example file, open Thermo-Calc and select **Help > Examples Files** or click **Example Files** on the **My Project Configuration** window.
  2. Open the *Precipitation Module (TC-PRISMA)* folder. Double-click the example file to open it.
  3. In Thermo-Calc at the bottom center of the **Configuration** window click **Perform Tree**.



## ABOUT THE EXAMPLE

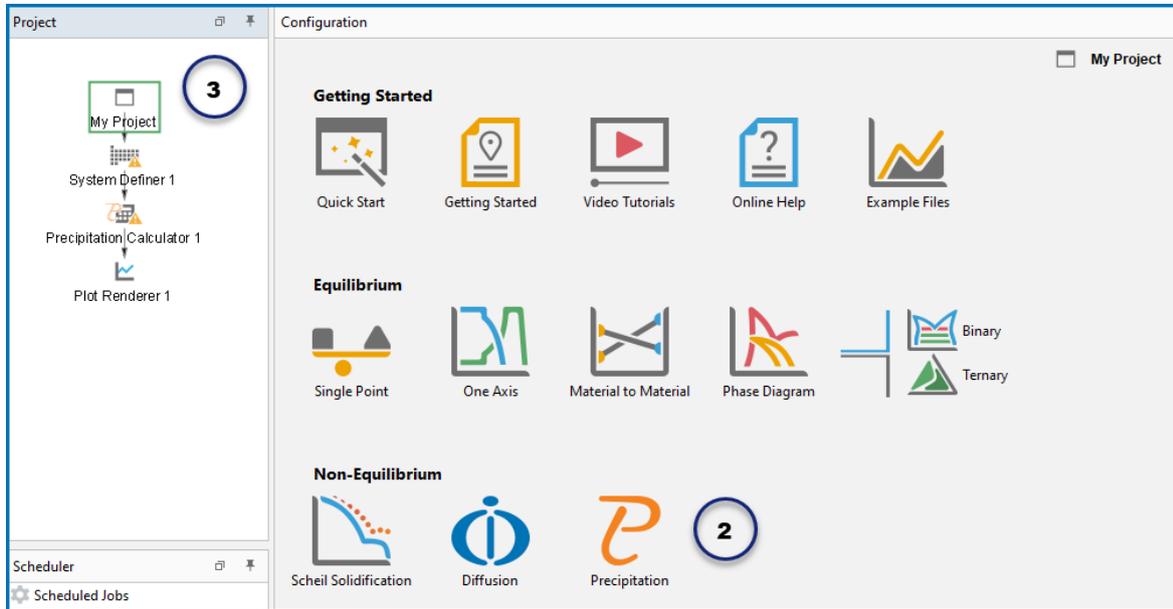
This example shows you how to simulate a CCT diagram for gamma prime ( $\gamma'$ ) precipitation in a Ni-Cr-Al alloy using the Precipitation Module known as TC-PRISMA.

CCT stands for Continuous Cooling Transformation and is a calculation that maintains the same cooling rate the entire time.

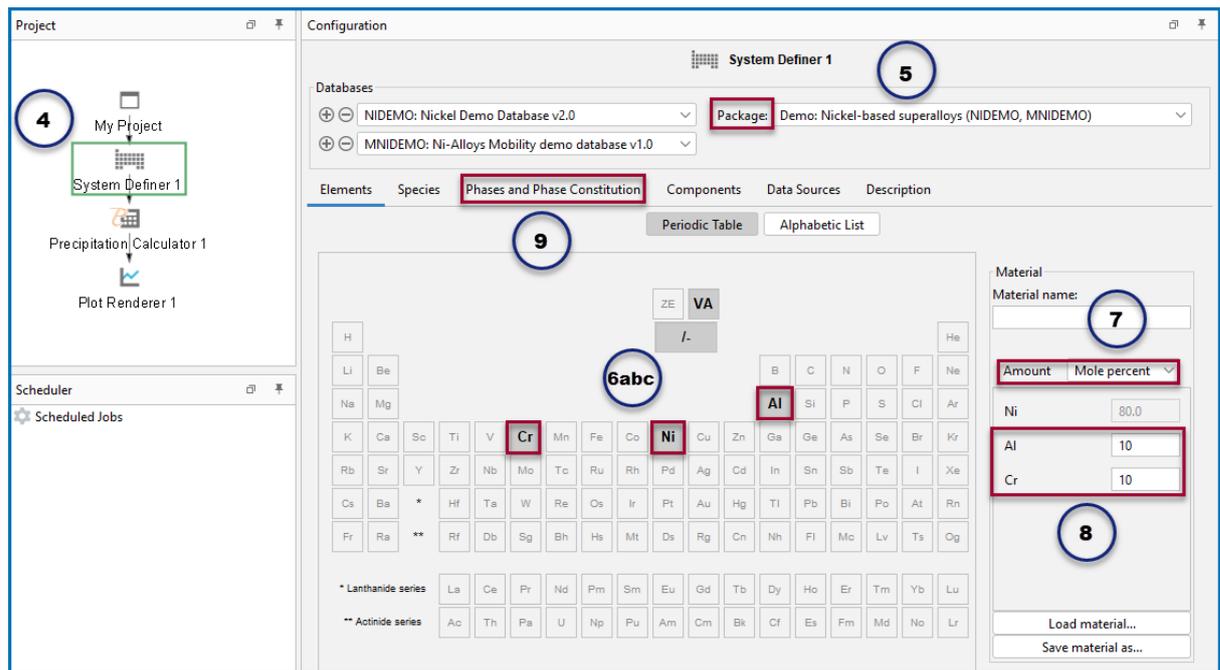
The system is a Ni-10Al-10Cr  $\gamma$  -  $\gamma'$  alloy and it is calculated and plotted with superimposition of the cooling rate values using the Precipitation Module (TC-PRISMA).

## SETTING UP THE SYSTEM

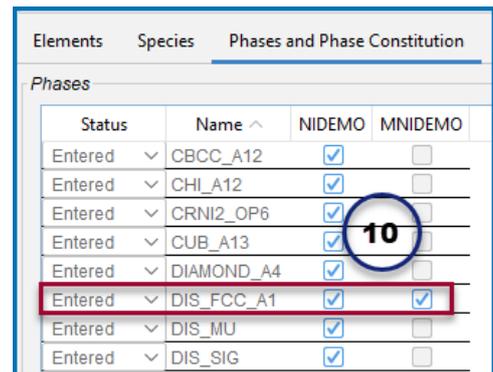
1. Open Thermo-Calc in Graphical Mode.
2. On the **My Project Configuration** window under **Non-Equilibrium**, click **Precipitation**.
3. All the nodes for a precipitation calculation are added to the **Project** window:



4. In the **Project** window, click the **System Definer 1** node.
5. On the **Configuration** window from the *Package* list, select **Demo: Nickel-based alloys (NIDEMO, MNIDEMO)**. This loads both thermodynamic and kinetic demonstration nickel databases.
6. From the **Periodic Table**, select the elements in the following order. Select Ni first so that it is the dependant element.
  - a. **Ni** (nickel)
  - b. **Al** (aluminium) then
  - c. **Cr** (chromium).
7. From the *Amount* list (to the right of the Periodic Table), select **Mole percent**.
8. Enter 10 for **Al** and 10 for **Cr**, which automatically sets **Ni** to 80 mole percent.



9. Click the **Phases and Phase Constitution** tab.
10. Under *Phases* click to select both check boxes next to the disordered FCC phase, **DIS\_FCC\_A1**.



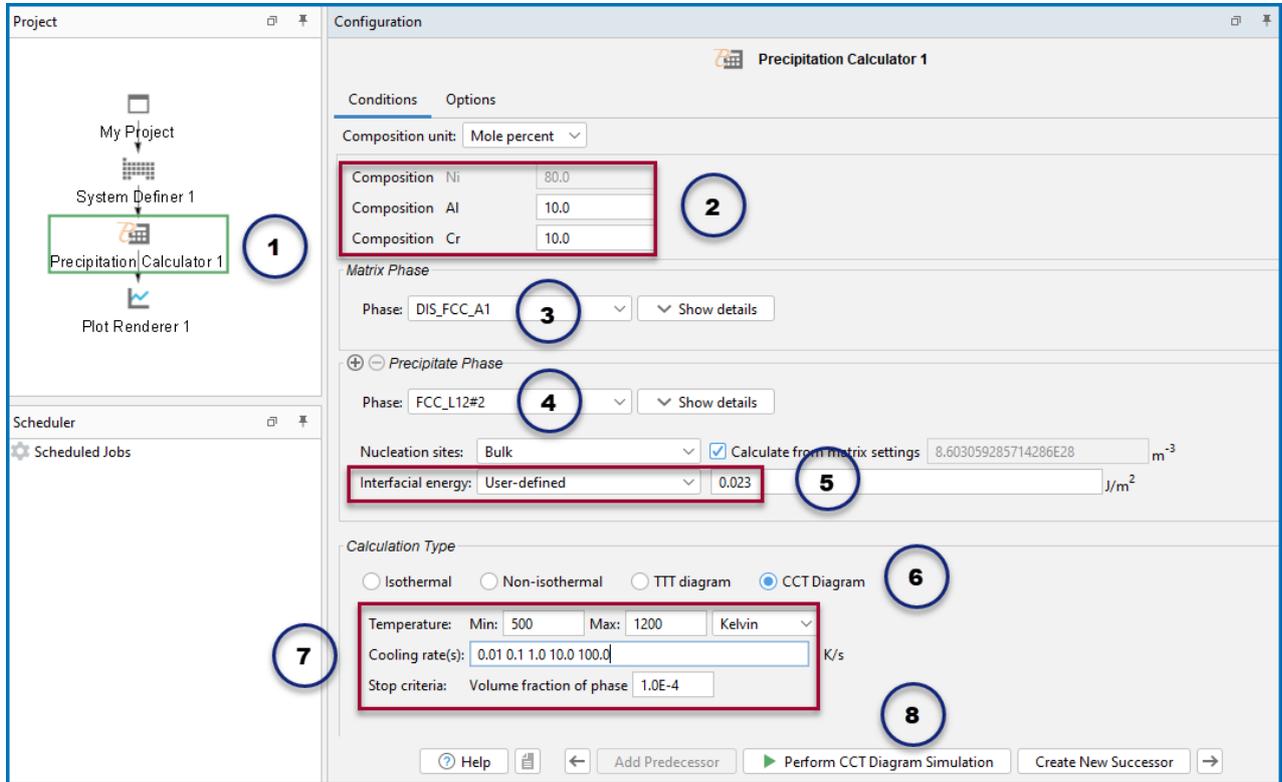
Status	Name ^	NIDEMO	MNIDEMO
Entered	CBCC_A12	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	CHI_A12	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	CRNI2_OP6	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	CUB_A13	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	DIAMOND_A4	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	DIS_FCC_A1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Entered	DIS_MU	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Entered	DIS_SIG	<input checked="" type="checkbox"/>	<input type="checkbox"/>

The system is now defined. The next step is to set up the precipitation calculations.

## SETTING UP THE PRECIPITATION CALCULATION

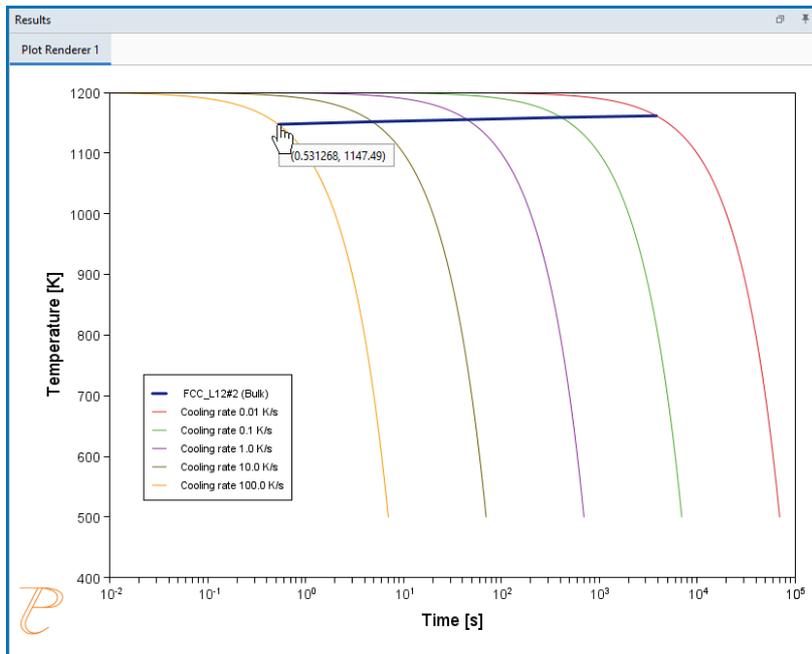
1. In the **Project** window, click the **Precipitation Calculator 1** node.
2. Notice that the composition set in the *System Definer* auto-populated here.
3. Under *Matrix Phase*, from the *Phase* list select **Dis\_FCC\_A1**, which is the disordered FCC phase.
4. Under *Precipitate Phase* from the *Phase* list scroll to select **FCC\_L12#2**.
5. Change the *Interfacial energy* to **User-defined** then enter 0.023 in the field.
6. Under *Calculation Type*, click **CCT Diagram**, which stands for Continuous-Cooling-Transformation and means that the same cooling rate is maintained throughout the calculation.
7. In the fields set:
  - a. *Min* (minimum) *Temperature* to 500.
  - b. *Max* (maximum) *Temperature* to 1200.
  - c. Keep the default **Kelvin** as the temperature unit.
  - d. Enter several cooling rates for the calculation. The rates are separated by a space. Enter these values as shown: .01 .1 1 10 100.

- e. Keep the default **Stop criteria** of  $1E-4$  volume fraction of the  $\gamma'$  phase.
8. The calculation is now set. At the bottom, center of the **Configuration** window click **Perform CCT Diagram Simulation**.



The screenshot shows the Configuration window for Precipitation Calculator 1. The Configuration section is divided into Conditions and Options. The Conditions section includes Composition (Ni: 80.0, Al: 10.0, Cr: 10.0), Matrix Phase (DIS\_FCC\_A1), and Precipitate Phase (FCC\_L12#2). The Options section includes Nucleation sites (Bulk), Interfacial energy (User-defined, 0.023 J/m<sup>2</sup>), and Calculation Type (CCT Diagram). The Temperature range is set from 500 to 1200 Kelvin, and the Cooling rate is set to 0.01, 0.1, 1.0, 10.0, and 100.0 K/s. The Stop criteria is set to Volume fraction of phase 1.0E-4. A blue box highlights the 'Perform CCT Diagram Simulation' button.

## INTERPRETING THE RESULTS OF THE PRECIPITATION CALCULATION



In this plot, the blue, horizontal line shows the time it takes for gamma prime ( $\gamma'$ ) to transform at each of the cooling rates according to the stop criteria, which we set as  $1e-4$  volume fraction.

In this plot, the blue line shows the time it takes for  $\gamma'$  to transform at each of the cooling rates according to the stop criteria, which is set as  $1e-4$  volume fraction. The cooling rates are represented by the multi-coloured curved lines.

If you hover your mouse over the intersection of the blue line and any of the vertical lines, a yellow box shows the approximate time it takes for  $\gamma'$  to transform according to the stop criteria, which is a volume fraction of  $1e-4$ , followed by the approximate temperature. In the image above, you can see a time of 0.531268 seconds and a temperature of 1147.49 Kelvin for the cooling rate of 100.0 K/s, which is represented by the yellow line.

## SHOWING THE RESULTS AS A TABLE

You can also view these results in the form of a table, which gives you more precise results.

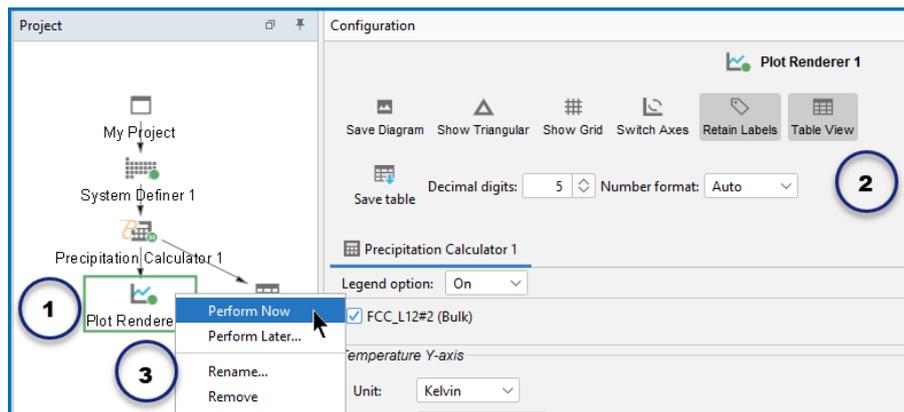
1. In the **Project** window, right-click the **Precipitation Calculator 1** node and select **Create New Successor>Table Renderer**.
2. Right-click the **Table Renderer 1** node and select **Perform Now**.

The table is shown in the **Results** window and shows the same information as in the plot - for each cooling rate the temperature and the time it takes for  $\gamma'$  to transform according to the stop criteria, which is a volume fraction of  $1e-4$ .

Results	
Plot Renderer 1	Table Renderer 1
Time [s]	FCC_L12#2 (Bulk)
0.53291	1147.52034
4.83553	1151.74191
44.76534	1155.24636
411.49295	1158.85356
3851.40185	1161.48826

Tip! You can also convert the plot data into a table format that includes all the points.

1. On the **Project** window, click **Plot Renderer 1**.
2. On the **Configuration** window, click the **Table View** button
3. After clicking the button, click **Perform** at the bottom of the window or right-click the **Plot Renderer 1** node and select **Perform Now**.
4. View the converted data on the **Results** window.



4

Results					
Plot Renderer 1		Table Renderer 1			
Time [s]	FCC_L12#2 (Bulk)	Time [s] [1]	Cooling rate 0.01 K/s	Time [s] [2]	Cooling rate 0.1 K/s
0.53291	1147.52034	0.01000	1199.99990	0.01000	1199.99900
4.83553	1151.74191	0.01082	1199.99989	0.01070	1199.99893
44.76534	1155.24636	0.01172	1199.99988	0.01145	1199.99886
411.49295	1158.85356	0.01268	1199.99987	0.01225	1199.99878
3851.40185	1161.48826	0.01373	1199.99986	0.01311	1199.99869

## P\_08: Precipitation of Cu-Ti CU4TI with Assumptions of Sphere and Needle Morphologies

---

In this isothermal calculation example, the precipitation of Cu<sub>4</sub>Ti phase in a Cu-Ti binary alloy is calculated. To make a comparison, two separate simulations are performed, one assuming spherical morphology without elastic strain energy, and the other assuming needle morphology whose shape, determined by competition between interfacial energy and elastic strain energy, is changed during the simulation. The transformation strain is obtained from Borchers [1999Bor]. The results are compared with experiment results from Kampmann et al. [1987Kam].



For more details about the background theory, see [Precipitation Morphology](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_08\_Precipitation\_Cu-Ti\_CU4TI1\_Sphere\_Needle.tcu*

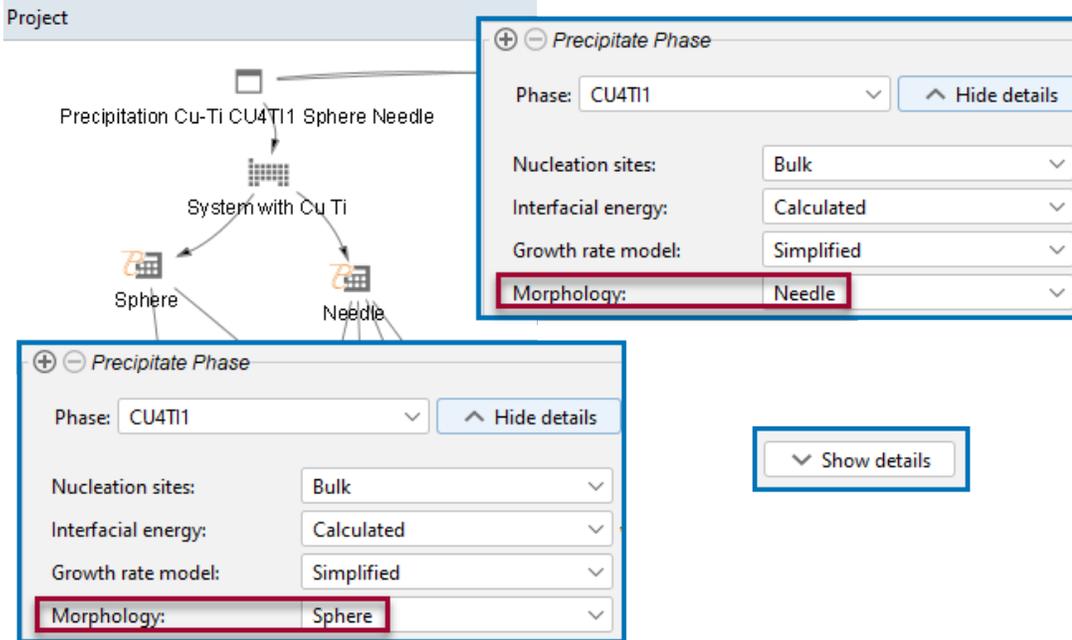


Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

### Example Settings



To ensure that the settings are done on the correct Precipitation Calculators, the **Sphere** and **Needle** nodes are renamed from **Precipitation Calculator** to match their morphology. The morphology is set in the **Precipitate Phase** section when you click **Show Details**.



**Project**

Precipitation Cu-Ti CU4Ti1 Sphere Needle

System with Cu Ti

Sphere Needle

**Precipitate Phase**

Phase: CU4Ti1 [^ Hide details](#)

Nucleation sites: Bulk

Interfacial energy: Calculated

Growth rate model: Simplified

Morphology: Needle

**Precipitate Phase**

Phase: CU4Ti1 [^ Hide details](#)

Nucleation sites: Bulk

Interfacial energy: Calculated

Growth rate model: Simplified

Morphology: Sphere

[v Show details](#)

### System (System Definer)

Database package	Demo: Copper-based alloys (CUDEMO and MCUDEMO)
Elements	Cu, Ti

### Sphere and Needle Conditions (Precipitation Calculator)

Composition	Cu-1.9Ti Mole percent
Matrix phase	FCC_L12
Precipitate phase	CU4Ti1

### Matrix Phase Data Parameters (Precipitation Calculator)

Mobility adjustment > Prefactor (click <b>Show Details</b> to display this setting)	Keep the default, Same for all elements, then enter 100 for the Prefactor.
---	--

### Precipitate Phase Data Parameters (Precipitation Calculator)

Nucleation sites	Bulk
Interfacial energy	The default
Morphology (click <b>Show Details</b> to display this setting)	For the <b>Sphere</b> node (renamed from Precipitation Calculator), keep the default.

Transformation strain (click <b>Show Details</b> to display this setting)	<p>For the <b>Needle</b> node (renamed from Precipitation Calculator), <b>Needle</b> is selected.</p> <p>For the <b>Sphere</b> node (renamed from Precipitation Calculator), keep the default.</p> <p>For the <b>Needle</b> node (renamed from Precipitation Calculator), <b>User defined</b> is selected. In this example, the following settings are defined:</p> <ul style="list-style-type: none"><li>• <math>\epsilon_{11}</math> and <math>\epsilon_{22}</math> are set to <b>0.022</b></li><li>• <math>\epsilon_{33}</math> is set to <b>0.003</b></li></ul>
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal
Temperature	350° C
Simulation time	10,000 seconds
<b>Datasets (Experimental File Reader)</b>	
Borchers Mean radius vs Time and Borchers Number density vs Time	Data sets included with this example and imported to two Experimental File Readers. These data sets are used for the Mean Radius and Number Density plots, respectively.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer**

---

or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



In this example, all nodes are renamed in the **Project** window, and this matches each of the tab names in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

---

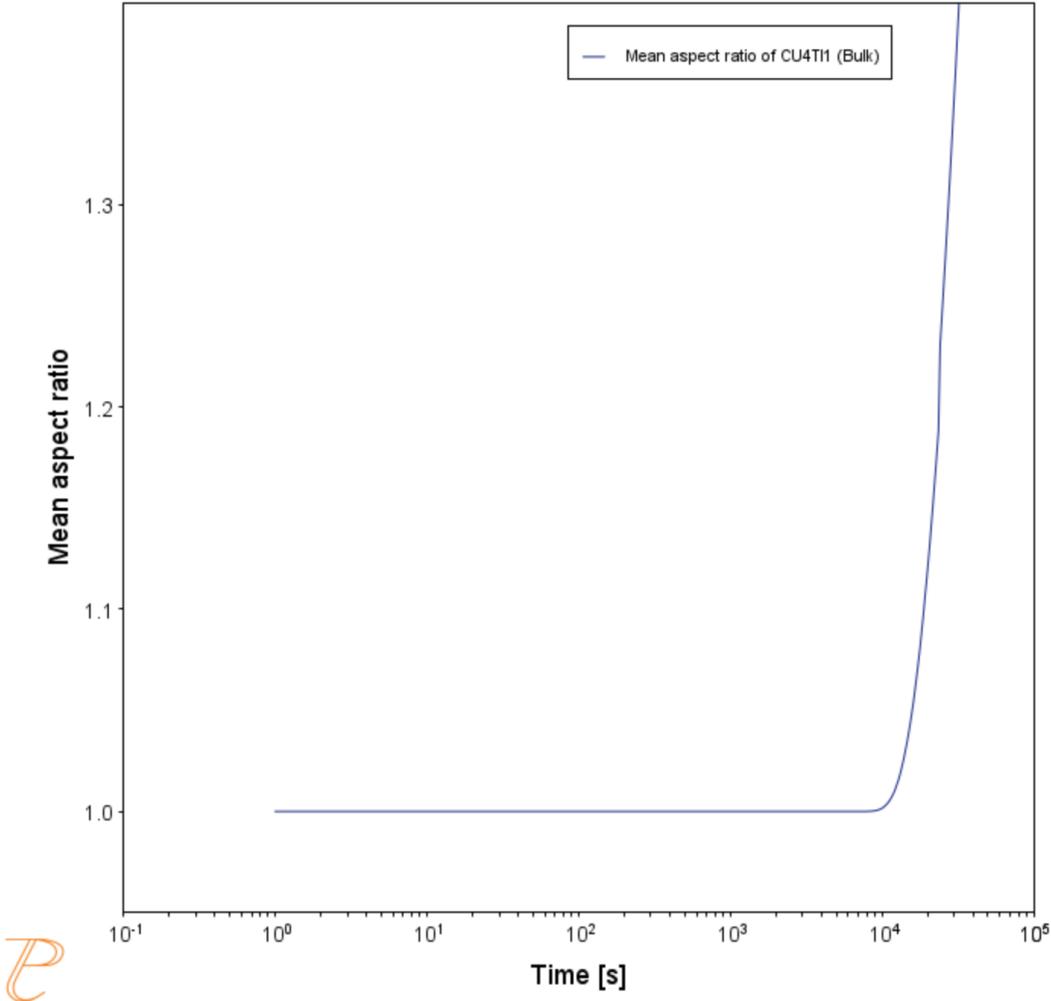


Figure 19: The predicted evolution of the morphology of needle shaped precipitates, showing how the mean aspect ratio changes during the isothermal heat treatment.

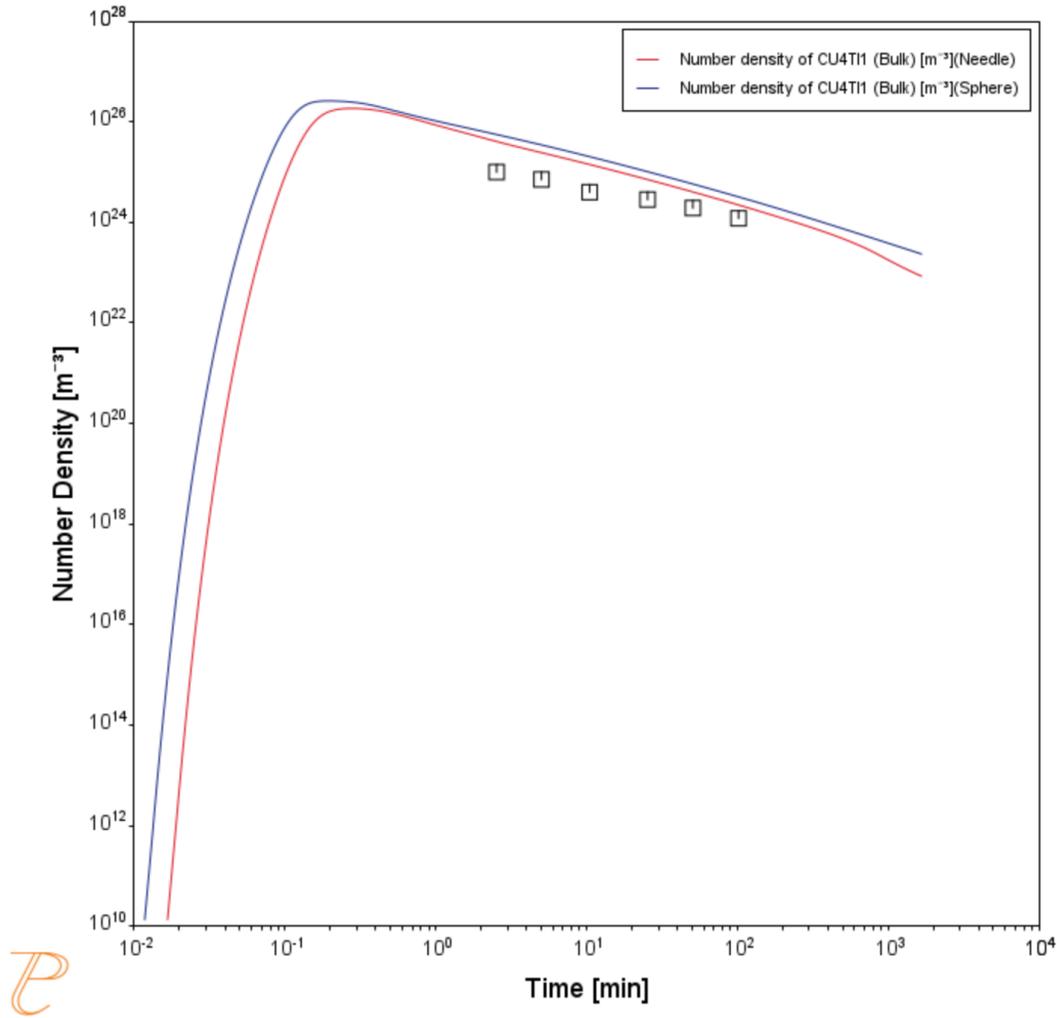


Figure 20: The results compare the predicted evolution of the mean size of precipitates with the experimental data from [1987Kam], comparing the spherical and needle shaped approximations of the precipitate morphology. Note how the needle shape approximation is closer to the experimental data.

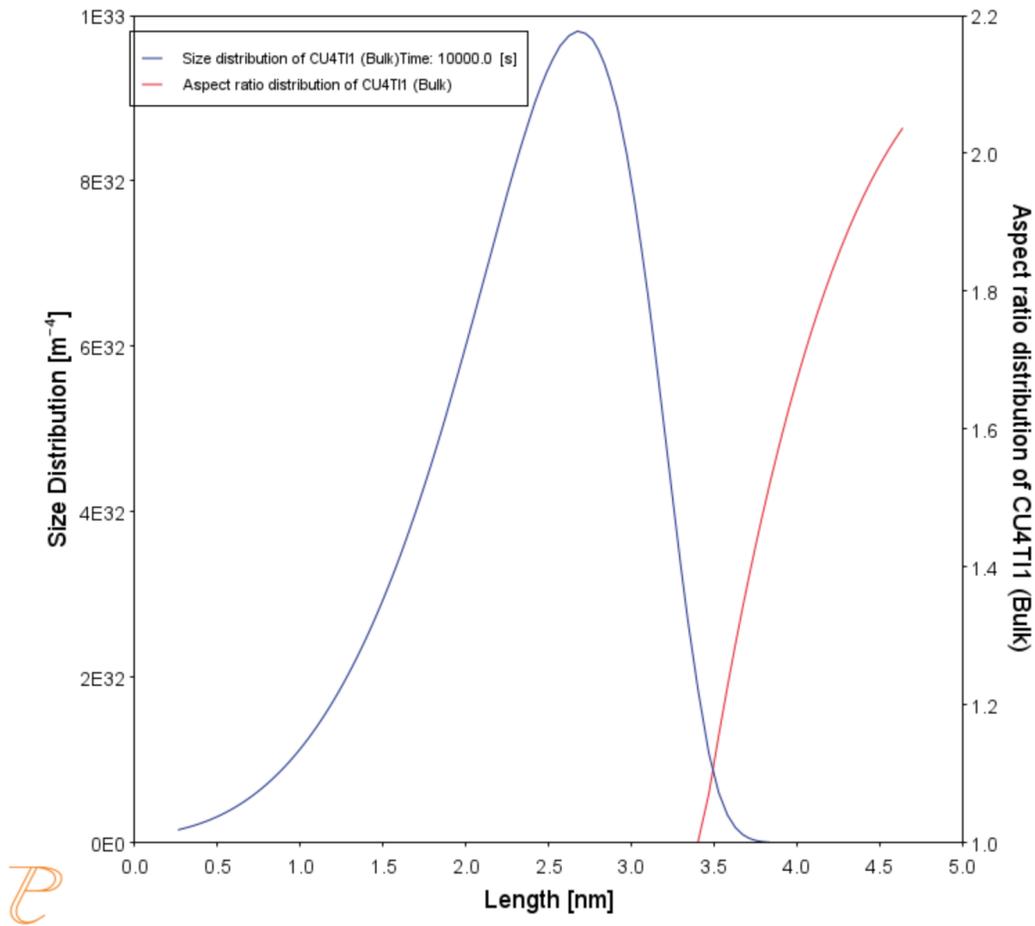


Figure 21: These results show both the predicted final particle size distribution from the needle shape simulation and the predicted relationship between the length of the precipitate and the needle aspect ratio.

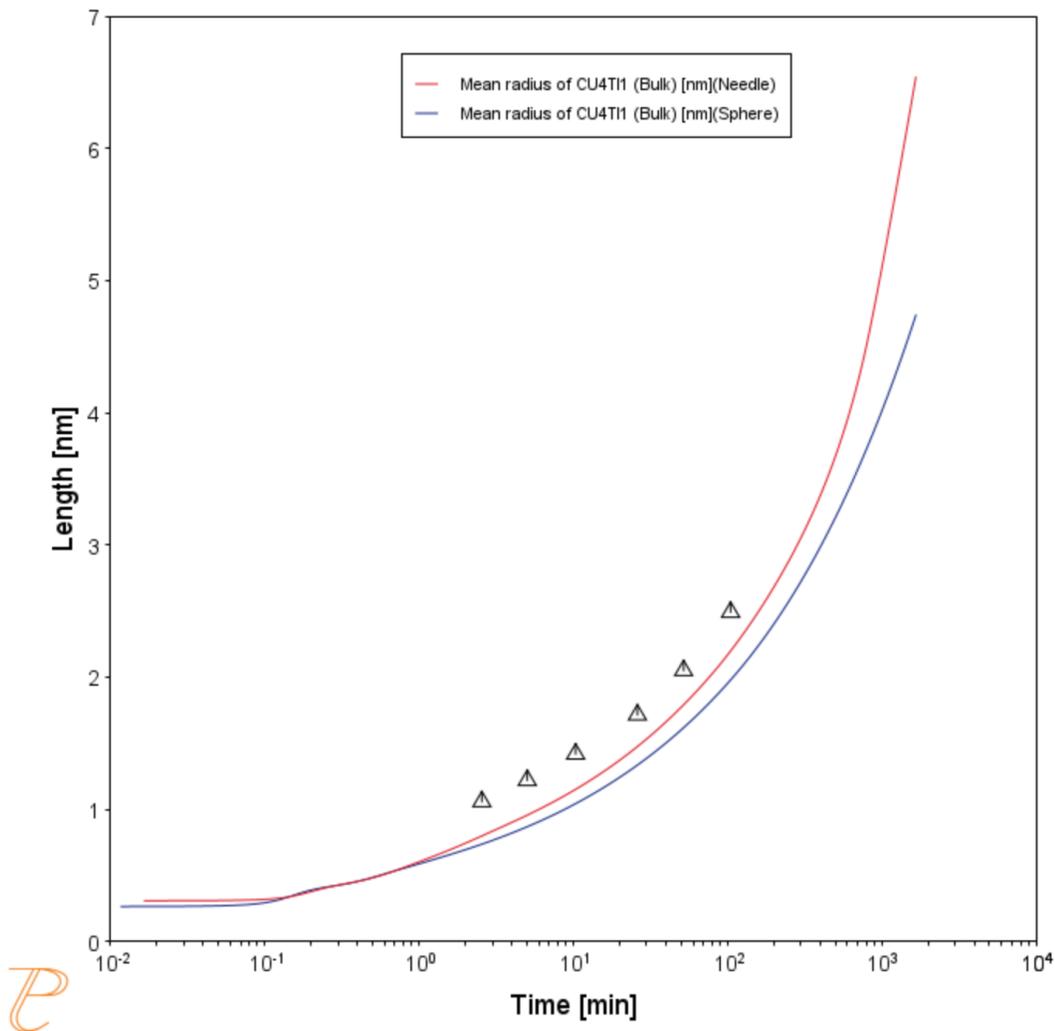


Figure 22: This shows the approximates the size of the needle shaped precipitates as a sphere of equivalent volume, and compares the experimental results of [1987Kam] with those obtained from a spherical and needle shaped approximation.

## References

- [1987Kam] R. Kampmann, H. Eckerlebe, R. Wagner, 1987. "Precipitation Kinetics in Metastable Solid Solutions - Theoretical Considerations and Application to Cu-Ti Alloys." Mat. Res. Soc. Symp. Proc. 57: 525-542.
- [1999Bor] C. Borchers, Catastrophic nucleation during decomposition of Cu-0.9at.% Ti. Philos. Mag. A. 79, 537-547 (1999).

## P\_09: Precipitation of Al-Sc AL3SC with Assumption of Sphere and Cuboid Morphologies

In this isothermal calculation example, the precipitation of Al<sub>3</sub>Sc phase from FCC\_A1 matrix phase in an Al-Sc binary alloy is simulated. To make a comparison, two separate calculations are performed, one assuming spherical morphology without elastic strain energy, and the other assuming cuboid morphology whose shape is determined by competition between interfacial energy and elastic strain energy. The simulation results are compared with experimental data collected from Marquis and Seidman [2001Mar] and Novotny and Ardell [2001Nov]. In addition, mean cubic factor and cubic factor distribution are also plotted for cuboid shape to illustrate the spherical-cuboidal transition during precipitation.



For more details about the background theory, see [Precipitation Morphology](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### ***Project File, Step-By Step Instructions, and Video tutorial Information***

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_09\_Precipitation\_Al-Sc\_AL3SC\_Sphere\_Cuboid.tcu*



This example is included as a Precipitation Module (TC-PRISMA) tutorial on our [website](#) and as part of the playlist on our [YouTube channel](#).



You can also use [the step-by-step instructions](#) included in a PDF to follow the video or compare to the project file in Thermo-Calc.

## **Example Settings**



To ensure that the settings are done on the correct Precipitation Calculators, the **Sphere** and **Cuboid** nodes are renamed from **Precipitation Calculator** to match their morphology. The morphology is set in the **Precipitate Phase** section when you click **Show Details**. See P\_08 for an example of this.

**System (System Definer)**

Database package	Demo: Aluminum-based alloys (ALDEMO, MALDEMO)
------------------	---

Elements	Al, Sc
----------	--------

**Sphere and Cuboid Conditions (Precipitation Calculator)**

Composition	Al-0.18Sc Mole percent
-------------	------------------------

Matrix phase	FCC_A1
--------------	--------

Precipitate phase	AL3SC
-------------------	-------

**Matrix Phase Data Parameters (Precipitation Calculator)**

Elastic properties (click <b>Show Details</b> to display this setting)	<p>For the <b>Sphere</b> node (renamed from Precipitation Calculator), the default, <b>Disregard</b> is kept.</p> <p>For the <b>Cuboid</b> node (renamed from Precipitation Calculator), choose <b>Cubic</b>. Then enter the elastic constants accordingly. Default elastic constants are given based on the major element of the alloy system. In this example that is</p> <ul style="list-style-type: none"> <li>• c11 is <b>108.2</b> GPa</li> <li>• c12 is <b>61.3</b> GPa</li> <li>• c44 is <b>28.5</b> GPa</li> </ul>
--	---

**Precipitate Phase Data Parameters (Precipitation Calculator)**

Nucleation sites	Bulk
------------------	------

Interfacial energy	The default
--------------------	-------------

Morphology (click <b>Show Details</b> to display this setting)	<p>For the <b>Sphere</b> node (renamed from Precipitation Calculator), keep the default.</p> <p>For the <b>Cuboid</b> node (renamed from Precipitation Calculator), <b>Cuboid</b> is selected.</p>
--	--

Transformation strain (click <b>Show Details</b> to display this setting)	<p>For the <b>Sphere</b> node (renamed from Precipitation Calculator), keep the default.</p> <p>For the <b>Cuboid</b> node (renamed from Precipitation Calculator), <b>Calculate from molar volume</b> is selected to obtain a purely dilatational strain.</p>
---	--

**Calculation Type (Precipitation Calculator)**

Calculation type	Isothermal
------------------	------------

Temperature	350° C
-------------	--------

Simulation time	1.0E9 seconds
<b>Datasets (Experimental File Reader)</b>	
Dataset 1 and Dataset 2	Data sets included with this example and imported to one Experimental File Reader. It is used for the Mean Radius plot.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the **Precipitation Calculator** tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



In this example, all nodes are renamed in the **Project** window, and this matches each of the tab names in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

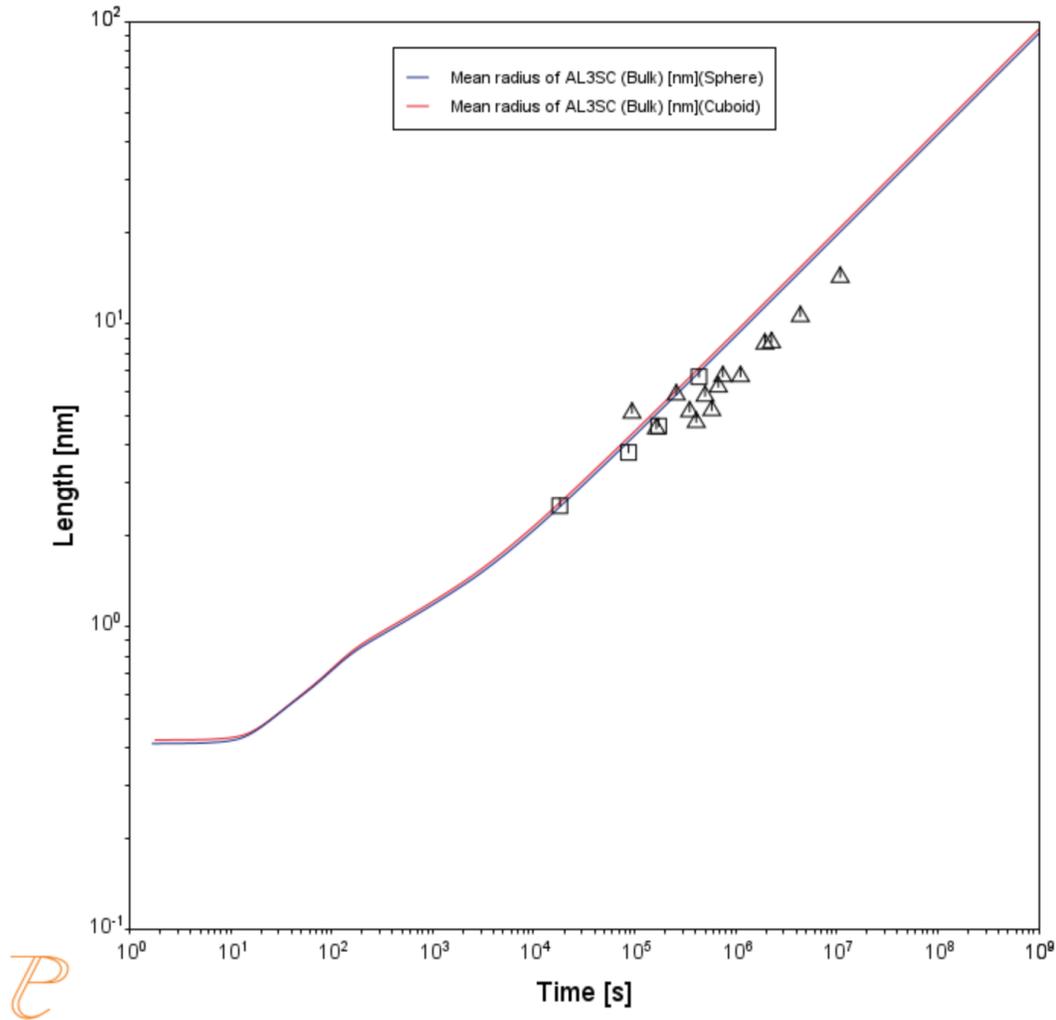


Figure 23: This shows the predicted evolution of the mean size of precipitates with the experimental data from [2001Mar, 2001Nov], comparing the spherical and cuboidal shaped approximations of the precipitate morphology. Although there is not significant difference in the results, capturing the correct geometry of the precipitate is important in subsequent calculations of precipitate spacing used in property calculations.

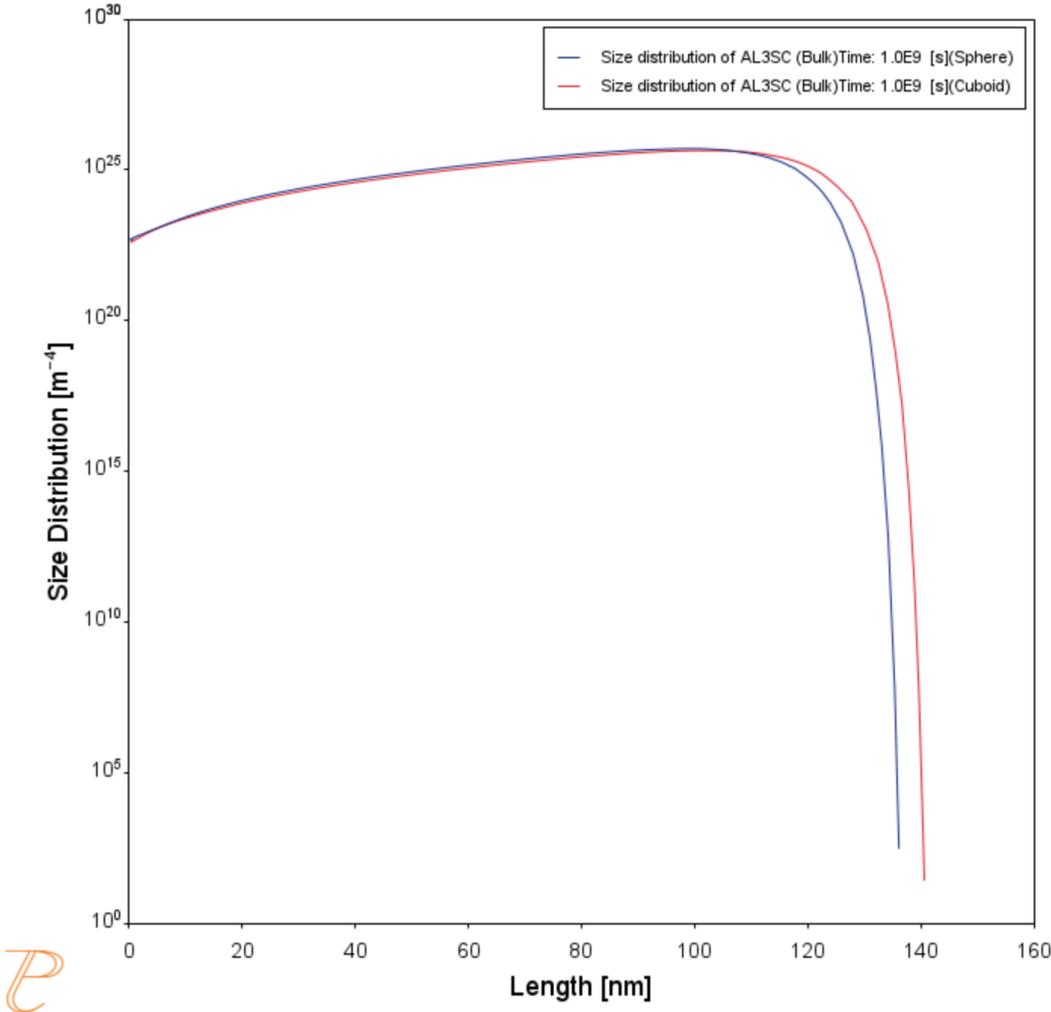


Figure 24: These results compare the predicted size distributions after heat treatment considering the spherical and cuboidal approximations of the precipitate morphology.

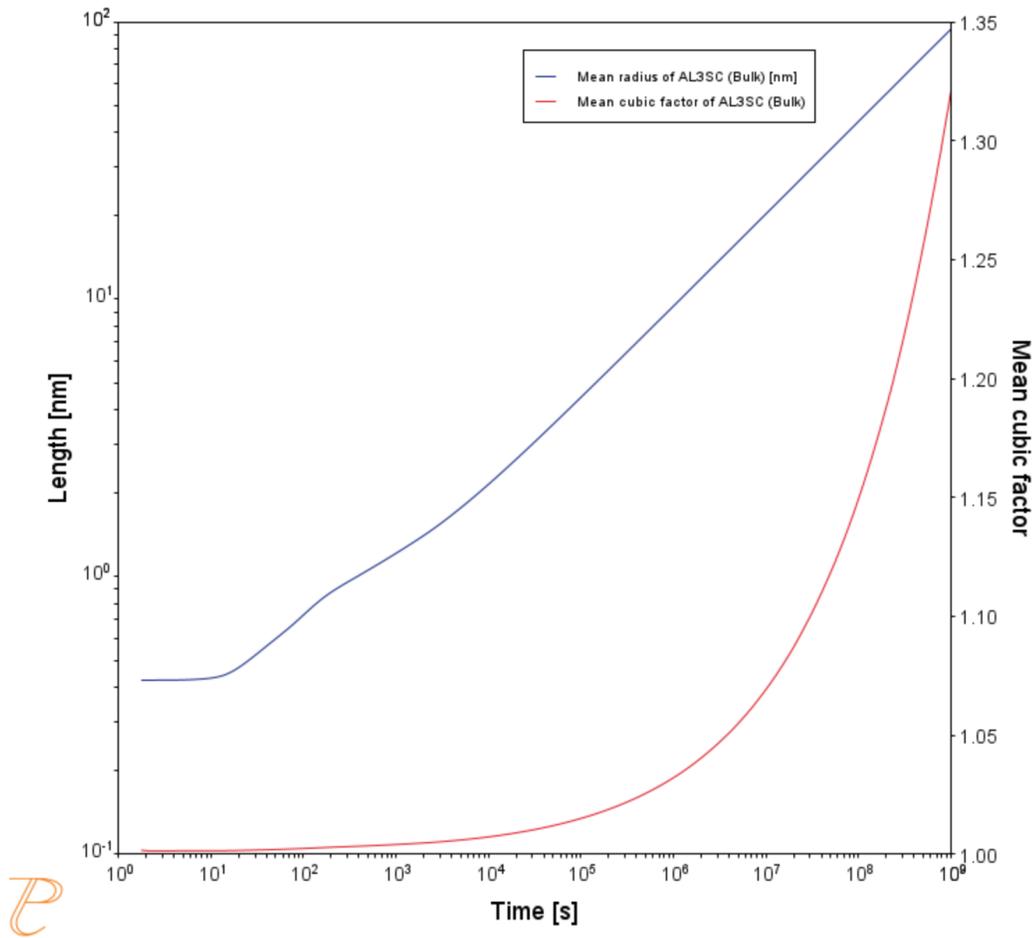


Figure 25: These results compare the predicted evolution of the mean particle radius and mean aspect ratio from using the cuboidal approximation of the precipitate morphology. In this case, the radius of a sphere of equivalent volume to the cuboidal precipitate is shown.

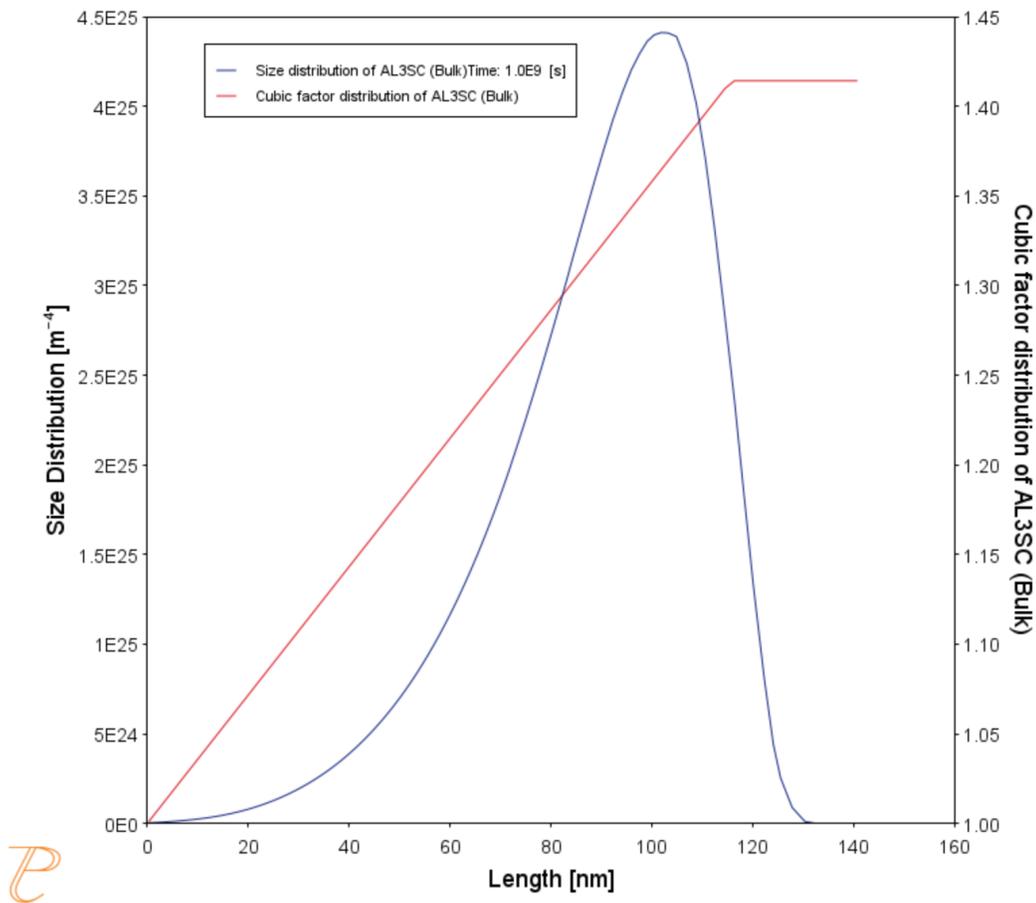


Figure 26: These results show the predicted size distribution of the precipitates obtained from the cuboidal approximation of the precipitate morphology. Also shown is the calculated size dependent cubic factor of the cuboids associated with this dispersion.

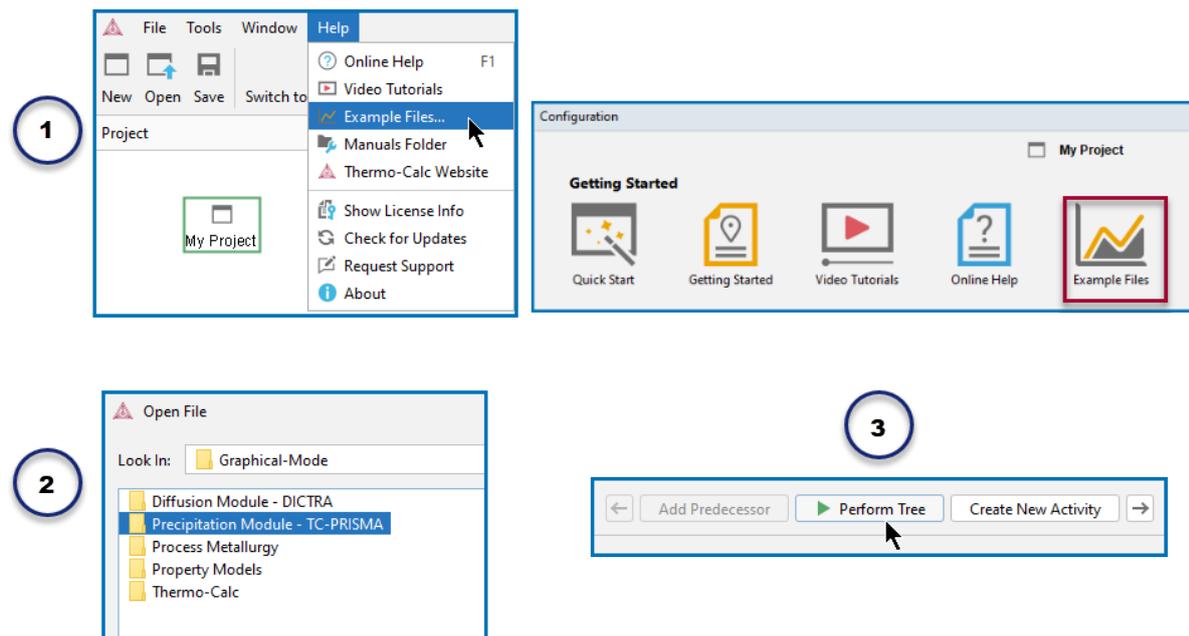
## References

- [2001Mar] E. A. Marquis, D. N. Seidman, Nanoscale structural evolution of Al<sub>3</sub>Sc precipitates in Al(Sc) alloys. *Acta Mater.* 49, 1909–1919 (2001).
- [2001Nov] G. M. Novotny, A. J. Ardell, Precipitation of Al<sub>3</sub>Sc in binary Al–Sc alloys. *Mater. Sci. Eng. A Struct. Mater. Prop. Microstruct. Process.* 318, 144–154 (2001).

# Precipitation of Al-Sc AL3SC with Assumption of Sphere and Cuboid Morphologies: Precipitation Example P\_09

## HELPFUL INFORMATION

- All users can run this calculation, even those who do not have a license for the Precipitation Module (TC-PRISMA).
  - A companion video is available for this example, which can be watched here: [https://www.youtube.com/playlist?list=PLfv6McToaTGSpqvUoY3b\\_UV-8xpgLUKj](https://www.youtube.com/playlist?list=PLfv6McToaTGSpqvUoY3b_UV-8xpgLUKj)
  - This calculation is based on Precipitation Module example *P\_09 – Precipitation\_Al-Sc\_AL3SC\_Sphere\_Cuboid*, which is included in your installation.
1. To run the example file, open Thermo-Calc and select **Help > Examples Files** or click **Example Files** on the **My Project Configuration** window.
  2. Open the *Precipitation Module (TC-PRISMA)* folder. Double-click the example file to open it.
  3. In Thermo-Calc at the bottom center of the **Configuration** window click **Perform Tree**.



## ABOUT THE EXAMPLE

This example shows you how to calculate the precipitation of Al<sub>3</sub>Sc phase from FCC\_A1 matrix phase in an Al-Sc binary alloy using the Precipitation Module (TC-PRISMA).

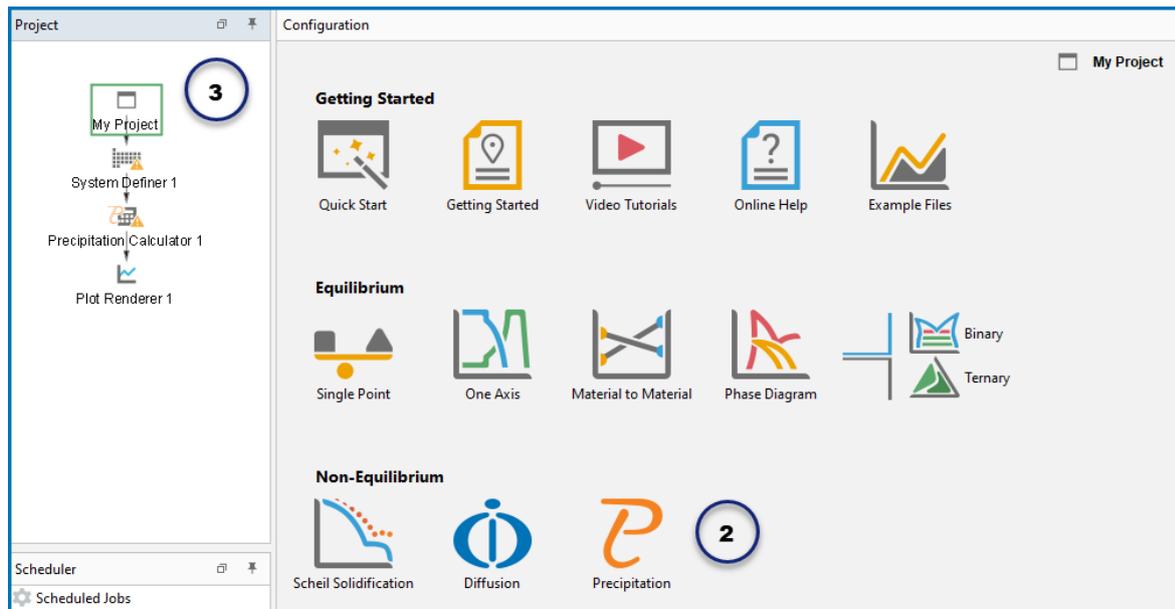
Two separate calculations are performed so you can make a comparison of the results, one assuming spherical morphology without elastic strain energy, and the other assuming cuboid morphology whose shape is determined by competition between interfacial energy and elastic strain energy.

In addition, mean cubic factor and cubic factor distribution are plotted for cuboid shape to illustrate the spherical-cuboidal transition during precipitation.

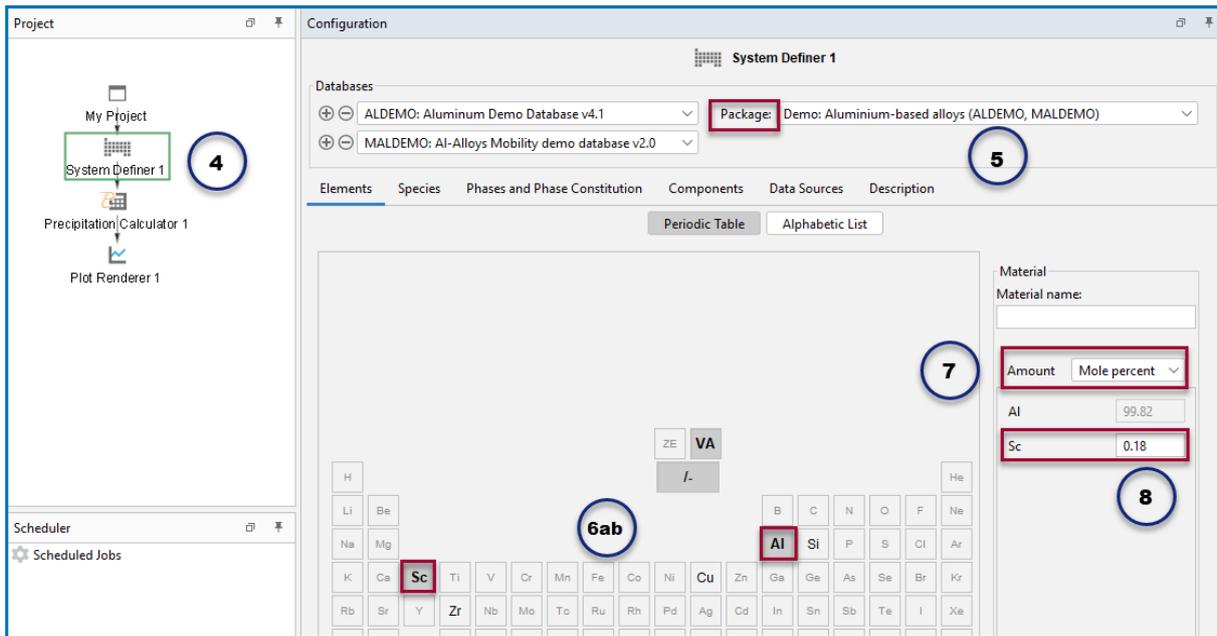
If you run the example file included in your software, the simulation results are compared with experimental data collected from Marquis and Seidman (2001) and Novotny and Ardell (2001).

## SETTING UP THE SYSTEM

1. Open Thermo-Calc in Graphical Mode.
2. On the **My Project Configuration** window under **Non-Equilibrium**, click **Precipitation**.
3. All the nodes for a precipitation calculation are added to the **Project** window:



4. In the **Project** window, click the **System Definer 1** node.
5. Set the database *Package* to **Demo: Aluminium-based alloys (ALDEMO, MALDEMO)**, which loads both thermodynamic and kinetic demonstration aluminium databases.
6. From the **Periodic Table**, select the elements in the following order.  
Select Al first so that it is the dependant element:
  - a. **Al** (aluminium)
  - b. **Sc** (scandium).
7. From the *Amount* list (to the right of the Periodic Table), select **Mole percent**.
8. Enter 0.18 for **Sc**. This automatically sets **Al** to 99.82.



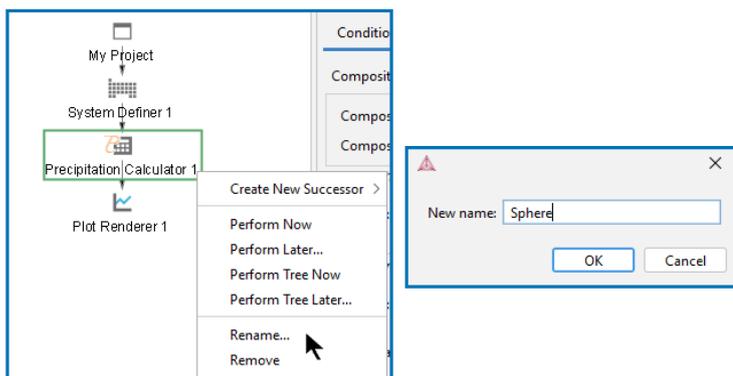
Our system is now defined. Now set up the precipitation calculations.

## SETTING UP THE PRECIPITATION CALCULATIONS

There are two precipitation calculations in this example, one assuming spherical morphology without elastic strain energy, and the other assuming cuboid morphology whose shape is determined by competition between interfacial energy and elastic strain energy. The results are then compared for both calculations.

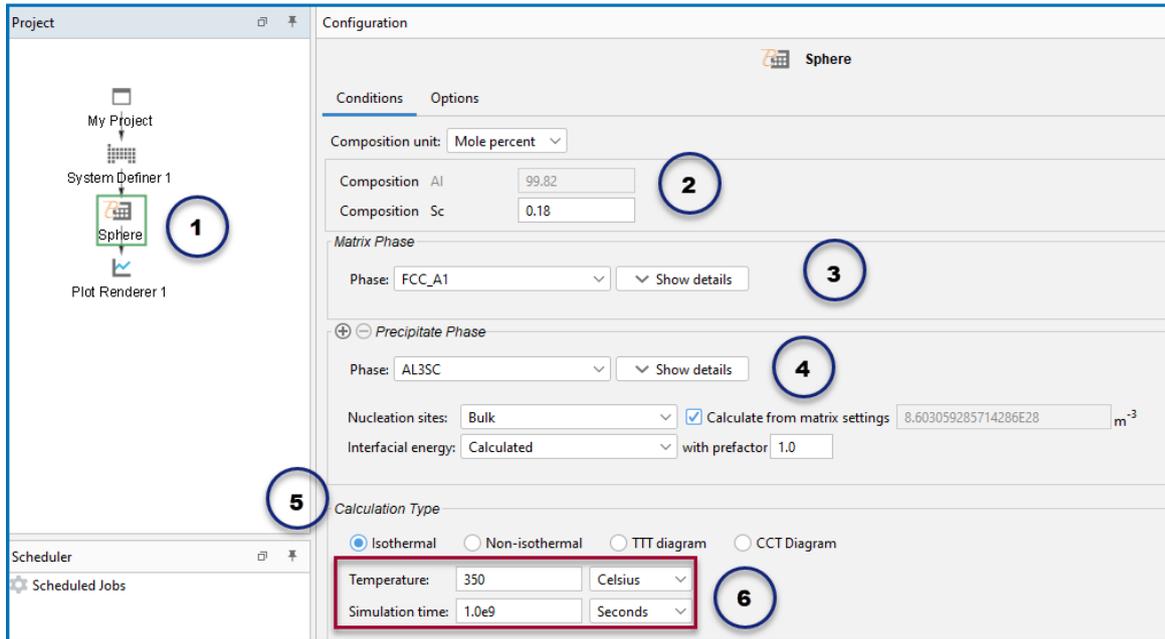
### Setting up the Precipitation Calculation with Spherical Morphology

1. In the **Project** window, right-click the **Precipitation Calculator 1** node and select **Rename**. Enter *Sphere*, then press enter or click **OK**.



2. Notice that the composition set in the *System Definer* auto-populated on the Precipitation Calculator.
3. Under *Matrix Phase* from the *Phase* list it defaults to **FCC\_A1**.
4. Under *Precipitate Phase* from the *Phase* list select **AL3SC**.
5. Under *Calculation Type*, make sure **Isothermal** is selected, which means that the same temperature is maintained throughout the calculation.
6. Enter
  - a. 350 as the *Temperature* and select **Celsius**.

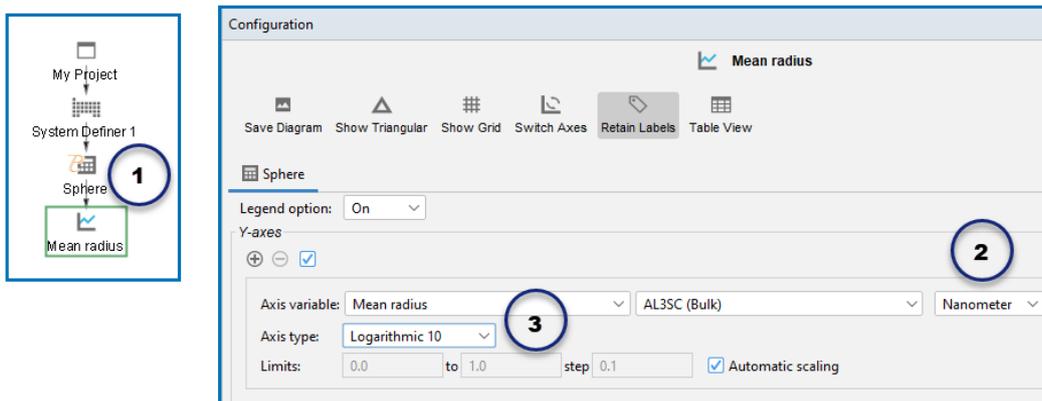
b.  $1.0 \times 10^9$  as the *Simulation time* and select **Seconds**.



Our first calculation is set, so next configure the plot.

### Configuring the Plot

1. In the **Project** window, right-click **Plot Renderer 1** and select **Rename**. Name the plot **Mean radius** then press enter or click **OK**.
2. Under **Y-axes** change the units to **Nanometer (nm)**.
3. From the **Axis type** list select **Logarithmic 10**.

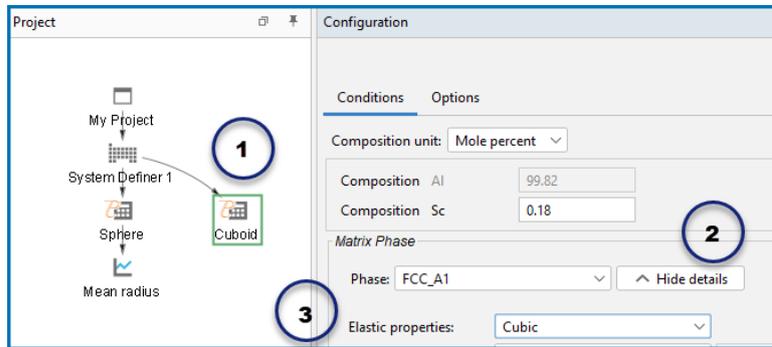


The plot is now set, but before running the simulation you set up another calculation and link it to the same Plot Renderer.

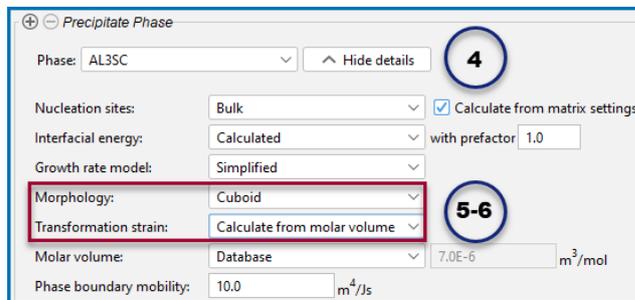
### Setting up the Precipitation Calculation with Cuboid Morphology

1. In the **Project** window, right-click the **Sphere** node and select **Clone**.
  - a. Right-click the **Sphere 1** node and select **Rename**.
  - b. Name the node **Cuboid** and press enter or click **OK**.
2. On the **Configuration** window to the right of **Matrix Phase** click **Show Details**.

- From the *Elastic Properties* list select **Cubic**. Keep the suggested default values. Click **Hide details**.



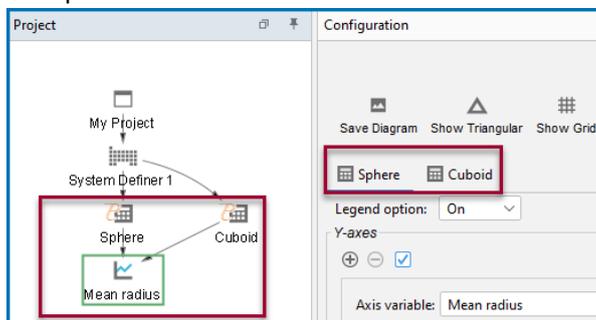
- To the right of *Precipitate Phase*, click **Show details**.
- From the *Morphology* list select **Cuboid**.
- From the *Transformation strain* list select **Calculate from molar volume**.



The second calculation is now set up, so now link it to the Plot Renderer node.

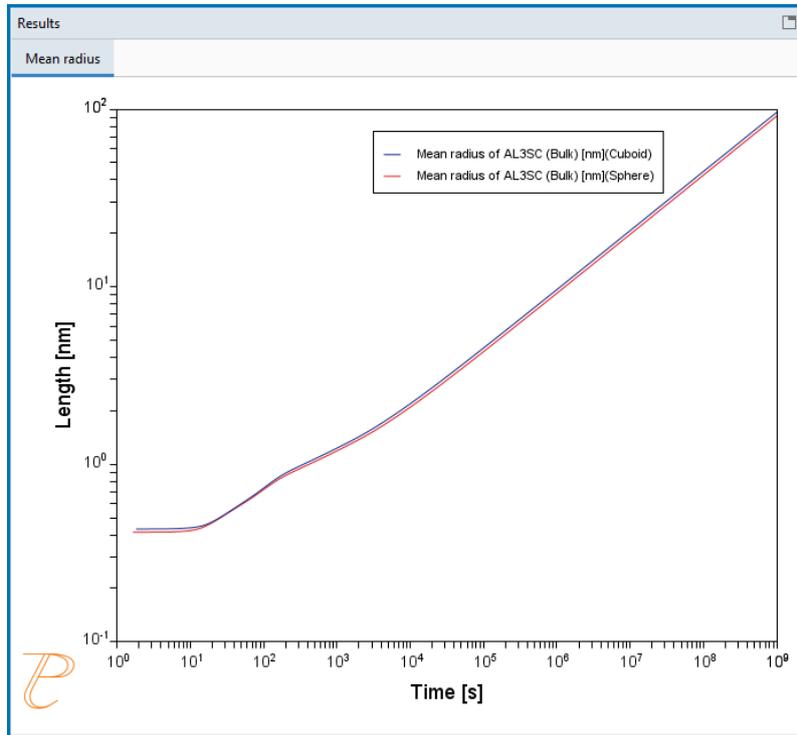
### **Linking the Plot to the Calculation with Cuboid Morphology**

- In the **Project** window, right-click the **Mean radius** node and select **Add Predecessor >Cuboid**.
- Notice that in the **Configuration** window there are two tabs associated with this plot, one for **Sphere** and one for **Cuboid**. The Sphere settings are already configured, so you just need to set up the Cuboid tab.



- Click the **Cuboid** tab.
- Under *Y-axes* change the units to **Nanometer (nm)**.
- From the *Axis type* list select **Logarithmic 10**.
- The calculation is now ready to run. Right-click the **Mean radius** node and select **Perform Now** or click **Perform** on the **Configuration** window.

## INTERPRETING THE RESULTS OF THE PRECIPITATION CALCULATIONS



In this plot, the blue line represents the sphere calculation and the red line represents the cuboid calculation. You can see that, in this instance, they are almost identical.

In this plot, the blue line represents the sphere calculation and the red line represents the cuboid calculation. You can see that, in this instance, these are almost identical.

By 'radius for non-spherical particles' it means the radius of equivalent spheres with the same volume.

If you run the example file that is included in your software, the plot also contains an experimental file, which you can see closely matches the calculations.

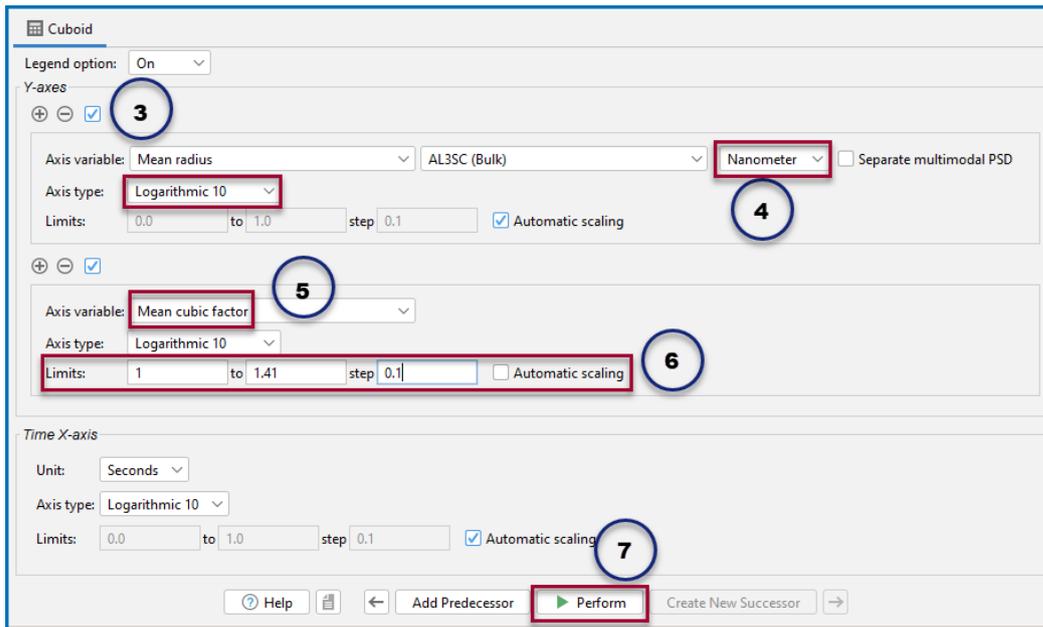
## PLOTTING ADDITIONAL VARIABLES

Now you use the same calculation to set up additional plots using the many available variables. The examples below include two Y-axes each.

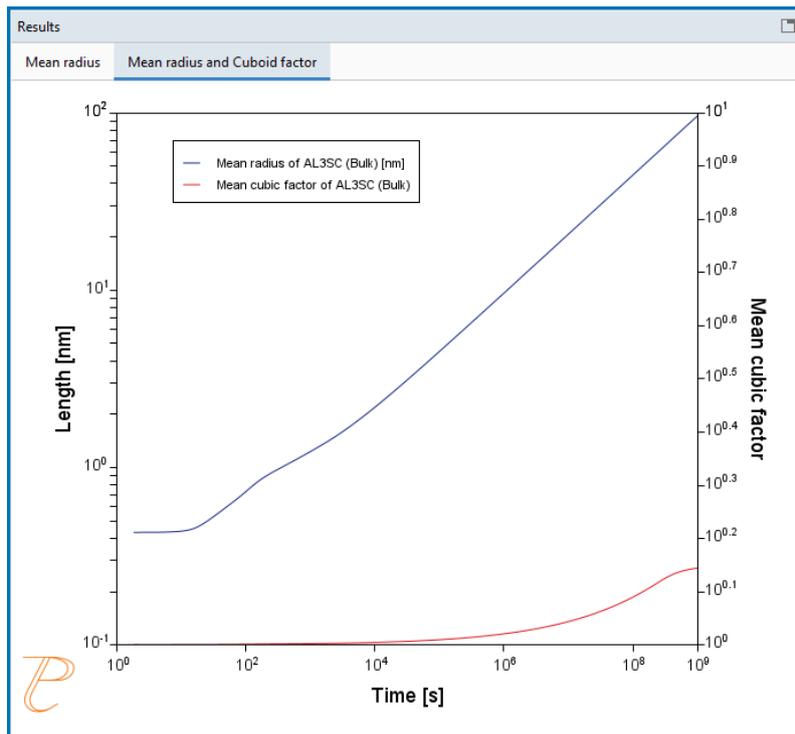
### ***Plotting Mean Radius and Cuboid Factor***

This plot has two Y-axes: *Mean radius* and *Cuboid factor*.

1. Right-click the **Cuboid** node and add a **Plot Renderer**.
2. Right-click the new node and rename it `Mean radius and Cuboid factor`.
3. Under *Y-axes* click the plus sign to add another axis.
4. Keep *Mean radius* as the first axis variable but change the units to **Nanometer (nm)** and set the axis type to **Logarithmic 10**.
5. Set the second *Axis variable* to **Mean cubic factor**.
6. Click to clear the *Automatic scaling* check box. In the fields, enter the *Limits* from 1 to 1.41 and the *step* to 0.1.
7. The plot is now set. Click **Perform**.



### Interpreting the Results of Mean Radius and Cuboid Factor



In this plot, the blue line represents the evolution of the average radius of the cubic particles as a function of time. The red line shows the average cubic factor as a function of time.

The blue line represents the evolution of the average radius of the cubic particles as a function of time. Again, for non-spherical precipitates this means the radius of equivalent spheres with the same volume.

The red line shows the average cubic factor as a function of time. A value of one represents a spherical shape. A square root of two represents a cubic shape.

The evolution of the shape when the particles grows is determined by competition between interfacial energy and elastic strain energy. In general, the shape is close to spherical at small particle sizes because the interfacial energy term dominates.

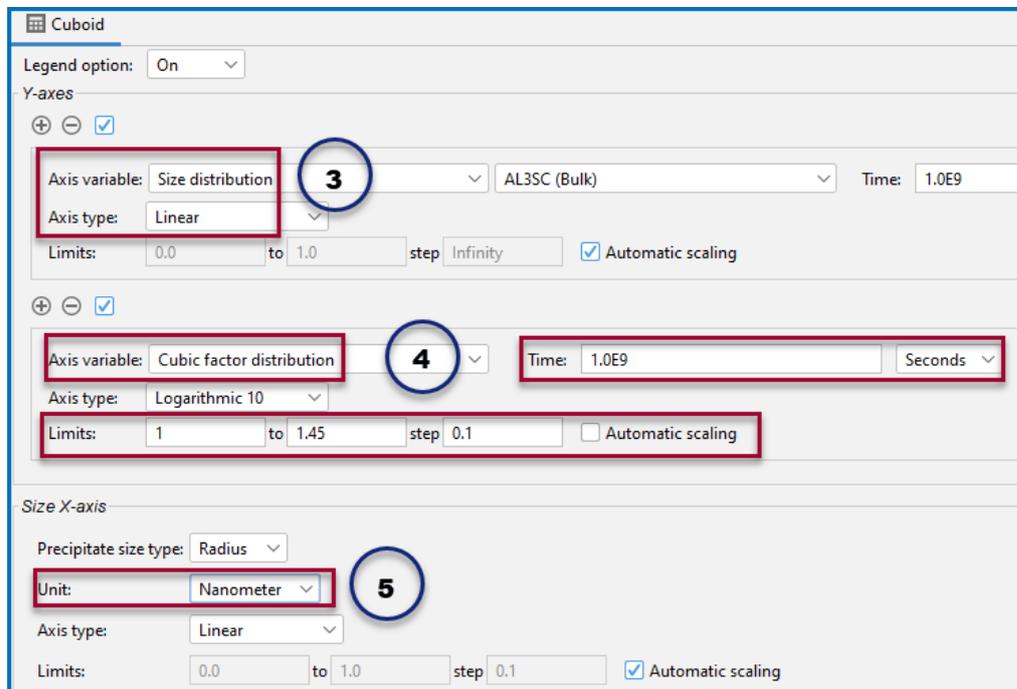
At large sizes the elastic energy dominates and it is therefore more favourable with a non-spherical shape. You can see that the particles get a more cubic form at later times when they grow to larger sizes.

### ***Plotting Particle Size and Cubic Factor Distribution***

This final plot shows the particle size distribution (PSD) and cubic factor distribution at the end of the simulation.

This plot also has two Y-axes, so you can clone the previous plot.

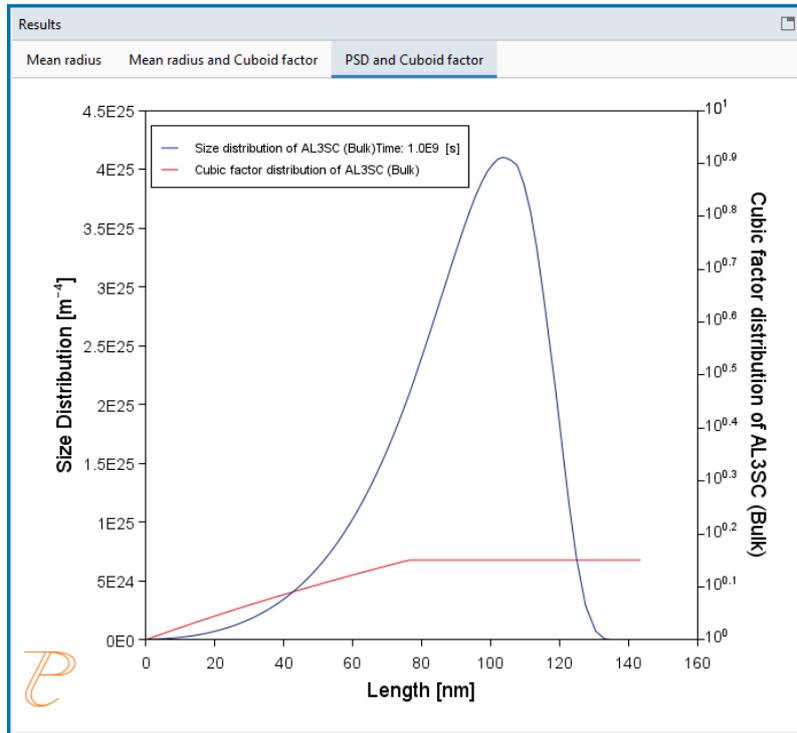
1. Right-click the **Mean radius and Cuboid factor** node and select **Clone**.
2. Right-click the new node and rename it **PSD and Cuboid factor**.
3. Under the *Y-axes*
  - a. Select **Size distribution** as the first *Axis variable*.
  - b. Change the *Axis type* to **Linear**.
4. Set the second Y-axis to **Cubic factor distribution** and enter a *Time* of **1e9 Seconds**.
5. Keep the *Automatic Scaling* check box cleared. Enter the *Limits* from 1 to 1.45 and the *step* to 0.1.
6. Under *Size X-axis*, change the unit to **Nanometer (nm)**.
7. Click **Perform** at the bottom center of the **Configuration** window.



The screenshot shows the 'Cuboid' configuration window with the following settings:

- Legend option:** On
- Y-axes:**
  - Axis 1: **Size distribution** (Axis variable), **Linear** (Axis type), Limits: 0.0 to 1.0, step: Infinity,  Automatic scaling.
  - Axis 2: **Cubic factor distribution** (Axis variable), **Logarithmic 10** (Axis type), **1.0E9** (Time), **Seconds** (Unit), Limits: 1 to 1.45, step: 0.1,  Automatic scaling.
- Size X-axis:**
  - Precipitate size type: Radius
  - Nanometer** (Unit)
  - Axis type: Linear
  - Limits: 0.0 to 1.0, step: 0.1,  Automatic scaling.

### ***Interpreting the Results of Particle Size Distribution and Cuboid Factor***



In this plot, the blue curve shows the particle size distribution and the red line shows the cubic factor distribution.

The blue curve, which shows the particle size distribution, is close to the regular LSW size distribution that is expected for spherical particles.

The cubic factor distribution shows that the smallest particles are closer to spherical and that the larger ones get more and more cubic.

### ***References***

1. Marquis, E.A, and D.N Seidman. 2001. "Nanoscale Structural Evolution of Al<sub>3</sub>Sc Precipitates in Al(Sc) Alloys." *Acta Materialia* 49 (11): 1909–19.
2. Novotny, Gabriel M., and Alan J. Ardell. 2001. "Precipitation of Al<sub>3</sub>Sc in Binary Al–Sc Alloys." *Materials Science & Engineering, A: Structural Materials: Properties, Microstructure and Processing* 318 (1–2): 144–54.

## P\_10: Initial Particle Size Distribution of Fe-Cr-C

---

This example demonstrates the effect of initial particle size distribution (PSD) of the precipitate phases on the overall precipitation kinetics. It uses two Precipitation Calculators to simulate and compare carbide precipitations from a ferritic BCC\_A2 matrix in a Fe-0.1C-12Cr alloy. Three carbides, CEMENTITE, M23C6, and M7C3, are included in the calculations for competitive precipitations, and the precipitation kinetics are compared with or without initial particle size distribution.

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Project File Information**

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_10\_Precipitation\_Initial\_PSD\_FeCrC.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

### **Example Settings**

The example illustrates the use of the PSD setting. You can import data from a spreadsheet or text file (.xls, .xlsx, .csv or .txt formats are acceptable). The **Preexisting Particle Size Distribution** window shown below, provides a graphical representation of the radius versus corresponding frequencies.



[Particle Size Distribution \(PSD\)](#)

---

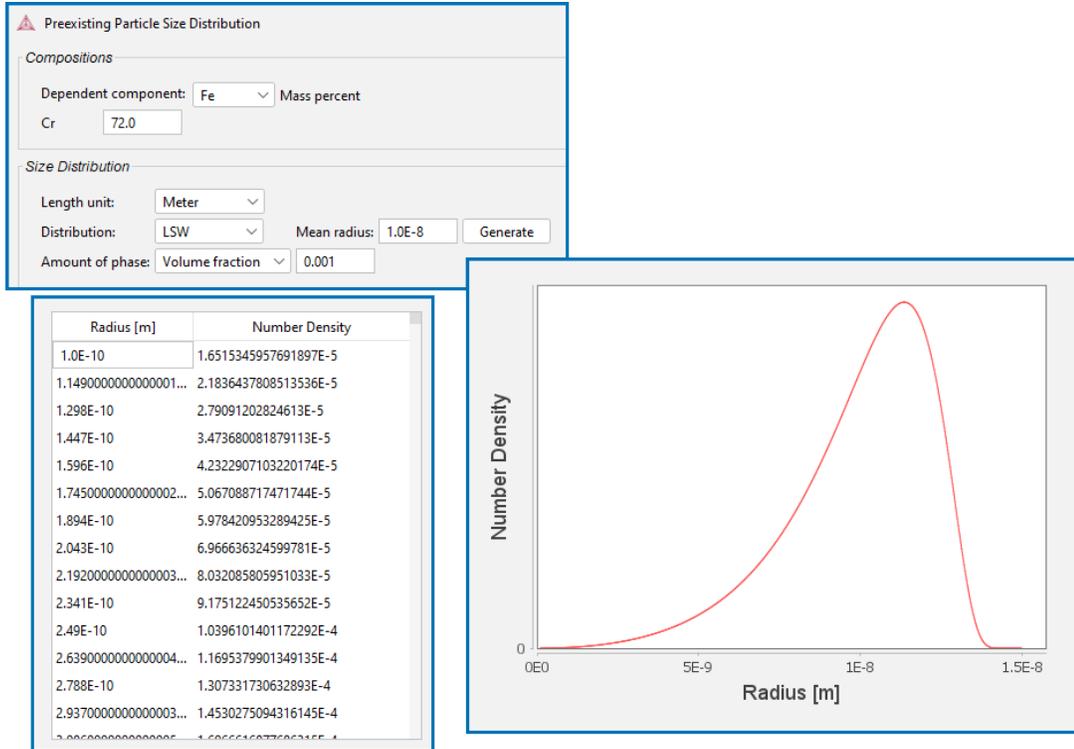


Figure 27: The Preexisting Particle Size Distribution (PSD) settings window for example P\_10.

System (System Definer)	
Database package	Demo: Steels and Fe-alloys (FEDEMO and MFEDEMO)
Elements	Fe, C, Cr
Conditions (Precipitation Calculator)	
Composition	Fe-0.1C-12Cr Mass percent
Matrix phase	BCC_A2 All other defaults are kept.
Precipitate phases	CEMENTITE, M23C6 and M7C3
Precipitate Phase Data Parameters (Precipitation Calculator)	
Nucleation sites	Grain boundaries (all calculations): Calculated from the matrix settings with a wetting angle of 90°
Interfacial energy	User-defined function f(r,T) (all calculations): <ul style="list-style-type: none"> <li>CEMENTITE: 0.167 J/m<sup>2</sup></li> </ul>

	<ul style="list-style-type: none"><li>• M23C6 0.252 J/m<sup>2</sup></li><li>• M7C3 0.282 J/m<sup>2</sup></li></ul>
Preexisting size distribution (click <b>Show Details</b> to display this setting)	<p>For the Precipitation Calculator including particle size distribution, and for all precipitate phases, this checkbox is selected.</p> <p>For each precipitate phase (CEMENTITE, M23C6 and M7C3), click <b>Edit particle size distribution</b> to make changes to the parameters. A window opens with a graphical representation of the radius vs number density.</p>
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal
Temperature	1053 K
Simulation time	400 000 seconds

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.

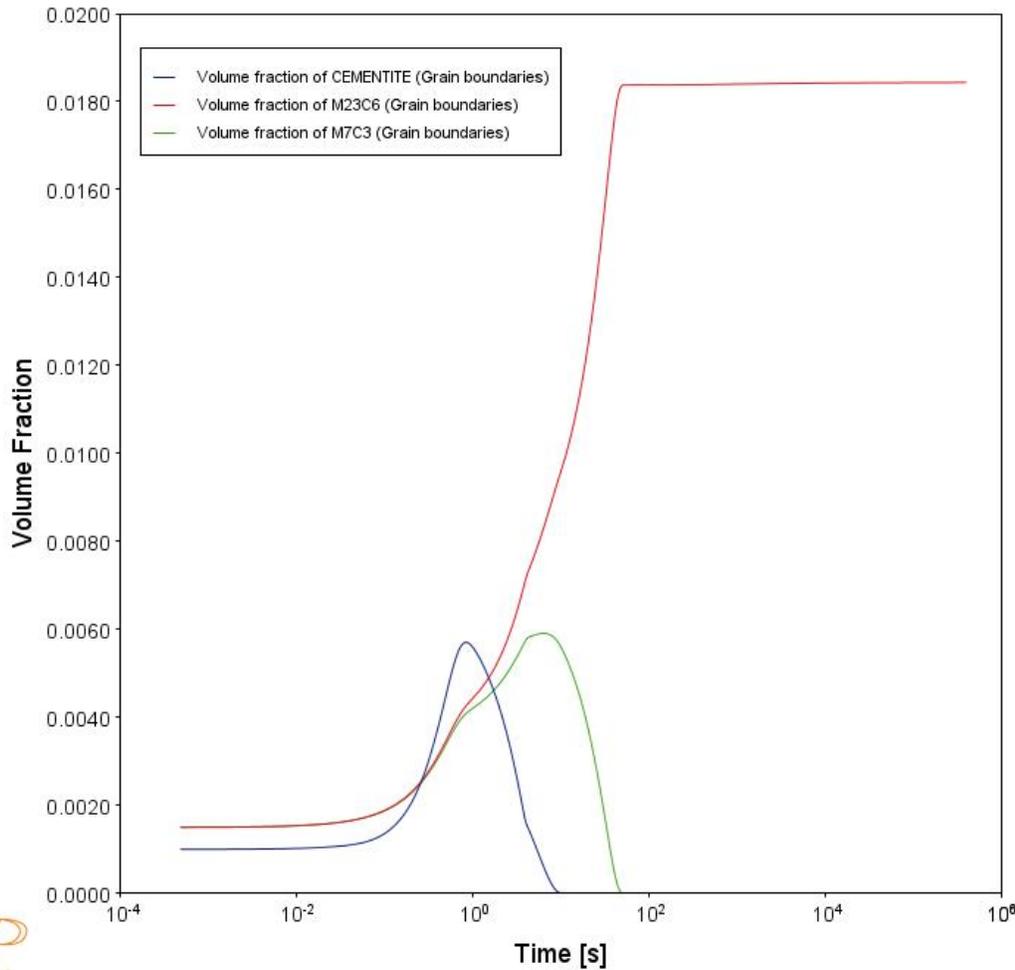


For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

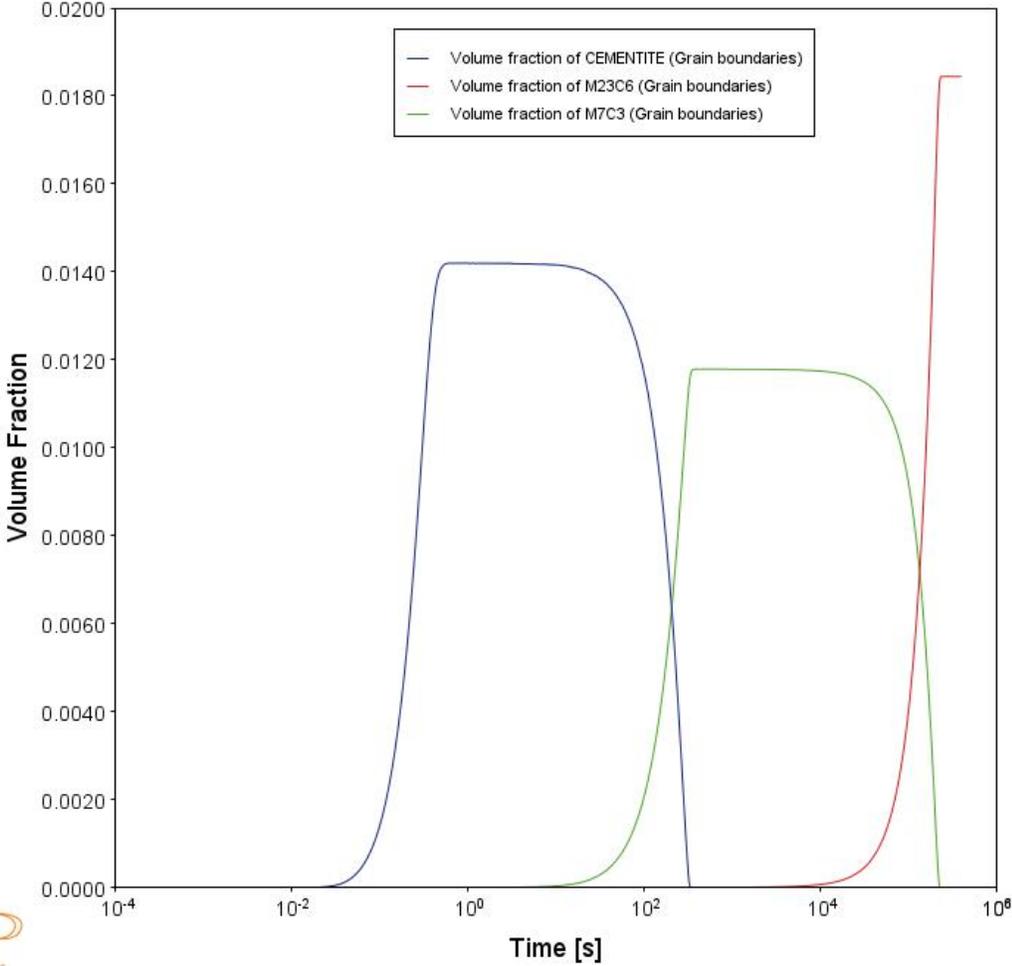
- **Plot or Table results:** After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.



*Figure 28: These results show the predicted volume fraction of cementite, M23C6, and M7C3 during an isothermal aging with an initial particle size distribution of each precipitate phase. The predicted kinetics are sensitive to the initial size, volume fraction, and composition of these precipitates.*



P

Figure 29: The predicted volume fraction of cementite, M23C6, and M7C3 during an isothermal aging with no precipitates in solution in the initial condition.

## P\_11: Interfacial Energy Function

In some cases, interfacial energy may be a function of temperature and/or particle radius. This example uses four Precipitation Calculators at four temperature points in 30 K increments: 673 K, 703 K, 733 K, and 763 K. It is an isothermal calculation to examine the mean radius of an Al-0.12Sc system. It uses an FCC\_A1 matrix phase and AL3SC precipitate phase with bulk nucleation sites and user-defined interfacial energy function. The user defined interfacial energy function uses an error function to set a smooth transition of the interfacial energy from 0.065 J/m<sup>2</sup> to 0.085 J/m<sup>2</sup> for particle radii below and above 1e<sup>-8</sup>m and 5e<sup>-8</sup>m, respectively.

A dataset based on Iwamura and Miura [2004Iwa] data is compared with the calculated results.



[Interfacial Energy Anisotropy](#)

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_11\_Interfacial\_energy\_function.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Example Settings

### System (System Definer)

Database package

Demo: Aluminum-based Alloys (ALDEMO, MALDEMO)

Elements	Al, Sc
<b>Conditions (Precipitation Calculator)</b>	
Composition	Al-0.12Sc Mole percent
Matrix phase	FCC_A1 All other defaults are kept.
Precipitate phase	AL3SC Nucleation sites (all calculations): Bulk (6.025E28 m <sup>-3</sup> ) Interfacial energy (all calculations): User-defined function f(r,T): 0.075+0.011*erf((r-3e-8)/1e-8 J/m <sup>2</sup> )
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal (all calculations)
Temperature	Four temperature points in 30 K increments: 673 K, 703 K, 733 K, and 763 K.
Simulation time	1 000 000 seconds (all calculations)
<b>Datasets (Experimental File Reader)</b>	
Wamura 2004 (Dataset 1)	Data set included with this example and imported to one Experimental File Reader.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

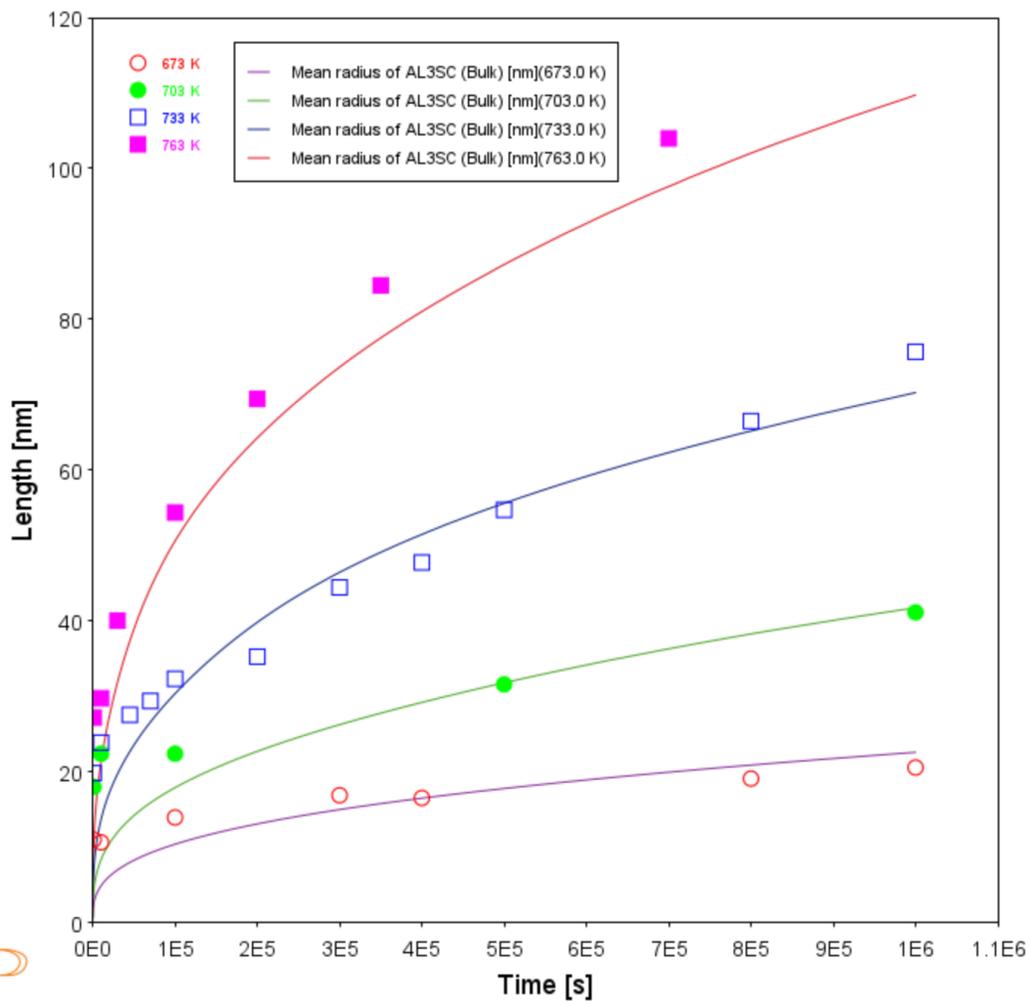


Figure 30: The results of an isothermal calculation to examine the mean radius of an Al-0.12Sc system with experimental data from [2004Iwa].

## Reference

[2004Iwa] Iwamura, S, and Y Miura. 2004. "Loss in Coherency and Coarsening Behavior of Al<sub>3</sub>Sc Precipitates." *Acta Materialia* 52 (3): 591–600.

---

## P\_12: Comparing Growth Rate Models for an Al-Zr System

This example compares the **Simplified**, **General**, and **Advanced** growth rate models for an Al-Zr system. The resulting plot compares the mean radius of the spheres for each AL3ZR\_D023 precipitate phase calculated for each type of growth rate model.

All models treat a spherical particle (precipitate) of stoichiometric composition or with negligible atomic diffusivity. Local equilibrium at the precipitate-matrix interface is assumed.

When you use the *Advanced* model, the velocity of a moving phase interface and the operating tie-line are solved together from flux-balance equations. This model can treat both high supersaturation and cross diffusion rigorously. It can also capture the transition between NPLE (non-partitioning local equilibrium) and PLE (partitioning local equilibrium) without any *ad hoc* treatment.

The *Simplified* model is based on the quasi-steady state diffusion approximation, and estimates solute partitioning with matrix composition and nuclei composition instead of time-consuming stepwise tie-line calculations. It also neglects cross diffusion for simplicity.

The *General* model can be considered the same theoretical approximation as, but an improvement over, the *Simplified* model, with cross-diffusion terms taken into account, as well as adjustment of Gibbs-Thomson effect and effective diffusivity implemented. A dataset based on Knipling et al. [2008Kni] data is compared with the calculated results.



For more background information, see the theory described in [Growth](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_12\_Precipitation\_Al-Zr\_GrowthRateModel\_comparison.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Example Settings

System (System Definer)	
Database package	Demo: Aluminum-based Alloys (ALDEMO, MALDEMO)
Elements	Al, Zr
Conditions (Precipitation Calculator)	
Composition	Al-0.2Zr Mole percent
Matrix phase	FCC_A1 All other defaults are kept.
Precipitate phase	AL3ZR_D023 Click <b>Show Details</b> to select the <b>Growth rate model (Simplified, Advanced and General)</b> . All other defaults are kept.
Calculation Type (Precipitation Calculator)	
Calculation type	Isothermal
Temperature	425 Celsius
Simulation time	400 hours
Datasets (Experimental File Reader)	
2008 Knipling	Data set included with this example and imported to one Experimental File Reader.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the **Precipitation Calculator** tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

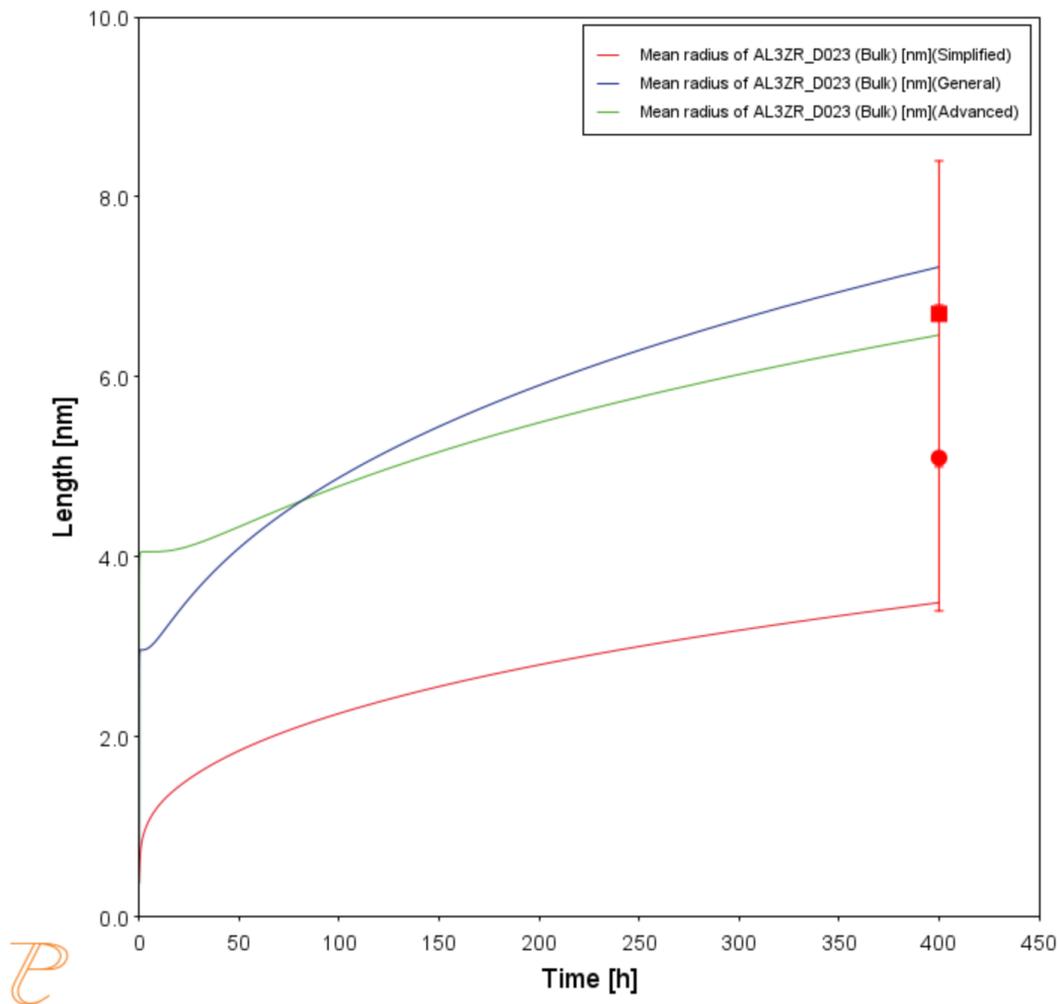


Figure 31: This shows the impact of the available growth rate settings options and compares the mean radius measurements of [2008Kni] with results obtained from the simplified, general, and advanced growth rate models. The experimental data includes measurements taken from different dendrites.

## Reference

[2008Kni] K. E. Knipling, D. C. Dunand, D. N. Seidman, Precipitation evolution in Al–Zr and Al–Zr–Ti alloys during isothermal aging at 375–425°C. *Acta Mater.* 56, 114–127 (2008).

## P\_13: Paraequilibrium Precipitation of Cementite Fe-C-Cr

In this example, the precipitation of cementite during tempering of a Fe-Cr-C steel is simulated considering two interface conditions: one is the usual ortho-equilibrium condition; the other is the para-equilibrium condition. The simulation results are compared with the experimental data from Sakuma et al. [1980Sak].

This example demonstrates that the early stage of the cementite precipitation can only be accounted for by a simulation applying the para-equilibrium condition, under which the precipitation kinetics are controlled by the diffusion of C. The comparison also shows a later stage gradual transition from the para-equilibrium condition to the ortho-equilibrium condition, and if the tempering time is long enough the diffusion of Cr has a dominating effect on the coarsening of cementite.



For more background information, see the theory described in [Growth](#).

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it can take over two hours to complete the calculations.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_13\_Precipitation\_Fe-C-Cr\_Paraequilibrium\_Precipitation\_of\_Cementite.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Example Settings

### System (System Definer)

Database package	Demo: Steels and Fe-alloys (FEDEMO, MFEDEMO)
------------------	--

Elements	Fe, Cr, C
<b>Conditions (Precipitation Calculator)</b>	
Composition	Fe-0.95Cr-1.065C Mass percent
Matrix phase	BCC_A2 All other defaults are kept.
Precipitate phase	CEMENTITE Click <b>Show Details</b> to select the <b>Growth rate model (Simplified and Para-eq)</b> . All other defaults are kept.
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal
Temperature	773 Kelvin
Simulation time	20 hours for the paraequilibrium model and 600 hours for the simplified model.
<b>Datasets (Experimental File Reader)</b>	
1980 Sakuma	Data set included with this example and imported to one Experimental File Reader.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- 
- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

---

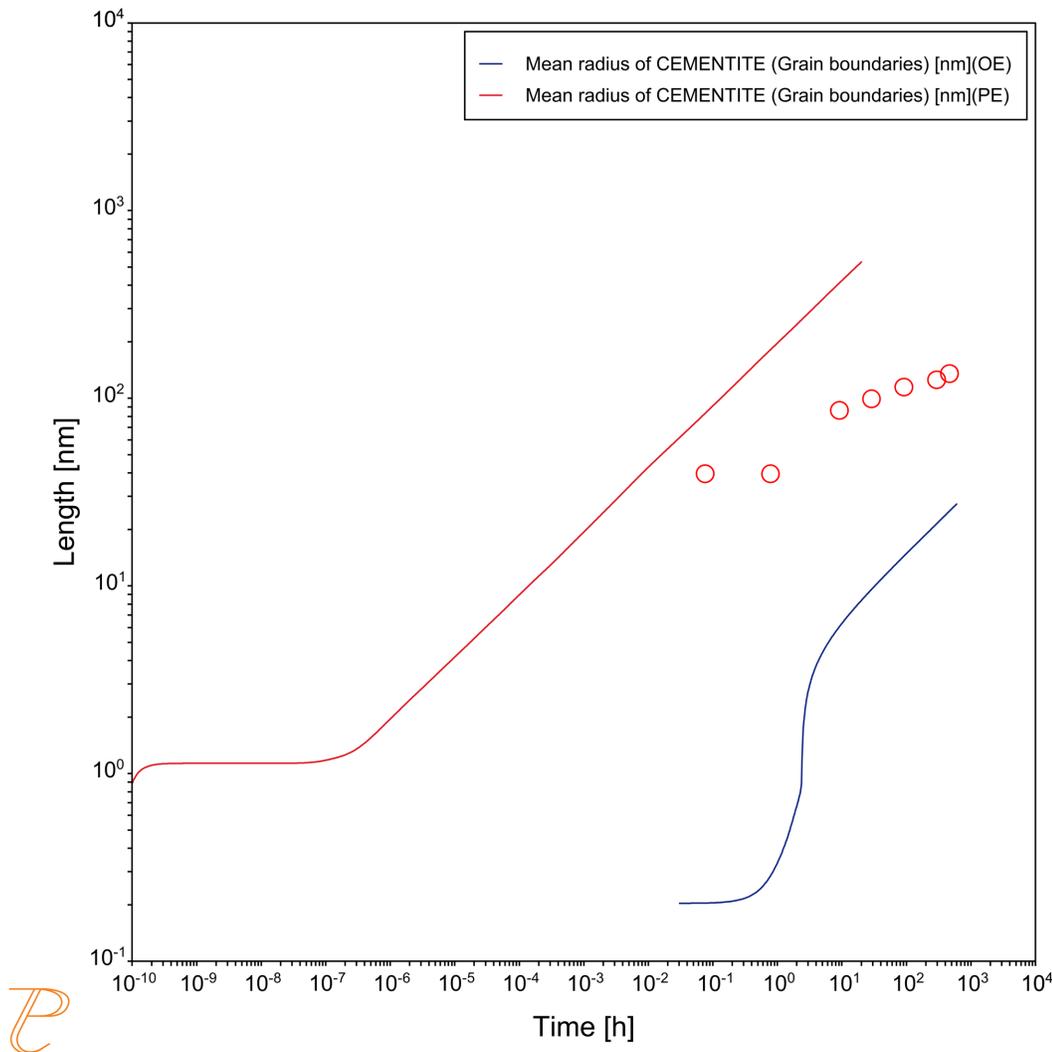


Figure 32: These results compare the predicted and measured evolution of the mean radius of cementite with the experimental data from [1980Sak] to simulation predictions obtained from the ortho-equilibrium (OE), and para-equilibrium (PE) approximations of the precipitate composition. The precipitate growth kinetics exhibited by cementite can be rationalized by the transition from PE to OE.

## Reference

[1980Sak] T. Sakuma, N. Watanabe, T. Nishizawa, The Effect of Alloying Element on the Coarsening Behavior of Cementite Particles in Ferrite. *Trans. Japan Inst. Met.* 21, 159–168 (1980).

## P\_14: Grain Growth and the Zener Pinning Effect

This example demonstrates the simulation of normal grain growth and the pinning effect [1948Smi; 1998Man] of precipitated second-phase particles on the grain boundary motion.

To investigate the grain growth and Zener pinning effect, an Fe-0.2C (wt.%) binary alloy, with a BCC\_A2 matrix phase and CEMENTITE precipitate phase, is simulated and uses the demonstration steel databases, FEDEMO and MFEDEMO. These databases are available to all users (i.e. you do not need a license for the Precipitation Module (TC-PRISMA)) and contain the necessary thermodynamic and kinetic data needed for the calculation.



Open and run this example in Thermo-Calc to follow along with the description and to experiment with the available settings.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_14\_Precipitation\_Fe-C-Ferrite-Grain\_Growth\_with\_Zener\_Pinning.tcu*



This example is included as a Precipitation Module (TC-PRISMA) tutorial on our [website](#) and as part of the playlist on our [YouTube channel](#).

### Example Settings

#### System (System Definer)

Database package	Demo: Steels and Fe-alloys (FEDEMO and MFEDEMO)
------------------	---

Elements	Fe, C
----------	-------

#### Conditions (Precipitation Calculator)

Composition	Fe-0.2C Mass percent
-------------	----------------------

Matrix phase	BCC_A2 <ul style="list-style-type: none"><li>Grain boundary energy (J/m<sup>2</sup>): 0.5</li><li>Grain boundary mobility: Prefactor(m<sup>4</sup>/Js): <math>2 \times 10^{-15}</math></li><li>Click to <b>Edit grain size distribution</b>. Then Initial grain size distribution: Hillert distribution with average radius of <math>3.2 \times 10^{-6}</math> m</li><li>Mobility adjustment: Same for all elements. Prefactor 0.08.</li></ul>
Precipitate phase	CEMENTITE <ul style="list-style-type: none"><li>Zener pinning parameters: Cutoff size (m): <math>8.0 \times 10^{-7}</math></li></ul>
<b>Calculation Type (Precipitation Calculator)</b>	
Calculation type	Isothermal
Temperature	722 °C
Simulation time	35 hours
<b>Datasets (Experimental File Reader)</b>	
[1975Hel]	Data sets included with this example and imported to two Experimental File Readers.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

## Results Discussion

The calculated equilibrium volume fraction of CEMENTITE at 722 °C (0.02786), using FEDEMO, matches that (0.02787) of a Fe-0.2C-0.004S-0.0004O-0.001N-0.001Al (wt.%) multicomponent commercial alloy, calculated using the latest version of the TCS Steel and Fe-alloys Database (TCFE), an alloy that was studied by Hellman and Hillert [1975Hel]. The mobility adjustment assures the precipitation kinetics of CEMENTITE phase in Fe-C system matches that of experimental data [1975Hel] of the commercial counterpart (and shown in [Figure 33](#))

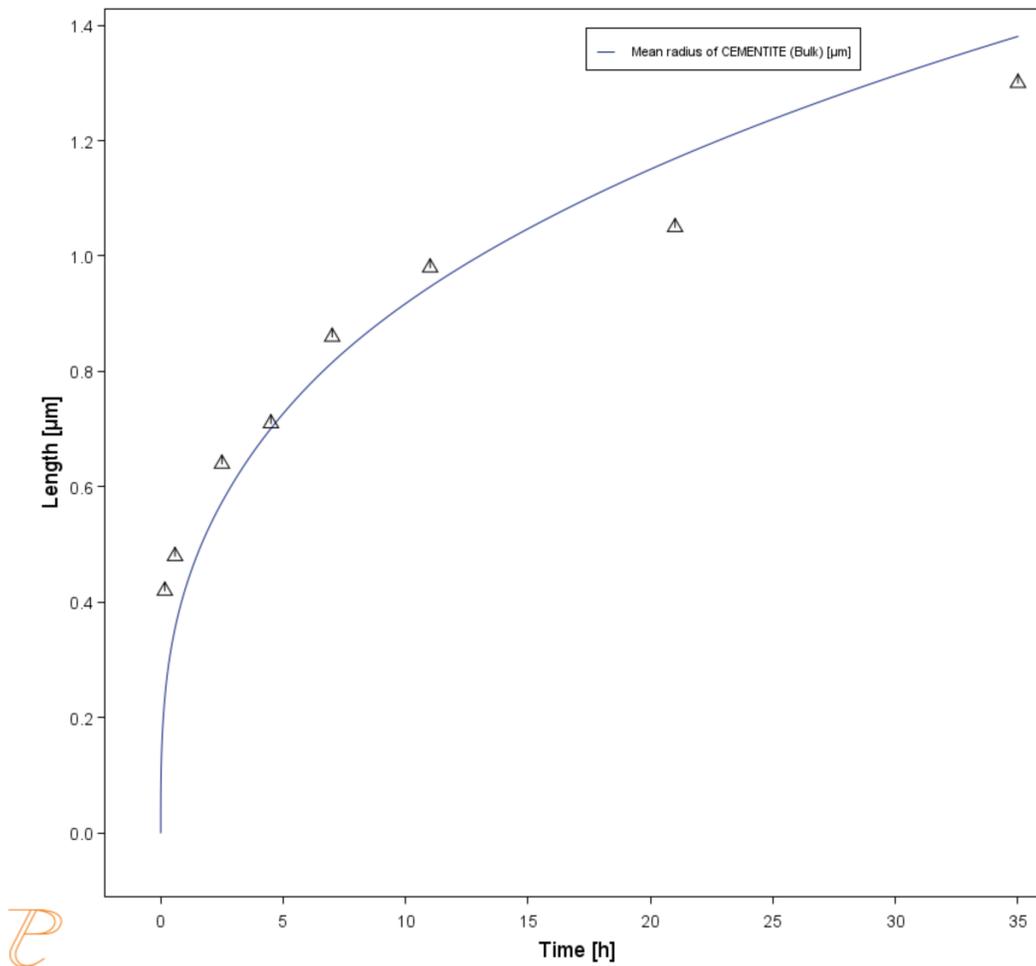


Figure 33: The evolution of the mean radius of cementite comparing the experimental data from [1975Hel] with the Precipitation Calculator results.

The grain boundary energy was chosen to be a reasonable value of  $0.5 \text{ J/m}^2$ . There is a large discrepancy, in several orders of magnitude, among experimental data regarding the grain boundary mobility. In the present calculation, a value of  $2 \times 10^{-15} \text{ m}^4/\text{Js}$  was chosen.

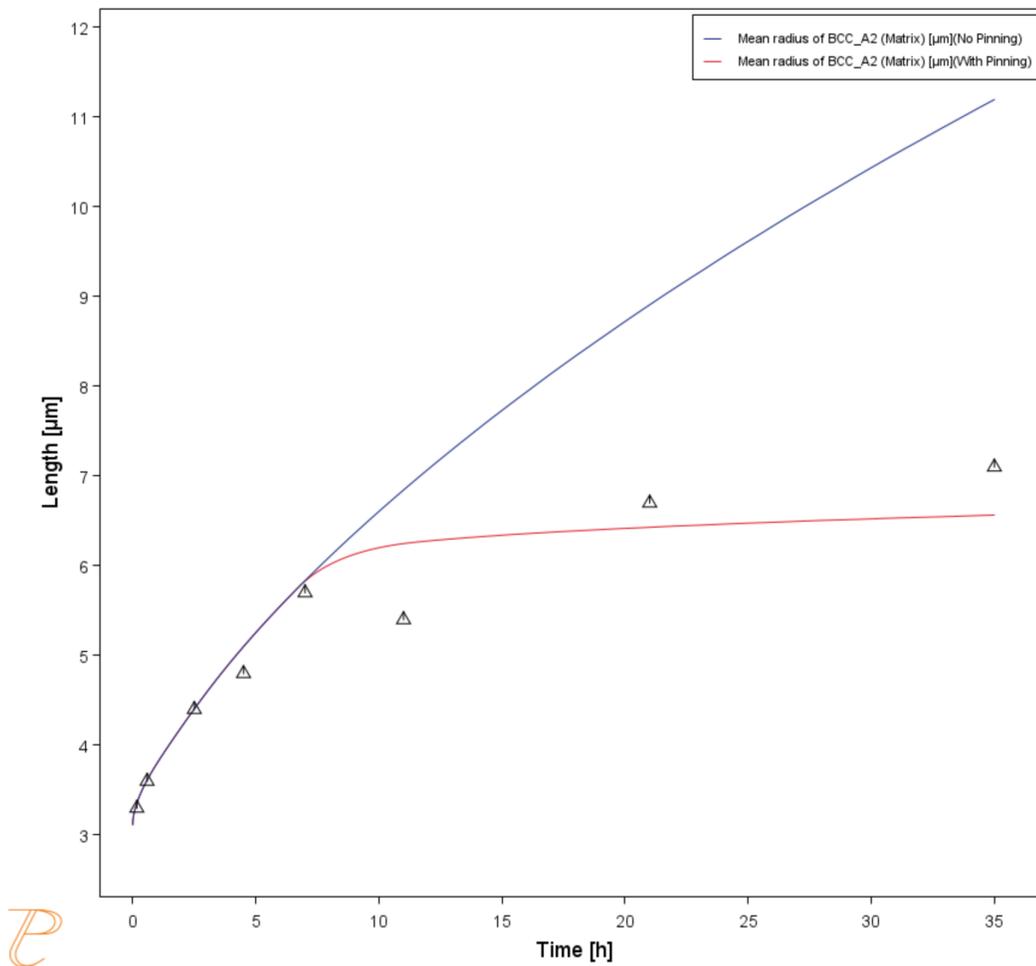


Figure 34: The evolution of the mean grain radius of ferrite comparing the experimental data from [1975Hel] with predictions made with and without Zener pinning from an evolving dispersion of cementite precipitates.

## References

- [1948Smi] C. S. Smith, Grains, Phases, and Interfaces - an Interpretation of Microstructure. Trans. AIME. 175, 15–51 (1948).
- [1975Hel] P. Hellman, M. Hillert, On the Effect of Second-Phase Particles on Grain Growth. Scand. J. Metall. 4, 211–219 (1975).
- [1998Man] P. A. Manohar, M. Ferry, T. Chandra, Five Decades of the Zener Equation. ISIJ Int. 38, 913–924 (1998).

## P\_15: Smooth Transition from Paraequilibrium to Ortho-equilibrium

---

In this example, the precipitation of cementite during tempering of an Fe-Mn-C steel is simulated considering three interface conditions: the usual ortho-equilibrium (OE) condition; paraequilibrium (PE) condition; and a smooth transition from paraequilibrium to ortho-equilibrium condition (PE-OE). The simulation results are compared with the experimental data from Miyamoto et al. [2007Miy].



The Precipitation Calculator nodes are renamed in the example to match the abbreviations for the three interface conditions considered: OE, PE, and PE-OE. Also see the video to learn how to create this example.

This example demonstrates that the early stage of the cementite precipitation follows a PE condition, under which the precipitation kinetics is controlled by the diffusion of C. At a later stage, gradual transition from PE condition to OE condition occurs, and if the tempering time is long enough the diffusion of Mn has a dominating effect on the coarsening of cementite. While a *Simplified* model follows the OE condition and a *Para-eq* model follows PE condition, the *PE Automatic* model enables the smooth transition from PE condition at early stage to OE condition at a late stage.

The example uses the demonstration steel databases, FEDEMO and MFEDEMO. These databases are available to all users (i.e. you do not need a license for the Precipitation Module (TC-PRISMA)) and contain the necessary thermodynamic and kinetic data needed for the calculation.



Open and run this example in Thermo-Calc to follow along with the description and to experiment with the available settings.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Project File Information

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_15\_Precipitation\_Fe-C-Mn\_PE-OE\_Precipitation\_of\_Cementite.tcu*



This example is included as a Precipitation Module (TC-PRISMA) tutorial on our [website](#) and as part of the playlist on our [YouTube channel](#).

## Example Settings

System (System Definer)	
Database package	Demo: Steels and Fe-alloys (FEDEMO and MFEDEMO)
Elements	Fe, Mn, C
Conditions (Precipitation Calculator)	
Composition	Fe-1.96Mn-0.61C mass percent
Matrix phase	BCC_A2 Click <b>Show Details</b> : <ul style="list-style-type: none"><li>Grain size: Average radius 1.0E-7m</li><li>Grain aspect ratio: 100.0</li><li>Mobility adjustment: Keep the default, Same for all elements, then enter 0.008 for the Prefactor, and -7e4 J/mol for the Activation energy.</li></ul> All other defaults are kept.
Precipitate phase	CEMENTITE Click <b>Show Details</b> to select the Growth rate model (Simplified, Para-eq, and PE Automatic). All other defaults are kept.
Calculation Type (Precipitation Calculator)	
Calculation type	Isothermal
Temperature	923.15 Kelvin
Simulation time	1E6 seconds for Simplified and PE Automatic models, and 5 seconds for Para-eq model
Datasets (Experimental File Reader)	
Mean Radius Exp	Data set including experimental mean radius of cementite
Mn Concentration	Data set including experimental Mn composition in cementite

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab (s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the **Precipitation Calculator** tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

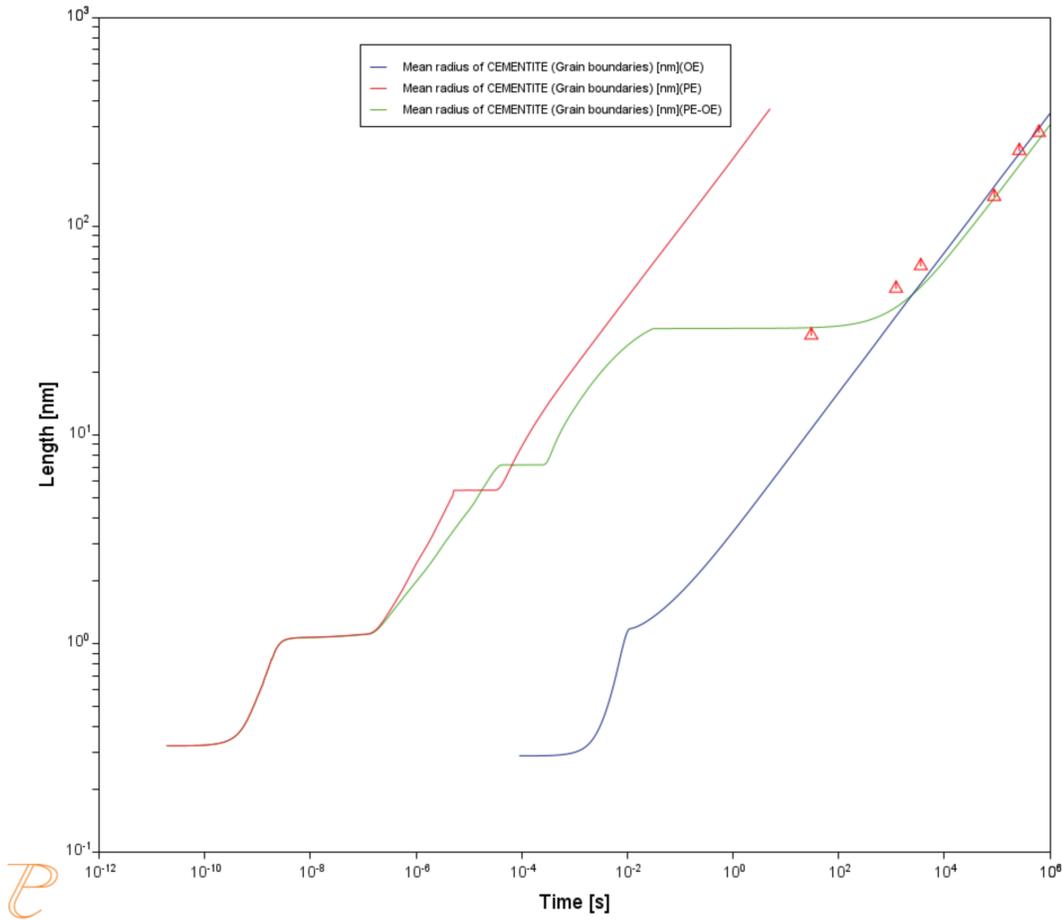


Figure 35: A comparison between the measurements of mean cementite radius from [2007Miy] with the simplified (OE), para-eq (PE), and PE Automatic (PE-OE) growth models. The PE-OE model is needed to capture the nucleation, growth, and coarsening behavior of cementite.

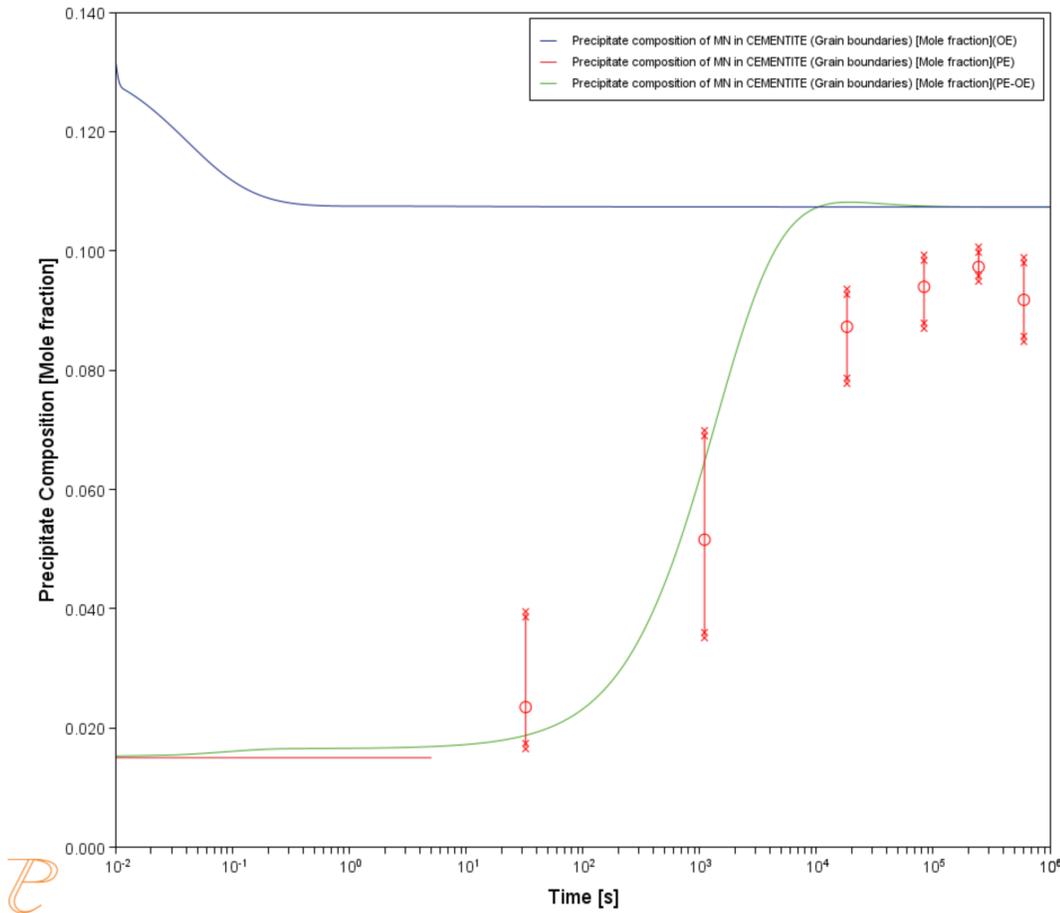


Figure 36: A comparison of the evolution of the cementite Mn composition during isothermal aging at 923.15 K between the experimental data from [2007Miy] with model predictions obtained from the simplified (OE) growth model, the para-eq (PE) growth model, and the PE Automatic (PE-OE) growth model. The PE-OE model captures the behavior shown in the experimental data, where the Mn composition transitions from the initial ferrite composition towards the equilibrium cementite composition.

## Reference

[2007Miy] G. Miyamoto, J. Oh, K. Hono, T. Furuhashi, T. Maki, Effect of partitioning of Mn and Si on the growth kinetics of cementite in tempered Fe–0.6 mass% C martensite. *Acta Mater.* 55, 5027–5038 (2007).

## P\_16 Isothermal Coarsening and a 3D to 2D Stereological Conversion

---

This example demonstrates the use of the **3D → 2D** stereological conversion functionality to compare Precipitation Module (TC-PRISMA) predictions with experimental data obtained from Scanning Electron Micrographs (SEM).



The **3D → 2D** setting is available on a Plot or Table Renderer that is a successor to the Precipitation Calculator, and when certain axis variables are selected for a spherical morphology.



For more background information, see the theory described in [Stereology](#).

The isothermal coarsening kinetics of  $\gamma'$  precipitates in a Ni-Al-Cr alloy is modeled. The experimental data is from [2008Sud], which investigates isothermal coarsening kinetics at a temperature of 873 K for up to 264 h.

It is assumed that there are no precipitates in solution at the beginning of the isothermal heat treatment.



Open and run this example in Thermo-Calc to follow along with the description and to experiment with the available settings.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### **Project File Information**

- Folder: **Precipitation Module - TC-PRISMA**
- File name: *P\_16\_Precipitation\_Ni-Al-Cr\_Stereology.tcu*



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## Example Settings



The example also includes an Equilibrium Calculator and Table Renderer to first confirm the stability of the precipitate phase for this system (FCC\_L12#2), then determine particle composition at 1073 K, and this information is used with the Precipitation Calculator set up.

System (System Definer)	
Database package	Demo: Nickel-based superalloys (NIDEMO and MNIDEMO)
Elements	Ni, Al, Cr
Conditions (Precipitation Calculator)	
Composition	Ni-10I-8.5Cr Mole percent
Matrix Phase	FCC_L12 <ul style="list-style-type: none"><li>Click <b>Show Details</b> and for the <b>Mobility adjustment</b> enter 0.5 for the <b>Prefactor</b>.</li><li>All other defaults are kept.</li></ul>
Precipitate Phase	FCC_L12#2 Click <b>Show Details</b> and adjust the default settings as indicated: <ul style="list-style-type: none"><li><b>Interfacial energy</b>: Select <b>User-defined</b> and enter 0.01 J/m<sup>2</sup>.</li></ul>
Calculation Type (Precipitation Calculator)	
Calculation Type	Isothermal
Temperature	1073 Kelvin

Simulation time | 300 h

### Datasets (Experimental File Reader)

2008 Sudbrack et al

Data sets are included with this example and are imported using the Experimental File Reader activity. There are 2D size distributions comparing results after a total of 16 h aging, 64 h aging, and 264 h of aging, and a comparison of the experimental and predicted 2D mean radius versus time.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation.

- **Thermal Profile:** When setting up a calculator on a **Configuration** window for **Isothermal** or **Non-isothermal Calculation Types**, you can preview the profile and adjust settings as needed. When you click a calculator node in the **Project** window, the matching name of the node is on the tab(s) displayed in the **Visualizations** window.



For **TTT Diagram** and **CCT Diagram** calculations there is nothing shown for the Precipitation Calculator tab in the **Visualizations** window as there is no Thermal Profile to be defined.

- **Plot or Table** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, either click a **Plot Renderer** or **Table Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.



For this **Isothermal** example, its **Thermal Profile** is also displayed in the **Visualizations** window showing the constant temperature entered for this calculation type. The tab names match the node names in the **Project** window.

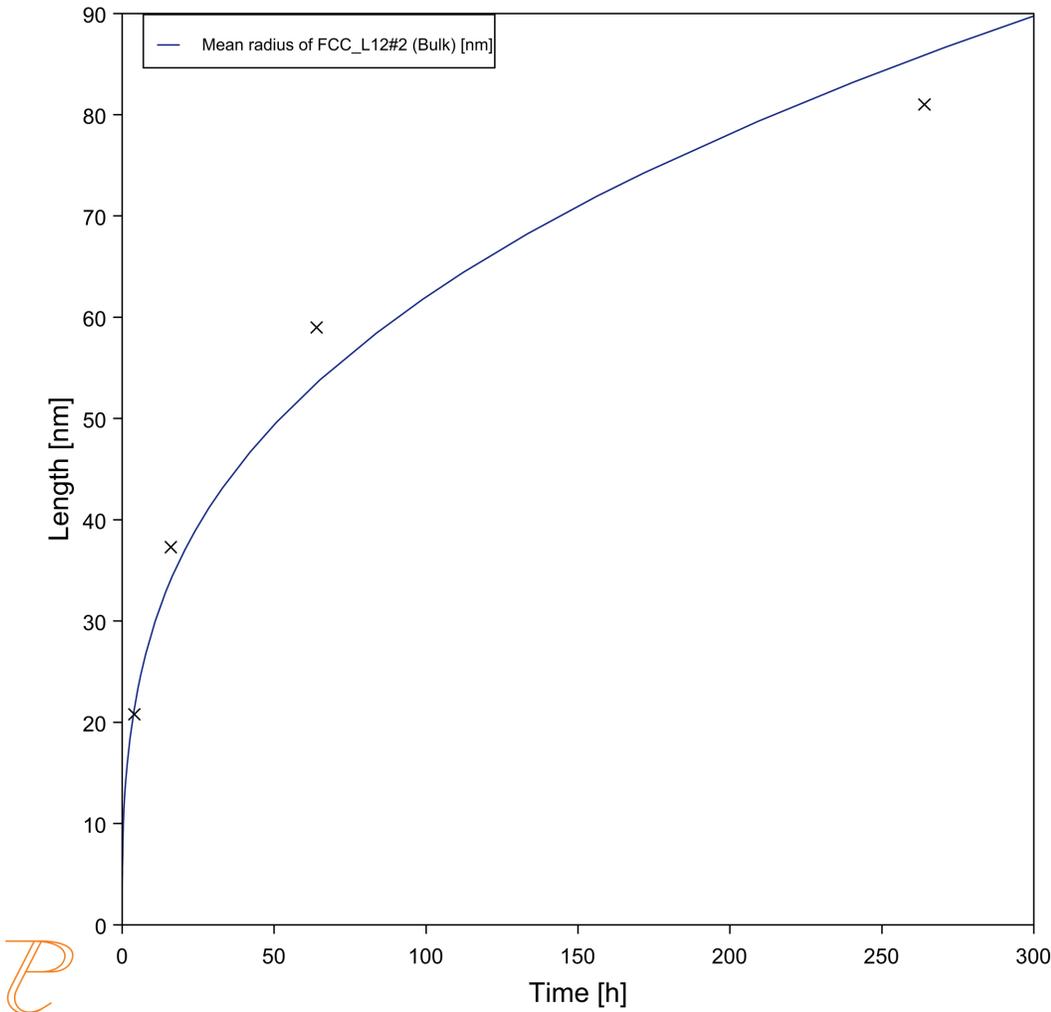


Figure 37: A comparison of the evolution of the mean 2D precipitate radius measurements of [2008Sud] with model predictions.

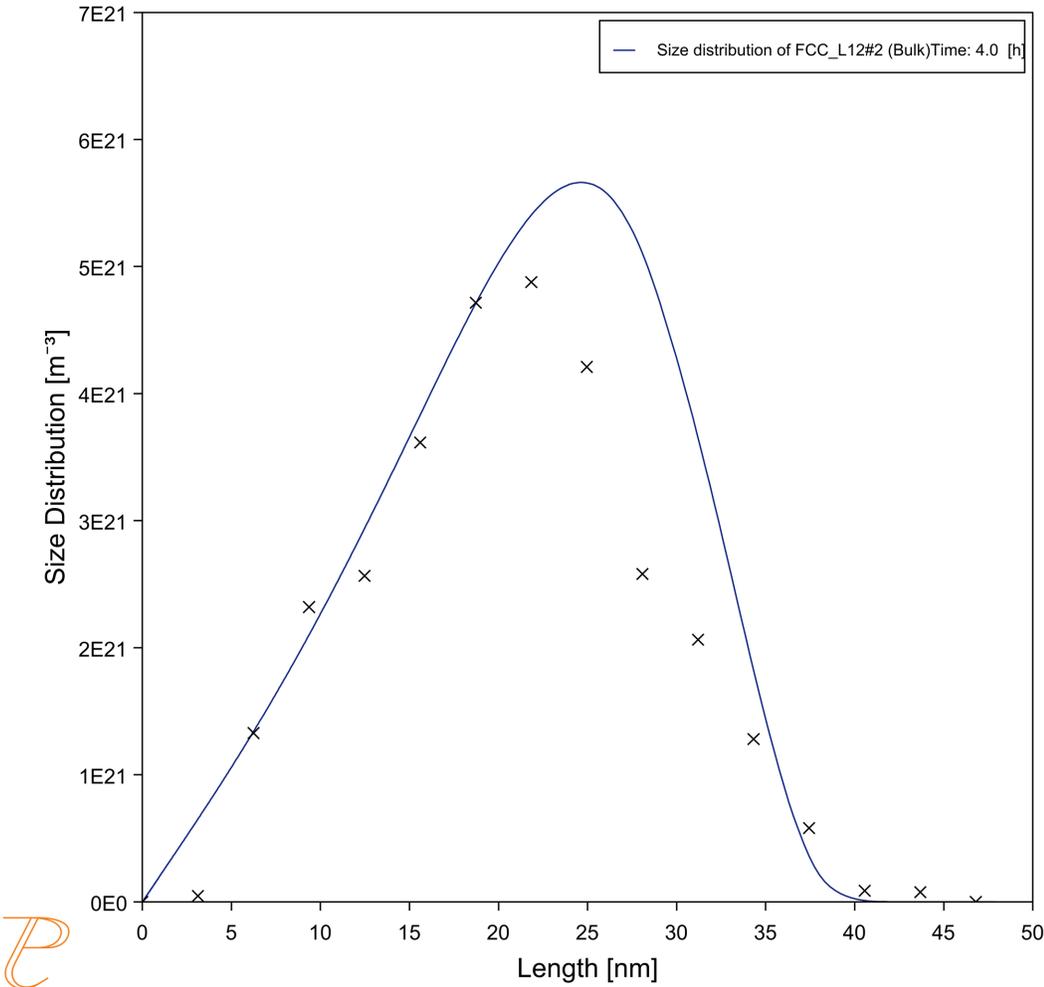


Figure 38: A comparison of the predicted and measured 2D size distribution after 4 h of aging.

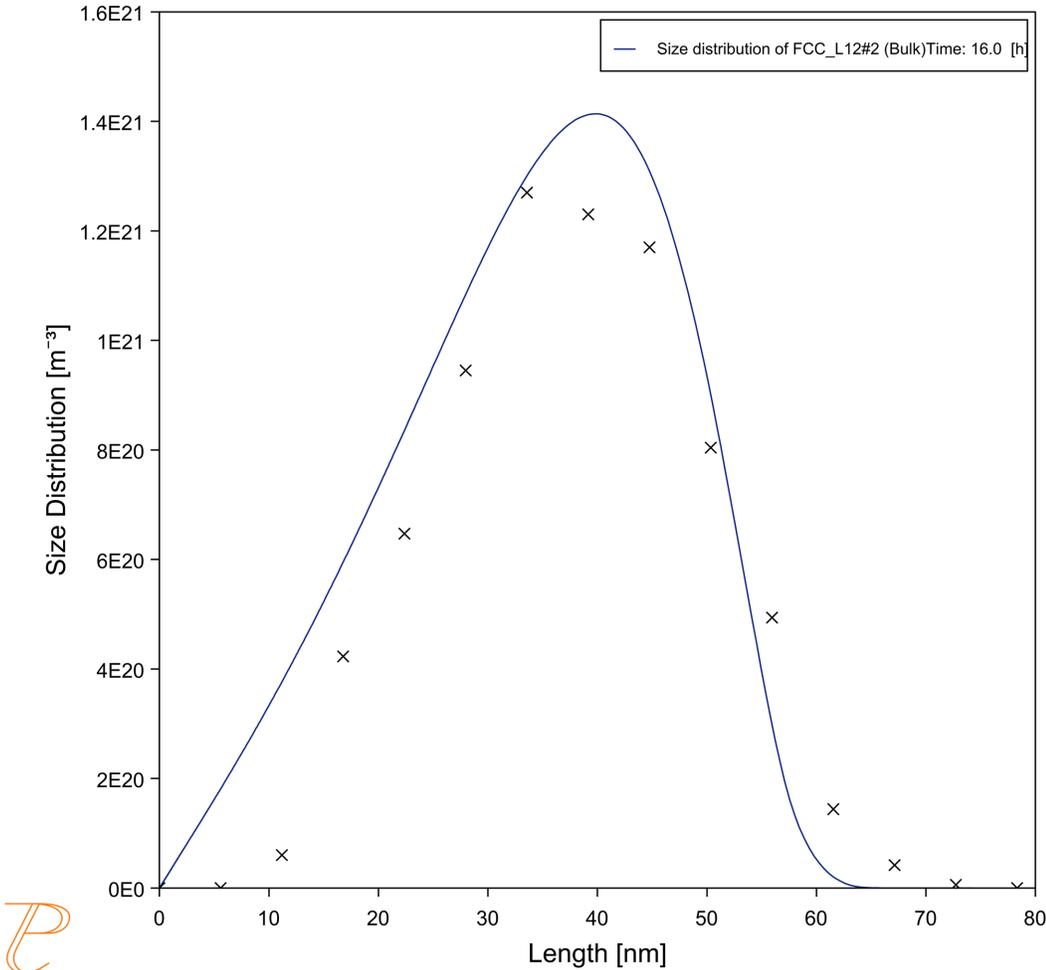


Figure 39: A comparison of the predicted and measured 2D size distribution after 16 h of aging.

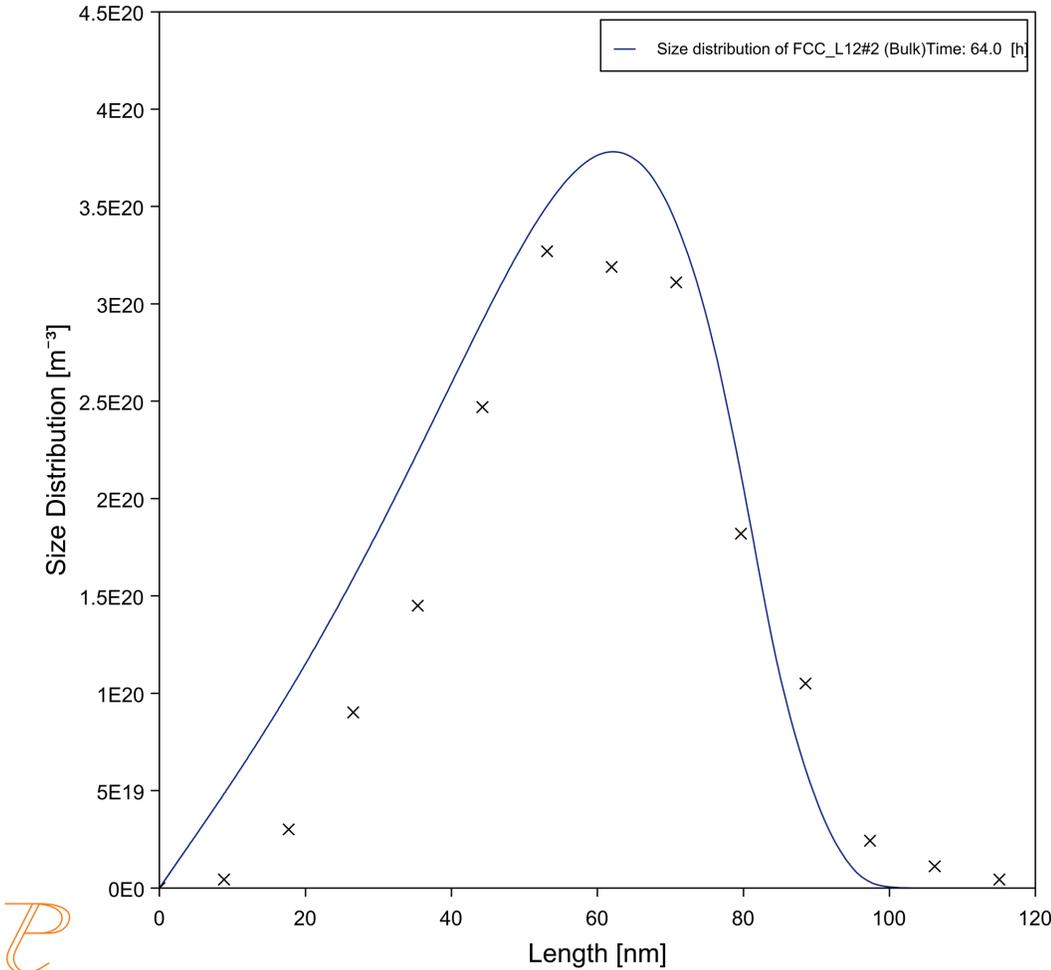


Figure 40: A comparison of the predicted and measured 2D size distribution after 64 h of aging.

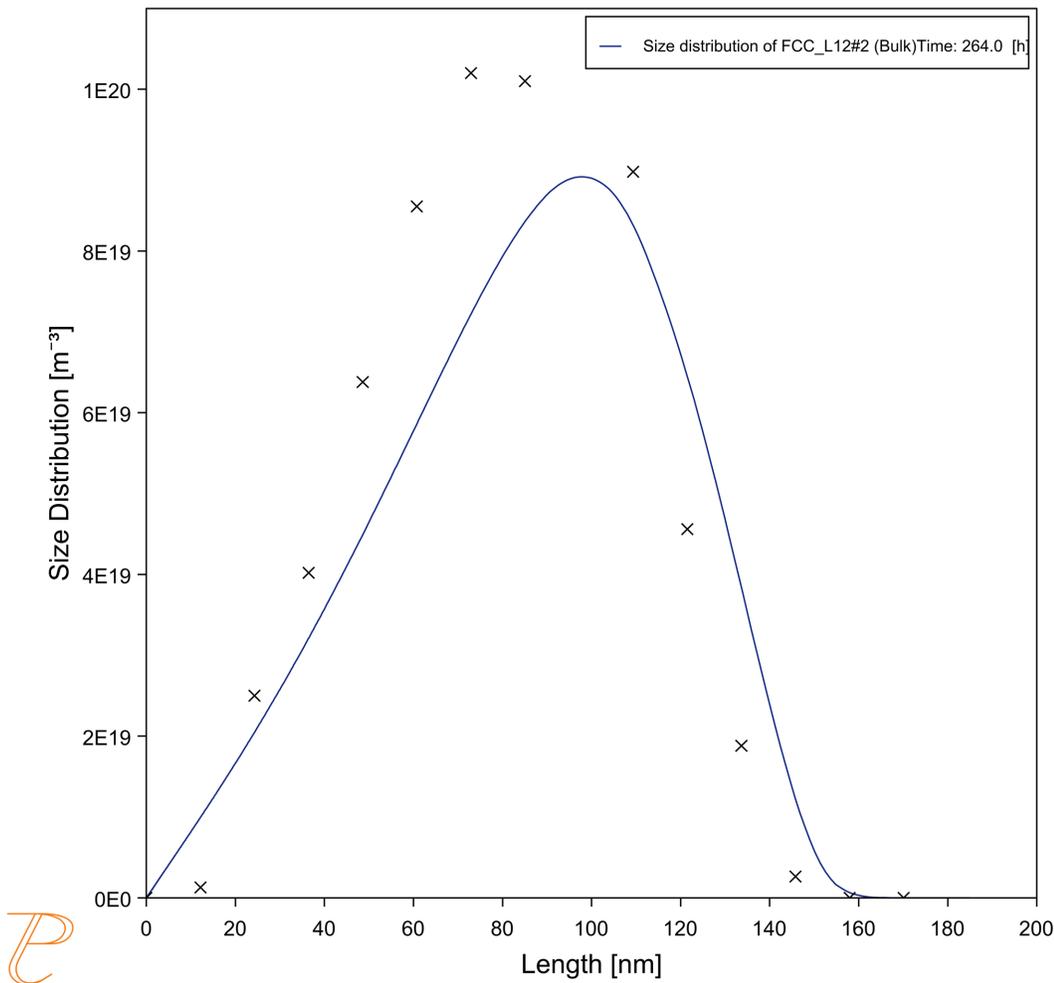


Figure 41: A comparison of the predicted and measured 2D size distribution after 264 h of aging.

## Reference

[2008Sud] C. K. Sudbrack, T. D. Ziebell, R. D. Noebe, D. N. Seidman, Effects of a tungsten addition on the morphological evolution, spatial correlations and temporal evolution of a model Ni–Al–Cr superalloy. *Acta Mater.* 56, 448–463 (2008).

# Additive Manufacturing (AM) Module

## Examples Collection

---



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. Without a license you are in *Demo Mode* where you can, for example, open and view example set ups, run some examples, add templates and nodes to the Project window, adjust some Configuration settings, and preview some functionality on the Visualizations window.



Read more about the [Additive Manufacturing \(AM\) Module](#) on our website including the details about database compatibility or to watch an [introductory webinar](#). You can also use the [Getting Started Guide](#) to learn about the key features available.

In this section:

---

AM_01: Transient Simulation of a Single Track .....	330
AM_02: Transient and Steady-state Simulations of a Single Track .....	338
AM_03: Steady-state Simulations .....	345
AM_04: Scheil Transient Steady-state .....	353
AM_05: Using AM Calculator Probe Data with the Diffusion Module (DICTRA) .....	364
AM_06a: Calibrating a Heat Source for a 316L Steel .....	375
AM_06b: Using the Calibrated Heat Source for a 316L Steel .....	381

---

## AM\_01: Transient Simulation of a Single Track

---

The following example is part of a series showing some of the features of the Additive Manufacturing (AM) Module and the **AM Calculator**. In this example, a single track simulation is performed using the **Transient** model of the AM Calculator.



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. Without a license you are in *Demo Mode* where you can, for example, open and view example set ups, run some examples, add templates and nodes to the Project window, adjust some Configuration settings, and preview some functionality on the Visualizations window.



Some examples (AM\_01, AM\_02, AM\_03, and AM\_06b) use predefined Material Properties libraries that are embedded with the examples for use for all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO mode. See [Available Options](#) for details.

### **Project File Information**

- Folder: **Additive Manufacturing**
- File name: *AM\_01\_Transient\_DE\_No\_Marangoni.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Material Properties**

These properties are the same for both AM\_01 and AM\_02.

- Inconel 738LC: Ni-15.8 Cr-8.6 Co-2.7 W-3.3 Ti-3.6 Al-1.8 Ta-1.8 Mo-0.04 Fe-0.8 Nb-0.002 Mn-0.1 C Mass percent.
- Database: TCNI12
- The material property was pre-calculated, and stored as a built-in material library of the Additive Manufacturing (AM) Module.

## Reference

[2021Gra] D. Grange, A. Queva, G. Guillemot, M. Bellet, J.-D. Bartout, C. Colin, Effect of processing parameters during the laser beam melting of Inconel 738: Comparison between simulated and experimental melt pool shape. *J. Mater. Process. Technol.* 289, 116897 (2021).

## Model Configuration

The transient model solves for the heat equation in the entire 3D domain, including the melt pool, and is therefore, computationally expensive to solve. The fluid flow inside the melt pool due to the Marangoni effect is not included in this example. The double ellipsoidal, or the so-called Goldak heat source model, is used to predict melt pool size and temperature distribution during single track scanning. The parameters for the double ellipsoidal heat source are computed using optimization in a steady-state case for the given process parameters (power and scanning speed) and the melt pool size reported in the paper by Grange et al. [2021Gra]. The resulting parameters are shown in [Figure 1](#).



Heat Source						
Double ellipsoidal	Power: 230.0	W Absorptivity: 88.06	% ar: 97.3	μm a: 99.04	μm b: 41.67	μm c: 181.11

Figure 1: Optimized heat source parameters entered on the AM Calculator for the experiment given in Grange et al. [2021Gra] with power = 230 W and scanning speed = 960 mm/s.

The geometry for a transient simulation, as given in the paper by Grange et al. [2021Gra], is shown in [Figure 2](#). The base plate is 2 mm in length, 0.5 mm in width, and with a thickness of 0.95 mm. A layer of powder with a thickness of 55 μm is deposited on the solid substrate. The powder layer, however, has the same material properties as the solid substrate. The laser beam with a power of 230 W develops linear scanning with constant velocity 960 mm/s. The trajectory evolves from initial position  $(x, y) = (0.25, 0.25)$  mm to final position  $(1.75, 0.25)$  mm on the top surface.

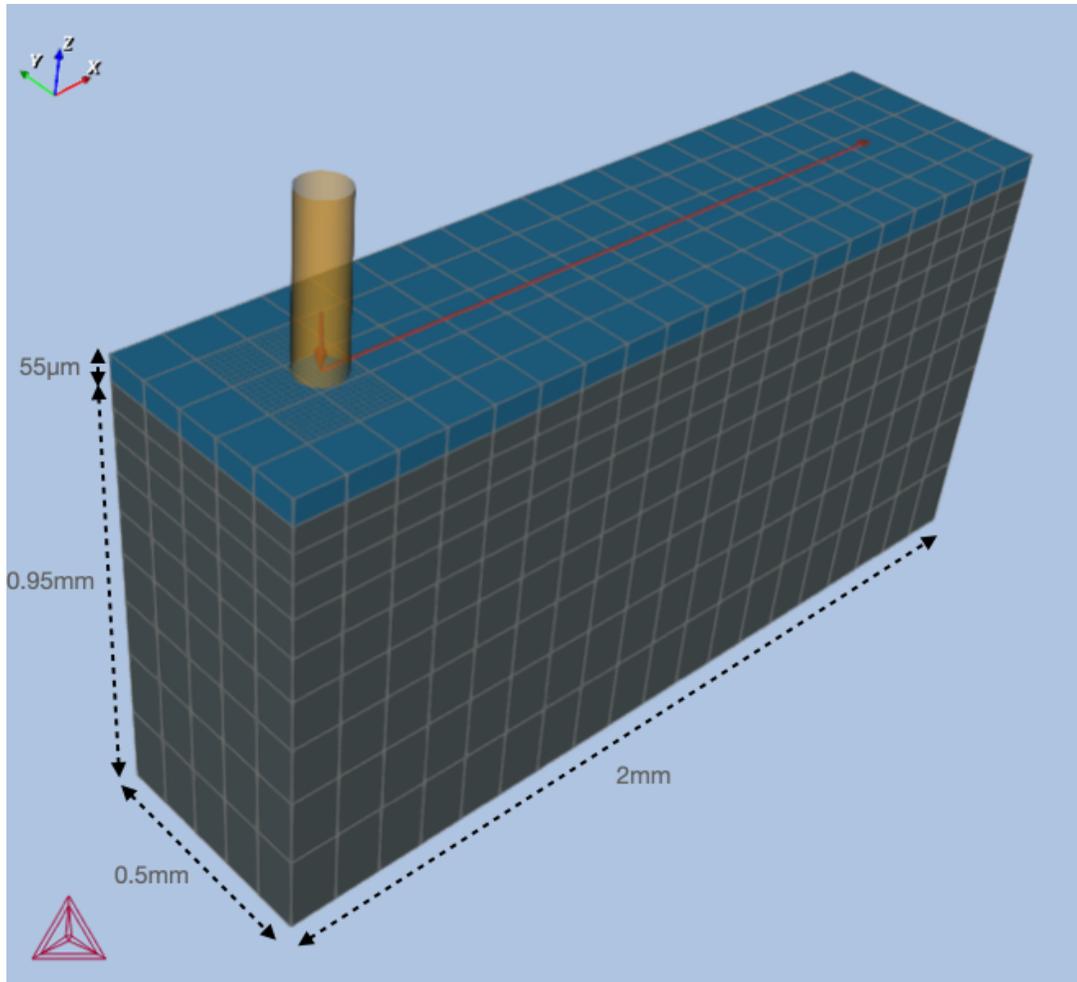


Figure 2: The geometry used for the Additive Manufacturing (AM) Module examples AM\_01 and AM\_02.

Two probes monitor the temporal evolution of temperature at positions shown in [Figure 3](#). The results can be presented by tabulated data or as 2D graph plots.

**Probe Positions**

<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	x: 0.5	mm	y: 0.25	mm	z: 1.005	mm	<input type="button" value="Pick coordinate"/>
<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	x: 0.5	mm	y: 0.29	mm	z: 1.005	mm	<input type="button" value="Pick coordinate"/>

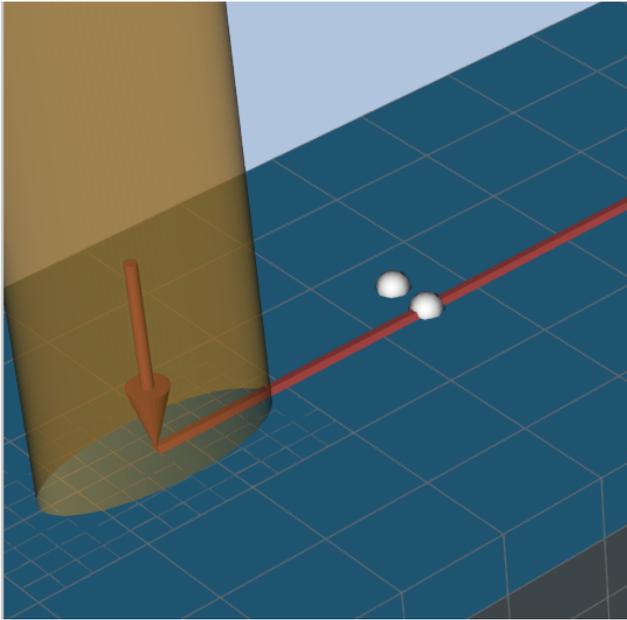


Figure 3: Probe positions for Probes P1 and P2, where P1 is placed exactly on the scanning track while P2 is placed at a distance of 0.04 mm from the track, as shown in a close up of the probes on the Visualizations window.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation(s).

- **Geometry:** View the geometry and adjust parameters visually by changing inputs on the Configuration window. Add probes as shown in [Figure 4](#).
- Melt pool and other dynamically changing features can also be visualized and changed.
- **Plot** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, click a **Plot Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

[Figure 4](#) shows the surface plot of temperature distribution after the scanning process is completed. It is noted that the melt pool shape reaches a steady-state when the laser beam approaches the end of the scanning track. The melt pool dimensions at the end of scanning track is given in the table.

Melt Pool Dimension	Size (mm)
Width	0.125
Depth	0.144
Length	1.12



If you are in the project file, click the **Probe plot** node in the Project window and the **3D Plot** tab in the Visualizations window.

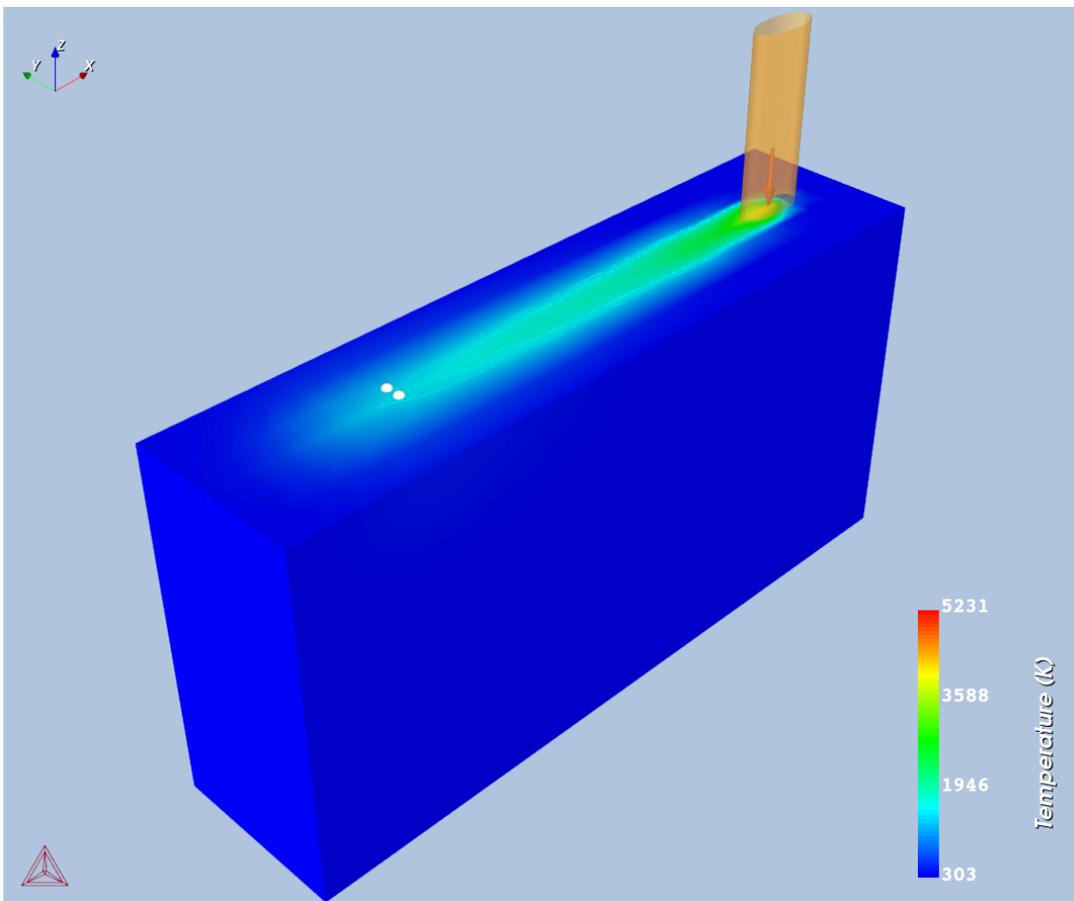
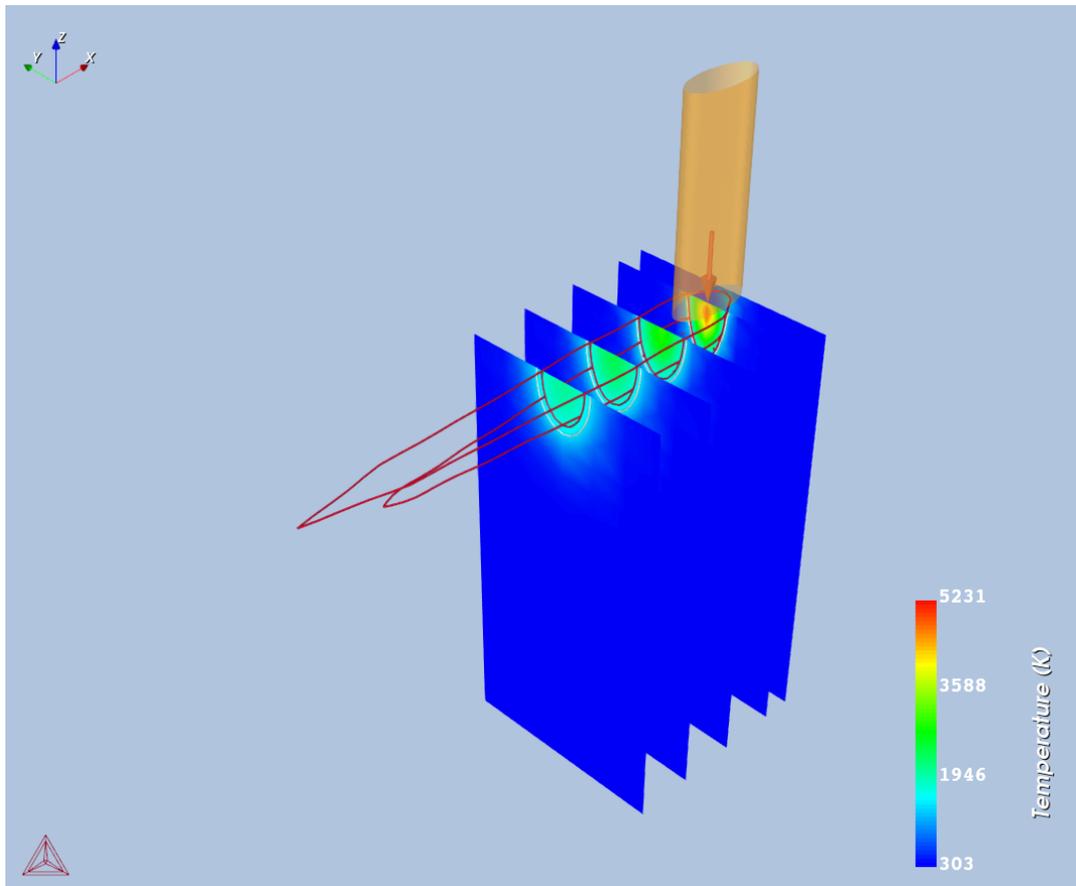


Figure 4: Surface plot of temperature distribution after the single track scan is completed. Probes P1 and P2 can also be seen on the top surface.

It is also possible to plot quantities at different cross-sections inside the domain by selecting the **Slice** checkbox on the Plot Renderer Configuration window under the 3D Plot tab. [Figure 5](#) shows the temperature slices plotted at cross-sections orthogonal to the scanning direction.



*Figure 5: Slices of the computational domain, orthogonal to the scanning direction, showing temperature distribution at different cross-sections. Red tubes show the contour of the melt pool (at liquidus temperature).*

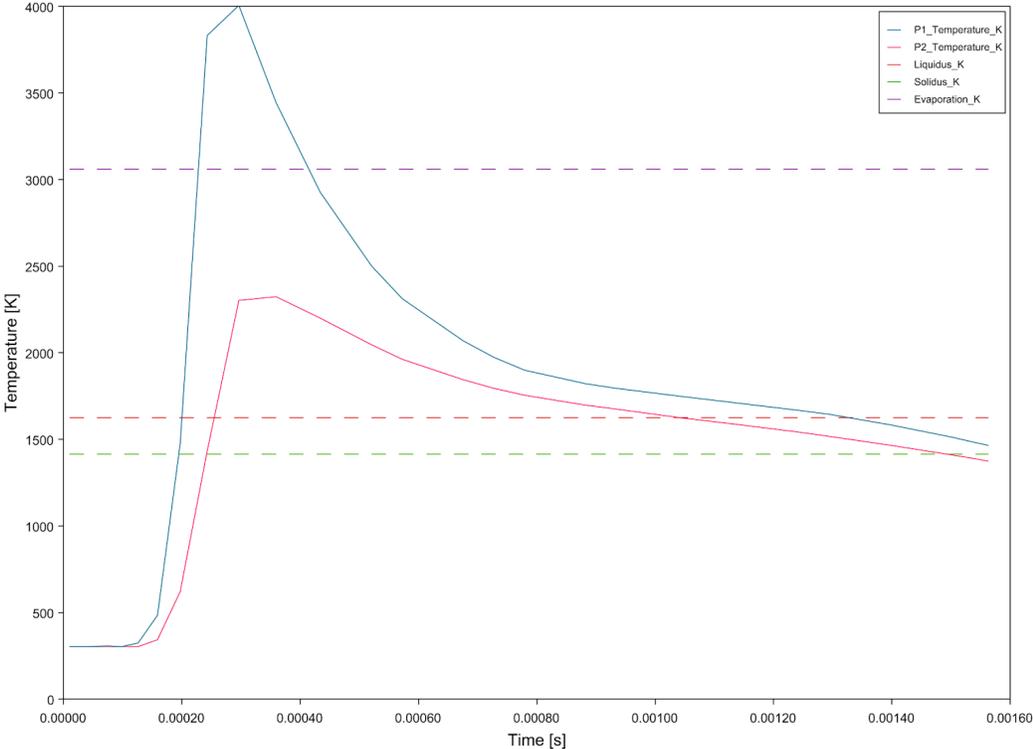


Figure 6: 2D probe plot showing temperature evolution as a function of time at probes P1 and P2.

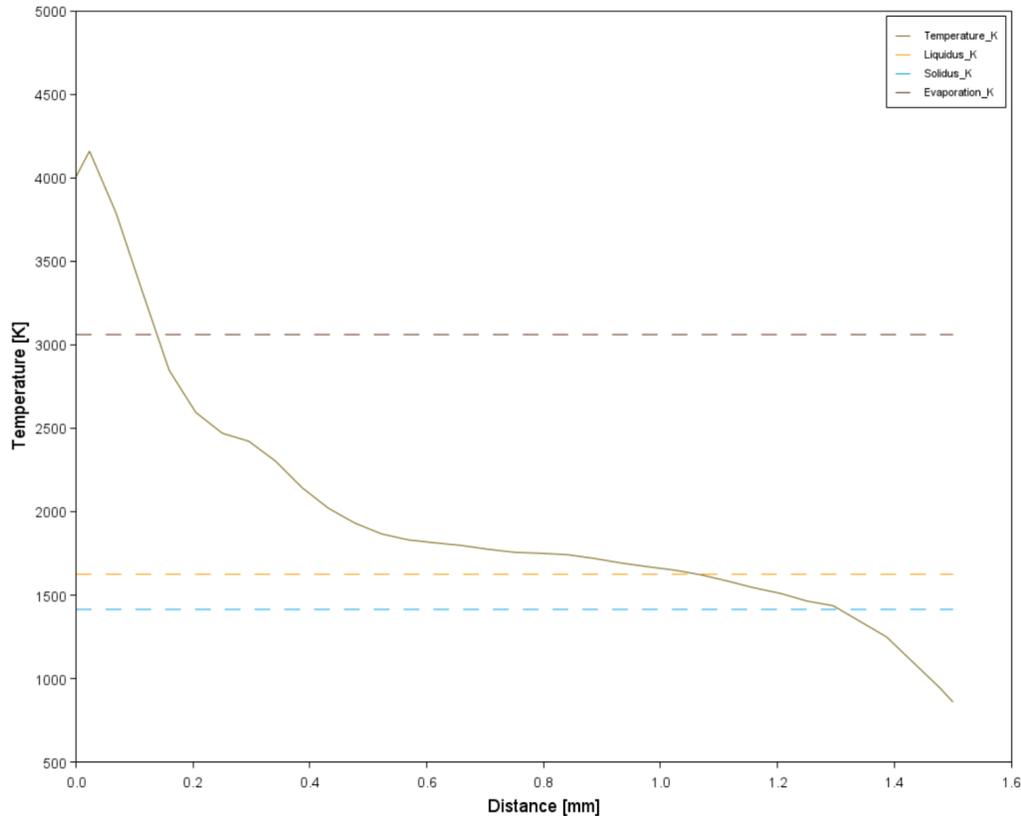


Figure 7: 2D plot over line showing temperature in the tail of the melt pool as a function of distance plotted along the scanning track.

## Other Resources



Read more about the [Additive Manufacturing \(AM\) Module](#) on our website including the details about database compatibility or to watch an [introductory webinar](#). You can also use the [Getting Started Guide](#) to learn about the key features available.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## AM\_02: Transient and Steady-state Simulations of a Single Track

---

The following example is part of a series showing some of the features of the Additive Manufacturing (AM) Module and the **AM Calculator**. This example builds on the work done by Grange et al. [2021Gra] and is similar to example AM\_01. The difference however is that in this example a single track simulation is performed using **Transient with heat source from Steady-state** model instead of the **Transient** model.

The **Transient with heat source from Steady-state** model exploits the assumption that the melt pool size and temperature distribution reaches a steady-state almost instantly and thus first solves for a **Steady-state** case with the given process parameters. The result from the **Steady-state** in the form of temperature distribution in the melt pool is then mapped as a heat source in the transient simulation. This novel approach is much faster and efficient than the approach used in example AM\_01 where the heat equation is solved in the entire 3D domain.



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. Without a license you are in *Demo Mode* where you can, for example, open and view example set ups, run some examples, add templates and nodes to the Project window, adjust some Configuration settings, and preview some functionality on the Visualizations window.



Some examples (AM\_01, AM\_02, AM\_03, and AM\_06b) use predefined Material Properties libraries that are embedded with the examples for use for all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO mode. See [Available Options](#) for details.

### **Project File Information**

- Folder: **Additive Manufacturing**
- File name: *AM\_02\_TransientSS\_DE.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Material Properties**

These properties are the same for both AM\_01 and AM\_02.

---

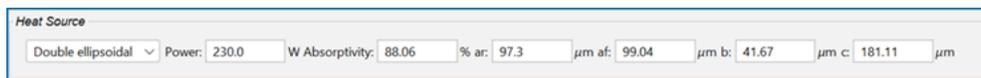
- Inconel 738LC: Ni-15.8 Cr-8.6 Co-2.7 W-3.3 Ti-3.6 Al-1.8 Ta-1.8 Mo-0.04 Fe-0.8 Nb-0.002 Mn-0.1 C Mass percent.
- Database: TCNI12
- The material property was pre-calculated, and stored as a built-in material library of the Additive Manufacturing (AM) Module.

## Reference

[2021Gra] D. Grange, A. Queva, G. Guillemot, M. Bellet, J.-D. Bartout, C. Colin, Effect of processing parameters during the laser beam melting of Inconel 738: Comparison between simulated and experimental melt pool shape. *J. Mater. Process. Technol.* 289, 116897 (2021).

## Model Configuration

The transient model solves for the heat equation in the entire 3D domain, including the melt pool, and is therefore, computationally expensive to solve. The fluid flow inside the melt pool due to the Marangoni effect is not included in this example. The double ellipsoidal, or the so-called Goldak heat source model, is used to predict melt pool size and temperature distribution during single track scanning. The parameters for the double ellipsoidal heat source are computed using optimization in a steady-state case for the given process parameters (power and scanning speed) and the melt pool size reported in the paper by Grange et al. [2021Gra]. The resulting parameters are shown in [Figure 8](#).



Parameter	Value
Heat Source Model	Double ellipsoidal
Power	230.0 W
Absorptivity	88.06 %
% ar	97.3
μm a	99.04
μm b	41.67
μm c	181.11 μm

*Figure 8: Optimized heat source parameters entered on the AM Calculator for the experiment given in Grange et al. [2021Gra] with power = 230 W and scanning speed = 960 mm/s.*

The geometry for a transient simulation, as given in the paper by Grange et al. [2021Gra], is shown in [Figure 9](#). The base plate is 2 mm in length, 0.5 mm in width, and with a thickness of 0.95 mm. A layer of powder with a thickness of 55 μm is deposited on the solid substrate. The powder layer, however, has the same material properties as the solid substrate. The laser beam with a power of 230 W develops linear scanning with constant velocity 960 mm/s. The trajectory evolves from initial position (x, y) = (0.25, 0.25) mm to final position (1.75, 0.25) mm on the top surface.

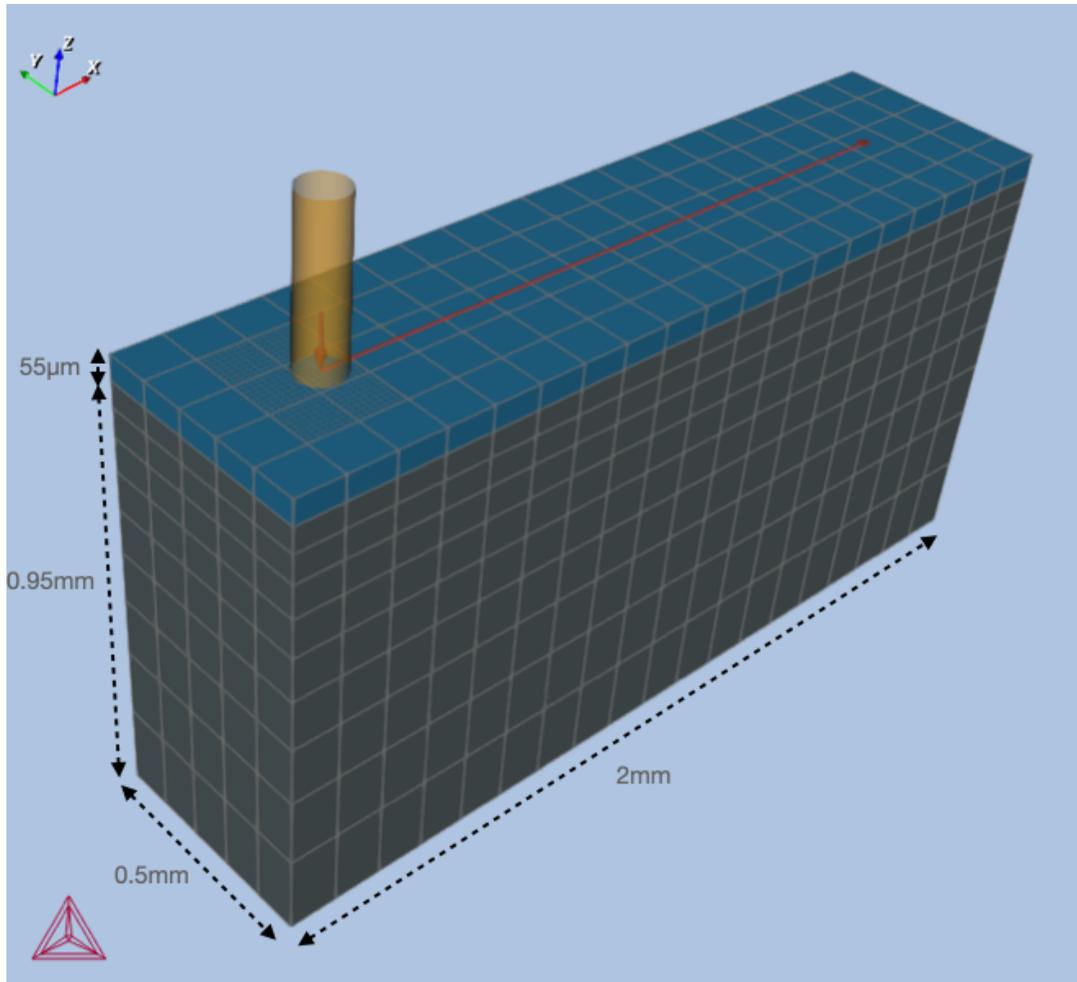


Figure 9: The geometry used for the Additive Manufacturing (AM) Module examples AM\_01 and AM\_02.

Two probes monitor the temporal evolution of temperature at positions shown in [Figure 10](#). The results can be presented by tabulated data or as 2D graph plots.

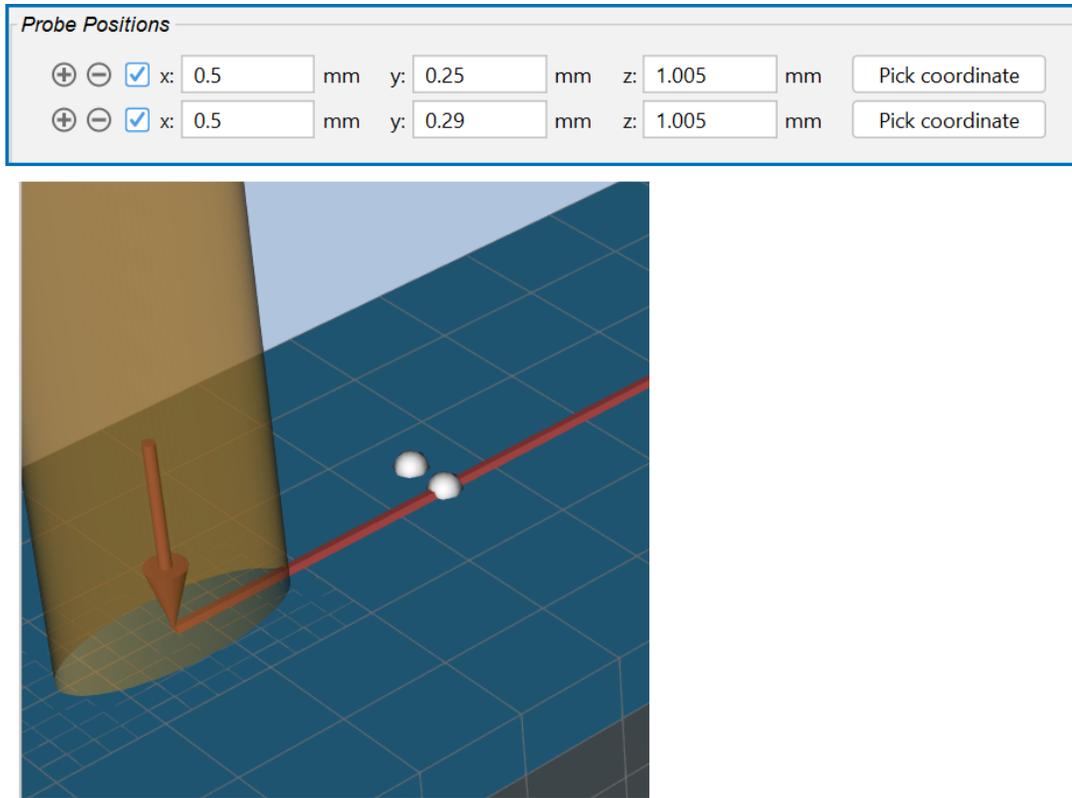


Figure 10: Probe positions for Probes P1 and P2, where P1 is placed exactly on the scanning track while P2 is placed at a distance of 0.04 mm from the track, as shown in a close up of the probes on the Visualizations window.

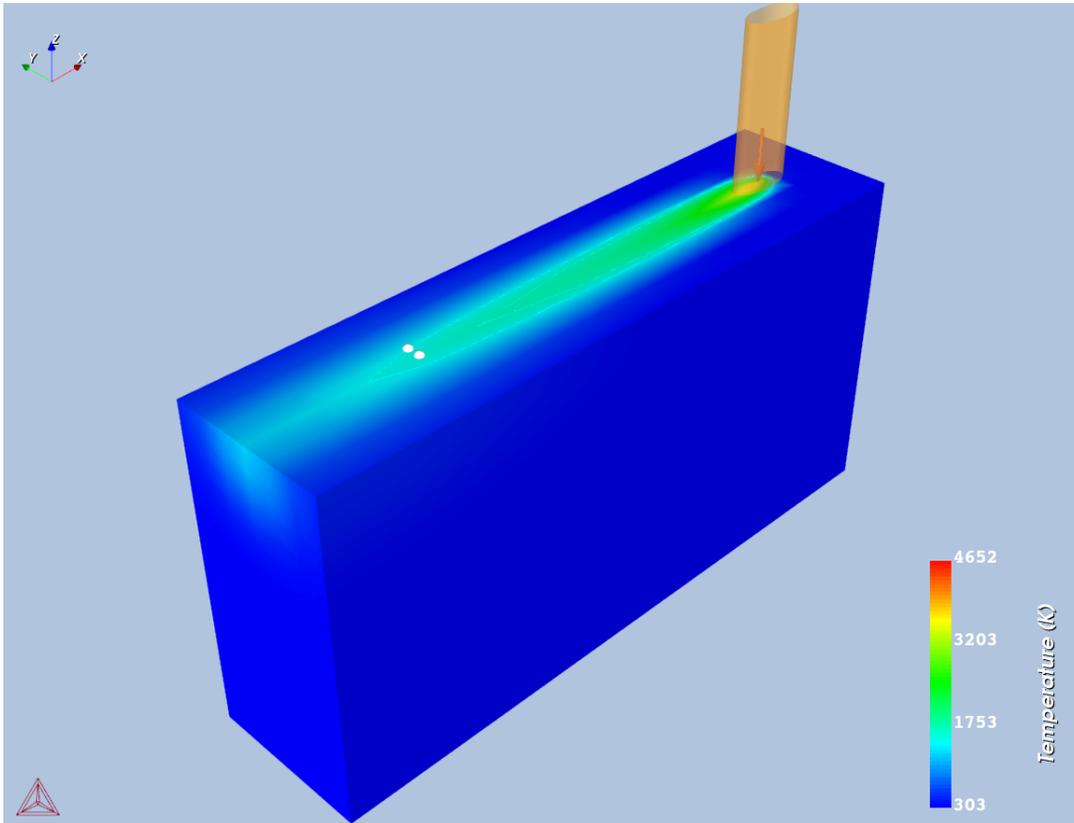
## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation(s).

- **Geometry:** View the geometry and adjust parameters visually by changing inputs on the Configuration window.
- Melt pool and other dynamically changing features can also be visualized and changed.
- **Plot** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, click a **Plot Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

With the example open and after it is run, the following is a summary of what is visible on the **Visualizations** window.

[Figure 11](#) shows the surface plot of temperature distribution after the single track scanning is completed. The temperature profile is qualitatively similar to the one in example AM\_01, however the maximum temperature is slightly lower in example AM\_02. This could be due to selection of a coarse mesh in both examples which sometimes could affect the maximum temperature.



*Figure 11: Surface plot of temperature distribution after the single track scan is completed. Probes P1 and P2 can also be seen on the top surface.*

The melt pool dimensions for AM\_02 are given in the following table, which are very similar to the melt pool dimensions obtained in example AM\_01.

Melt Pool Dimension	Size (mm)
Width	0.127
Depth	0.129
Length	1.12

A qualitative comparison between AM\_01 and AM\_02 can also be made by looking at probe plots. The probe plot for AM\_02 is given in [Figure 12](#).

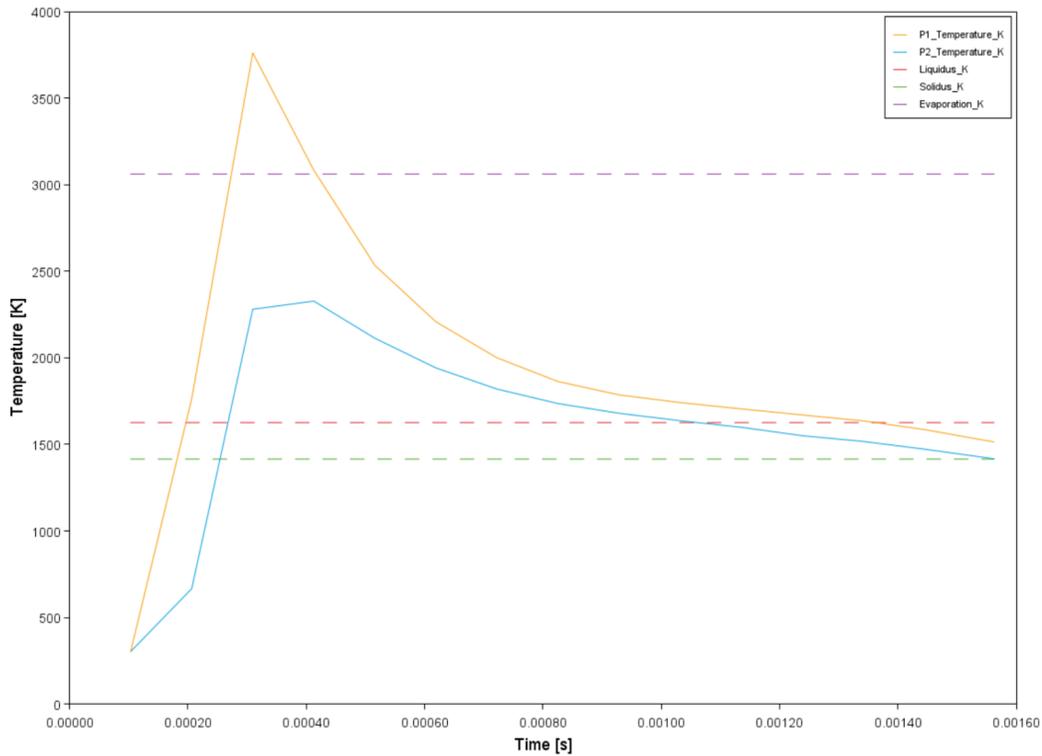


Figure 12: 2D probe plot showing temperature evolution as a function of time at probes P1 and P2.

Despite the fact that maximum temperature is higher in AM\_01, the temperature profile in the tail of the melt pool and close to the melt pool is very similar. This can also be concluded by comparing plot over lines for the two examples, where the plot over line for AM\_02 is given in [Figure 13](#).

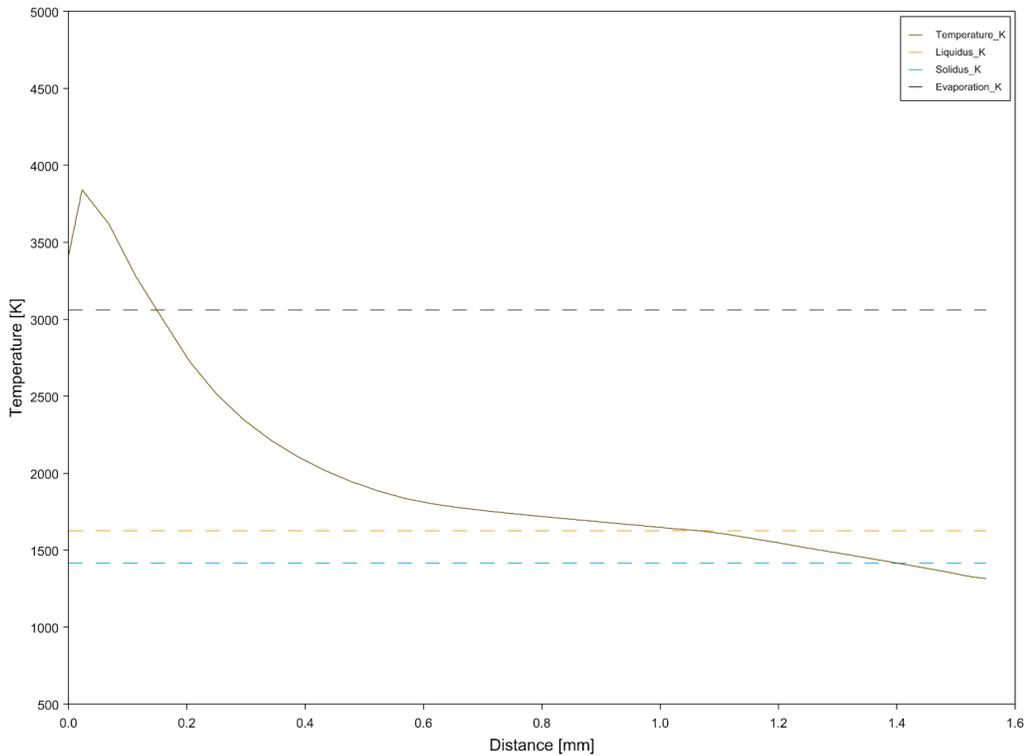


Figure 13: 2D plot over line showing temperature in the tail of the melt pool as a function of distance plotted along the scanning track.

## Other Resources



Read more about the [Additive Manufacturing \(AM\) Module](#) on our website including the details about database compatibility or to watch an [introductory webinar](#). You can also use the [Getting Started Guide](#) to learn about the key features available.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## AM\_03: Steady-state Simulations

---

The following example is part of a series showing some of the features of the Additive Manufacturing (AM) Module and the **AM Calculator**.



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. Without a license you are in *Demo Mode* where you can, for example, open and view example set ups, run some examples, add templates and nodes to the Project window, adjust some Configuration settings, and preview some functionality on the Visualizations window.



Some examples (AM\_01, AM\_02, AM\_03, and AM\_06b) use predefined Material Properties libraries that are embedded with the examples for use for all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO mode. See [Available Options](#) for details.

In this example steady-state simulations are performed for IN625 with different conditions i.e.

- i. without fluid flow in the melt pool,
- ii. with fluid flow in the melt pool due to Marangoni effect, and
- iii. using separate material properties for the powder.

and then the results are compared to demonstrate the effects of fluid flow and separate material properties for the powder on the temperature distribution as well as on the shape of the melt pool.



For the first two simulations, the same material properties are used for both powder and solid substrate while for the third simulation no fluid flow is included in the melt pool.



In steady-state simulations it is assumed that the heat source (laser beam) moves with a uniform speed relative to the base plate. In this case, the model is solved by modifying governing equations to a reference frame attached to a moving heat source. Furthermore, in steady-state mode you can make use of symmetry along the width of the geometry and solve for only half of the domain.

### ***Project File Information***

- Folder: **Additive Manufacturing**
- File name: *AM\_03\_Steady.tcu*

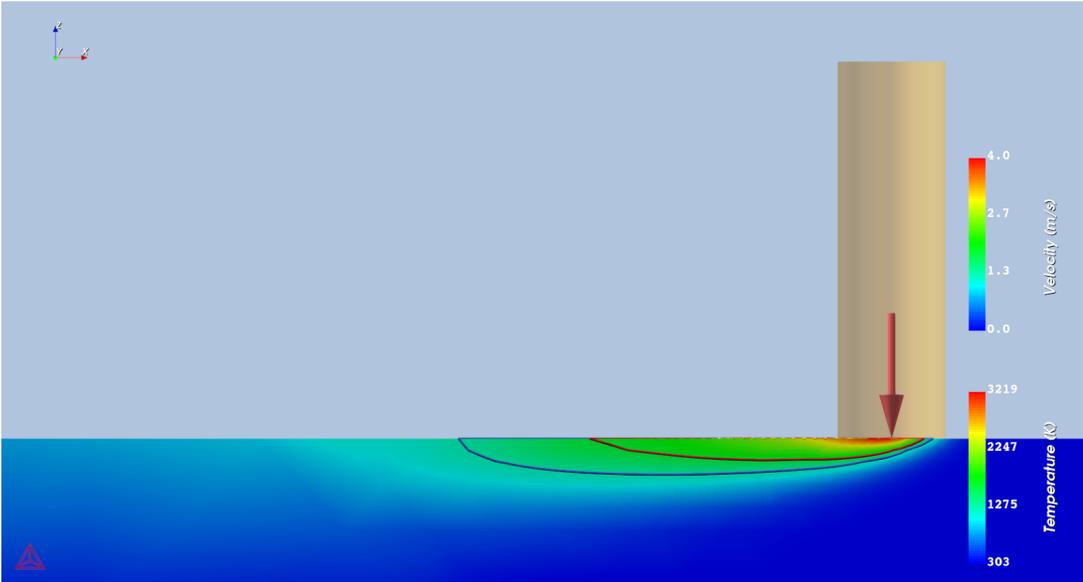
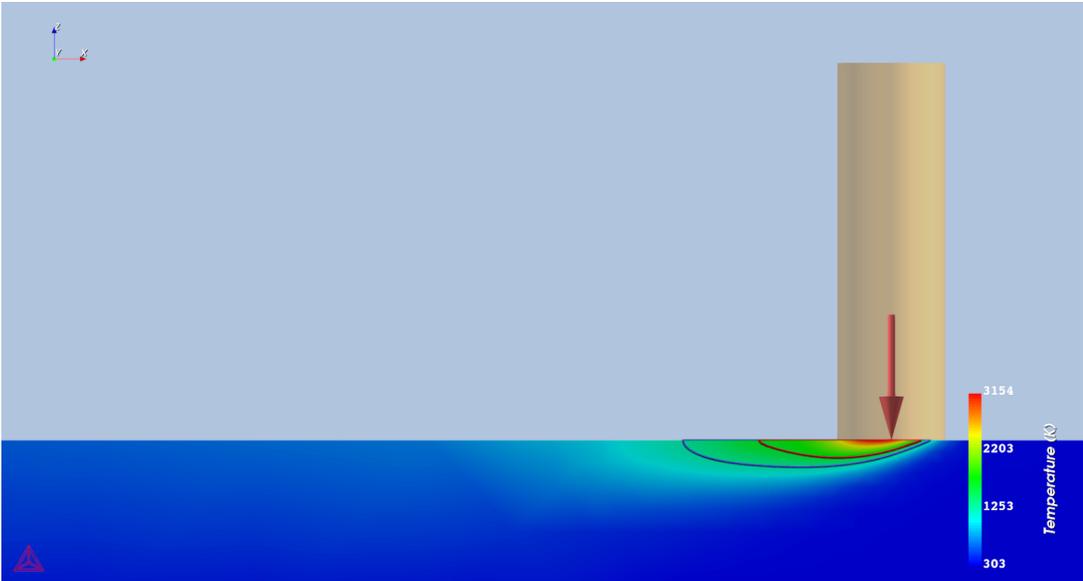
- 



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

---

*Results Discussion*



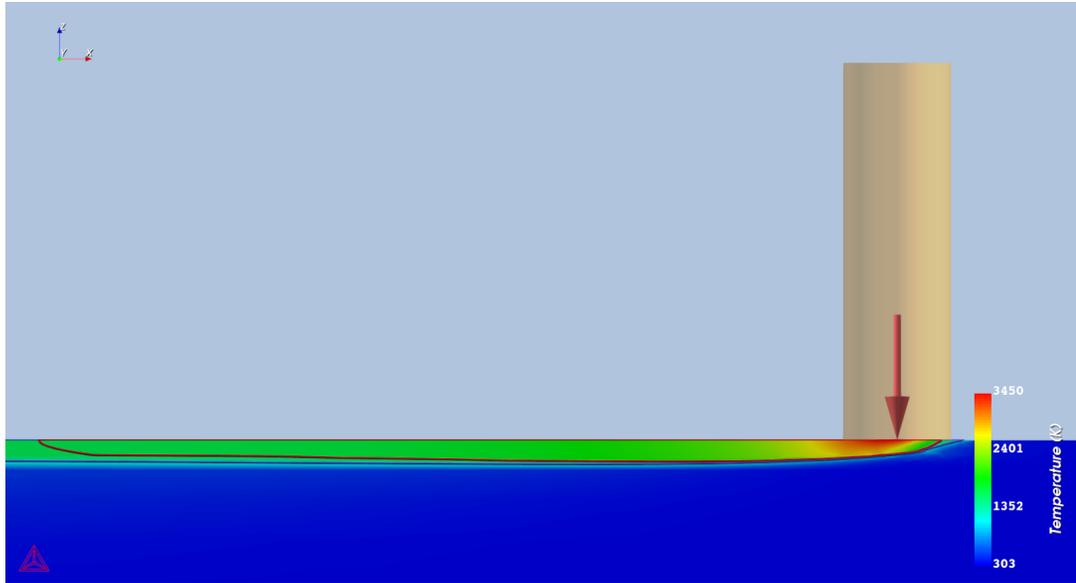


Figure 14: A comparison of the melt pool depth for different simulation conditions; (top) without fluid flow, (middle) with fluid flow in the melt pool and (bottom) with different material properties for the powder layer. Isocontours show the boundaries of (inner) melt pool and (outer) mushy zone.

The table shows a comparison of the melt pool dimensions for different simulation conditions.

Simulation set up	Melt Pool Dimensions (mm)		
	Width	Depth	Length
Without fluid flow	0.156	0.032	0.300
With fluid flow	0.188	0.040	0.619
Separate powder material properties	0.250	0.039	1.670

Figure 14 shows the effect of fluid flow on temperature distribution and size of the melt pool for the given process parameters. A quantitative comparison is also given in the table. It can be seen that with the addition of fluid flow due to gradient in surface tension (Marangoni effect), the melt pool tends to increase its dimensions in all three directions. This is because the negative gradient in surface tension with respect to the temperature creates an outward flow on the surface of the melt pool which consequently increases the width and length of the melt pool, as shown in Figure 15.

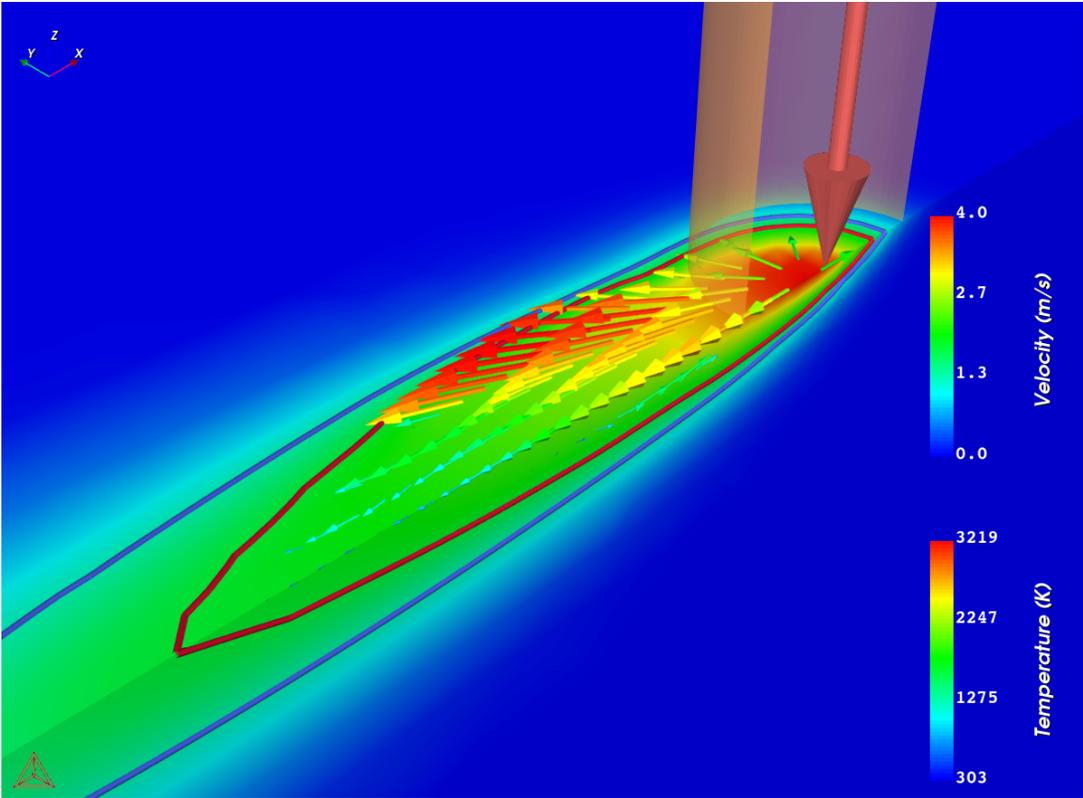
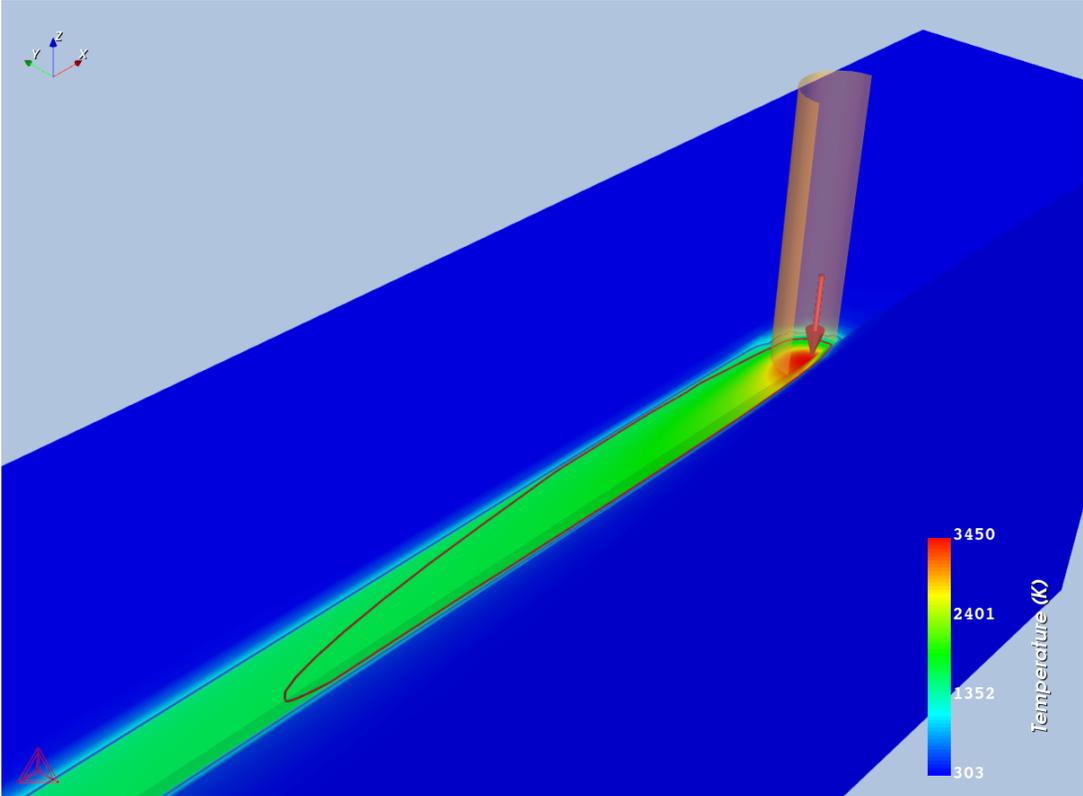
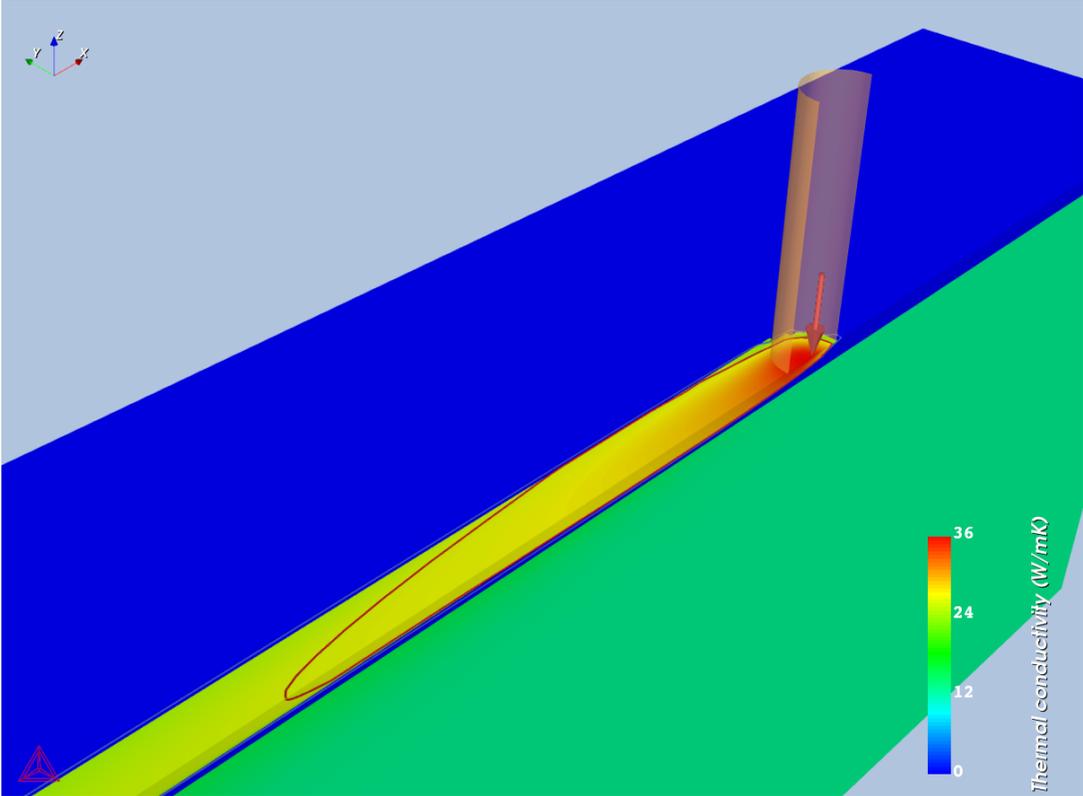


Figure 15: Vectors showing flow field in the melt pool due to gradient in surface tension. The colours of the vectors represent the magnitude of the velocity. Surface colormap shows the temperature distribution.



*Figure 16: (top) Surface colormap of temperature dependent thermal conductivity showing different values in powder and liquid/solid and (bottom) the corresponding temperature distribution.*

Using separate material properties for powder has a significant effect on both the temperature distribution and the size of the melt pool. This is because a lower thermal diffusivity in the powder makes the temperature to diffuse more towards the length of the melt pool hence causing a significant increase in the length of the melt pool (shown in [Figure 16](#)) as compared to the first two cases. There is also an obvious increase in the width of the melt pool while the depth of the melt pool is almost the same as with the fluid flow. Furthermore, the maximum temperature is also highest in the case with separate material properties.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation(s).

- **Geometry:** View the geometry and adjust parameters visually by changing inputs on the Configuration window.
- Melt pool and other dynamically changing features can also be visualized and changed.
- **Plot** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, click a **Plot Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

## Other Resources



Read more about the [Additive Manufacturing \(AM\) Module](#) on our website including the details about database compatibility or to watch an [introductory webinar](#). You can also use the [Getting Started Guide](#) to learn about the key features available.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help** → **Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## AM\_04: Scheil Transient Steady-state

---

In this example, three different simulations are performed: Steady-state, Transient Single track, and Transient Multilayer. Both Transient\_Single track and Transient\_Multilayer use **Transient with heat source from Steady-state** model to compute time-dependent temperature distribution in the given geometry. The primary difference between this example and the previous examples (AM\_01 to AM\_03) is that, in this case, the material properties data is retrieved from the Scheil Calculator whereas in the previous examples the material properties are taken from the preinstalled material library. Furthermore, this example simulates scanning of two layers of powder where the second layer is spread on the top of the first layer when scanning of the first layer is completed.



Material chosen for this example is Ti-6Al-4V. To run this example requires both version TCTI5.0 and newer of the TCS Ti/TiAl-based Alloys Database (TCTI) in addition to a license for the Additive Manufacturing (AM) Module. Some portions of this example are also covered in the [Getting Started Guide](#) on our website.

### Project File Information

- Folder: **Additive Manufacturing**
- File name: *AM\_04\_Scheil\_TransientSS.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes several hours to complete the calculations.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation(s).

- **Geometry:** View the geometry and adjust parameters visually by changing inputs on the Configuration window.
- Melt pool and other dynamically changing features can also be visualized and changed.

- **Plot** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, click a **Plot Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window.

### *Steady-state*

The Steady-state calculator of this example computes temperature distribution in a steady state for the power of 100W and scanning speed of 600mm/s. The beam radius in this example is taken as 100 $\mu$ m. For the geometry, the steady-state model only needs the height of the solid substrate as the input, which is taken as 2.0mm. The solid substrate is covered with a powder layer of thickness 55 $\mu$ m. This example also takes into account the effect of fluid flow due to Marangoni forces inside the melt pool.

---

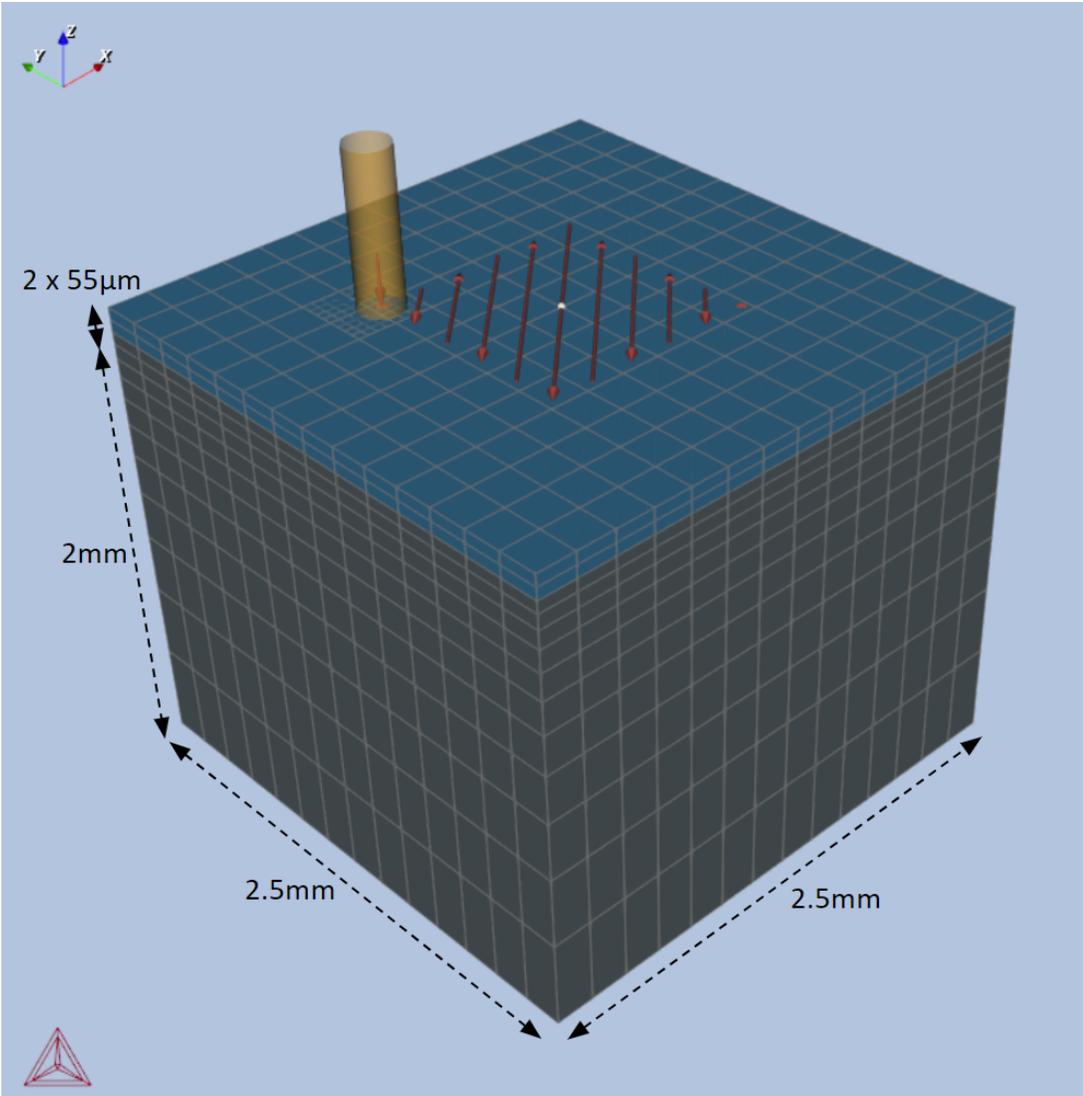


Figure 17: Geometry definition for Transient Multilayer. The scanning strategy shows the scanning pattern on the top most layer.

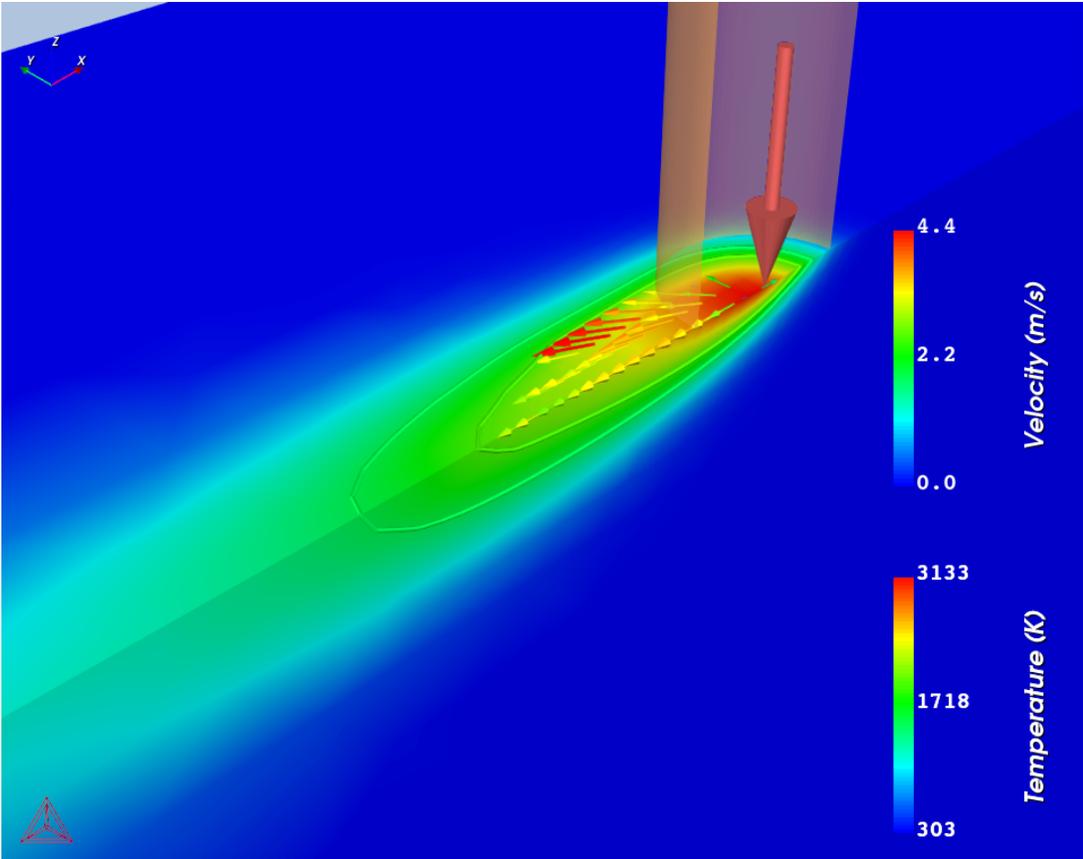


Figure 18: Temperature distribution around the melt pool and mushy zone for the Steady-state case. Velocity vectors representing the fluid flow inside the melt pool can also be seen.

Transient Single Track

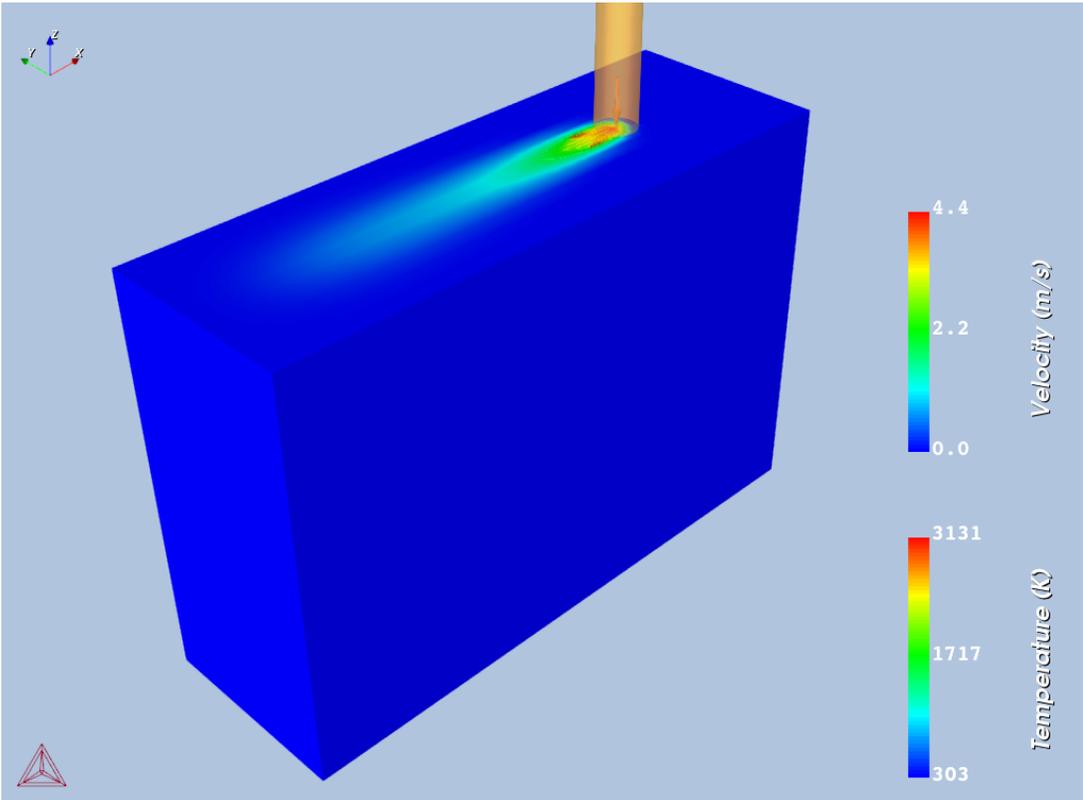


Figure 19: Surface colormap showing temperature distribution at the end of single track transient simulation. The velocity vectors around the heat source show the direction of fluid flow in the melt pool.

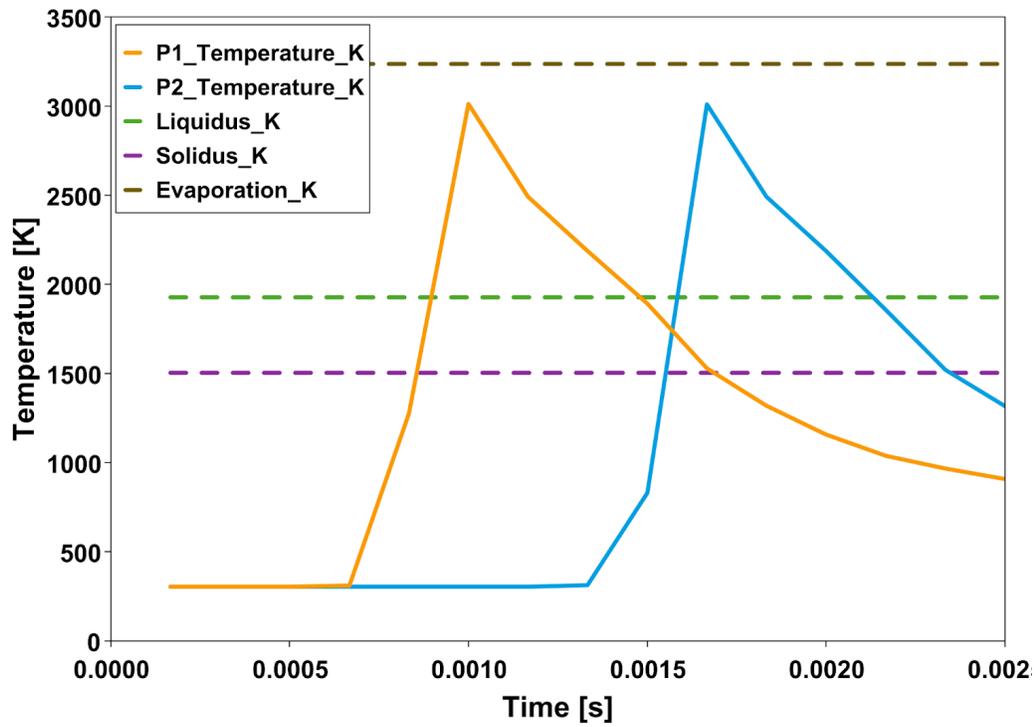


Figure 20: 2D Plots showing temperature as a function of time from Probe 1 (orange) and Probe 2 (blue) after the single track scan is completed.

Transient Multilayer

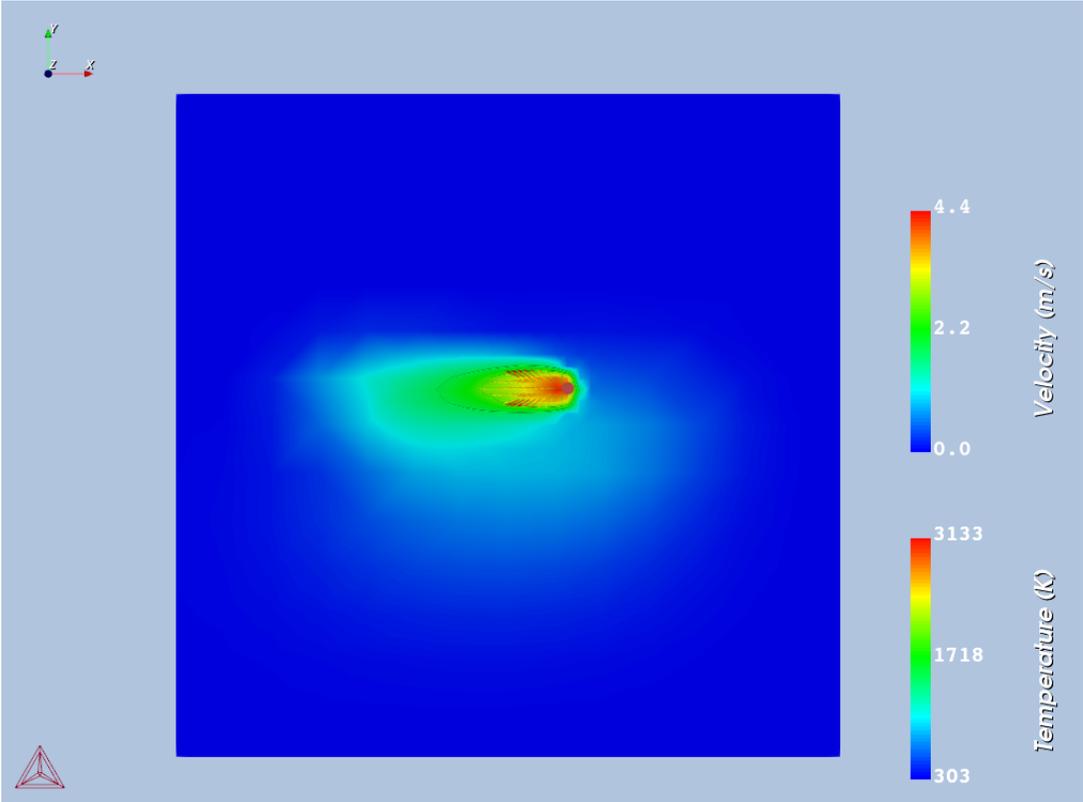


Figure 21: Temperature distribution on the top surface of the workpiece during scanning of the first layer.

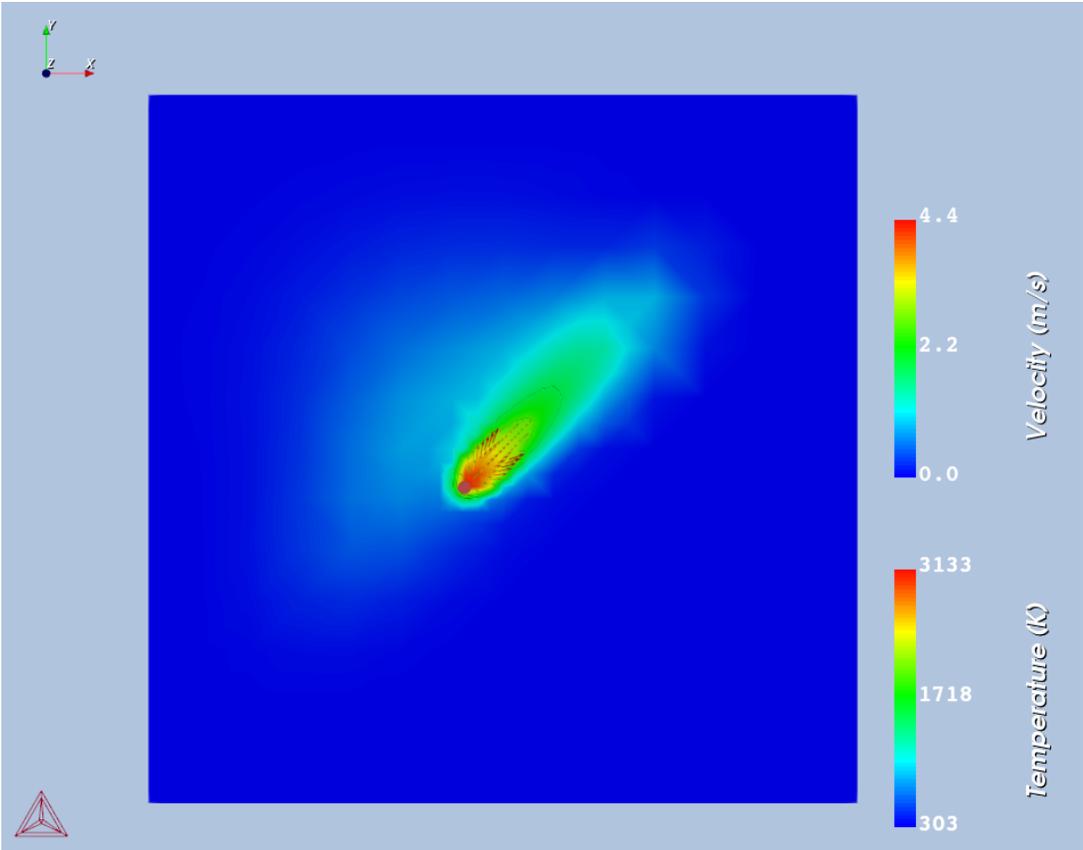


Figure 22: Temperature distribution on the top surface of the workpiece during scanning of the second layer.

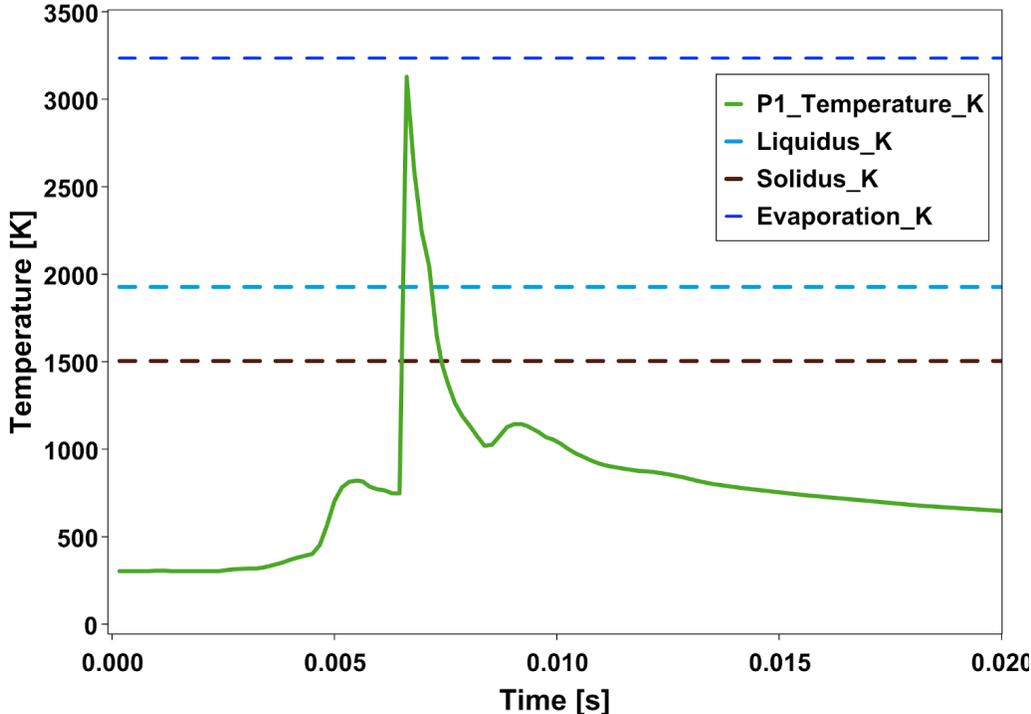


Figure 23: 2D Plots showing temperature as a function of time from Probe 1 after scanning the first layer.

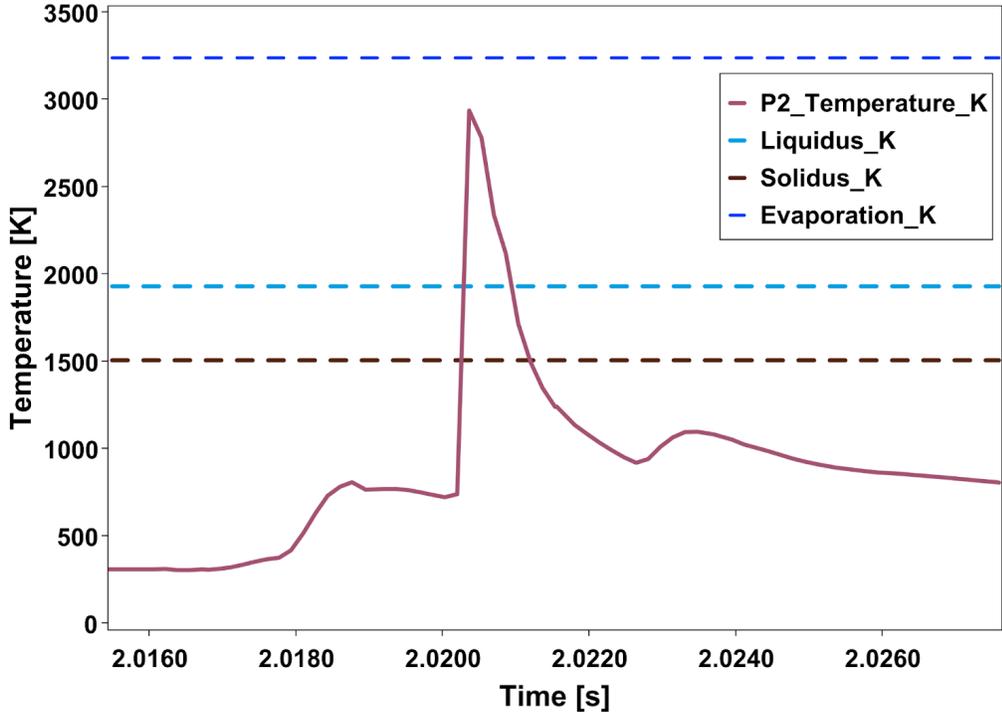


Figure 24: 2D Plots showing temperature as a function of time from Probe 2 after scanning the second layer.

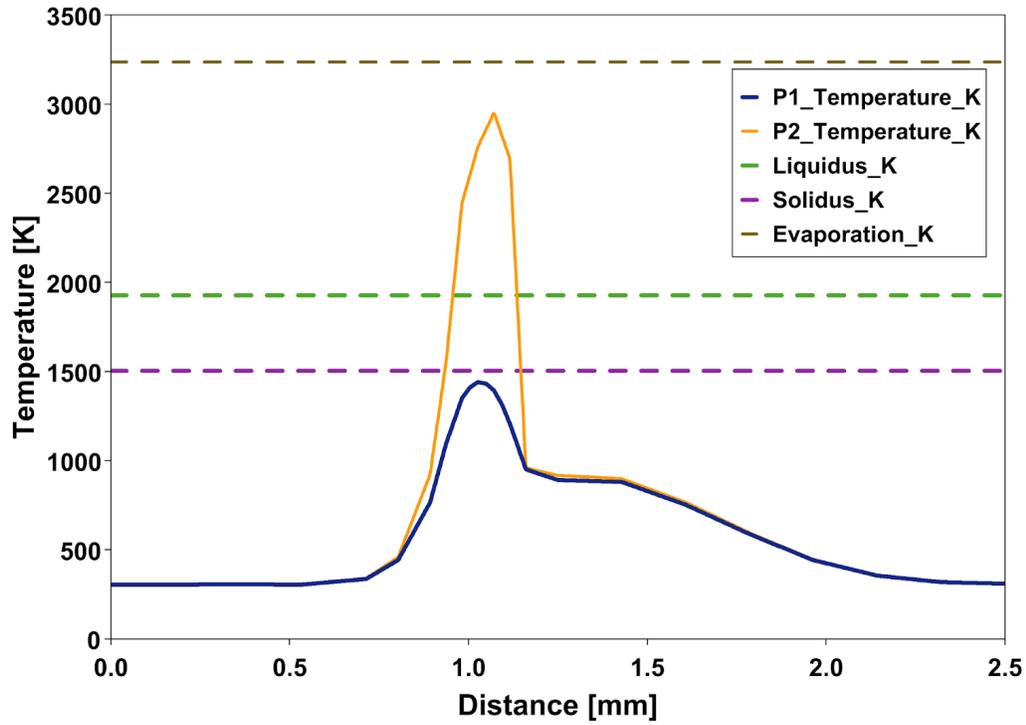


Figure 25: 2D Plot over line showing temperature profiles plotted along the lines, parallel to the width of the geometry placed on the top of the first layer (P1) and the second layer (P2).

## AM\_05: Using AM Calculator Probe Data with the Diffusion Module (DICTRA)

---

The microstructure of a hot-work tool steel additively manufactured using laser powder-bed fusion (L-PBF) is studied in detail by microstructure characterization and computational thermodynamics and kinetics by C.-Y. Chou et al. [2021Cho].

This example demonstrates the application of adding probes to an AM Calculator and then using this data via the Thermal Profile that is set up on a connected Diffusion Calculator. The example also uses a Scheil Calculator to collect some materials data that is then further used with the AM Calculator prior to using the probe data generated by the AM Calculator.



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. This example also requires a license for the Diffusion Module (DICTRA), plus additional database licenses for the TCS Steel and Fe-alloys Database (TCFE) (TCFE13 and newer), and the TCS Steels/Fe-Alloys Mobility Database (MOBFE) (MOBFE8 and newer).

### **Project File Information**

- Folder: **Additive Manufacturing**
- File name: *AM\_05\_AM\_Probe\_to\_Diffusion.tcu*

Open the example project file and click **Perform Tree** to generate the plots associated with it.

### **Background**

According to the thermodynamics of this steel,  $\delta$ -ferrite is the solid phase stable at the highest temperature and would be the first solid phase to form during solidification from a thermodynamic point of view, see [Figure 26](#).

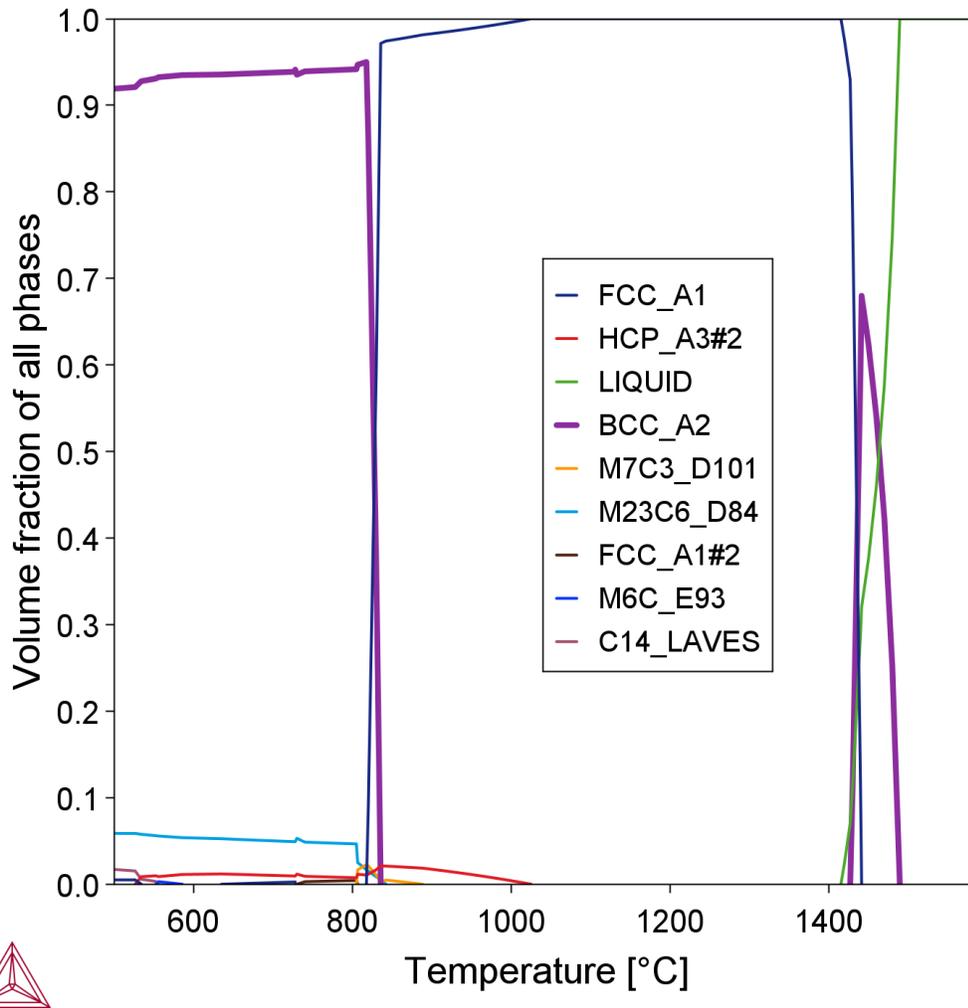


Figure 26: Equilibrium property diagram calculated in Thermo-Calc of the tool steel showing  $\delta$ -ferrite would be the first solid phase to form during solidification according to equilibrium.

However, the high solidification and cooling rates during the L-PBF process lead to suppression of  $\delta$ -ferrite and instead solidification of an austenite phase directly containing a cellular substructure where the alloying elements have segregated to the inter-cellular regions.

The microsegregation can be predicted by reducing the complex solidification behavior to a diffusion problem in one dimension enabling comparisons with the measured segregation profiles quantified at a nanometer scale.

## Configuration and Calculation Set Up

For all calculations, the simplified composition Fe-0.35C-4.93Cr-0.45Mn-2.24Mo-0.25Si-0.54V (mass%) was used with  $\delta$ -ferrite suspended on the System Definer.

The **Additive Manufacturing** template was used to first add the System Definer, Scheil Calculator, AM Calculator and Plot Renderer to the Project window tree. Additionally, a Diffusion Calculator was added as a successor to the AM Calculator in order to automatically import the time-temperature profile from the AM simulation into the diffusion simulation. See [Figure 27](#) (or refer to the example if you have it open) to see the layout.

---

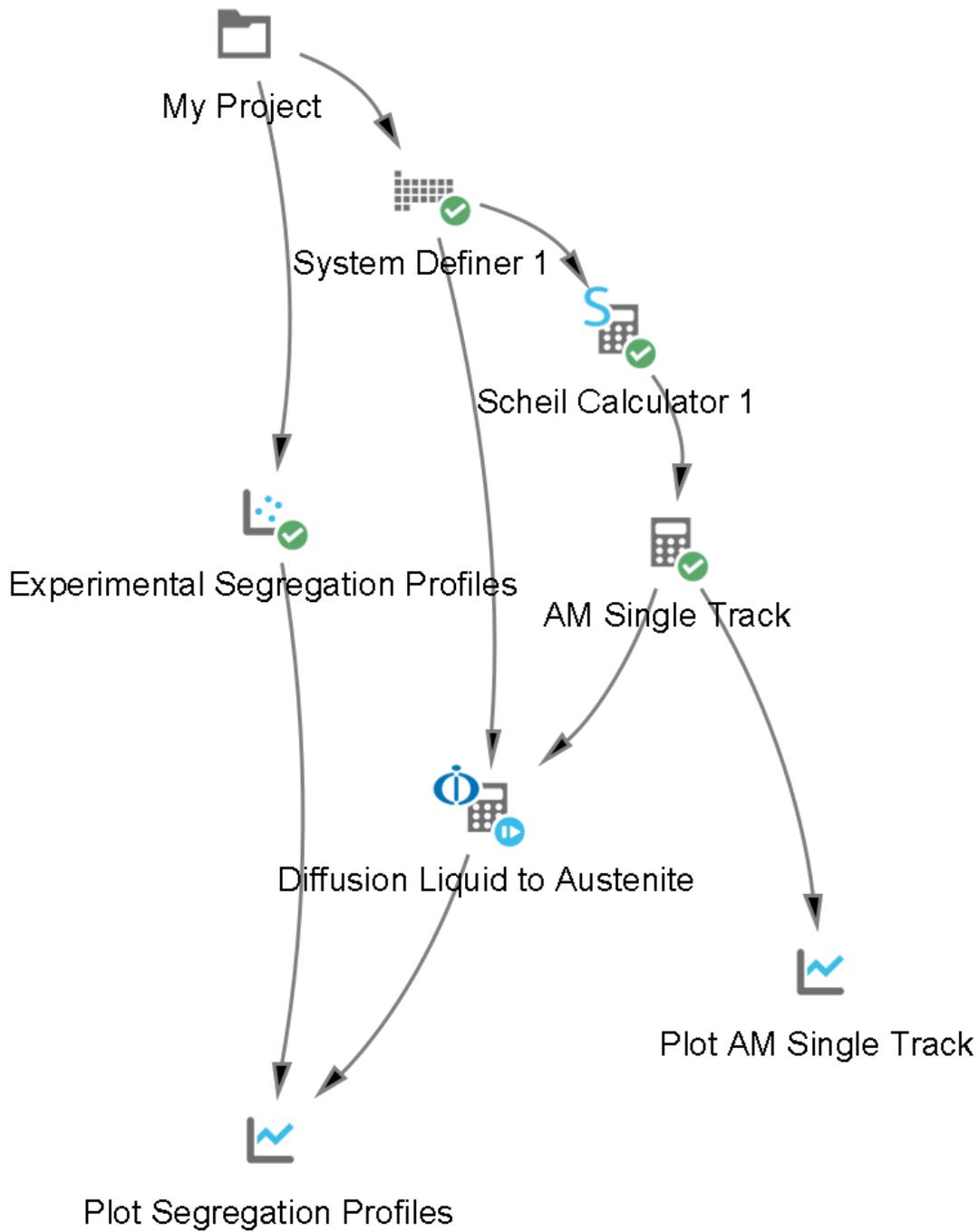
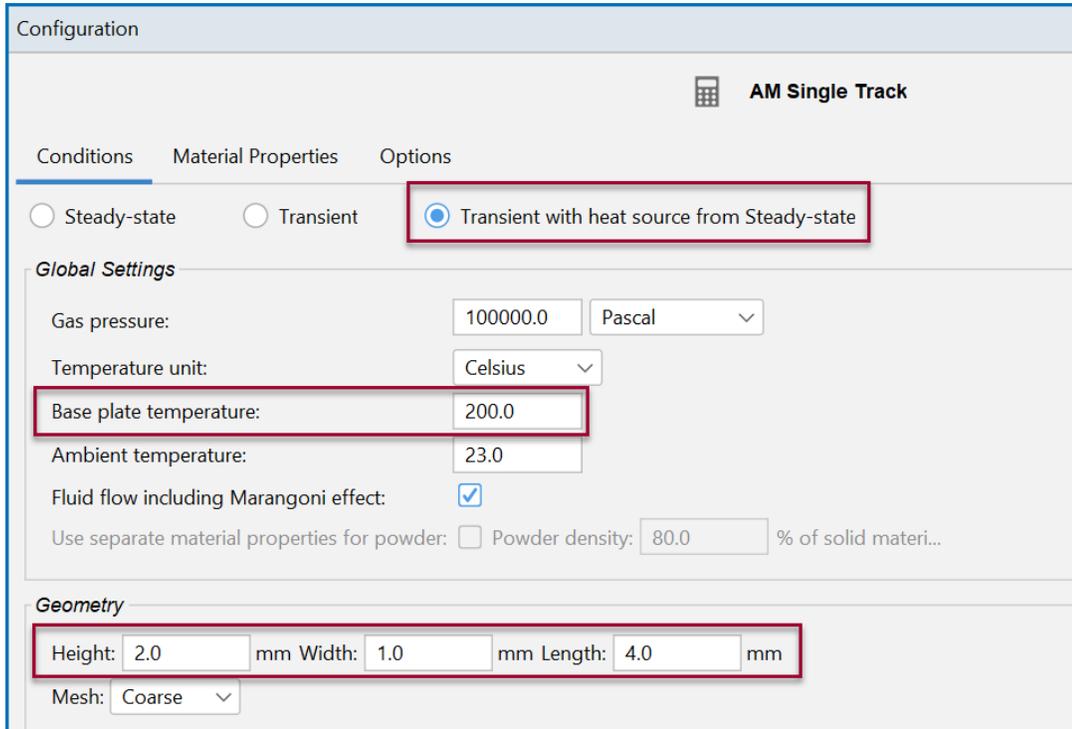


Figure 27: Project tree of the example containing activities from the default Additive Manufacturing template with additional Diffusion Calculator and Experimental File Reader nodes added, where the segregation profiles datasets are pulled from the Experimental File Reader.

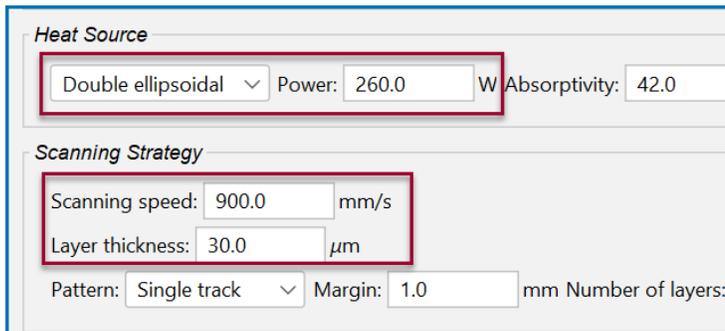
On the **AM Calculator Configuration** window, the calculation type **Transient with heat source from Steady-state** is selected with a single track scan pattern and a **Geometry** of 2 mm x 1 mm x 4 mm (height x width x length). The **Base plate temperature** is set to 200 °C.



The screenshot shows the 'Configuration' window for 'AM Single Track'. The 'Conditions' tab is active, and the 'Transient with heat source from Steady-state' radio button is selected. The 'Global Settings' section includes: Gas pressure: 100000.0 Pascal; Temperature unit: Celsius; Base plate temperature: 200.0; Ambient temperature: 23.0; Fluid flow including Marangoni effect: checked; Use separate material properties for powder: unchecked; Powder density: 80.0 % of solid materi... The 'Geometry' section shows: Height: 2.0 mm, Width: 1.0 mm, Length: 4.0 mm, and Mesh: Coarse.

Additional settings are then made on the **Configuration** window as follows.

The **Double ellipsoidal** volume heat source was previously assessed as a function of linear energy density for a similar material and directly applied to the experimental conditions of a volumetric energy density of 80 J/mm<sup>3</sup>. The laser **Power** is entered at 260 W, **Scanning speed** at 900 mm/s, and powder **Layer thickness** 30 µm. Below is a snapshot of the settings.



The screenshot shows the 'Heat Source' and 'Scanning Strategy' sections. The 'Heat Source' section has 'Double ellipsoidal' selected, Power: 260.0 W, and Absorptivity: 42.0. The 'Scanning Strategy' section has Scanning speed: 900.0 mm/s, Layer thickness: 30.0 µm, Pattern: Single track, Margin: 1.0 mm, and Number of layers: (empty).

A probe was then added in the center of the scan track at a distance of 1.3 mm from the edge. The **Probe Positions** can either be entered directly on the Configuration window or click the **Pick coordinate** button and then click on the geometry in the **Visualizations** window to directly choose the probe point (or points if there is more than one probe added).

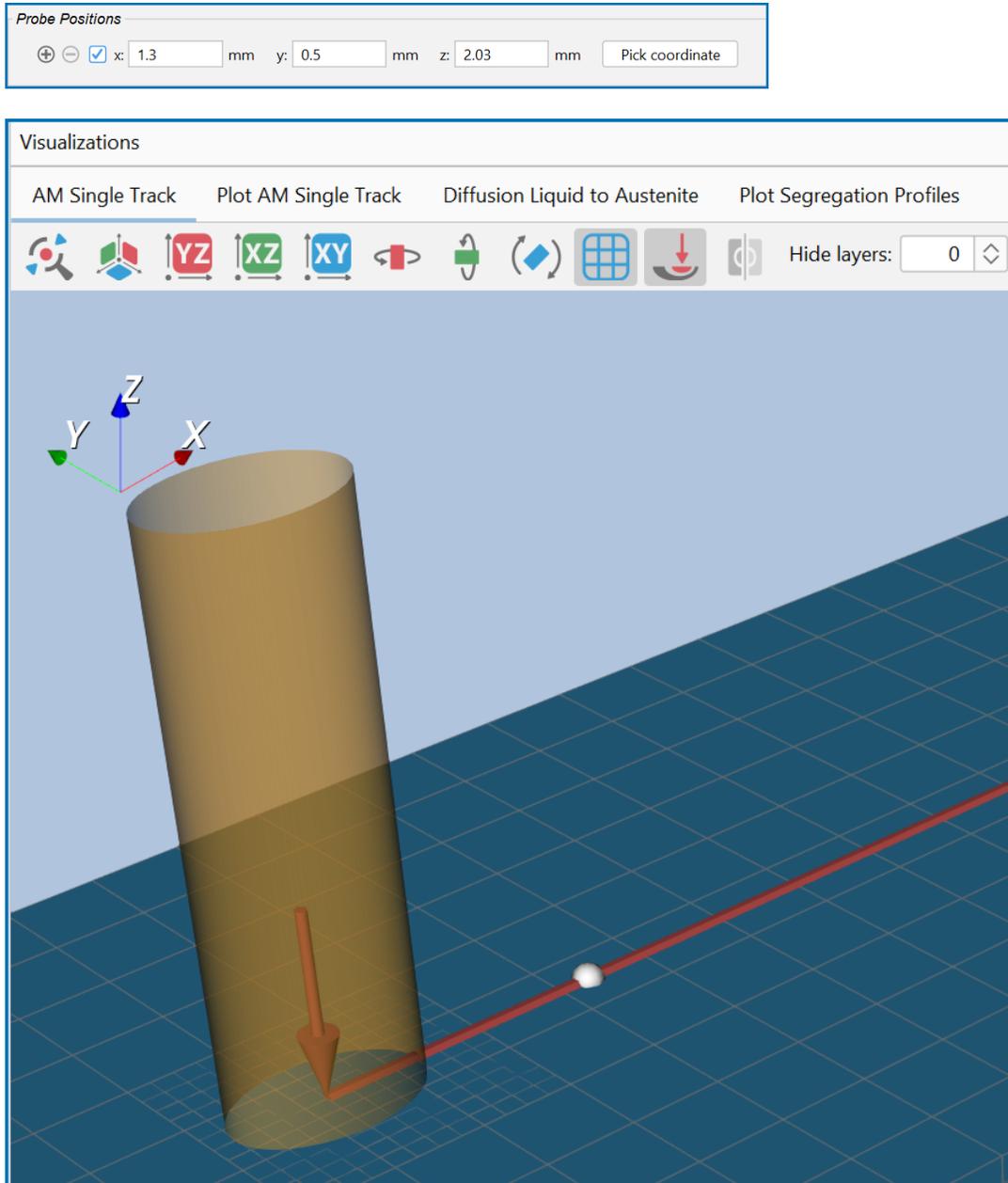


Figure 28: Close up of the probe point added to the geometry for the single track AM simulation.

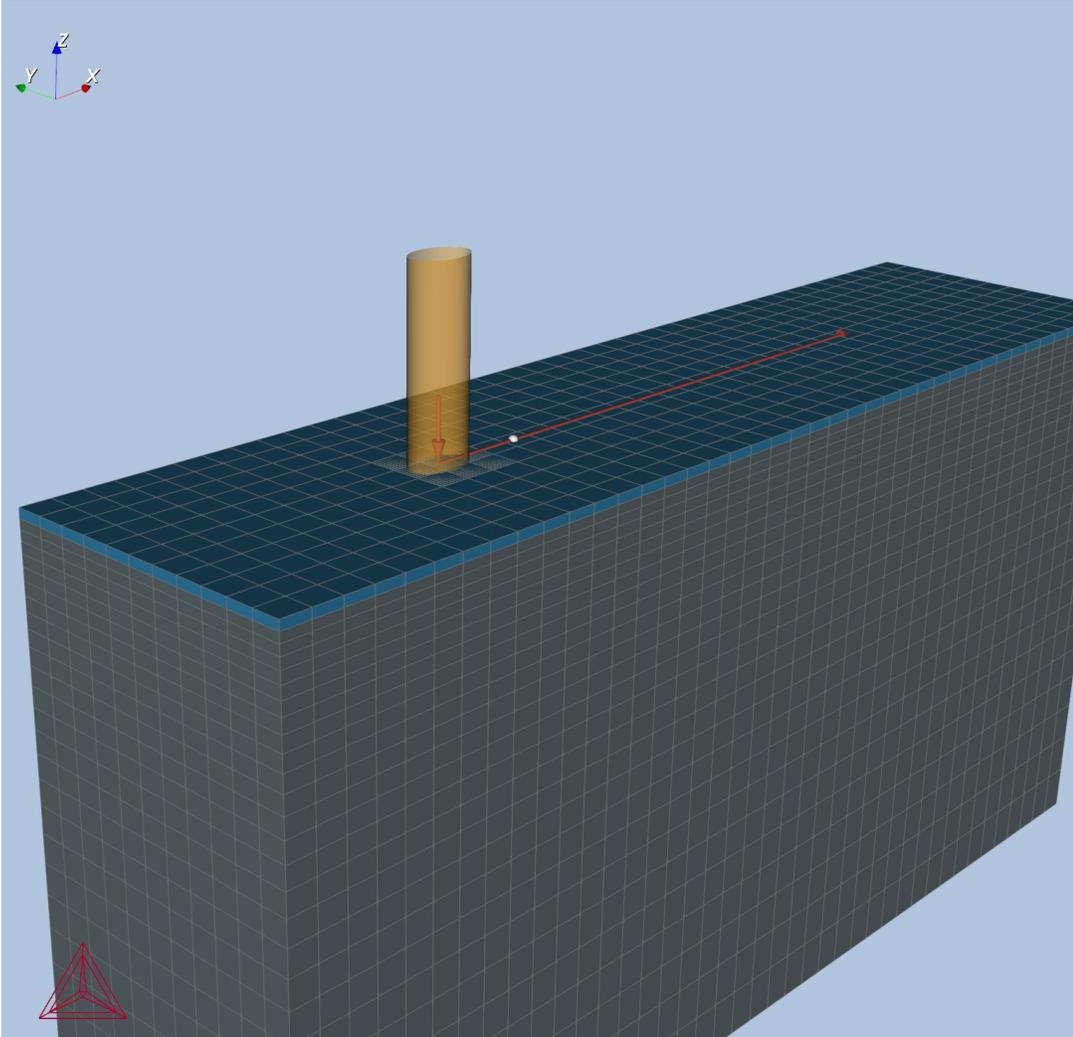


Figure 29: The full geometry set up for the single track AM simulation with a probe point added to the middle of the scan. Open the example to better see the set up and to try working within the window.

## Diffusion Calculations

For the Diffusion Module (DICTRA) solidification calculations using the **Diffusion Calculator**, only the liquid to austenite (FCC) transformation was simulated. On the **Diffusion Calculator Configuration** window, a **Region** of 140 nm with liquid and FCC allowed to form at the right boundary is set.

Configuration

 **Diffusion Liquid to Austenite**

region 1

Region1

*Left Interface Boundary*

▼ Show left interface boundary

**Region**

Name:

Width:  nm  points and type

Phase:

In the *Thermal Profile* section, the time temperature from the probe was selected between a **Start time** of 5.0E-4 and **Simulation time** of 8.5E-4 s in order to only simulate the solidification, which can be viewed on the **Visualizations** window as in [Figure 30](#). Note that the probes need to be added and then run in the AM Calculator to make this data available in for this part of the calculation.

*Thermal Profile*

Temperature unit:

Start time:  Simulation time:

Probes:

Condense time-temperature data:

Time [s]	Temperature [°C]
0.0	30.0
1.11111111111111...	30.082552233050023

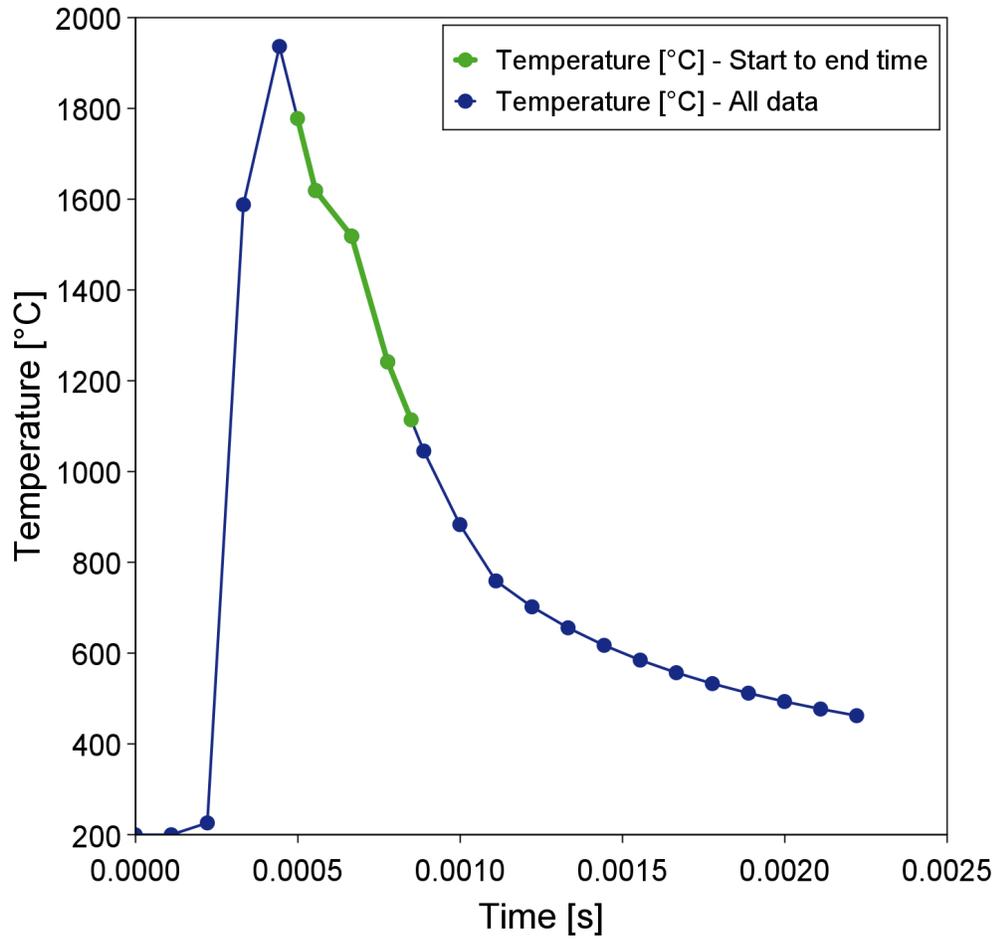


Figure 30: Time temperature profile of the probe in the AM simulation. The green curve show the part of the profile used during the diffusion simulation with the Diffusion Calculator.

The calculated segregation profiles are finally compared with the STEM-EDS line scans as shown in [Figure 31](#).

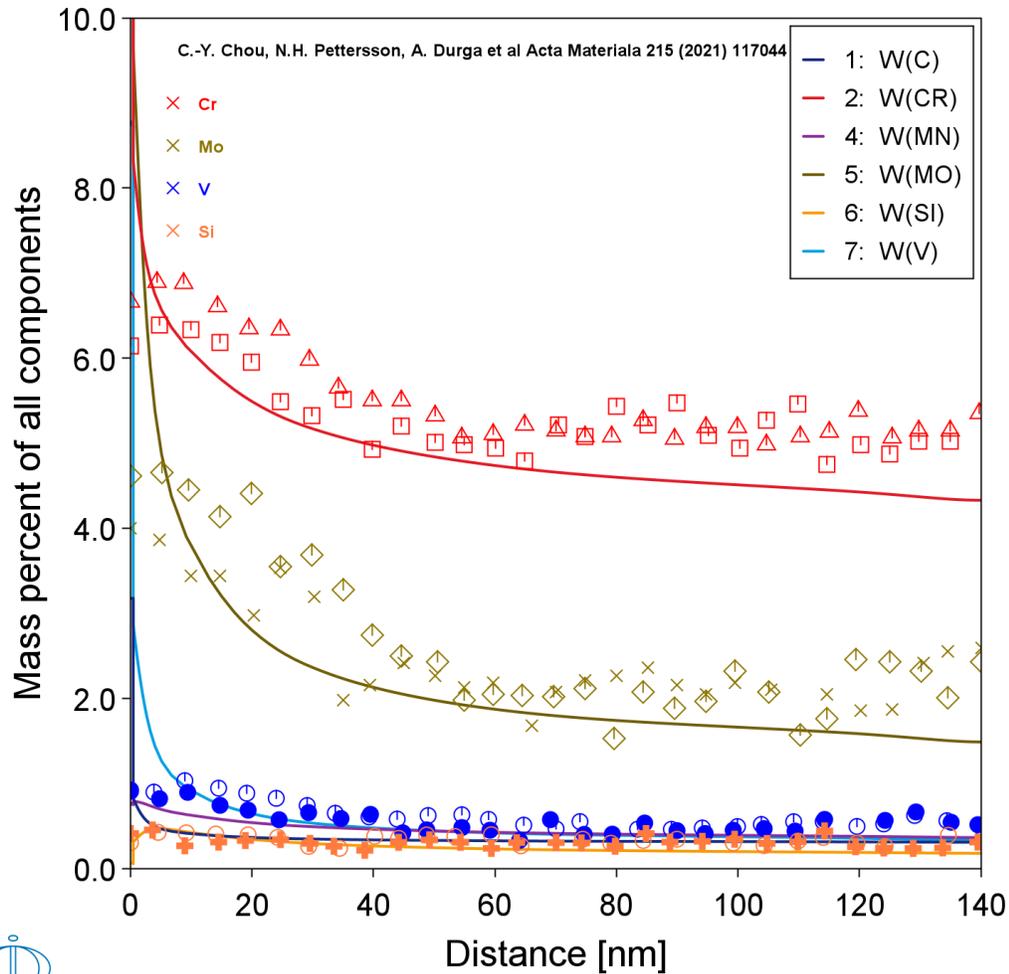


Figure 31: The simulated segregation profile with overlaid experimental STEM-EDS line scans from C.-Y. Chou et al. [2021Cho].

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation(s). This example highlights some of these capabilities:

- **Probes:** Probes are added to the AM Calculator during the set up either by entering coordinates or directly adding these to the geometry. See [Figure 28](#) and [Figure 29](#) for example.
- **Thermal Profile:** The Diffusion Calculator Thermal Profile settings automatically include the probe data and you can visualize to help you continue the simulation. See [Figure 30](#).

- **Plot** results: After completing the set up and performing the calculation, to view the matching name of the node on tab(s) in the **Visualizations** window, click a **Plot Renderer** node in the **Project** window or click the tabs individually in the **Visualizations** window. See [Figure 31](#).

## Reference

[2021Cho] C.-Y. Chou, N. H. Pettersson, A. Durga, F. Zhang, C. Oikonomou, A. Borgenstam, J. Odqvist, G. Lindwall, Influence of solidification structure on austenite to martensite transformation in additively manufactured hot-work tool steels. Acta Mater. 215, 117044 (2021).

## Other Resources



Read more about the [Additive Manufacturing \(AM\) Module](#) on our website including the details about database compatibility or to watch an [introductory webinar](#). You can also use the [Getting Started Guide](#) to learn about the key features available.



Many of our Graphical Mode examples have video tutorials, which you can access in a variety of ways. When in Thermo-Calc, from the menu select **Help → Video Tutorials**, or from the main My Project window, click **Video Tutorials**. Alternately, you can go to the [website](#) or our [YouTube channel](#).

## AM\_06a: Calibrating a Heat Source for a 316L Steel

This example demonstrates the calibration of the heat source using the Gaussian as well as Double ellipsoidal heat sources. In this example, experimental data is taken from Hu. et al. [2019Hu] where the authors performed single-track experiments using the selective laser melting (SLM) process on 316L stainless steel for varying processing conditions. The power ranges from 50 W to 100 W while the scanning speed varies from 400 mm/s to 2800 mm/s. The data imported from [2019Hu], consisting of 21 experimental points, is saved in the project file.



Also see the companion example, [AM\\_06b: Using the Calibrated Heat Source for a 316L Steel](#).

### Project File Information

- Folder: **Additive Manufacturing**
- File name:AM\_06a\_Calibrate\_Heat\_Sources\_316L.tcu



An Additive Manufacturing Module license is needed to run AM\_06a.



Some examples (AM\_01, AM\_02, AM\_03, and AM\_06b) use predefined Material Properties libraries that are embedded with the examples for use for all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO mode. See [Available Options](#) for details.

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it can take about an hour to complete the calculations.

## Material Properties

- SS316L: Fe-17.7Cr-12.6Ni-2.67Mo-0.027C-0.73Si-1.64Mn-0.15Cu-0.01P Mass percent
- Database: TCFE13
- The material properties are precalculated, and stored as a built-in material library with the Additive Manufacturing (AM) Module.

## Configuration and Calculation Set Up

### *AM Calculator Configuration Settings*

The AM\_06a example contains two AM Calculators, which are renamed to **AM Calibration Gaussian** and **AM Calibration Double Ellipsoidal**.

- **AM Calibration Gaussian** uses a **Gaussian** heat source with the **keyhole model**.
- **AM Calibration Double Ellipsoidal** uses a **Double ellipsoidal** heat source
- The heat source parameters for both heat sources are calibrated for the given experimental data.
- For both AM Calculators, the **Base plate temperature** and the **Ambient temperature** use a value of 353 K and the **Layer thickness** is 10  $\mu\text{m}$ . The rest of the Configuration settings keep the original defaults.

For **AM Calibration Gaussian**, all of the 21 experimental points that are imported to the *Experiment Data* table are selected to be used in the calibration. This is because, for the Gaussian heat source with the keyhole model, one wants to obtain a single constant value for beam radius as well as absorptivity to correctly predict the melt pool size for a range of power and scanning speed. It is also in order to obtain the best possible value it is good to include as many experiments as you can to calibrate the heat source parameters.

For **AM Calibration Double Ellipsoidal**, only five (5) experiment points are selected to perform the calibration. Two experiments are selected from the lower energy density, two from the higher energy density, and then one experiment is selected in the middle region of the two extremes.

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation(s).



See [Visualizing Heat Source Calibrations](#) for an extended overview of what is available and how to navigate the many options for visualizing the heat source calibration data.

### Calibrated Functions

For **AM Calibration Gaussian**, a comparison of the melt pool dimensions from the experiments and calibration simulation is shown in [Figure 32](#).

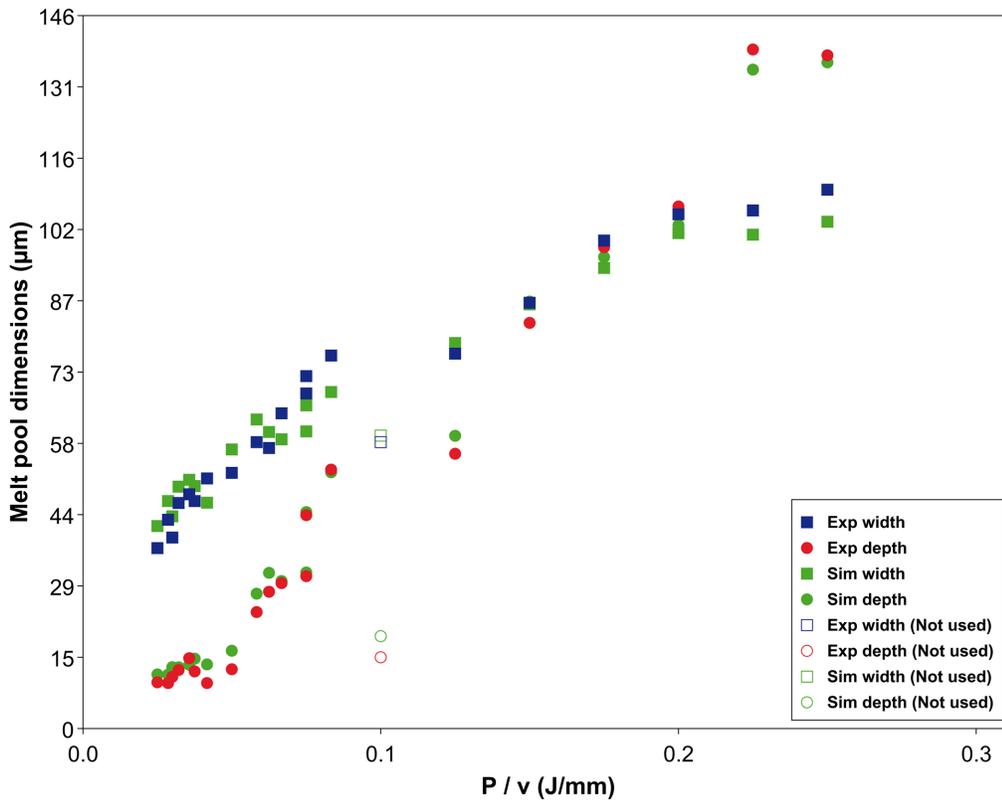


Figure 32: A comparison of melt pool dimension for Gaussian heat source optimization. One of these experimental points, number 10, is later excluded from the double ellipsoidal calculation as it is considered an outlier.

The resulting calibrated heat source parameters for **AM Calibration Gaussian**, are shown in [Figure 33](#).

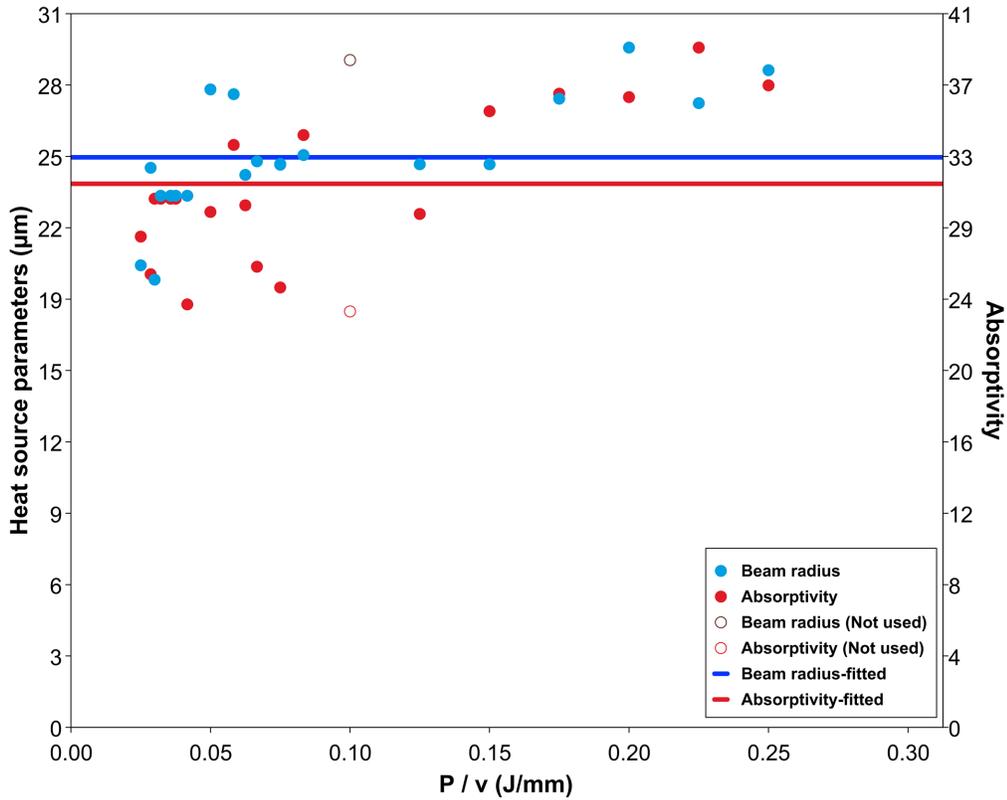


Figure 33: Calibrated heat source parameters for Gaussian heat source optimization.

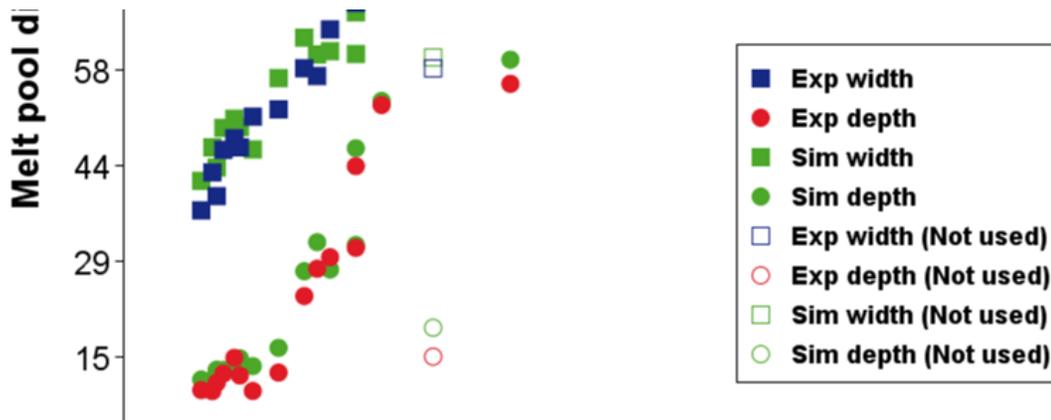


Figure 34: This is an example of how the visualization can change dynamically by using the checkboxes. This is a close up of the original [Figure 32](#) melt pool dimension for a Gaussian heat source optimization where experiment #10 is now not used. This is after the checkbox is deselected in the 'Use in fit' column for row 10 on the Plot Renderer Configuration window. This is also indicated in the legend and by the hollow squares (width) and circles (depth) in the plot.

In order to obtain the expressions for **Absorptivity** and **Beam radius**, one of the experiments (*Experiment # 10*) is not going to be included. This is because *Experiment # 10* is apparently an outlier as can be seen in the melt pool dimensions plot in [Figure 32](#) and then generally shown when not included for the Gaussian plot as per [Figure 34](#).

For both Absorptivity and Beam radius the Constant function of  $P/v$  is chosen to be saved as a calibrated heat source. Absorptivity of the material at the printing conditions is usually unknown in these experiments which is why it is regarded as a fitting parameter. Beam radius, on the other hand, is most often known, but does not match the definition of beam radius in the Gaussian heat source model used in the simulation, which is why a different value is obtained as compared to the one used in the experiment.

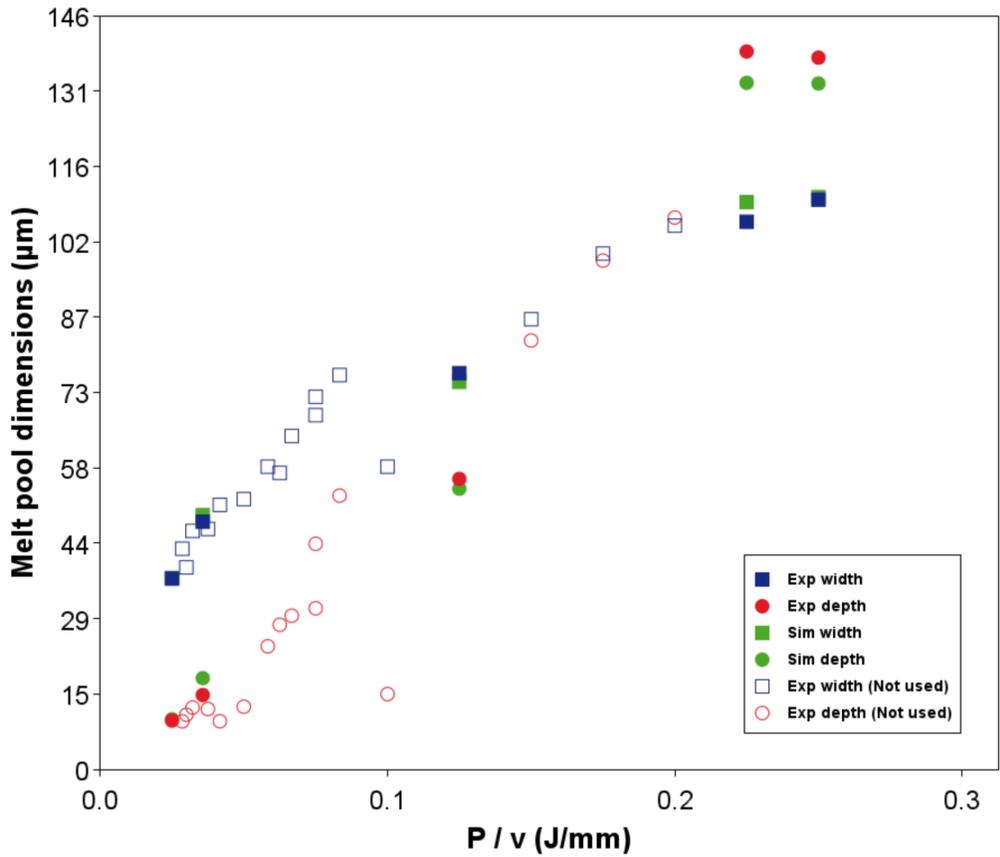


Figure 35: A comparison of melt pool dimension for double ellipsoidal heat source optimization.

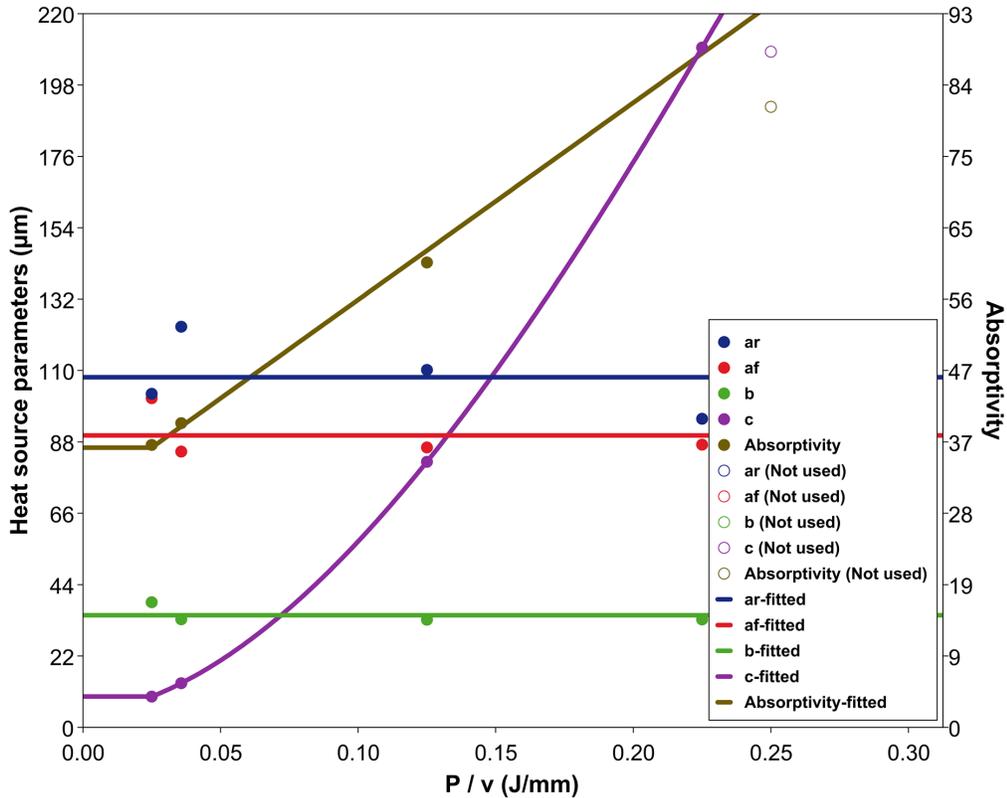


Figure 36: Calibrated heat source parameters for double ellipsoidal heat source optimization.

While selecting the functions for heat source parameters for double ellipsoidal, it was seen that **ar**, **af** and **b** remained almost unchanged for all the experiments. Therefore the Constant function of P/v for ar, af and b is used. For Absorptivity, a Linear function of P/v gave the best R2 value while for c a Cubic function of P/v gave the best R2 value. One of the experiments (experiment # 21) was excluded (i.e. the checkbox is deselected) to get a suitable curve for all functions.

## Reference

[2019Hu] Z. Hu, B. Nagarajan, X. Song, R. Huang, W. Zhai, J. Wei, Formation of SS316L Single Tracks in Micro Selective Laser Melting: Surface, Geometry, and Defects. Adv. Mater. Sci. Eng. 2019, Article ID 9451406, 1–9 (2019).

## AM\_06b: Using the Calibrated Heat Source for a 316L Steel



This example does not require an additional license to run it, although its companion example, AM\_06a, does.



[AM\\_06a: Calibrating a Heat Source for a 316L Steel](#)

This example demonstrates the use of the two calibrated heat sources that are saved in the example AM\_06a. One heat source is of the Gaussian type while the other heat source is Double ellipsoidal. For both heat sources, expressions are saved as a function of energy density ( $P/v$ ) for all heat source parameters as well as the absorptivity. The example consists of four AM Calculators, two of which use a Gaussian heat source while the other two use a Double ellipsoidal heat source, and where the processing conditions for all four AM Calculators are taken from Hu et al [2019Hu].

### Project File Information

- Folder: **Additive Manufacturing**
- File name: AM\_06b\_Use\_Calibrated\_Heat\_Sources\_316L.tcu

Open the example project file and click **Perform Tree** to generate the plots associated with it.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

### Material Properties

- SS316L: Fe-17.7Cr-12.6Ni-2.67Mo-0.027C-0.73Si-1.64Mn-0.15Cu-0.01P Mass percent
- Database: TCFE13
- The material properties are precalculated, and stored as a built-in material library with the Additive Manufacturing (AM) Module.

### Configuration and Calculation Set Up

There are four AM Calculators where all the calculators use the **Steady-state** simulation type and the **Single point Calculation Type**. The purpose of this example is to use the saved heat sources from AM\_06a and then validate the results by comparing the melt pool size from the simulations with the

experimental melt pool dimensions as given in Hu et. al [2019Hu].

The saved heat sources are selected from the *Heat Source* list on the AM Calculator, where a subsection called **Users heat sources** includes all previously saved heat sources. In the example, two calculators use the **Gaussian - 316L - beam d 15um** user-defined heat source, and two use the **Double ellipsoidal - 316L - beam d 15um** user-defined heat source. The **Base plate temperature** and **Ambient temperature** is set to 353 K, while the **Layer thickness** is set to 10  $\mu\text{m}$ .

## Visualizations

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation(s).



For this example, it is best to open and run this simulation to be able to view all the different plots and outputs to various tabs on both the various Configuration windows as well as the Visualizations window.



See [Visualizing Heat Source Calibrations](#) for an extended overview of what is available and how to navigate the many options for visualizing the heat source calibration data.

## Gaussian Heat Source and Keyhole Model

The different processing conditions are selected to simulate both the conduction mode as well as the keyhole mode. For **Gaussian - 316L - beam d 15um** heat source, with  $P = 100 \text{ W}$  and scanning speed = 2800 mm/s, the dominant mode of heat transfer is conduction, which is why a shallow melt pool is obtained. The results from the simulations are fairly close to the ones from the experiments, as can be seen in *Table 1*.

With  $P = 80 \text{ W}$  and scanning speed = 400 mm/s, a keyhole mode is observed in the experiments. This is also predicted from the simulations as can be seen in [Figure 37](#).

A comparison of the melt pool depth from the simulations and the experiments for the latter case shows that the calibrated Gaussian heat source with the keyhole model predicts the melt pool depth with reasonable accuracy. However, it underpredicts the melt pool width. This is because fluid flow due to Marangoni forces is not included in these simulations, which probably causes the melt pool to widen in the experiments.

Calculator Name	Power [W]	Scanning Speed [mm/s]	Melt Pool Width		Melt Pool Depth	
			Simulation [ $\mu\text{m}$ ]	Experiment [ $\mu\text{m}$ ]	Simulation [ $\mu\text{m}$ ]	Experiment [ $\mu\text{m}$ ]
AM use calibrated Gaussian P100 u2800	100	2800	53.79	47.82	14.7	14.36
AM use calibrated Gaussian P80 u400	80	400	90.47	105	103.42	106.57

Table 1: A comparison of the melt pool dimensions from the simulations and experiments using the calibrated Gaussian heat source.

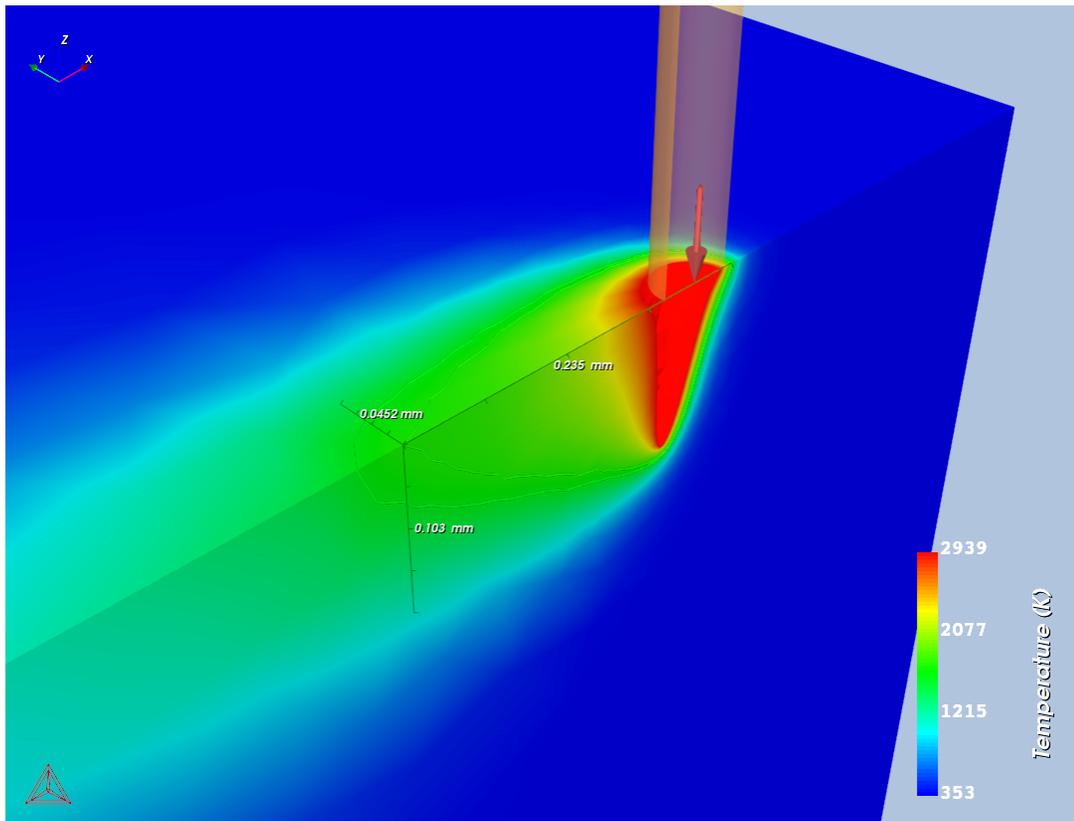


Figure 37: Steady-state temperature distribution around the melt pool for SS316L with  $P = 80$  W and scanning speed = 400 mm/s using the calibrated Gaussian heat source with the keyhole model. A keyhole can also be seen formed just below the location of the heat source.

## Double Ellipsoidal Heat Source

For the **Double ellipsoidal - 316L - beam d 15um** heat source, two different processing conditions are selected to simulate both the conduction mode as well as the keyhole mode. For both processing conditions, the prediction of the melt pool size from the simulations is in fairly good agreement with the results from the experiments as can be seen in *Table 2*.

The temperature distribution using the calibrated double ellipsoidal heat source with  $P = 80$  W and scanning speed = 400 mm/s is shown in [Figure 38](#).

Calculator Name	Power [W]	Scanning Speed [mm/s]	Melt Pool Width		Melt Pool Depth	
			Simulation [ $\mu\text{m}$ ]	Experiment [ $\mu\text{m}$ ]	Simulation [ $\mu\text{m}$ ]	Experiment [ $\mu\text{m}$ ]
AM use calibrated DE P80 u1200	100	1200	63.41	64.34	30.110	29.68
AM use calibrated DE P80 u400	80	400	99.56	105	107.44	106.57

*Table 2: A comparison of the melt pool dimensions from the simulations and experiments using the calibrated double ellipsoidal heat source.*

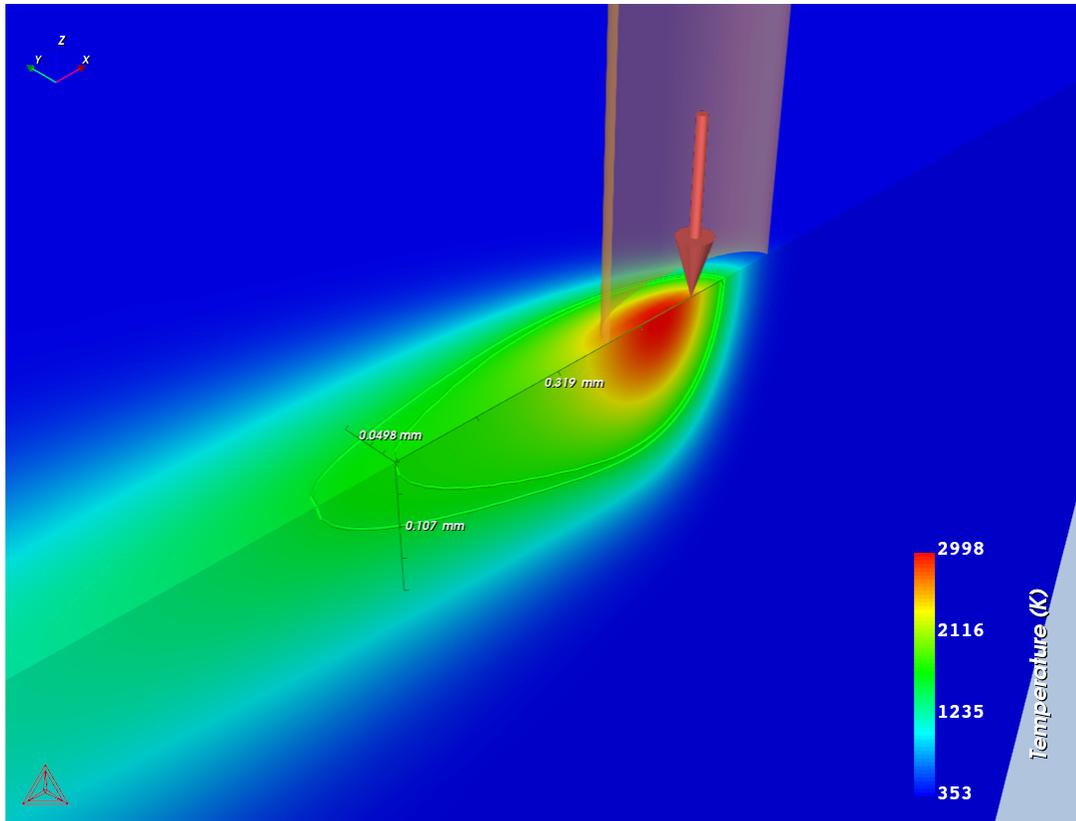


Figure 38: Steady-state temperature distribution around the melt pool for SS316L with  $P = 80$  W and scanning speed = 400 mm/s using the calibrated double ellipsoidal heat source.

## Reference

[2019Hu] Z. Hu, B. Nagarajan, X. Song, R. Huang, W. Zhai, J. Wei, Formation of SS316L Single Tracks in Micro Selective Laser Melting: Surface, Geometry, and Defects. *Adv. Mater. Sci. Eng.* 2019, Article ID 9451406, 1–9 (2019).

# **Thermo-Calc User Guide: Console Mode**

**Thermo-Calc Version 2024a**



---

# Working in Console Mode

---

**In this section:**

---

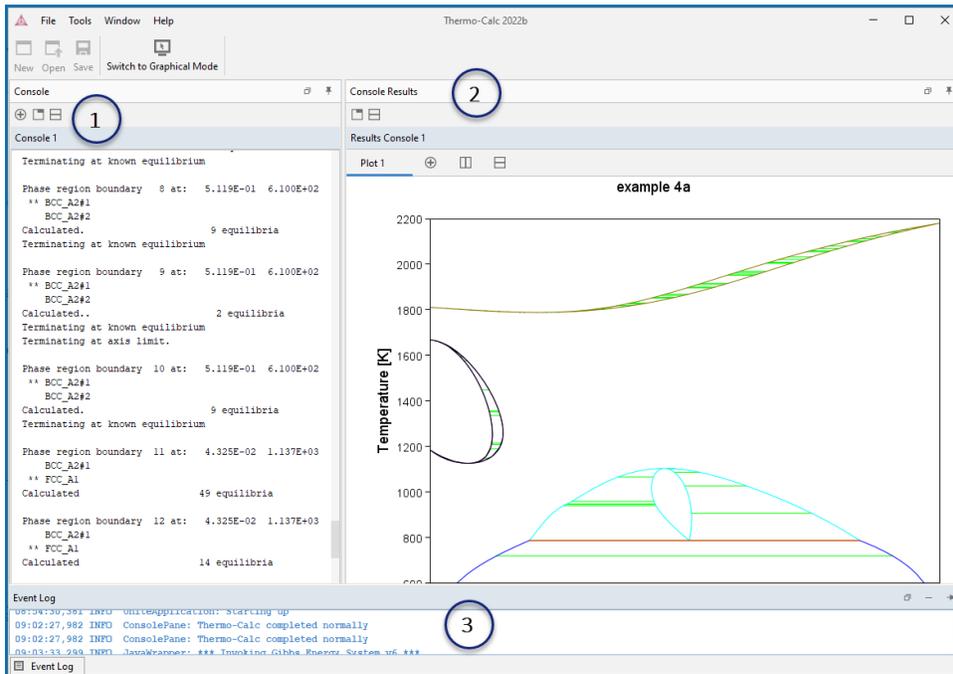
The Console Mode Layout .....	3
Console Window and Tabs .....	6
Console Mode Results Window .....	9
Event Log Window .....	10
Keyboard Shortcuts to Navigate Between Tabs .....	11

## The Console Mode Layout

By default the first time you launch Thermo-Calc it opens in Graphical Mode. After this, the mode defaults to the most recent one. You can switch back and forth between modes. To open the Console Mode interface, click **Switch to Console Mode** on the toolbar.

 [Keyboard Shortcuts to Navigate Between Tabs](#)

### The Console Mode User Interface



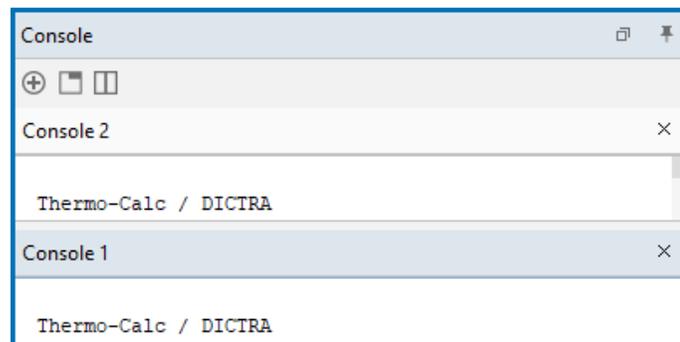
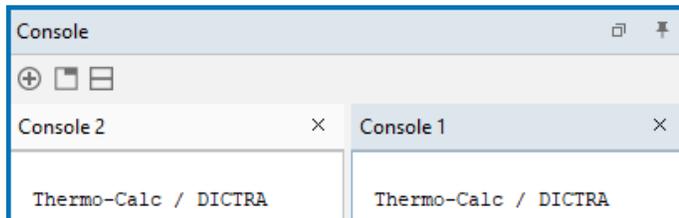
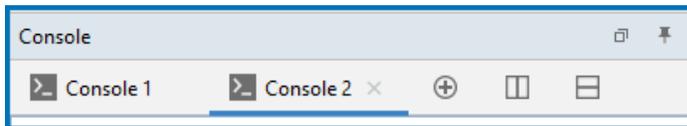
The currently selected window is called the *active* window. This has a darker title bar than the other windows. The windows are:

1. **Console**, with a **Console 1** tab.
2. **Console Results**, with a **Results Console 1** tab as well as a sub-tab (**Plot 1**), which is what is displayed in the window in the figure.
3. **Event Log**.

### Console Tabs

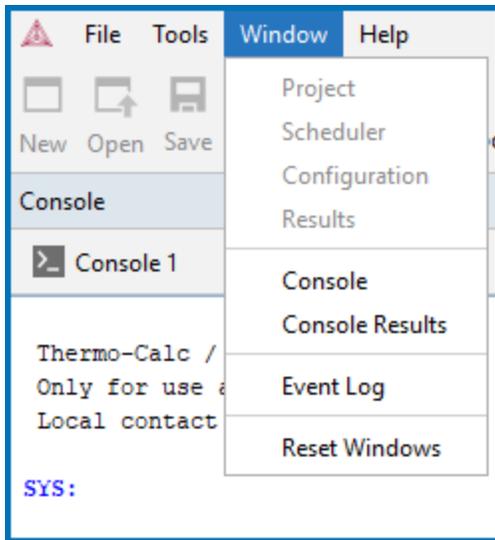
By default, the tabs are side-by-side. You can toggle to view the tabs

- Click the  button to display both windows side by side where you can see the information in all windows at the same time.
- Click the  button to stack the windows top to bottom where you can see the information in all windows at the same time.
- Click the  button to return to the default layout with tabs side by side and the focus on one window at a time.



## ***How to Open Console Windows***

Open a specific window from the **Window** main menu, or select **Reset Windows** to return to the default window layout.



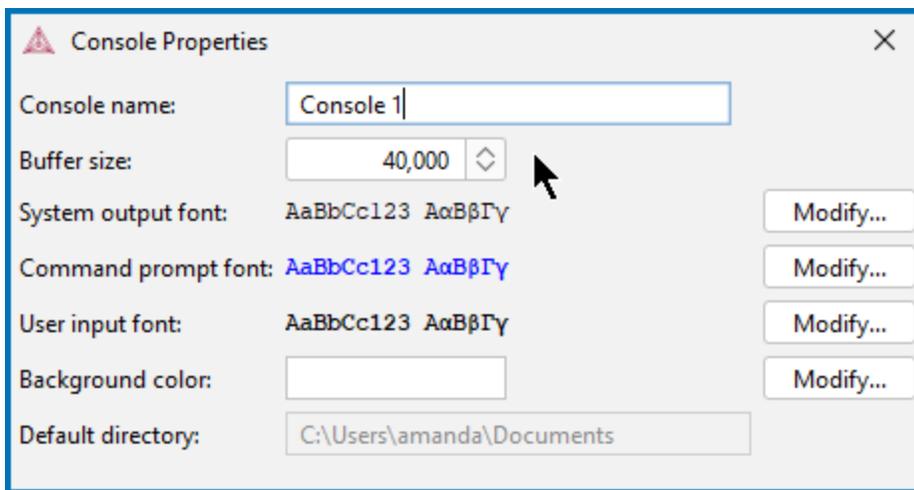
## Console Window and Tabs

By default the Console window is on the left-hand side of the console. This is where you enter commands and within the window there can be several Console tabs open at a time. When you right-click the tab, there are options available from the menu to edit the tab properties, open and close tabs, and view, copy, and save a command history.

### Edit the Console Tab Properties

Right-click any tab and select **Properties** from the menu to edit the Console tab properties.

 [Global Settings: Console Mode Default Appearance](#)



- Change the tab name in the **Console name** field to make it more descriptive for the work you are doing.
- Increase or decrease the buffer size.
- Edit fonts, sizes, and colors for various Console window text. Click **Modify** next to **System output**, **Command prompt**, and **User input**.
- Click **Modify** next to **Background color** to change the color for each Console window.
- To globally set the **Default directory** file path to where the various file types are saved, go to **Tools > Options > Console Mode**.

### Open and Close Console Tabs

There are options to add a **New Console** tab, **Close** one, or **Close all** tabs in the Console window.

---

## Open One or More Console Tabs

- Click the  button.
- Press <Ctrl+T> when the Console window is active.
- Right-click a tab header and select **New Console**.

## Close One or More Console Tabs

- Click the X button.
- Press <Ctrl+W> or <Ctrl+F4>.
- Right-click the tab header and select **Close**.
- To close all tabs except one, right-click the header of the tab you want to keep and select **Close Other**.
- To close all tabs and open a new one, right-click the header of any tab and select **Close All**.

## View, Copy, or Save a Command History

There are commands available to enter at the command line prompt as well as other ways to save or copy the full set of commands in any Console window tab.

### Copy Commands to a Clipboard

- Press `CTRL + Shift + C` in the window or tab where you want to copy the command history. Paste in a text editor as required.
- Right-click any tab and select **Copy Command History to Clipboard** from the menu. Paste in a text editor as required.
- Right-click any tab and select **Command History** from the menu to open the **Command History** window. Then click the **Copy** button () to copy all the commands to the clipboard. Paste in a text editor as required.

### View or Save Commands to a Macro File

Right-click any tab and select **Command History** from the menu to open the **Command History** window. Then click the **Save** button () , enter a file name and path to save all the commands in a `.TCM` file format.

## ***Enter Commands to View the History***

- To scroll through the last twenty commands used, at the command line prompt enter two exclamation marks (! !) and press <Enter>.
- To repeat the last  $n$  commands, at the command line prompt enter ! followed by the number of previous commands to be repeated.
- !<text> executes the most recent command starting with <text>. The < > is not entered at the prompt.
- !\*<n> or !\*<text> makes the command available for editing before execution. The < > is not entered.
- To scroll through previous performed commands, use the Up (↑) or Down arrow (↓) keys.
- To read more information about the command history functionality in Thermo-Calc, enter ! ? at the command line prompt.

---

## Console Mode Results Window

---

By default, the Console Results window is on the right-hand side of the screen. This is where plotted diagrams display. Each Console has its own Results tab in the Console Results window. All the plots and tables generated are presented in the Results tab.

- To open new **Plot** tabs press <Ctrl+Shift+T> or click the add  button.
- To open a new Plot tab for the currently active Console tab, press <Ctrl+Shift+T>. To close the Plot tab, press <Ctrl+Shift+W> or <Ctrl+Shift+F4>.

## Event Log Window

The **Event Log** window is closed by default. Information about what Thermo-Calc is processing or doing is in blue text, an error message is in red text, and caution/warning information is in yellow text.



### Global Settings

```

Event Log
14:11:08,366 INFO GoblerTask: Gobbled: 14:11:08,366 [pool-5-thread-1] INFO AbstractQuantity: condition T the value 1773.15
14:11:08,472 INFO GoblerTask: Gobbled: 14:11:08,472 [pool-5-thread-1] INFO GeneralEquilibriumCalculator: Calculating equilibrium
14:11:08,472 INFO GoblerTask: Gobbled: 14:11:08,472 [pool-5-thread-1] INFO AbstractQuantity: condition W(Mn) the value 0.5
14:11:08,473 INFO GoblerTask: Gobbled: 14:11:08,472 [pool-5-thread-1] INFO AbstractQuantity: condition T the value 1773.15
14:11:08,570 INFO ScheduledJob: Execution cancelled
14:11:08,571 INFO ScheduledJob: Cancelling activity: Equilibrium Calculator 1
14:11:08,570 INFO ScheduledJob: Cancelling activity: Plot Renderer 1
14:11:08,584 INFO GoblerTask: Gobbled: 14:11:08,583 [pool-5-thread-1] INFO GeneralEquilibriumCalculator: Calculating equilibrium
14:11:09,813 WARN ProcessExecutorAllocator: The subprocess completed with status code 2: the execution was cancelled by the user

```

- To open the Event Log window: Click the **Event Log** window once to expand it or select it from the **Window → Event Log** menu. Double-clicking the window maximizes it.
- To clear all messages in the window, right-click in the text area and select **Clear**.
- To specify the level of detail in the Event Log, open the **Options** window:
  - Windows: Select **Tools → ☰ Options**.
  - Mac: Select **Thermo-Calc <version> → Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **General** tab and set the **Log level** slide bar to anything between **Debug** (most detailed) and **Error** (least detailed).

## Keyboard Shortcuts to Navigate Between Tabs

Use keyboard shortcuts to move between the tabs available in the Console and Results windows.

### Windows

- Press CTRL + TAB to move forward sequentially through each open tab. When you are focused on the Console window, you move forward simultaneously through the Console Results tabs, e.g. If you are on Console 1 and press CTRL + TAB, you move forward to Console 2 and Results Console 2 and so forth.

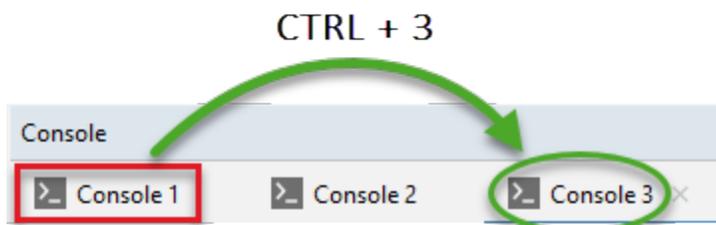


- Press CTRL + SHIFT + TAB to move back through the open tabs. This is useful if you have multiple tabs open, otherwise you can use the CTRL + TAB to cycle through the windows.
- Press CTRL + [tab number 1 to 9] to move directly to the numbered tab. For example, if you are focused on Console 1 and want to move to Console 3, press CTRL+3 and the focus jumps to that tab.



The tab number corresponds to the order of the tabs, not necessarily the name. That is, if Console 4 is first, then it is what is highlighted when CTRL + 1 is used.

For **EXAMPLE**, press CTRL + 3 on the keyboard to move focus from Console 1 to Console 3.



When you are focused on the **Console Results** window and use any of the shortcuts this only moves you through the open tabs on this window.

## ***Linux and macOS***

The same **Windows** keyboard shortcuts are used but must be pressed twice to move forward and backward.

---

# Modules and Commands

---

**In this section:**

---

Console Mode Workflow .....	14
Defining a System in Console Mode .....	16
Console Mode Modules .....	18
Moving Between Modules and Submodules .....	23
Entering Command Names or Abbreviations .....	24
Using the Command Line Prompt .....	26
Main Menu Commands .....	27
Wild Card Characters .....	28

---

## Console Mode Workflow

---

Working with Thermo-Calc involves moving between different modules. The workflow differs depending on the type of calculation:

- If you want to perform a calculation using the **POLY** module, you typically first retrieve thermodynamic data in the **DATA** module, perform the calculation in **POLY**, and visualize the results in the **POST** module. It is possible to define a system directly in **POLY** using the **DEFINE\_MATERIAL** and **DEFINE\_DIAGRAM** commands.
- If you want to calculate and plot a diagram using a response-driven module, such as **POTENTIAL** or **SCHEIL**, then you can go directly to that module. Response-driven modules prompt you to go through the steps to do the calculation and the required post-processing. These modules include **BIN**, **TERN**, **POTENTIAL**, **POURBAIX**, and **SCHEIL**. You typically end up in the **POST** module after having used a response-driven module. In the **POST** module you can modify the plots and save the diagram.
- If you want to tabulate a chemical substance, phase, or reaction, go directly to the **TAB** module.

The basic workflow is:

1. You are in the **SYS** module when you start Thermo-Calc in Console Mode.
2. In the **DATA** module define the system. Before performing a calculation, you must define the system and retrieve thermodynamic data. The data is retrieved from a database file (\*.TDB or \*.TDC).
3. The data needed to describe a system is then sent to the **GIBBS** module. This data is accessed by the **POLY** module when you instruct it to perform calculations.
4. In the **POLY** module, perform the calculations.
  - In **POLY**, the conditions are set for an equilibrium calculation (temperature, pressure, system composition, etc.), then an equilibrium calculation is done.
  - If you want to make a property or phase diagram, you set the conditions for a stepping or mapping operation and perform that operation.
  - This generates data that can be used to plot a property diagram (if a stepping operation is performed) or a phase diagram (if a mapping operation is performed).

5. In the **POST** module plot and visualize your data.
  - If a stepping or mapping operation is performed, then you can go to the POST module and plot a property or phase diagram.
  - The diagram can be plotted quickly using default settings, but you can also modify which variables to plot of the diagram axes and change the appearance of the diagram.
  - You can save the diagram either as a plain text file with data about all the coordinates or as an image file.

## Defining a System in Console Mode

---

*Defining a system* means to select the chemical components and to retrieve thermodynamic data about those components from an appropriate database in order to do the calculations. You use database(s) with thermodynamic data and then define what system elements as components. Once the system is defined, you retrieve the system's thermodynamic data from the database(s).

- If you use a *response-driven module* to perform your calculation, such as BIN or POURBAIX, then the module prompts you to select database(s) and define the system.
- When doing calculations in the POLY module, you typically have to manually select databases and define the system in the DATA module. However, it is also possible to define the system directly in POLY using the DEFINE\_MATERIAL and DEFINE\_DIAGRAM commands.



[Database \(TDB\) Module Commands](#)

### How to Define a System in Console Mode

1. Go to the DATA (TDB) module. At the `sys` prompt, use `GOTO_MODULE` and enter `Data`.
2. Change the default database unless you directly specify the name of the database as a parameter to the command. Type `SWITCH_DATABASE` and follow the prompts.

The second part of the command prompt indicates the current default database. For example, if the prompt is `TDB_TCFE10`, then the current database is TCFE10.

3. Use `DEFINE_ELEMENTS` followed by a list of the elements that you want in your system. (To list the elements that are available in your current database, use `LIST_DATABASE` and choose `Elements`). For example, if you want to have Fe and C in your system, type:

```
DEFINE_ELEMENTS Fe C
```

4. Use `REJECT` and choose `Phases` if you want to avoid retrieving any phases from the database. (To list the phases that are available in your current database, use `LIST_DATABASE` and choose `Phases`. To list the phases that can form in the defined system, use `LIST_SYSTEM` and choose `Constituent`.) For example, if you do not want the graphite phase to be retrieved, you type:

## REJECT PHASES GRAPHITE

If the number of phases to include is much lower than the total number of phases, then it can be convenient to first `Reject Phases *` and then restore the phases to include using `Restore Phases`.

5. Use `GET_DATA` to search the database and send the thermodynamic data about your system to the GIBBS workspace.

At this point you can proceed to the POLY module (with `GOTO_MODULE POLY`). However, you may want to add elements, phases and species to the GIBBS workspace from other databases. If so, then proceed to the next step.

6. Use `APPEND_DATABASE` to select the (additional) database from which you want to retrieve data. This command works exactly in the same way as the `SWITCH_DATABASE` command, with the exception that it does not reinitialize the DATA module and the GIBBS workspace, but instead appends or replaces the data that has already been retrieved with new data from the additional database.
7. Define your elements and specify whether to reject and restore any phases. Do this in exactly the same way when using the `SWITCH_DATABASE` command (see steps 2 and 3).
8. Use `GET_DATA` to search the database and add the thermodynamic data to the data that already exists in the GIBBS workspace.
9. Use `APPEND_DATABASE` again if you want to add data from yet another database. When you have retrieved all the data you need, you can proceed to the POLY module.

## Console Mode Modules

---



### [Graphical Mode Activities vs Console Mode Modules](#)

Below are each of the available modules you access in Console Mode. Most of the modules have commands, some are the same no matter what module you are in (e.g. general commands like BACK, GO TO MODULE, EXIT, HELP and so forth). Other commands have different prompts based on the module you are in how you define the system and so forth.



Most modules have several commands available. The exceptions are the BINARY, TERNARY, SCHEIL, POURBAIX and POTENTIAL modules which do not have any additional commands once you access the modules. These are response-driven modules that prompt you with a series of questions and then build the diagram automatically based on your responses.

### ***System Utilities (SYS)***

This is the module where you start. It also has several commands. Throughout the other modules there are some general commands, both of which are listed together.



### [System Utilities \(SYS\) Commands](#)

### ***Gibbs Energy System (GES)***

The Gibbs Energy System (GES) module is an important Thermo-Calc Console Mode module and consists of a comprehensive subroutine package for various thermochemical calculations. The module is interactively connected with all basic and advanced modules in Thermo-Calc and the add-on Diffusion Module (DICTRA). Most users will not directly use the GES module and commands unless you perform assessments.



The GES module is in essence the Thermo-Calc calculation engine, sometimes referred to as the *calculation core*. When working in Graphical Mode you are using the GES calculation engine behind the scenes, whereas in Console Mode, and if you are performing assessments, you are directly interacting with the module. This is also applicable if you are working with any of the software development kits (SDKs).



[About the Gibbs Energy System \(GES\) Module](#)



[Gibbs Energy System \(GES\) Commands](#)

### ***POLY3 (POLY) (Equilibrium)***

With the equilibrium calculation module, POLY3, it is possible to calculate many different kinds of equilibria and diagrams, in particular multicomponent phase diagrams.



[The Equilibrium Calculation Module \(POLY3\)](#)



[POLY Module Commands](#)

### ***Database (TDB)***

The Database module (full name is Database Retrieval) is where you select and retrieve various types of thermodynamic and kinetic information from the selected databases you are going to use for your system. Once you enter the module the short name is `TDB`.



[Database \(TDB\) Module Commands](#)



[Defining a System in Console Mode](#)



Also see the [Database Manager User Guide](#) and the available information for the specific databases.

### ***POST - A POLY3 Submodule***

The POST module (post-processor) is a submodule to the POLY module and has its own set of commands. The TAB, BIN, TERN, POT, SCHEIL and POURBAIX modules use the POST module features as part of the automatic generation of the plots.

---

 [Using the POST Module](#)

 [POST Module Commands](#)

## **Scheil (SCHEIL) Simulations**

The available Scheil-Gulliver solidification simulations are *classic* Scheil, Scheil with *back diffusion* in the primary phase, or Scheil with *solute trapping*. There are also configuration options available with *fast diffusers* for the classic and back diffusion models.



There is also connectivity to the Diffusion Module (DICTRA) and *homogenization* where you collect data from a Scheil solidification calculation, which is then used as part of the kinetic homogenization simulation of segregated composition profiles.



[Scheil in Graphical Mode vs Console Mode](#)



The Scheil with back diffusion feature is only available for systems with diffusion data, i.e. this model requires the use of a mobility database.

In Thermo-Calc, the Scheil calculation allows the calculation of:

- The solidification range of an alloy.
- Depression of the solidus temperature due to segregation.
- Composition of the last liquid to disappear in segregation pockets.
- Phases formed on final solidification in segregation pockets.
- The composition gradient in the primary solid phase(s) (segregation profile).
- Solute trapping to simulate deviation from local equilibrium for the primary phase
- Kinetic homogenization simulation of segregated composition profile (diffusion calculations)



[About the Scheil-Gulliver Solidification Simulations](#)



[SCHEIL Module Commands](#)

---

## ***Pourbaix Diagram (POURBAIX)***

The POURBAIX module lets you construct Pourbaix diagrams and other types of property diagrams for heterogeneous interaction systems with aqueous solution phases. Once you have calculated the Pourbaix diagram, you can also use the saved results of the calculation (the POLY3-file), and make a property diagram based on stepping calculation. You can also plot additional diagrams based on the results of the same calculation, but with different axis variables than Ph and Eh.

 [About Aqueous Solutions and the Pourbaix Diagram](#)

## ***Potential Diagram (POT)***

The POTENTIAL\_DIAGRAM module (short name, the POTENTIAL module) is a potential phase diagram calculation module. A potential diagram uses the activities of two major species in the gaseous mixture phase as mapping variables.

 [About Potential Diagrams](#)

## ***Binary Diagram (BIN)***

The BINARY\_DIAGRAM module (short name, the BIN module) enables you to quickly calculate a simple binary phase diagram. Binary phase diagrams are by far the most common found in the literature and calculated by computers. However, most real systems have more than two components and thus binary diagrams are more a kind of exercise in order to calculate more realistic phase diagrams.

 [About Binary Phase Diagrams](#)

## ***Ternary Diagram (TERN)***

The TERNARY\_DIAGRAM module (short name, the TERN module) enables you to quickly calculate a simple ternary phase diagram. There are many types of ternary phase diagrams but perhaps the most common is the isothermal section. In this case, the temperature is constant, which means that the tie-lines are in the plane of the calculation.

 [About Ternary Phase Diagrams](#)

---

## ***PARROT (Optimization)***

The PARROT module makes use of the GES module to handle the models of the various phases that might form in a multicomponent system, as well as of the POLY module to store and calculate complex heterogeneous equilibria.

-  [PARROT Module Commands](#)
-  Also see the [Data Optimization User Guide](#)

## ***Tabulation Reaction (TAB)***

The TABULATION\_REACTION (TAB) module can be used to tabulate thermodynamic functions of any type of substance, stoichiometric phase or solution phase, or reaction. The module can tabulate thermodynamic functions of pure stoichiometric phases or solution phases at fixed composition, as well as various kinds of partial derivatives for a pure stoichiometric phase or solution phase at a given composition.

-  [About the Tabulation Reaction Module](#)
-  [Tabulation Reaction \(TAB\) Commands](#)

## ***Reactor Simulator (REACTOR)***

Simulate chemical reaction processes in several feed-forward steady-state stages or in several dynamic stages.

-  [REACTOR \(REACTOR\\_SIMULATOR\) Commands](#)

## ***DICTRA Monitor (DICTRA)***

DICTRA Monitor (DICTRA) and DIC Parrot (optimization for DICTRA): These are available with the add-on Diffusion Module.

## Moving Between Modules and Submodules

---

To go to a specific module, you typically use `GOTO_MODULE` (or just `GOTO`) followed by the name of the module. For example, to go to the `DATA` module, type `GOTO DATA`.

The exceptions are the submodules `POST` and `ED-EXP`:

- To go to the `POST` module, you must enter the `POST` command from within the `POLY` module or `DICTRA` module (if you have the add-on license).
- To go to the Edit Experiments module you must enter the `ED-EXP` command from within the `PARROT` module.

To go back to the previous module type `Back`. For example, if you are in the `DATA` module and entered it from the `SYS` module, when you type `Back`, you return to the `SYS` prompt. This command is also available to exit from the submodules `POST` and `ED-EXP`.



Without a valid Thermo-Calc license, you cannot leave the `SYS` module. To enter the `DICTRA` and `DIC_PARROT` modules, you also need a valid Diffusion Module (`DICTRA`) license key.

## Entering Command Names or Abbreviations

The name of a command typically consists of terms linked with underscores, for example, LIST\_EQUILIBRIUM. Note the following:

- UPPER or lower case letters are OK.
- Hyphens (-) instead of underscores (\_) are OK, e.g. LIST-EQUILIBRIUM.
- You can abbreviate commands as long as it is unambiguous, e.g. when more than one command begins with LIST\_, you need to type the next letter or word to distinguish between the commands.
- Each word can be abbreviated e.g., L\_E for LIST\_EQUILIBRIUM.

### Examples

Examples of how to abbreviate commands at the command line prompt in Console Mode.

<i>Command Name</i>	<i>Abbreviation</i>
CALCULATE_EQUILIBRIUM	C-E
CALCULATE_ALL_EQUILIBRIA	C-A
LIST_EQUILIBRIUM	L-E
LIST_INITIAL_EQUILIBRIUM	LI-I-E
LOAD_INITIAL_EQUILIBRIUM	LO-I-E
LIST_PHASE_DATA CBCC	L-P-D CBC
LIST_PHASE_DATA CEMENTITE	L-P-D CEM
SET_ALL_START_VALUES	S-A-SS-AL
SET_AXIS_VARIABLE 1 X(FCC,FE) 0 0.89 0.025	S-A-V 1 X(F,FE) 0 .89 .025
SET_START_CONSTITUENT	S-S-C
SET_START_VALUE	S-S-V
SET_AXIS_PLOT_STATUS	S-A-P

---

<i>Command Name</i>	<i>Abbreviation</i>
SET_AXIS_TEXT_STATUS	S-A-T-SS-A-TE
SET_AXIS_TYPE	S-A-TY
SET_OPTIMIZING_CONDITION	S-O-C
SET_OPTIMIZING_VARIABLE	S-O-V
SET_OUTPUT_LEVEL	S-O-LS-OU

## Using the Command Line Prompt

---

The command line prompt in the Console window tells you which module you are currently in. When Thermo-Calc is first opened in Console Mode, you are in the SYS module, which is indicated with the command line prompt `SYS`.

To use Console Mode, you type in commands at the command line prompt. The available commands depend on which module you are in. You can list the commands by typing `?` at the prompt and then press <Enter>.



[Console Window and Tabs](#) also includes information about saving or copying the Command history.

- [Moving Between Modules and Submodules](#)
- [Entering Command Names or Abbreviations](#)
- [Specifying Parameters](#)
- [Wild Card Characters](#)
- [Keyboard Shortcuts to Navigate Between Tabs](#)
- [Working with the Console Mode Output](#)

## Main Menu Commands

---



In Console Mode you use commands to access these menu options. If there is an equivalent command, it is listed below.



The **File** menu and toolbar are reserved for Graphical Mode. The **Tools**, **Window** and **Help** menus are described in [Menu, Toolbar, and Keyboard Shortcuts](#).

### How to Open Files

- In the SYS module: [MACRO\\_FILE\\_OPEN](#)
- In the SYS module: [OPEN\\_FILE](#)



[Opening Console Mode Macro Files](#)



Drag and drop files into the Console window to open or run a macro, for example. File types that can be opened this way are macro files (\*.TCM and \*.DCM), database files (\*.TDB), and workspace files (\*.POLY3, \*.GES5, \*.PAR, and \*.DIC). Note that \*.DCM and \*.DIC file extensions are applicable to the Diffusion Module (DICTRA).

### Save (Workspace)

This command depends on the module you are in. Type ? at the prompt to look for a command such as [SAVE\\_GES\\_WORKSPACE](#) (GIBBS module).

### Exit

In any module: EXIT

### Close

In the SYS module: [CLOSE\\_FILE](#)

### Display License Info

In the SYS module: [DISPLAY\\_LICENSE\\_INFO](#)

## Wild Card Characters

You can sometimes use the asterisk (\*) as a wild card character to, for example, refer to all components, phases, or species, when showing calculated properties in the POLY module, or when you set axis variables for plotting diagrams in the POST module.

When showing calculated properties in the POLY module, you can also use the dollar sign (\$) to refer to all stable phases.



For activity and/or chemical potential properties, the \* and \$ normally do not work properly as wild card characters.

### Examples

These are examples of the wild card characters you can use to search commands at the command line prompt.

Command	Action
SHOW_VALUE B(*)	Lists mass (gram) of all components in the system.
SET_AXIS_VAR Y B(*) *	Sets mass (gram) of all components in the system as Y-axis variable.
SHOW_VALUE MUR(*)	Lists chemical potentials of all components in the system.
SET_AXIS_VAR Y ACR(*) *	Sets activities of all components in the system as Y-axis variable.
SHOW_VALUE HM(*) .T	Lists heat capacities of all phases.
SET_AXIS_VAR Y GM(*) *	Sets molar Gibbs free energies of all phases as Y-axis variable.
SHOW_VALUE TC(\$)	Lists curie temperature of all stable phases.
SHOW_VALUE W(*,*)	Lists mass fractions of all components in all phases.
SHOW_VALUE W(\$,*)	Lists mass fractions of all components in all stable phases.
SHOW_VALUE W(FCC,*)	Lists mass fractions of all components in the FCC phase.

---

<i>Command</i>	<i>Action</i>
SHOW_VALUE W(*,FE)	Lists mass fractions of the Fe component in all phases.
SHOW_VALUE W(\$,FE)	Lists mass fractions of the Fe component in all stable phases.
SHOW_VALUE Y(*,*)	Lists site fractions of all species in all sublattices of all phases.
SHOW_VALUE Y(\$,*)	Lists site fractions of all species in all sublattices of all stable phases.
SHOW_VALUE Y(*,*)	Lists site fractions of all species in all sublattices of all phases.

---

# File Types

---

**In this section:**

---

Working with the Console Mode File Types .....	31
Opening Console Mode Macro Files .....	32
Log Files .....	36
Macro Files .....	37
Workspace Files .....	40
Editing the Default Console Directory .....	43
Experimental Data Files (*.POP, *.DOP) .....	46

## Working with the Console Mode File Types



For DICTRA Console Mode see *File Formats* in the *Diffusion Module (DICTRA) User Guide*

In Console Mode, Thermo-Calc uses different types of files, including:

- [Macro Files](#) and [Log Files](#) both with the file extension \*.TCM. Log files are created using the SET\_LOG\_FILE command.
- [Workspace Files](#) in POLY (\*.POLY3), GIBBS module (\*.GES5), and the PARROT module (\*.PAR). (For the GIBBS module, this cannot currently be done with GES6).
- [Experimental Data Files \(\\*.POP, \\*.DOP\)](#) are a type of workspace file available with the PARROT module and have the file extension of \*.POP. In the Diffusion Module (DICTRA), you work with .DOP files instead of .POP. These .POP and .DOP files are referred to as experimental data files.



Another type of experimental data file with the \*.EXP file extension are also generated when working with DATAPLOT. See [About the DATAPLOT Graphical Language](#).



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.



Drag and drop files into the Console window to open or run a macro, for example. File types that can be opened this way are macro files (\*.TCM and \*.DCM), database files (\*.TDB), and workspace files (\*.POLY3, \*.GES5, \*.PAR, and \*.DIC). Note that \*.DCM and \*.DIC file extensions are applicable to the Diffusion Module (DICTRA).

## Opening Console Mode Macro Files



The macro files (with the extension \*.TCM) include comments, which you can either run in Thermo-Calc or open and read in a text editor.

You can open the macro files (\*.TCM or \*.DCM files) in different ways.

- From the main menu,
- Using a command,
- By dragging and dropping the file into the Console window, or
- Navigate to the file location on your computer and double-click to open it. If you already have Thermo-Calc open, the macro opens in a new tab. Otherwise a new instance of Thermo-Calc is launched.



If you are using the advanced pipe or redirect option ([Technical Note: Pipe or Redirect Output Data from Console Mode](#)) then a new instance of Thermo-Calc is always started.

- You can also read the commands listed in the macro file using a text editor.



If you only open and read the macro file in a text editor, you do not see the output that Thermo-Calc gives in response to the commands stored in the macro file.

## Opening a Macro from the Main Menu

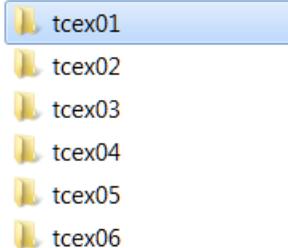
To open a Thermo-Calc example macro file:

1. Open Thermo-Calc.
2. Confirm you are in Console Mode. If not, click the **Switch to Console Mode** button.
3. From the main menu, select **File** → or **Help** → **Examples Files**. The Console Mode

folder opens.

4. Open the **Thermo-Calc** folder to see the available examples contained in separate folders.

Name



For the Diffusion Module (DICTRA) examples (\*.DCM macro files), open the **Diffusion Module - DICTRA** folder.

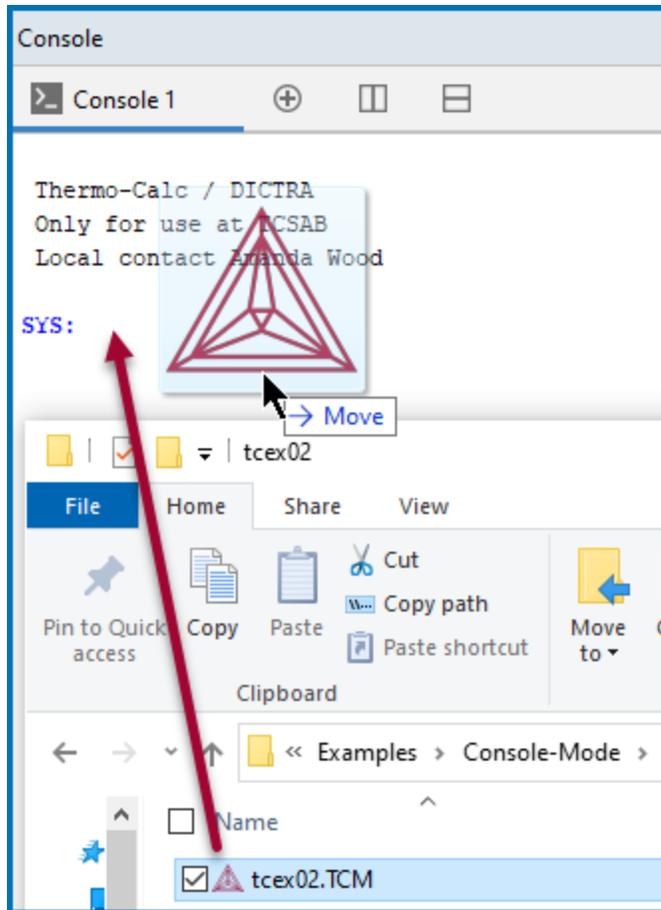
5. In the folder, double-click the \*.TCM file or click **Open**.
6. Follow the prompts in the Console window i.e. keep pressing <Enter> to run the macro and produce a plot in the **Console Results** window.

## Opening a Macro Using a Command

At the `sys` prompt, type `MACRO_FILE_OPEN` and press <Enter>. In the file dialogue window that opens, navigate to and select the macro file to open.

## Drag and Drop a Macro File into the Console Window

For example, on your computer, navigate to the examples folder. Drag the file from its location into the Thermo-Calc Console window. When you release (drop) the file, the macro starts running. Follow the prompts in the Console window i.e. keep pressing <Enter> to run the macro and produce a plot in the **Console Results** window.



```

Console
> Console 1
Thermo-Calc / DICTRA
Only for use at TCSAB
Local contact Amanda Wood

SYS:
SYS:MACROD "C:\Users\amanda\Documents\Thermo-Calc\2022b\Examples\Console-Mode\Thermo-Calc\tcex02\tcex02.TCM"
SYS: set-echo
SYS: @@
SYS: @@ Plotting thermodynamic functions
SYS:
SYS: @@ This example shows how to plot thermodynamic
SYS: @@ functions in unary, binary and ternary systems.
SYS: @@ It also involves working with partial derivatives
SYS: @@ and partial quantities.
SYS:
SYS: set-log ex02,,
SYS:
SYS: go da
... the command in full is GOTO_MODULE
THERMODYNAMIC DATABASE module

```



Drag and drop files into the Console window to open or run a macro, for example. File types that can be opened this way are macro files (\*.TCM and \*.DCM), database files (\*.TDB), and workspace files (\*.POLY3, \*.GES5, \*.PAR, and \*.DIC). Note that \*.DCM and \*.DIC file extensions are applicable to the Diffusion Module (DICTRA).

---

## Log Files

---

Log files with the file extension \*.TCM are plain text files used to save a sequence of commands. Log files can be edited in a text editor.

- To start saving your input into such a file, use SET\_LOG\_FILE in the SYS module, followed by the name of the file that you want to save your command sequence to.
- If you want to save the output in the log file as well, use SET\_ECHO before the SET\_LOG\_FILE command. Doing this is useful if you want use the log file later as a macro file.

## Macro Files

---

Macro files (with the file extension of either \*.TCM or \*.LOG) are plain text files used to save a sequence of commands that can be loaded and executed. Macro files can be edited in a text editor. When creating a command sequence, you add comments to the file by starting a line with @@.

Macro files include comments, which you can either run in Thermo-Calc or open and read in a text editor.



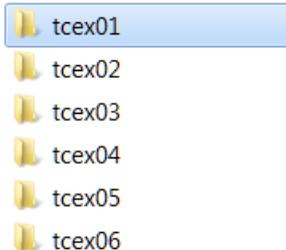
If you only open and read the macro file in a text editor, you do not see the output that Thermo-Calc gives in response to the commands stored in the macro file. It is recommended you run the macro to view all the details.

### ***How to Open a Console Mode Example Macro File***

To open a Thermo-Calc example macro file:

1. Open Thermo-Calc.
2. Confirm you are in Console Mode. If not, click the **Switch to Console Mode** button.
3. From the main menu, select **File** → or **Help** → **Examples Files**. The Console Mode folder opens.
4. Open the **Thermo-Calc** folder to see the available examples contained in separate folders.

Name



For the Diffusion Module (DICTRA) examples (\*.DCM macro files), open the **Diffusion Module - DICTRA** folder.

5. In the folder, double-click the \*.TCM file or click **Open**.
6. Follow the prompts in the Console window i.e. keep pressing <Enter> to run the macro and produce a plot in the **Console Results** window.

## Macro File Commands

Action	Command	Description and Comments
To load a macro file	In the SYS module, <a href="#">MACRO_FILE_OPEN</a> followed by the name of the macro file.	Thermo-Calc starts the command sequence that the file contains.
Regain control of the console	<a href="#">SET_INTERACTIVE</a>	End a macro file with this command to have control of the Console returned to you.
Add comments to a macro file	Start a line with @@, or enclose it between an @ (-line and a @)-line.	When lines are enclosed the software ignores them when running the macro.
Open a macro file	MACRO_FILE_OPEN	When creating the log file, you can make a macro file load up to five other macro files.
Nest macro files inside of each other	MACRO_FILE_OPEN SET_INTERACTIVE	If a nested macro file ends with SET_INTERACTIVE, then Thermo-Calc resumes with the higher-level macro file at the command immediately following the MACRO_FILE_OPEN command that loaded and executed macro file that has just been terminated. If a nested macro file doesn't end with SET_INTERACTIVE (but with an end-of-file character), then the console shuts down and the macro is stopped.

## Macro File Control Characters

The @? character allows you to make a macro interactive by allowing input. At the @? character, which is placed where a parameter value or argument is normally put, Thermo-Calc prompts to input the value of a parameter or argument. You can enter a string immediately following the @? character. This string is presented to the user when prompted to enter the parameter value or argument. The entered value is used by Thermo-Calc as input to the command in question. For example, you can request the user to specify the temperature range of a stepping calculation by entering the following in a macro file:

```
SET_AXIS_VARIABLE 1 T @?Low-temperature-limit: @?High-temperature-limit:
```

You can also use up to nine variables in your macro file and prompt the user to enter values that can be assigned to these variables. Use the @#n character when you want to prompt the user to provide a value to the variable, where the n is a digit between 1 and 9. You can then use this value by with the ##n character. For example, you can request the user to provide the first element of a system by entering the following in the macro file:

```
@#3First-element?
```

You can then use this variable with the entering the character ##3 later in the macro file. For example, you can write:

```
SET_AXIS_VARIABLE 1 x(##3) 0 1,,,
```

Finally, there is the @& pause character. Thermo-Calc pauses and waits for input from the user when this pause character is read from a macro file. Inserting pause characters is useful if you want to allow the user to monitor what is happening when Thermo-Calc is running the macro.

## ***Response-Driven Modules and Macros***

Macro files that are created while you use a *response-driven module* begin with the module-entering command (for example, `Goto_Module Scheil`) followed by a number of lines with responses to the module's questions and requests. The file terminates with the commands [POST](#) or [SET\\_INTERACTIVE](#) (this command gives you back control of the Console). An empty line cannot be edited with any input rather than the default answer (to a specific question). Comment lines, commands to open other macro files, pause characters or input-controlling characters cannot be inserted between these empty lines (otherwise the response-driven module cannot be executed properly).

## Workspace Files

---



For DICTRA Console Mode see *File Formats* in the *Diffusion Module (DICTRA) User Guide*

Workspaces files allow you to save all the data in your current workspace. A workspace contains all the data, specified settings and the result of any calculations performed. The saved data includes original and modified thermodynamic data, the last set of conditions and options, and the results of any simulations. You use the command `SAVE_WORKSPACES` to create this file type, which is available in the POLY, GIBBS and PARROT modules.

When in Thermo-Calc Console Mode these are the workspace file extensions:

- POLY module (\*.POLY3)
- GIBBS (GES) module (\*.GES5)
- PARROT module (\*.PAR and \*.POP)



For DICTRA in Console Mode, the PARROT module file extension is \*.DOP.

The workspace has all the settings specified and the result of any stepping or mapping operations performed after using `SAVE_WORKSPACES`. Consequently, the command opens a workspace in which all data is saved which is generated after the `SAVE_WORKSPACE` command is executed. The saved data includes original and modified thermodynamic data, the last set of conditions and options, and calculation results.

To load the data and calculation results of a workspace file, use the `READ_WORKSPACES` command available in the POLY module. The file can be used for calculation in POLY, visualization in POST, or data manipulation in GES. You are also given the option in the POURBAIX and SCHEIL modules to open a previously saved workspace file. This allows you to make new POURBAIX or SCHEIL calculations on the same chemical system as before but with different temperature, pressure, and composition conditions, or to plot new property diagrams based on the previous calculations.



A reminder that Linux and macOS are case sensitive; when applicable ensure you enter file extensions with capital letters.



Drag and drop files into the Console window to open or run a macro, for example. File types that can be opened this way are macro files (\*.TCM and \*.DCM), database files (\*.TDB), and workspace files (\*.POLY3, \*.GES5, \*.PAR, and \*.DIC). Note that \*.DCM and \*.DIC file extensions are applicable to the Diffusion Module (DICTRA).

## Links to Workspace Commands

### ***POLY module***

- [READ\\_WORKSPACES](#)
- [SAVE\\_WORKSPACES](#)

### ***Gibbs (GES) module***

- [READ\\_GES\\_WORKSPACE](#)
- [READ\\_WORKSPACES](#)
- [SAVE\\_GES\\_WORKSPACE](#)
- [SAVE\\_WORKSPACES](#)

### ***PARROT module***

- [READ\\_PARROT\\_WORKSPACES](#)
- [SAVE\\_PARROT\\_WORKSPACES](#)

### ***ED-EXP module (submodule to PARROT)***

- [READ\\_WORKSPACES](#)
- [SAVE\\_WORKSPACES](#)

### ***Reactor module***

- [READ\\_WORKSPACE](#)
- [SAVE\\_WORKSPACES](#)

---

## ***Important Information About Map and Step Commands and the Save\_Workspaces Command***



A STEP or MAP command automatically saves the workspace with the most recently specified name. Do not use the SAVE\_WORKSPACES command after a MAP or STEP command.

The results from the STEP or MAP commands are destroyed by the SAVE\_WORKSPACE command. You can append several results obtained by sequential STEP or MAP calculations without destroying the previous results, whilst SAVE\_WORKSPACE erases them all. Keeping this in mind is important and useful particularly for calculating various isothermal (or isoplethal) sections and plotting them on the same diagram in a single Thermo-Calc run.

To suspend some of the STEP or MAP results, use the AMEND\_STORED\_EQUILIBRIA command.

## Editing the Default Console Directory

---

You can add a startup parameter to a shortcut when launching Thermo-Calc on any of the platforms. When you launch the revised shortcut, the file path is set to a user-defined default directory. An example of why you might want to have a custom directory path is if you use batch projects or if you have macro files for batch jobs. The custom directory file path can also be viewed from within Console Mode.



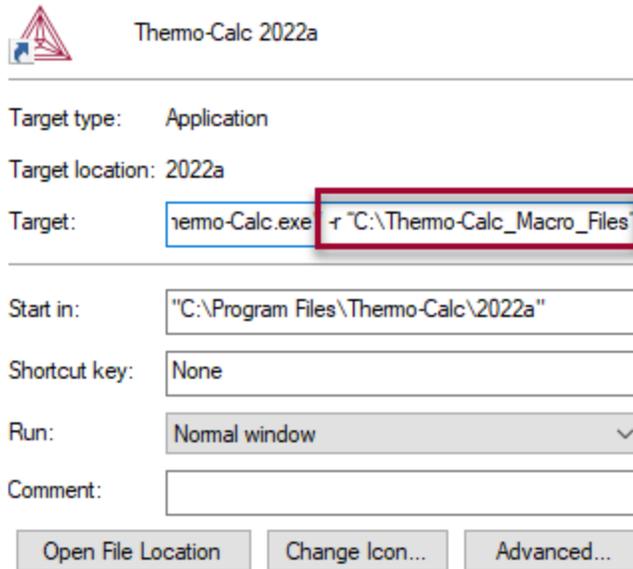
The shortcut is only valid throughout the existing session; it is not saved as a default. Each time you want to use this file path you must launch Thermo-Calc from the customized shortcut.

The following uses Windows for the example.

### ***How to Set up the Default Console Directory***

1. Right-click a shortcut to Thermo-Calc and click **Properties**.
2. In the **Thermo-Calc 2024a Properties** dialog, **Target** field, add the following to the end of the string: `-r` plus the *“path to the default directory”*. Include a space between the `-r` and path, and use quotation marks around the full file path. A simple example:

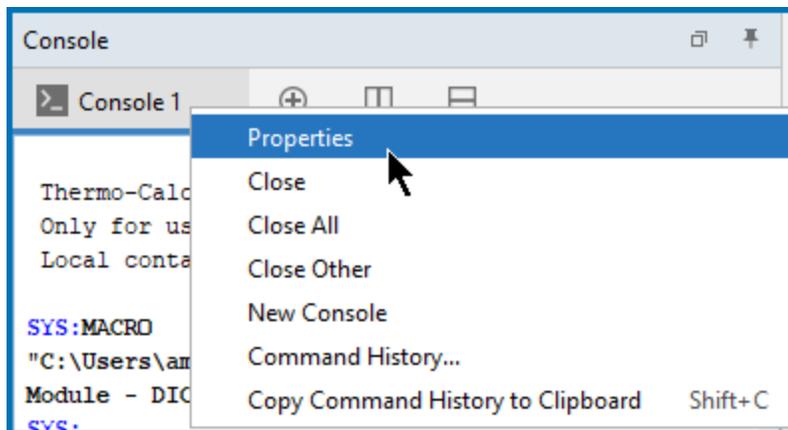
`-r "C:\Thermo-Calc_Macro_Files"`



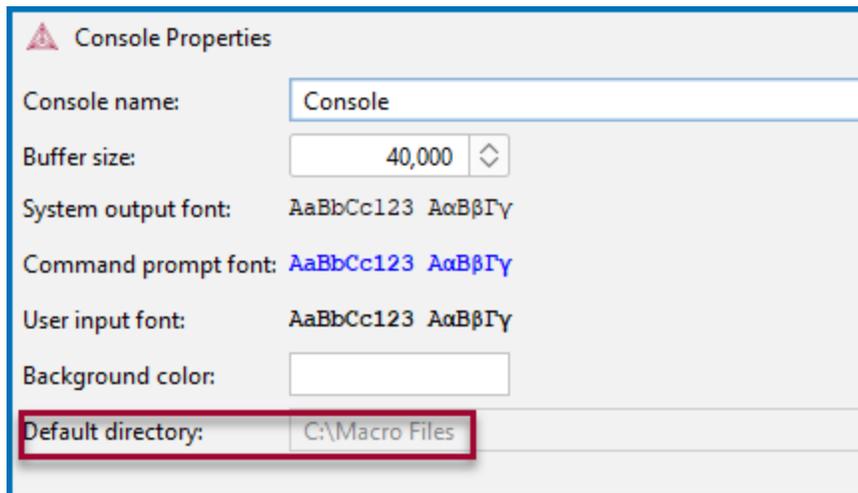
3. Click **Apply** and **OK**.
4. Launch Thermo-Calc from the shortcut where you added the parameter. You can confirm that it is set correctly from within Thermo-Calc.

### ***How to View the File Path in Thermo-Calc***

1. Launch Thermo-Calc from the short cut where you added the parameter.
2. In Console Mode, right click the **Console** window and choose **Properties**.



3. In the **Console Properties** window the **Default directory** section displays the custom file path. This default is set in the **Options** window, Console Mode section.



To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  on the keyboard.

## Experimental Data Files (\*.POP, \*.DOP)

---



There is also another type of experimental data file (\*.EXP) generated when working with DATAPLOT. See [About the DATAPLOT Graphical Language](#).



If you are using Graphical Mode, the [Experimental File Reader](#) activity node is used to plot experimental data on one or more plots.



See the [Tips and Tricks: How to Plot Experimental Data](#) blog post on our website for useful guidance about this topic.

In order to conduct an assessment for a system, the experimental data is described with a syntax, which is similar to the way that one calculates an equilibrium in the POLY module or you edit an experimental point in the ED\_EXP module. Similar to a Thermo-Calc macro file (\*.TCM), an experimental data file (i.e. the \*.POP or \*.DOP file) is a basic text file and can be opened and edited by any text editor. For this reason, a \*.POP or \*.DOP file is not hardware dependent.

A POP or DOP file consists of various commands from the POLY and ED\_EXP modules, as well some special commands which can only be used in such experimental data files. Various experimental information can be inputted in an \*.POP or \*.DOP file as different tables; the same type of experimental data are usually documented in the same table (see below).

A \*.POP or \*.DOP file is used in the PARROT module to provide experimental information for the optimization process, and is checked by a *syntax checker* when the PARROT command [COMPILE\\_EXPERIMENTS](#) is proceeded.

The special commenting identifier, i.e. a dollar sign \$ which starts a line, is used to document comment lines at any position in a \*.POP or \*.DOP file. You may remember that this feature is universally the same as in almost all kinds of text files in Thermo-Calc software, e.g. in \*.TDB, \*.DAT, \*.TCM, \*.POP/\*.DOP and \*.EXP files. Such a sign and all information afterwards in the same comment line is skipped and ignored by the syntax checker.

Many POLY and ED\_EXP commands can be directly used in a POP or DOP file. But there are some differences with the set of POLY or ED\_EXP commands, and some specially designed commands as described in this topic can only be used in the \*.POP or \*.DOP files.

---

## Frequently Used POLY Commands

Most of the commands in the experimental data file are the same as in the POLY module. The most frequently used are listed below:



The last command in an \*.POP or \*.DOP file must always be the SAVE command.

- [CREATE\\_NEW\\_EQUILIBRIUM](#)
- [CHANGE\\_STATUS](#)
- [SET\\_CONDITION](#)
- [SET\\_REFERENCE\\_STATE](#)
- [ENTER\\_SYMBOL](#)
- [SAVE\\_WORKSPACES](#)

## Less Often Used POLY Commands

Other legal commands from the POLY module that are used less often are:



The DEFINE-COMPONENTS command must be always used as the first command in an \*.POP or \*.DOP file, as it automatically reinitiates the whole workspace.

- [DEFINE\\_COMPONENTS](#)
- [EVALUATE\\_FUNCTIONS](#)
- [SET\\_ALL\\_START\\_VALUES](#)
- [SET\\_NUMERICAL\\_LIMITS](#)
- [SET\\_START\\_VALUE](#)
- [ADVANCED\\_OPTIONS](#)

---

## ***Frequently Used Edit Experiments (ED\_EXP) Commands***

Most of the special commands for the ED\_EXP module are also often used in an \*.POP or \*.DOP file. For example:

- [EXPERIMENT](#)
- [EXPORT](#)
- [IMPORT](#)
- [LABEL\\_DATA](#)
- [SET\\_ALTERNATE\\_CONDITION](#)
- [SET\\_WEIGHT](#)

## ***Do not use these Edit Experiment (ED\_EXP) Commands***



The following special ED\_EXP commands are illegal and should not be used in any \*.POP or \*.DOP file:

- COMPUTE-ALL-EQUILIBRIA
- MAKE-POP-FILE
- READ (READ-BLOCK OR READ-WORKSPACE)
- TRANSFER-START-VALUES

---

# Calculations

---



## [Graphical Mode vs Console Mode: Calculation Types](#)

### In this section:

---

About the Gibbs Energy System (GES) Module .....	50
Equilibrium Calculations .....	52
Property Diagrams .....	60
Phase Diagrams .....	66
T0 Temperature Simulations .....	76
Paraequilibrium .....	80
Potential Diagrams .....	85
Scheil Simulations .....	91
Aqueous Solutions and Pourbaix Diagrams .....	105
Tabulation Reactions .....	120

## About the Gibbs Energy System (GES) Module

The Gibbs Energy System (GES) module is an important Thermo-Calc Console Mode module and consists of a comprehensive subroutine package for various thermochemical calculations. The module is interactively connected with all basic and advanced modules in Thermo-Calc and the add-on Diffusion Module (DICTRA). Most users will not directly use the GES module and commands unless you perform assessments.

The GES module is in essence the Thermo-Calc calculation engine, sometimes referred to as the *calculation core*. When working in Graphical Mode you are using the GES calculation engine behind the scenes, whereas in Console Mode, and if you are performing assessments, you are directly interacting with the module. This is also applicable if you are working with any of the software development kits (SDKs).

The objective of the GES module is to provide a unified set of subroutines to be used in any application program that needs thermochemical data. All kinds of thermodynamic models for various types of substances are implemented. However, most model-dependent features are hidden inside the module, and the application programmer can use a standardized set of subroutines, i.e. using the Thermo-Calc SDKs such as TC-Python, TQ-Interface, TC-API and TC-Toolbox for MATLAB® to calculate the integral Gibbs energy or any partial derivative thereof for each phase at any composition, temperature and pressure.

There are subroutines to analytically calculate the first and second partial derivatives of integral Gibbs energy with respect to any variable. You can enter and modify phase descriptions, model connections, basic thermodynamic parameters, and so on.

POLY and all other basic and special modules, as well as the add-on Diffusion Module (DICTRA), interactively call the subroutines within the GES module to calculate any thermodynamic quantities.

### Calculation Engine: GES5 vs GES6



GES6 is a re-write and new implementation of the Gibbs Energy System module of Thermo-Calc and it is the default engine. The previous version, GES5, is still available and will continue to work in the background of the program where necessary. The main purpose of GES6 is to support faster development of new features. However, not all GES5 functionality is implemented in GES6. In such cases (except for the POURBAIX module), GES6 falls back silently and uses functionality from GES5 in the background.



To globally set the default from GES6 back to GES5, go to the **Tools → Options → Global General Settings** tab and click **Version 5** for the *Preferred Gibbs Energy System* setting. This is useful if you are using the POURBAIX module (aqueous solutions), or if you have a custom database that is not compatible with GES6. The `SET_GES_VERSION` Console Mode command can also be used to change GES versions for a single Thermo-Calc session.



Contact Thermo-Calc Software AB at [info@thermocalc.com](mailto:info@thermocalc.com) for more information about the changes or if you encounter any issues you think are related to the new version of the calculation engine.

---

## Equilibrium Calculations

---

### In this section:

---

The Equilibrium Calculation Module (POLY3) .....	53
Setting Conditions for Equilibrium Calculations .....	54
Calculating an Equilibrium .....	57
Calculating an Equilibrium with a Fixed Phase .....	58
Calculating an Equilibrium with Suspended or Dormant Phases .....	59

---

## The Equilibrium Calculation Module (POLY3)

An equilibrium describes what the composition of the end state of a system is, given a full specification of state variables such as temperature, pressure, initial composition, system size, etc. An equilibrium calculation is normally done in POLY-3 according to the *Global Minimization Technique*, which ensures that the most stable minimum under the specified conditions is computed.

For an equilibrium calculation to be performed, the state variables must all be set as conditions for the calculations. Such conditions include, for example, temperature, pressure, and system composition. When you calculate an equilibrium in the POLY module, you have to set these conditions manually.

Knowledge of the thermodynamic equilibrium is an important factor for understanding properties of materials and processes. With a database of thermodynamic model parameters, it is possible to predict such properties and also to obtain driving forces for diffusion-controlled phase transformations and other dynamic processes.

With the equilibrium calculation module, POLY3, it is possible to calculate many different kinds of equilibria and diagrams, in particular multicomponent phase diagrams.

Different types of databases can be used with the POLY module, and it can be used for alloys or ceramic systems, as well as gaseous equilibria, aqueous solution involved heterogeneous interaction systems. Up to 40 elements and 1000 species can be defined into a single system for equilibrium calculations.

All normal thermodynamic state variables can be used to set as conditions in calculating equilibria, and as axes in plotting diagrams. You can set the composition or any property of an individual phase as a condition. Any state variable can be varied along an axis in order to generate a diagram. During calculations of a diagram, complete descriptions of all calculated equilibria are stored, and in the diagram any state variable can be used as an axis.

The *Global Minimization Technique* is used to ensure that the present minimum in an equilibrium calculation is the most stable minima for the specified conditions. This is based on the traditional GEM (Gibbs Energy Minimization) technique.

A direct global minimization can be performed on conditions: N, n(comp), B, b(comp), w(comp), x(comp), T, and P, but not when combined conditions as e.g.  $w(a)^3 \cdot w(b) = 1$  are used or when an activity or potential condition is used. For all other types of conditions where regular minimization converges, Indirect Global Minimization, i.e. global test and corrections, if necessary, are performed until the lowest minimum is found.

- Direct Global Minimization: From the mesh of Gibbs energy, find the set of grid points that gives the lowest energy solution under the specified conditions. This set of grid points provides starting combination of phases and their constitutions for

regular minimization to find the exact equilibrium solution. This solution is then subject to a global test as described below.

- **Indirect Global Minimization:** Under certain conditions, a direct approach is impossible. In this case, regular minimization is performed first and then a check is performed in order to see if the found local minimum is a global one by checking if all grid points are above the equilibrium Gibbs energy plane. If not, then recalculate by including these grid points until no grid point is above the equilibrium Gibbs energy plane from the previous step.

The full-scale and full-scope usage of the *Global Minimization Technique* has been extended from for only single-point calculations for all types of calculations (of single-points, property diagram stepping and phase diagram mapping) in Thermo-Calc.

The use of *Global Minimization Technique* may increase the computation time. The main cost in time comes from the calculation of Gibbs energy at each grid point generated by properly meshing the composition space for each entered phase. An additional (but much smaller) cost in time comes from finding the set of grid points in the above mesh that give the lowest energy solution. This solution is where POLY starts its ordinary minimization. When POLY has found an equilibrium, the equilibrium Gibbs energy surface is compared to the mesh to assure that no grid point is below the surface, i.e. a global minimization has been reached.

Global minimization is performed by default in single-point or stepping or mapping equilibrium calculations.

Together with the PARROT module, the POLY module is also used for critical assessment of experimental data in order to develop thermodynamic databases. The POLY module uses the Gibbs Energy System (GES) for modeling and data manipulations of the thermodynamic properties of each phase.

## Setting Conditions for Equilibrium Calculations

Setting a condition normally involves giving a single state variable a specific value. For example, you can set the temperature to 1273.5 Kelvin ( $T=1273.5$ ). Alternatively, setting a condition can involve giving a linear expression with more than one state variable a specific value. For example, you can set the mole fraction of the S component to be the same in the liquid and the pyrrohotite phases ( $X_{(LIQ, S)} - X_{(PYRR, S)} = 0$ ).

```

POLY_3:1-e
OUTPUT TO SCREEN OR FILE /SCREEN/:
Options /VWCS/:
Output from POLY-3, equilibrium =      1, label A0 , database: TCFE7

Conditions:
T=1000, P=1E5, X(C)=5E-2, N=1
DEGREES OF FREEDOM 0

Temperature 1000.00 K ( 726.85 C), Pressure 1.000000E+05
Number of moles of components 1.00000E+00, Mass in grams 5.36552E+01
Total Gibbs energy -4.07975E+04, Enthalpy 2.41176E+04, Volume 7.21740E-06

Component      Moles      W-Fraction Activity Potential Ref.stat
C                5.0000E-02  1.1193E-02  2.1816E-01 -1.2659E+04 SER
FE               9.5000E-01  9.8881E-01  6.1895E-03 -4.2279E+04 SER

BCC_A2                Status ENTERED   Driving force 0.0000E+00
Moles 9.5069E-01, Mass 5.3063E+01, Volume fraction 9.6312E-01 Mass fractions:
FE 9.99844E-01 C 1.55514E-04

GRAPHITE                Status ENTERED   Driving force 0.0000E+00
Moles 4.9313E-02, Mass 5.9230E-01, Volume fraction 3.6881E-02 Mass fractions:
C 1.00000E+00 FE 0.00000E+00

```

The number of state variables that need to set is determined by the *Gibbs Phase Rule*. Typically, the state variables to give values to are:

- temperature (in K)
- pressure (in Pascal)
- system size in number of moles (in mole) or mass (in grams)
- the fraction of each component (in number of moles or mass)

If you fix the phase of the equilibrium and all but one of the state variables, you can discover the value of the other state variable at equilibrium.

It is possible to specify a set of conditions that does not have any equilibrium and the program detects this by failing to reach equilibrium during the calculation.

## State Variables

The following table lists some of the available state variables you can use to set conditions:

State Variable	SET_CONDITION Parameter
temperature in the system (in K)	T

State Variable	SET_CONDITION Parameter
pressure in the system (in Pascal)	P
system size (mole number in moles)	N
system size (mass in grams)	B
number of moles of a component in the system	N(<component>)
mole fraction of a component in the system	X(<component>)
mass fraction of a component in the system	W(<component>)
activity of a component in the system	ACR(<component>)
chemical potential of a component in the system	MUR(<component>)
mole fraction of a component in a phase	X(<phase>,<component>)
mass fraction of a component in a phase	W(<phase>,<component>)
activity of a species referred to a phase at ambient temperature and pressure	ACR(<species>,<phase>)
chemical potential of a species referred to a phase at ambient temperature and pressure	MUR(<species>,<phase>)
enthalpy in the system (in J)	H
enthalpy of a phase (in J/mol)	HM(<phase>)

## Calculating an Equilibrium

This topic explains how to calculate an equilibrium in POLY. To calculate an equilibrium means to calculate the equilibrium composition of your system, given a full specification of conditions that reduce the degrees of freedom of the calculation to zero.



If you are running a network client installation of Thermo-Calc and cannot enter the POLY module, this may be because all the license instances on that the license server makes available have been checked out.



### [POLY Module Commands](#)

1. Define your system.



#### [Defining a System in Console Mode](#)

2. Go to the POLY module. At the SYS prompt, type `GOTO_MODULE` and enter `POLY`.
3. Use `SET_CONDITION` followed by conditions and value assignments to set the conditions of your calculation. For example, to set temperature, pressure, initial composition and system size for a Fe-Cr-C system, you might enter the following:

```
SET_CONDITION T=1200 P=1E5 W(CR)=0.18 W(C)=0.0013 N=1
```

This sets the temperature (T) to 1200 K, the pressure to 1 bar (100,000 Pascal), the mass fraction of Cr to 18 mole percent, the mass fraction of C to 0.13 weight percent, and the total amount of material to 1 mole. The fraction of Fe in the system is calculated from the fractions of Cr and C. You have to set the fraction of all the components in your system except one.

4. Use `COMPUTE_EQUILIBRIUM` to run the calculation.
5. Use `LIST_EQUILIBRIUM` to see the results of the calculation.

---

## Calculating an Equilibrium with a Fixed Phase

There are many ways to calculate equilibriums in Thermo-Calc. The following is one example.

You can calculate an equilibrium that has a certain amount of a certain stable phase. Use [CHANGE\\_STATUS](#) and choose `Phase` to specify the phase and the amount of that phase (in normalized mole number) that you want to set as fixed. For example, if you want to find out at what temperature a system starts to melt, enter the following:

```
CHANGE_STATUS PHASE LIQUID=FIX 0
```

You must leave the state variable whose equilibrium value you are interested in unspecified. However, if you have already specified that state variable, you can make it unspecified again by using [SET\\_CONDITION](#) and set that state variable to `NONE`. For example, if you have given temperature a value, you can type:

```
SET_CONDITION T=NONE
```

The calculated equilibrium includes the value of the unspecified variable at which the equilibrium enters the phase that is fixed.



For an example where an equilibrium is calculated with a fixed phase, see Console Mode example `tcex07`.

---

## Calculating an Equilibrium with Suspended or Dormant Phases

You can calculate an equilibrium under the assumption that one or several phases are `SUSPENDED` or `DORMANT` using `CHANGE_STATUS` and choose `Phase`. For example, to specify that all phases except one should be suspended, you can first suspend all phases and then enter a single phase in the following way:

```
CHANGE_STATUS PHASE *=SUSPENDED
CHANGE_STATUS PHASE FE_LIQUID=ENTERED
```



For an example where this is done, as well as where the status of phases is set to be dormant, see the [Single Equilibrium Calculation Examples](#).

---

## Property Diagrams

---

### In this section:

---

About Property Diagrams .....	61
Calculating and Plotting a Property Diagram .....	62
Calculating a Property Diagram One Phase at a Time .....	64
Calculating Several Properties in the Same Diagram .....	65

---

## About Property Diagrams

When you calculate and plot a property diagram, there is only one independent state variable. Many different properties can be plotted as a function of this independent variable. For example, if the independent state variable is temperature, then the mole fractions of all phases can be plotted as a function of temperature. Or the composition of a specific phase may be plotted relative to temperature, or the activity of a component in the system as a whole or in a specific phase may be plotted relative to it.

A property diagram is plotted based on a series of equilibria that is computed while the value of one state variable is varied between a minimum and a maximum value. This variable is referred to as the stepping axis variable. In Thermo-Calc, first calculate one initial equilibrium, and then new equilibria are calculated at incremental steps in both directions on the stepping axis from the initial equilibrium. This continues until the stepping operation has covered the length of the axis between a minimum and a maximum value that is specified.



For an example of the calculation of a property diagram, see [tcex08](#).

## Calculating and Plotting a Property Diagram

This topic explains how to calculate a property diagram in POLY. To calculate a property diagram means to calculate a series of equilibria while the value of the stepping axis variable varies between a minimum and a maximum value. With the exception of the stepping axis variable, all the state variables that you set when you calculate the initial equilibrium retain the values when the new equilibria are calculated.

### [POLY Module Commands](#)

1. Calculate an initial equilibrium in the POLY module.
2. Use `SET_AXIS_VARIABLE` to set the axis variable, the minimum and the maximum stepping variable values and the step length. The first parameter of `SET_AXIS_VARIABLE` is the axis number. Since a property diagram only has one axis variable, this is set to 1. For example, if you want to set the axis variable to temperature, and calculate an equilibrium at every 50 K between a minimum temperature of 100 K and a maximum temperature of 2000 K, enter

```
SET_AXIS_VARIABLE 1 T 100 2000 50
```



An axis variable must be a state variable that is set when you calculated the initial equilibrium. For example, if you set the fraction of a component in number of moles, then you cannot set the mass fraction of this component as an axis variable. Also, the minimum stepping variable value must be smaller than, and the maximum value larger than, the value that you set the state variable to when calculating the initial equilibrium.

3. Use `STEP_WITH_OPTIONS` and choose `Normal` to perform the stepping operation.

The phase regions are listed along the stepping axis, the phases contained in each region, and the number of equilibria calculated in that region. Each region is defined by the phases it contains.



The number of equilibria calculated is not just the number of steps that are performed according to the step length and the minimum and maximum values on the axis variable. This is because Thermo-Calc calculates extra equilibria when a new phase is discovered in order to determine the phase region boundary more precisely.

```
POLY_3:s-a-v 1 T 1000 2000 100
POLY_3:step
Option? /NORMAL/:
  No initial equilibrium, using default
  Step will start from axis value    1000.00
...OK

Phase Region from    1000.00    for:
  BCC_A2
  GRAPHITE
Global check of adding phase at  1.01118E+03
Calculated    4 equilibria

Phase Region from    1011.18    for:
  BCC_A2
  FCC_A1
  GRAPHITE
Calculated    2 equilibria

Phase Region from    1011.18    for:
  FCC_A1
  GRAPHITE
Global test at  1.09000E+03 .... OK
Global check of removing phase at  1.15727E+03
Calculated    17 equilibria
```

- To plot the diagram, open the POST module and then use PLOT\_DIAGRAM. By default, the X-axis represents the stepping axis variable and Y-axis represents the sum of the mole fractions of all phases (NP(\*)).

---

## Calculating a Property Diagram One Phase at a Time

There are many ways to calculate property diagrams in Thermo-Calc. One example is where you can calculate a property diagram with a separate stepping operation being performed for each phase at a time in its default most stable composition (the major constitution). This is useful if you want to create a property diagram for a heterogeneous system with both ordered phases and their disordered pairs.

To calculate a property diagram one phase at a time, use [STEP\\_WITH\\_OPTIONS](#) and when prompted choose `One_phase_at_time` to perform the stepping operation.

## Calculating Several Properties in the Same Diagram

There are many ways to calculate property diagrams in Thermo-Calc. The following is another example. If you perform several stepping calculations after each other, the results of these calculations are all saved in your workspace file. This allows you to do the following:

- Calculate (and then plot) missing parts of a specific property inside the first property diagram. These parts are calculated inside the range of the stepping variable with a different control condition.
- Calculate (and then plot) two or more sets of a specific property on the same property diagram for the same system. These can be calculated under different control conditions with stepping operations being performed across the same stepping axis variable range.
- Calculate phase boundary lines and then plot them in a corresponding phase diagram for the same system. This can be especially useful for some defined secondary phase-transformations. For example, if you want to find the phase boundary between BCC\_A1 and BCC\_B2, or the equal-Gm for two specific phases, or the equal-fraction or equal-activity for two specific phases of a certain species, then this can be useful.

Unless you have opened a new workspace file previously using `SAVE_WORKSPACES`, the results of the stepping calculations are saved in a `RESULT.POLY3` file. To save the results of several stepping calculations in a file different from the workspace file that your results are currently saved to, then use `SAVE_WORKSPACES` before you perform the first stepping calculation.



`SAVE_WORKSPACES` overwrites and deletes the results of all previous stepping calculations.



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.

---

# Phase Diagrams

---

**In this section:**

---

About Phase Diagrams .....	67
About Binary Phase Diagrams .....	69
About Ternary Phase Diagrams .....	70
About Quasi-Binary and Quasi-Ternary Phase Diagrams .....	72

## About Phase Diagrams

Phase diagrams have two or more independent axis variables. Any state variable that has already been set can be used as the mapping variable for a mapping calculation and then as the axis variable for a phase diagram. From a mapping calculation, many types of phase diagrams can be plotted, with one of the mapped variables as one axis variable, and with other mapped variables or any varied property (state or derived variables) or entered symbol (variables, functions or table values) as the other axis variables.

All phase diagrams consist of zero phase fraction lines. There are two distinct types of phase diagrams: those with the tie-lines in the plane of the diagram and those where the tie-lines are not in the plane. The former includes binary phase diagrams and ternary isotherms. The latter includes more general isopleth diagrams with one or more fixed extensive variables (normally, this is a composition).

- The BIN and TERN modules calculate binary and ternary phase diagrams.
- The POTENTIAL and POURABIX modules calculate the more general isopleth diagrams and other related diagrams.

The following topics describe how to calculate the types of phase diagrams in the POLY\_3 module. In POLY, you can calculate phase diagram for systems with up to 40 components and with thousands of phases. You can combine activity conditions and fixed phase status and fraction conditions in any way. Thermo-Calc can calculate any arbitrary 2D section through composition space.



There is no guarantee that the conditions set will result in a calculation that reaches an equilibrium.



[Binary Phase Diagram Examples](#)



[Ternary Phase Diagram Examples](#)



[Potential Diagram Example](#)



[Pourbaix Diagram Examples](#)

## Calculating and Plotting a Phase Diagram

This topic describes how to calculate a phase diagram in the POLY module. The commands are described in [POLY Module Commands](#) and [POST Module Commands](#).

1. Calculate an initial equilibrium in the POLY module.
2. Use SET\_AXIS\_VARIABLE to set the first axis variable, its minimum and maximum mapping value and the length of each incremental step on this axis. The first parameter of SET\_AXIS\_VARIABLE is the axis number and is set to 1. For example, suppose you want the mass fraction of C as your first axis variable. You want it to vary between 0 and 0.1, with the length of each incremental step being no more than 0.002. Then enter:

```
SET_AXIS_VARIABLE 1 W(C) 0 .1 0.002
```



An axis variable must be a state variable that is set when you calculated the initial equilibrium. For example, if you set the fraction of a component in number of moles, then you cannot set the mass fraction of this component as an axis variable.

3. Use SET\_AXIS\_VARIABLE to set the second axis variable and specify its minimum, maximum, and step length values. The axis number is set to 2. For example, suppose you want temperature as your second axis variable and you want it to vary between 900 K and 1900 K, with an incremental step of 25 K. Then enter:

```
SET_AXIS_VARIABLE 2 T 900 1900 25
```

4. If you want more than two axis variables, then use SET\_AXIS\_VARIABLE until you have set all the axis variables. You can set up to five axis variables. Axis variables 3, 4 and 5 must be set temperature, pressure or to the chemical potentials of components.

You may want to save your workspace with SAVE\_WORKSPACES before you perform the mapping calculation. This saves the axis variables you have set. However, it overwrites the results of any previous stepping or mapping calculations done.

5. Use MAP to perform the mapping calculation. The console lists the phase region boundaries and the phases contained on one side of each boundary.

```

Phase region boundary 30 at: 1.363E+03 2.012E-01
  FCC_A1
  ** GRAPHITE
Calculated. 3 equilibria
Terminating at known equilibrium

Phase region boundary 31 at: 1.990E+03 2.526E-01
  LIQUID
  ** GRAPHITE
Calculated. 13 equilibria
Terminating at known equilibrium

```

- To plot the diagram, use POST to open the POST module and then use PLOT\_DIAGRAM. By default, the X-axis represents mapping axis variable 1 and the Y-axis represents mapping axis variable 2.

## About Binary Phase Diagrams

Binary phase diagrams are by far the most common found in the literature and calculated by computers. However, most real systems have more than two components and thus binary diagrams are more a kind of exercise in order to calculate more realistic phase diagrams.

You can calculate many types of binary phase diagrams with POLY, but the traditional one has one composition axis and the temperature on the other axis. This is calculated by first specifying a single equilibrium by setting conditions such as

```
SET_CONDITION T=1200, P=1E5, W(C)=.02, N=1
```

This means that the temperature should be 1200 K, pressure 1 bar, mass fraction of C is 0.02 (i.e., 2 weight percent) and the system contains one mole of atoms. Note that one does not set a condition on the amount of the second component. This is simply taken to be the rest of the system.

After setting these conditions one can calculate the equilibrium by giving the command:

```
CALCULATE_EQUILIBRIUM
```

When the calculation has converged, one continues by setting the axes for the diagram calculation. In most cases, one is interested in the complete composition range but the temperature range may vary.

```
SET_AXIS_VARIABLE 1 W(C) 0 .1 0.002
SET_AXIS_VARIABLE 2 T 900 1900 25
```

This means that the axis number 1 is taken to be the mass fraction of C which may vary between 0 and 0.1 with a maximum increment of 0.002. The axis number 2 is the temperature varied between a minimum 900 and maximum 1900 K with a maximum increment of 25.

Before starting the mapping, it is a good rule to SAVE the workspaces. If this is done just before the MAP command, and the MAP command fails for some reason, the user can restart from the situation just before mapping and try to find a better start point.

```
SAVE
MAP
```

The diagram is plotted in the post-processor. The post-processor will automatically set the same axes for plotting as used for mapping.

```
POST
PLOT
```

## BINARY\_DIAGRAM

The BINARY\_DIAGRAM module (short name, the BIN module) enables you to quickly calculate a simple binary phase diagram. Access to specific databases designed for BIN, such as TCBIN, is required.

To enter the module, at the SYS prompt type `GOTO_MODULE BINARY`. There are no other commands for this module. Follow the prompts to plot a diagram.

Syntax	BINARY_DIAGRAM
Prompts	Database: /TCBIN/
	First element
	Second element
	Phase diagram, Phase fraction (FE), G- OR A-curves (G/A): /Phase_Diagram/

## About Ternary Phase Diagrams

There are many types of ternary phase diagrams but perhaps the most common is the isothermal section. In this case, the temperature is constant, which means that the tie-lines are in the plane of the calculation. Isothermal sections and monovariant lines on the liquidus surface can be calculated in traditional way. They can also be calculated in the TERN module, but this requires a special database (such as PTERN).

In order to set up calculation of a ternary isothermal section in the system Al-Mg-Si, do the following (after retrieving the data from the database (TDB) module):

```
SET_CONDITION T=823,P=1E5,N=1,X(MG)=.01,X(SI)=.01
```

---

C\_E

Note that there is no condition on the amount of Al as that is the "rest". N=1 specifies that the system is closed. This calculation gives one point inside the system. Now define the axis:

```
S-A-V 1 X(MG) 0 1 0.01
```

```
S-A-V 2 X(SI) 0 1 0.01
```

Of course, you might not be interested in the whole section, and in such cases specify different lower and upper limits for the axes. Then give a SAVE command before the MAP command:

```
SAVE almgssi-823
```

```
MAP
```

The result can be plotted in the POST module. Ternary isopleths are equally simple to calculate. Use the same conditions as above if you are interested in a section of Al-Si at 1 mole percent of Mg, just define a different set of axis:

```
S-A-V 1 X(SI) 0 1 .01
```

```
S-A-V 2 T 500 2000 25
```

```
SAVE almgssi-1mg
```

```
MAP
```

In the resulting diagrams, the tie-lines are not in the plane of the diagram, thus the lever rule etc. cannot be applied. But you can calculate more elaborate sections. An interesting diagram is the section from pure Al to Mg<sub>2</sub>Si, which is almost a quasi-binary section. In order to calculate this, set up the conditions differently:

```
SET-CONDITION T=823, P=1e5, N=1, X(MG)-2*X(SI)=0, W(MG)=.01
```

```
C-E
```

Note that there is no condition on the amount of Si as this is given by the relation  $X(\text{MG})-2*X(\text{SI})=0$ . Also note that you can mix conditions in X and in W. Then set the axis, save the workspace and map.

```
S-A-V 1 W(MG) 0 .1 .01
```

```
S-A-V 2 T 500 1500 10
```

```
SAVE al-mg2si
```

```
MAP
```

A special type of ternary phase diagrams is liquidus surfaces. This is still not simple to calculate as a separate start point is needed for each separate line at each temperature.

However, you can easily calculate the monovariant lines where the liquid is stable together with two condensed phases:

```
SET-CONDITION T=800, P=1e5, N=1, X(MG)=0.01, X(SI)=.01
```

```

C-E
C-S P LIQ=FIX 0.3
S-C T=NONE
C-E

```

Now there should be an equilibrium where liquid is just stable. Set the axis to be the compositions.

```

S-A-V 1 X(MG) 0 1 .01
S-A-V 2 X(SI) 0 1 .01
SAVE almsgsi-luni
MAP

```

## *TERNARY\_DIAGRAM*

The TERNARY\_DIAGRAM module (short name, the TERN module) enables you to quickly calculate a simple ternary phase diagram.

To enter the module, at the SYS prompt type GOTO\_MODULE TERNARY. There are no other commands for this module. Follow the prompts to plot a diagram.

Syntax	TERNARY_DIAGRAM
Prompts	Database: /FEDEMO/
	First element: Second element: Third element:
	Phase diagram, Monovariants, or Liquidus Surface: /Phase_Diagram/
	Temperture (C) /1000/
	Global minimization on: /Y/

## About Quasi-Binary and Quasi-Ternary Phase Diagrams

Quasi-binary diagrams, as well as quasi-ternary diagrams and so forth, have often been misunderstood and misused. A real quasi-binary phase diagram is for a ternary system where one component has a fixed activity or chemical potential, and a real quasi-ternary phase diagram is for a quaternary system where one component has a fixed activity or chemical potential.

A typical example of quasi-ternary system is a quaternary system Fe-Cr-Ni-C calculated at fixed carbon activity. The condition can be set as follows:

```

SET-REF-STATE C GRAPH , , , ,
SET-CONDITION T=1273, P=1e5, N=1, X(CR)=.1, X(NI)=.1, ACR(C)=.002

```

---

In some cases, the activity may vary in a quasi-binary section if one has a phase fixed instead (and where the phase may vary in composition). The advantage with the quasi-binary section is that the tie-lines will be in the plane of the diagram and thus the lever rule etc is obeyed.

A quasi-binary section in Ca-Fe-O when the liquid oxide is in equilibrium with liquid Fe can be specified as follows:

```
DEFINE-COMPONENTS CAO FE FEO
SET-CONDITION T=1850, P=1e5, N=1, X(CAO)=.1
CHANGE-STATUS PHASE FE-LIQ=FIX 0
```

and the mapping can be done with the following axis

```
S-A-V 1 X(CAO) 0 1 .01
S-A-V 2 T 1500 2000 25
```

However, it is a problem as FE-LIQ is not the stable phase for pure iron below 1811. One will have to calculate separate sections depending on the stable modification of Fe.

---

## Calculating a Quasi-Binary Phase Diagram

There are many ways to calculate phase diagrams in Thermo-Calc. The following is an example where a quasi-binary phase diagram is used for calculations on a ternary system in which one component has an activity or chemical potential that is fixed (although if you have fixed a phase and the phase composition varies, then the activity or chemical potential may also vary). The tie-lines in a quasi-binary diagram are in the diagram's plane of the diagram. This means that the calculation follows the lever rule as well as other rules.

When calculating a quasi-binary phase diagram it is necessary to set a condition on either activity or chemical potential of the third component. To specify a meaningful value it is recommended that you first change the component's reference state using SET\_REFERENCE\_STATE. For example, to calculate a phase diagram for a quasi-binary Fe-Cr-O system with fixed oxygen activity, you can enter the following:

```
SET_REFERENCE_STATE O GAS , ,
```

You must also use SET\_CONDITION to set the activity of the component whose activity or chemical potential you have fixed. Do this by assigning a value to the ACR state variable. For example, if the component is oxygen, you might enter

```
SET_CONDITION ACR(O)=0.2
```

## Alternative Way to Set the Component Activity

As an alternative to set the activity of the component, you can Use CHANGE\_STATUS to set a phase to fixed. For example, suppose you want to compute the iron-saturated section in "FeO"-SiO<sub>2</sub>. You then enter:

```
CHANGE-STATUS FCC=FIX 0
```

---

## Calculating a Quasi-Ternary Phase Diagram

There are many ways to calculate phase diagrams in Thermo-Calc. The following is an example of a quasi-ternary phase diagram used for calculations on a quaternary system where one component has a fixed activity or fixed chemical potential.

When calculating a quasi-ternary phase diagram it is necessary to set a condition on either activity or chemical potential of the fourth component. To specify a meaningful value it is recommended that you first change the component's reference state using SET\_REFERENCE\_STATE. For example, to calculate a phase diagram for a quasi-ternary Fe-Cr-Ni-C system with fixed carbon activity, you can enter the following:

```
SET_REFERENCE_STATE C GRAPH , ,
```

You must also use SET\_CONDITION to set the activity of the component whose activity or chemical potential you have fixed. Do this by assigning a value to the ACR state variable. For example, if the component is carbon, you might enter

```
SET_CONDITION ACR(C)=0.002
```

---

## T0 Temperature Simulations

---

**In this section:**

---

About T0 Temperature Simulations .....	77
Making a T0 Temperature Simulation .....	78

---

## About T0 Temperature Simulations

The T0-temperature is the temperature at which two phases have the same Gibbs energy for a certain composition in a multicomponent system. The  $T_0$ -temperature is located within the two-phase field between the phases and it is the theoretical limit for a transformation without diffusion. T0 temperatures are often of interest if you want to understand how diffusionless transformations work. The T0 temperature in a multicomponent system with a fixed composition is temperature at which the tangent lines of the Gibbs energies of the two target phases are the same.

If the composition of one or two components varies, the common Gibbs energy for the two phases in partial equilibrium of a diffusionless transformation becomes a plane or surface, and the T0 temperature becomes a line or plane.



See Console Mode examples tcex 23 and tcex41.

## Making a T0 Temperature Simulation

This describes how to perform a T0 temperature simulation and apply a stepping calculation to its initial result.

Before you perform a T0 temperature calculation, you must have defined your initial system and be in POLY. The initial overall composition must have appropriate settings for the desired T0 temperature calculation for the two target phases.

### [POLY Module Commands](#)

1. Use SET\_CONDITION to set the conditions of your calculation in the same way as when you calculate an ordinary equilibrium.
2. Use COMPUTE\_EQUILIBRIUM to run the calculation. The calculation does not need to reach an equilibrium in which any of the target phases is stable.
3. When you have calculated your initial equilibrium, use ADVANCED\_OPTIONS and choose T-zero.
4. When prompted, specify the names of the two target phases for which the T0 temperature (where the phases' Gibbs energies are equal) is to be calculated.
5. If the T0 temperature between the two target phases is calculated, this temperature (in Kelvin) is shown in the Console window.
6. If you want to do a stepping calculation based on this initial T0 temperature calculation, then use SET\_AXIS\_VARIABLE to specify which state variable to use as the stepping variable. This is done the same way as when you calculate a normal property diagram.



When a T0 temperature simulation is run, you cannot set temperature as the stepping variable.

7. Use STEP\_WITH\_OPTIONS and choose T-zero to initiate the stepping calculation. You are prompted to specify the names of the target phases for which the T0 temperature is calculated.

During the STEP T-ZERO calculation procedure, the calculated T0 values are presented next to the stepping variable.

For example, part of the output might look like this:

```
Phase Region from 1.000000E-01 for:  
BCC_A2
```

---

FCC_A1	
1.000000E-01	940.24
9.250000E-02	941.20
2.500000E-03	977.61
7.500000E-09	979.34

When the stepping calculation has finished, you can move to the POST module and plot the T0 line against the stepping composition variable or another varying axis value. Or you can impose the calculated T0 line onto a normal phase diagram.

---

# Paraequilibrium

---

**In this section:**

---

About Paraequilibrium .....	81
Calculating a Paraequilibrium .....	82

---

## About Paraequilibrium

A paraequilibrium is a partial equilibrium where one interstitial component (such as carbon C and nitrogen N) in a multicomponent alloy can diffuse much faster than the other components (the substitutional elements, including the matrix element and alloying elements). The chemical potential for the interstitial component in two partially equilibrated phases is equal in such a case, but this is not true for the other components. In such a paraequilibrium state, it is possible to have a transformation that is partly without partitions, where a new phase of the mobile component can form with different content but where the slowly diffusing components retain their compositions.

A paraequilibrium calculation is useful when, for example, you want to study phase transformations in systems with large differences in the diffusivities of different elements. Transformations occurring under paraequilibrium states can be much more rapid than if full local equilibrium holds at the phase interface.



### [Calculating a Paraequilibrium](#)

## Calculating a Paraequilibrium

This topic describes both how to calculate an initial paraequilibrium with two target phases, and how to base a stepping calculation on this initial calculation.

Before you do any paraequilibrium calculations, you must have defined your initial alloy system and be in POLY. The initial overall composition must have appropriate settings for the desired paraequilibrium calculation for the two target phases.

1. Use [SET\\_CONDITION](#) to set the conditions of your calculation just like when you calculate an ordinary equilibrium.
2. Use [CHANGE\\_STATUS](#) to set the status of the chosen interstitial components to SPECIAL. For example, if C is an interstitial component, enter:

```
CHANGE_STATUS COMPONENTS C=SPECIAL
```

This gives you a clear picture on u-fractions of the substitutional and interstitial components, which are different from the overall composition in the system. The SPECIAL status means that the specified components are not included in summations for mole or mass fractions. Therefore, all the composition variables plotted from paraequilibrium calculations are u-fraction related quantities.

3. Use [COMPUTE\\_EQUILIBRIUM](#) to run the calculation.
4. When you have calculated your initial equilibrium, use [ADVANCED\\_OPTIONS](#) Paraequilibrium to perform the paraequilibrium calculation.
5. Specify the names of the target phases of the paraequilibrium state. For example, FCC#1 BCC and FCC#2 M23C6.



Both phases must have similar interstitial/vacancy sublattices that the fast-diffusion interstitial components occupy. Both target phases should have phase constitution definitions that cover all the defined substitutional and interstitial components of the current alloy system.

6. Specify the names of one or more fast-diffusing components. These components must be located on the interstitial/vacancy sublattices in both of the chosen phases.



Interstitial components (for instance C and N combined) may have significantly different behaviours depending on how they are partitioned in different structured phases.

If the paraequilibrium between the two specified phases is successfully calculated, then a message is displayed, for example:

```
NP(FCC) = 0.3586 with U-fractions C = 2.71821E-02 N =
4.1548129E-03

NP(BCC) = 0.6414 with U-fractions C = 7.10061E-04 N =
2.3781027E-04

All other compositions are the same in both phases
```

The first two lines show the phase amounts expressed in mole-percent [NP(phase)] and the contents of the interstitial components C and N in each phase. These contents are expressed in u-fractions. The third line states that the compositions of the matrix component and all the remaining compositions (regarding substitutional components) are equal in the two target phases at the paraequilibrium state.

7. If you want to do a stepping calculation based on this initial paraequilibrium, then use [SET\\_AXIS\\_VARIABLE](#) to specify which state variable to use as the stepping variable. This is done exactly in the same way as when you calculate a normal property diagram.
8. Use [STEP\\_WITH\\_OPTIONS](#) → `Paraequilibrium` to initiate the stepping calculation. You are prompted to specify the names of the target phases of the paraequilibrium states as well as the fast-diffusing components.

The stepping calculation is performed, and the different phase regions are listed with columns for the value of the stepping variable, the amounts of the two target phases, the u-fractions of interstitial(s) in each of the two phases, and the LNACR value(s) of the interstitial component(s).

This command calculates a paraequilibrium between two phases.  
 You must calculate an equilibrium with the overall composition first.  
 Name of first phase: fcc\_a1  
 Name of second phase: bcc\_a2  
 Fast diffusing component: /C/: C  
 Fast diffusing component: /NONE/:  
 Output during stepping is:  
 axis value, phase amounts, u-fractions of interstitial(s) in phase 1 and 2,  
 and LNACR value(s) of interstitial(s)

```
Phase Region from 900.000 for:
  BCC_A2
  FCC_A1
9.000000E+02 0.176 0.824 4.977501E-02 6.609390E-04 -2.303643E-01
8.900000E+02 0.161 0.839 5.421135E-02 6.889794E-04 -1.689777E-02
8.800000E+02 0.148 0.852 5.872152E-02 7.140931E-04 1.943989E-01
8.700000E+02 0.137 0.863 6.329608E-02 7.362105E-04 4.040959E-01
8.600000E+02 0.127 0.873 6.792661E-02 7.552871E-04 6.126989E-01
8.500000E+02 0.119 0.881 7.260558E-02 7.713010E-04 8.206607E-01
8.400000E+02 0.111 0.889 7.732629E-02 7.842505E-04 1.028392E+00
8.300000E+02 0.105 0.895 8.208278E-02 7.941530E-04 1.236267E+00
8.200000E+02 0.099 0.901 8.686971E-02 8.010431E-04 1.444633E+00
8.100000E+02 0.093 0.907 9.168236E-02 8.049714E-04 1.653814E+00
8.000000E+02 0.089 0.911 9.651648E-02 8.060032E-04 1.864114E+00
```



It is possible to now move to the POST module and plot some of the results of the calculation. For an example of a paraequilibrium calculation and how it can be plotted, see Console Mode example tcex23.

---

## Potential Diagrams

---

### In this section:

---

About Potential Diagrams .....	86
POTENTIAL_DIAGRAM .....	86
Calculating a Potential Diagram .....	88
Calculating a Potential Diagram with Different Pressure .....	90

## About Potential Diagrams

If you have a metal-oxide/sulphide-gas interaction system, you might want to calculate a so-called potential phase diagram given certain temperature and pressure. A potential diagram uses the activities of two major species in the gaseous mixture phase as mapping variables. The activities (that is, the fugacities at 1 bar) of these two species are typically set as the X- and Y-axis when the diagram is plotted. The phase relations between the gaseous mixture and various metal forms, metal-oxides, metal-sulphides, or other metal-bearing solids, are represented by different fields controlled by these two potential quantities.

This topic describes how to calculate such potential diagrams with the POTENTIAL module, and how to later modify the pressure of such a calculation in POLY. The calculation done by the POTENTIAL module always assumes a pressure of 1 bar.

## POTENTIAL\_DIAGRAM

The POTENTIAL\_DIAGRAM (POT or POTENTIAL) module is a potential phase diagram calculation module.

If you have a metal-oxide/sulphide-gas interaction system, you might want to calculate a so-called potential phase diagram given certain temperature and pressure. A potential diagram uses the activities of two major species in the gaseous mixture phase as mapping variables. The activities (that is, the fugacities at 1 bar) of these two species are typically set as the X- and Y-axis when the diagram is plotted. The phase relations between the gaseous mixture and various metal forms, metal-oxides, metal-sulphides, or other metal-bearing solids, are represented by different fields controlled by these two potential quantities.

To enter the module, at the SYS prompt type `GOTO_MODULE POTENTIAL`.



There are no commands for this module. Follow the prompts and see [About Potential Diagrams](#) for details about how to calculate using potential diagrams.

Syntax	POTENTIAL_DIAGRAM
Prompts	Database: /POT/ Press <Enter> and the program lists the available databases. Type one of the database names and press <Enter> to select. The following default elements are then based on the chosen database.
	Matrix element: /Fe/
	First potential species: /S1O2/
	Second potential species: /O2/

Syntax	POTENTIAL_DIAGRAM
	Temperature: /1000/

---

## Calculating a Potential Diagram

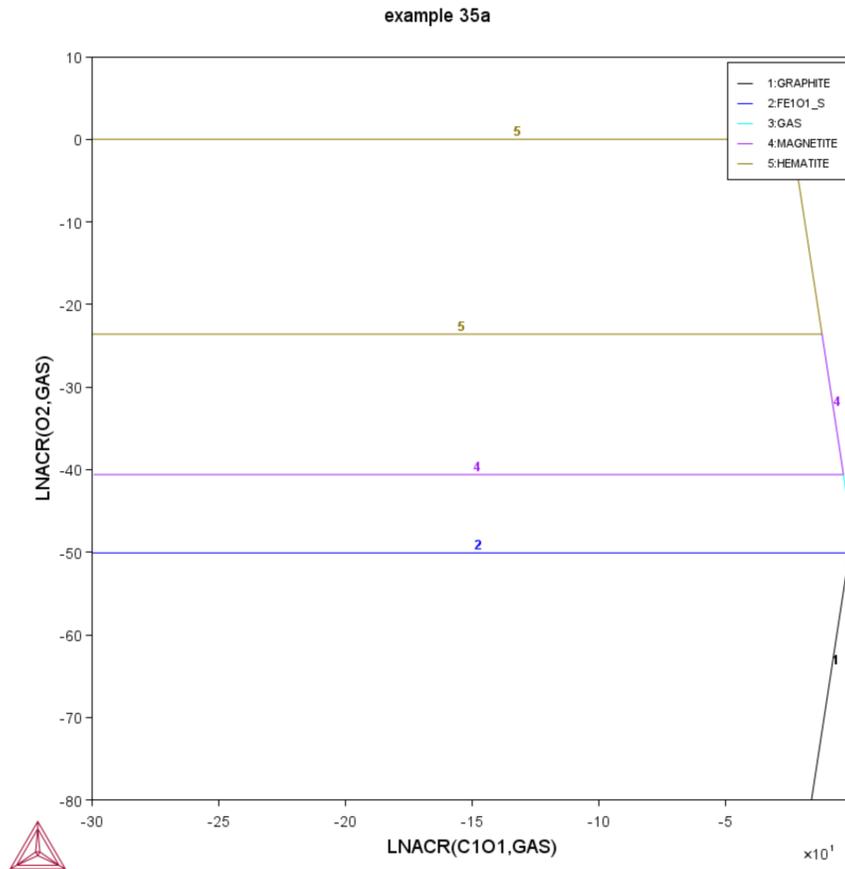
You need access to a substance or solution database that contains a gaseous mixture phase, metals (or alloys), and solids such as metal-oxides, sulphides, sulphates, carbonates, silicates, nitrides or nitrates (stoichiometric or solution phases).

1. Use `GOTO_MODULE` and enter `POTENTIAL`.
2. Specify which substance or solution database to use. Any substance or solution database can be used that has a gaseous mixture phase, metals (or alloys), and solids such as metalxides, sulphides, sulphates, carbonates, silicates, nitrides, and nitrates (stoichiometric or solution phases).
3. Specify the matrix element (a metal) and the two gaseous species that define you metal-gas interaction system. The names of these components must be available in the database you are using.
4. Set the temperature (in Kelvin). (The pressure is always 1 bar.)

A potential diagram with the activities of the two gaseous species as mapping variables is calculated. These mapping variables are also set as the axis variables in the plotted diagram.



An example diagram (Console Mode example 35).



A POLY3 file is also automatically saved in the current working directory with the default name POT.POLY3.

5. When the diagram is plotted, it opens the POST module where you can modify the plotted diagram in any way you like.

## Calculating a Potential Diagram with Different Pressure

It is possible to modify both the calculation and the plotted diagram that the POTENTIAL module generates in the POLY and POST modules. You can do this by opening a POLY3-file with the results from a POTENTIAL module calculation in either POLY or POST. You can modify the calculation in the same way that you can modify any ordinary mapping calculation or phase diagram. For example, this is useful for calculating a potential diagram at a pressure other than 1 bar.

### [POLY Module Commands](#)

You must have calculated a potential diagram already, and have the saved POLY3-file available.

1. Use GOTO\_MODULE and enter `POLY`.
2. Use READ followed by the name of the POLY3 file that contains the results of the potential diagram calculation that you want to modify.



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.

3. This opens the workspace file and loads the previous settings and results. The activities of the two gaseous species is now part of the conditions, and these conditions are set as the mapping variables.
4. Use SET\_CONDITION to specify the pressure under which you want the potential diagram to be calculated. For example, to set the pressure to 1000 bar, enter:

```
SET_CONDITION P=1e8
```

5. Use COMPUTE\_EQUILIBRIUM to compute the initial equilibrium.



You may also want to use ADD\_INITIAL\_EQUILIBRIUM to calculate some additional equilibria that the mapping calculations are based on.

6. Use MAP to perform the mapping calculations.
7. Use GOTO\_MODULE and enter `POST` to set conditions for visualization and to plot the diagram.

---

## Scheil Simulations

---

### In this section:

---

About the Scheil-Gulliver Solidification Simulations .....	92
Scheil in Graphical Mode vs Console Mode .....	97
Examples of Scheil-Gulliver Calculations .....	98
Simulating a Scheil Solidification Process .....	100
State Variables for Scheil .....	102

## About the Scheil-Gulliver Solidification Simulations



In Console Mode most of the set up for the Scheil module is via the [START WIZARD](#) command where you are prompted to answer a series of questions. You can also perform Scheil calculations in Graphical Mode using the Scheil Calculator, which is recommended for most users. See [Scheil in Graphical Mode vs Console Mode](#) to compare.

Thermo-Calc is often used to perform equilibrium calculations, but some non-equilibrium transformations or partial-equilibrium transformations can also be simulated.

A well-known example of a non-equilibrium calculation is the *Scheil-Gulliver solidification simulation*. In Thermo-Calc, the available Scheil-Gulliver solidification simulations are *classic Scheil*, Scheil with *back diffusion* in the primary phase, or Scheil with *solute trapping*. There are also configuration options available with *fast diffusers* for the classic and back diffusion models.

### Classic Scheil Simulation

This is a Scheil simulation based on the well-known *Scheil-Gulliver* model. If this model is used without "fast diffusers" it is *Classic Scheil* simulation with the following assumptions:

- Diffusion in the liquid phase is assumed to be very fast, that is, infinitely fast.
- Diffusion in the solid phases is so slow that it can be ignored, that is, diffusion is assumed to be zero.
- The liquid/solid interface is in thermodynamic equilibrium.

In this Classic Scheil simulation the temperature is decreased step-by-step. When the temperature drops below the liquidus temperature the equilibrium amount and composition of solid and liquid phase is calculated. The solid phase is removed from the system and only the amount and composition of the liquid phase is used for the next calculation step at a lower temperature. Again, the equilibrium amount and composition of solid and liquid phase is calculated and again the solid phase is removed from the system for the next step. This procedure is repeated until the last liquid disappears.

This calculation procedure is equivalent to assuming that there is no diffusion in the solid phase and infinitely fast diffusion of all elements in the liquid phase. It has been shown to be a good approximation of the solidification of most alloys such as Ni-superalloys, Cu alloys, Al alloys, Mg alloys, and others. However, in materials with interstitial elements, such as carbon, ignoring diffusion in the solid causes discrepancies with experimental results.

Interstitial elements have much faster diffusion rates. The assumption of no diffusion in the solid phase during solidification is thus not correct at most industrial or lab-scale solidification rates.

For this reason, a variant of the Scheil simulation was developed and implemented in Thermo-Calc, where one or more elements can be defined as *fast diffusers*. See [Other Options: Fast Diffusers and Scheil Simulations](#).

### *Scheil Simulation with Back Diffusion in Primary Phase*

The *Scheil simulation with back diffusion in primary phase* kinetic model quantitatively takes into account the real back diffusion of all elements in the primary solid phase (typically the FCC or BCC phase). This model requires the use of both a thermodynamic and mobility database. The calculation also requires the cooling rate to be specified; a fast cooling rate allows for less time for back diffusion and the simulation result is similar to the classic Scheil-Gulliver solidification simulation. A very slow cooling rate allows for almost complete back diffusion, the solidification simulation is then close to an equilibrium calculation. It also requires that the size of the solidification domain is specified. In most cases this domain size corresponds to the secondary arm spacing, as this is where typically the liquid is trapped during solidification.



The secondary arm spacing also depends on the cooling rate. For this reason, a simple empirical relation between cooling rate and secondary arm spacing is predefined in Thermo-Calc.

For the *Scheil simulation with back diffusion in primary phase* model the following assumptions are made.

- Diffusion of all elements in the liquid phase is infinitely fast.
- Diffusion of all elements in the primary solid phase are quantitatively calculated using mobility data, a cooling rate and a domain size (typically this is the secondary arm spacing).
- The liquid/solid interface is in thermodynamic equilibrium.

Also see [Other Options: Fast Diffusers and Scheil Simulations](#).

### *Scheil with Solute Trapping*

The *Scheil with solute trapping* model simulates deviation from local equilibrium for the primary phase. It is useful for high solidification speeds, such as those seen in additive manufacturing applications and is based on the Aziz [1982] model for solute trapping and

extended to multicomponent systems.

The model is based on the main assumption that the primary phase is the only dendrite forming phase affected by solute trapping.

For equilibrium- and Scheil-type solidification simulations thermodynamic equilibrium is established at the solid-liquid interface. This means that solutes are partitioned between the solid and liquid phases according to solidus and liquidus lines of the phase diagram. The assumption of thermodynamic equilibrium at the solid-liquid interface however breaks down for very fast solidification rates, that are encountered for example during additive manufacturing. Aziz [1982, Aziz; 1988, Aziz], proposed a model that links partitioning at the solid-liquid interface to solidification rate / interface velocity. Basically, the faster the solidification rate, the lower is the partitioning at the interface. Put very simply, what is thought to happen, is that the solid-liquid interface moves so quickly, that the solutes in the liquid are trapped in the advancing solid phase, thus the name “solute trapping”.

Aziz [1982] proposed a model to account for solute trapping based on balancing the overall kinetics at the solid-liquid interface. The two terms to be balanced are the velocity of the interface, that is calculated based on the cooling rate and considerations of the solid-liquid geometry (dendrite arm spacing) the interface kinetics, that follows from the trans-interface diffusion (meaning how fast solutes travel from one side of the solid-liquid interface to the other) and the change of phase from liquid to solid without changing the composition.

For a *Scheil with solute trapping* model the following assumptions are made.

- Only one primary solid phase forms dendrite, NOT necessarily the first solid phase
- Solute trapping in primary solid phase only. Other solid phases have equilibrium compositions following Scheil model
- Amounts of solid phases are dependent on solute trapping and solidification speed
- Dynamic liquidus for primary solid phase is dependent on solute trapping and solidification speed

### ***Solute Trapping References***

[1982Azi] Aziz, M. J. (1982). Model for solute redistribution during rapid solidification. *Journal of Applied Physics*, 53(2), 1158–1168.

[1988Azi] Aziz, M. J., & Kaplan, T. (1988). Continuous growth model for interface motion during alloy solidification. *Acta Metallurgica*, 36(8), 2335–2347.

## Other Options: Fast Diffusers and Scheil Simulations

If a *Classic Scheil* or *Scheil simulation with back diffusion in primary phase* model is selected plus one or more *Fast diffuser* checkboxes selected, then Thermo-Calc takes into account the concept of the *fast diffuser*. Typically, C is defined as a fast diffuser, but also other elements such as N, O, or others can also be defined.



*Fast diffuser* does not quite define this since diffusion is not actually simulated. It is simply assumed that the “fast diffusers” distribute throughout the solid and liquid phase according to thermodynamic equilibrium. This corresponds to *infinitely fast diffusion* for the selected elements.

For a *Classic Scheil* model with the compositions defined with *Fast diffuser* the following assumptions are made.

- Diffusion of all elements in the liquid phase is infinitely fast.
- Diffusion of all elements in the solid phases, except the ones defined as “fast diffusers”, is zero.
- Diffusion of the elements defined as “fast diffusers” is infinitely fast in the solid phase.
- The liquid/solid interface is in thermodynamic equilibrium.

For a *Scheil simulation with back diffusion in primary phase* model with the compositions defined with *Fast diffuser* the following assumptions are made.

- Diffusion of all elements in the liquid phase is infinitely fast.
- Diffusion of all elements in the primary solid phase are quantitatively calculated using mobility data, a cooling rate and a domain size (typically this is the secondary arm spacing).
- Diffusion of the elements defined as “fast diffusers” is infinitely fast in the solid phase.
- The liquid/solid interface is in thermodynamic equilibrium.

The Classic Scheil simulation and Classic Scheil simulation with fast diffuser models can be considered *thermodynamic* models. There are clear limitations as a certain amount of back diffusion always takes place in the solid phase at real solidification.

You can also use the more sophisticated *Scheil simulation with back diffusion in primary phase* kinetic model. Adding the fast diffuser option to this can be useful, for example for interstitials like C in steels where these then equilibrate in all phases and not just the primary phase.

### *Homogenization, Segregation Profiles, and the Diffusion Module*



Scheil calculations can be used in diffusion simulations with the Diffusion Module (DICTRA).

Homogenization is a process where you want to get rid of the composition segregation from solidification with a homogenization heat treatment. This composition segregation can be simulated with Scheil. Then with the Diffusion Module (DICTRA), you input the secondary dendrite arm spacing to get the right diffusion distances. With composition profile from file the idea is that you should be able to input for example a measured composition profile (from EDS/WDS, micro-probe etc). This could also be a segregation profile but also composition profiles over a weld, substrate/coating, diffusion couple, and so forth.

### *Summary*

The available Scheil-Gulliver solidification simulations are *classic* Scheil, Scheil with *back diffusion* in the primary phase, or Scheil with *solute trapping*. There are also configuration options available with *fast diffusers* for the classic and back diffusion models.



There is also connectivity to the Diffusion Module (DICTRA) and *homogenization* where you collect data from a Scheil solidification calculation, which is then used as part of the kinetic homogenization simulation of segregated composition profiles.



[Scheil in Graphical Mode vs Console Mode](#)



The Scheil with back diffusion feature is only available for systems with diffusion data, i.e. this model requires the use of a mobility database.

In Thermo-Calc, the Scheil calculation allows the calculation of:

- The solidification range of an alloy.
- Depression of the solidus temperature due to segregation.
- Composition of the last liquid to disappear in segregation pockets.

- Phases formed on final solidification in segregation pockets.
- The composition gradient in the primary solid phase(s) (segregation profile).
- Solute trapping to simulate deviation from local equilibrium for the primary phase
- Kinetic homogenization simulation of segregated composition profile (diffusion calculations)

## Scheil in Graphical Mode vs Console Mode

Type of Calculation	Console Mode	Graphical Mode
Classic Scheil	Use the START_WIZARD command to answer prompts	Scheil Calculator > select the Classic Scheil button on the Configuration window.  On the My Project Configuration window under Getting Started, click Quick Start and select Scheil solidification simulation.
Scheil with back diffusion in the primary phase	USE_BACK DIFFUSION command (turn on) then use the START_WIZARD command to answer prompts	Scheil Calculator > select the Scheil with back diffusion in the primary phase button on the Configuration window then enter the settings as needed. A mobility database is required.
Scheil with solute trapping	SOLUTE_TRAPPING command (turn on) then use the START_WIZARD command to answer prompts	Scheil Calculator > select the Scheil with solute trapping button on the Configuration window then enter the settings as needed.
Fast diffusers and Classic Scheil	Use the START_WIZARD command to answer the prompt <code>Fast diffusing components</code> where you enter the elements that diffuse fast in solid phases and can assume to be in equilibrium in the whole phase. E.g. C in a steel.	Scheil Calculator > select the Classic Scheil button plus one or more of the Fast diffuser checkboxes next to the composition(s) on the Configuration window then enter the settings as needed.
Fast diffusers and Scheil with back diffusion in the primary phase	USE_BACK DIFFUSION command (turn on) then use the START_WIZARD command to answer the prompt <code>Fast diffusing components</code> where you enter the elements that diffuse fast in solid phases and can assume to be in equilibrium in the whole phase. E.g. C in a steel.	Scheil Calculator > select the Scheil with back diffusion in the primary phase button plus one or more of the Fast diffuser checkboxes next to the composition(s) on the Configuration window then enter the settings as needed. A mobility database is required.
Homogenization	Use the EVALUATE_SEGREGATION_PROFILE	Use the <b>Homogenization</b> template

Type of Calculation	Console Mode	Graphical Mode
	<p>command before START_WIZARD to generate the segregation profile to file. In DICTRA_MONITOR use INPUT_SCHEIL_PROFILE for use of segregated profile in simulation. An additional license to the Diffusion Module (DICTRA) is needed to work with more than three components.</p>	<p>under <b>Non-Equilibrium</b> to automatically set up the project tree with the defaults needed to work with this. An additional license to the Diffusion Module (DICTRA) is needed to work with more than three components.</p>

## Examples of Scheil-Gulliver Calculations



[About the Scheil-Gulliver Solidification Simulations](#)

### ***tcex15 - Simulating the Solidification of a Cr-Ni Alloy***

This is an example of a solidification simulation of a Cr-Ni alloy. There is no back diffusion in the solid, i.e. this is a classic Scheil-Gulliver simulation. The FEDMO database is used and included with your installation.

These commands are of note and used in this example:

- [START\\_WIZARD](#)
- [COMPUTE\\_TRANSITION](#)
- [ADD\\_INITIAL\\_EQUILIBRIUM](#)
- [ENTER\\_SYMBOL](#)
- [ADVANCED\\_OPTIONS](#)
- [STEP\\_WITH\\_OPTIONS](#)
- [MAKE\\_EXPERIMENTAL\\_DATAFILE](#)
- [CHANGE\\_STATUS](#)

### ***tcex30a and 30b - Scheil Calculation for an Al-4Mg-2Si-2Cu Alloy***



[The Equilibrium Calculation Module \(POLY3\)](#) and [About the Scheil-Gulliver Solidification Simulations](#)

These examples show two ways to do a Scheil calculation for an Al-4Mg-2Si-2Cu alloy. In part A, you use the POLY3 module and the STEP\_WITH\_OPTIONS command with an EVALUATE option. Then in part B you use the SCHEIL module commands to do the same thing. Data from the COST2 database is used and included with the example in the file named *tcex30\_cost2.TDB*.

### ***tcex48 - Scheil Solidification with C Back Diffusion***

The FEDEMO database is used and the example is available to all users.

This is an example of Scheil solidification with simulated back diffusion of carbon in solid phases. The results are compared between a simple Scheil and equilibrium calculation. First perform a Partial Equilibrium Scheil by assigning carbon as a fast diffusing element. Plot solidification curve and a microsegregation diagram and save to file (so called .exp files). The "back diffusion" of C in solid phases is not from any calculation involving kinetics, but comes from the equilibration of chemical potential of C at each step in temperature, thus allowing each already solidified part of the system to change its C concentration.



See *tcex54* for an example that calculates "real" back diffusion. This model requires a kinetic (mobility) database in addition to the thermodynamic one. Also see [About the Scheil-Gulliver Solidification Simulations](#) for details about the different Scheil options.

### ***tcex54 - Scheil Solidification of an Al-2.1Cu-1Si Alloy including Back Diffusion***

This is an example of a solidification simulation of a Al-2.1Cu-1Si alloy. The back diffusion option in the Scheil module is used, i.e. there is back diffusion in the primary phase. The example uses both the thermodynamic ALDEMO and mobility MALDEMO demo databases included with your installation.

### ***tcex57 - Scheil Solidification with Solute Trapping***

This is an example of a solidification simulation of a Al-7.5Si-0.2Cu system. The solute trapping option in the Scheil module is used. The example uses the thermodynamic ALDEMO demo database included with your installation.



See the Graphical Mode example [T\\_13: Scheil Solidification with Solute Trapping](#) for an additional comparison to Classic Scheil.

## Simulating a Scheil Solidification Process



[SCHEIL Module Commands](#) and [About the Scheil-Gulliver Solidification Simulations](#)

Simulating a Scheil-Gulliver solidification process involves calculating the liquid composition of a higher-order multicomponent system at each step of a cooling process, and resetting the liquid composition as the composition of the entire system (after having removed all amount of solid phase).



The step-by-step instructions below are intended as a simple example of setting up a calculation using the wizard in Console Mode. You can also use the macros available and briefly described in [Examples of Scheil-Gulliver Calculations](#).



The following does not include any prompts related to using back diffusion or solute trapping. See [START\\_WIZARD](#) for additional details about when these options are enabled with the [USE\\_BACK\\_DIFFUSION](#) and [SOLUTE\\_TRAPPING](#) commands.

1. At the prompt type `GOTO_MODULE SCHEIL`.
2. Type `START-WIZARD` and press `<Enter>`.
3. Specify which `Database` to use. Type `?` for a list of available databases based on your license.
  - Type `FEDEMO` for this example.
4. Specify the `Major element or alloy` in your system. For example, enter `Fe` (steel/Fe-alloy), `Al` (Al-based alloy), or `Ni` (Ni-based super alloy).
  - Type `Fe` for this example.
5. Set whether to specify the `Composition input in mass (weight) percent (the default) or in Mole percent`. Press `<Enter>` to keep the default for mass (weight) percent or type `N` to use mole percent.
  - Keep the default for this example (press `<Enter>`).
6. Specify the name of the `1st alloying element`. You can directly specify mass or mole percent after the name. If this is not specified, then you are prompted to enter it separately.
  - Type `Cr 5` for this example.

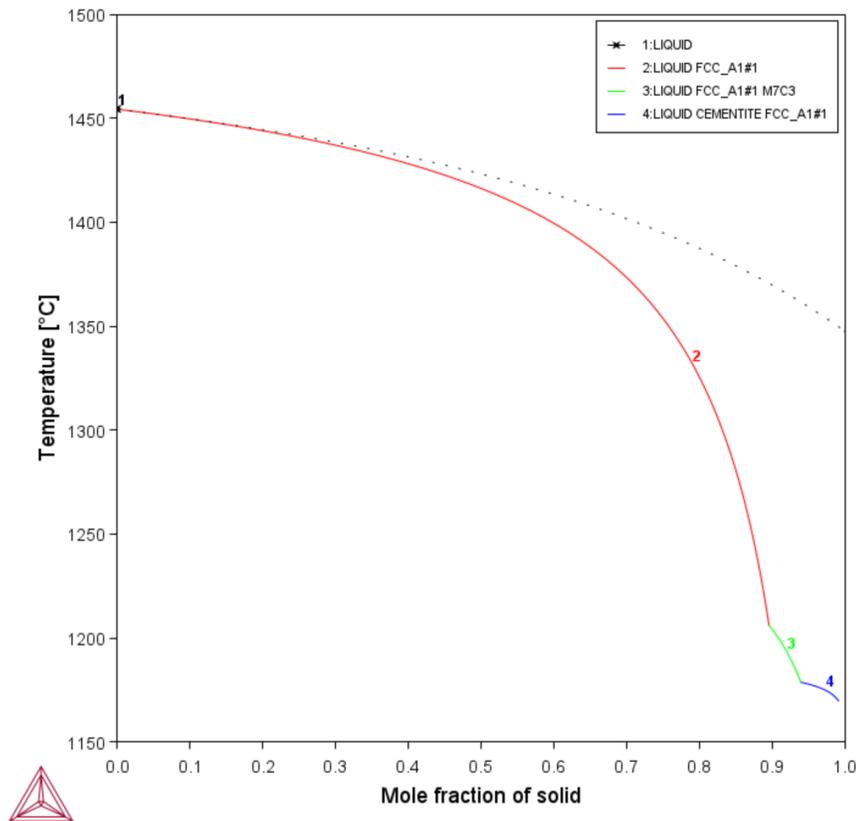
7. Specify the other alloying elements in the same way as you specify the first.
  - Type `C 1` for this example.
8. After you have specified your last alloying element, press `<Enter>`. This ends the process of defining the bulk composition of the alloy system.



You can also specify all the alloying elements and their corresponding compositions on the same line when you are prompted to specify your first alloying element. As a general example, you can enter `Cr 5 Ni 1 Mo 0.5 C 0.01 N 0.02`.

- Press `<Enter>` for this example.
9. Specify the starting `Temperature (C)` in Celsius. The default is `2000`.  
This value should be sufficiently high so that the solidification simulation starts with the alloy system in the liquid single-phase region.
    - Keep the default for this example (press `<Enter>`).
  10. At the `Reject phase(s)` prompt enter the name of the phases to reject or press `<Enter>` for the default `NONE`.
    - Keep the default `s` for this example (press `<Enter>`).
  11. At the `Restore phase(s)` prompt you may want to restore a phase that you rejected when you ran the simulation earlier, or you may want to restore a phase that is rejected by default in your database. Enter the name of the phases to restore or press `<Enter>` for the default `NONE`.
    - Keep the default for this example (press `<Enter>`).
  12. When prompted `OK?` type `N` to go back to reconsider which phases to reject or restore. Type `Y` to continue. The thermodynamic data about the alloy system you defined is retrieved from the database.
    - Keep the default for this example (press `<Enter>`).
  13. At the `Should any phase have a miscibility gap check` prompt, press `<Enter>` for the default `N` or type `Y` then specify the `Phase with miscibility gap`.
    - For this example, type `Y` then enter `FCC` as the `Phase with miscibility gap`.
  14. If you answer `Y` to the miscibility gap check, then next enter the following for `Major constituent(s) for sublattice 1` and `Major constituent(s) for sublattice 2`.

- Enter `Cr` for sublattice 1 and `c` for sublattice 2 in this example. Then at the Phase with miscibility gap prompt press `<Enter>`. The system responds with `LIQUID PHASE NAME: LIQUID`.
15. Specify the names of any Fast-diffusing components. Press `<Enter>` for `NONE` or type all the names on the same line then press `<Enter>`. Keep the default for this example (press `<Enter>`).



After the Scheil-Gulliver solidification simulation runs, the diagram Temperature versus Mole Fraction of Solid is plotted in the Console Results window.

## State Variables for Scheil

Post processing is done the same way as POLY-3: specific variables for plotting in the Scheil module are listed.



[POLY Module Commands](#)



For a comparable list in Graphical Mode, see [Scheil Calculator Available Plot Variables](#).

The variables can be plotted along the X- and Y-axes. In the table, *ph* is an abbreviation of *phase*, and *el* is an abbreviation of *element*. An asterisk (\*) can be used as a wild character for *el* and *ph*. Use the abbreviations of the variables when you specify what the X- and Y-axes represent in a diagram.

<i>Abbreviation</i>	<i>State Variable</i>
T	Temperature in Celsius
NL/BL/VL	Mole/mass fraction of liquid
NS/BS/VS	Mole/mass fraction of all solid phases
NS(ph)/BS(ph)	Mole/mass fraction of a solid phase
VS(ph)	Volume fraction of a solid phase
W(ph,el)	Weight fraction of an element in a phase
X(ph,el)	Mole fraction of an element in a phase
Y(ph,el)	Site fraction of an element in a phase
NN(ph,el)	Distribution of an element in a phase
NH/BH	Total heat per mole/gram (top curve) and Latent heat per mole/gram (bottom curve)
CP/BCP	Apparent heat capacity per mole/gram
NV/NV(ph)	Molar volume of the system or a phase
DS/DS(ph)	Average density of the system or a phase
BT	Apparent volumetric CTE of the system
DVIS(ph)	Dynamic viscosity of a phase
KVIS(ph)	Kinematic viscosity of a phase
SURF(ph)	Surface tension of a liquid phase

---

<i>Abbreviation</i>	<i>State Variable</i>
ELRS/ELRS(ph)	Electrical resistivity of the system or a phase
ELCD/ELCD(ph)	Electrical conductivity of the system or a phase
THCD/THCD(ph)	Thermal conductivity of the system or a phase
THRS/THRS(ph)	Thermal resistivity of the system or a phase
THDF/THDF(ph)	Thermal diffusivity of the system or a phase
DGV	Driving force for evaporation
DHV	Evaporation enthalpy
MMG	Molar mass of gas
XAVG(el)	Mole fraction of an element in solid phases
WAVG(el)	Mass fraction of an element in solid phases

---

## Aqueous Solutions and Pourbaix Diagrams

---

### In this section:

---

About Aqueous Solutions and the Pourbaix Diagram .....	106
Calculating a Pourbaix Diagram .....	109
Plotting More Pourbaix Diagrams .....	112
Stepping Calculations in an Aqueous Solution .....	114
POURBAIX_DIAGRAM Module .....	115

## About Aqueous Solutions and the Pourbaix Diagram



This command or module is not available with GES6.



GES6 is a re-write and new implementation of the Gibbs Energy System module of Thermo-Calc and it is the default engine. The previous version, GES5, is still available and will continue to work in the background of the program where necessary. The main purpose of GES6 is to support faster development of new features. However, not all GES5 functionality is implemented in GES6. In such cases (except for the POURBAIX module), GES6 falls back silently and uses functionality from GES5 in the background.

Heterogeneous equilibrium calculations involving aqueous solutions are not different from other equilibrium calculations, but the setup of the equilibrium conditions and the reference states of the system is more complex. For example, the pH and Eh values of the aqueous solution phase are normally complex functions of bulk composition, temperature and pressure conditions.

For some simple systems, thermodynamic approximations of pH-Eh calculations can be used. But Thermo-Calc also has comprehensive thermodynamic models and databases for various non-ideal solution phases. These let you accurately calculate pH and Eh property diagrams (and many other types of property diagrams) in complex aqueous solutions in a variety of conditions.

A Pourbaix diagram is a kind of phase diagram that shows the stability boundaries for a metal-aqueous interaction system. The phase boundaries are shown as a function of pH (acidity) and Eh (standard hydrogen electronic potential). An aqueous solution phase is always present in such a system. At a given pH and Eh, a metal may lose its stability to a soluble or corrosive aqueous solution, or be in equilibrium with either the aqueous solution (insoluble/immune) or with a secondary-phase file that has formed (consisting of oxides, hydroxides, sulphides or other solids). In the latter case, further dissolution of the passive or protective metal is prevented.

A Pourbaix diagram is a pH-Eh phase diagram where the tie-lines are not in the plane. The chemical potentials or activities of two system components ( $H^+$  and EA) are used as mapping variables and the pH and Eh quantities are normally plotted on the X- and Y-axes of the diagram. You can use many other properties as axis variables in various property diagrams for the interaction system at a specific temperature and pressure. The phase relations

between the aqueous solution and various metal forms, oxides, hydroxides, sulphides, sulphates, nitrates, silicates, carbonates and other solids, or gaseous mixtures, are represented as different fields defined by acidity and electric potential.

The POURBAIX module lets you construct Pourbaix diagrams and other types of property diagrams for heterogeneous interaction systems with aqueous solution phases. The compositional definition and variations and the basic solvent and electron components H<sub>2</sub>O, H+1 and ZE are automatically determined by the aqueous solution phase in the Pourbaix module.



See Console Mode examples [tcex40](#) and [tcex53](#).

## Key Components

These key components are defined as follows:

Component	Component status	Reference state	Ref-T(K)	Ref-P(Pa)
H <sub>2</sub> O	ENTERED	AQUEOUS	*	100000
H+1	ENTERED	SER		
ZE	ENTERED	REFERENCE_ELECTRODE	*	100000

Besides the aqueous solution phase, your system may also contain a gaseous mixture phase. Other neutral and charged gaseous species may also be constituents. For multicomponent systems there are normally also some interacting solid phases (stoichiometric or solution) as the so-called secondary phases. These are typically the matrix and/or precipitate phases in the alloys of interest.

Some databases have been specially developed for the Pourbaix module's diagram calculations. Such a database contains all necessary phases, i.e., an aqueous solution phase, a gaseous mixture phase, some alloy solid solution phases and some secondary solid phases. Databases developed for the Pourbaix module include *PAQ* (TCS Public Aqueous Solution Database) or *PAQS* (TGG Public Aqueous Solution Database).



If such a dedicated Pourbaix module is not used, then an aqueous solution database such as *TCAQ* (TC Aqueous Solution Database), *AQS* (TGG Aqueous Solution Database), or any user-specified aqueous databases can be used.

This database contains at least the AQUEOUS and REFERENCE\_ELECTRODE phases. The aqueous solution database needs to be appended with additional required data and phases, such as interacting alloy (solid) solution phases and secondary (solid and liquid) phases. Additional data is from one or more databases, such as SSUB/SSOL/TCFE or other substances/solutions databases.

---

## Calculating a Pourbaix Diagram

The POURBAIX module lets you construct Pourbaix diagrams and other types of property diagrams for heterogeneous interaction systems with aqueous solution phases.

When calculating a Pourbaix diagram, you need to have a database with an aqueous solution phase (that is, a phase with water as the dissolving solvent and with aqueous cation/anion and complex species as the dissolved solutes). Due to restrictions of aqueous solution models (SIT, HKF, Pitzer) used in the Thermo-Calc software, the aqueous solution database must be designed in the same Thermo-Calc database format as that used in the PAQ, PAQS, TCAQ and AQS databases for the aqueous solution phase.

Pourbaix diagram calculations are done in the advanced Pourbaix module.

### *How to Calculate a Pourbaix Diagram*

1. At the `sys` prompt, enter `GOTO_MODULE POURBAIX`.
2. At the `Need any instruction on the POURBAIX module?` prompt, press `<Enter>` to skip or enter `y` to learn more.
3. Press `<Enter>` at the prompt `Enforce a PAUSE after plotting when running a MACRO?`.
4. Press `<Enter>` to accept the default `1` to `Start a completely new POURBAIX diagram calculation` or enter `3` to `Open an old file and make another POURBAIX calculation`.

To choose `3`, you must have already done a Pourbaix calculation and have it saved in a `POLY3` file. The new calculation uses the same system definition as the previous calculation, but the other conditions, such as the bulk composition, temperature and pressure, can be modified in the next step.

5. Type `y` or `n` to set whether to `Consider the GAS phase (gaseous mixture phase) in the calculation`.

The gaseous mixture phase should at least contain `H2O`, `O2` and `H2`. If a `GAS` phase is not considered, then only the interactions between the aqueous solution phase and various solid phases (alloy phases and secondary phases) are calculated. Such a calculation may not accurately present all the heterogeneous interactions. Some secondary phases are therefore usually needed. Such phases exist in the PAQ or PAQS Public Aqueous Solution Databases. They can also be appended from the `SSUB/SSOL/TCFE` or other appropriate substances/solutions databases.

6. Type `Y` or `N` to set whether to Use `single database (Y)` or a multiple database (`N`).

If you choose to use a single database, at the `Combined Database: /PAQ2/` prompt specify a database that is specially developed for the Pourbaix module's diagram calculations.

If you choose to use a multiple database, then you are prompted to specify an aqueous solution database and to append any additional required data and select necessary phases, such as various interacting alloy (solid) solution phases, and secondary (solid and liquid) phases, for example.

7. Specify all your solutes and the molality of each solute when prompted to do so. This can be done in terms of elements (such as Fe, Ni, Na, Cl, C or S) or arbitrary complex species (such as NaCl, CO<sub>2</sub>, CoCl<sub>3</sub>, Fe<sub>0.93</sub>S, NaSO<sub>4</sub>-1, or H<sub>2</sub>SO<sub>4</sub>). You can enter the element/species and the molarity on one line (for an example, NaCl 2.5).

The definition of your system and its initial bulk composition is now presented.



The first letter of the element abbreviation must always be upper case. Any second letters must be lower case. When you have finished your specification, press `<Enter>` when asked to specify `Next solute`.

8. Type `Y` to confirm the defined system and its bulk composition.

The thermodynamic data for your system is now retrieved from the database(s) chosen.

9. Answer the questions that you are prompted regarding various parameters of your calculation. You are asked to do any of the following:

- `Rejectphase(s)` OR `Restore phases(s)`
- Check for miscibility gaps on any phase (Should any phase have a miscibility gap check?)
- Enforce the Global Minimization Technique in calculations?
- Save all functions, variables and tables in POLY3 file?
- Set numerical limits on the calculation
- Confirm defined conditions?
- Accept the default settings for two mapping variables

- 
10. The Pourbaix calculation starts. Answer the question `Any missing phase boundary you could possibly think of?` Type `Y` or `N` to set whether you want to add starting points to the calculation. If you suspect that the calculation has missed some phase boundaries, this may help the program to detect them.
  11. Answer questions about how the diagram is plotted and how to save the results of your calculation. You are asked to do any of the following:
    - `Change the pH/Eh steps for smoother curves?`
    - `Zoom in on a specific region of the diagram`
    - `Change the curve-label option for the diagram`
    - `Add label-texts onto the Pourbaix diagram`
    - `Change the subtitle of the diagram`
    - `Change the axis-texts`
    - `Further refine the diagram in the POST Module`
    - **Print the diagram (save a Hard copy of the diagram)**
    - `Save the X-Y coordinates of curve on a text file`
    - `Modify the diagram`
    - **Calculate and plot another Pourbaix diagram (Any more diagram?)**

## Plotting More Pourbaix Diagrams

When you have calculated a Pourbaix diagram, you can plot additional diagrams based on the results of the same calculation, but with different axis variables than Ph and Eh.

The variables can be plotted along the X- and Y-axes. Use the abbreviations of the variables when you specify what the X- and Y-axes represent in a diagram. In the table, *AQsp* refers to the name of a specific aqueous species. The asterisk (\*) can be used as a wild card character for all such species. If neither *AQsp* nor \* is entered in parenthesis, then all species are searched.

You must have plotted a Pourbaix diagram to plot a diagram with other variables than Ph and Eh on the Y-axis and the X-axis. You can either plot such additional diagrams directly after having finished the plotting and saving/printing of a Pourbaix Ph-Eh diagram or by choosing option 2 (Open an old file and plot other property diagrams) when you enter the POURBAIX module and opening a POLY3 workspace file.

### Abbreviations for Aqueous Solution Variables

Abbreviation	Variable
pH	Acidity
Eh	Electronic Potential (V)
Ah	Electronic Affinity (kJ)
Pe	Electronic Activity (log <sub>10</sub> ACRe)
IS	Ionic Strength
TM	Total Concentration
Aw	Activity of Water
Oc	Osmotic Coefficient
MF(AQsp)	Mole Fractions of Aqueous Species
ML(AQsp)	Molalities of Aqueous Species
AI(AQsp)	Activities of Aqueous Species

---

Abbreviation	Variable
RC(AQsp)	Activity Coefficients of Aqueous Species

### ***How to Plot an Aqueous Solution Diagram***

1. Type `Y` when prompted whether to plot any more diagrams, or select option 2 when you enter the POURBAIX module.
2. Specify which variable you want the X-axis to represent. Use the abbreviation specified in the preceding table. When you specify the name of an aqueous species, you can use the asterisk (\*) to select all species.
3. Specify which variable you want the Y-axis to represent. The diagram is then automatically plotted.
4. Answer questions about how the diagram is plotted and how to save it. You are asked to do any of the following:
  - a. change the axis-type (linear, logarithmic or inverse) on the X-axis or Y-axis
  - b. zoom in on a specific region of the diagram
  - c. change the curve-label option for the diagram
  - d. add label-texts onto the Pourbaix diagram
  - e. change the subtitle of the diagram
  - f. change the axis-texts
  - g. refine the diagram in the POST Module
  - h. print the diagram (save a hard copy)
  - i. save the X-Y coordinates of curve on a text file
  - j. modify the plotted diagram
  - k. calculate and plot another aqueous solution diagram

## Stepping Calculations in an Aqueous Solution



Performing the stepping calculation overwrites all the previous settings and stepping results on the POLY3-file. To avoid this, use [SAVE\\_WORKSPACES](#) to create a new workspace.

When you have calculated a Pourbaix diagram, you can use the saved results of the calculation (the POLY3 file), and make a property diagram based on stepping calculation.

Since the program normally sets the calculated initial equilibrium point as the starting point, the minimum value for the stepping variable must be specified as smaller or equal to the specific initial condition, and the maximum value as larger or equal to that condition. If pH is set as the stepping variable, the program automatically calculates the equilibrium constant in logarithmic ( $\log_{10}K_a$ ) of the electrolysis reaction  $\text{H}_2\text{O (water)} = \text{H}^+ + \text{OH}^-$  for the real aqueous solution under the current temperature-pressure-composition conditions, and thus this constant  $\log_{10}K_a$  value is considered as the highest pH in specifying the maximum value along stepping. Subsequently, confirm or modify an increment factor with a default value; for pressure and mole of a component, specify the step in logarithmic ( $\log_{10}P$  or  $\log_{10}N$ ).

### State Variable Abbreviations

You can use any of these state variables as a stepping variable.

State Variable	Abbreviation
Temperature (K)	T
Pressure (bar)	P
Acidity	pH
Electronic potential (V)	Eh
Mole number of a dissolving component (such as Fe, Na, Cl, or NaCl) in 1 kg of water.	N(Comp) (for example, $N(\text{Fe})$ )

## How to Do a Stepping Calculation

You must have plotted a Pourbaix diagram before making a stepping calculation on an aqueous solution.

1. Use GOTO\_MODULE POURBAIX to enter the POURBAIX module.
2. Type 4 to open an old file and make another STEPPING calculation.
3. Choose which POLY3 file to load.
4. Specify the conditions of the calculation by answering the questions that follows.

The system definition is the same as in the previous Pourbaix calculation, but you can modify conditions such as bulk composition, temperature and pressure, and change some component definitions if necessary.

5. Specify which variable to use as the stepping variable (see the preceding table), its minimum and maximum value, and the length of each incremental step.

The settings and stepping results are written to the opened POLY3 file, and a first default property diagram is plotted.

## POURBAIX\_DIAGRAM Module



This command or module is not available with GES6.



GES6 is a re-write and new implementation of the Gibbs Energy System module of Thermo-Calc and it is the default engine. The previous version, GES5, is still available and will continue to work in the background of the program where necessary. The main purpose of GES6 is to support faster development of new features. However, not all GES5 functionality is implemented in GES6. In such cases (except for the POURBAIX module), GES6 falls back silently and uses functionality from GES5 in the background.

### Getting started with the Pourbaix module

After you enter the POURBAIX\_DIAGRAM module, you are asked if you want to have step-by-step instructions to help walk you through the module.

```
Need any instruction on the Pourbaix module? /N/: Y
```

---

If you answer  $\forall$  a series of notes are displayed. The following is a more comprehensive summary of these same notes.

### WHAT IS A POURBAIX DIAGRAM?

The Pourbaix diagram is a calculated equilibrium phase diagram mapped and plotted with the independently-varied electropotential (Eh, as defined with regard to the standard hydrogen electrode as its reference) and acidity (pH), that represents all the equilibrated phase boundaries among aqueous solution, gaseous mixture, and various primary and secondary solids (modeled as either complex solution or simple stoichiometric phases) in a certain multicomponent heterogeneous interaction system, under a defined T-P condition and a specific initial bulk composition (which is, by default, always set as 1 kg of water dissolving a specified amount of metals/alloys and other acids/alkalines/salts).

### WHAT DATABASE SHOULD I USE?

At least one database is required that contains an aqueous solution phase (with thermodynamic data for water and various aqueous solutes), that shall be selected from TCAQ (PAQ) or AQS (PAQS) [which use the SIT Model or the Complete Revised HKF Model, respectively] within the Thermo-Calc databases.

### CHOOSING A CUSTOM AQUEOUS SOLUTION DATABASE

You can also use a custom database but it must be in the Thermo-Calc TDB format. Due to the restrictions of aqueous solution models used within Thermo-Calc, such a database must be designed in the same format as in the default TCAQ (PAQ2) for aqueous solution phases. Among other things, keep the following requirements in mind:

The ELECTRON is defined as a special element (ZE) and as the only constituent in its reference phase REFERENCE\_ELECTRODE (for determining the electro-potential that is defined as Eh with the standard hydrogen electrode as the reference), but it is not defined as an aqueous species;

The vacancy (VA) is unnecessary for AQUEOUS solution phase and it should be avoided in the definition of phase-constituents in the AQUEOUS phase;

The AQUEOUS solution phase should always be defined as a constitutional solution phase, implying that all the aqueous solution species must be included in a single site, rather than in two or multiple sublattices.

Beside the AQUEOUS solution phase, there shall exist a GAS mixture phase containing at least H<sub>2</sub>O, O<sub>2</sub> and H<sub>2</sub>; and for multicomponent systems, normally there shall also contain some solid (stoichiometric or solution) phases. Of course, if desired, you could also choose

to calculate and generate a Pourbaix diagram without considering the GAS mixture phase entirely; however, such a plot is not really a complete Pourbaix diagram, due to the fact that thermodynamically-stable phase boundaries between the AQUEOUS solution phase and GAS mixture phase will then not be calculated at all!

All the required thermodynamic data for calculations of Pourbaix diagrams or other diagrams must be retrieved either from one (Single) database which consists of an AQUEOUS solution phase, a GASEous mixture phase, a REF\_ELECTRODE phase, and some SOLID phases (being solutions and/or stoichiometric compounds; for primary metals/alloys and for secondary products formed from heterogeneous chemical/electrochemical interactions, or from several (Multiple) databases that respectively contain various solutions/compounds (as listed above).

Such databases suitable for calculations of aqueous-bearing heterogeneous interaction systems can be those default-prompted ones [i.e., in the Single-Database case, the PAQ or PAQS; and in the Multiple-Database case, the PAQ2 or as primarily-switched database, plus the SSUBx as firstly-appended one and the SSOLx as secondly-appended one if it is necessary; even more databases can be appended). Of course, you could also choose to append required data from other appropriate databases (such as TCFE, TCSLD, TCNI/TTNi, TCAL/TTAI, TCMG/TTMg, TTZr, TCMP, SLAG, etc.) for GASEous mixture phase and for various solid solution and stoichiometric compound phases. Furthermore, an experienced user can also utilize his/her own USER-specified databases in various cases.

The advanced POURBAIX module is designed so that it requires you to just answer some necessary questions, rather than to go through the other Console Mode modules (i.e., TDB, GES5, POLY3 and POST) although you still enter commands at the prompts. Beside the default plotted Pourbaix diagram, it also allows the user to easily and quickly plot many different properties of the system, stable phases and aqueous species, varied along the calculated phase boundaries for the same defined heterogeneous interaction system. Moreover, it permits the user to directly change some plotting settings and manipulate all kinds of plotted diagrams.

The current advanced POURBAIX-Module has been extended so that it is additionally able to directly perform a normal STEPPING calculation (varied with a specified independent variable) and to easily generate various types of property diagrams, for the same heterogeneous interaction system that has been defined in a previous POURBAIX or TDB-GES5-POLY3-POST calculation.

### **COPY AND RENAME THE POLY3 FILE**

At the end of a Pourbaix module calculation, a new POLY3 file is automatically saved.



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.



If you are using a POLY3 file more than once (as is done in the extended examples), it is recommended that you copy and rename the previously-generated POLY3 file. This is so that the required POLY3 file structure and Pourbaix results are not lost. Do this outside of Thermo-Calc i.e. do not save from within Thermo-Calc.



The reason you do not save from within Thermo-Calc is because of how the file is saved. When the command sequence (GO POLY3>SAVE <FILE NAME>.POLY3>Y) is used, the POLY3 workspace of the POURBAIX calculation results in the POLY3 module monitor and all the initial equilibrium points and the mapping/stepping calculation results are lost as well as parts of the POLY3 file structure (e.g., some definitions of the previously defined POURBAIX-type calculation system).

## ENFORCE A PAUSE

After the initial set of information is displayed, you are asked:

```
Enforce a PAUSE after plotting when running a MACRO? /N/
```

- If you choose **Y** a pause is always enforced after each diagram is plotted;
- If you choose **N**, all the command-lines (answers) for the session continue without stopping or until the SET\_INTERACTIVE\_MODE line in the macro file is reached.

Whenever running a Thermo-Calc macro (TCM) file, you may want to pause after a specific diagram is plotted on screen. This gives you the flexibility to:

- Print it (of EMF/PS format) to a printer
- Convert it (of EMF format) to (PDF) graphical files;
- Save it as an EMF graphical file;
- Dump it as a PNG or BMP graphical file;
- Set background colour for the current diagram
- You can also set the default font/size and change plot layers for sequential plots.

## *Pourbaix Main Options*

If you answer `y` to the Pause prompt above, these options are displayed:

1. Start a completely new Pourbaix diagram calculation.
2. Open an old file and plot other property diagrams.
3. Open an old file and make another Pourbaix calculation.
4. Open an old file and make another stepping calculation.



[POURBAIX\\_DIAGRAM Commands for Options 1 to 4](#)

---

## Tabulation Reactions

---

**In this section:**

---

About the Tabulation Reaction Module .....	121
Tabulating a Reaction .....	123
Tabulating a Substance or Solution Phase at Fixed Composition .....	124

---

## About the Tabulation Reaction Module

The TABULATION\_REACTION (TAB) module can be used to tabulate thermodynamic functions of any type of substance, stoichiometric phase or solution phase, or reaction. The module can tabulate thermodynamic functions of pure stoichiometric phases or solution phases at fixed composition, as well as various kinds of partial derivatives for a pure stoichiometric phase or solution phase at a given composition.

The TAB module begins by evaluating the most stable species of a substance in either a homogeneous state or in a reaction. Results are presented in a table or plotted as a graph.

### [Tabulating a Reaction](#)

A reaction tabulation presents you with the following properties, on rows representing different temperature levels:

- Heat capacity change / Delta-Cp (Joule/K) in column 2.
- Enthalpy change / Delta-H (Joule) in column 3.
- Entropy change / Delta-S (Joule/K) in column 4.
- Gibbs energy change / Delta-G (Joule) in column 5.

In the case of a tabulation of a substance or solution phase at a fixed composition, the properties presented are the heat capacity, enthalpy, entropy and Gibbs energy, rather than the degree of change in those properties.

### [Tabulating a Substance or Solution Phase at Fixed Composition](#)

You can add a 6th column to the table that shows the values of a property or user-defined function of your choice.

```
*****
  T      Delta-Cp      Delta-H      Delta-S      Delta-G
  (K)    (Joule/K)     (Joule)     (Joule/K)   (Joule)
*****
298.15  -4.44006E+01  -9.18800E+04  -1.98115E+02  -3.28120E+04
300.00  -4.43267E+01  -9.19621E+04  -1.98389E+02  -3.24452E+04
400.00  -3.92294E+01  -9.61533E+04  -2.10482E+02  -1.19604E+04
500.00  -3.34122E+01  -9.97861E+04  -2.18613E+02  9.52022E+03
600.00  -2.77768E+01  -1.02842E+05  -2.24200E+02  3.16779E+04
700.00  -2.26324E+01  -1.05358E+05  -2.28088E+02  5.43040E+04
800.00  -1.81080E+01  -1.07390E+05  -2.30808E+02  7.72568E+04
900.00  -1.41889E+01  -1.09000E+05  -2.32710E+02  1.00438E+05
1000.00 -1.08095E+01  -1.10245E+05  -2.34025E+02  1.23779E+05
1100.00 -7.77802E+00  -1.11169E+05  -2.34908E+02  1.47229E+05
1200.00 -5.07556E+00  -1.11807E+05  -2.35464E+02  1.70750E+05
1300.00 -2.93467E+00  -1.12203E+05  -2.35782E+02  1.94314E+05
1400.00 -1.19414E+00  -1.12407E+05  -2.35934E+02  2.17901E+05
1500.00 2.55400E-01  -1.12452E+05  -2.35966E+02  2.41497E+05
1600.00 1.49022E+00  -1.12363E+05  -2.35909E+02  2.65091E+05
1700.00 2.56484E+00  -1.12159E+05  -2.35785E+02  2.88676E+05
1800.00 3.51909E+00  -1.11854E+05  -2.35611E+02  3.12246E+05
1900.00 4.38259E+00  -1.11458E+05  -2.35397E+02  3.35797E+05
2000.00 5.17775E+00  -1.10980E+05  -2.35152E+02  3.59325E+05
```

## Tabulating a Reaction

A tabulation of a given chemical reaction provides you with data about the rate of change of the standard tabulation properties at various levels of temperature. You can also add a column for a property of your choice to the table.

1. Use `GOTO_MODULE TAB` to enter the TAB module.
2. If you want to add one thermodynamic functions to the tabulation calculation, which is present in a sixth column in the table, use `ENTER_FUNCTION`.

You are prompted to enter the name of the column that is given at its head, and the function itself. The following state variables can be used to define the function: G, H, S, T, P, V and H298.

3. Use `TABULATE_REACTION` followed by a formula that specifies of the reacting species and products to perform the tabulation calculation. The name of the elements must be entered in upper case letters only. Terminate the chemical reaction with a semi-colon. For example, you can enter:

```
TABULATE_REACTION 3H2+N2=2N1H3;
```

Elements that are designated with a single letter must be followed by a stoichiometry factor, even if this factor is 1. `CO` hence be interpreted as Cobalt rather than as carbon monoxide. Carbon monoxide is entered as `C1O1` or `O1C1`.

4. Set the pressure, the range of temperature that you want the tabulation to cover, and change in temperature between each row in the table. You can also specify if you want to output the calculation results to an EXP-file and whether you want any graphical output, and if so, which column you want to be plotted against temperature.

The tabulation calculation is performed and you are given a table that shows the various properties at each temperature level within the range defined.

If you have chosen to receive graphical output, then the Console Results window presents a graph that plots the column of you chose (on the Y-axis) against temperature (on the X-axis). When this graph is plotted, you are now in the POST module, where you can modify the graph in various ways.

## Tabulating a Substance or Solution Phase at Fixed Composition

A tabulation of a substance or solution phase at fixed composition provides you with data about the standard tabulation properties at various levels of temperature for a given chemical reaction. You can also add one extra property column of your choice to the table.



It is important to have already defined the system and retrieved thermodynamic data from a database before using the TAB command.

1. Use `GOTO_MODULE TAB` to enter the TAB module.
2. Use `SWITCH_DATABASE` to switch to a database that contains solution-based data, such as TCFE for example.



You cannot use the default SSUB database when tabulating solution phases at a fixed composition.

3. If you want to add one thermodynamic function to the tabulation calculation, which is present in a sixth column in the table, use `ENTER_FUNCTION`.

You are prompted to specify the column header and the function itself. The following state variables can be used to define the function: G, H, S, T, P, V and H298.

4. Use `TABULATE_SUBSTANCE` followed by the name of the substance or solution phase to perform the tabulation calculation.
5. Set the pressure, the range of temperature that you want the tabulation to cover, and change in temperature between each row in the table. You can also specify if you want to output the calculation results to an EXP-file and whether you want any graphical output, and if so, which column you want to be plotted against temperature.

The tabulation calculation is performed and a table shows the various properties at each temperature level within the defined range.

If you have chosen to receive graphical output, then the Console Results window presents a graph that plots the column you chose (on the Y-axis) against temperature (on the X-axis). ). When this graph is plotted, you end up in the POST module, where you can modify the graph in other ways.

---

# Results

---

**In this section:**

---

Console Mode Results .....	127
Using the POST Module .....	129
Plotting Diagrams .....	130
Saving and Printing the Output in Console Mode .....	131
Modifying Diagrams .....	134
Loading Saved Diagrams .....	136
Working with the Console Mode Output .....	137

---

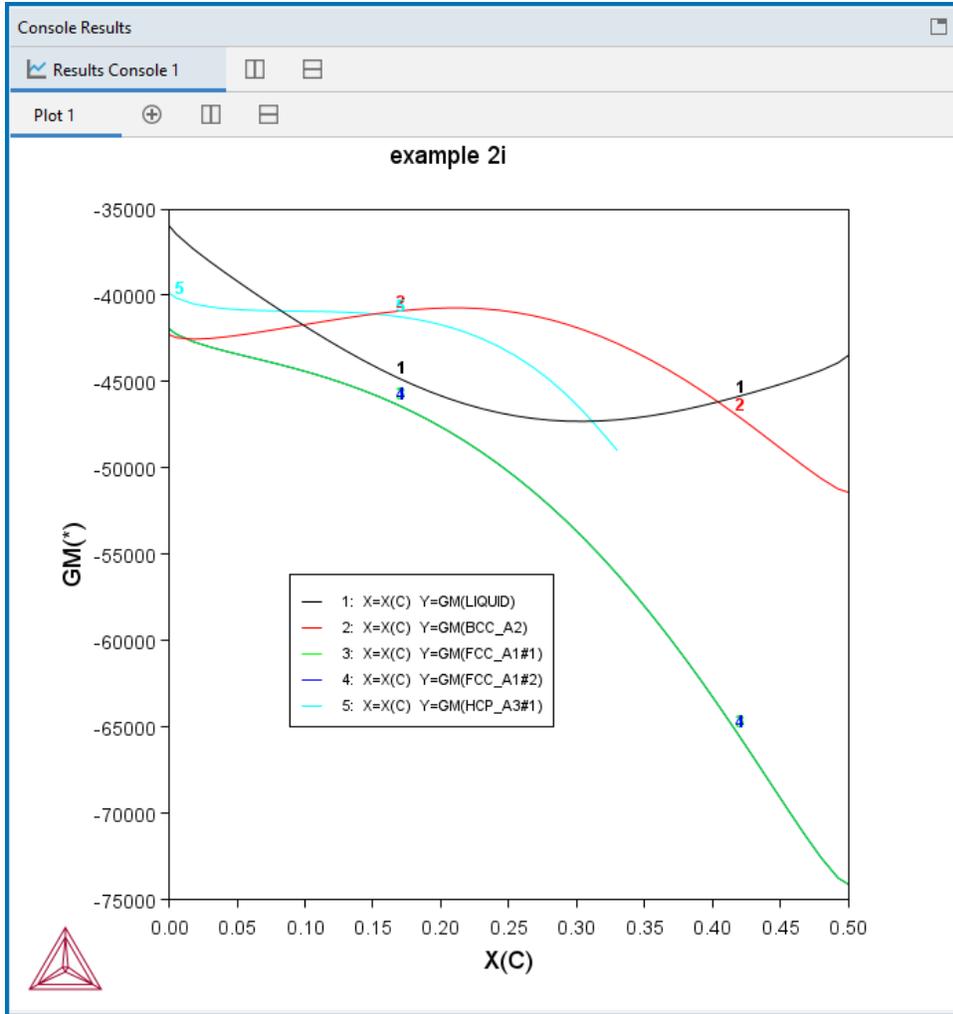
## Console Mode Results

---

The results of a calculation in Thermo-Calc can be plotted by the post-processor module, also called the POST module. The diagram is displayed in the **Console Results** window.

In the POST module, you can define and plot various kinds of diagrams based on a calculation. You can modify diagram type, which variables to plot on each axis and between which minimum and maximum values, append experimental data from an EXP-file, change labels and symbols in the diagram, zoom in on (and out from) a certain region in a diagram, etc.

If you calculate a property or phase diagram in POLY, then you have to open the POST module yourself. If your calculation was performed by a response-driven module such as SCHEIL or POURBAIX, then the plotting of a diagram based your calculation is part of the dialogue of questions and answers you have in that module (but under the hood, the response-driven module is actually utilizing the POST module to get the plotting done).



Example of output to the Console Results window.

## Using the POST Module

---

Use the POST module after a [STEP\\_WITH\\_OPTIONS](#) or [MAP](#) calculation to generate *graphical* or *tabulated* presentations on screen or into files.

The graphical files are available in different formats (e.g. PS, EMF, PNG, BMP, PDF, JPG, TIF) or data forms (e.g. EXP, TAB and WRL). The graphical output can be phase diagrams, property diagrams, diffusion profiles and other types of plots. The tabulated presentation is into a basic text file (TXT) or a spreadsheet (e.g. an MS Excel file with an extension of .XLS).



The POST module (post-processor) is a submodule to the POLY module and has its own set of commands. The TAB, BIN, TERN, POT, SCHEIL and POURBAIX modules use the POST module features as part of the automatic generation of the plots.

Within the POST module, you can choose any state variable, any derived variable, or any entered symbol (functions or variable) as the X/Y-axis.

When a diagram is plotted, appearance parameters for defining a high-standard graph can be further specified, e.g. curve labelling options, diagram titles and subtitles, plot size, axis length, axis types, axis-tic type, tie-line status, automatic or manual scaling and zooming, semi-automatic or manual labelling on phase boundaries and phase regions, graphic formats, text fonts, colours, raster plot, etc.

You can append experimental data onto a plotted diagram. You can also save the coordinates onto a textual file, which can be edited and used as an experimental data file to be merged onto another diagram or as a part of a setup file for PARROT-module assessment.

Phases can be suspended or restored in a plotted diagram. The reference states for components can also be modified for the resulted diagrams. Moreover, you can translate a plotted property diagram (after a stepping calculation) into a tabulated form, such as a simple list on screen or a textual file, and a spreadsheet .

## Plotting Diagrams

---

Plotting a diagram is to graphically represent how several variables (typically two variables) co-vary. Each axis in a property diagram or a phase diagram represents the value of one variable. The default variables represented by the X- and Y-axes are the following:

- If you plot a diagram based on the results of a mapping operation (a phase diagram), then by default the X- and Y-axes represent the mapping axis variables 1 and 2 (set using [SET\\_AXIS\\_VARIABLE](#) in POLY).
- If you plot a diagram based on the results of a stepping operation (a property diagram), then by default, the X-axis represents the stepping variable (set using [SET\\_AXIS\\_VARIABLE](#)) and the Y-axis represents the sum of the mole fractions of all phases (that is, NP(\*)).



[POLY Module Commands](#)

### **How to Plot a Diagram**

To plot a diagram, you must have already made a stepping or mapping operation in POLY (or in some cases, made a calculation as part of some opening a response-driven module), and you must have entered the POST module.

The POST command is available inside POLY. Use POST to enter the POST module

1. If you want to directly plot your diagram with the default variables represented by the X- and Y-axes, type `PLOT` and press <Enter>. This plots a property or phase diagram with the default variables represented at each axis. If you do not want to plot your diagram with these default variables, go to the next step.
2. Use `SET_DIAGRAM_AXIS` to set which variables to plot on the X-axis and the Y-axis. For example, if you want to plot a phase diagram with pressure on the Y-axis and temperature on the X-axis, type:

```
SET_DIAGRAM_AXIS X T
```

```
SET_DIAGRAM_AXIS Y P
```

3. Use `PLOT_DIAGRAM` to plot your diagram.

---

## Saving and Printing the Output in Console Mode

---



### [Global Settings: Graphical and Console Mode Plotting](#)



Output can refer to results in the form of tabulated data in a table or a diagram.



The terms *diagram* and *plot* are interchangeable. The commands in Console Mode generally use the term diagram and in Graphical Mode it is also called a plot (e.g. the Plot Renderer).

## Commands to Save and Print Diagrams

Use the following commands to create diagrams, which then display in the **Console Results** window. From this window, you can right-click the diagram and choose **Save as** or **Print**. You can also use the following commands for these actions.

### ***PLOT\_DIAGRAM***

After you use the SET\_PLOT\_FORMAT command to define the default graphic device (the printer) you can use a [PLOT\\_DIAGRAM](#) command to either save the calculation output to a file or display the output in the **Console Results** window.

### ***DUMP\_DIAGRAM***

[DUMP\\_DIAGRAM](#) is an alternative command used to create output. After a diagram is displayed to the **Console Results** window with the PLOT\_DIAGRAM command, you can save (dump) it to a file using this command.

### ***MAKE\_EXPERIMENTAL\_DATAFILE***

Use [MAKE\\_EXPERIMENTAL\\_DATAFILE](#) to save all the data about the plotted diagram in an EXP-file. An EXP-file is a plain text file that describes the diagram in the DATAPLOT graphical language. Note that information about the underlying calculations that the diagram is based on is not saved in an EXP-file.



*DATAPLOT User Guide* included with this documentation set.



See the [Tips and Tricks: How to Plot Experimental Data](#) blog post on our website for useful guidance about this topic.

## **PRINT\_DIAGRAM**

First use PLOT\_DIAGRAM to create output that displays in the **Console Results** window. Then after entering the [PRINT\\_DIAGRAM](#) command, the **Print** window opens where you can choose additional settings for a hard copy of the diagram.

## **Commands to Save Tables**

Tables or tabulated data can be listed in the **Console** window. You then use commands to save the text to different file formats (txt, html, or xls) where you can then print the information.

- [LIST\\_DATA\\_TABLE](#): Use this command to output to the screen or to file.
- [TABULATE](#)

## **Saving Diagrams**

You can save the output in a variety of file formats. For JPG and PNG images you can also adjust the image quality.



In Console Mode, the diagram is output to the **Console Results** window after using the PLOT\_DIAGRAM command.



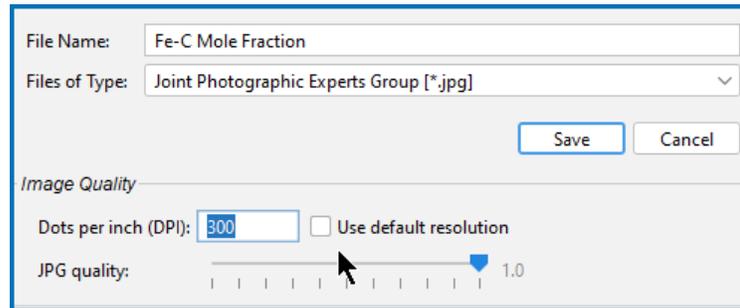
In Graphical Mode, the diagram is output to the **Visualizations** window after performing a tree that includes a Plot Renderer.

1. In the **Visualizations** window, right-click the diagram and select **Save as**.



In the Graphical Mode **Configuration** window you can also click the **Save Diagram**  (or **Save table** ) buttons to open the **Save** window.

2. In the **Save** window:
  - a. Navigate to where you want to save the diagram.
  - b. Enter a **File name**.
  - c. From the **Files of type** list choose **png** (the default), **jpg**, **ps**, **pdf**, **gif**, **svg**, or **emf**.
  - d. For PNG and JPG files only, under the **Image Quality** section, choose to improve the resolution of the image. For JPG files you can also adjust the file size.
    - To change the resolution of a PNG or JPG image, click to clear the **Use screen resolution** checkbox. Then enter a numerical value in the **Dots per inch (DPI)** field.
    - For JPEG files, you can also use the slider to choose a number between 0 and 1 for the **JPG quality**. The default is 0.9. The lower the number, the smaller the file.



- e. When you are ready, click **Save**.

## Printing Diagrams

In the **Console Results** window, right-click the diagram and choose **Print**. Select additional settings as required on the **Print** window.



Use the [LIST\\_DATA\\_TABLE](#) command to save to file and then print the file.

## Modifying Diagrams

---

Modifying a diagram amounts to changing the settings that determine how the POST module presents the calculated results and plots a new diagram. Suppose that you have already plotted a binary phase diagram with the mapping axis variables temperature and mole fraction of one of your components (the Fe component, say) on the X- and Y-axes. If you then want to modify your diagram so that the Y-axis instead represents, say, the mass fraction of Fe rather than the mole fraction, then you use [SET\\_DIAGRAM\\_AXIS](#) to set diagram axis  $Y$  to  $w(\text{Fe})$ , and then use [PLOT\\_DIAGRAM](#).

There are many ways in which you can modify your diagram and how it is presented. Any modification made in the plot settings is taken into account the next time you use PLOT.

### Example Commands

The following are some of the modifications you can make. Use:

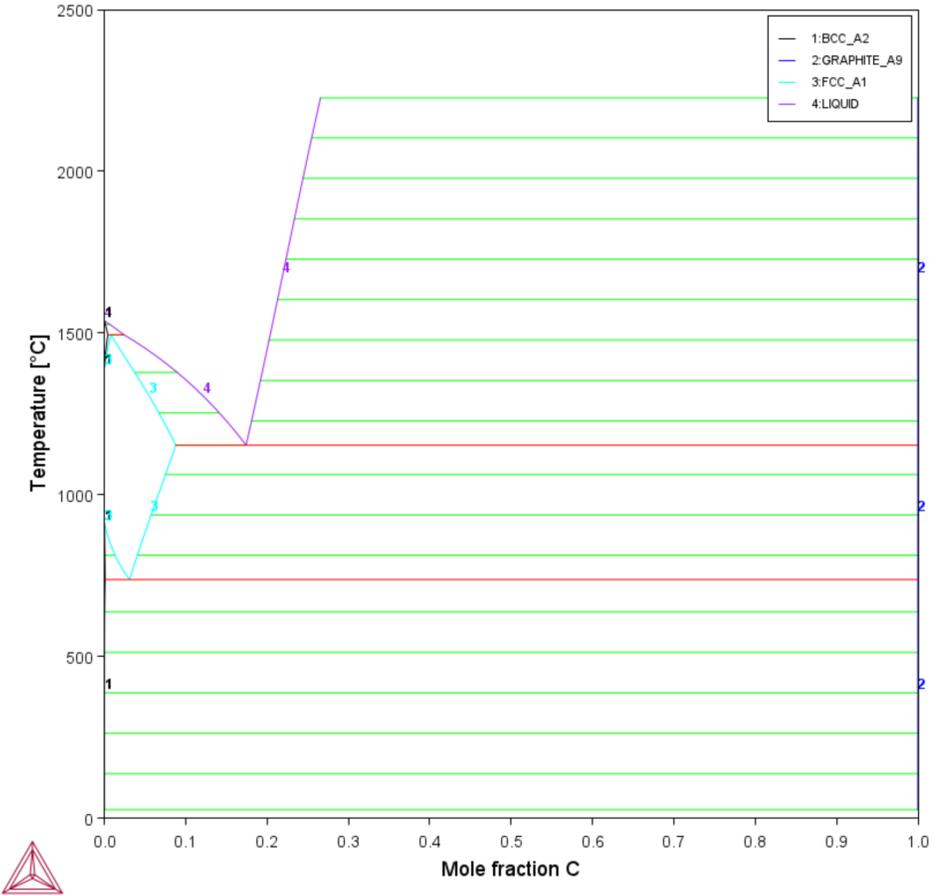
- [SET\\_TITLE](#) to set add a title or to change an already existing title. The title is displayed above the plotted diagram in the Console Results window.
- [ADD\\_LABEL\\_TEXT](#) to add a label text that starts at a certain X- and Y-coordinate. If you have plotted a phase diagram, then you can let Thermo-Calc set the label text that specifies the names of the stable phases at that coordinate.
- [SET\\_AXIS\\_TYPE](#) to set whether the scale of an axis is linear, logarithmic or inverse.
- [SET\\_DIAGRAM\\_TYPE](#) to plot a triangular diagram with the X-axis on the triangle's base and the Y-axis on the triangle's left side. Such a diagram is often useful if you want to plot the fractions of two components in a ternary diagram.
- [SET\\_SCALING\\_STATUS](#) to modify the range of values that are displayed on an axis. In effect, you can use it to zoom in or out of the diagram. For example, to set the scale of the X-axis to range from 0 to 0.3, type:

```
SET_SCALING_STATUS X N 0 0.3
```

The  $X$  specifies which axis to change the scaling on, the  $N$  turns off Thermo-Calc's automatic scaling, and  $0$  and  $0.3$  specifies the minimum and maximum values on the scale. You can turn on automatic scaling again by setting the second parameter to  $Y$ .

- [SET\\_LABEL\\_CURVE\\_OPTION](#) to label each set of curves that have the same stable phases with a certain digit. A legend is also added, where you specify which phases each digit designates.

C FE



---

## Loading Saved Diagrams

---

If you have previously saved a diagram in an EXP-file, then you can load and superimpose the diagram on another diagram. Both the following command requires you to specify which prologue and dataset(s) that are loaded from an experimental data file.

Use [APPEND EXPERIMENTAL DATA](#) to plot selected data from an experimental data file (EXP-file). If you have set the axes of the diagram already, the labels on the diagram are not changed when you plot the additional data from the EXP-file, irrespectively of the data in that file.

Use [QUICK EXPERIMENTAL PLOT](#) to plot selected data from an experimental data file (EXP-file). If you have set the axes of the diagram already, these settings are overwritten with the axes settings from the EXP-file.

---

## Working with the Console Mode Output

---

### Pause, Resume or Stop Output to the Console Window

In response to a command or a lengthy calculation, Thermo-Calc may display a lot of text in the Console window. Use these keyboard shortcuts to pause, resume, or stop a process.

- To pause the text on printing press <Ctrl+S>.
- To resume the printing on screen, press <Ctrl+Q>.
- When Thermo-Calc is performing a stepping or mapping operation, the results are continuously printed in the Console window. To stop the calculation of the current region of the mapping and stop the output, press <Ctrl-A> (Windows) or <Ctrl-C> (Linux).

### Controlling Console Output

In response to a command or a lengthy calculation, Thermo-Calc may display a lot of text in the Console window. Use these keyboard shortcuts to pause, resume or stop a process.

- To pause the text on printing press <Ctrl+S>.
- To resume the printing on screen, press <Ctrl+Q>.
- When Thermo-Calc is performing a stepping or mapping operation, the results are continuously printed in the Console window. To terminate the calculation of the current region of the mapping and stop the output, press <Ctrl-A> (Windows) or <Ctrl-C> (Linux).

In Console Mode, sometimes the output overflows the window text buffer. This is how to increase the buffer size.

1. In the Console window right-click the tab to edit. For example, if this is the first tab, it is labeled **Console 1**.
2. Select **Properties** and either enter a number in the field or use the arrows to increase or decrease the **Buffer size**.

---

## Commands by Module

---

**In this section:**

---

System Utilities (SYS) Commands .....	139
Database (TDB) Module Commands .....	163
POLY Module Commands .....	177
Gibbs Energy System (GES) Commands .....	280
SCHEIL Module Commands .....	338
PARROT Module Commands .....	355
EDIT_EXPERIMENTS (ED_EXP) Submodule Commands .....	401
Tabulation Reaction (TAB) Commands .....	458
REACTOR (REACTOR_SIMULATOR) Commands .....	484
POST Module Commands .....	495

---

## System Utilities (SYS) Commands

---

### In this section:

---

ABOUT .....	140
BACK .....	141
CLOSE_FILE .....	142
DISPLAY_LICENSE_INFO .....	143
EXIT .....	144
GOTO_MODULE .....	145
HELP (? and ??) .....	146
HELP .....	147
HP_CALCULATOR .....	148
INFORMATION .....	149
MACRO_FILE_OPEN .....	150
OPEN_FILE .....	153
SET_COMMAND_UNITS .....	154
SET_ECHO .....	155
SET_GES_VERSION .....	156
SET_INTERACTIVE .....	157
SET_INTERACTIVE_MODE .....	158
SET_LOG_FILE .....	159
SET_PLOT_ENVIRONMENT .....	160
SET_TC_OPTIONS .....	161
STOP_ON_ERROR .....	162

## **ABOUT**

The ABOUT command lists basic information about the development history and ownership of the Thermo-Calc software as well as version information.

## **BACK**

 [GOTO\\_MODULE.](#)

Use the BACK command to switch control back to the most recent module. Going from the POST module, BACK goes only to the TAB or POLY module (from where the POST module is entered).

---

## CLOSE\_FILE

Close an opened text file.

Syntax	CLOSE_FILE
Prompt	Unit number: <Unit number> A unit number given in an OPEN_FILE command must be specified.

## DISPLAY\_LICENSE\_INFO



You can also access this same information from the top menu. Choose **Help** → **Show License Info**.

Use this command to launch a license diagnostics program implemented in the SYS module. It detects the license information for the Thermo-Calc installations on the current client computer (and on the connected license server), and lists the details, including the current status of available license file and all its included license features for the current installation (s), as well as the Safenet Environment Variables and detailed Local (Client) Information, on screen or into a textual file (\*.TXT).

This is useful for a list of license details, and to obtain basic technical support by sending the \*.TXT file to [support@thermocalc.com](mailto:support@thermocalc.com). If you are running a network client installation of Thermo-Calc, you can see how many of the client example licenses that are currently checked out and how many of them are left. You can also see which clients computers it is that have checked these client example licenses out.

Syntax	DISPLAY_LICENSE_INFO
Prompt	Output to screen or file /Screen/: <Mylicenseinfo.txt> Press <Enter> to display a list of license information or into a *.TXT file under a specific file name which can later on be opened and edited by any basic text editor.

## **EXIT**

Use the EXIT command to leave the program and return to the operating system. Use a SAVE command before exiting otherwise all data and results are lost (in either the GIBBS, POLY, PARROT or REACTOR module).

## GOTO\_MODULE

Switch between modules. The name of the module must be typed. To get a list of available modules, press the <Enter> key.

Syntax	GOTO_MODULE
Prompt	Module name: <Module name>
	If the <Enter> key is pressed without typing a unique module name (or typing a ? mark) it prompts for the <code>MODULE NAME</code> and lists the available modules.

---

## HELP (? and ??)

Enter ? or ?? to get text help about the current prompt (whether this is the prompt of a module, or a prompt that requests you to specify the value of some parameter).



For some commands and prompts, more detailed help is given when two ?? are entered.

## HELP

Lists the available commands or explains a specified command. Specifying a unique command displays an explanation on screen. Typing a command abbreviation that is not unique lists all matching commands. Get the command information by typing a unique abbreviation or the complete command name.

Syntax	HELP
Prompt	<p>Command: &lt;Command name&gt;</p> <p><i>Command name</i> is the name of the command (one of the-module commands) to get help.</p> <p>If the &lt;Enter&gt; key is pressed without typing a command name, then all the available commands are listed.</p>

---

## HP\_CALCULATOR

The HP-CALCULATOR command is the QBA simple interactive calculator using reverse Polish notations.

Available OPCODEs (HPC codes) are listed by entering the command HELP.

Use the BACK command to quit and go back to the SYS module.

Input are numbers, + - \* / and ^ and OPCODEs. Several numbers and operations can be given on one line. The content of the X register is displayed after each operation.

For example, to compute  $2 * \text{EXP}(1.5^{**}3 - 30000 / (8 * 1273))$ , you should input the following:  
30000 8/1273/chs 1.5 3 ^ + exp 2 \*

---

## INFORMATION

Get basic information about topics related to the module you are in.

Syntax	INFORMATION
Prompt	Which subject /Purpose/: <Topic name> Specify a subject (or its abbreviation as long as it is unique). Enter a question mark ? for a list of topics.

## MACRO\_FILE\_OPEN



Some of the information in this topic is related to the add-on Diffusion Module (DICTRA).

The macro file functionality is a way to predefine sequences of Thermo-Calc commands stored in a macro file (a basic text file usually with the default extension \*.TCM or \*.DCM for DICTRA) and then executing all of the sequences using this command (preceded by the macro file name). This command can be used within various modules (i.e. SYS, POLY, PARROT, TAB and DICTRA\_Monitor).

Syntax	MACRO_FILE_OPEN
	<p>&lt;Name of a macro file *.tcm&gt;</p> <p>An <b>Open</b> window displays to specify the filename of the macro file (*.TCM) with the macro command, so that the path (in the <b>Look in</b> field) and <b>File name</b> can be specified.</p> <p>The <b>Files of type</b> *.TCM for Thermo-Calc (Console Mode) calculations and *.DCM for DICTRA simulations, cannot be changed. Click <b>Open</b> or <b>Cancel</b> to continue.</p> <p>A window such as <b>Save</b>, <b>Open</b>, or <b>Print</b> opens if the macro file contains commands to:</p> <ul style="list-style-type: none"> <li>• set log files (these have a file extension .TCM)</li> <li>• save or read GIBBS, POLY or PARROT workspaces</li> <li>• switch user databases</li> <li>• compile experiments (from existing *.POP files)</li> <li>• create new *.PAR files</li> <li>• append *.EXP files</li> <li>• plot diagrams, etc.</li> </ul> <p>If required, opening these windows can be avoided by issuing the file names (preferably with file-type extensions; if the files are not in the current working area where the macro is located, the appropriate and complete paths of the files should also be specified) and sometimes with the required options after the corresponding commands or parameters/options.</p> <p>For details, see the related commands and modules.</p> <p>When using a macro file that intended to plot graphs on screen, but the command SET_PLOT_FORMAT is used to alter the plotting environment from the default value, it is important to first use the command SET_PLOT_FORMAT again to change back to the default value.</p>

This is useful when the same or similar calculations are made frequently with small changes [in terms of system definitions, data manipulations, conditions (for single points, stepping or mapping calculations), plotting settings, etc.]. For example, use this feature when calculating phase/property diagrams during an assessment of thermodynamic data.

A macro file can be automatically generated in Console Mode if in the SYS module the SET\_LOG\_FILE command is used and a log file name (with a file extension \*.TCM) is given before any other SYS, DATA, TAB, GIBBS, POLY, POST, PARROT, or ED-EXP command or any special-module command (e.g. BIN, TERN, POTENTIAL, SCHEIL, POURBAIX, or REACTION). Such a log file generated in Console Mode is a text file. You can use a text editor (such as Notepad++) to remove unnecessary command lines, modify some commands, settings and definitions, add some pause points, add comment lines that start with @@, and so forth. Then save it as a macro file with the standard file extension TCM.

Experienced users can also write/edit an appropriate macro file for calculations/simulations, using any basic textual editor outside the Thermo-Calc program.

All commands can be used in a macro file.



The file can be terminated with EXIT or interrupted with SET\_INTERACTIVE.

Within a macro file you can use comment-lines (for describing the problems and for explaining commands and inputs/outputs), which should always start with the @@ signs in the beginning of each comment-line. Such comment-lines help to document the macro file, while these are not considered as command lines and thus do not affect the proceeding of all the normal commands in modules when the file is called by Thermo-Calc.

You can put multiple-line comment-blocks in a macro file between the @ (" and "@") signs. The former sign indicates where the comment begins and the latter sign indicates where the comment ends. A comment-block begins from a line started with the *begin comment* sign @ (and ends with the *finish comment* sign @); all the lines written in between are ignored, and the line started with @) is also ignored.

Another use for a macro file is to allow you to interact at some specific points, using the "@?" sign starting a text line (note that the text describing the expected user-specification/input should be written as a continuous string without any empty space), for user's on-time specifications of arguments/parameters or inputs of parameter-values that are requested by a certain command. The macro temporarily stops at the "@?" sign, prompt on screen the text given after "@?", and waits for the specified argument/parameter/value. Thermo-Calc then uses specified argument/parameter/value as the input(s) for the associated command. For example, you can input the values of lower and higher temperature limits for the second axis-variable as follows:

```
GO POLY-3
SET-AXIS-VAR 2 T @?Low-temperature-limit:
@?High-temperature-limit:
```

You can have macro-variables denoted by the signs of @#n (for definition) and ##n (for usage); and up to nine variables inside a single macro file. Such a macro-variable can be assigned with its desired value, as for example:

```
@#3First-element?
```

This writes the text (note that the text describing the expected user specification/input is written as a continuous string without any empty spaces) after the “@#3” sign as prompted on screen and then wait for a specification. The input is assigned to the macro-variable ##3, which is directly called in different parts within the current macro file.

For example, the content of the macro-variable ##3 is inserted in the command:

```
DEFINE-SYSTEM ##3
```

It is also useful in more complicated commands, such as:

```
SET AXIS VAR 1 x(##3) 0 1,,,
```

This command sets the mole fraction of macro variable 3 as axis 1.

A macro file can have any number of pauses at the @& signs, for the purpose of checking the details/results of executing certain commands when running the macro file. However, you can also prevent the software from temporarily stopping at any pause by typing any character (except for the Y character for confirming a Yes answer to a command prompt) after specifying the name of a macro file.

A macro file can have a maximum of five nested levels, i.e. a macro file can call another macro file, and if one sub-level macro is terminated by the command it resumes at the next command in the previous macro. If it is terminated by `end-of-file`, the Thermo-Calc (Console Mode) software stops.

This feature is especially useful for alloy design that requires many (hundreds) of calculations/simulations on similar material system/processes (specified in many different but appropriately-documented macro files that are organized in up to five levels) during a certain period of time (e.g. in an evening), you can run the main macro (on the top level) at a certain time (e.g. before leaving the office) and afterwards (e.g. next morning) you can systematically and efficiently check/compare/analyse the results (saved as graphical files, and/or EXP, TXT, XLS... files).

By adding the SYS command SET\_ECHO at the beginning of a macro file (or in the primary macro file on the top level if any sub-level(s) of macro files are used), it is useful to automatically display the complete/detailed meaning of various commands in all the sequential operations in the software, all enforced according to the macro file(s).

---

## OPEN\_FILE

A text file is opened for use in other commands where a unit number is necessary. The program automatically assigns a unit number.

Syntax	OPEN_FILE
Prompt	File name: <File name> A legal file name must be specified.

## SET\_COMMAND\_UNITS

This command is useful for reading inputs already prepared by a textual editor on a file. Such inputs can be a table of values or a large number of parameters. The file must be opened with `OPEN_FILE`.



The first two lines of an input file are skipped before any input is read.

Syntax	SET_COMMAND_UNIT
Prompt	<p>Input unit number /5/: &lt;Input unit number&gt;</p> <p>Specify the input unit number returned from an <code>OPEN_FILE</code> command. The next command is taken from the file connected to this unit number. The last command on such a file must be <code>EXIT</code> or <code>SET_INTERACTIVE</code> when you are in <code>POLY</code>, <code>POST</code> or <code>SYS</code> module, in order to go back to read input from the keyboard. The default value is the current input unit.</p>
	<p>Output unit number /6/: &lt;Output unit name&gt;</p> <p>Specify the output unit number returned from an <code>OPEN_FILE</code> command.</p>

---

## SET\_ECHO



This command is not applicable if macro files are not used.

Add the SET\_ECHO command at the beginning of a macro \*.TCM file (or to the primary macro file on the top level if any sub-level(s) of macro files are used).

Automatically displays on screen the complete details of various commands in all the sequential operations in the software, that are enforced according to the macro file(s).

## SET\_GES\_VERSION

This command enables switching between the two versions of the Gibbs Energy System (GES) calculation engine. As of Thermo-Calc version 2019b, GES6 is the default. Also see [About the Gibbs Energy System \(GES\) Module](#).



GES6 is a re-write and new implementation of the Gibbs Energy System module of Thermo-Calc and it is the default engine. The previous version, GES5, is still available and will continue to work in the background of the program where necessary. The main purpose of GES6 is to support faster development of new features. However, not all GES5 functionality is implemented in GES6. In such cases (except for the POURBAIX module), GES6 falls back silently and uses functionality from GES5 in the background.

Syntax	SET_GES_VERSION
Prompt	USE GES VERSION 5 OR 6 /6/: The default is version 6. If you change the version to GES5, it is set for the current session of Thermo-Calc only. If you launch a new Console window, the version resets to GES6.
	 <p>To globally set the default from GES6 back to GES5, go to the <b>Tools</b> → <b>Options</b> → <a href="#">Global General Settings</a> tab and click <b>Version 5</b> for the <i>Preferred Gibbs Energy System</i> setting. This is useful if you are using the PARROT module (optimization) or if you have a custom database that is not compatible with GES6.</p>



Contact Thermo-Calc Software AB at [info@thermocalc.com](mailto:info@thermocalc.com) for more information about the changes or if you encounter any issues you think are related to the new version of the calculation engine.

## **SET\_INTERACTIVE**

Use the SET\_INTERACTIVE command to reset the input and output units to the initial values, i.e. keyboard and screen. Add this as the last command to the macro files.

---

## SET\_INTERACTIVE\_MODE

Use the SET\_INTERACTIVE\_MODE command to reset the input and output units to the initial values, i.e. keyboard and screen.



Remember to add this as the last command to your macro files, in order to stop execution of the command file in the POST module. It has no meaning in interactive mode.

## SET\_LOG\_FILE

Use it to save, in a basic text file, everything typed into Thermo-Calc. When having problems executing a command sequence, this command can be used to save the command typed onto a log file with the file extension \*.TCM.

This command also makes the system echo the full command for all commands typed. This feature is useful when demonstrating the system since the abbreviated commands are often difficult.

The saved log file can then be edited as a macro file by using a basic text editor. This is useful to run the macro file(s) for similar calculations (the same system but slightly varied temperature-pressure-composition conditions), or run the example macro files.



### [Macro Files](#)

Syntax	SET_LOG_FILE
Prompt	A <b>Save</b> window opens to specify a file name for the *.TCM file, so that the path (in the <b>Save in</b> field) and <b>File name</b> can be specified. If a log file with the same name exists in the current working directory, it is overwritten. The <b>Files of type</b> (i.e. TCM) cannot be changed. Click <b>Save</b> or <b>Cancel</b> as applicable.

## SET\_PLOT\_ENVIRONMENT

Use this command to set the plot devices to save or print plots to, which varies for by installation type.



This command must be terminated with an empty line or two commas.

Syntax	SET_PLOT_ENVIRONMENT
Prompts	<p>Default plot device number /Default number/: &lt;Device number&gt;</p> <p>The number given is unique for each type of graphical device. The default graphical device number should normally be selected. A question mark ? gives you a list of all available graphical devices used.</p>
	<p>Pseudo file name: &lt;Pseudo-file name&gt;</p> <p>The name given here is a symbol which can be used to refer to a physical graphics device when asked for graphical output file.</p>
	<p>Plot device number /1/: &lt;Device number&gt;</p> <p>The number that specifies the type of graphical device.</p>
	<p>Plot file name: &lt;File name or printer name&gt;</p> <p>The name of the graphical device on system level (name of file or printer).</p> <p><b>Example</b></p> <pre>SET_PLOT_ENVIRONMENT 1 lasp 5 a0tr,,</pre> <p>This example sets the default plot device number to 1, and defines an alias named <code>lasp</code> with plot device 5 and connected to a printer named <code>a0tr</code>.</p>

## SET\_TC\_OPTIONS

Predefine the default values of three general options for performing all the sequential single-points, stepping and mapping calculations.

Syntax	SET_TC_OPTIONS
Prompts	<p><code>Use_global_minimization /Y/:</code></p> <p>By default it global minimization is used where possible (depending on the conditions setting). A global minimization test is performed when an equilibrium is reached. This costs more computer time but the calculations are more robust. Enter N to prevent global minimization being used to reach the equilibrium.</p>
	<p><code>Global_test_interval /0/: &lt;An integer value&gt;</code></p> <p>The integer number determines how often the global minimization should be used during calculations with STEP_WITH_OPTIONS and MAP. If it is set to 0 (zero), the recommended global test interval is used: i.e. every tenth step and at each phase change during step calculations, and only at node points during map calculations. Any other positive integer number, n, suggests it performs global minimization tests at every n<sup>th</sup> step during step and map calculations. Of course, the Global Minimization test is always carried out at a phase change or a node point. The software is initially configured with 0 (zero) as the default value, but such a configuration for the default value can be changed by this option.</p>
	<p><code>Automatically_add_initial_equilibria /Y/:</code></p> <p>When ADD_INITIAL_EQUILIBRIUM is not used before a map calculation, a mesh of INITIAL_EQUILIBRIUM is automatically added before the mapping procedure itself takes place. This mesh is only available when the two independent axis-variables have already been defined by the POLY command.</p>
	<p><code>Set_axis_variable</code></p> <p>If Y is entered, the POLY module always enforces the automatic procedures for adding initial equilibrium points during mapping. For N, no initial equilibrium points are then added during mapping.</p>

---

## STOP\_ON\_ERROR

Useful for batch jobs in order to prevent that an erroneous sequence of commands to the program causes waste of computer activity.

Syntax	STOP_ON_ERROR
	Give the preferred argument ( <code>ON</code> or <code>OFF</code> ) after the <code>STOP</code> command. If the default argument <code>ON</code> is given, the program terminates after an illegal or ambiguous command. It is possible to reset the effect of this command by giving the argument <code>OFF</code> .

---

## Database (TDB) Module Commands

---



These commands are related to the DATABASE\_RETRIEVAL module, which is also known as the TDB module.

Search the online help for step-by-step instructions to define a system.

### In this section:

---

AMEND_SELECTION .....	164
APPEND_DATABASE .....	165
DATABASE_INFORMATION .....	166
DEFINE_ELEMENTS .....	167
DEFINE_SPECIES .....	168
DEFINE_SYSTEM .....	169
GET_DATA .....	170
LIST_DATABASE .....	171
LIST_SYSTEM .....	172
REJECT .....	173
RESTORE .....	174
SET_AUTO_APPEND_DATABASE .....	175
SWITCH_DATABASE .....	176

## AMEND\_SELECTION

 This command is for the DATABASE\_RETRIEVAL (TDB) module.

Use this command after defining the elements, species or the system (with the commands DEFINE\_ELEMENTS, DEFINE\_SPECIES, or DEFINE\_SYSTEM). The prompts allow changes to the predefined system. By answering Y or N, each of the selected elements, species or phases can be accepted or rejected (although this is not the case for the constituents or the entire system).

Syntax	AMEND_SELECTION
Prompt	Keep <Name1> No/Quit/Yes/ Keep <Name2> No/Quit/Yes/
Options	Keyword-- Elements/Species/Phases name& Names of the pre-defined or pre-selected elements/species/phases

## APPEND\_DATABASE



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Appends data to the current set of data already read from another database. The syntax is the same as for the SWITCH\_DATABASE command.

This command enters all additional functions and parameters to already existing phases, e.g. atomic mobilities, phase constituents, excess models, GO and interaction parameters.

This command is most commonly used to read mobility data from stand-alone mobility databases.

This command should be used with some caution if thermodynamic data is appended and the application is run with Gibbs Energy System version 5 (see SET\_GES\_VERSION in the SYS module). It can change calculated values in an unexpected way. This can happen e.g. if the appended database has the same function name as a function in the data retrieved from a database opened with SWITCH\_DATABASE.

Syntax	APPEND_DATABASE
Prompt	New database name /XYZ/: The name of either an existing database (e.g. FEDEMO) or USER.

## **DATABASE\_INFORMATION**



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Use the DATABASE\_INFORMATION command to obtain a short description of the current database. This can include information on covered systems, used models, valid ranges for parameters in temperature and composition, major applications, and so on.

## DEFINE\_ELEMENTS



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Define the system in terms of elements. All possible species that can be formed by the given elements are retrieved from the database. The names of elements must be separated with a space or comma. It is possible to use a wildcard \* after a common part of element names so that all the elements, which start with that common part and are available in the currently switched or appending database, are defined in the system. Up to 40 elements can be defined into a single system.



Also see [Defining a System in Console Mode](#) for step-by-step instructions to define a system with this command.

Syntax	DEFINE_ELEMENTS
Prompt	Element& <Element1, Element2, ...> A list of elements to be defined into the system.
	 When appending database(s), this command, or DEFINE_SPECIES or DEFINE_SYSTEM must be repeated with the same or similar elements as defined in the first switched database.

## DEFINE\_SPECIES



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Define the system in terms of species. Only those species given are retrieved. The different names of species must be separated with a space or comma. It is possible to use a wildcard \* after a common part of species names so that all the species, which start with that common part and are available in the currently switched or appending database, are defined in the system. Up to 1000 species can be defined in a single system.



### Defining a System in Console Mode

Syntax	DEFINE_SPECIES
Prompt	Species& <Species1, Species2, ...> A list of species to be defined in the system.
	 When appending database(s), this command, DEFINE_ELEMENTS or DEFINE_SYSTEM must be repeated with the same or similar elements as defined in the first switched database.

## DEFINE\_SYSTEM



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Define the system in terms of either elements (equivalent to DEFINE\_ELEMENTS) or species (equivalent to DEFINE\_SPECIES). Certain databases have a default value of the keyword (as either ELEMENTS or SPECIES) reflecting what is most appropriate when defining a system.

The different names of elements or species must be separated with a space or comma. It is possible to use a wildcard \* after a common part of elements or species names so that all the elements or species, which start with that common part and are available in the currently switched or appending database, are defined in the system.

When appending database(s), this command (or DEFINE\_ELEMENTS or DEFINE\_SPECIES) must be repeated with the same or similar elements as defined in the first switched database. Different databases might contain different elements and have different species definitions: avoid defining elements/species that are missing in the appending database(s). Otherwise, the program indicates these missing elements/species, and ignores them in subsequent steps. But additional elements/species, and additional phases not available in the first switched database can be defined and retrieved from the appending database(s). Up to 40 elements and 1000 species can be defined in a single system.



Also see [Defining a System in Console Mode](#) for step-by-step instructions to define a system with this command.

Syntax	DEFINE_SYSTEM
Prompt	Elements: <Element1, Element2, ...>, or Species: <Species1, Species2, ...>
Options	Description
Species or Elements	Default keyword.
Element&	Specify a list of elements for the defining system.
Species&	Specify a list of species for the defining system.

---

## GET\_DATA



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Enter the defined system's elements, species, phases, and the connected parameters obtained from either the primary switched or additionally appending database(s) to the GIBBS and/or DICTRA workspace. Then use the GET\_DATA command to get all information concerning a defined system from the databank.



Only after executing this command can you go to any of the application programs such as GES, POLY or DICTRA and use the retrieved data. When appending database(s), this command must be repeated in order to obtain the additional system definitions, parameters and functions.

## LIST\_DATABASE



This command is for the DATABASE\_RETRIEVAL (TDB) module.

List all elements, species, phases or phase constituents in the database.

Syntax	LIST_DATABASE
Options	Description
Keyword	One of the keywords Elements, Species, Phases or Constituent must be used to indicate what to list
Elements	All available elements, the reference state, atomic mass, H298-H0 and S298. Some elements have spaces in the column for the reference state. This implies that there are no parameters stored for this element.
Species	All available species together with the stoichiometric factors.
Phases	All available phases together with the number of sublattices and the number of sites in each sublattice.
Constituent	All available phases, the number of sublattices, the number of sites in each sublattice and the species dissolved in each phase. Species in different sublattices are separated with a colon (:). It is important to realize, for example, a phase can consist of Fe, Mo, V, and Cr, and its thermodynamic parameters can come from the binary systems Fe-Mo, Fe-V, Fe-Cr, and Mo-Cr. These data can give a relatively good description of the corners of the Fe-Cr-Mo system, but would most certainly give a bad one for the system Mo-Cr-V, due to the fact that the interaction parameters are, by default, set to zero, which originates from binary systems not included in the database.

## LIST\_SYSTEM

 This command is for the DATABASE\_RETRIEVAL (TDB) module.

List all elements, species, phases or phase constituents in the defined system. It works only after a system is defined.

Syntax	LIST_SYSTEM
Keyword	One of the keywords <code>Elements</code> , <code>Species</code> , <code>Phases</code> or <code>Constituent</code> must be used to indicate what to list.  <a href="#">LIST_DATABASE</a>

## REJECT



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Reject elements, species, phases or phase constituents that can form from the defined elements and species. Phases/species/constituents that are possible to form in the defined system are removed from the list of system phases/species/constituents (shown by the command LIST\_SYSTEM). Phases/species/constituents that are not included in the list cannot be entered without first being restored. The different names must be separated with a space or comma. You can use a wildcard \* after a common part of names so that all the elements/species/phases/constituents that start with that common part and are available in the currently switched or appending database, are rejected from the defined system.

This command can also reject a defined system, and thus reinitiate the entire DATA module memory and GES workspace.

Syntax	REJECT
Prompt 1	(Keyword = Elements or Species or Phases) If keyword: <Name1, Name2, ...>
Prompt 2	(if keyword = Constituent) Phase: <Phase name> Sublattice number: <Sublattice number in the phase> Constituent: <Constituent(s) in the sublattice of the phase>
Prompt 3	If keyword = SYSTEM the DATA module is reset to its initial state; GIBBS is reinitiated and data already entered to GES is lost.
<b>Options</b>	<b>Description and additional information</b>
Keyword	One of the keywords <code>Elements</code> , <code>Species</code> , <code>Phases</code> , <code>Constituent</code> or <code>System</code> must be used to indicate what is rejected.
Name&	Names of the pre-defined elements/species/phases/constituents/system
Elements	The given elements are rejected.
Species	The given species are rejected, making it impossible to form them from the defined elements.
Phases	The given phases are rejected, making it impossible to form them from the defined elements or species.
Constituent	The given constituent in one phase are rejected. Add the following when prompted: Phase: The name of the phase containing the constituent to be rejected. Sublattice number: The sublattice where the constituent enters (the first sublattice is 1). The question is omitted if there exists only one possible sublattice. Constituent: The name(s) of the constituent(s) to be rejected.

## RESTORE



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Restore already explicitly rejected elements, species, phases or constituents; it is the opposite of the command REJECT although it does not restore a completely rejected system.

Phases/species/constituents that are possible to form from the defined elements or species are entered to the list of system phases/species/constituents (shown by the command LIST\_SYSTEM). Phases/species/constituents that are not included on the list can now be added to the list. The different names must be separated with a space or comma. It is possible to use a wildcard \* after a common part of names so that all the elements/species/phases/constituents, which start with that common part and are available in the currently switched or appending database, are restored in the defined system.

Syntax	RESTORE
Prompt1	(if keyword = ELEMENTS or SPECIES or PHASES) Keyword: <Name1, Name2, ...>
Prompt2	(if keyword = CONSTITUENT) Phase: <Phase name> Sublattice number: <Sublattice number in the phase> Constituent: <Constituent(s) in the sublattice of the phase>
<b>Options</b>	<b>Description and Information</b>
Keyword	One of the keywords Elements, Species, Phases or Constituent must be used to indicate what is rejected.
Name&	Names of the pre-defined elements, species, phases, and constituents
Elements	The given elements are restored.
Species	The given species are restored and thus possible to form from the defined elements.
Phases	The given phases are restored and thus possible to form from the defined elements or species.
Constituent	The given constituent in one phase are restored. Add the following when prompted: Phase: The name of the phase containing the constituent to be restored. Sublattice number: The sublattice where the constituent enters (the first sublattice is 1). Constituent: The name(s) of the constituent(s) to be restored.

## SET\_AUTO\_APPEND\_DATABASE



This command is for the DATABASE\_RETRIEVAL (TDB) module.

Enforce an automatic action that appends thermodynamic data from a chosen database regarding the system to the data you already have about the system from the current database (that is, the default database or the database last chosen with SWITCH\_DATABASE or APPEND\_DATABASE).



This is useful to simultaneously retrieve both thermodynamic and mobility data for a defined system when performing a DICTRA module simulation.

This command should be used before defining a system (with the commands DEFINE\_ELEMENTS, DEFINE\_SPECIES, or DEFINE\_SYSTEM) and retrieving the data with the GET\_DATA command from the primary switched database.

Syntax	SET_AUTO_APPEND_DATABASE
Prompt	Database name /XYZ/: <Additional database name>
Option	Additional database name The name of an existing database or a User database that corresponds to the automatically-appending database.

It works in a way that is similar to the APPEND\_DATABASE<DATABASE-NAME> command sequence, but only the phases that are also available in the primary switched database are appended. There is no possibility to manually list, reject and/or restore any phase that is available in the appending database. When retrieving data with GET\_DATA afterwards, all the phases that exist in the appending database but not in the primary switched database are automatically rejected. The prompt is kept as for the primary switched database, TDB\_XYZ: (where XYZ stands for the name of the primary switched database) until the execution of GET\_DATA.

Therefore, to selectively append more phases from a secondary database to a system that is defined and retrieved from a primary switched database, then APPEND\_DATABASE and sequential commands (such as DATABASE\_INFORMATION, DEFINE\_SYSTEM, DEFINE\_ELEMENT, DEFINE\_SPECIES, LIST\_SYSTEM, REJECT, RESTORE and GET\_DATA) should be used instead, before this command.

Although it is impossible to manually list, reject and restore any phase from the appending database, the DATA module automatically ignores all the phases that do not exist in both the primary switched database and the appending database, and append the data for the phases (that are also available in the primary switched database) from the appending database, as informed on screen when executing GET\_DATA afterwards.

## SWITCH\_DATABASE

 This command is for the DATABASE\_RETRIEVAL (TDB) module.

Switch (or change) from the current database to a new one. It also reinitializes the entire module.

The database can be either a standard database (the ones in `tc_initd.tdb`) or a so-called user database. To open your own (user) database, type `USER` followed by the path to the user `.TDB` file or press <Enter> and select it in the window that opens.

Syntax	SWITCH_DATABASE
Prompt	New database name /XYZ/: The name of either an existing database (e.g. FEDEMO) or USER.

---

## POLY Module Commands

---

### In this section:

---

ADD_INITIAL_EQUILIBRIUM .....	181
ADVANCED_OPTIONS .....	184
AMEND_STORED_EQUILIBRIA .....	185
CHANGE_STATUS .....	187
COMPUTE_EQUILIBRIUM .....	192
COMPUTE_TRANSITION .....	194
CREATE_NEW_EQUILIBRIUM .....	196
DEFINE_COMPONENTS .....	197
DEFINE_DIAGRAM .....	198
DEFINE_MATERIAL .....	201
DELETE_INITIAL_EQUILIBRIUM .....	205
DELETE_SYMBOL .....	206
ENTER_SYMBOL .....	207
EQUILIBRIUM_CALCUL .....	210
EVALUATE_FUNCTIONS .....	212
GLOBAL_MINIMIZATION .....	213
KEEP_COMP_SET_NUMBERS .....	215
LIST_AXIS_VARIABLE .....	216
LIST_CONDITIONS .....	217
LIST_EQUILIBRIUM .....	218
LIST_INITIAL_EQUILIBRIA .....	219
LIST_PHASE_ADDITION .....	220
LIST_STATUS .....	221

---

LIST_SYMBOLS .....	222
LOAD_INITIAL_EQUILIBRIUM .....	223
MAKE_COMPONENT_ENTERED .....	224
MAKE_COMPONENT_SUSPENDED .....	225
MAJOR_CONSTITUENTS .....	226
MAP .....	227
NEW_COMPOSITION_SET .....	229
OUTPUT_FILE_FOR_SHOW .....	231
PARAEQUILIBRIUM .....	232
PHASE_ADDITION .....	235
POST .....	236
PRESENT_PHASE .....	237
READ_WORKSPACES .....	238
REINITIATE_MODULE .....	239
SAVE_WORKSPACES .....	240
SELECT_EQUILIBRIUM .....	242
SET_ALL_START_VALUES .....	243
SET_AXIS_VARIABLE .....	245
SET_CONDITION .....	247
SET_INPUT_AMOUNTS .....	250
SET_INTERACTIVE .....	251
SET_NUMERICAL_LIMITS .....	252
SET_REFERENCE_STATE .....	254
SET_START_CONSTITUTION .....	256
SET_START_VALUE .....	257
SHOW_FOR_T= .....	258

---

---

SHOW_VALUE .....	260
STABILITY_CHECK .....	261
STEP_AND_MAP .....	262
STEP_WITH_OPTIONS .....	264
TABULATE .....	277
TOGGLE_ALTERNATE_MODE .....	278
T-ZERO_TEMPERATURE .....	279



## ADD\_INITIAL\_EQUILIBRIUM

 This command is for the POLY module.

Add initial equilibrium points from which a phase diagram is calculated (through the MAP command).

For each initial equilibrium the first stage of the mapping varies the value of the given axis variable condition, changing it in the given direction. When a phase change is found it is saved as a start equilibrium, and if the symbol ">" was used, it continues the search until the end of the axis variable.



If no axis variable conditions differ between initial equilibria then the map results in several overlaid diagrams, in accordance with the selected conditions and the values.

The ADD\_INITIAL\_EQUILIBRIUM default command implicitly makes the MAP command search for phase changes according to settings set by ADVANCED\_OPTIONS STEP\_AND\_MAP command. Default for STEP\_AND\_MAP are three initial equilibrium equidistantly along each axis variable, along the edge of the axis variable rectangle.

By default these default start points are used in MAP. If any ADD\_INITIAL\_EQUILIBRIUM is used these default start points are not used. They are again used if the command ADD default is used together with another ADD\_INITIAL\_EQUILIBRIUM command.

The ADD command with the default direction scans along the axis variables and generates start points each time the scan procedure crosses a phase boundary. In addition, it generates four start points, scanning cross the middle of each axis, if there is any solubility line that does not reach the axes. At the MAP command, a search for lines in the diagram is made along each direction of the axis variables in the diagram.

In this way, it should guarantee that all possible phase boundary lines in a phase diagram are found. It may take a little longer to execute than using the minimum number of start points, as some lines may be calculated more than once. But the POLY module remembers all node points and subsequently stops calculations along a line when it finds a known node point.

It is also possible to create a sequence of start points from one initial equilibria by appending a > after the direction at the ADD command. For example:

```
Direction /Default/: 2>  
Direction /Default/: -2>
```

This generates one start point for each set of phase change in the positive direction of the axis 2 (or negative direction of the axis 2); this ensures finding all possible phase boundary lines (not just the first one) along such an axis direction.

This is particularly useful when you have a phase diagram with several lines with no intersection. It is thus possible to calculate e.g. an isopleth for a much more limited composition range. It is also useful for calculating CVD diagrams and Pourbaix diagrams.

Syntax	ADD_INITIAL_EQUILIBRIUM
Prompt	<pre>Direction /Default/: &lt;Direction code&gt;</pre> <p>Direction code(s):</p> <ul style="list-style-type: none"> <li>• 1 or 2 for positive direction of axis 1 or 2, respectively.</li> <li>• -1 or -2 for negative direction of axis 1 or 2, respectively.</li> <li>• <code>Default</code> for MAP to start looking for phase changes from each corner of the axis variable rectangle.</li> </ul> <p>The direction is important when the initial equilibrium point is in a single-phase region or when the diagram is an isopleth (tie-lines not in the plane of calculation). In such cases, the program will search for a line in the diagram (i.e., a line where the amount of a phase is zero) in the given direction.</p>

Normally, this command is not needed to calculate a property diagram (with the `STEP_WITH_OPTIONS` → `NORMAL` command-sequence), as the `STEP` procedure starts from the current equilibrium state already calculated. In many cases, the `ADD_INITIAL_EQUILIBRIUM` command is not required for `MAP` commands either.

In order to calculate a simple property diagram, set the equilibrium conditions and the stepping axis variable, and then give a `STEP` command. But if a phase diagram has disconnected lines, `ADD_INITIAL_EQUILIBRIUM` may still be needed to add two or more initial equilibria so that the `MAP` calculation starts from such initial equilibria at the specified directions to find all phase boundary lines.

This command becomes unnecessary in most cases, as the `MAP` routines that by default use the Global Minimization Technique handles all the initial equilibrium points in a robust and automatic way. Therefore, you do not need to have a good guess of the starting point and to add any initial equilibrium point prior to the calculations of various types of phase diagrams. However, if preferred, use this command to add any initial equilibrium points in certain directions, the `POLY` module uses the specified starting point(s) and corresponding user-added initial equilibrium point(s) for a mapping calculation; in such a case, the automatic procedure in the rewritten `MAP` routines are not enforced while the Global Minimization Technique can still be applied. Similar situations occur when a specific database, in which there are some definitions of initial-equilibrium adding direction(s) for accessed binary and/or ternary subsystems in its `ACCESSED_SYSTEM` section, is used in the `BINARY` or `TERNARY` module for calculating a binary or ternary phase diagram.

All the initial equilibrium points generated by `ADD_INITIAL_EQUILIBRIUM` (previously and presently; saved in the current POLY workspace) can be easily listed on screen with `LIST_INITIAL_EQUILIBRIA`. A certain initial equilibrium point (including its conditions and equilibrium results) can be loaded into the current equilibrium, if needed, with `LOAD_INITIAL_EQUILIBRIUM`. Any specific or all of the initial equilibrium points can be deleted from current POLY workspace, if desired, with `DELETE_INITIAL_EQUILIBRIUM`.

## ADVANCED\_OPTIONS



This command is available with the POLY and ED-EXP modules.

Syntax	ADVANCED_OPTIONS
Prompt	Which option? /Step_and_Map/: <Option>

Several options are available after you enter this command at the prompt. The default is STEP\_AND\_MAP.

```
POLY:advanced-options
Which option? /STEP_AND_MAP/:?
EQUILIBRIUM_CALCUL      NEW_COMPOSITION_SET      STABILITY_CHECK
GLOBAL_MINIMIZATION     OUTPUT_FILE_FOR_SHOW     STEP_AND_MAP
IGNORE_COMPOSI_SET_ORDER PARAEQUILIBRIUM         T-ZERO TEMPERATURE
KEEP_COMP_SET_NUMBERS   PHASE_ADDITION           TOGGLE_ALTERNATE_MODE
LIST_PHASE_ADDITION     PRESENT_PHASE
MAJOR_CONSTITUENTS      SHOW_FOR_T=
Which option? /STEP_AND_MAP/:
```

- [EQUILIBRIUM\\_CALCUL](#)
- [GLOBAL\\_MINIMIZATION](#)
- [KEEP\\_COMP\\_SET\\_NUMBERS](#)
- [LIST\\_PHASE\\_ADDITION](#)
- [MAJOR\\_CONSTITUENTS](#)
- [NEW\\_COMPOSITION\\_SET](#)
- [OUTPUT\\_FILE\\_FOR\\_SHOW](#)
- [PARAEQUILIBRIUM](#)
- [PHASE\\_ADDITION](#)
- [PRESENT\\_PHASE](#)
- [SHOW\\_FOR\\_T=](#)
- [STABILITY\\_CHECK](#)
- [STEP\\_AND\\_MAP](#)
- [TOGGLE\\_ALTERNATE\\_MODE](#)
- [T-ZERO\\_TEMPERATURE](#)

## AMEND\_STORED\_EQUILIBRIA

 This command is for the POLY module.

This command gives information about the calculated blocks (and phase regions included in blocks) after the STEP or MAP calculation(s). It allows you to list all or part of the calculation results, to suspend all or parts of the calculation results that are redundant or where metastable equilibria are calculated, and to restore all or parts of the calculation results (if having been suspended by another AMEND\_STORED\_EQUILIBRIA command).

The workspace for storing equilibria may overflow during stepping or mapping, and is then written to a file as blocks. Each block usually contains one or more ranges of equilibrium regions.

Use one these options:

- **L** to list the calculated equilibria (all or a specified block)
- **S** to suspend everything (all blocks and their regions)
- **Q** to suspend each set of equilibria individually (specified blocks and/or regions)
- **R** to restore everything (all blocks and their regions)

Syntax	AMEND_STORED_EQUILIBRIA
Prompts	<p>Name: &lt;Name of a defined table&gt;</p> <p>List (<b>L</b>) the calculated equilibria, suspend everything (<b>S</b>) or suspend each set of equilibria individually (<b>Q</b>), or restore everything (<b>R</b>).</p>
	<p>Block /*/: &lt;Block number&gt;</p> <p>Specify a block number in the option <b>L</b>, <b>S</b>, <b>Q</b> or <b>R</b>. Or include all the blocks in the amending option, by accepting the wildcard * (press &lt;Enter&gt;).</p> <p>For <b>L</b>, a block (if a block number is specified) or all the blocks (if the wildcard * is accepted/used), with the regions and the equilibrium details, that are calculated during stepping or mapping is listed on screen or in a textual file.</p> <p>For <b>Q</b>, each ranger in a block (if a block number is specified) or in all blocks (if the wildcard * is accepted/used) then choose:</p>
	<p>S (uspend) K(eep) /K/: &lt;S or K&gt;</p> <p>For <b>S</b> or <b>R</b>, if a block is specified, you are asked what region in the block should be suspended or restored. However, if the wildcard * is accepted/used, you are asked the following:</p> <p>Really suspend all /N/:</p> <p>Really restore all /N/:</p> <p>In both cases the default is N and you are prompted for which block(s) and which region(s) in a certain block to execute the <b>S</b> or <b>R</b> action.</p> <p>Really suspend all /N/:</p>

Syntax	AMEND_STORED_EQUILIBRIA
	<p>Y suspends everything.</p> <p>Really restore all /N/:</p> <p>Y restores everything.</p>
	<p>Range: &lt;Range(s) of region&gt;</p> <p>Specify one or more ranges to be suspended (the <b>S</b> option) or restored (the <b>R</b> option). The wildcard <b>*</b> suspends or restores all ranges in the specified block.</p> <p>In order to know the ranges the <b>LIST</b> command must first be used.</p>
	<p>S(suspend) K(eep) /K/: &lt;S or K&gt;</p> <p>The Query suspend option (<b>Q</b>) asks for each region in a certain block if it should be suspended or kept. Suspended regions are not included on sequentially generated plots.</p>
	<p>Output file: /Screen/: &lt;File name&gt;</p> <p>This prompt is only for listing (the <b>L</b> option), and after it the command is terminated. The file name must be given here; or accept the default Screen (terminal) by pressing &lt;Enter&gt;. A list of stored equilibria as various blocks (with all their ranges) are shown out on screen or the file.</p>

## CHANGE\_STATUS



This command is available with the POLY and ED-EXP modules.

In the POLY module, set the status for components, species and phases in the defined system for all the sequential calculations (single-point, stepping, and mapping) in equilibrium or local/partial equilibrium state. Each component, species and phase has a status. The default status is `ENTERED`.

The most important use is to calculate metastable equilibria and metastable phase diagrams by setting some phases (that would otherwise be stable) to the `SUSPENDED` or `DORMANT` phase-status. Another important applications is to calculate paraequilibria by setting some components to the `SPECIAL` component-status.

For a component and for a species, the status can be one of the following:

- `ENTERED`: the component(s) or species are included in the calculation. This is the default status.
- `SUSPENDED`: the component(s) or species are not considered in the calculation.
- `SPECIAL`: the specified component(s) are not included in summations for mole or mass fractions. It only works for component(s).



Only component(s) can have the status `SPECIAL`, which implies that these are not included in summations for mole or mass fractions.

For example, for the *u-fractions* or other normalized fractions, when one or more of the components are excluded from the summation, you must specify which component(s) should be excluded from the calculation of mole or mass fraction. This component status is particularly useful when calculating paraequilibrium states. Such component(s) are normally interstitial component, and must have the status `SPECIAL`. This is assigned by the `CHANGE_STATUS` command.

Syntax	CHANGE_STATUS
Prompts	<p>For phases, species or components? /Phases/: &lt;Keyword&gt;</p> <p><i>Keyword</i> = phase or species or components</p> <p>Phase name(s): &lt;Name(s) of the phase(s)&gt;</p> <p>For <i>phase</i> as the keyword, the names of the phases that have their status changed must be given (all on one line). A comma or space must be used as separator. The status to be assigned to the phases can also be given on the same line if preceded with an equal sign (=).</p>

Syntax	CHANGE_STATUS
	<div data-bbox="363 260 1395 432" style="border: 1px solid blue; padding: 10px;">  <p>An asterisk, *, can be used to denote all phases. The special notations *S, i.e. a wildcard * directly followed by an S, sign, means all suspended phases. In the same way, *D means all dormant phases, and *E means all entered phases.</p> </div>
	<p>Name(s): &lt;Name(s) of the Species or Component(s)&gt;</p> <p>For species or components as the keyword, the names of the species or components that have their status changed must be given (all on one line). A comma or space must be used as separator. Similarly to the case of phase as the keyword, the status to be assigned to the species or components can also be given on the same line if preceded with an equal sign =.</p> <div data-bbox="363 646 1395 848" style="border: 1px solid blue; padding: 10px;">  <p>An asterisk, *, can be used to denote all species or components. The special notations *S, i.e. a wildcard * directly followed by an S, sign, means all suspended species or components. In the same way, *E means all entered species or components.</p> </div>
	<p>Status /Entered/: &lt;New status&gt;</p> <p>The new status to be assigned must be given.</p> <p>For species, the values ENTERED or SUSPENDED can be used.</p> <div data-bbox="363 1022 1395 1255" style="border: 1px solid blue; padding: 10px;">  <p>For the POLY module, using the Entered or Suspended status here has the same result as using the commands <a href="#">MAKE_COMPONENT_ENTERED</a> and <a href="#">MAKE_COMPONENT_SUSPENDED</a>. The exception is that any existing conditions that depend on a suspended component are removed when using the MAKE_COMPONENT_SUSPENDED command.</p> </div> <p>For components, the status ENTERED, SUSPENDED or SPECIAL can be given. SPECIAL means that this component is excluded from sums for mole fractions and mass fractions, which is useful when calculating the <i>ufractions</i> or other normalized fractions of system components.</p> <p>For phases, the status ENTERED, SUSPENDED, DORMANT or FIXED can be given. DORMANT means the same as suspended but the driving force is calculated. FIXED means that it is a condition that the phase is stable at a certain amount.</p> <p>For example, for the <i>ufractions</i>, when one or more of the components are excluded from the summation, you must specify which component should be excluded from the calculation of the mole fraction. This component must have the status SPECIAL. This is assigned by the CHANGE_STATUS command: <code>Change_Status comp C=special.</code></p>
	<p>Start value, number of mole formula units /0/: &lt;Initial amount&gt;</p> <p>For ENTERED phases, an initial amount of the phase can be given. Normally, 0 is given if the phase is not likely to be stable, and 0.5 or 1 or any positive number if the phase could be stable, but such an initial amount is only used as the rough starting estimation in the equilibrium calculations.</p>
	<p>Number of mole formula units /0/: &lt;Equilibrium amount&gt;</p>

Syntax

## CHANGE\_STATUS

For FIXED phases, the *equilibrium amount* of the phase [always using an initial estimation being the  $NPF(\text{phase})$  value which it is the normalized mole number of components (per mole formula unit) of the specific status-fixed phase] must be given. If the equilibrium amount is zero, then the phase is at its stability limit.

### Note when specifying a FIXED phase status

Special attention should be paid when specifying a FIXED phase status in equilibrium calculations (for single points, stepping or mapping calculations), as described below.

The phase amount variables,  $NP(\text{phase})$ ,  $BP(\text{phase})$  and  $VP(\text{phase})$ , as well as all their M/W/V-suffixed quantities, should not be used as conditions. Instead, use the CHANGE\_STATUS command to set a relevant condition, e.g. `CHANGE_STATUS phase <phase>=fix <amount>` where the fixed <amount> is roughly the same as the F-suffixed quantity  $NPF(\text{phase})$ .

The  $NPF(\text{phase})$  quantity is the normalized mole number of components (per mole formula unit) of the specific phase in the defined system, which unlike other F-suffixed state variables [e.g.  $GF(\text{phase})$ ,  $HF(\text{phase})$  and  $DGF(\text{phase})$ ] cannot be directly applied in any POLY command, implying that it cannot be directly evaluated or listed/shown. If intended to be shown such a normalized phase amount value in an equilibrium state, you should use a properly-entered symbol (function or variable), for example:  $NPF_{abc} = NP(abc)/NA$  or  $NPF_{abc} = NPM(abc)/NA*N$ .  $N$  is the total system size (in mole). The  $NA$  value is a quantity that is phase-dependent (and sometimes also equilibrium-dependent for ionic solution phases), and is the total atomic number in a mole-formula-unit of the specific phase  $abc$  (excluding interstitial component and, of course, vacancy). For example, the SIGMA, FCC, BCC and LIQUID phases (among others) in a defined Fe-Cr-Ni-C-N-O system (retrieved from a specific database) may be modeled by certain models, and their  $NA$  values must be evaluated in different ways, as described below:

```
LIQUID (C,Cr,CrO3/2,Fe,FeO,FeO3/2,N,Ni,NiO)1 NA = 1
FCC_A1 (Cr,Fe,Ni)1(Va,C,N,O)1 NA = 1
BCC_A2 (Cr,Fe,Ni)1(Va,C,N,O)3 NA = 1
SIGMA (Fe,Ni)8(Cr)4(Cr,Fe,Ni)18 NA = 30
```

If in the same Fe-Cr-Ni-C-N-O system the liquid solution phase is modeled by the Two-Sublattice Ionic Liquid Model, i.e.:

```
IONIC_LIQ (Cr+3,Fe+2,Ni+2)p(VA,C,N,O-2,FeO3/2)q,7
```

then the evaluation of its  $NA$  value becomes even more complicated:

$$NA = p + q*y_{C2} + q*y_{N2} + q*y_{O2e} + q*y_{FeO2} / 2$$

where the stoichiometric coefficients  $p$  and  $q$  are also dependent upon the real equilibrium state (rather than having fixed values in the system). Similar situations occur for other (solid) phases which are described by a multiple sublattice model with ionic constituents, such as SPINEL and HALITE phases in some databases.

There is no strange thing when using a zero value [i.e. 0] in a FIXED phase-status, since it means the specified phase is stable in equilibrium state but has a zero-amount of mass in the equilibrium calculations; in other words, on a phase diagram, the specific phase is on a zero-fraction line (ZFL), i.e. it starts becoming stable on one side of a corresponding phase-boundary line or unstable on the other side of the same boundary. It is often and efficient to do so when calculating e.g. solidus equilibrium states.

However, when a non-zero value [it must always be positive; e.g. 1 or 0.5 or 0.3 or 1.5] is to be specified in a FIXED phase-status, it is unnecessarily the exactly same stable amount of the specific

Syntax	CHANGE_STATUS
	<p>FIXED-status phase in a calculated equilibrium state any longer; instead, the &lt;equilibrium amount&gt; value is the NPF(phase) value that is only roughly used as the estimated starting-value of the FIXED-status phase in the equilibrium calculations.</p> <p>Therefore, a FIXED-status for a liquid phase being unity does not necessarily imply that it is a liquidus equilibrium state (where the liquid phase is in equilibrium with some solid phases but the liquid phase takes all the mass in the defined system). A unity value for setting the liquid phase status in calculating liquidus equilibrium state can only be used when the liquid mixture phase is predefined as a single-sublattice solution phase (such as metallic liquid phase in multicomponent alloy systems) and the total system size as one mole (i.e. N=1).</p> <p>When a phase is described by a solution model in which two or more sublattices are considered and these sublattice sites may also have different stoichiometric coefficients [meaning that the mixture phase could have more than one atom in formula [NA&gt;1; see some examples above], the unity value should not be used when setting the FIXED status for the phase; instead, you should use an appropriate value that ranges from 0 to a NPF(phase) value that equals to or is smaller than 1/NA (if the total system size N=1) or 1/NA*N (if N differs from unity). For this reason, if a multicomponent system bears an IONIC_LIQUID phase that is described by the Two-Sublattice Ionic Liquid Model (or any other multiple-sublattice ionic solution phases), it is difficult to use a proper NPF(ION_LIQ) value in setting its FIXED phase-status, because that should be less than (or equal to) the complex value of</p> $N/[p + q*y_{C2} + q*y_{N2} + q*y_{O22} + q*y_{FeO2}^{3/2}].$

## Examples

For example, to obtain the metallic fraction in a system with carbon as an interstitial component, you can set the component status for carbon as `SPECIAL`:

```
Change_Status comp C=special
```



The `SUSPENDED` status for components and species does not always work as expected.

For a phase, it may have one of these statuses:

- **ENTERED**: the phase(s) are included in the equilibrium calculations and these are stable if that minimizes the total Gibbs energy in the defined system. This is the default status for all phases already retrieved from the chosen database(s). An ENTERED phase-status is always associated with an initially-estimated amount [in mole number; normally, as 0 if the phase is not likely to be stable, and as 0.5 or 1 or any positive number if the phase could be stable] but it is only used as the rough starting value in the equilibrium calculations.

- **SUSPENDED:** the phase(s) are not considered in the equilibrium calculations.
- **DORMANT:** the phase(s) are not considered in the equilibrium calculations but their driving forces for precipitation are calculated.
- **FIXED:** it is an equilibrium condition that the status-fixed phase must be stable, and be in equilibrium at a specified amount [always using an initial estimation being the  $NPF(\text{phase})$  value which it is the normalized mole number of components (per mole formula unit) of the specific status-fixed phase]. See more descriptions at the end of this command.

## COMPUTE\_EQUILIBRIUM

 This command is for the POLY module.

The full equilibrium state is calculated for the given set of conditions. The Global Minimization Technique is by default enforced in this command (C\_E), while it can be disabled temporarily (for the current single-point equilibrium calculation) if using C\_E – or C\_E \* command-combination, or permanently (for all the sub sequential single-point calculations or stepping/mapping calculations within the current TCC run) if having decisively switched it off by a user (or possibly in some special modules) through changing the minimization option using the ADVANCED\_OPTIONS → MINIMIZATION\_OPTION command-sequence.

Syntax	COMPUTE_EQUILIBRIUM
Prompt	<p>The C_E command can be used, in order to enforce the ordinary POLY minimization routines in an equilibrium calculation; this is because of that the ordinary C_E command is now associated with the Global Minimization Technique, and only after the Global Minimization</p> <p>If technique is permanently switched off the C_E command makes no difference from the C_E – command-combination.</p> <p>Only certain types of equilibrium conditions [e.g. T, P, N, N(&lt;component&gt;), X(&lt;component&gt;), B, B (&lt;component&gt;), and W(&lt;component&gt;)] are fully supported in the Global Minimization mode (called <i>Direct Global Minimization</i>); and when other types of equilibrium conditions are used, after the initial POLY, a Global Minimization test and corrections are performed until the lowest minimum is found (called <i>Indirect Global Minimization</i>).</p> <p>If there is any problem with convergence, you may try the C_E * command-combination. The character * enforces the command to use an advanced technique to obtain a complex equilibrium. However, after a successful C_E * calculation, you may repeat the C_E command and can check the status of phases, species or components (with LIST_STATUS → CPS command-sequence) and equilibrium conditions (with LIST_CONDITIONS) and list out the calculation results (with LIST_EQUILIBRIUM), because such actions may tell you how to further modify various settings for your current calculation. This command-combination is not that useful anymore, because the Global Minimization Technique that is always associated with the C_E command is even more powerful and more precise in finding the most-stable equilibrium state in a complex heterogeneous interaction system; therefore, the C_E * command-combination is functional and can be used only after the Global Minimization mode has already been disabled temporarily or permanently.</p> <p>Some phases that are not stable in the current equilibrium state may not have their most favorable composition after this command, and thus their driving forces may not be correct. You can force the program to correctly calculate the driving forces of metastable phases, by giving repeated C_E commands until the number of iterations (that is shown on screen after this command) is reduced to 2.</p> <div style="border: 2px solid blue; padding: 10px; margin-top: 10px;"> <p> Also see the POLY command SET_NUMERICAL_LIMITS which can set the approximate driving force for metastable phases option on or off in all the subsequent POLY calculations within the current TCC run.</p> </div>

---

Syntax	COMPUTE_EQUILIBRIUM
	<div data-bbox="354 260 1398 373" style="border: 1px solid blue; padding: 5px;"> If an equilibrium state for the defined system is not found, an error message is given.</div> <p data-bbox="354 401 1398 491">You can try repeating this command a few times, or change some of settings for the numerical limits, for starting variables and starting values, for starting constitutions of certain phases and for reference states of certain components, or to verify some of the defined conditions.</p>

## COMPUTE\_TRANSITION

 This command is for the POLY module.

This command is a combination of the CHANGE\_STATUS, SET\_CONDITION and COMPUTE\_EQUILIBRIUM commands. It allows a direct calculation when a new phase may form by varying one of the already-set conditions. It can be used only after at least one equilibrium is calculated successfully; otherwise, you are informed on the necessity of first making an equilibrium calculation to find out the stable phases under the current conditions.

When this command is used, the program calls the command CHANGE\_STATUS to temporarily change the phase status of a specified phase as FIXED at the *zero* amount, and at the same time to temporarily release one of the existing equilibrium conditions (which is chosen by you). The program calculates a new equilibrium in which that specific phase is stable but its equilibrium amount in the system is zero. The released condition is then assigned with a calculated value that ensures the calculated equilibrium. After wards, the program automatically changes the phase status of that specific phase back to ENTERED, and resets the temporarily released condition as one of the conditions and assigns it with the value that is calculated to ensure the zero-amount formation of that specific phase.

This command is useful to find melting temperature, boiling temperature, or solubility limits, and generally when you want to set the most optimal conditions for calculating an equilibrium where a specific phase becomes stable. It can also be used when you want to know exactly how far away the defined conditions are from the value that can ensure a zero-amount of a specific phase in the system when other conditions remain the same.

After a successful calculation, you can issue a COMPUTE\_EQUILIBRIUM calculation to assure the calculated transitional equilibrium is a really stable one, and can also use LIST\_EQUILIBRIUM to see the details of transitional equilibrium state.

Syntax	COMPUTE_TRANSITION
	<p>Phase to form: &lt;Phase name&gt;</p> <p>A new phase name, e.g. BCC that is expected to form, is specified here. This changes the status of this new phase to be FIXED as 0 amount, and the program shows the information such as:</p> <pre>You must remove one of the these conditions P=100000, T=800, N=1, X(Fe)=.5    DEGREE OF FREEDOM 0</pre>
Prompts	<p>If the key word ANY is used when prompted for Phase to form (instead of a specific phase name), it is possible to find out any new phase to be formed, in a given varying direction sign and at an estimated change of the released condition: a negative sign means at a lower value of the released condition any new phase is to be found, and a positive sign at a higher value; an estimated change of the released condition implies where any new phase is expected (but it is only estimated value, so any value within its reasonable scale would be enough). Such calculations can be repeated if required. This feature is useful to find out all possible phase transformations along a certain released condition.</p>

Syntax	COMPUTE_TRANSITION
	<p>Give the state variable to be removed /T/: &lt;One condition&gt;</p> <p>One condition must be removed, in order to calculate the transition equilibrium where the specified (or any) new phase to be formed at a calculated value of this released variable.</p>
	<p>Therefore, the message may display (after a successful calculation) if, for example, <math>x(\text{Fe})</math> is entered:</p> <pre>To form BCC the condition is set to X(Fe)=.48605791769</pre> <p>This calculated value is assigned as the parameter of that removed condition, in this case, the <math>x(\text{Fe})</math> variable. If the LIST_CONDITIONS command is typed this message displays:</p> <pre>P=100000, T=800, N=1, X(Fe)=4.86057918E-1 DEGREES OF FREEDOM 0</pre> <p>If the key word ANY (instead of a specific phase) is given as the phase name when it is prompted for Phase to form, the line is prompted for a given varying direction sign and an estimated change of the released condition before the calculation of transition equilibrium:</p>
	<p>Estimated change (with sign) /1/: &lt;+/-#&gt;</p> <p>A given varying direction sign and an estimated change of the released condition, in this case as <math>x(\text{Fe})</math>, must be given here: a negative sign means at a lower value of the released condition any new phase is to be found, and a positive sign at a higher value; an estimated change of the released condition implies where any new phase is expected (but it is only estimated value, so any value within its reasonable scale would be enough). For example, if a combination of -0.02 is input, this message may display (after a successful calculation):</p> <pre>To form BCC_A2#1 the condition is set to X(Fe)=.493708756187</pre> <p>This calculated value is then assigned as the parameter of that removed condition, in this case, the <math>x(\text{Fe})</math> variable. The message is shown if the LIST_CONDITIONS command is typed:</p> <pre>P=100000, T=800, N=1, X(Fe)=4.93708756E-1 DEGREES OF FREEDOM 0</pre>

## CREATE\_NEW\_EQUILIBRIUM

 This command is available with the POLY and ED-EXP modules.

During data-assessments using the ED\_EXP module (a submodule of PARROT), you can, in the POLY module, create several equilibria with different sets of conditions and phases (but normally with the same set of components). By default, there is one equilibrium. To keep the set of conditions and phase for this equilibrium, create another one using this command, and use another set of conditions for that. Two equilibria may be useful to calculate easily the enthalpy difference between two states. In the PARROT module, the experimental information is stored as a sequence of equilibria.

Syntax	CREATE_NEW_EQUILIBRIUM
Prompts	<p>Equilibrium number /2/: &lt;A new equilibrium number&gt;</p> <p>Each equilibrium in the POLY workspace is identified by a unique integer number. Such an equilibrium number can be recalled with SELECT_EQUILIBRIUM.</p>
	<p>Initiation code /2/:</p> <p>When an equilibrium is created, you can choose to <b>ENTER</b> all components and phases (initiation code 2), <b>ENTER</b> the components only (initiation code 1) or <b>SUSPEND</b> everything (initiation code 0). No other values are legal.</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;"> <p> The entered components and phases can later be changed with CHANGE_STATUS.</p> </div>

## DEFINE\_COMPONENTS



This command is available with the POLY and ED-EXP modules.

Change the set of components. By default, the elements are used as components. The set of components can be important because some conditions are set using components, for example, the amounts, activities or chemical potentials.

For example, in the system Fe-Si-O, you can define FEO, FE2O3 and SIO2 as components, thus replacing the default FE, SI and O.



This implies a command REINITIATE\_MODULE and it should be given as the first command in the POLY module.

Syntax	DEFINE_COMPONENTS
Prompt	<p>Give all new components / existing components/: &lt;New components&gt;</p> <p>The <i>new components</i> must be specified all on one line. These replace the existing component definitions.</p> <div data-bbox="370 1045 417 1106" data-label="Image"> </div> <p>The number of components cannot be changed with this command. Use CHANGE_STATUS instead.</p>
	<p>To keep some existing components, it is recommended that you also enter these on the line. Otherwise, the added new components may not be defined correctly. This is especially important if some components are built out of several elements.</p> <div data-bbox="370 1318 417 1379" data-label="Image"> </div> <p>The new components do not have to be present as species.</p>

## DEFINE\_DIAGRAM

This allows automatic calculation and plotting of a diagram with a single command. It is the same as the DEFINE\_MATERIAL command up to when the first equilibrium is calculated. The alloy OPTION feature is also available in this command to specify alloying compositions for a special alloy predefined by the OPTION keyword in a selected database (e.g. the TCNI Ni-based Superalloys Database).

Use this command to calculate all types of phase diagrams after specifying all composition value and an initial temperature (if temperature is used as an axis). However, for binary and ternary diagrams, the special BIN and TERN modules may be preferred.

It then lists all the independent variables for the defined system (i.e. temperature and the components) and asks for a variable as the X-axis. You must also specify a maximum and minimum for the X-axis. The second axis (Y-axis) can be another composition (or the temperature if that is not on the X-axis) from the independent variable list. The program then calculates and plots a *phase diagram*, as there are two independent quantities on the axes.

Alternatively, select a dependent quantity as the Y-axis variable from the second list on screen (e.g. the amount of all phases, composition of a specific phase, or fractions of a component in all phases), and the program calculates and plots how this quantity depends on the condition on the X-axis. This is a *property diagram*.

This command ends up within the POST module monitor. You can refine the calculated phase diagram or property diagram.

Moreover, many more property diagrams with axes other than compositions can also be plotted (after the calculation), using SET\_AXIS\_VARIABLE in the sequent POST monitor.

Syntax	DEFINE_DIAGRAM
Prompts	<pre> These prompts are given:  Same elements as before /Y/? Mole percent of &lt;Element&gt; /##/: &lt;Value&gt;  or  Mass percent of &lt;Element&gt; /##/: &lt;Value&gt; Database /ABCDE/: &lt;Database name&gt; Major element or alloy: &lt;Element name&gt; Composition in mass (weight) percent? /Y/: 1st alloying element: &lt;Element name&gt; Mass (weight) percent: &lt;Amount of the above specified element&gt; 2nd alloying element: &lt;Element name&gt; Next alloying element: &lt;Element name&gt; Mass (weight) percent: &lt;Amount of the above specified element&gt; </pre>

Syntax	DEFINE_DIAGRAM
	<p>Temperature (c) /1000/: &lt;Temperature of interest in oc&gt;  Reject phase(s) /none/: &lt;List of phase(s) to be rejected&gt;  Restore phase(s) /none/: &lt;List of phase(s) to be restored&gt;  OK? /Y/: &lt;Y or N&gt;  Should any phase have a miscibility gap check? /N/:  Phase with miscibility gap: &lt;phase name&gt;  Major constituent(s) for sublattice #: /aa/: &lt;Constituent(s)&gt;  Phase with miscibility gap: &lt;Phase name&gt;</p> <p>The first equilibrium is calculated, as with DEFINE_MATERIAL. Then a list of all independent conditions suitable to be chosen as X/Y-axis variables is given by the program.</p>
	<p>Quit? /Y/: &lt;Y or N&gt;</p> <p>This question is asked only when the axis variables are already defined, or if DEFINE_DIAGRAM is used. It then offers an opportunity to quit (Y) the calculation or to continue (N) the calculation but by defining other axes.</p>
	<p>Give the number of the condition to vary /1/: &lt;A condition index&gt;</p> <p>Select one of the independent conditions by giving its index on the condition list as the X-axis variable.</p>
	<p>Minimum value /XXX/: &lt;minimum value for X-axis&gt;</p> <p>Specify the minimum value of the chosen X-axis variable. A default value is shown automatically by the program; press &lt;Enter&gt; to accept it or input another value.</p>
	<p>Maximum value /YYY/: &lt;maximum value for X-axis&gt;</p> <p>Specify the maximum value of the chosen X-axis variable. A default value is shown automatically by the program; press &lt;Enter&gt; to accept it or input another value. Then another list with some dependent quantities is given by the program, which can be selected as the Y-axis variable.</p>
	<p>Give the number of the quantity on the second axis /#/: &lt;##&gt;</p> <p>Select one of the independent conditions or dependent quantities as the Y-axis variable, by giving its corresponding index given on the condition lists.</p> <div data-bbox="363 1339 1393 1451" style="border: 2px solid red; padding: 5px; margin: 10px 0;">  It must be different from the X-axis variable already selected. </div> <p>If selecting one of dependent quantities (by giving the corresponding number from the second list) as the Y-axis, then a property diagram is to be automatically calculated (through a normal stepping procedure) and generated. For composition of a phase the phase name is asked for further specification subsequently.</p> <p>If selecting any of the other independent variables (conditions) on the first list as the Y-axis, then a phase diagram is automatically calculated (through a mapping procedure) and plotted.</p>
	<p>Name of phase: /ABC/: &lt;phase name&gt;</p> <p>This is prompted only in case of that the composition of a phase is selected as the Y-axis variable. The phase name for which the composition varied along with X-axis variable should be specified.</p>

Syntax	DEFINE_DIAGRAM
	<pre>Multiple start points? /Y/:</pre> <p>By default, the program automatically generates and uses multiple start points for mapping the defined phase diagram.</p>
	<pre>Save file /Result/: &lt;File name&gt;</pre> <p>The file name where the calculations are stored (saved as a *.POLY3 file) should be specified; the default file name is <code>RESULT.POLY3</code>.</p> <p> Linux and Mac are case sensitive so ensure you enter <code>.POLY3</code>.</p>

## DEFINE\_MATERIAL

 This command is for the POLY module.

Read data for a system from a database in the POLY module. It is convenient to use for alloys when there is a major component and the amount of the other elements is known in mass (weight) fraction. The command reads the system from the specified database, sets the composition and temperature (and pressure equal to 1 bar) and calculates the equilibrium state before the prompting for a new command. You can list the results with LIST\_EQUILIBRIUM or set a new composition or set axis for a STEP or MAP command.



You cannot append data from different databases in this way. Use this command with data from a USER database.

Syntax	DEFINE_MATERIAL
Prompts	<p>Same elements as before /Y/?</p> <p>This question is asked only if some data is already read from the database, or if the command DEFINE_MATERIAL or DEFINE_DIAGRAM is used. It then offers a convenient way to change the composition and temperature with one command.</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;">  This command only works properly in cases where the composition of the material system is already defined as in the mole-percent or mass-percent unit.         </div>
	<p>Mole percent of &lt;Element&gt; /##/: &lt;Value&gt;</p> <p>or</p> <p>Mass percent of &lt;Element&gt; /##/: &lt;Value&gt;</p> <p>If you have decided to use the same materials system (available in the current POLY3 workspace) by accepting the default (Y) to the previous prompt Same elements as before /Y/?, one of the alternative prompts display for each of the components in the defined system, depending on how the composition is defined (either in mole-percent, or in mass-percent).</p> <p>Prompts are repeated until all the defined components are completed. Then, the program prompts to specify the temperature condition.</p>
	<p>Database /ABCDE/: &lt;Database name&gt;</p> <p>The database with the description for the material must be given, or press &lt;Enter&gt; if using the current database. It is possible to give a USER database.</p>
	<p>Major element or alloy: &lt;Element name&gt;</p> <p>The material must have a major element, usually the element which is present in the largest amount. The fraction of this element is not set but is the rest.</p>

Syntax	DEFINE_MATERIAL
	<p>In some databases there are the alloys predefined. An alloy has a default major element and have limits of the amounts of the alloying elements. If you stay within these limits the calculation gives reasonable results.</p>
	<p>Composition in mass (weight) percent? /Y/:</p> <p>By default the input is taken as mass percent. Choose N to change to mole percent.</p> <div style="border: 2px solid red; padding: 10px; margin: 10px 0;">  Composition should be given in PERCENT not FRACTION, as it is required for the <math>w</math> and <math>x</math> state variables in the SET_CONDITION command.         </div>
	<p>1st alloying element: &lt;Element name&gt;</p> <p>The first alloying element must be given.</p> <p>All alloying elements are asked for in a sequence. These can be given in any order. You must know if you are present as assessed systems in the database. There is no error or warning messages if data are missing. Check the documentation of the database selected.</p> <p>If an alloy is selected, a list of legal alloying elements and their maximum percent is listed on-line.</p>
	<p>Mass (weight) percent: &lt;Amount of the above specified element&gt;</p> <p>The amount of the alloying element in mass (weight) percent. Using DEFINE_MATERIAL you cannot use the normal flexibility for conditions, but all must be given in mass percent. However, you can after wards change the conditions using SET_CONDITION.</p>
	<p>2nd alloying element: &lt;Element name&gt;</p> <p>The second alloying element must be given. If only one, press &lt;Enter&gt;. If an element name is given then the program asks for its mass fraction.</p>
	<p>Mass (weight) percent: &lt;Amount of the above specified element&gt;</p> <p>The amount of the above specified alloying element in mass (weight) percent.</p>
	<p>Next alloying element: &lt;Element name&gt;</p> <p>Continue giving elements and mass (weight) fractions until all elements specified. When all alloying elements and their compositions (as in the above prompt) are specified, press &lt;Enter&gt; as answer to this question to finish the materials definition.</p>
	<p>Temperature (C) /1000/: &lt;Temperature of interest in oC&gt;</p> <p>POLY makes the first calculation after retrieving the data for this temperature. By pressing &lt;Enter&gt; to accept the default temperature. The value should be given in Celsius (°C).</p> <div style="border: 2px solid blue; padding: 10px; margin: 10px 0;">  The pressure is set to 1 bar.         </div>
	<p>Reject phase(s) /None/: &lt;List of phase(s) to be rejected&gt;</p> <p>This is a question generated by the database allowing you to select the phases. Normally, all phases should be included when you press &lt;Enter&gt;. If a phase is to be rejected, the name of the phase must be supplied. Several phase names can be specified in one line. It is possible to reject all phase</p>

Syntax	DEFINE_MATERIAL
	<p>by giving an asterisk *. If the number of phases to be included is much smaller than the total number of phases, it may be convenient to first reject all phases and then restore those that should be included. The question is repeated until you press &lt;Enter&gt; after rejecting all unwanted phases or an asterisk *.</p>
	<p>Restore phase(s) /None/: &lt;List of phase(s) to be restored&gt;</p> <p>You can restore phases that are accidentally or deliberately rejected. It may also be possible to restore some hidden phases.</p> <p>If phases are to be restored the name of the phases must be supplied. Several phase names can be specified in one line. It is possible to restore all phase by giving an asterisk *. The question is repeated until you press &lt;Enter&gt; after restoring all desired phases.</p>
	<p>OK? /Y/:</p> <p>By default, all phases to be selected from the database have the thermodynamic data retrieved and the references from the chosen database are listed. Enter N if there are any errors or you want to amend the selection then you are returned to the prompt about rejecting phase(s).</p>
	<p>Should any phase have a miscibility gap check? /N/:</p> <p>The database usually creates two or more composition sets for phases that can have miscibility gaps. However, for some phases this is not done automatically, for example the miscibility gap in the bcc phase in Fe-Cr is usually ignored. But if it is important to include a Cr-rich bcc phase, specify this here. It costs some computation time and may make the subsequent MAP or STEP more difficult to converge.</p> <p>If you do not want to have any phase with a miscibility gap in the calculation, press &lt;Enter&gt;. Then, DEFINE_MATERIAL starts calculating the equilibrium, and is terminated.</p>
	<p>To set such a phase with miscibility gaps in the calculation, answer Y. Then the software asks questions about the phase names and their constitutions, such as:</p> <p>Phase with miscibility gap: &lt;Phase name&gt;</p> <p>You must supply the phase name, which has a miscibility gap under the specified system and conditions.</p>
	<p>Major constituent(s) for sublattice #: /AA/: &lt;Constituent(s)&gt;</p> <p>The software shows a default constituent in the sublattice # (1, 2, 3, ...), according to the existing phase definition in the chosen database. You can specify one or more major constituents for the sublattice # in the phase.</p> <p>This question is repeated until all sublattices are specified.</p>
	<p>Phase with miscibility gap: &lt;Phase name&gt;</p> <p>You can supply another phase name with a miscibility gap under the specified system and conditions, and answer the questions concerning the major constituent(s) in associated sublattice (s).</p> <p>By pressing &lt;Enter&gt;, the command starts calculating the equilibrium, and then terminates.</p>

Syntax

DEFINE\_MATERIAL



It is also possible to use this command to select an alloy from a specific database (e.g. the TCNI Ni-based superalloys database). Such alloys are predefined by the `OPTION` keyword inside the database, and have their default major elements and composition limits of their alloy elements.

Such alloys available in the selected database (at the prompt `Database /ABCDE/` can be listed on the screen if typing a `?` mark on the prompt `Major element` or `alloy`. When a specific predefined alloy (instead of a major element) is selected, the major element is staked from the alloy definition and shown on the screen (with a message like `Alloy found with major element NI`).

You can only specify alloying elements and the compositions (weight percent or mole percent). Typing a `?` at any of the prompts for the alloying element names, e.g. `1st alloying element`, `2nd alloying element`, lists all the alloying elements and the composition limits in the alloy. If the composition of an alloying element is outside of its limit, there is a message (such as `Amount above limit: 30.0000`) and a prompt `Override limit ? /N/`. If you decide to enforce the override by answering `Y` on this prompt (i.e. accepting the over-limit alloying composition), another warning message (such as `Amount of major element below limit: 70.0000`) and prompt `Override limit ? /N/`. Then further decide if enforcing the overriding: if `Y` then accepting the major element's composition below the limit; if `N` then using the predefined major element composition limit.

## DELETE\_INITIAL\_EQUILIBRIUM

 This command is for the POLY module.

Delete ONE specific initial equilibrium point or ALL of the initial equilibria. The initial equilibria are used as starting points for all the sub-sequential MAP and STEP calculations.

 [ADD\\_INITIAL\\_EQUILIBRIUM.](#)

Syntax	DELETE_INITIAL_EQUILIBRIUM
Prompt	Number /All/: <Number of an initial equilibrium>
	Specify the number of an initial equilibrium (as a specific numerical value e.g. 3, or ALL) to be deleted from the POLY3 workspaces. It is recommended to use LIST_INITIAL_EQUILIBRIA to figure out the numbers for all the existing initial equilibrium points that have already been added ( <i>manually</i> through ADD_INITIAL_EQUILIBRIUM, or <i>automatically</i> by some other POLY commands) and stored in the current POLY3 workspace. By default, ALL the initial equilibrium points are deleted.

## DELETE\_SYMBOL



This command is available with the POLY and ED-EXP modules.

Use the DELETE\_SYMBOL command to remove symbols i.e. constants, variables, functions or tables.

Syntax	DELETE_SYMBOL
Prompt	Name: <Name of a symbol> Specify the name of the symbol to be deleted. Only one symbol can be deleted each time.

## ENTER\_SYMBOL



This command is for the POLY module.



There are also GES and POST module commands with the same name.

Symbols are a useful feature of the POLY and POST modules to define quantities that are convenient. Symbols can be constants, variables, functions or tables.

Functions or tables (with defined functions as values) can be entered in the POST module after a stepping or mapping calculation, for purposes of plotting such entered functions or tables as axis variables.

Within the POLY module, symbols are normally defined prior to an equilibrium calculation (enforced by a C\_E command), stepping calculation (enforced by the command STEP\_WITH\_OPTIONS) or mapping calculation (enforced by MAP). These can be entered after an equilibrium calculation; however, for defined functions, variable or tables, it requires using EVALUATE\_FUNCTIONS before showing the corresponding values in the calculated equilibrium state.

The symbols entered in the POST module are not saved in the currently-loaded POLY3 workspaces. Therefore, if you want to apply such symbols in other similar calculations for the same defined system, you must use the ENTER\_SYMBOL command prior to the STEPPING or MAPPING calculation in the POLY module.



See [example 44](#) for an example of using variables and functions.

Symbols are a useful feature modules to define quantities that are convenient. Symbols can be constants, variables, functions, or tables.

Syntax	ENTER_SYMBOL
Prompt	<p>Constant, Variable, Function or Table? /Function/: &lt;Keyword&gt;</p> <p>The <i>Keyword</i> can be specified as CONSTANT, VARIABLE, FUNCTION, or TABLE.</p> <ul style="list-style-type: none"> <li>CONSTANTS can only be entered once and is a means of using a name for a numeric value. For example, the value of 1 atmosphere in Pascal can be denoted by P0 after the command ENTER_CONSTANT P0=101325. Defined constants can be used as values in condition assignments, for example, SET-COND P=P0.</li> <li>FUNCTIONS are expressions of state variables or other functions. These expressions are saved, and whenever a function value is requested all functions are evaluated. The reason for this is that they may depend on each other.</li> <li>VARIABLES are similar to functions because they can also be expressions of state variables. However, contrary to functions, they are only evaluated when they are entered or if they are explicitly named in an EVALUATE_FUNCTIONS command. It is possible to enter a variable</li> </ul>

Syntax	ENTER_SYMBOL
	<p>with a new expression any time. This expression is evaluated directly and its value stored as the value of the variable. Defined variables can be used as values in the SET_CONDITION command.</p> <ul style="list-style-type: none"> <li>• TABLES are used for listing results from the STEP or MAP commands. A table consists of a list of any number of state variables, functions, or variables. Defined tables can also be used in the POST (post-processor) module.</li> </ul> <div data-bbox="354 495 1393 667" style="border: 1px solid blue; padding: 10px; margin-top: 10px;">  There is a special connection between tables and variables. If a variable is used in a table, it is evaluated for each line of the table in the TABULATE command or when the table is used in a plot. </div>
	<p>Name: &lt;Name of the symbol&gt;</p> <div data-bbox="354 743 1393 949" style="border: 1px solid black; padding: 10px; margin-top: 10px;">  Each symbol has a unique name that must start with a letter and can have maximum 8 characters. Legal characters include letters (either UPPER or lower case), digits and underscore <code>_</code>. Any other special character, such as parentheses ( and ), plus +, minus -, slash / or \, full stop (.), are illegal for symbol names. </div> <p>You can enter the symbol name and the value or function on the same line; these must be separated with an equal sign =, for example, <code>TC=T-273.15</code> or <code>T_C=T273.15</code> which stands for a definition of temperature in Celsius. Otherwise, these questions are asked.</p> <div data-bbox="354 1092 1393 1234" style="border: 1px solid blue; padding: 10px; margin-top: 10px;">  For different types of symbols (constant, function, variable, or table), the questions have different prompts. </div>
	<p>Function: &lt;Definition for a function or variable&gt;</p> <p><i>Functions</i> and <i>variables</i> are evaluated from an expression of state variables or other functions, constants, or variables. The expression is a FORTRAN-like expression and operators +, -, *, = and ** can be used (** only with integer powers).</p> <p>Unary functions like LOG, LOG10, EXP, SIN, COS, and ERF can also be used. An expression can be continued on more than one line. An expression should be terminated by a semicolon (;) or an empty line (press &lt;Enter&gt; at the next prompt).</p> <p>Examples of function expressions:</p> <ul style="list-style-type: none"> <li>• <code>GM(LIQUID)</code>: The Gibbs energy of liquid per mole component</li> <li>• <code>H.T/4.184</code>: The heat capacity of the system in calories</li> <li>• <code>ACR(CR)/X(FCC,CR)</code>: The activity coefficient for Cr in FCC</li> <li>• <code>T-273.15</code>: The temperature in Celsius</li> </ul> <p>&amp;: &lt;Continuation of the definition for the symbol&gt;</p> <p>The ampersand &amp; means that you can continue to write the function on the new line if one line is</p>

Syntax	ENTER_SYMBOL
	not enough for the function. If you finish the function press <Enter> again.
	Value: <Value for a constant> A <i>constant</i> can only be assigned a numeric value once.
	Value or expression: <Value of expression for a variable> A <i>variable</i> can be assigned a numeric value or an expression. An expression is evaluated immediately and discarded. Only the numeric value is kept. This gives a possibility to save a value between calculations with different conditions because all state variables and functions are evaluated for the new conditions.
	Variable(s): <Variable(s) in a table> A <i>table</i> consists of a list of state variables or functions. One way to obtain results from a STEP command is through a table. Example: <pre>ENTER TABLE K=T,X(LIQ,C),X(LIQ,CR),ACR(C)</pre> Which means that the table called K contains four columns, i.e. the temperature, the mole fractions of C and Cr in the LIQUID phase, and the activity of C. To show the temperature in Celsius in a table, give the command ENTER_FUNCTION TC=T-273; and then use the symbol TC in the table. & <Continuation of the definition for the table> The ampersand & means that you can continue to write the table on the new line if one line is not enough for the table. If you finish the table press <Enter> again.

## EQUILIBRIUM\_CALCUL



This is an option that is available with the `ADVANCED_OPTIONS` command with the POLY and ED-EXP modules.



### ADVANCED\_OPTIONS

Use this command to decide how to perform the POLY minimization (i.e. the traditional Gibbs energy minimization). The *Global Minimization Technique* is used by default for various single point equilibrium calculations and for stepping or mapping calculations.

To permanently turn off the Global Minimization Technique (for all sequential calculations throughout the current run), use the `GLOBAL_MINIMIZATION` command and set the options to use only the local POLY minimization. You can also make other adjustments to control the steps in reaching Gibbs energy minima in an equilibrium state, for example.

When enforcing the Global Minimization Technique in the equilibrium calculations, you can further adjust how to assign grid points and to handle compositional set(s) for solution phases during stepping/mapping. In order to improve convergence, and because the Global Minimization Technique uses the normal POLY optimization routine for some calculations, you can also change how to control the steps in reaching Gibbs energy minima in an equilibrium state.

### *Settings for the Minimization of an Equilibria*

The following prompts show how the POLY optimization does the local POLY minimization to reach the minimum of an equilibrium state.

```
Force positive definite phase Hessian /Y/:
```

Choose `Y` or `N` to determine how to reach the minimum of an equilibrium state in a normal POLY optimization procedure. This is related to the special quantity, the phase stability function  $Q_F(\text{phase})$ , for all kinds of phases in an equilibrium state [a phase stability function for a phase is negative when the phase composition is inside a spinodal, and positive everywhere else].

If the composition of a solution phase is inside its spinodal, the enforcement of positive definite eigenvalues of a Hessian matrix (i.e. when `Y` is selected) makes the step be taken towards the minima rather than the local maxima. Furthermore, if an eigenvalue (for a phase) of the Hessian matrix is near `0.0`, the step size is large this sets a lower limit to this eigenvalue, reducing the step-size and improving the convergence.



The POLY command sequence `SHOW_VALUE QF (phase)` shows the size of the lowest eigenvalue of a phase in an equilibrium state.

If you answer:

- `Y` , it helps the local POLY minimization to converge (if there is a problem), especially if  $QF(\text{phase})=0$  for phases.
- `N` the local POLY minimization routine is applied.

For each phase in a defined system, the molar Gibbs energy of the phase is a function of the temperature-pressure condition and its composition:

$$G_m(X) = G_m(T, P; y_1, y_2, \dots, y_n)$$

Compute a Hessian matrix, which describes the curvature of the Gibbs energy curve of this phase at the defined composition X:

$$\partial^2 G_m / \partial y_i \partial y_j$$

Diagonalize this matrix and call the Eigenvalues as  $e_1, e_2, \dots, e_n$ . The  $QF(\text{phase})$  quantity for this phase is the lowest of these eigenvalues divided by the same values for a corresponding ideal phase:

$$QF(ph) = \min(\{e_1, e_2, \dots, e_n\}) / \min(\{se_1, se_2, \dots, se_n\})$$

Here,  $se_1, se_2, \dots, se_n$  are the Eigenvalues for the ideal phase. Therefore, for an ideal phase, the  $QF(\text{phase})$  should always be 1.0.

Control stepsize during minimization /Y/:

Choose `Y` or `N` to determine whether to introduce a control of step size while reaching the minimum of an equilibrium state in a normal POLY optimization procedure. If you choose `Y` it might help the POLY optimization to converge, especially if some site fractions are less than  $1E-4$ .

## EVALUATE\_FUNCTIONS



This command is available with the POLY and ED-EXP modules.

The value of one or more or all entered functions or variables are evaluated and listed.

Syntax	EVALUATE_FUNCTIONS
Prompt	<p>Name(s): &lt;Name(s) of defined function(s)&gt;</p> <p>The names of one or more entered functions or variables must be specified. By typing a wildcard *, all functions and variables are evaluated.</p> <div data-bbox="354 625 1393 739" style="border: 1px solid blue; padding: 5px;"> Variables are evaluated only if these are explicitly named.</div>

## GLOBAL\_MINIMIZATION



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

Designed to decide how to perform the so-called Global Minimization in the `POLY` module. By default, the `POLY` module uses the Global Minimization technique for various single-point equilibrium calculations and for stepping or mapping calculations. You can use this to permanently (for all sequential calculations throughout the current software run) turn off the Global Minimization technique and consequently use only the Ordinary `POLY` Minimization, and additionally adjust the ways to proceed the normal minimization (mainly in terms of how to control the steps in reaching Gibbs energy minima in an equilibrium state). When enforcing the Global Minimization technique in the equilibrium calculations, you can adjust the manners for assigning grid-points and for handling new possible compositional set(s) for solution phases during stepping/mapping; since the Global Minimization technique also uses the normal `POLY` optimization routine for some calculations, it is also possible to change the ways on how to control the steps in reach Gibbs energy minima in an equilibrium state, in order to improve convergence.

The following prompts affect how the Global Minimization is done. Settings for global minimization:

```
Use global minimization as much as possible /Y/:
```

```
Use global minimization for test only? /N/:
```

```
Maximum number of gridpoints per phase /2000/: <Integer number>
```

```
Use global minimization as much as possible /Y/:
```

Choose Y or N to decide if using the Global Minimization Technique from start of calculation or not. The default value on start-up may be changed by `SET_TC_OPTIONS` in the `SYS` module.

If Y the calculation is done when possible (depending on the condition settings), and a Global Minimization test is always performed when an equilibrium is reached. This costs more computer time but the calculations are more accurate.

If N the calculation is not used to reach the equilibrium state. Use global minimization for test only? determines if the Global Minimization test is made against the calculated equilibrium states obtained by the Ordinary `POLY` Minimization calculation.

```
Use global minimization for test only? /N/:
```

If Y a calculated equilibrium state obtained by the Ordinary POLY Minimization calculation is tested against the Global Minimization Technique after it is reached; and if found it as unstable, there is an error message.

If N the Global Minimization Technique is never tested for or done for all sequential calculations throughout the current TCC run. This implies that Global Minimization Technique is permanently turned off, and that consequently only the Ordinary POLY Minimization routine is used.

Maximum number of gridpoints per phase /2000/: <Integer number>

Specify the maximum number of grid points that are computed for each of the phases in the currently defined system, during the calculations enforced by the global minimization. More grid points give a higher robustness and take more computation time. The total number of computed grid points in all phases is limited to 2E6.

## KEEP\_COMP\_SET\_NUMBERS



This is available with the `ADVANCED_OPTIONS` command in the POLY module.



[ADVANCED\\_OPTIONS](#)



[COMPUTE\\_EQUILIBRIUM](#)

This setting affects the `COMPUTE_EQUILIBRIUM` command and turns on/off the functionality to keep the composition set numbers from the previous equilibrium calculation. The setting is on by default.

To turn the setting on or off enter the `ADVANCED_OPTIONS` command in POLY, then enter this option and either press `<Enter>` to keep the default `Y` or enter `N` at the prompt:

```
Keep composition set numbers /Y/:
```

If this setting is on, performing the `COMPUTE_EQUILIBRIUM` command keeps the composition set number for any phases that were also present in the last result from `COMPUTE_EQUILIBRIUM`.



The composition set numbers are kept when the constitution of a phase in the new equilibrium result is similar to the constitution of one of the composition sets of that phase in the previous result. Making major changes to the conditions between calculations makes it therefore less likely that the composition set numbers are kept. The major constituents, given in the thermodynamic database, are used if no previous result exists.

---

## LIST\_AXIS\_VARIABLE



This command is for the POLY module.

Syntax	LIST_AXIS_VARIABLE
	Lists all the axis variables for a stepping or mapping calculation that have already been set by SET_AXIS_VARIABLE.

## LIST\_CONDITIONS

 This command is available with the POLY and ED-EXP modules.

All the conditions that are set by the SET\_CONDITION command and the command-sequence CHANGE\_STATUS PHASE =FIXED <0 or 1 or alike>, are listed. The current conditions are also listed by the LIST\_EQUILIBRIUM command. The degree of freedom in the defined system is also shown.

- If this is zero, you can perform a COMPUTE\_EQUILIBRIUM command.
- If it is larger than zero, some more conditions are required, and you must further set additional ones, using SET\_CONDITION or CHANGE\_STATUS.
- If it is negative, then too many conditions are defined and the unnecessary ones conditions need to be removed using the SET\_CONDITION command (with a value of NONE for the to-be-deleted condition) or the CHANGE\_STATUS command (i.e. changing a FIXED status of a phase to another type of phase status, ENTERED or DORMANT or SUSPENDED).

Syntax	LIST_CONDITIONS
Example	<pre>P=100000, T=800, N(NI)=1E-1, N=1 FIXED PHASES FCC_A1=1 LIQUID=0 DEGREE OF FREEDO 0</pre>

## LIST\_EQUILIBRIUM



This command is available with the POLY and ED-EXP modules.

The result (always in SI units) from the last calculated equilibrium is listed on screen or in a text file.



You can also execute this command if no calculation is made or if the calculation fails. Make sure to interpret the results accordingly.

Syntax	LIST_EQUILIBRIUM
Prompt	Output to screen or file /Screen/: <File name> The name of the text file where the list of the calculation results shall be written.
	Options /VWCS/: <Option(s)> Select the output units and formats by optionally specifying a combination of these letters. <ul style="list-style-type: none"> <li>• <i>Fraction order</i>: V means VALUE ORDER; A means ALPHABETICAL ORDER.</li> <li>• <i>Fraction type</i>: W means MASS FRACTION; X means MOLE FRACTION.</li> <li>• <i>Composition</i>: C means only COMPOSITION; N means CONSTITUTION and COMPOSITION.</li> <li>• <i>Phase</i>: S means including only STABLE PHASES; P means including ALL NON-SUSPENDED PHASES.</li> </ul>
	Default options are VWCS. If the output fraction type should be in mole fraction (rather than mass fraction), then give VXCS or type X (implying that in this case the options V, C and S are accepted as the listing manners for fraction order, composition and phase). If accepting all the default options, or if accepting all the altered options that had already changed when using this command previously, you can type L_E,,, or LIST_EQUILIBRIUM ,,, <div data-bbox="370 1388 417 1444" data-label="Image"> </div> <p>LIST_EQUILIBRIUM displays ORD (ordered) or DISORD (disordered) (when available). See <a href="#">CHANGE_LEGEND</a> for details.</p>

---

## LIST\_INITIAL\_EQUILIBRIA

 This command is for the POLY module.

List all the equilibria added with ADD\_INITIAL\_EQUILIBRIUM.

Syntax	LIST_INITIAL EQUILIBRIA
	All the initial equilibria are used for the MAP (and STEP) calculations.

---

## LIST\_PHASE\_ADDITION



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

The values set as additional contributions (given by `PHASE_ADDITION`) to Gibbs energy  $G_m$  (J/mol formula unit) to all the phases (stoichiometric or solution) are listed at the current calculated equilibrium.



`LIST_PHASE_ADDITION` has no additional prompt.

## LIST\_STATUS



This command is for the POLY module.



There is also an ED-EXP module command with the same name.

List the status of components, species or phases.

Syntax	LIST_STATUS
Prompt	<p>Specify what to list:</p> <p>Option /CPS/ :&lt;keyword(s)&gt;</p> <p>keyword = C or P or S, or any combination</p> <ul style="list-style-type: none"> <li>• C: list component status</li> <li>• P: list phase status</li> <li>• S: list species status</li> <li>• CS: list both components and species</li> </ul> <p>Default is CPS.</p>
	<p>The results depend on the key word specified in the options for CHANGE_STATUS, a table with the current status of phases or species or components, or the combinations, is shown:</p> <ul style="list-style-type: none"> <li>• For components, the statuses and reference states are listed.</li> <li>• For species, only the status is listed.</li> <li>• For ENTERED and FIXED phases, the status, driving forces and equilibrium amounts are listed.</li> </ul>
	<div style="border: 2px solid blue; padding: 10px;">  <p>The metastable phases are listed in descending order of stability. To avoid long outputs only 10 metastable phases (in ENTERED status) are listed by lines, while all other less stable phases are merged onto one line. For DORMANT phases, their phase names and driving forces are listed. For SUSPENDED phases, only the phase names are listed.</p> </div>

## LIST\_SYMBOLS

- >\_ This command is for the POLY module.
- >\_ There are also POST and ED-EXP module commands with the same name.

Use this command to list the definitions for all constants, functions, variables and tables. In order to find the value of a function or variable, use SHOW\_VALUE or EVALUATE\_FUNCTIONS. A table is tabulated with the TABULATE command.

Syntax	LIST_SYMBOLS
	The defined variables are listed up together with the defined functions, but variable names are followed by a percentage sign %.

## LOAD\_INITIAL\_EQUILIBRIUM



This command is for the POLY module.

Copies all conditions and calculated results from a specific added initial equilibrium to the current equilibrium. The current conditions and calculation results are lost, and the newly loaded initial equilibrium point gets into the POLY workspace.

Syntax	LOAD_INITIAL_EQUILIBRIUM
Prompt	Number: <Number of an initial equilibrium> Specify the number of an initial equilibrium to be loaded as current. The number can be found with LIST_INITIAL_EQUILIBRIA.

## MAKE\_COMPONENT\_ENTERED

 This command is for the POLY module.

The MAKE\_COMPONENT\_ENTERED command sets the status for the components to be ENTERED in the defined system for all the sequential calculations (single-point, stepping, and mapping) in equilibrium or local/partial equilibrium states. ENTERED means that the component(s) are included in the calculation and these are stable if that minimizes the total energy.



This command has the same results as when using [CHANGE\\_STATUS](#) Entered option.

Syntax	MAKE_COMPONENT_ENTERED
Prompts	Name(s) <Name(s) of the components> The names of the components to set to ENTERED (all on one line). A comma or space must be used as separator. An asterisk, "*", can be used to denote all components.

## MAKE\_COMPONENT\_SUSPENDED

 This command is for the POLY module.

The MAKE\_COMPONENT\_SUSPENDED command sets the status for the components to be SUSPENDED in the defined system for all the sequential calculations (single-point, stepping, and mapping) in equilibrium or local/partial equilibrium states. SUSPENDED means that the component(s) are not considered in the calculation. Any conditions that depend on the suspended components are automatically removed.



Removing relevant conditions is unique to this command and does not occur when suspending components with the [CHANGE\\_STATUS](#) command and `Suspended` option.

Syntax	MAKE_COMPONENT_SUSPENDED
Prompts	Name(s) <Name(s) of the components> The names of the components to set to SUSPENDED (all on one line). A comma or space must be used as separator. An asterisk, "*", can be used to denote all components.

This command is useful when an element has a very low amount because you can now ignore it in the calculation instead of reading the entire system from the database. The command is also of use when working in an API with many calculations.

## MAJOR\_CONSTITUENTS



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

Use this to set the major constituent(s) of a composition sets in a miscibility gap of a solution phase. Normally, the major constituents are specified when a new composition set is created by the `NEW_COMPOSITION_SET` option; but for the first composition set, this option may be needed before using `NEW_COMPOSITION_SET`.

The following prompts are to specify the major constituent(s) on each of the sublattice sites for a composition set of a certain solution phase:

Phase name: <name of a phase>

Specify the name of the solution phase with a new set of major constituents.

Composition set number /1/: <#>

The default value for the composition set number (#) is usually /1/ as the other composition sets are given major constituents when creating them. Each phase has initially one composition set.

Major constituent(s) for sublattice 1: /XX/: <YY>

The major constituents (YY) on each sublattice of a solution phase can be given. This may simplify giving start values when calculating the equilibrium as phases with miscibility gaps should have different major constituents for each composition set.

This question is repeated for each sublattice in the phase.

## MAP



This command is for the POLY module.

This command starts the mapping procedure for making a calculation of phase diagrams in a defined multicomponent heterogeneous system, from one or more initial equilibria. A phase diagram is usually mapped within a specific space that is constructed by two (or more) defined independent mapping axis-variables.

Syntax	MAP
	A phase diagram consists of mapped phase boundary lines/curves; on one side of each such phase-boundary line/curve, the amount of one specific stable phase is zero (i.e. the zero-fraction lines). From a single MAP calculation, many different types of phase diagrams in the defined multicomponent heterogeneous system can be plotted, with some desired properties (that vary along the calculated phase-region boundaries) plotted as X/Y-axis variables. All different types of phase diagrams are generated by the mapping calculations through this command.

Normally, you need to have calculated at least one initial equilibrium point and have also defined at least two independent varying variables (i.e. the controlling conditions in the system) that are set with SET\_AXIS\_VARIABLE. You can also have three, or four or maximum five independent varying variables that are defined by the SET\_AXIS\_VARIABLE command).

This lists the current values of each of the independent axis variables for each of the calculated equilibrium points along each of the mapped phase boundaries, and also lists the corresponding information when the set of stable phases changes.

The Global Minimization Technique is by default enforced in this command while it can be disabled if having decisively switched it off (or possibly in some special modules) through changing the minimization option using the ADVANCED\_OPTIONS → MINIMIZATION\_OPTION command-sequence. You can choose how often to do a Global Minimization test (via the Global Test Interval option) in the ADVANCED\_OPTIONS STEP\_AND\_MAP command-sequence.

During a MAP calculation, the values of mapped axis-variables for presenting each phase boundary (lines/points) of the calculated phase-regions are listed, and the corresponding stable-phase sets are shown up.



You can terminate the mapping of a line by pressing a single <Ctrl-A> (Windows) or <Ctrl-C> (Linux). This can be useful in order to stop a longish calculation without losing what is already calculated.

If there is any convergence problem in finding a stable solution at a certain stage during a calculation procedure enforced by this command, these messages display on screen:

```
Convergence problems, increasing smallest site-fraction from 1.00E-30
to hardware precision 2.00E-14. You can restore using SET-NUMERICAL-
LIMITS
```

This implies that smallest site fraction in the current POLY3 workspace is automatically increased from the default value 1.00E-30 to the hardware-dependent precision (under Linux, as 2.00E-14). For other subsequent POLY-module calculation in the current TC run, you can use SET\_NUMERICAL\_LIMITS to restore or reset the smallest site fraction to the previous or another preferred value, as well as to reset other numerical limits.

In particular, for phase diagrams with tie-lines in the plane (i.e. most binary systems and ternary isotherms), there is a MAP procedure which checks for the best phase to use as axis variables in order to ensure reasonable increments between the tie-lines. This produces smoother curves and also gives a better stability in finding adjacent phase-regions.

Complex miscibility gaps of various solution phases are automatically detected during a mapping calculation, and two or more composition sets for each of such solution phases are automatically generated as well, through an automatic Global Minimization Test procedure (by specifying a Global Test Interval value and confirming the `Automatically add initial equilibria` in the ADVANCED\_OPTION STEP\_AND\_MAP command-sequence). As a particular case, such an automatic Global Minimization Test ensures that you can start calculating from high temperatures in an austenitic steel (metallic FCC\_A1 solution phase) where the MC carbides/nitrides/carbonitrides (i.e. the C-/N-rich sides of the FCC\_A1 miscibility-gap, often (while not always) being referred to as FCC\_A1#2, FCC\_A1#3, etc.) are not stable, and during the MAP command the MC carbides/nitrides/carbonitrides may first become metastable with a composition different from the metallic FCC\_A1 phase and later also stable. The advanced mapping procedure (enforced by the Global Minimization Technique) inside the Thermo-Calc software (both the Console Mode and Graphical Mode) can efficiently and effectively handle complex miscibility gaps in multicomponent heterogeneous systems.

To get a complete phase diagram, sometimes it may be necessary to have multiple starting equilibrium-points, and/or to have added multiple initial-equilibrium points (through ADD\_INITIAL\_EQUILIBRIUM) inside various phase-region boundaries under certain direction(s).

## NEW\_COMPOSITION\_SET



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

A solution phase that can exist with two (or more) different compositions simultaneously must have two (or more) composition sets in its phase descriptions. Normally the database creates as many composition sets as is necessary but use this command to add or delete more composition sets. A complex solution phase in a defined multicomponent system can have up to 9 different composition sets for the purpose of appropriately handling its possible miscibility gap(s) under various temperature-pressure-composition conditions.

This option is unnecessary if the Global Minimization Technique is in use, as it can automatically detect all possible miscibility gap(s) for complex solution phases (normal or disordered/ordered phases) and then automatically add required composition sets for such phases in the defined system.

This option can automatically create composition sets also for disordered phase when it is executed for the ordered phase.

If you have your own data file, this option must be used to indicate that a solution phase can have a miscibility gap (or may exhibit some complex phase separations as to more than two composition sets), unless that the possibly-additional composition set(s) for the solution phase must have already been added inside the TDB file (through a `TYPE_DEFINITION` command to amend composition-set) or been manipulated through the `GIBBS` module (using the `AMEND_PHASE_DESCRIPTION COMPOSITION_SET` command-sequence). This option asks for default major constituents for the new sets and it is important that this is set correctly, otherwise the test for miscibility gaps may fail.

The following prompts are to specify the additional composition set(s) to handle possible miscibility gap(s) of a certain solution phase:

```
Phase with miscibility gap: <Name of a phase>
```

Specify the name of the phase with miscibility gap.

```
New highest composition set number /2/: <#>
```

The default value is usually one higher than the current value. Each phase has initially one composition set. If a lower value is given, composition sets are deleted. You cannot take away the first composition set.

A message displays to show that you need to specify the composition for the composition set # (2,3,...).

```
Major constituent(s) for sublattice 1: /XX/: <YY>
```

The major constituents (YY) in each sublattice can be given. This may simplify giving start values when calculating the equilibrium as phases with miscibility gaps should have different major constituents for each composition set.

This prompt is repeated for each sublattice in the phase, sometimes even for all sublattices in the first composition set if such major constituents have not been specified.

---

## OUTPUT\_FILE\_FOR\_SHOW



This is an option that is available with the `ADVANCED_OPTIONS` command with the POLY and ED-EXP modules.



### ADVANCED\_OPTIONS

The name of a text file is asked for and all the results output from the command `SHOW_VALUE` is written to this file.

When you use this command, a window opens with a default name for the file `tc_show.dat`. Choose a location to save the file and click **Open**.

## PARAEQUILIBRIUM



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

This calculates a paraequilibrium between two specific phases in an alloy system with one or more interstitial component(s) as fast diffusion species. Under the paraequilibrium state, two partially-equilibrated phases have the same chemical potential (but different contents) for one or more interstitial components (such as C, N, O, S, etc., as individual or combined)], along varied temperature or along a composition variable (of the matrix or one substitutional component) which has already set as the stepping variable with `SET_AXIS_VARIABLE`.

To ensure a successful point calculation of paraequilibrium state between two specific phases in a defined alloy system, it is important that you first have made a single-point equilibrium calculation with an initial overall composition in the current system before performing this advanced-option calculation; however, it is unnecessary to obtain an equilibrium in which either one or both of the target phases is stable. The initial overall composition must have a reasonable setting for the desired paraequilibrium calculation for the two target phases. This is especially true for cases where there are more than one interstitial components to be considered in the paraequilibrium state, because different interstitial components (for example C and N combined) may have significant different behaviors as partitioning into different structured phases; otherwise, for one chosen interstitial component the initial overall composition is OK for the paraequilibrium calculation between the specified two phases, but for other chosen interstitial component(s) it might be impossible to calculate the paraequilibrium state.

Note the following:

- Always check if the chosen phases A and B have the exactly same definition of elements and if the chosen interstitial components are all in the vacancy sublattice sites of the two phases; otherwise the program cannot find the paraequilibrium state (as it is impossible to correctly calculate u-fractions).
- Always have a comprehensive understanding of the normal phase diagram for the currently investigated system, so that you make the appropriate choice of the phase pair and starting bulk composition for the system.
- Always set the status of the chosen interstitial components as `SPECIAL` using the `POLY` command: `CHANGE_STATUSComponent <interstitial component> = SPECIAL`. By doing this, you get a clear picture of u-fractions of various substitutional and interstitial components, which are different from the overall composition in the

system. The SPECIAL status means that specified component(s) are not included in summations for mole or mass fractions. Therefore, all the composition variables plotted from paraequilibrium calculations are u-fraction related quantities.

## Settings

Name of first phase: <Phase A>

Name of second phase: <Phase B>

The names of the two target phases A and B, between which the paraequilibrium state is to be calculated, must be entered subsequently or on the same (first) line at once then separated by an empty space, e.g. FCC#1 BCC or FCC#2 M23C6.



You need to understand what you are dealing with in terms of calculating a paraequilibrium state between the two specified phases.

Specifically, there are four distinguished cases to understand: (1) both chosen phases must have similar interstitial/vacancy sublattices where the fast-diffusion interstitial component (s) occupy; (2) the choice on the target phase pair must be reasonable for the defined system and specified initial overall composition; (3) both target phases should have phase constitution definitions that cover all the defined substitutional and interstitial components of the current alloy system; or (4) it is impossible to calculate the paraequilibrium state between the target phase pairs with given interstitial component(s) in the currently defined system.

Fast diffusing component: /c/: <interstitial component(s)>

Fast diffusing component: /none/: <interstitial component(s)>

The name(s) of the fast-diffusing component(s) (C as the default single component) must be given at the above prompts subsequently or at the same (first) prompt. It is possible to specify more than one interstitial component as fast diffusion species.

Note the following:

Such specified interstitial component(s) must be appropriately defined according to the phase constitution definitions of the two selected phases: these must be located on the interstitial/vacancy sublattices in both chosen phases;

If there is only one fast-diffusing component which is carbon, press the <Enter> key to accept the default input at the first prompt; if the single fast-diffusing component is another element (e.g. N), type its name at the first prompt;

If there are two or more fast-diffusing components (e.g. C and N), type their names at the above prompts subsequently or at the same (first) prompt (separated by an empty space, such as C N);

To finish the input of fast-diffusing elements, accept NONE at a repeated prompt, i.e. by pressing <Enter> key to start the paraequilibrium point calculation;

If NONE or a non-existing component name is typed at the first prompt, it means no back diffusion is to be considered, and the paraequilibrium calculation is thus canceled entirely.

If the paraequilibrium state between the two specified phases is successfully calculated, the messages displays e.g.

```
NP(FCC) = 0.3586 with U-fractions C = 2.71821E-02  N = 4.1548129E-03
NP(BCC) = 0.6414 with U-fractions C = 7.10061E-04  N = 2.3781027E-04
All other compositions the same in both phases
Note: LIST-EQUILIBRIUM is not relevant
```

The first and second lines list the phase amounts expressed in mole-percent [NP(phase)] and the contents of the interstitial components C and N in a specific phase expressed in the so-called u-fractions [u-f(phase,C) and u-f(phase,N)], for the phase A (in this case as FCC) and phase B (in this case as BCC), respectively. The third line states that the compositions of the matrix component and all the remaining compositions (regarding substitutional components) in both the target phase A and target phase B are the same at the current paraequilibrium state, while these are not shown on screen. The last line indicates that after this advanced-option calculation the LIST\_EQUILIBRIUM command is irrelevant and does not list the paraequilibrium state for the system at the current condition.

However, if the single-point calculation of the paraequilibrium state between the two specified phases has failed, these messages display:

```
*** Error 4 in ns01ad
*** Numerical error
```

This implies that the chosen target phase pair may be unreasonable for the defined alloy system or for the defined initial overall composition, or one or both phases may have inappropriate phase constitution definitions regarding the specified interstitial component (s). Then, you must either modify the settings of initial overall composition or specify the reasonable target phase pair with an appropriate choice of the fast diffusion interstitials in the defined alloy system.

## PHASE\_ADDITION



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

Sometimes it is interesting to add a constant contribution to the Gibbs energy of a phase (stoichiometric or solution). This can be done in the Database module (`DATA`) or Gibbs Energy module (`GIBBS`), for a stoichiometric phase, or for a pure end-member in a solution phase. However, if the addition is related to the equilibrium state, for example, strain energies or surface energies, interfacial energies, or deformation energies, it may be more convenient to have this quantity related to the equilibrium state rather than the thermodynamic data.

Give a value of an addition to the Gibbs energy of a phase. The value should always be constant (implying that the addition is not a function of phase composition or temperature-pressure conditions in the equilibrium state) and always be given in the unit of J/mol formula unit of the phase.

The following prompts are to specify the additional energy term (always being a constant) of a given phase.

Phase name: <Name of a phase>

Specify the name of the (stoichiometric or solution) phase with the addition.

Addition to G per mole formula unit: <xxxxxx>

The value (`xxxxxx`) given is added to the Gibbs energy of the (stoichiometric or solution) phase. It can represent a nucleation barrier, surface tension, elastic energy, etc.



It is not composition-, temperature-, or pressure-dependent.

## POST



The POST module (post-processor) is a submodule to the POLY module and has its own set of commands. The TAB, BIN, TERN, POT, SCHEIL and POURBAIX modules use the POST module features as part of the automatic generation of the plots.



[POST Module Commands](#)



[Using the POST Module](#)

Syntax	POST
	Switches to the POST module, which has its own command set.

---

## PRESENT\_PHASE



This is an option that is available with the `ADVANCED_OPTIONS` command with the POLY and ED-EXP modules.



### ADVANCED\_OPTIONS

Specify the name of the present phase and the name of the phase that should be present at all calculated equilibria:

Present phase:

Phase name: <Name of a phase>

## READ\_WORKSPACES

 This command is used in both the POLY and GES (GIBBS) modules.

 [Workspace Files](#)

The POLY3 and GIBBS workspaces and the calculated results from the MAP and STEP commands can be READ from a file where they are saved with SAVE\_WORKSPACES. Such a \*.POLY3 file is not printable.

Syntax	READ_WORKSPACES
Options	<p><code>File name</code> is the name of a saved POLY3-file where the POLY3 and GIBBS workspaces shall be read from must be specified. You do not need to type the extension if it is the default *.POLY3, otherwise type the whole POLY-file name.</p> <p> A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.</p> <p>A window opens so that the path (in the <b>Look in</b> field) and <b>File name</b> can be specified. The <b>Files of type</b> (i.e. POLY3) cannot be changed. Click <b>Open</b> to open the POLY3 and GIBBS workspaces from the saved *.POLY3 file.</p>
	<p>When reading back an original POLY3 workspace that has already been saved as an *.POLY3 file in the current Thermo-Calc (Console Mode) run or had been read from an existing POLY3 file under the current work area, while some additional changes in the settings may be made but do not need to be kept in further steps in the current Thermo-Calc (Console Mode) run or any diagram is plotted in the POST module, you can type <code>READ,,</code> or <code>READ_WORKSPACE ,,</code></p>

## REINITIATE\_MODULE



This command is for the POLY module.

This command makes sure that the whole POLY module (and thus the entire POLY workspace) is reinitiated to the state as it was when first entered.

Syntax	REINITIATE_MODULE
	All the defined components, defined conditions, changed status, entered symbols, defined independent axis-variables, calculated starting equilibrium-points, added initial equilibrium points, stepped/mapped equilibrium data, and so forth, are removed completely. The saved file name is restored to the default.

## SAVE\_WORKSPACES



This command is available for both the POLY and GES (GIBBS) modules.



### [Workspace Files](#)

You can save the current status and workspaces, including thermodynamic data, conditions, options and results from a single, stepping or mapping calculation on a \*.POLY3 file. The workspaces are saved to a file with this command. This is useful so you have access to the workspace for later use or when you need to terminate the current Console Mode session.

- In the Gibbs (GES) module workspace, all thermochemical data are stored.
- In the POLY module workspace, all the last set of conditions and equilibrium state, changed status, entered symbols, advanced options, defined stepping/mapping variables, added initial equilibria, stepped/mapped results, etc., are stored, so it also contain the GES workspace.
- When you are in a response-driven module such as POTENTIAL or SCHEIL for example, a workspace file is automatically opened. In the workspace file, system definitions, conditions for the calculation, calculation results, and plot settings are saved. The file is saved in the current working directory, and is named after the name of the module that created it. For example, the POTENTIAL module saves a workspace file called `POT.POLY3` ; the POURBAIX module saves a file called `POURBAIX.POLY3` and so forth.



After a SAVE\_WORKSPACES command, you can always come back to exactly the state you had when you issued the command by giving a READ\_WORKSPACE command. For example, after saving the POLY and GIBBS workspaces to a file, you can leave the program and at a later time READ the file and continue from the saved state.

### *Important Information About Map and Step Commands*



A STEP or MAP command automatically saves the workspace with the most recently specified name. Do not use the SAVE\_WORKSPACES command after a MAP or STEP command.

The results from the STEP or MAP commands are destroyed by the SAVE\_WORKSPACE command. You can append several results obtained by sequential STEP or MAP calculations without destroying the previous results, whilst SAVE\_WORKSPACE erases them all. Keeping this in mind is important and useful particularly for calculating various isothermal (or isoplethal) sections and plotting them on the same diagram in a single Thermo-Calc run.

To suspend some of the STEP or MAP results, use the AMEND\_STORED\_EQUILIBRIA command.

Syntax	SAVE_WORKSPACES
Options	<p>A <code>file name</code> must be specified. The default extension of the POLY workspace file is *.POLY3 but you can have any other file extension.</p> <p> Linux and Mac are case sensitive so ensure you enter <code>.POLY3</code>.</p> <p>A <b>Save</b> window displays if a file name is not given after the command, so that the path (in the <b>Save in</b> field) and <b>File name</b> can be specified. The <b>Files of type</b> cannot be changed.</p> <p>If there is already a file with the same file name under the directory a warning message displays. Click <b>OK</b> to overwrite the current POLY or GIBBS workspace onto the existing file. Click <b>Cancel</b> to return to the module. You can use the SAVE command with an unspecified name (i.e. through <code>SAVE , , ,</code> command-sequence) to save the POLY or GIBBS workspace.</p> <div style="border: 2px solid blue; padding: 10px; margin-top: 10px;"> <p> When saving a POLY3 workspace under a name that already exists under the current work area, which is saved by default (after running a special module, e.g. BIN, TERN, POT, SCHEIL and POURBAIX) or in an earlier stage of the current Console Mode session or in a previous run (which is READ into the current POLY3 workspace), you can type <code>SAVE , , y</code> or <code>SAVE_WORKSPACE , , y</code>. However, this must also be avoided if some results from previous MAP or STEP calculations shall not be destroyed.</p> </div>

## SELECT\_EQUILIBRIUM



This command is available with the POLY and ED-EXP modules.

If you create more than one initial equilibrium (during data-assessments using the PARROT/ED\_EXP modules), you can switch between them using this command.

Syntax	SELECT_EQUILIBRIUM
Prompt	Number /Next/: <Choice on equilibrium> Options: FIRST, LAST, NEXT, PREVIOUS or PRESENT. Most commands affect only the PRESENT equilibrium. However, the REINITIATE_MODULE and DEFINE_COMPONENTS commands remove all the stored equilibria.

## SET\_ALL\_START\_VALUES



This command is available for the POLY and ED-EXP modules.

Set start values, e.g. if the calculation fails or if you have a miscibility gap or ordering. If temperature and pressure are not conditions, you are asked for values of them. Then for each phase prompt on if it should be stable and on its constitution.

Syntax	SET_ALL_START_VALUES
	<p>T /XXXX/: &lt;Temperature in K&gt;</p> <p>If the temperature is not a condition, supply a guess of its final value (in K).</p>
	<p>P /100000/: &lt;Pressure in Pa&gt;</p> <p>If the pressure is not a condition supply a guess of its final value (in Pa).</p>
Prompts	<p>Automatic start values for phase constitutions? /N/: &lt;Y, N or F&gt;</p> <ul style="list-style-type: none"> <li>For N, enter an initial amount of each entered phase and the major constituents or site fraction of each constituent.</li> <li>For Y, the program automatically sets the start values for phase constitutions in all possible phases.</li> <li>F is for force. In some cases the calculation fails because impossible conditions are set, e.g. <math>w(C)=1.5</math> [<math>w(C)</math> is mass fraction and must thus be less than unity. The program tries to fulfill this condition by putting maximum amount of carbon in all phases, but can fail anyway. When you detect the error and set <math>w(C)</math> to 0.015, the calculation may still fail because it could start from the previous values. To get back to 'fresh' start values, choose F.</li> </ul>
	<p>Should &lt;phase&gt; be stable /N/: &lt;Y/1 or N/2&gt;</p> <p>A guess if this phase should be stable or not is requested. All entered phases are prompted for this question and next two.</p> <div style="border: 2px solid red; padding: 10px; margin: 10px 0;"> <p> You cannot have more phases stable than you have components, but at least one (which dissolves all constituents) must be set as stable. For backward compatibility, this question can be answered by 1 (for Yes) or 0 (for No).</p> </div> <p>The phase name may have a hash sign # followed by a digit, e.g. BCC_A2#2. For phases with miscibility gaps, there should be two phases with the same name but with different numbers after the hash sign.</p>
	<p>Major constituent(s): &lt;Name of major constituent(s) in the phase&gt;</p> <p>The constituent with the largest fraction in the phase should be specified. If there are more than one constituent with a large fraction, give them all on the same line. If the default major constituents should be used answer with an asterisk (*). By giving \$ the constitution is not changed. If there should be no major constituent give NONE and or if the major constituent(s) are improperly specified, you are asked for individual fractions in the phase.</p>

Syntax	SET_ALL_START_VALUES
	<pre data-bbox="358 258 948 281">Y(&lt;phase&gt;,&lt;constituent&gt;) /.XXXXXXXXX/: &lt;.YYYYY&gt;</pre> <p data-bbox="358 300 1398 359">The current value (.XXXXXXXXX) is default. You can accept the default by pressing &lt;Enter&gt; or give a new value (.YYYYY).</p> <p data-bbox="358 378 1398 495">The phase name or constituent name may have a hash sign # followed by a digit, e.g. Y(BCC_A2#2,FE), Y(BCC_A2#2,C#2). For phases with miscibility gaps, there should be two phases with the same name but with different numbers after the hash sign. For phases with sublattices, the constituents in sublattice 2 or higher is also be suffixed with a hash sign # followed by a digit.</p>

## SET\_AXIS\_VARIABLE

In order to calculate a diagram, set at least one axis variable in a stepping calculation, or at least two axis variables in a mapping calculation. For property diagrams, one axis is enough; for phase diagrams two or more are necessary. Any condition that can be set to calculate an equilibrium can be used as an axis variable (with its lower and upper limits and step length) by using SET\_AXIS\_VARIABLE, and the POLY program does, after a STEP or MAP command, vary the value of the condition between the limits set on its related axis variable. As an extraordinarily unique and powerful feature of the Thermo-Calc software, up to five independent axis variables can possibly be used in a mapping calculation of a multicomponent heterogeneous system; however, the axis numbers 3, 4 and 5 must have chemical potentials of components (or temperature or pressure) as conditions.

Syntax	SET_AXIS_VARIABLE
	<p>Axis number /#/ : &lt;An axis number&gt;</p> <p>Specify a number between 1 and 5. The axis numbers 3, 4 and 5 must have chemical potentials of components (or temperature or pressure) as conditions.</p>
Prompts	<p>Condition /None/ : &lt;One condition&gt;</p> <p>Here the condition that should be varied along the axis must be given. The condition is specified as in SET_CONDITION, for example <math>w(C)</math> for mass fraction of carbon. By accepting NONE, the axis is removed.</p>
	<p>Min value /0/ : &lt;Min value&gt;</p> <p>Specify the minimum value of the axis condition.</p>
	<p>Max value /1/ : &lt;Max value&gt;</p> <p>Specify the maximum value of the axis condition.</p>
	<p>Increment /.025/ : &lt;Step length&gt;</p> <p>Specify the maximum step length. By default, this is 1/40 of the total axis length.</p>

You can give SET\_AXIS\_VARIABLE without having set a condition on the axis variable. Under such circumstances, the relevant condition is automatically created and the value set between the minimum and maximum axis limits; however, as a side effect, the POLY module creates two conditions,  $P=1e5$  and  $N=1$  (these have not been defined as a condition yet), in case you set an axis variable which is not already a condition.

You can use a logarithmic axis during calculations. This is useful for low fractions like in a gas phase where  $1e-7$  to  $1e-2$  might be an interesting range. The pressure is also suitable for logarithmic step. You specify the logarithmic axis by giving an asterisk \* after the increment value.



The increment in this case is treated as a factor.

For example,

```
S-A-V 1 P 1E5 1E25 5*
```

This makes axis 1 a logarithmic axis where the difference between two calculated values makes a factor 5.



The factor must be larger than 1.0.

In some cases, such as when the DEFINE\_DIAGRAM command is used or a special advanced module (e.g. BIN, TERN, POT, SCHEIL or POURBAIX) is called, some axis variables are automatically set by the program, not necessarily by this command.

## SET\_CONDITION

 This command is available with the POLY and ED-EXP modules.

Specify the equilibrium conditions for the calculation. All kinds of state variables, as well as most of the  $M/W/V/F$ -suffixed state variables (for normalization) and  $R$ -suffixed state variables (with respect to chosen reference states), can be used as conditions. In addition to temperature and pressure conditions, a multicomponent system can have mixed types of conditions. You should repeat this command for a multicomponent system until the degree of freedom in the defined system becomes zero. When a FIXED phase status is used on a specific phase (use CHANGE\_STATUS), it is equivalent to one condition (implying that particular phase is stable in the defined system).

Syntax	SET_CONDITION
	<p>Each condition must be given explicitly, but can be given on the same line (separated by a comma (,) or an empty space) or on separate lines with each one started with the command.</p> <p><b>Example</b></p> <pre>SET_COND T=1273, P=1E5, W(C)=.0015, X(LIQ,Cr)=.22, ACR(N)=.2</pre> <p>or</p> <pre>SET_COND T=1273, P=1E5 SET_COND W(C)=.0015, X(LIQ,Cr)=.22 ACR(N)=.2</pre> <p>In this example, the temperature is set to 1273 K, the pressure to 1 bar (1E5 Pascal), the mass (weight) fraction of C to 0.0015 and the mole fraction of Cr to 0.22 and the activity of N to 0.2.</p>
Prompt	<p>State variable expression: &lt;State variable name or linear expression&gt; or a linear expression of state variables.</p> <p>Some of the state variables that can be used in conditions are:</p> <ul style="list-style-type: none"> <li>• <b>T</b>: temperature in the system (in K)</li> <li>• <b>P</b>: pressure in the system (in Pascal)</li> <li>• <b>N</b>: system size (mole number in moles)</li> <li>• <b>B</b>: system size (mass in grams)</li> <li>• <b>N</b>(&lt;component&gt;): mole number of a component in the system</li> <li>• <b>X</b>(&lt;component&gt;): mole fraction of a component in the system</li> <li>• <b>W</b>(&lt;component&gt;): weight fraction of a component in the system</li> <li>• For the SET_CONDITION command this is <b>ACR</b>(&lt;component&gt;): activity of a component in the system. For the SET-ALTERNATIVE_CONDITION command, this is <b>ACR</b>(&lt;phase&gt;,&lt;component&gt;) (activity of a component in a phase)</li> <li>• For the SET_CONDITION command this is <b>MUR</b>(&lt;component&gt;): chemical potential of a component in the system. For the SET-ALTERNATIVE_CONDITION command, this is <b>MUR</b>(&lt;phase&gt;,&lt;component&gt;) (chemical potential of a component in a phase)</li> </ul>

Syntax	SET_CONDITION
	<ul style="list-style-type: none"> <li>• <math>X(\langle\text{phase}\rangle, \langle\text{component}\rangle)</math>: mole fraction of a component in a phase</li> <li>• <math>W(\langle\text{phase}\rangle, \langle\text{component}\rangle)</math>: weight fraction of a component in a phase</li> <li>• <math>ACR(\langle\text{species}\rangle, \langle\text{phase}\rangle)</math>: activity of a species referred to a phase</li> <li>• <math>MUR(\langle\text{species}\rangle, \langle\text{phase}\rangle)</math>: chemical potential of a species referred to a phase</li> <li>• <math>H</math>: enthalpy in the system (in J)</li> <li>• <math>HM(\langle\text{phase}\rangle)</math>: enthalpy of a phase (in J/mol)</li> </ul> <p>There are many state variables that can be used in conditions. For more information, enter an <code>INFO STATE_VARIABLES</code> command. A condition is normally a value of a single state variable with its value.</p> <p><b>Example</b></p> <pre>T=1273.15 P=1E5 X(C)=.002 W(CR)=0.5 ACR(CR)=0.85 X(FCC,C)=.001 H=-250000 HM(BCC)=-225000</pre> <p>A condition can also be a value of a linear expression involving more than one state variable. For example,</p> <pre>X(LIQ,S)-X(PYRR,S)=0</pre> <p>This means that it is a condition that the mole fraction of <code>s</code> component should be the same in the LIQUID and PYRRHOTITE phases. In practice, that should be the congruent melting point.</p> <div style="border: 2px solid blue; border-radius: 10px; padding: 10px; margin-top: 10px;">  After the equal sign only a numeric value is allowed in the expression. </div>
	<p>Factor: &lt;A factor for the state variable, or a continuation&gt;</p> <p>This question means you did not answer the previous question. The program is then expecting a single state variable or a complete state variable expression, or the numeric factor in an expression with only one state variable. In a state variable expression a state variable may be preceded by a constant factor. An example of this is:</p> <pre>2*MUR(Fe)+3*MUR(O)=-35000</pre> <p>This means that it should be a condition that two times the chemical potential of <code>Fe</code> plus three times the chemical potential of <code>O</code> should be -35000 J/mol.</p>
	<p>State variable: &lt;A specified state variable, or a continuation&gt;</p> <p>This question is prompted if a single state variable name has not given in either the prompt <code>State variable expression</code> or <code>Factor</code>, or a state variable expression is given but the expression is incomplete, for example, <code>T-</code> or <code>2*MUR(Fe)+</code>, for which the program is then expecting a continuation of the unfinished expression. You need to specify a state variable or a complete state variable expression, or complete the unfinished state variable expression. If a numeric factor is given before</p>

---

Syntax	SET_CONDITION
	this prompt, only one state variable can be specified; otherwise, the program only takes the first state variable to complete the expression (i.e. the factor times the state variable).
	Value /X/: <A numeric value, a constant or a variable>  The value of the condition. This can be a numeric value, a constant or a variable. A suggestion is given as the default value. The special value <code>NONE</code> means that the condition is removed; for example <code>T=NONE</code> takes away the temperature condition.

## SET\_INPUT\_AMOUNTS

Specify how a system is made up from mixing of various substances. It is useful with a substance database. In the C-H-O-N system, you can, for example, give:

```
S-I-A N(H2)=10, N(H2O)=25, N(C1O2)=5, N(N2)=100
```

The POLY module automatically converts this into conditions for the current set of components. In the case when the elements are defined as the components, the command above is equivalent to

```
SET-CONDITION N(H)=70, N(O)=35, N(C)=5, N(N)=200
```

Syntax	SET_INPUT_AMOUNTS
Prompts	<p>Quantity: &lt;N(&lt;specie&gt;) or B(&lt;specie&gt;)&gt;</p> <p>You can give the amount also preceded with an equal sign = [e.g. N(H2)=10 or B(H2O)=1000], or press &lt;Enter&gt; for the next prompt on the amount of the quantity.</p>
	<p>Amount: &lt;Value of the quantity&gt;</p> <p>Specify the numerical value of the quantity.</p> <div style="border: 2px solid blue; padding: 5px; margin-top: 10px;">  You can give negative amounts in the SET-INPUT-AMOUNTS command.         </div>

## **SET\_INTERACTIVE**



In the POST module this is called SET\_INTERACTIVE\_MODE.

Use the SET\_INTERACTIVE command to use it in demonstration or macro files to stop the execution of the command file and pass over input focus to the keyboard.

## SET\_NUMERICAL\_LIMITS

 This command is available for the POLY, Scheil, and ED-EXP modules.

Change the criteria for convergence. This is to speed up a calculation in a complex system.

If there is any convergence problem in finding a stable solution at any stage during a calculation procedure enforced by COMPUTE\_EQUILIBRIUM, STEP\_WITH\_OPTIONS, MAP or ADVANCED\_OPTIONS), this message displays:

```
Convergence problems, increasing smallest site-fraction from 1.00E-30
to hardware precision 2.00E-14. You can restore using SET-NUMERICAL-
LIMITS
```

This implies that smallest site fraction in the current POLY workspace is automatically increased from the default value 1.00E-30 to the hardware-dependent precision (under Linux, as 2.00E-14). For other subsequent POLY module calculations, you can use this command to restore or reset the smallest site fraction to the previous or another preferred value, as well as to reset other numerical limits.

Syntax	SET_NUMERICAL_LIMITS
Prompt	<p>Maximum number of iterations /500/:</p> <p>By default, the program tries 500 iterations before it gives up. As some models give computation times of more than 1 CPU second/iteration, this number is also used to check the CPU time and the calculation stops if 500 CPU seconds/iterations are used.</p>
	<p>Required accuracy /1E-6/:</p> <p>This is a relative accuracy, and the program requires that the relative difference in each variable must be lower than this value before it has converged. A larger value normally means fewer iterations but less accurate solutions. The value should be at least one order of magnitude larger than the machine precision.</p>
	<p>Smallest fraction /1E-12/:</p> <p>This is the value assigned to constituents that are unstable. It is normally only in the gas phase you can find such low fractions.</p> <p>The default value for the smallest site-fractions is 1E-12 for all phases, except for IDEAL phase with one sublattice site (such as the GAS mixture phase in many databases) for which the default value is always as 1E-30 (unless you have used the SET_NUMERICAL_LIMITS command to reset an even-lower value (e.g. 1E-45, that is naturally enforced to all the phases in the system).</p>
	<p>Approximate driving force for metastable phases /Y/:</p> <p>Y is the default. Enter N to change the default as required and based on the options described below.</p> <p>This setting involves the convergence of metastable phases and affects their driving forces. It can also have an effect on when an equilibrium is considered successful.</p> <p>The default is to allow an equilibrium with metastable phases that have not converged, as long as the stable phases have converged. This is efficient but often causes approximate values of the</p>

---

Syntax	SET_NUMERICAL_LIMITS
	<p>driving forces for the metastable phases.</p> <p>If you change the default, it enforces metastable phases to converge. This gives accurate driving forces for metastable phases as well as stable phases. It can however take a slightly longer time, and if metastable phases do not converge it causes the equilibrium calculation to fail.</p>

## SET\_REFERENCE\_STATE



This command is available for the POLY, ED-EXP and POST modules.

The reference state for a component is important when calculating activities, chemical potentials, and enthalpies and is determined by the database being used. For each component the data must be referred to a selected phase, temperature, and pressure, i.e. the reference state.



All data in all phases where this component dissolves must use the same reference state. However, different databases can use different reference states for the same element/component. It is important to be careful when combining data obtained from different databases.

By default, activities, chemical potentials, and so forth are computed relative to the reference state used by the database. If the reference state in the database is not suitable for your purposes, use this command to set the reference state for a component using `SER`, i.e. the Stable Element Reference (which is usually set as default for a major component in alloys dominated by the component). In such cases, the temperature and pressure for the reference state is not needed.

In order to specify conditions in the specific reference state, you can append an `R` to the state variables. Thus, `AC` is the activity (of a system component or of a species in a phase) with respect to the default reference state, and `ACR` is the activity with respect to the selected reference state; `MU` is the chemical potential of a system component with respect to the default reference state, and `MUR` stands for the chemical potential with respect to the selected reference state. The `AC` and `ACR` variables, for both components in a system and species in a phase, can be specified in natural logarithm, e.g. `LNAC(Fe)`, `LNACR(C)`, `LNAC(O2,GAS)`, `LNACR(O2,GAS)`.

For the POST module, and after calculating with the `STEP_WITH_OPTIONS` or `MAP` commands, the reference state for a component can also be changed to plot various properties of the component in the entire system or in a specific phase. Then you can set the diagram axes as the chemical potential or activity quantities with the `R` suffix, i.e. `MUR(comp)`, `MUR(comp,ph)`, `ACR(comp)`, `ACR(comp,ph)` or the common logarithms [e.g. `LNACR(comp,ph)`].

Syntax	SET_REFERENCE_STATE
Prompts	Component: <Name of the component> The name of the component must be given.
	Reference phase: <Name of a phase used as the new reference state>

Syntax	SET_REFERENCE_STATE
	<p>Enter the Name of a phase for the system.</p> <div data-bbox="363 317 1398 520" style="border: 1px solid blue; padding: 10px; margin: 10px 0;">  <p>For a phase to be usable as a reference for a component, the component needs to have the same composition as an end member of the phase. The reference state is an end member of a phase. The selection of the end member associated with the reference state is only performed once this command is executed.</p> </div> <p>If a component has the same composition as several end members of the chosen reference phase, then the end member that is selected at the specified temperature and pressure will have the lowest Gibbs energy.</p> <p>The following are example using temperature. Pressure is used in the same way:</p> <ul style="list-style-type: none"> <li>• In the Fe-C system, BCC can be a reference state for Fe but not for C since BCC can exist as pure Fe but cannot exist as pure C.</li> <li>• If GAS is chosen as the reference phase of component O at such a high temperature that O1 has the lowest energy, then O1 remains the reference state even at calculations performed at lower temperatures where O2 has a lower energy than O1.</li> <li>• Setting the reference state for component O as gas (one sublattice) with constituents O1, O2 and O3 results in O2 being the reference state if, at the present temperature, pure O2 has the lowest energy of O1, O2 and O3. If the reference state is set above a critical higher temperature, then O1 has the lowest energy and consequently becomes the reference state.</li> </ul>
	<p>Temperature /*/: &lt;Temperature for the reference state&gt;</p> <p>Select the Temperature (in K) for the reference state. The wildcard value * means the current temperature is used at the time of evaluation of the reference energy for the calculation.</p> <div data-bbox="363 1194 1398 1373" style="border: 1px solid blue; padding: 10px; margin: 10px 0;">  <p>The temperature set with this command does not affect the temperature used when evaluating the energy of the reference state during the end member selection process.</p> </div>
	<p>Pressure /1E5/: &lt;Pressure for the reference state&gt;</p> <p>Select the Pressure (in Pa) for the reference state. The wildcard value * means the current pressure is used for evaluating the reference energy at the time of calculation.</p>
	<p><b>Examples</b></p> <pre>S-R-S Fe SER S-R-S Cr FCC * 100000 S-R-S H2O AQUEOUS * 100000 S-R-S ZE REF_ELECTRODE * 100000</pre>

## SET\_START\_CONSTITUTION



This command is available with the POLY and ED-EXP modules.

This command is similar to SET\_ALL\_START\_VALUES, but is used for an individual phase that may have a wrong constitution.

Syntax	SET_START_CONSTITUTION
Prompts	<p>Phase name: &lt;Name of a phase, and possible major constituent(s)&gt;</p> <p>Specify the name of a phase for which the constitution shall be set.</p> <p>If there is a major constituent of the phase, you must specify this on the same line as the phase name. By giving an*, select the default major constituents. A \$ means keeping the same constitution, and NONE means to give individual site-fractions.</p>
	<p>Y(&lt;Phase&gt;#&lt;Composition_set&gt;,&lt;Constituent&gt;#&lt;Sublattice&gt;) /xxx/: &lt;SF&gt;</p> <p>The site fraction (SF) of the constituent shall be given. The default value xxx is the last calculated one.</p>

## SET\_START\_VALUE



This command is available with the POLY and ED-EXP modules.



This command is not often required as the automatic start values can be handled for most problems.

Set a start value of an individual variable.

Syntax	SET_START_VALUE
Prompts	State variable: <Name of a state variable> The state variable must be given.
	Value Specify the value to be assigned the state variable as start value.

## SHOW\_FOR\_T=



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

Display various thermodynamic properties (state variables, derived/partial variables or entered symbols) of the currently calculated (stable/meta-stable) equilibrium state but under a different temperature condition. This is useful for knowing, for example, a volume-related property of a frozen (stable/meta-stable) equilibrium state at a certain temperature, where the equilibrated phase assemblage and all the phase compositions are not adjusted while only the temperature condition is changed.

Use this option carefully. You must have successfully calculated a real equilibrium state under one temperature condition (normally the temperature for the last heat treatment). This can then be used to obtain the value(s) of any specified state variable(s) or derived/partial variable(s) or defined symbol(s) for thermodynamic properties of the entire system, of components, or of phases for the currently-defined system (being in a frozen state) under another temperature (normally at room temperature). No real equilibrium is re-calculated through this option, and thus the phase amounts and compositions in the system are the same as at the last real equilibrium calculation.

### Settings

Temperature (K) /298.15/: <temperature\_in\_K>

Specify the new temperature condition (in K) under which the values of some specific state variable(s) or derived/partial variable(s) or defined symbol(s) for various thermodynamic properties of the entire system, of components, or of phases) in the currently-defined system (being in a *frozen* state) is shown on screen.

State Variable or symbol /VM/: <State variable or symbol name(s)>

Specify the name(s) of the desired state variable(s) or derived/partial variable(s) or defined symbol(s) for various thermodynamic properties of the entire system, of components, or of phases) in the currently-defined system. More than one state variable or symbol of interest can be simultaneously specified on the same line. For example, you can choose to show the values of `VM` or `GM` (i.e. molar volume or molar Gibbs energy of the entire system), or of `VM(*)` or `HM(*)`.T (i.e. molar volumes or isobaric heat capacity of all phases), or of `ACR(*)`,

DGM(\*) and LNACR(\*,FCC) [i.e. activities of all system components, driving forces for all phases, and activities (in logarithm) of all system components in the FCC solution phase], under the new temperature condition (being in a frozen state).

## SHOW\_VALUE



This command is available for the POLY and ED-EXP modules.

Show the current value of any state variable, function or variable.

Syntax	SHOW_VALUE
Prompt	<p>State variable or symbol: &lt;Name(s) of state variable(s) or symbol(s)&gt;</p> <p>A single or several state variables or symbols (entered functions or variables) can be specified.</p>
	<div style="border: 1px solid #00AEEF; padding: 5px; margin-bottom: 10px;">  The wildcard * can be used to denote all phases or all components.         </div> <p>You can use the dollar-sign \$ to denote all stable phases. Thus, the command SHOW W(*,*) lists the mass fraction of all phases, whereas SHOW W(\$, *) lists the mass fraction of all stable phases.</p> <div style="border: 1px solid #00AEEF; padding: 5px; margin-bottom: 10px;">  If you SHOW a function, all functions are evaluated using the current values of the state variables. But if you SHOW a variable, it retains its value from the time it was ENTERED or last EVALUATED.         </div> <p>Therefore to save a value from one calculation to another, ENTER it to a variable. This is frequently done in the PARROT module to transfer values between equilibria.</p>

## STABILITY\_CHECK



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

In some composition ranges of a multicomponent system, it often happens that an unstable solution phase region locates inside a miscibility gap, and the stability limit (the *spinodal curve* or *spinodal*) may be not easy to find. A system inside a spinodal is thermodynamically unstable with respect to compositional fluctuations, and the system may experience the *spinodal decomposition* (i.e. decomposing to a mixture of regions with the two stable compositions, one on each side of the miscibility gap).

This option makes it possible to automatically check internal stability of both stable and unstable phases in all subsequent single-point equilibrium and MAP/STEP calculations. It can find out if any phase is subject to spinodal decomposition during the subsequent calculations. If there is an unstable phase located inside a miscibility gap in a calculation, it gives a warning so that you suspend the unstable phase, or use `FORCED` automatic start values for phase constituents (i.e. `SET_ALL_START_VALUES` → `FORCE` command-sequence), or create other composition sets; you can also ignore the warning message if you know that the unstable phase is not formed in the current calculations.

### Settings

```
Stability check on? /Y/:
```

```
Check also for unstable phases? /Y/:
```

If the automatic stability-check option is switched on choose to also check the stability for unstable phases. The default is Y and if an unstable phase is found to be located in a miscibility gap during a subsequent single-point equilibrium or MAP/STEP calculation, a warning message informs you to selectively make adjustments in the calculation settings [e.g. suspending the unstable phase, or using `FORCED` automatic start values for phase constituents by the `S_A_S_V F` command-sequence, or creating other composition sets, etc.]. By answering N the stability check is enforced only to stable phases in the system.

## STEP\_AND\_MAP



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

`STEP_AND_MAP` is the default `ADVANCED_OPTIONS` choice and is used to determine how to perform Global Minimization test and how to handle initial equilibrium points.

Settings for `MAP` and `STEP`:

Global test interval /0/: <Integer number>

Settings for `MAP`:

Automatically add initial equilibria /Y/:

Number of mesh along an axis /3/: <Integer number>

Use inside meshing points /N/:

Global test interval /0/: <Integer number>

The integer number determines how often the Global Minimization should be used during `STEP_WITH_OPTIONS` and `MAP` calculations. If it is set to 0 (zero), the recommended global test interval is used: i.e. every tenth step and at each phase change during `STEP` calculations, and only at node points during `MAP` calculations. Any other positive integer number,  $n$ , suggests it performs a Global Minimization test at every  $n$ -th step during `STEP` and `MAP` calculations. Of course, the Global Minimization test is always carried out at a phase change or a node point. The Thermo-Calc software is installed with 0 (zero) as the default value, but such a configuration for the default value can be changed by using `SET_TC_OPTIONS` in the `SYS` module.

Automatically add initial equilibria /Y/:

When [ADD\\_INITIAL\\_EQUILIBRIUM](#) is not used before `MAP`, a mesh of initial equilibrium points are added before the mapping itself takes place. This mesh is only available when two (2) axes are already defined. The default on start-up may be changed by `SET_TC_OPTIONS` in the `SYS` monitor.

Number of mesh along an axis /3/: <Integer number>

An integer for how many intervals of initial equilibrium points to be added along an edge in the diagram. For example, if set to 2 (two), initial equilibrium points are added at the beginning, center, and end of each axis-variable forming 2 intervals.

Use inside meshing points /N/:

If N the INITIAL\_EQUILIBRIUM mesh consists of initial equilibrium points added only along the edge/border of the diagram defined by the axis-variables. If Y, the initial equilibrium points added are also added to the inside edge.

## STEP\_WITH\_OPTIONS

 This command is for the POLY module.

The STEP\_WITH\_OPTIONS (short name, STEP) command initiates the stepping procedure to make a calculation of property diagrams in a multicomponent heterogeneous system. A property diagram is usually stepped under one defined independent stepping axis-variable, and is often constructed by the defined independent mapping axis-variable against a chosen property (for the system, or for a specific phase, or for a certain species).

In the POLY module, it is possible to compute a series of equilibria with one condition as the stepping axis variable. This is achieved by first calculating a single equilibrium, and then selecting one of the conditions as the axis variable. The minimum and maximum value and the incremental step along the axis must also be given. By default, the global minimization technique is used .



A property diagram consists of property lines that change (normally) along with the stepping axis-variable. From a single STEP calculation, many different types of property diagrams in the defined multicomponent heterogeneous systems can be plotted. Various types of property diagrams are generated by the stepping calculations.



### [About Property Diagrams](#)

Normally, before the command can be invoked, you need to have already calculated an initial equilibrium point and set one independent variable (i.e. the controlling condition in the system) . This is done using SET\_AXIS\_VARIABLE. You then list the current values of the axis variable for each calculated equilibrium, as well as the corresponding information when the set of stable phases changes.

By default, the global minimization technique is enabled, You can turn this off by using the command-sequence ADVANCED\_OPTIONS → GLOBAL\_MINIMIZATION.



You can choose how often to do a global minimization test using the command-sequence ADVANCED\_OPTIONS → STEP\_AND\_MAP → GLOBAL TEST INTERVAL.

Syntax

STEP\_WITH\_OPTIONS

Prompt

Option? /Normal/: <A chosen option, N, I, S, O, E, M, T, or P>

Syntax	STEP_WITH_OPTIONS
	N = Normal, I = Initial equilibria, S = Separate phases, O = One phase at a time, E = Evaluate, M = Mixed Scheil, T= T0, and P Paraequilibrium

## Normal (N)

When you accept the default, Normal (N), it allows a stepping calculation with the chosen independently-varying equilibrium condition (axis-variable). Only the step axis value is changed between each step.

When you repeat the STEP → NORMAL command-sequence, you can make several stepping calculations while still keep all the stepping-calculation results in the current POLY3 workspaces.



The calculation results are saved in the default `RESULT.POLY3` file or you can specify to save it in the `<MyName>.POLY3` file. This is as long as there is no `SAVE_WORKSPACES` command before the second set of command sequences (STEP → NORMAL). Repeated stepped results can be restored into the same current POLY3 workspaces/file.



Linux and Mac are case sensitive so ensure you enter `.POLY3`.

The addition of several steps enables you to generate special types of property diagrams and to calculate and plot special phase boundary lines on phase diagrams, such as:

- The missing part(s) of a specific property [that are calculated under another controlling condition inside the range of the stepping-variable] on the same property diagram.
- Two or more sets of a specific property [that are calculated under different controlling conditions and stepped over the same stepping-variable range] on the same property diagram for the same system.
- Calculate some special phase-boundary lines [especially for some defined secondary phase transformations or alike, e.g. the BCC\_A1 ↔ BCC\_B2 phase-boundary, or equal-Gm for two specific phases, or equal-fraction or equal-activity for a certain species in two specific phases, that shall be calculated under different controlling-conditions and also be stepped over different stepping variable(s)], and then plot (or

impose) them onto a corresponding phase diagram (which can be resulted from several such STEP → NORMAL calculations or even from a separate MAP calculation) for the same system.

### ***Initial\_Equilibria (I)***

An initial equilibria is added at each calculated point. This can be useful, for instance, to generate a set of isothermal calculations.

When you choose **I**, an initial equilibrium is stored at each calculated equilibria. It is intended to generate a matrix of calculated initial equilibria, first stepped under a STEP INITIAL\_EQUILIBRIA command-sequence and then repeated by another STEP NORMAL command-sequence (or even continued by a MAP command).

For example, you can first use the temperature condition as the stepping axis-variable with this option, which calculates a number of initial equilibrium points and automatically adds each one as an initial equilibria for another STEP → NORMAL command-sequence (or even for a MAP command). Before the second STEP command (i.e. the STEP → NORMAL command-sequence), you can chose a compositional condition as the new stepping axis-variable; then the STEP → NORMAL command-sequence uses those created starting equilibrium points (at different temperatures) and steps in such a new compositional variable. As a result, these two STEP commands together give a matrix of values.

### ***Separate\_Phases (S)***

In this way, you can calculate how the Gibbs energy for a number of phases varies for different compositions. This is particularly useful to calculate Gibbs energies for complex phases with miscibility gaps and for an ordered phase that is never disordered (e.g. SIMGMA-phase, G-phase, MU-phase, etc.).

### ***One\_Phase\_at\_Time (O)***

This option is recommended for STEPPING heterogeneous systems with both ordered phases and the disordered pairs.

When you choose **O**, the stepping calculation is conducted individually for one phase at a time. That is, it is stepped over the whole stepping variable range, but the stepping is repeated for all the entered phases in the defined system. This is particularly useful to properly step in composition for an equilibrium heterogeneous system with both ordered phases and the disordered pairs. Thus, instead of calculating for the same composition for each of the phases at each composition step, it starts the stepping in each of the phases in the default most-stable compositions (the major constitutions).

## Evaluate (E)



For Scheil-Gulliver simulations it is recommended you use the SCHEIL module.

Evaluate **E** is an advanced option and is useful when additional conditions (instead of the stepping variable) require changing during a stepping calculation. It allows a stepping calculation in a single axis with simultaneous evaluation of one or more variables after each step. As variables can be used as conditions, it means that you can change the conditions during the stepping. After a successful EVALUATE calculation, you can go to the POST module to manually define and plot property diagrams.



The variables given are evaluated after each step, which may change other conditions. This can be useful to perform the Ordinary ScheilGulliver Approach (non-equilibrium transformations) for simulating alloy solidification without back diffusion of fast-diffusing interstitial components in solid phases. However, it is recommended you use the SCHEIL module.

Specify the prompt:

Variable name(s): <Variable name>

The names of the variables that shall be evaluated after each step must be typed here.



The miscibility gap test is automatically used during stepping (see [ADVANCED OPTIONS](#)) if a phase has two or more composition sets.

The miscibility gap test means that you can start calculating from high temperatures in a steel where the MC carbide is not stable. During the calculation, the MC carbide first becomes metastable with a composition different from the metallic FCC phase and later also becomes stable.



See Console Mode example tcex30A.

## Mixed\_Scheil (M)



For Scheil-Gulliver simulations it is recommended you use the SCHEIL module.

This is an advanced stepping calculation designed for the Extended Scheil-Gulliver Approach (partial-equilibrium transformations) to simulate alloy solidifications with back diffusion of one or more fast-diffusing interstitial components (such as C, N, O, S, etc.) in solid phases is performed. It also will consider BCC-->FCC phase transformation (practically for steels) along the alloy solidification process.

To ensure a successful stepping calculation of mixed Scheil-Gulliver simulation of solidification process of a defined alloy system with a certain initial overall composition, it is important to note the following:

- For solution phases with possibility of miscibility gap existence, you have appropriately added the necessary composition set(s);
- You have already set the temperature condition as the stepping SET\_AXIS\_VARIABLE command, with minimum and maximum temperature points as well as an appropriate s temperature step for the cooling process).
- There must be a composition condition for each of the back diffusion components (i.e. fast-diffusing interstitial components, such as C, N and/or other elements) that are already defined in terms of mole-fraction or mass-fraction, e.g.  $X(C)$  and  $W(N)$ . Otherwise, a stepping calculation with this option can fail.
- An initial equilibrium in which the LIQUID mixture phase is the only stable must be calculated. This is why it is good to start with a relatively high temperature condition for the initial equilibrium calculation. It is always necessary to reject or suspend the GAS mixture phase (if it exists) before the C-E and STEP commands, to avoid its formation along with LIQUID.
- In cases where fast-diffusing interstitial components are specified as back diffusion components, you can choose to allow BCC>FCC phase transformation (practically for steels) during the alloy solidification process.

These are the prompts for this option:

```
Fast diffusing components: <Fast diffusion interstitial(s)>
```

Specify one or more interstitial component(s) as the back diffusion component(s).

- If there is only one fast-diffusing interstitial component (e.g. C or N), type its name;
- If there are two or more fast-diffusing interstitial components (e.g. C, N and S), always type the names on the same line at once (separated by an empty space in between, e.g. C N S);
- If there is no fast-diffusing interstitial component to be considered, type `NONE` to ignore back diffusion entirely, meaning a normal Scheil-Gulliver simulation is performed.



Such specified back diffusion interstitial component(s) must be appropriately defined according to the phase constitution definitions of some major alloy solution phases (e.g. FCC, BCC, HCP, etc.). These are located on the interstitial/vacancy sublattices in such alloy solution phases; otherwise, such a Scheil-Gulliver simulation may not make sense at all.

Allow BCC -> FCC ? /Y/:

By default the BCC-->FCC phase transformation is allowed. This is usually a typical phenomenon along the cooling processes of steels/Fe-alloys.

During the calculation the simulated solidification path, including the locally-equilibrated phase assemblage (region), captured temperature condition ( $T$  in K), remaining liquid phase ( $N_L$ , mole fraction), overall formed solid phases ( $N_S$ , mole fraction) and latent heat formation ( $N_H$ , J/mol) along the cooling process are shown after the information on starting temperature point of the alloy solidification process, e.g.

```

Solidification starts at 1743.15 K
Phase Region from 1.744150E+03 for: LIQUID
Phase Region from 1.742525E+03 for:
LIQUID
FCC_A1#1
1742.5250 0.9960 0.0040 -1.1824
1742.4000 0.9795 0.0205 -202.1585 . . . . .
Phase Region from 1.733150E+03 for:
LIQUID
BCC_A2
FCC_A1#1
1733.1500 0.3294 0.6707 -8032.6240
1733.0250 0.3237 0.6763 -8095.1490 . . . . .

```

After a successful calculation, you can go to the POST module to manually define and plot property diagrams for the alloy solidification process according to the chosen Scheil-Gulliver model, or to impose existing relevant experimental information or other types of calculated solidification results [e.g. Scheil-Gulliver simulation with or without considering fast-diffusing interstitial component(s), full-equilibrium approach, or Diffusion Module (DICTRA)-type simulation with moving-boundary conditions] onto the same plotted property diagrams. Normally, the solidus temperature ( $T$  in Celsius) is set as one of the axis variables (usually as Y-axis in most cases), while the other plotted quantity on the other axis can be the amount of overall formed solid alloy phases ( $N_S$  in mole-fraction or  $B_S$  in mass-fraction), the amount of remaining liquid phase ( $N_L$  in mole-fraction or  $B_L$  in mass fraction), the heat of latent along the solidification process ( $N_H$  in J/mol or  $B_H$  in J/gram), among many other properties in the solidified alloy phases or in the whole alloy system.

## T-zero (T)



See Console Mode example 41.

This advanced option allows a stepping calculation of the so-called T0 (T-zero) line in a diffusionless transformation (where two specific partially-equilibrated phases have the same Gibbs energy) along a composition variable which has already set as the stepping variable with [SET\\_AXIS\\_VARIABLE](#).



The temperature condition cannot be set as the stepping variable if you want to make a STEP T-ZERO calculation.



Prior to performing the step T0 command sequence, and to ensure a successful calculation of T0 line between two specific phases after an initial equilibrium calculation in the current system, it is sometimes recommended to make a single T0 point calculation using `ADVANCED_OPTIONS → T-ZERO`.

Specify these prompts:

Name of first phase: <Phase A>

Name of second phase: <Phase B>

The names of the target phases, for which the Gibbs energies equal to each other at each point on the T0 line, must be entered.

During the calculation, the T0 values are shown after the corresponding scanned conditions (of the stepping composition variable), e.g.

```
Phase Region from 1.000000E-01 for:
  BCC_A2
  FCC_A1
1.000000E-01      940.24
9.250000E-02      941.20
. . . . .
2.500000E-03      977.61
7.500000E-09      979.34
Phase Region from 1.000000E-01 for:
  BCC_A2
  FCC_A1
1.000000E-01      940.24
1.075000E-01      939.62
. . . . .
2.950000E-01      1084.87
3.000000E-01      1080.99
```

After a successful calculation, you can go to the POST module to plot the T0 line against the stepping composition variable or another varying axis, or to impose the calculated T0 line onto a normal phase diagram [normally plotted as a T-x isopleth].

## ***Paraequilibrium (P)***



See Console Mode examples 42 and 43.

This is an advanced option. Select **P** to calculate the paraequilibrium state between two specific phases (the *paraequilibrium lines*). It allows a stepping calculation of paraequilibrium lines where the chemical potential for one or more interstitial components (that are considered as back diffusion elements, such as C and N) but not for the other (s) in two partially equilibrated phases are equal.

More than one interstitial component can be treated as fast-diffusion species (such as C, N, O, S, etc., as individual or combined) in a paraequilibrium stepping calculation.



The composition condition for the interstitial components cannot be set as the stepping variable if you want to make this calculation.

To ensure a successful stepping calculation of paraequilibrium states between two specific phases in a defined alloy system, it is important to first make a starting-point equilibrium calculation with an initial overall composition in the current system. However, it is unnecessary to obtain an equilibrium in which either one or both of the target phases is stable.

The initial overall composition must have a reasonable setting for the desired paraequilibrium calculation for the two target phases. This is especially true where there is more than one interstitial component to be considered in the paraequilibrium state, because different interstitial components (for example C and N combined) may have significant different behaviours as partitioning into different structured phases; otherwise, for one chosen interstitial component the initial overall composition is OK for the paraequilibrium calculation between the specified two phases, but for other chosen interstitial component(s) it might be impossible to calculate the paraequilibrium state.

Always check if the chosen `phaseA` and `phaseB` have the exact same definition of elements and if the chosen interstitial components are all in the vacancy sublattice sites of the two phases; otherwise the program cannot find the paraequilibrium state (as it is impossible to correctly calculate u-fractions).

Always have a comprehensive understanding of the normal phase diagram for the currently investigated system, so that you make the appropriate choice of the phase pair and starting bulk composition for the system.

Always set the status of the chosen interstitial components as SPECIAL using the command:

```
CHANGE_STATUS COMPONENT <interstitial component> = SPECIAL
```

By doing this, you get a clear picture on u-fractions of various substitutional and interstitial components, which are different from the overall composition in the system. The SPECIAL status means that specified component(s) are not included in summations for mole or mass fractions. Therefore, all the composition variables plotted from the paraequilibrium calculations are u-fraction related quantities.

Always make a single-point paraequilibrium calculation by the command-sequence of `ADVANCED_OPTIONS → PARAEQUILIBRIUM`, prior to performing this `STEP → PARAEQUILIBRIUM` command-sequence, although this might not be always necessary for some systems.

With this stepping calculation specify the following:

```
Name of first phase: <Phase A>
```

Name of second phase: <Phase B>

The names of the two target phases <sub>A</sub> and <sub>B</sub>, between which the paraequilibrium states establish, must be typed at the above two prompts subsequently or on the same (first) line at once (separated by an empty space, e.g. FCC BCC, FCC#2 M23C6).



It is important to understand that this is the calculation of a paraequilibrium state between the two specified phases.

Particularly, there are these issues to consider:

1. Both chosen phases must have similar interstitial/vacancy sublattices where the fast-diffusion interstitial component(s) occupy.
2. The choice on the target phase pair must be reasonable for the defined system and specified initial overall composition.
3. Both target phases should have phase constitution definitions that cover all the defined substitutional and interstitial components of the current alloy system; or
4. It is impossible to calculate the paraequilibrium state between the target phase pairs with given interstitial component(s) in the currently defined system.

Fast diffusing component: /C/: <Interstitial component(s)>

Fast diffusing component: /None/: <Interstitial component(s)>

The name(s) of the fast-diffusing interstitial component(s) (c as the default single component) must be given either one by one at each of the above prompts subsequently or in a combination (separated by an empty space in between, e.g. C N S) at the same (first) prompt; otherwise, press <Enter> key to start the paraequilibrium line stepping-calculation.



Note the following:

Such specified fast-diffusing interstitial component(s) must be appropriately defined according to the phase constitution definitions of the two selected phases: These must be located on the interstitial/vacancy sublattices in both chosen phases.

If there is only one fast-diffusing component which is carbon, press <Enter> to accept the default value (c) at the first prompt; if the single fast-diffusing interstitial component is another element (e.g. N), appropriately type interstitial component the name at the first prompt;

If there are two or more fast-diffusing components (e.g. C and N), type their names at the above prompts subsequently or at the same (first) prompt (separated by an empty space, such as C N);

To finish the input of fast-diffusing elements, accept NONE at a repeated prompt;

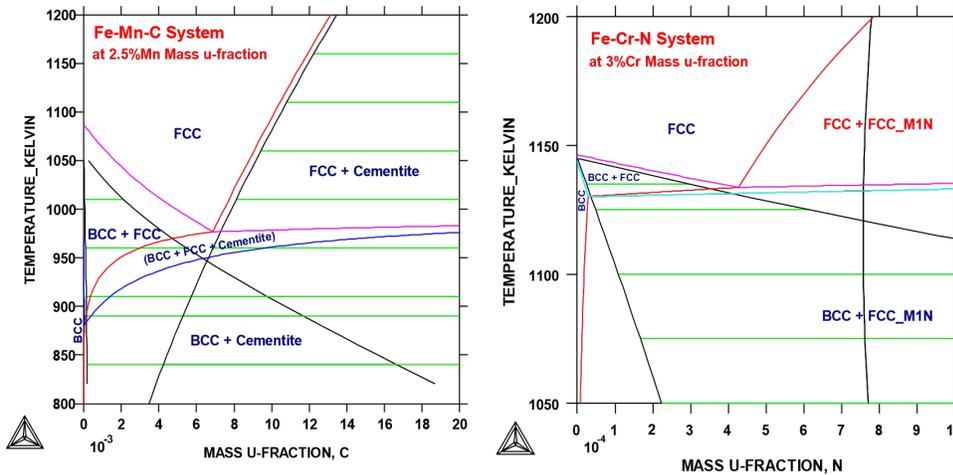
If NONE or a non-existing component name is typed at the first prompt, it means no back diffusion is to be considered, and the para-equilibrium calculation is thus cancelled entirely.

During the STEP → PARAEQUILIBRIUM calculation procedure, the calculated paraequilibrium states include the:

- corresponding stepping conditions (e.g. temperature in K, or the stepping composition variable of the matrix component or of one of substitutional components) which has already set as the stepping variable by the SET\_AXIS\_VAR command);
- amounts of the phase A in mole number [i.e. NP (phaseA) ];
- amounts of the phase B in mole number [i.e. NP (phaseB) ];
- contents of the interstitial component(s) in the phase A expressed in u-fractions [i.e. u-f (phaseA, component (s) )];
- contents of the interstitial component(s) in the phase B expressed in u-fractions [i.e. u-f (phaseB, component (s) )];
- LNACR (component) value(s) for the interstitial(s).

After a successful STEP → PARAEQUILIBRIUM calculation, you can go to the POST module to plot a paraequilibrium phase diagram, or to impose the calculated paraequilibrium states onto a normal phase diagram [normally plotted as a T-X isopleth with one of the considered interstitial component(s) as the x-axis variable].

However, the stepping calculation procedure for the currently specified paraequilibrium states between the two specified phases may find difficulties at some steps due to some possible numerical problems (normally regarding to the u-fractions of the interstitial components under some composition ranges), or it may completely fails. In the latter case, it implies that the chosen target phase pair may be unreasonable for the defined alloy system or for the defined initial overall composition, or one or both phases may have inappropriate phase constitution definitions regarding the specified interstitial component(s) and substitutional components. Then, you must either modify the settings of initial overall composition or specify the reasonable target phase pair with an appropriate choice of the fast diffusion interstitials in the defined alloy system.



The figure shows the plot of a paraequilibrium calculation of isopleths: Formation of para-pearlite in two Fe-based alloys, with equilibrium results appended.



The calculated paraequilibrium phase boundaries (black lines) are shown with tie-lines (in green-colour). In the Fe-2.5Mn-C alloy system, C is considered as fast diffusing interstitial component in the BCC+FCC and FCC+Cementite paraequilibrium stepping calculations; while in the Fe-3Cr-N alloy system, N is treated as fast diffusing interstitial components in the BCC+FCC and FCC+FCC\_M1N (i.e. FCC#1+FCC#2) paraequilibrium stepping calculations.



During a STEP calculation, the value of the stepping axis variable for each calculated equilibrium is listed and also the set of stable phases.



You can terminate the mapping of a line by pressing a single <Ctrl-A> (Windows) or <Ctrl-C> (Linux). This can be useful in order to stop a longish calculation without losing what is already calculated.

If there is any convergence problem in finding a stable solution at any stage during a calculation procedure enforced by a STEP\_WITH\_OPTIONS command-sequence, this message displays:

```
Convergence problems, increasing smallest site-fraction from 1.00E-30 to
hardware precision 2.00E-14. You can restore using SET-NUMERICAL-LIMITS
```

---

This implies that smallest site fraction in the current POLY3 workspace is automatically increased from the default value  $1.00\text{E}-30$  to the hardware-dependent precision (under Linux, as  $2.00\text{E}-14$ ). For other subsequent POLY-module calculation in the current TC run, you can use the POLY command `SET_NUMERICAL_LIMITS` to restore or reset the smallest site fraction to the previous or another preferred value, as well as to reset other numerical limits.

Complex miscibility gaps of various solution phases are automatically detected during a stepping calculation, and two or more composition sets for each of such solution phases are automatically generated as well, through an automatic Global Minimization Test procedure (by specifying a `Global Test Interval` value and confirming the `Automatically add initial equilibria` in the `ADVANCED_OPTION STEP_AND_MAP` command-sequence). As a particular case, such an automatic Global Minimization Test ensures that you can now start calculating from high temperatures in an austenitic steel (metallic `FCC_A1` solution phase) where the MC carbides/nitrides/carbonitrides (i.e. the C-/N-rich sides of the `FCC_A1` miscibility-gap, often (while not always) being referred to as `FCC_A1#2`, `FCC_A1#3`, etc.) are not stable, and during the `MAP` command the MC carbides/nitrides/carbonitrides may first become metastable with a composition different from the metallic `FCC_A1` phase and later also stable.

Occasionally, it may be necessary to have a specified increment other than the default value for the defined stepping axis-variable, in order to calculate/plot a smoother property diagram.

To get a complete property diagram, sometimes (especially in some earlier Thermo-Calc versions) it may also be required to have more than one starting equilibrium-points (under another starting equilibrium condition), and repeat the stepping calculation over the same stepping axis-variable range (while without over-writing the same POLY3 workspaces via a `SAVE_WORKSPACE` command). However, using the current `STEP` procedure such a requirement become less necessary.

## TABULATE



This command is available for a table entered in either the POLY or POST module.

For any entered table, this command gives a table of values from equilibria calculated with a STEP command.



Unlike the TAB module's command (also called TABULATE) it is not possible to plot columns from the tabulated tables. For this use normal POST commands.

Syntax	TABULATE
Prompts	Name: <Name of a table entered in either POLY or POST> Specify a table name that is entered in either the POLY or POST module.
	Output on screen or file /Screen/: Specify a file name if you want to save on a file the table values along the defined STEP calculation, or press <Enter> if want to see the table values displayed.

---

## TOGGLE\_ALTERNATE\_MODE



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

Toggle the `ALTERNATE` mode for experimental equilibrium calculation between `DEFAULT`, `ALWAYS` and `NEVER`, during data-assessments using the `PARROT` optimization.

Set alternate toggle to `DEFAULT`, `ALWAYS`, or `NEVER`?

Set global toggle to `DEFAULT`, `ALWAYS`, or `NEVER`?

- `DEFAULT`: the experiment points are calculated according to the Alternate Technique depending on the `SET_ALTERNATE` command.
- `ALWAYS`: the experiment points are always calculated according to the Alternate Technique even if the Alternate Mode is switched off in the `PARROT` module.
- `NEVER`: the experiment points are calculated as normal equilibria even if you `SET_ALTERNATE` in the `PARROT` module.

---

## T-ZERO\_TEMPERATURE



This is an option that is available with the `ADVANCED_OPTIONS` command with the `POLY` and `ED-EXP` modules.



### ADVANCED\_OPTIONS

This calculates the temperature when two specific phases have the same Gibbs energy, i.e. the so-called `T0` temperature. You must calculate an equilibrium state at an estimated temperature before performing this advanced-option calculation; however, it is unnecessary to obtain an equilibrium state in which either one or both of the target phases is stable.

The following prompts are shown.

```
Name of first phase: <Phase A>
```

```
Name of second phase: <Phase B>
```

The names of phases A and B must be given, for which the `T0` temperature (where the Gibbs energies are equal) is to be calculated.

If the `T0` temperature between the two specified phases is successfully calculated, a message displays, e.g.

```
The T0 temperature is 840.82 K. Note: LIST-EQUILIBRIUM is not relevant
```

The first message shows the calculated `T0` temperature between the two specified phases.

The second message indicates that after this option calculation with `LIST_EQUILIBRIUM` is irrelevant and does not list the equilibrium for the system at the `T0`-temperature.

## Gibbs Energy System (GES) Commands

---

### In this section:

---

Solution Models in the GIBBS (GES) Module .....	283
About the ENTER_PHASE Command .....	291
ADD_COMMENT .....	295
AMEND_ELEMENT_DATA .....	296
AMEND_PARAMETER .....	298
AMEND_PHASE_DESCRIPTION .....	301
AMEND_SYMBOL .....	305
CHANGE_STATUS .....	308
ENTER_ELEMENT .....	309
ENTER_FUNCTION .....	310
ENTER_PARAMETER .....	312
ENTER_PHASE .....	314
ENTER_SPECIES .....	316
ENTER_SYMBOL .....	317
LIST_CONSTITUTION .....	321
LIST_DATA .....	322
LIST_PARAMETER .....	325
LIST_PHASE_DATA .....	327
LIST_STATUS .....	328
LIST_SYMBOL .....	331
READ_GES_WORKSPACE .....	332
READ_WORKSPACES .....	333
REINITIATE .....	334

---

SAVE_GES_WORKSPACE .....	335
SAVE_WORKSPACES .....	336



---

## Solution Models in the GIBBS (GES) Module

### AMEND\_PHASE\_DESCRIPTION

The AMEND\_PHASE\_DESCRIPTION command is used to specify/amend phase descriptions if a phase has a miscibility gap, uses a special excess energy model, or has a specific additional contribution to its Gibbs energy, for example.

This topic outlines the application of each option and includes important information about each command.

There are many solution models implemented in the GIBBS module (type `INFO MODELS` to see details about various thermodynamic models). However, these are switched on and handled differently via various GIBBS routines/commands:

- The first models the non-ideality as excess parts, i.e. by amending the phase's EXCESS\_MODEL and/or TERNARY\_EXTRAPOLAT models.
- The second models the non-ideality as ADDITIONAL parts, i.e. by amending the phase's other subjects in this command, e.g. MAGNETIC\_ORDERING for the Magnetic Ordering Model, DISORDERED\_PART for the CVM Approach in modeling chemical ordering/disordering phenomenon, QUASICHEM\_IONIC for using a quasi-chemical entropy term for an ionic two-sublattice liquid solution phase, QUASICHEM\_FACT00 or QUASICHEM\_IRSID for describing a substitutional liquid solution phase, DEBYE\_HUCKEL for the DHLL/SIT Model in describing a dilute aqueous solution phase, etc.
- The third implements the model entirely or partially into the GIBBS module and related database(s), such as for the electrostatic contribution in an AQUEOUS solution phase by the Complete Revised\_HKF Model, the Murngham Model, Birch-Murngham Model or Generalized PVT Model for high-pressure/temperature volume contribution in a solid or liquid phase, the SUPERFLUID Model for the non-ideal EOS and non-ideal mixing in a gaseous mixture.

### **NEW\_CONSTITUENT**

Use this if you want to add a new constituent to a phase. It is illegal to add new constituents to an ionic two-sublattice liquid phase.

### **RENAME\_PHASE**

Use this to change the names of some specific phases. For example, a phase called *FE3O4\_S* may be better named *Magnetite* to help identify it during the calculations and postprocessing. This is also a way to delete a phase by hiding it under a new name.

---

## ***SITE\_RATIOS***

Use this to change the number of sites (i.e. the stoichiometric coefficients of various sublattices) in a sublattice phase.

## ***COMPOSITION\_SETS***

Use this for solution phases that may have miscibility gap(s). However, this is less important today and often unnecessary to define additional composition set(s), since the implemented Global Minimization Technique can usually detect and add such composition set(s) in an automatic manner where it is really necessary during equilibrium calculations (of single-points, stepping or mapping).

## ***MAJOR\_CONSTITUENT***

Use this to set major constituent(s) on each sublattice in each composition set for a solution phase. This is useful to make calculations converge faster because it may simplify giving start values when calculating the equilibrium as those phases with miscibility gaps should have different major constituents for each composition set. The databases often set major constituents for several phases automatically when data are retrieved.

## ***FRACTION\_LIMITS***

Use this to limit the application range (in terms of mole-fractions of all the involved elements) of a particular solution phase. This is useful to avoid automatic creations (enforced by the Global Minimization Technique) of additional composition sets for some solution phases (such as AQUEOUS solutions or dilute Fe-based liquid mixtures) of which the used models [e.g. the `SIT Model` or the `Complete Revised_HKF Model` for AQUEOUS solution, or the modified dilute solution parameters (plus a quadratic term, according to Hillert (1986) based on the SigworthElliot Model [1974Sig] for Fe-rich liquid mixture] cannot be appropriately applied on a full scale.

Globally set the composition limits (in terms of mole fractions of various elements) in a specific solution phase, so that whenever the program finds a potential phase composition or a new composition set of possible miscibility gap(s) but that is out of this globally-set composition range, the program automatically ignores such a phase composition in an equilibrium calculation. This can be done either permanently inside a database (by enforcing this option for amending the phase-description of the solution phase) or temporarily within the GIBBS module (by using this phase-description amendment option).

This is important for solution phases for which the thermodynamic models and assessed data for the phases are for specific composition ranges, for example, the FE\_LIQUID phase in the SLAG database is only applicable for Fe-rich liquid mixture, and the AQUEOUS phase in the TCAQ/PAQ and AQS/PAQS databases is only applicable for H<sub>2</sub>O-dominant aqueous solution phase.

Specify the low and high mole-fraction limits for each of the elements possibly available within the considered phase (in the currently defined system, or in a certain database). Do this for all the possible elements (defined in the phase) in a single AMEND\_PHASE\_DESCRIPTION FRACTION\_LIMITS command-sequence, for example:

```
TYPE_DEFINITION R GES AM_PH_DES FE_LIQUID FRACTION_LIMITS Fe 0 0.6
    Ag 0 0.01 Al 0 0.05 Ca 0 0.05 Co 0 0.01 Cr 0 0.01
    Cu 0 0.02 Mg 0 0.05 Mn 0 0.05 Mo 0 0.05 Nb 0 0.05
    Ni 0 0.05 Pb 0 0.05 Si 0 0.10 Sn 0 0.02 Ti 0 0.05
U 0 0.01 V 0 0.02 W 0 0.02 Zr 0 0.03
    B 0 0.01 C 0 0.01 H 0 0.01 N 0 0.01 O 0 0.01
P 0 0.01 S 0 0.01 !
```

## ***DISORDERED\_PART***

This command is needed for the special treatment of chemically-ordered phases where the contributions from the disordered state are described by a phase without ordering sublattices (the disordered phase name must be specified).

Several checks are made that the ordered and disordered phases are compatible (sublattices, sites and constituents). A link is then created between the phases, and the disordered phase is hidden from application programs. The Gibbs energy for the ordered phase also includes the Gibbs energy for the disordered phase.

Phases which can have an order/disorder transformation have parameters split on two phases and are referred to as the *two phase* description in the GIBBS module. One of them has sublattices for chemical ordering, the other one represents the disordered state. Normally, the ordered BCC and FCC or HCP phases may have either 2 or 4 substitutional sublattice (plus one additional interstitial site), that are handled by the Two Substitutional-Sublattice Ordering Model or Four Substitutional-Sublattice Ordering Model, respectively.

*Two phase* means that the *ordered* phase has parameters that describe the ordering. The *disordered* phase has all parameters for the reference state and those for describing the disordered phase. The ordered phase may occur only in some systems whereas the disordered phase may occur frequently, typical examples are the disordered FCC and BCC

which may become ordered as L12 or B2 in certain systems. In order to treat multicomponent systems where some subsystems have ordering it would be necessary to transform all parameters of BCC into a B2 model.

The parameters describing the disordered phase are not changed but the Gibbs Energy system is informed that the Gibbs Energy for the two phases should be added.

The method used calculates the Gibbs Energy of a two phase model with two or four substitutional sublattices for ordering.

$$G_m = G_m^{ord}(y'_i, y''_i) + G_m^{dis}(x_i) - G_m^{ord}(y'_i = x_i, y''_i = x_i)$$



The mole fractions  $x_i$  are calculated from the site fractions  $y_i$ . The last term means that the contribution from  $G_m^{ord}$  in a disordered state (same site fraction, equal to the mole fraction, on both sublattices) is subtracted. The effect of this is that the parameters in the ordered phase have no contribution to the disordered state.

## **MAGNETIC\_ORDERING**

Change the magnetic ordering parameter for a certain phase with magnetic ordering contribution, in terms of its anti-ferromagnetic factor. By default this is  $-1$  for BCC phase and  $-3$  for all other phases (FCC, HCP, etc.). The fraction value of the total enthalpy (due to short-range ordering above the magnetic transition temperature) is by default 0.40 for BCC phase and 0.28 for all other phases (FCC, HCP, etc.).

## **EXCESS\_MODEL**



See [example tcex52](#).

Use this to change the default Excess Model (for interaction energies) from the default (REDLICH-KISTER\_MUGGIANU). The model handles the excess interaction energies in a solution phase is chosen from:

- REDLICH-KISTER\_MUGGIANU: for binary (R-K) & ternary (R-K-M) interactions
- REDLICH-KISTER\_KOHLER: for binary (R-K) & ternary (R-K-K) interactions
- FLORY-HUGGINS POLYMER MODEL: for interactions in a polymer mixture phase (F-H)

- MIXED-EXCESS-MODELS: (R-K default) for mixed binary excess model (R-K, Legendre & Polynom) of a substitutional solution phase
- HKF: for interaction in an AQUEOUS solution phase (HKF)
- PITZER: for interaction in an AQUEOUS solution phase (PIT)
- CENTRAL\_ATOM\_MODEL: for interaction in a liquid slag solution phase (C-A-M)

For extrapolations of excess energies from related binary systems to ternary or higher-order systems, the Redlich-Kister binary excess interaction parameters may be extrapolated with either a Muggianu extension (i.e. the default REDLICH-KISTER\_MUGGIANU model) and a Kohler extension (i.e. the alternative REDLICH-KISTER\_KOHLER model), where there is no ternary, quaternary or higher-order interaction parameter.



The REDLICHKISTER\_KOHLER model is implemented only for ternary systems.

The MIXED-EXCESS-MODELS option works only for a substitutional phase with no sublattice (such as the metallic LIQUID phase) and it can be used to invoke asymmetrical simple or Legendre polynomial as binary excess energy models in addition to the default symmetrical Redlich-Kister model for the chosen pair of constituents in a substitutional phase with no sublattice. Asymmetrical here means that the power series depend only on one of the constituents, for example with expansions based on the  $[1-2*X(B)]$  term rather than  $[X(A)X(B)]$ .



Binary interaction parameters for such a pair of constituents must be entered prior to turning on the non-default Legendre or Polynom models.

## ***TERNARY\_EXTRAPOLAT***



See [example tcex52](#).

Use this to change the extrapolation method from the default REDLICH-KISTER\_MUGGIANU to another extrapolation model. This method extrapolates from binary to ternary (and higher-order) excess interaction parameters in a solution phase and is chosen from:

- MUGGIANU for Muggianu Extrapolation from R-K parameters
- TOOP-KOHLER for Toop-Kohler Extrapolation Model
- KOHLER-ALL for Kohler Extrapolation Model
- MUGGIANU\_RESTOR for Muggianu-Restor Extrapolation Model



Only when all the relevant binary excess energies in the current solution phase are treated by the default Redlich-Kister Model (i.e. the Mixed-Excess-Model should have not been used), the MUGGIANU\_RESTOR method for ternary extrapolations is equivalent to the Redlich-Kister\_Muggianu Model, or the KOHLER-ALL method to the RedlichKister\_Kohler Model.

The default ternary extrapolation method MUGGIANU\_RESTOR applies to a solution phase as a whole, whatever the binary excess model(s) enforced to each of individual binary pairs in the phase. In case that all the binary pairs in the phase use the default binary excess model REDLICH-KISTER, then the ternary extrapolation method becomes the so-called Redlich-Kister\_Muggianu Model (in short as R-K-M), for extrapolations from binary parameters to ternary (and higher-order) excess energy terms; when no ternary L parameter is entered for that, such a default R-K-M Model is always used.

The KOHLER-ALL ternary extrapolation method can be turned on also for a solution phase as a whole, whatever the binary excess model(s) enforced to each of individual binary pairs in the phase. In case that all the binary pairs in the phase use the default binary excess model.

REDLICH-KISTER, then the ternary extrapolation method becomes the so-called Redlich-Kister\_Kohler Model (in short as R-K-K), for extrapolations from binary parameters to ternary (and higher-order) excess energy terms; when no ternary L parameter is entered for that, such an R-K-K Model is used.

The TOOP-KOHLER ternary extrapolation method is applied in a special way: for a specific ternary system (e.g. A-B-C) in a solution phase, specify which two constituents as the so-called Kohler constituents and the remaining constituent as the so-called Toop constituent on a given sublattice. If for the A-B-C ternary system you have specified A and B, or B and A, as the Kohler constituents (entered as the basis constituent and first interacting constituent) and C as the Toop constituent, only the A-B binary interaction parameters are used in accordance with the Kohler ternary extrapolation formula for A-B-C ternary interaction, while any other binary interaction parameters involving the Toop species C (i.e. of A-C and B-C binaries) are used in line with the Toop-Kohler ternary extrapolation formula (for the A-C-B and B-C-A ternary interactions).

---

## **DEBYE\_HUCKEL**

To use the DHLL (Debye-Hückel Limiting Law) model and SIT (Specific Ionic Interaction Theory) model for a dilute AQUEOUS solution phase, switch the DEBYE\_HUCKEL part on, and it removes previously set (if any) ADDITIONAL part on the non-ideality for the chosen AQUEOUS phase.

## **HKF\_ELECTROSTATIC**

To use the hypothetical electrostatic contribution for the chosen phase (it must be an aqueous solution phase using the Complete Revised\_HKF Model). It removes previously set (if any) ADDITIONAL part on the non-ideality for the chosen AQUEOUS solution phase.

## **QUASICHEM\_IONIC**

To use the Quasichemical-Ionic model for the chosen liquid phase, you must have already entered/retrieved a liquid phase as an ionic two-sublattice liquid solution phase (normally, that is the IONIC\_LIQ phase), and then use this amending option to switch QUASICHEM\_IONIC on. It creates a completely new liquid solution phase [namely the QUAS\_IONIC phase which uses the Quasichemical Model for the entropy, according to Mats Hillert [2001Hil], while the original IONIC\_LIQ phase remains in the system and is not changed by this option.

## **QUASICHEM\_FACT00**



See example [tcex49](#).

To use the Quasichemical Model developed by Kongoli et al. [1998Kon], you must have entered/retrieved a liquid phase using a normal substitutional liquid model with specified associates or species (on a single lattice site), and then turn on the option QUASICHEM\_FACT00. This removes the previously-set (if any) ADDITIONAL part on the non-ideality for the chosen liquid phase.

## **QUASICHEM\_IRSID**

To use the Kapoor-Frohberg-Gaye Quasichemical Cell Model (i.e. the Quasichemical Model developed by ISRID, France) for a liquid SLAG solution phase, you can use a normal liquid model with specified associates or species.

### ***GLASS\_TRANSITION***

To use the special model for glass transition of a liquid phase. It removes previously set (if any) `ADDITIONAL` part on the non-ideality for the chosen liquid phase.

### ***REMOVE\_ADDITION***

To remove all the selected `ADDITIONAL` part from the Gibbs energy description for the chosen phase. If preferred, set a specific phase-status bit for a phase, use the `PHASE_BITS` option as long you know the restrict meaning of each part of a phase- status bits.

### ***DEFAULT\_STABLE***

Set phases as default-stable, which helps you to have a better guess of which phases that should be stable at the first calculation.

### ***References***

- [1974Sig] G. K. Sigworth, J. F. Elliott, The Thermodynamics of Liquid Dilute Iron Alloys. *Met. Sci.* 8, 298–310 (1974).
- [1998Kon] F. Kongoli, A. D. Pelton, Y. Dessureault, Thermodynamic modeling of liquid Fe-Ni-Cu-Co-S mattes. *Metall. Mater. Trans. B.* 29, 591–601 (1998).
- [2001Hil] M. Hillert, The compound energy formalism (1). *J. Alloys Compd.* 320, 161–176 (2001).

## About the ENTER\_PHASE Command



### ENTER\_PHASE

With the command ENTER\_PHASE, the phase name, phase-type, sublattice number, and constituents or constituent array(s) for the phase are entered into the GIBBS workspaces.

### **About Phase Names**

A thermochemical system must have at least one phase (which is a homogeneous part of the system with a uniform composition and structure). In the GIBBS module, any number of phases can be entered for a system and for each phase there must be a description of how its Gibbs energy varies with temperature, pressure and composition. A phase has a large amount of data connected to it, e.g. it starts with a phase name:

- It may be treated as a special phase-type;
- It may have structural information about sublattice(s), etc.,
- There must be a list of constituents (for a substitutional phase with no sublattice) or of constituent arrays (for a sublattice phase);
- There may be basic information on what kind of EXCESS\_MODEL (polynomial expression) is used to describe the binary, ternary and/or higher-order interactions between constituents;
- There may be so-called Additional contributions to the Gibbs energy of the phase from special physical phenomena, e.g. magnetic ordering, hypothetical electrostatic interaction, and so forth;
- There must exist all the parameters required for the descriptions of thermochemical properties (i.e.  $G$  terms for standard Gibbs energies, and  $L$  terms for binary, ternary or higher-order interaction excess energies) and of some special physical properties (e.g. the Curie temperature  $T_C$  and Bohr magneton number  $B_{MAGN}$  (or  $B_M$ ) for magnetic ordering,  $V_0-V_A-V_C-V_K$  parameters for volume contributions, Born functions for hypothetical electrostatic interaction in an aqueous solution phase) stored in connection with the phase.

### **About Phase Type Code**

A G phase (gaseous mixture) or an A phase (aqueous solution) is usually treated as a substitutional phase without sublattice, and that an L phase (ordinary liquid solution) is normally (but not always) modeled as a substitutional phase without sublattice, too.

The F and B phase-types are useful for ordered FCC (or HCP) and BCC solution phases handled by the so-called Four Substitutional-Sublattice Ordering Model, which always requires that the solution phase must have four sublattices for substitutional ordering and can additionally have an interstitial sublattice.

For ordered FCC or HCP phases, these four substitutional sublattices represent four corners of the regular tetrahedron on these lattices all of which are the nearest neighbours. A *Normal 4-Sublattice Model* requires that all the G parameters for each of end-members with the same elements but distributed on different sites be given separately. However, as these corners are identical lattice points, the phase-type option F means that the G parameters need be given only once. The possible permutations are handled automatically. To be more clarified: An A-B binary solution phase (with the element A locates on one sublattice site and B on three sublattice sites) treated by the Normal 4-Sublattice Model has to have 4 G parameters for 4 end-members, i.e. G(phase,A:B:B:B), G(phase,B:A:B:B), G(phase,B:B:A:B) and G(phase,B:B:B:A), because of that in the general case these G parameters can be different from each other. But for the FCC and HCP orderings, these are identical and thus all G parameters of such end-members need be given only once, and the possible permutations are then automatically handled by the GIBBS module. This significantly simplifies the usage of this model (*Four Substitutional-Sublattice Ordering Model*) in multicomponent alloys.

There are restrictions on the excess parameters allowed with the phase-type option F. You can only have excess parameters of these types:

L(phase,A,B:\*:\*:\*;0...9)

L(phase,A,B:C,D:\*:\*;0...9)

The asterisk \* means that the interaction parameter is independent on the constituents on its corresponding sublattice. No ternary interaction parameters (i.e. with three elements on one sublattice site) are allowed. The reason for this restriction is that it would be too complicated to handle all possible permutations. In the current *Four Substitutional Sublattice Ordering Model*, the binary interaction between A and B atoms is thus independent of the constituents on the other sublattices, where there are many other parameters to model the composition-dependence of the Gibbs energy (both in the ordered and disordered parts of the phase). The model for these ordered phases are always partitioned in a disordered part (with a single substitutional sublattice) and an ordered part (with four substitutional sublattices for ordering).

For ordered BCC phases, the phase-type option B means the same thing but the situation is more complicated, as the 4-sublattice ordering phase represents an irregular tetrahedron with two pairs of sites that are next nearest neighbours. Thus, for an A-B binary solution phase (with the element A locates on two sublattice site and B on other two sublattice sites) treated by the Normal 4-Sublattice Model, the end-member described by the G

(phase,A:A:B:B) term has four nearest neighbour bonds between A and B atoms, whereas the end-member described by the G(phase,A:B:A:B) term has two nearest neighbour bonds between A and B atoms and two next nearest neighbour bonds



For detailed reference visit the Crystal Lattice Structure web page  
<https://www.nrl.navy.mil/mstd/6390>.

The first end-member (described by the G(phase,A:A:B:B) term) represents B2- ordering and the second (described by the G(phase,A:B:A:B) term) stands for B32-ordering. There are two permutations of the G(phase,A:A:B:B) term and four permutations of the G(phase,A:B:A:B) term, automatically conducted in the Four Substitutional-Sublattice Ordering Model. And there are also two kinds of reciprocal interaction parameters, i.e.

$L(\text{phase}, A, B : C, D : * : * ; 0 \dots 9)$   $L(\text{phase}, A, B : * : C, D : * ; 0 \dots 9)$

An advanced feature with the phase-type options F and B is that a composition set that represents the solution phase has a suffix (indicating that it is really as an ordered or disordered phase) that is automatically added to its phase name in some listings of equilibrium calculations (when performing either single-point or stepping or mapping calculations, and when plotting the calculated property diagrams or phase diagrams). Such suffix indications can be:

- `_L12` or `_L10` for ordered FCC
- `_A1` for disordered FCC
- `_B2`, `_B32`, `_D03` or `_L21` for ordered BCC
- `_A2` for disordered BCC
- `_D019` and `_B19` for ordered HCP
- `_A3` for disordered HCP

Solution Phase	Suffix for Disordered Phase	Suffix for Ordered Phase
FCC PHASE	PHASE_A1	PHASE_L12
		PHASE_L10
BCC PHASE	PHASE_A2	PHASE_B2
		PHASE_B32
		PHASE_D03
		PHASE_L21

Solution Phase	Suffix for Disordered Phase	Suffix for Ordered Phase
HCP PHASE	PHASE_A3	PHASE_D019
		PHASE_B19

## **ADD\_COMMENT**



This command is for the Gibbs Energy System (GES) module.

Use the ADD\_COMMENT command to add a comment or make notes about parameters.

## AMEND\_ELEMENT\_DATA

 This command is for the Gibbs Energy System (GES) module.

The data for an element (in the SER, the Stable Element Reference, state) can be changed by this command. It should only be used for the elements that do not have any data in the database because the element data available in the database is set by the database-developer for the purpose of internal consistency. Except for the mass, the other values have no influence on the calculations.

Syntax	AMEND_ELEMENT_DATA
Prompts	Element name: <Element name> Specify the name of the element for which you want to change the data.
	New stable element reference /ABCD/: <Name of ser>  Press <Enter> to accept the default SER or specify a new SER for the element. Important: The default name should not be changed if this data is retrieved from a database. This name is used when parameters for a phase are listed and the database assumes that the stable element reference is the same as in the database. Only if the element's data have not been fetched from a database, e.g. entered manually, can you enter a new SER.
	New atomic mass /xx.xxxx/: <yyyyyy>  Press <Enter> to accept the default atomic mass or specify a new value for the element. The atomic mass of the element is given in g/mol.
	New h(298.15)-h(0) /xxx.xxx/: <yyyyyy>  Press <Enter> to accept the default H(298.15)-H(0) or specify a new value for the element. H(298.15)-H(0) is the enthalpy difference between 298.15 K and 0 K for the element in its SER state.
	New s(298.15) /xx.xxxx/: <yyyyyy>  Press <Enter> to accept the default S(298.15) or specify a new value for the element. S(298.15) is the absolute value of the entropy at 298.15 K for the element in its SER state.
	Default element reference state symbol index /#/: <index>  <div style="border: 2px solid red; padding: 10px; margin: 10px 0;">  The index only changes the symbol, not any value. Normally the index is set correctly by the database. Only when manually entering data, you must set the index to get the correct symbol.         </div> Specify an index for the default listing parameters (symbol), or press <Enter> to accept the pre-set index. 0 is for G, 1 is for H <sub>298</sub> and 2 is for H <sub>0</sub> . The index is to define the symbol printed in parameter listings. The symbol can be: <ul style="list-style-type: none"> <li>• G the data are referred to Gibbs energy at a variable temperature (also called <i>lattice stability</i>).</li> </ul>

---

Syntax	AMEND_ELEMENT_DATA
	<ul style="list-style-type: none"><li>• <math>H_{298}</math> the data are referred to the enthalpy of the element at 298.15 K and 1 bar.</li><li>• <math>H_0</math> is the same as <math>H_{298}</math> but at the temperature 0 K.</li></ul>

## AMEND\_PARAMETER

-  This command is for the Gibbs Energy System (GES) module.
-  There is also a PARROT module command with the same name.

Use this command to interactively change/amend the temperature-pressure function of a parameter. This is useful to correct typing errors because the old function is made available on the terminal for interactive editing.



### About the ENTER\_PARAMETER Command

Syntax	AMEND_PARAMETER
Prompts	<p>Parameter: &lt;Parameter name&gt;</p> <p>Specify a correct parameter name. If a parameter name is not acceptable or you only press &lt;Enter&gt;, the error message displays:</p> <p>Error, please re-enter each part separately</p> <p>The program prompts for separate input for each part for a parameter name.</p>
	<p>Identifier (/X/): &lt;G or L, or TC, or BM, or V0 or VA or VC or VK&gt;</p> <p>Specify one of these legal identifiers as described for ENTER_PARAMETER.</p>
	<p>Phase name (/ABCD/): &lt;Phase name&gt;</p> <p>Specify the phase name.</p>
	<p>Constituent (in sublattice # /ABC/): &lt;Species name&gt;</p> <p>Specify the constituent name.</p>
	<p>Interacting constituent (in sublattice # /xyz/): &lt;Species name&gt;</p> <p>Specify the interacting constituent name: If there is no interacting constituent, press &lt;Enter&gt;.</p>
	<p>Interacting constituent (in sublattice # /xyz/): &lt;Species name&gt;</p> <p>If there are more than one interacting constituents, specify them; otherwise press &lt;Enter&gt;.</p>
	<p>Degree /#/ : &lt;Degree&gt;</p> <p>Specify a numerical number as the degree of the phase parameter.</p>
	<p>After the parameter name is specified correctly, the program lists its current definition (either present in the database or defined by the ENTER_PARAMETER command), such as:</p> <pre>L (PHASE2, AL, MG; 1) = 298.15&lt;T&lt;2000.00: +5000 2000.00&lt;T&lt;4500.00: +4500 4500.00&lt;T&lt;6000.00: +4000</pre> <p>Then you are prompted to change the parameter definition, as shown below:</p>

Syntax	AMEND_PARAMETER
	<p>Do you want to change the number of ranges /No/:</p> <p>Enter Y if you want to change the number of ranges for the chosen function, or change some of the temperature limits in the definition. Then retype both the low/high temperature limits and functions.</p> <p>If you do not want to change the number of ranges but want to change the function(s) in one or more ranges, press &lt;Enter&gt; to accept the default N, then the whole definition of the chosen parameter in all ranges (if any) is listed on screen, such as:</p> <pre>DIFFERENT FUNCTIONS IN THESE RANGES 298.15&lt;T&lt;2000.00 2000.00&lt;T&lt;4500.00 4500.00&lt;T&lt;6000.00</pre> <p>The prompt is given:</p> <p>Do you want to change range limits /No/:</p> <p>If there is more than one range, this question is prompted press &lt;Enter&gt;.</p>
	<p>Range number (0 to exit) /0/: &lt;Range number&gt;</p> <p>If the function of a parameter is different in two or more temperature ranges, you must specify the range of the function of which you want to amend. Or press &lt;Enter&gt; or type 0 to exit this command without making any changes.</p>
	<p>Function:</p> <p>The previous function is available for editing. The editing is performed within the general subroutine FOOLED, as described in <a href="#">AMEND_SYMBOL</a>. This routine prompts as follows:</p> <pre>1:++&gt;</pre> <p>The prompt consists of the current position in the string and the character at that position between colons, (::).</p> <h3>Commands</h3> <p>These commands can be given:</p> <ul style="list-style-type: none"> <li>• Help: ?</li> <li>• Move CP to last or first character: &lt;+/-&gt; A</li> <li>• Delete characters from CP: &lt;+-#characters&gt; D</li> <li>• Exit: E</li> <li>• Find: &lt;#occurrences&gt; F&lt;string&gt;@</li> <li>• Insert: I&lt;string&gt;@</li> <li>• Move: &lt;+-#positions&gt; M</li> <li>• Restore string: R</li> <li>• Substitute: S&lt;OLD&gt;@&lt;NEW&gt;@</li> <li>• Type string: T</li> </ul> <p>Where CP denotes the current position in a string, # means number of, @ is a terminator of an</p>

---

Syntax	AMEND_PARAMETER
	input or search string.
	 When the string is typed the character at the current position is replaced by an underscore _.
	To finish the editing of the current function, type <code>␣</code> at the prompt.

## AMEND\_PHASE\_DESCRIPTION



This command is for the Gibbs Energy System (GES) module.



Also see [Solution Models in the GIBBS \(GES\) Module](#) for detailed information about this command and its options.

Specify/amend phase descriptions if a phase has a miscibility gap, uses a special excess energy model, or has a specific additional contribution to its Gibbs energy, for example.

Syntax	AMEND_PHASE_DESCRIPTION
Prompts	<p>Phase name: &lt;Phase name&gt;</p> <p>Specify the name of the phase.</p> <p>Amend what /Composition_Set/: &lt;Subject&gt;</p> <p>Several subjects for the phase can be amended but most often this command is used to enter two or more composition sets for a phase. If a phase has a miscibility gap it is necessary to have two composition sets, one for each possible composition that can be stable simultaneously.</p> <p>Enter a question mark at the prompt to get a list of all possibly amended subjects for a phase.</p>
	<p> There are no additional prompts for DEBYE_HUCKEL, HKF_ELECTROSTATIC, GLASS_TRANSITION, QUASICHEM_FACT00, QUASICHEM_ISRID, REMOVE_ADDITION, and DEFAULT_STABLE.</p>
RENAME_PHASE	<p>New phase name /ABCD/: &lt;Phase name&gt;</p> <p>Give a new phase name for the chosen phase, or press &lt;Enter&gt; to keep the default shown.</p>
Site_ratios	<p>Sites in first sublattice /XX/ : &lt;YY&gt; Sites in second sublattice /XX/ : &lt;YY&gt;</p> <p>Specify the site numbers for each of the prompted sublattices for the current phase. Press &lt;Enter&gt; to accept the previous definitions.</p>
New_constituent	<p>Sublattice /#/: &lt;Sublattice number&gt;</p> <p>Specify the sublattice where the new constituents are located.</p> <p>Species: &lt;Species name&gt;</p> <p>Give a valid species name.</p>
Composition_sets	<p>New highest set number /#/: &lt;Set number n&gt;</p> <p>The default value (#) is usually one higher than the current value. All phases have initially one composition set. If a lower value (i.e. lower than the default one) is given, that specific composition sets are deleted.</p>

Syntax	AMEND_PHASE_DESCRIPTION
	<div style="border: 2px solid blue; padding: 10px; margin-bottom: 10px;">  You cannot take away the first composition set. </div> <p>Major constituent(s) for sublattice #: /AB/: &lt;Major constituent(s)&gt;</p> <p>Specify the new major constituent(s) for the sublattice #, or press &lt;Enter&gt; to accept the default which was automatically set according to the specified composition set of the phase.</p> <p>This prompt is repeated for all available sublattices in the chosen phase. The major constituents in each sublattice can be given. This may simplify giving start values when calculating the equilibrium as phases with miscibility gaps should have different major constituents for each composition set.</p>
Major_ constituent	<p>Composition set /1/: &lt;Composition set number&gt;</p> <p>Give the composition set (digit number) for the chosen phase, or press &lt;Enter&gt; if you want to set major constituents for the specified composition set.</p> <p>Major constituent(s) for sublattice #: /AB/: &lt;Major constituent(s)&gt;</p> <p>Specify the new major constituent(s) for the sublattice #, or press &lt;Enter&gt; to accept the default which is automatically set according to the specified composition set of the phase.</p> <p>This prompt is repeated for all available sublattices in the chosen phase for the specified composition set.</p> <p>The major constituents in each sublattice can be specified. This is useful in order to make calculations converge faster and more easily (because it may simplify giving start values when calculating the equilibrium as those phases with miscibility gaps should have different major constituents for each composition set). The databases often set major constituents for several phases automatically when data are retrieved.</p>
Fraction_ limits	<p>Element : &lt;el1&gt;</p> <p>Low fraction limit /0/ : &lt;Appropriate low limit&gt;</p> <p>High fraction limit /1/ : &lt;Appropriate high limit &gt;</p> <p>Element : &lt;el2&gt;</p> <p>Low fraction limit /0/ : &lt;Appropriate low limit&gt;</p> <p>High fraction limit /1/ : &lt;Appropriate high limit&gt;</p> <p>Element : &lt;eln&gt;</p> <p>Low fraction limit /0/ : &lt;Appropriate low limit&gt;</p> <p>High fraction limit /1/ : &lt;Appropriate high limit &gt;</p>
Magnetic_ ordering	<p>The antiferromagnetic factor /XX/: &lt;YY&gt;</p> <p>Specify the anti-ferromagnetic factor for the chosen phase. This should be <math>-1</math> for BCC phase and <math>-3</math> for all other phases (FCC, HCP, etc.).</p> <p>Short range order fraction of the enthalpy /XX/: &lt;YY&gt;</p> <p>The magnetic ordering is a second-order transformation and part of the enthalpy due to this transformation is due to short-range order. This value is the fraction of the total enthalpy that is due to short-range ordering above the magnetic transition temperature.</p> <p>The default value (xx) is 0.40 for BCC phase, and 0.28 for all other phases (FCC, HCP, etc.).</p>

Syntax	AMEND_PHASE_DESCRIPTION
Disordered_part	<p>Disordered phase name: &lt;Phase name&gt;</p> <p>Give the disordered phase name for which there is no ordering sublattice.</p>
Excess_model	<p>Model name /ABCDEFG/: &lt;Model name&gt;</p> <p>The default model is the pre-set model for the solution phase, normally the REDLICH-KISTER_MUGGIANU model, or choose another model for the phase to be amended:</p> <ul style="list-style-type: none"> <li>• REDLICH-KISTER_MUGGIANU</li> <li>• REDLICH-KISTER_KOHLER</li> <li>• FLORY-HUGGINS POLYMER MODEL</li> <li>• MIXED-EXCESS-MODELS (R-K default)</li> <li>• HKF</li> <li>• PITZER</li> <li>• CENTRAL_ATOM_MODEL</li> </ul> <p>The MIXED-EXCESS-MODELS option has sub-options:</p> <p>First (the independent) constituent: &lt;Constituent name&gt;</p> <p>Second (the dependent) constituent: &lt;Constituent name&gt;</p> <p>Specify the binary pair of constituents in the current substitutional solution phase (the first one as the so-called independent constituent, and the second one as the dependent constituent), for which you wish to change the binary excess model from the default REDLICH-KISTER model to another model (LEGENDRE or POLYNOM).</p> <p>You are prompted with the first sub-option (i.e. <i>First (the independent) constituent</i> after you have specified the desired binary Excess model type. To make further changes of binary excess model for other specific binary pairs in the current substitutional solution phase, press &lt;Enter&gt; (implying that there are no more changes of binary excess model for all other possibly-remaining binary pairs that shall still use the default REDLICH-KISTER model).</p> <p>Excess model type: /Legendre/: &lt;Desired binary excess model type&gt;</p> <p>For the currently-specified binary pair, choose one the legal binary excess models: LEGENDRE, POLYNOM or REDLICH-KISTER. If the default binary excess model REDLICH-KISTER is used for a certain binary pair, you can either specify the model name (or in short as R-K) or press &lt;Enter&gt;. After this sub-option, it returns to the first sub-option <i>First (the independent) constituent</i>: either for further change(s) of binary excess model of any other binary pair(s) or for termination (by pressing &lt;Enter&gt;) in the MIXED-EXCESS-MODELS option.</p>
Ternary_extrapolat	<p>Extrapolation method: /Toop-Kohler/: &lt;Ternary model&gt;</p> <p>Choose a ternary extrapolation model for the current solution phase:</p> <ul style="list-style-type: none"> <li>• TOOP-KOHLER for Toop-Kohler Extrapolation Model</li> <li>• KOHLER-ALL for Kohler Extrapolation Model</li> <li>• MUGGIANU_RESTOR for Muggianu-Restor Extrapolation Model</li> </ul> <p>The TOOP-KOHLER option further prompts:</p> <p>Constituent in sublattice #: &lt;Basis constituent name&gt;</p> <p>First interaction constituent: &lt;Interacting constituent name&gt;</p>

Syntax	AMEND_PHASE_DESCRIPTION
	<p>Toop constituent: &lt;Toop constituent name&gt;</p> <p>Specify the so-called Kohler constituents (entered as the basis constituent and first interacting constituent) and Toop constituent on a given sublattice in the current solution phase.</p>
Quasichem_ionic	<p>To use the Quasichemical Model for the chosen liquid phase, you must first enter it as an ionic liquid phase, and then use this amending option. It creates a completely new phase, the original IONIC_LIQUID phase remains in the system and is not changed.</p> <p>Number of bonds/atom /2/: &lt;Number of bonds per atom&gt;</p> <p>Specify the Z value (number of bonds per atom).</p>
Status_bits	<p>Correct the phase status bits. It is important to fully understand this command before using it:</p> <p>New status word (in hexadecimal) /04204000/: &lt;New status bits&gt;</p> <p> <a href="#">LIST_STATUS</a></p> <p>Normally, these should never be changed in this way, because such phase status bits are automatically set on when all the phase descriptions (predefined in databases and amended in the GIBBS module) are defined properly. The only exceptions are for the Kapoor-Frohberg-Gaye cell model for liquid SLAG solution phase, which can be set only as 04208000; and for the complete Revised HKF AQUEOUS solution model, which can be set as 02084000.</p>

## AMEND\_SYMBOL

 This command is for the Gibbs Energy System (GES) module.

 The same command is also available for the PARROT module.

Interactively calculate the current values of a function or table (predefined by the switched database or previously entered with ENTER\_SYMBOL) at the current temperature and pressure condition, and also possible to modify the definitions of variables or functions.



The current values of the temperature and pressure used to calculate the functions or tables are the ones listed with LIST\_STATUS. There is no way to change the current temperature and pressure values interactively.

To modify the definitions of entered symbols (variable, functions or parameters), the performance of this command is slightly different one from another, as described below:

- For variables, the values can be changed.
- For functions the low and high temperature limits in various ranges (if any), as well the expressions, can be changed.
- For parameters the low and high temperature limits in various ranges (if any), as well the expressions, can be changed.

Syntax	AMEND_SYMBOL
Prompts	<p>Name: &lt;Symbol name&gt;</p> <p>Specify the name of an entered symbol.</p> <p>For tables and functions (or parameters which are treated as functions after these are entered by the ENTER_SYMBOL command with the <code>Parameter</code> keyword, but not by the ENTER_PARAMETER command), after the symbol name is specified here, the program automatically calculates the values under the current temperature and pressure conditions, and lists the current values, such as:</p> <pre>FUNCTION VALUE  2.52500000E+01 TABLE VALUE    1.56000000E+02</pre> <p>For variables and functions (or parameters), there are additional prompts and depend on the symbol type.</p> <p>For a variable, its current value is displayed, and you can change it to a new value:</p> <pre>Value /Current value/: &lt;New value&gt;</pre> <p>For a function (or a parameter entered as a symbol):</p> <pre>Do you want to change the number of ranges /No/:</pre>

Syntax	AMEND_SYMBOL
	<p>Enter Y to change the number of ranges for the function, or change some of the temperature limits in the definition, then retype both the low/high temperature limits and functions (see all the remaining details in the ENTER_SYMBOL command).</p> <p>If you do not want to change it, press &lt;Enter&gt;. The definition of the chosen function in all ranges (if any) is listed, for example:</p> <pre>DIFFERENT FUNCTIONS IN THESE RANGES 298.15&lt;T&lt;2000.00 2000.00&lt;T&lt;4500.00 4500.00&lt;T&lt;6000.00</pre> <p>and this message displays:</p> <pre>Do you want to change range limits /No/:</pre> <p>If a function is different in two or more temperature ranges, you must specify the range of the function of which you want to amend. Or press &lt;Enter&gt; or type 0 to exit this command without making any change.</p>
	<pre>Range number (0 to exit) /0/: &lt;Range number&gt; Function:</pre> <p>The previous function is available for editing. The editing is performed within the general subroutine FOOLED. This routine prompts as follows:</p> <pre>1:+:&gt;</pre> <p>The prompt consists of the current position in the string and the character at that position between colons (::).</p> <h3>Commands</h3> <p>These commands can be given:</p> <ul style="list-style-type: none"> <li>• Help: ?</li> <li>• Move CP to last or first character: &lt;+/-&gt; A</li> <li>• Delete characters from CP: &lt;+-#characters&gt; D</li> <li>• Exit: E</li> <li>• Find: &lt;#occurrences&gt; F&lt;string&gt;@</li> <li>• Insert: I&lt;string&gt;@</li> <li>• Move: &lt;+-#positions&gt; M</li> <li>• Restore string: R</li> <li>• Substitute: S&lt;OLD&gt;@&lt;NEW&gt;@</li> <li>• Type string: T</li> </ul> <p>Where CP denotes the current position in a string, # means number of, @ is a terminator of an input or search string.</p>

---

Syntax	AMEND_SYMBOL
 When the string is typed the character at the current position is replaced by an underscore _.	
To finish the editing of the current function, type <code>E</code> at the prompt.	

## CHANGE\_STATUS



This command is for the Gibbs Energy System (GES) module.

The status of an element or species or phase can be either ENTERED or SUSPENDED. The suspended status can be either implicit or explicit. The implicitly suspended status can be set e.g. for a species if any of the elements in its chemical formula is explicitly suspended. A species that is implicitly suspended becomes entered automatically if all its elements are set entered. After this command, a message shows which elements/species/phases are suspended or restored (entered). Consequently, the status bits for the specified elements or species or phases are changed, as listed with LIST\_STATUS.

Syntax	CHANGE_STATUS
Prompts	For Element, Species or Phase /Species/: <Keyword> Specify the keyword (element, species or phase).
	Suspend /Y/: By default the status is changed from ENTERED to SUSPENDED. If an element is suspended all species with this element become implicitly SUSPENDED too. A phase may become implicitly suspended if all its constituents or all constituents in a sublattice are suspended.
	List of elements/species/phases: <Name of elements or species or phases> Specify the names or indices of those elements or species or phases that shall become suspended or active. For names, these should be separated by a space and terminated with a semicolon (;) or an empty line. For indices it is possible to give a range by separating two numbers by a hyphen (-). The list should be terminated by a semicolon (;). Example: 5 1 7-12 FE;

## ENTER\_ELEMENT

Specify a system interactively. The program searches the currently switched or preset database for data for the given elements. The data for the elements in the database are the:

- mass in g/mol
- name of the Selected Element Reference State (SER) which normally is the stable phase for the element at 298.15 K
- enthalpy difference for the element in the SER state at 298.15 K and zero K
- absolute entropy for the element in the SER state at 298.15 K.

The two predefined elements, i.e. electrons and vacancies, have the chemical symbols /- and  $v_A$ , respectively. Initially, these are suspended but can be entered either by this command or CHANGE\_STATUS.

If an aqueous solution phase is involved, in the GIBBS module you enter a special aqueous electron called, ZE. This is specially designed for appropriately calculating the standard electric potential in the aqueous solution system.

Syntax	ENTER_ELEMENT
Prompt	<p>Element name: &lt;Element name&gt;</p> <p>Specify several elements on one line. The name of an element is its chemical symbol. The chemical symbols must be separated by spaces. Fictitious element names are legal but naturally no data are found in the database for them.</p>
	<div style="border: 2px solid blue; padding: 10px;"> <p> An element name (maximum 2 characters) can either have its first letter in upper and the second (if any) in lower case (i.e. Lower Case Mode) or both letters can be in upper or lower case (i.e. Upper Case Mode). The Upper or Lower Case Mode is selected by the command REINITIATE, which removes all data, and should be executed before any other command.</p> </div>

## ENTER\_FUNCTION



This command is available with the GES and TAB modules.

Used to define a new thermodynamic function to be included in the sixth column of tabular output. The last entered function is listed in the 6th column of all tables, and can be plotted (if choosing the Plot Column value 6) in all graphs, for all subsequently tabulated substances or reactions.

Syntax	ENTER_FUNCTION
Prompt	Name: <Name> Function: <Definition>
	A limited number of state variables, G, H, S, T, P, V and H298 can be used in the function definition. For example, $-(G-H)/T$ can be defined as a function which is identical to S for a substance or $\Delta S$ for a reaction, $G+T*S-P*V$ can be defined as a function which is U (internal energy) for a substance or $\Delta U$ (internal energy change) for a reaction.
Options	Name of the function (maximum 8 characters) and Definition of the function. A long function can be typed over several lines. A function should be terminated by a semicolon (;) or an empty line.

### Example

For the pure substance Fe (using the SSUB database), by entering a function as  $G+T*S-P*V$ , the following table is obtained by typing <Enter> in the prompt Output file /Screen/:, and the figure by typing 6 in the prompt Plot column /2/:

```
TAB: ENTER-FUNCTION
Name: InEnergy
Function: G+T*S-P*V;
TAB: TABULATE_SUBSTANCE
Substance (phase): Fe
Pressure /100000/: <Enter>
Low temperature limit /298.15/: <Enter>
High temperature limit /2000/: <Enter>
Step in temperature /100/: <Enter>
Output file /try1/: try2
Graphical output? /Y/: <Enter>
Plot column? /2/: 6
```

This gives the following output:

O U T P U T F R O M T H E R M O - C A L C

Column 6: InEnergy G+T\*S-P\*V

Phase : FE\_S Pressure : 100000.00

Specie: FE

\*\*\*\*\*

T (K)	Cp (Joule/K)	H (Joule)	S (Joule/K)	G (Joule)	InEnergy
298.15	2.48446E+01	2.17972E-06	2.72800E+01	-8.13353E+03	2.17972E-06
300.00	2.48905E+01	4.60049E+01	2.74338E+01	-8.18414E+03	4.60049E+01
400.00	2.71299E+01	2.64957E+03	3.49085E+01	-1.13138E+04	2.64957E+03
500.00	2.93561E+01	5.47211E+03	4.11976E+01	-1.51267E+04	5.47211E+03
600.00	3.19293E+01	8.53245E+03	4.67701E+01	-1.95296E+04	8.53245E+03
700.00	3.50985E+01	1.18777E+04	5.19207E+01	-2.44667E+04	1.18777E+04
800.00	3.92042E+01	1.55830E+04	5.68623E+01	-2.99068E+04	1.55830E+04
900.00	4.49645E+01	1.97726E+04	6.17903E+01	-3.58387E+04	1.97726E+04
1000.00	5.42147E+01	2.46891E+04	6.69619E+01	-4.22728E+04	2.46891E+04
1100.00	4.55851E+01	2.99025E+04	7.19412E+01	-4.92328E+04	2.99025E+04 \$
Stable phase is FE_S2					
1200.00	3.40840E+01	3.51037E+04	7.64466E+01	-5.66322E+04	3.51037E+04
1300.00	3.49398E+01	3.85549E+04	7.92086E+01	-6.44162E+04	3.85549E+04
1400.00	3.57994E+01	4.20918E+04	8.18293E+01	-7.24692E+04	4.20918E+04
1500.00	3.66636E+01	4.57149E+04	8.43287E+01	-8.07780E+04	4.57149E+04
1600.00	3.75330E+01	4.94247E+04	8.67226E+01	-8.93314E+04	4.94247E+04 \$
Stable phase is FE_S3					
1700.00	4.05217E+01	5.41173E+04	8.95609E+01	-9.81363E+04	5.41173E+04
1800.00	4.12595E+01	5.82055E+04	9.18975E+01	-1.07210E+05	5.82055E+04 \$
Stable phase is FE_L					
1900.00	4.60000E+01	7.74165E+04	1.02377E+02	-1.17099E+05	7.74165E+04
2000.00	4.60000E+01	8.20165E+04	1.04736E+02	-1.27456E+05	8.20165E+04

## ENTER\_PARAMETER

 This command is for the Gibbs Energy System (GES) module.

 There is also a PARROT module command with the same name.

 [About the ENTER\\_PARAMETER Command](#)

Use this command to enter TP-function(s) for a specific parameter for a phase interactively with this command. If there is already a parameter expression defined (for this parameter), that is deleted and replaced with newly entered one.

The entered TP-Function(s) for the parameter can be changed later with the command AMEND\_PARAMETER.

Syntax	ENTER_PARAMETER
Prompts	<p>Parameter: &lt;Parameter name&gt;</p> <p>As explained in <a href="#">About the ENTER_PARAMETER Command</a>, specify a correct and complete Parameter Name, which should contain all the necessary parts of the general form:</p> <p>&lt;Identifier&gt;&lt;Phase&gt;,&lt;Constituent array&gt;;&lt;Digit&gt;</p> <p>If a parameter name is not acceptable or &lt;Enter&gt;, is pressed, a message displays:</p> <pre>*** Error, please re-enter each part separately</pre> <p>and you are prompted for input for each required part for a parameter name.</p>
Options	Description or Information
Identifier	<p>Identifier (/X/): &lt;G or L, or TC, or BM, or V0 or VA or VC or VK&gt;</p> <p>If this command is used one or more times, the previous value on this prompt is set as default. Press &lt;Enter&gt; for the same type identifier or specify a new type.</p>
Phase name	<p>Phase name (/ABCD/): &lt;Phase name&gt;</p> <p>Each parameter is valid for a specific phase only. The name of that phase must be supplied (maximum 24 characters). The name can be abbreviated.</p>
Constituent	<p>Constituent (in sublattice # /ABC/): &lt;Species name&gt;</p> <p>A parameter is identified by the constituents on a specified sublattice site of the given phase, the stoichiometric coefficients of which are multiplied with the parameter. The name of the constituent can be abbreviated.</p> <div style="border: 2px solid blue; padding: 10px; margin: 10px 0;">  It is the species name, not the stoichiometric formula required here. </div> <p>If this command is used one or more times, the previous value on this prompt is set as default. Accept it by pressing &lt;Enter&gt; if the constituent is the same, or specify a new species name.</p>

Syntax	ENTER_PARAMETER
Interacting constituent	<p>For phases with several sublattices, the program asks for one constituent in each sublattice.</p> <p>Interacting constituent (in sublattice # /XYZ/): &lt;Species name&gt;</p> <p>If this command is used one or more times, the previous value on this prompt is set as default. Press &lt;Enter&gt; to accept it if the constituent is the same, or specify a new species name.</p> <div data-bbox="410 436 1395 575" style="border: 2px solid blue; padding: 10px; margin: 10px 0;">  To cancel the default value of the interacting constituent type <code>NONE</code> or the name of another constituent. </div> <p>This question is repeated until all the interested interacting constituent(s) on a specific sublattice in the phase are specified, and finally an &lt;Enter&gt; is enforced.</p>
Degree	<p>Degree /#/ : &lt;Degree&gt;</p> <p>Degree is model-dependent. Specify an integer number (a value from 0 through 9) as the degree of composition-dependent interaction contribution for the phase parameter. This is valid for excess energy (<code>L</code>), Curie temperature (<code>TC</code>) and Bohr magneton number (<code>BMAGN</code>), as well as for volume-related parameters (<code>V0</code>, <code>VA</code>, <code>VC</code> or <code>VK</code>).</p> <p>For binary interaction parameters, the degree is usually the power in the Redlich-Kister expression. For ternary interaction parameters, it is usually the Hillert ternary index.</p> <p>For a standard <code>G</code> parameter for a pure component (end-member) its degree should be always 0 and this prompt should not display.</p>
Phase parameter	<p>After this prompt, the program echoes on the screen the full TP-Function of the phase parameter.</p> <p>Low temperature limit /298.15/: &lt;Lowest temperature limit in K&gt;</p> <p>Specify the lowest temperature limit (in Kelvin), or the lowest-pressure limit (in Pascal but entered as a negative number), for the current TP-Function.</p> <p>Function: &lt;Definition for a function&gt;</p> <p>A TP-Function consists of terms in T and P.</p> <p>&amp; &lt;Continuation of the definition for the current function&gt;</p> <p>Continuation of a TP-Function definition.</p> <p>High temperature limit /6000/: &lt;High temperature limit in K&gt;</p> <p>Specify the high temperature limit (in Kelvin), or the high-pressure limit (in Pascal; only if the <i>lowest limit</i> is entered as a negative number) for the current TP-Function.</p> <p>Any more ranges? /N/:</p> <p>Enter Y for more function(s) or N to end this command.</p>

## ENTER\_PHASE



See [About the ENTER\\_PHASE Command](#) for details about the phase-type and other prompts.

With this command, the phase name, phase-type, sublattice number, and constituents or constituent array(s) for the phase are entered into the GIBBS workspaces. However, for the remaining parts of thermodynamic descriptions (i.e. thermochemical and physical parameters, excess model, and possibly additional contribution term) for the phase, the GIBBS commands ENTER\_PARAMETER and AMEND\_PHASE\_DESCRIPTION, should be used sequentially.

Normally, data for a system are retrieved from the preset database after the elements and species are specified. This is automatically done through GET\_DATA. In some cases, specify a phase interactively.

Syntax	ENTER_PHASE
Options	Description and Information
Phase name	<p>Name of phase: &lt;Phase name&gt;</p> <p>The phase name is any string starting with a letter and containing only letters, digits or the underscore character. The phase name must be unique.</p> <p>Type code: &lt;Phase-type code&gt;</p> <p>The <code>Phase-type</code> code must be specified for the phase if it is not an ordinary solution phase. Press &lt;Enter&gt; if the phase is ordinary.</p>
Phase-type	
G	<p>Gaseous mixture phase.</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;">  There may only exist one gas phase in a system.         </div>
A	Aqueous solution phase.
L	Liquid solution phase but not A (aqueous) or Y (ionic liquid)
Y	Ionic liquid solution phase (that is specially treated by the Ionic Two Sublattice Liquid Model).
I	A phase with charged species but not G (gaseous), A (aqueous) or Y (ionic liquid).
F	An ordered FCC or HCP solution phase with four substitutional sublattices (additionally, such a phase can also have an interstitial sublattice).
B	An ordered BCC solution phase with four substitutional sublattices (additionally, such a phase

Syntax	ENTER_PHASE
	can also have an interstitial sublattice).
Sublattice number	<p>Number of sublattices /1/: &lt;Sublattice number&gt;</p> <p>Phases with stoichiometric constraints usually have two or more sublattices. On each sublattice one or more species can be entered as constituents. The number of sublattices must not exceed 10. Substitutional phases with no sublattices are treated as phases with one sublattice.</p> <p>Number of sites on sublattice # /1/: &lt;Site number&gt;</p> <p>For phases with sublattices, the ratio of the sites on each sublattice (donated with #) must be given. The program asks for values for all sublattices (by repeating this question for each sublattice), but these may have a common factor extracted. It is recommended to use integer numbers as sites if possible. For substitutional phases with no sublattices, this question does not show up.</p>
Constituents	<p>Name of constituent: &lt;Constituent name(s)&gt;</p> <p>For each sublattice specify at least one species as constituent. Several constituents can be given consequently (on the repeated prompt) or on the same line separated by a space. This question is repeated till pressing &lt;Enter&gt; or giving a semi-colon (;), meaning that the list of constituents is terminated by (;) or an empty line.</p> <p>Will you add constituents later /N/:</p> <p>For Y add other constituents to the phase by specifying new components in e.g. an ENTER_PARAMETER command.</p> <p>If illegal constituents are used, you get a message e.g. when parameters are entered.</p> <p>Do you want a list of possible parameters /N/:</p> <p>Enter Y for a list of all possible parameters up to the fifth interaction order for the entered phase.</p>

## ENTER\_SPECIES

Specify a species from the already entered elements. The stoichiometry of the species is the chemical formula of the species. For each species, its name and chemical formula must be given.



All elements are automatically entered simultaneously as species. The constituents of a phase must be species, but there is no thermochemical data associated with the species except as constituents of a phase.

Syntax	ENTER_SPECIES
Prompt	<p>Species name: &lt;Species name&gt;</p> <p>Enter a unique species name. This is usually its chemical formula but it can be any string which starts with a letter and contains only letters or digits. Parenthesis () and an underscore _ are permitted. The combination /- or /+ used to denote charge.</p> <p><b>Species Names Examples</b></p> <p>NA<sub>2</sub>CL<sub>2</sub>, FE<sub>2</sub>O<sub>3</sub>, FE<sub>0.5</sub>O<sub>3</sub>, FE<sub>1.5</sub>O<sub>1</sub>, CA<sub>0.5</sub>MG<sub>0.5</sub>SI<sub>1.0</sub>O<sub>3</sub>,</p> <p>AL<sub>1</sub>H<sub>1</sub>O<sub>1</sub>_AL(OH) , AL<sub>1</sub>H<sub>1</sub>O<sub>1</sub>_HALO, C<sub>2</sub>H<sub>4</sub>O<sub>3</sub>_124TRIOXOLANE, NA<sup>+</sup>, SiO<sub>4</sub><sup>-4</sup>, H<sub>1</sub><sup>+1</sup>, H<sub>1</sub>O<sub>1</sub><sup>-1</sup>, AGC<sub>2</sub>N<sub>2</sub>-1.</p> <p>The species name can be used to separate isomers with the same stoichiometry, e.g. C<sub>2</sub>H<sub>2</sub>Cl<sub>2</sub>, (CHCl)<sub>2</sub>_cis, (CHCl)<sub>2</sub>_trans, CH<sub>2</sub>_CCl<sub>2</sub>.</p>
	<p>Stoichiometry /Species name/: &lt;Chemical formula&gt;</p> <p>The elements in the chemical formula are normally separated by stoichiometric numbers. Neither parenthesis () nor an underscore _ is allowed in the chemical formula, while the special combination /- or /+ can be used.</p> <p>In upper case mode (see <a href="#">REINITIATE</a>), the stoichiometric number unity can be excluded only if the element symbol consists of two letters. In lower case mode, the element must be typed with the first letter in UPPER case and the second in lower, and it is thus possible to distinguish CO (carbon monoxide) from Co (cobalt) without any stoichiometry number.</p> <div style="border: 1px solid red; padding: 5px; margin-top: 10px;">  All elements in a species must be entered prior to entering the species.         </div>

## ENTER\_SYMBOL



This command is for the Gibbs Energy System (GES) module.



There are also a POLY and POST module commands with the same name. However, symbols entered in the GIBBS module are not the same symbols as defined in the POLY and POST modules.

In the GIBBS module, a symbol can be used to represent a numeric quantity (e.g. a *variable*), a *function*, or a *table*. The entered symbol (with a symbol name of maximum eight characters) can be used later when parameters are entered, and this is a flexible way to manipulate thermodynamic functions. Symbols are especially useful if the same function or table is used in several thermochemical parameters. A *parameter* symbol is used to assign the values of phase parameters (which are defined with the command ENTER\_PARAMETER) to specific characterized symbols. If you want to define a constant quantity, enter it as a simple function with a constant value in all (temperature) ranges.

The symbols entered in the GIBBS module can also be listed and used in the PARROT module (for data optimization) where you can use the GIBBS-entered symbols (variables, functions or parameters) to define parameters that shall be optimized.



See [example 44](#) for an example of using variables and functions.

Syntax	ENTER_SYMBOL
Prompt	<p>Variable, function, table or parameter? /Function/: &lt;Keyword&gt;</p> <p>The keyword can be a variable, function, table or parameter.</p> <ul style="list-style-type: none"> <li>• <b>Variables</b> are similar to functions because these can also be expressions of state variables. It is possible to enter a variable with a new expression anytime.</li> <li>• <b>Functions</b> are expressions of state variables or other functions.</li> <li>• <b>Tables</b> are used for listing results. A table consists of a list of any number of state variables, functions or variables.</li> <li>• <b>Parameter</b> symbols are used to assign the values of phase parameters (which are defined by the command ENTER_PARAMETER) to specific characterized symbols. If the phase parameter is not defined yet, the parameter symbol is assigned zero or a symbol which is valued as zero. In this way it is easy to refer to entered phase parameters in further defining other phase parameters.</li> </ul> <p>Name: &lt;Name of the symbol&gt;</p> <p>Each symbol has a unique name that must start with a letter and can have maximum 8 characters.</p>

Syntax	ENTER_SYMBOL
	The following prompts are based on the type of symbol.
Variable symbol	<p>Value: &lt;Numeric value of a variable&gt;</p> <p>Only a constant numeric value, not an expression, is accepted.</p>
Function symbol	<p>Enter the lower-temperature limit, the expression, an upper-temperature limit and if there is another expression above this upper limit:</p> <p>Low temperature limit /298.15/: &lt;Lowest temperature limit in K&gt;</p> <p>Specify the lowest-temperature limit, below which its (first) expression is not applicable. The default lowest limit of all kinds of data is 298.15 K.</p> <p>If a negative number is given as the lowest-temperature limit, it is assumed that there are breakpoints in pressure for this function; in such a case, it is interpreted as the lowest-pressure limit (in Pascal), and the other limits in the current function are also taken as pressure limit values (in Pascal).</p> <p>The temperature/pressure limits for the functions are checked during calculations. An indicator is set if the actual temperature/pressure condition is below the lowest temperature/pressure limit or above the highest temperature/pressure limit; and in such a case, an extrapolation is conducted using the TP-FUNCTIONS valid in the nearest temperature/pressure range.</p> <p>Function: &lt;Definition for a TP-function&gt;</p> <p>A TP-Function consists of terms in T and P. The expression is a FORTRAN-like expression and operators +, -, *, = and ** can be used (** only with integer powers). Unary-functions LN or LOG (both for natural logarithm) and EXP (for exponential) can also be used. An expression can be continued on more than one line. A PT-function must be terminated by a semicolon (;) or an empty line. No more than 78 characters can be written on a line. If this is not sufficient space on one line, press &lt;Enter&gt; and continue on the next line. The program prompts with an ampersand &amp; when a continuation is expected.</p> <div data-bbox="386 1157 1393 1325" style="border: 2px solid blue; padding: 10px; margin: 10px 0;"> <p> A function can have several temperature ranges, and a different expression (i.e. a TP-Function) for each region; if required, a single function can have up to ten different temperature ranges (and thus have up to ten different TP-Functions).</p> </div> <p>This question is repeated if the current function has more than one expression (TPFunction) in various temperature ranges [or pressure ranges; only if the <i>lowest limit</i> is entered as a negative number)].</p> <p>&amp; &lt;Continuation of the definition for the current TP-Function&gt;</p> <p>This is for a continuation of the definition for the current TP-Function. The ampersand &amp; means that you can continue to write the function on the new line if one line is not enough for the current TP-Function. If you are finished the current TP-Function, press &lt;Enter&gt; again.</p> <p>High temperature limit /6000/: &lt;High temperature limit in K&gt;</p> <p>For a TP-Function, specify the high-temperature limit, above which its current expression is not applicable. The default high-temperature limit for all kinds of data is always 6000 K.</p> <p>This question is repeated if the function has more than one expression (TPFunction) in various temperature ranges.</p>

Syntax	ENTER_SYMBOL
	<div data-bbox="402 302 444 359" style="border: 1px solid blue; padding: 5px; margin-bottom: 10px;">  All the temperature limits for the presently-be-entered TP-Function MUST be in an increasing order; otherwise the whole function is given a single value of zero. </div> <p>If a negative number is given as the lowest-temperature limit, it is interpreted as the low-pressure limit (in Pascal), and the high-temperature limit is also taken as the high-pressure limit (in Pascal) for the current TP-Function.</p> <p>The high-temperature/pressure limit for the current applicable expression (TPFunction) is the low temperature/pressure limit for the next range if several ranges are needed to describe the temperature/pressure-dependence of this function.</p> <p>The temperature/pressure limits for the functions are checked during calculations. An indicator is set if the actual temperature/pressure condition is below the lowest temperature/pressure limit or above the highest temperature/pressure limit; and in such a case, an extrapolation is conducted using the TP-Function valid in the nearest temperature/pressure range.</p> <p>Any more ranges /N/: &lt;Y or N&gt;</p> <p>Answer N to finish the definition of the current function. If Y the program asks for a new function valid above the last high temperature limit and below a new high temperature limit. The maximum number of temperature ranges is 10.</p> <p>Important note: Make sure that the function and its first derivatives are continuous across break points.</p>
Table symbol	<p>Specify the low and high temperature limits, temperature step and tabled value at each specified temperature:</p> <p>Low temperature limit /298.15/: &lt;Low temperature limit in K&gt;</p> <p>The lowest temperature where the table has a value.</p> <p>High temperature limit /6000/: &lt;High temperature limit in K&gt;</p> <p>The highest temperature where the table has a value.</p> <p>Step in temperature /100/: &lt;Temperature step&gt;</p> <p>The step in temperature between each value in the table. This step must be constant for the whole table. If a table cannot be described with a single step it must be divided into several tables for different temperature ranges. The upper and lower limits are truncated to give reasonable values where the table values must be specified.</p> <p>Table value at XX /YY/: &lt;Table value&gt;</p> <p>The value of the table at the specified temperature (xx) must be given. The default value (yy) is the last value.</p>
Parameter symbol	<p>Input the correct name of a phase parameter (including its identifier, phase name, constituent and interacting constituent(s) in various sublattices (if exist), and degree of the parameter. Most of the settings are the same as in the <a href="#">ENTER_PARAMETER</a> command or the Function symbol prompts above.</p> <p>Parameter: &lt;Parameter name&gt;</p> <p>Specify a correct and complete parameter name, which should contain all the necessary parts of the general form:</p>

Syntax	ENTER_SYMBOL
	<p><code>&lt;Identifier&gt;&lt;Phase&gt;,&lt;Constituent array&gt;;&lt;Digit&gt;</code></p> <p>If a parameter name is not acceptable or &lt;Enter&gt; is pressed, this error message displays:</p> <pre>*** Error, please re-enter each part separately</pre> <p>The program prompts for separate input for each required part for a parameter name.</p> <p>Identifier (/X/): &lt;G or L, or TC, or BM, or V0 or VA or VC or VK&gt;</p> <p>Specify one of these types of legal identifiers.</p> <p>Phase name (/abcd/): &lt;Phase name&gt;</p> <p>Specify the phase name.</p> <p>Constituent (in sublattice # /abc/): &lt;Species name&gt;</p> <p>Specify the constituent name on the specified sublattice site of the given phase.</p> <p>Interacting constituent (in sublattice # /XYZ/): &lt;Species name&gt;</p> <p>Specify the interacting constituent name on the specified sublattice site of the given phase; if there is no interacting constituent, press &lt;Enter&gt;.</p> <p>Interacting constituent (in sublattice # /XYZ/): &lt;Species name&gt;</p> <p>If there is more than one interacting constituent on the specified sublattice site of the given phase, specify them; otherwise press &lt;Enter&gt;.</p> <p>Degree /#/: &lt;Degree&gt;</p> <p>Specify an integer number (a value from 0 through 9) as the degree for the phase parameter.</p> <p>Low temperature limit /298.15/: &lt;Lowest temperature limit in K&gt;</p> <p>Specify the lowest-temperature limit (in Kelvin), or the lowest-pressure limit (in Pascal but entered as a negative number), for the current TP-Function.</p> <p>Function: &lt;Definition for a function&gt;</p> <p>Specify the desired TP-Function for the current temperature range (or current pressure range; only if the <i>lowest limit</i> is entered as a negative number).</p> <p>&amp; &lt;Continuation of the definition for the current TP-function&gt;</p> <p>Specify the desired TP-Function continuation for the current temperature range (or current pressure range; only if the lowest limit is entered as a negative number).</p> <p>High temperature limit /6000/: &lt;High temperature limit in K&gt;</p> <p>Specify the high-temperature limit (in Kelvin), or the high-pressure limit (in Pascal; only if the lowest limit is entered as a negative number) for the current TP Function.</p> <p>Any more ranges /N/:</p>

## LIST\_CONSTITUTION



This command is for the Gibbs Energy System (GES) module.

Syntax	LIST_CONSTITUTION
	List all the constitutions (as site-fractions) for all the phases in the defined system.
	<div data-bbox="362 531 407 590"></div> <p>This is mainly for software managers. There are no commands in the GIBBS monitor to set the constitution.</p>

## LIST\_DATA

 This command is for the Gibbs Energy System (GES) module.

Use the LIST\_DATA command to retrieve and then list the data included with a database in the DATA module or directly entered in the GIBBS module. All data for the current system that is defined within the current GIBBS workspace is written in a readable manner in the output file. If no output file is specified, by default the data displays on screen. The behavior of this command differs slightly based on whether you are using GES6 (the default) or GES5.



For encrypted commercial databases you may not be able to use this command to list any retrieved data. However, it can be used to list some basic definitions (such as phase models and phase-constituents) retrieved from an encrypted database, as well as all the references related to original assessments (but not thermodynamic parameters) for the currently-defined system.

### Command Prompts

After entering the command in the console, the following prompt options are available.

Syntax	LIST_DATA
Prompts	<p>Output to screen or file /Screen/: &lt;File name&gt;</p> <p>Specify the name of a file in which the data shall be written in a readable manner. The default value is the screen (by pressing &lt;Enter&gt;).</p> <p>Options?: &lt;Option(s)&gt;</p> <p>Choose one or several of the following options for output.</p> <div style="border: 1px solid red; padding: 5px; margin: 10px 0;"> <p> See the section about the differences between GES5 and GES6 output for these options.</p> </div> <ul style="list-style-type: none"> <li><b>N</b> the output is written as a user database format. For GES6, and only for an unencrypted database, this option has a subcommand that allows filtering of phase parameters with respect to the property type e.g., to include only volume parameters and/or viscosity parameters in the output.             <ul style="list-style-type: none"> <li>The subprompt is <code>Property type /All/</code>. Choose <code>All</code> or enter the property type. The valid alternatives (when available) are: V0, VA, VB, VC, VK, TC, TN, BM, MQ, MF, DQ, DF, OQ, VS, G, L, THCD, ELRS, ESPD, VISC, GD, SIGM or XI</li> </ul> </li> <li><b>P</b> the output is written as a macro file for future input. This is useful for creating <code>*SETUP</code> files for data assessments.</li> </ul>

Syntax	LIST_DATA
	<ul style="list-style-type: none"> <li>• <code>S</code> the symbols are suppressed.</li> <li>• <code>R</code> the references for the parameters are listed (only for some databases in which references are available)</li> <li>• <code>L</code> the output is written suitable for a LaTeX preprocessor.</li> </ul>

### Output Differences: GES5 vs GES6



GES6 is a re-write and new implementation of the Gibbs Energy System module of Thermo-Calc and it is the default engine. The previous version, GES5, is still available and will continue to work in the background of the program where necessary. The main purpose of GES6 is to support faster development of new features.



As required you can set Thermo-Calc to use GES5 instead of the default with the [SET\\_GES\\_VERSION](#) command. The application may also automatically and temporarily switch to GES5 when a database is being used that is not compatible with GES6.

For both GES5 and GES6 and when you enter `P`, `S`, `R`, or `L`, the output data consists of a list of all elements and the data followed by a list of all phases and the data associated with each phase. The thermochemical parameters listed for each phase are always in SI units.

For GES6, and only when option `N`, `P`, or nothing is entered, the output data consists of a list of all elements and the data followed by a list of all phase definitions and the data associated with the phases, sorted by system. In these modes, and only for unencrypted databases, there is a subcommand that allows filtering of phase parameters with respect to the property type e.g., to include only volume parameters and/or viscosity parameters in the output.

Other differences between the GES5 and GES6 output are related to the output as shown below.

Output	GES5	GES6
Parameters	Includes implicitly defined zero-valued thermodynamic parameters	Only contains parameters that are actually defined in the database
Automatically	Does not include	Includes

---

<i>Output</i>	<i>GES5</i>	<i>GES6</i>
suspended phases		
References	Contains references of functions and parameters with generic names, such as "REF123"	Retains the original reference names that are used in the database

## LIST\_PARAMETER

-  This command is for the Gibbs Energy System (GES) module.
-  There is also a PARROT module command with the same name.

List the TP-function(s) of a specific parameter for a phase. You must supply the name of the phase parameter:

The parameter name: <Identifier>(<Phase>,<Constituent array>;<Digit>)



For encrypted commercial databases, you may not be able to use this command to list any retrieved parameter.

Syntax	LIST_PARAMETER
	<p>Parameter: &lt;Parameter name&gt;</p> <p>Specify a correct and complete parameter name, which should contain all the necessary parts of the general form: &lt;identifier&gt;(&lt;phase&gt;,&lt;constituent array&gt;;&lt;digit&gt;)</p>
Prompts	<p> See <a href="#">About the ENTER_PARAMETER Command</a> to learn how to define the parameter name.</p> <p>If a parameter name is not acceptable or &lt;Enter&gt; is pressed, an error message displays:</p> <pre>*** Error, please re-enter each part separately</pre> <p>The program prompts for separate input for each required part for a parameter name.</p>
	<p>Identifier (/X/): &lt;G or L, or TC, or BM, or V0 or VA or VC or VK&gt;</p> <p>Specify one of these types of legal identifiers.</p>
	<p>Phase name (/ABCD/): &lt;Phase name&gt;</p> <p>Specify the phase name.</p>
	<p>Constituent (in sublattice # /ABC/): &lt;Species name&gt;</p> <p>Specify the constituent name on the specified sublattice site of the given phase.</p>
	<p>Interacting constituent (in sublattice # /XYZ/): &lt;Species name&gt;</p> <p>Specify the interacting constituent name on the specified sublattice site of the given phase; if there is no interacting constituent, press &lt;Enter&gt;.</p>
	<p>Interacting constituent (in sublattice # /XYZ/): &lt;Species name&gt;</p> <p>If there is more than one interacting constituent on the specified sublattice site of the given phase, specify them; otherwise press &lt;Enter&gt;.</p>
	<p>Degree /#/ : &lt;Degree&gt;</p>

---

Syntax	LIST_PARAMETER
	Specify a numerical number as the degree for the phase parameter.

## LIST\_PHASE\_DATA

 This command is for the Gibbs Energy System (GES) module.

All data for a specific phase are written in a readable manner on screen. The thermochemical parameters listed for the phase are always in SI units.



For encrypted commercial databases you may not be able to use this command to list any phase data. However, this command can be used for listing some basic definitions (such as phase model and phase-constituents) for the specified phase, as well as all the references related to original assessments (but not thermodynamic parameters) for the currently-defined system.

Syntax	LIST_PHASE_DATA
Prompt	Phase name: <Phase name> Specify a phase name (if abbreviated, it should be unique).

### Output Differences: GES5 vs GES6



GES6 is a re-write and new implementation of the Gibbs Energy System module of Thermo-Calc and it is the default engine. The previous version, GES5, is still available and will continue to work in the background of the program where necessary. The main purpose of GES6 is to support faster development of new features.



As required you can set Thermo-Calc to use GES5 instead of the default with the [SET\\_GES\\_VERSION](#) command. The application may also automatically and temporarily switch to GES5 when a database is being used that is not compatible with GES6.

For GES6 the output data consists of the phase definition and the data associated with the phase, sorted by system. In GES6, there is a subcommand that allows filtering of phase parameters with respect to the property type e.g., to include only volume parameters and/or viscosity parameters in the output.

## LIST\_STATUS



This command is for the Gibbs Energy System (GES) module.

Syntax	LIST_STATUS
	<p>List the entered elements, phases and species with the status word. The command is included for system managers.</p> <div style="border: 1px solid blue; padding: 5px;"><p> Each element, species, phase and symbol has a set of status bits. The values of these bits are listed with this command.</p></div> <p>The bits are listed as hexadecimal, i.e. four bits are written as a hexadecimal number. Two hexadecimal numbers make a byte. In hexadecimal, 0 to 9 mean normal digits. A to F means values 10 to 15. The number <code>E4000000</code> has thus bit 1, 2, 3 and 6 equal to one. The bits are numbered starting with 1 for the most significant (leftmost) bit.</p>

### Element Status Word

Set the element status word (*Bit. Meaning*) if:

1. Element cannot be deleted (only vacancy and electron)
2. Suspended (inclusive OR of bit 3 and 4)
3. Explicitly suspended
4. Implicitly suspended (cannot occur for an element)

### Species Status Word

Set species status word (*Bit. Meaning*) if:

1. Species record for an element (each element has a species record)
2. Suspended (inclusive OR of bit 3 and 4)
3. Explicitly suspended
4. Implicitly suspended (e.g. if a species element is suspended)
5. Charged (inclusive OR of bit 6 and 7)
6. Negative charge

7. Positive charge
8. Vacancy
9. Component (by default the elements are the components)

### ***Phase Status Word***

Set phase status word (*Bit. Meaning*) if:

1. Ideal (no sublattices and no excess parameters)
2. Suspended (inclusive OR of bit 3 and 4)
3. Explicitly suspended
4. Implicitly suspended (e.g. if all constituents suspended)
5. Gas phase
6. Liquid phase
7. Solution phase
8. Only one constituent (in each sublattice)
9. Ions (inclusive OR of bit 10 and 12)
10. New ionic liquid model (charge balance handled internally)
11. Constituents cannot be added
12. External charge balance needed
13. Aqueous model
14. Charged species (ions)
15. Dilute entropy
16. Last calculation exceeded temperature range for any parameter
17. Kapoor-Frohberg-Gaye cell model
18. Turbo calculation used
19. Turbo calculation impossible
20. Turbo calculation illegal
21. Phase is not ideal
22. Current site fractions saved
23. This phase is the ordered part
24. This phase is the disordered part (bit 2 and 3 also set)

25. Shadow phase with diffusion data
26. Error in ionic model
27. Sometimes
28. CVM-SRO (short-range ordering) entropy expression
29. CVM initialization is made
30. Used to test quasi-chemical ordering model
31. Major constituents check even if no miscibility gap (ordering)
32. Hoch-Arpshofen model

### ***Symbol Status Word***

Set symbol status word (*Bit meaning*) if:

1. Constant
2. Variable
3. Function
4. Table
5. Value must not be amended
6. Undefined

## LIST\_SYMBOL



This command is for the Gibbs Energy System (GES) module.

List the symbols available in the current GIBBS workspace, which are defined (retrieved from database in the DATA module, or directly entered in the GIBBS module) and used in TP-functions for the entered model parameters for various phases in the defined system.



For encrypted commercial databases you may not be able to use this command to list any retrieved symbol.

Syntax	LIST_SYMBOL
	<p>Name: &lt;Symbol name&gt;</p> <p>Specify a symbol name (either as the full name of a defined symbol, or as the first few common characters of several symbols which have names starting with such common characters).</p>
Prompts	<div style="border: 1px solid blue; padding: 5px; margin-bottom: 10px;">  Only those symbols that match this name are listed.         </div> <p>Or press &lt;Enter&gt; for a list of all the symbols defined (retrieved from database in the DATA module, or directly entered in the GIBBS module) for the current system, which are available in the current GIBBS workspace.</p>
	<p>Output to screen or file /Screen/: &lt;File name&gt;</p> <p>Specify the name of a simple-textual file in which the entered symbols shall be written in a readable manner (such a basic textual file can later on be opened and edited by any text editor). The default value is the screen (by pressing &lt;Enter&gt;).</p>

## READ\_GES\_WORKSPACE



This command is for the Gibbs Energy System (GES) module.



[Workspace Files](#)

The data saved into a \*.GES5 with SAVE\_GES\_WORKSPACE can be read back into the GIBBS workspace.

Syntax	READ_GES_WORKSPACE
	<p>Enter a <b>File name</b> and specify the working directory where to save the file in the <b>Look in</b> field. Normally the default file-type in the <b>Files of type</b> field is the correct one for the GES workspace format.</p> <div data-bbox="342 726 1396 869" style="border: 1px solid blue; padding: 10px;"><p> The saved *.GES5 files are unique for each CPU type, and therefore a file saved on one CPU type cannot be read and used on another CPU type.</p></div>

## READ\_WORKSPACES

 This command is used in both the POLY and GES (GIBBS) modules.

 [Workspace Files](#)

The POLY3 and GIBBS workspaces and the calculated results from the MAP and STEP commands can be READ from a file where they are saved with SAVE\_WORKSPACES. Such a \*.POLY3 file is not printable.

Syntax	READ_WORKSPACES
Options	<p><code>File name</code> is the name of a saved POLY3-file where the POLY3 and GIBBS workspaces shall be read from must be specified. You do not need to type the extension if it is the default *.POLY3, otherwise type the whole POLY-file name.</p> <p> A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.</p> <p>A window opens so that the path (in the <b>Look in</b> field) and <b>File name</b> can be specified. The <b>Files of type</b> (i.e. POLY3) cannot be changed. Click <b>Open</b> to open the POLY3 and GIBBS workspaces from the saved *.POLY3 file.</p>
	<p>When reading back an original POLY3 workspace that has already been saved as an *.POLY3 file in the current Thermo-Calc (Console Mode) run or had been read from an existing POLY3 file under the current work area, while some additional changes in the settings may be made but do not need to be kept in further steps in the current Thermo-Calc (Console Mode) run or any diagram is plotted in the POST module, you can type <code>READ,,</code> or <code>READ_WORKSPACE ,,</code></p>

## REINITIATE



This command is for the Gibbs Energy System (GES) module.

All data in the stored GIBBS workspace are erased, and all variables are initiated to their default values.

Syntax	REINITIATE
Prompts	<p>Upper case only /Y/:</p> <p>Press &lt;Enter&gt; to keep the default where the name of elements and species is in only UPPER case. Or choose N to have the elements with two letter names with the first letter in Upper case and the second in lower case.</p> <div style="border: 2px solid blue; padding: 5px; margin: 10px 0;">  In upper case mode all input in lower case is automatically converted to upper case.         </div>
	<p>Lower temperature limit /298.15/: &lt;Lowest T in K&gt;</p> <p>This value is used as the lower temperature limit when data are entered interactively.</p>
	<p>Upper temperature limit /6000/: &lt;Highest T in K&gt;</p> <p>This value is used as the upper temperature limit when data are entered interactively.</p>
	<p>Default element reference state symbol index /1/: &lt;1 or 2 or 3&gt;</p> <p>Specify a proper index for the default listing parameters (symbol), or press &lt;Enter&gt; to accept the preset index 1.</p> <p>The <i>index</i> is for defining the symbol printed in listings of parameters. The symbol can be:</p> <ul style="list-style-type: none"> <li>• <math>G</math>: the data are referred to Gibbs energy at a variable temperature (also called <i>Lattice Stability</i>).</li> <li>• <math>H_{298}</math>: the data are referred to the enthalpy of the element at 298.15 K and 1 bar.</li> <li>• <math>H_0</math> is the same as <math>H_{298}</math> but at the temperature 0 K.</li> </ul> <div style="border: 2px solid blue; padding: 5px; margin: 10px 0;">  The index only changes the symbol, not any value. Normally the index is set correctly by the database. Only when manually entering data must you set the index to get the correct symbol. Set the index value to 0 to get the symbol <math>G</math>, to 1 to get the symbol <math>H_{298}</math> and to 2 to get <math>H_0</math>.         </div>

## SAVE\_GES\_WORKSPACE



This command is for the Gibbs Energy System (GES) module.



[Workspace Files](#)

The data used by the GES workspace can be saved in a file with a default extension `*.GES5`. The default file name is `RESULT.GES5` or equal to the one used in a previous `READ_GES_WORKSPACE` command.

Syntax	SAVE_GES_WORKSPACE
	<p>Enter a <b>File name</b> and specify the working directory where to save the file in the <b>Save in</b> field. Normally the default file-type in the <b>Files of type</b> field is the correct one for the GIBBS workspace format.</p> <div data-bbox="342 764 1396 877" style="border: 2px solid blue; padding: 5px;"> The output is not formatted and cannot be printed.</div>

## SAVE\_WORKSPACES



This command is available for both the POLY and GES (GIBBS) modules.



### [Workspace Files](#)

You can save the current status and workspaces, including thermodynamic data, conditions, options and results from a single, stepping or mapping calculation on a \*.POLY3 file. The workspaces are saved to a file with this command. This is useful so you have access to the workspace for later use or when you need to terminate the current Console Mode session.

- In the Gibbs (GES) module workspace, all thermochemical data are stored.
- In the POLY module workspace, all the last set of conditions and equilibrium state, changed status, entered symbols, advanced options, defined stepping/mapping variables, added initial equilibria, stepped/mapped results, etc., are stored, so it also contain the GES workspace.
- When you are in a response-driven module such as POTENTIAL or SCHEIL for example, a workspace file is automatically opened. In the workspace file, system definitions, conditions for the calculation, calculation results, and plot settings are saved. The file is saved in the current working directory, and is named after the name of the module that created it. For example, the POTENTIAL module saves a workspace file called `POT.POLY3` ; the POURBAIX module saves a file called `POURBAIX.POLY3` and so forth.



After a SAVE\_WORKSPACES command, you can always come back to exactly the state you had when you issued the command by giving a READ\_WORKSPACE command. For example, after saving the POLY and GIBBS workspaces to a file, you can leave the program and at a later time READ the file and continue from the saved state.

### *Important Information About Map and Step Commands*



A STEP or MAP command automatically saves the workspace with the most recently specified name. Do not use the SAVE\_WORKSPACES command after a MAP or STEP command.

The results from the STEP or MAP commands are destroyed by the SAVE\_WORKSPACE command. You can append several results obtained by sequential STEP or MAP calculations without destroying the previous results, whilst SAVE\_WORKSPACE erases them all. Keeping this in mind is important and useful particularly for calculating various isothermal (or isoplethal) sections and plotting them on the same diagram in a single Thermo-Calc run.

To suspend some of the STEP or MAP results, use the AMEND\_STORED\_EQUILIBRIA command.

Syntax	SAVE_WORKSPACES
Options	<p>A <code>file name</code> must be specified. The default extension of the POLY workspace file is *.POLY3 but you can have any other file extension.</p> <p> Linux and Mac are case sensitive so ensure you enter <code>.POLY3</code>.</p> <p>A <b>Save</b> window displays if a file name is not given after the command, so that the path (in the <b>Save in</b> field) and <b>File name</b> can be specified. The <b>Files of type</b> cannot be changed.</p> <p>If there is already a file with the same file name under the directory a warning message displays. Click <b>OK</b> to overwrite the current POLY or GIBBS workspace onto the existing file. Click <b>Cancel</b> to return to the module. You can use the SAVE command with an unspecified name (i.e. through <code>SAVE , , ,</code> command-sequence) to save the POLY or GIBBS workspace.</p> <div style="border: 2px solid blue; padding: 10px; margin-top: 10px;"> <p> When saving a POLY3 workspace under a name that already exists under the current work area, which is saved by default (after running a special module, e.g. BIN, TERN, POT, SCHEIL and POURBAIX) or in an earlier stage of the current Console Mode session or in a previous run (which is READ into the current POLY3 workspace), you can type <code>SAVE , , y</code> or <code>SAVE_WORKSPACE , , y</code>. However, this must also be avoided if some results from previous MAP or STEP calculations shall not be destroyed.</p> </div>

---

## SCHEIL Module Commands

---

### In this section:

---

CALCULATE_BELOW_SOLIDUS .....	339
CALCULATE_EVAPORATION_PROPERTIES .....	339
CALCULATE_FROM_START_TEMPERATURE .....	340
CALCULATE_FROM_GAS .....	340
DELTA_FERRITE_AUSTENITE_TRANSITION .....	341
EVALUATE_SEGREGATION_PROFILE .....	342
GLOBAL_MINIMIZATION .....	343
LIQUID_PHASE_NAME .....	344
SAVE_FILE_NAME .....	345
SET_NUMERICAL_LIMITS .....	346
SKIP_STEP_CALCULATION .....	347
SOLUTE_TRAPPING .....	347
START_WIZARD .....	348
TEMPERATURE_STEP .....	352
TERMINATION_CRITERIA .....	353
TEST_INTERVAL_FOR_GLOBAL .....	353
USE_BACK_DIFFUSION .....	354

## CALCULATE\_BELOW\_SOLIDUS

 This command is for the SCHEIL module.

The CALCULATE\_BELOW\_SOLIDUS command enables or disables the calculation of solid state properties at temperatures below the solidus temperature, using the phase compositions and fractions from the Scheil calculation. By default this is disabled.

The calculation is performed at a set number of temperatures between the solidus temperature and a lower temperature, which is by default set to room temperature. The end temperature and the number of points can be changed.

Syntax	CALCULATE_BELOW_SOLIDUS
Prompts	<p>Calculate properties below solidus? /N/:</p> <p>The default is N. Enter Y to calculate solid state properties below solidus .</p> <p>If Y is entered, then you are prompted:</p> <p>Final temperature (C) /25/:?</p> <p>Enter the final temperature in degrees Celsius. Must be 25 C or higher.</p> <p>Number of points /50/:?</p> <p>Enter the number of points where solid state properties are calculated.</p>

## CALCULATE\_EVAPORATION\_PROPERTIES

 This command is for the SCHEIL module.

The CALCULATE\_EVAPORATION\_PROPERTIES command enables or disables the calculation of evaporation properties. By default this is disabled. The evaporation properties include driving force for evaporation, evaporation enthalpy, and molar mass of gas.



To be able to calculate these properties the gas phase needs to be selected (in the [START\\_WIZARD](#)).

Syntax	CALCULATE_EVAPORATION_PROPERTIES
Prompts	<p>Calculate evaporation properties? /N/:</p> <p>The default is N. Enter Y to calculate evaporation properties.</p>

## CALCULATE\_FROM\_START\_TEMPERATURE

 This command is for the SCHEIL module.

The `CALCULATE_FROM_START_TEMPERATURE` command enables or disables the calculation of equilibria directly from the start temperature chosen in the `START_WIZARD`. By default this is disabled (set to `N`). Enable this (set to `Y`) if you want to obtain properties of the liquid phase (and gas phase if selected) before the solidification starts. Ordinarily the start temperature is only used as a start point in the search of the liquidus temperature, as the Scheil calculation starts from the liquidus temperature.

With this setting enabled single equilibrium calculations are performed at specific intervals above the liquidus temperature, and after that the Scheil solidification calculation proceeds as usual. The equilibrium calculations are performed at 50 Kelvin (K) intervals in the fully liquid region and at 5 K intervals in a possible two-phase (liquid/gas) region.

Syntax	<code>CALCULATE_FROM_START_TEMPERATURE</code>
Prompt	<pre>Perform equilibrium calculations between start and liquidus temperatures? /N/:</pre> <p>The default is <code>N</code>. Enter <code>Y</code> to perform equilibrium calculations from the start temperature until liquidus temperature is reached (i.e. solidification starts). The calculations are performed at 5 Kelvin intervals in the two-phase gas/liquid region and at 50 Kelvin intervals in liquid.</p>

## CALCULATE\_FROM\_GAS

 This command is for the SCHEIL module.

The `CALCULATE_FROM_GAS` command enables or disables the calculation of equilibria starting from the gas/liquid temperature, i.e. from the temperature above which the liquid phase is no longer stable. By default this is disabled (set to `N`). Enable this (set to `Y`) to perform single equilibrium calculations at 5 Kelvin (K) intervals in the two-phase region between the gas/liquid temperature and the liquid/gas temperature, which is the temperature below which the gas phase is no longer stable.

In the fully liquid region below the liquid/gas temperature single equilibrium calculations are performed at 50 K intervals until the liquidus temperature is reached. After that the Scheil solidification calculation proceeds as usual.

Syntax	<code>CALCULATE_FROM_GAS</code>
Prompts	<pre>Calculate from gas/liquid temperature? /N/:</pre> <p>The default is <code>N</code>. Enter <code>Y</code> to perform equilibrium calculations from the gas/liquid temperature.</p> <pre>Gas phase name /GAS/:</pre> <p>Enter the name of the gas phase (default <code>GAS</code>).</p>

Syntax	CALCULATE_FROM_GAS
	 <p>If you choose a phase that is not available in the system, Thermo-Calc checks if there is any phase present with the phase type code G.</p>

## DELTA\_FERRITE\_AUSTENITE\_TRANSITION

 This command is for the SCHEIL module.

The DELTA\_FERRITE\_AUSTENITE\_TRANSITION command enables or disables BCC to FCC transition.



This feature should only be used for such steels for which it is reasonable to assume infinitely fast diffusion (equilibrium conditions) when delta ferrite is present. When the **Allow delta ferrite to austenite transition in steel** checkbox is selected in Graphical Mode or the DELTA\_FERRITE\_AUSTENITE\_TRANSITION command used in Console Mode, this essentially assumes that diffusion can be considered (infinitely) fast when delta ferrite is present as a solid phase, i.e. assumes an equilibrium condition. After delta ferrite is no longer stable, the usual Scheil assumptions for the selected calculation type are in effect. This setting is therefore not suitable for steels where you DO NOT have a complete BCC to FCC transition during solidification. This setting is also NOT available in combination with solute trapping.

Syntax	DELTA_FERRITE_AUSTENITE_TRANSITION
Prompt	<p>Allow delta-ferrite -&gt; austenite ? /N/:</p> <p>The default is N. Enter Y to enable BCC to FCC transition.</p> <p>This feature should only be used for such steels for which it is reasonable to assume infinitely fast diffusion (equilibrium conditions) when delta ferrite is present. See the above for details.</p>

## EVALUATE\_SEGREGATION\_PROFILE

 This command is for the SCHEIL module.

 This feature is not supported with fast diffusers.

Use this command in order to evaluate a segregation profile while performing the Scheil calculation. The Scheil calculation itself does not contain a spatial dimension. However, if it is assumed that the molar volume is constant and equal for all elements, and if a length is provided over which solidification occurs (for example the secondary dendrite arm spacing) then the generated profile corresponds to the segregation that would be obtained under the assumptions of a Scheil calculation. The grid is then equidistant and can be used as input for a DICTRA simulation.

The generated text file contains, on each line, the mole fractions of all elements in alphabetical order at a certain grid point. If the composition is the same on several consecutive lines it means that the composition of the then remaining liquid has reached a state in which there are no degrees of freedom.

Syntax	EVALUATE_SEGREGATION_PROFILE
Prompt	<p>Evaluate segregation profile? /N/</p> <div style="border: 2px solid blue; padding: 10px; margin: 10px 0;"> <p> If the answer is Y, a segregation profile will be evaluated and saved to disc. This feature is not supported with fast diffusers. You cannot select fast diffusers if evaluation of the segregation profile is enabled.</p> </div>
	<p>Number of grid points /100/</p> <p>Enter the number of grid points for the segregation profile. A file dialogue window opens and allows you to specify the name of the file in which the segregation profile is written.</p> <p>In a macro file it is possible to enter all arguments on a single line, e.g. <code>EVALUATE_SEGREGATION_PROFILE Y 100 XF.TXT</code>, meaning that the segregation profile is evaluated over 100 grid points and saved to a file (XF.TXT) in the current working directory.</p>

## GLOBAL\_MINIMIZATION

 This command is for the SCHEIL module.

Global minimization is enabled by default (i.e. set to `y`) and global minimization is used throughout the simulation.



It is recommended to keep global minimization on as it makes the calculation more accurate. However, turn it off if you find that the calculation is taking too long for any particular system, and accuracy is not the highest priority.

For the GLOBAL\_MINIMIZATION command, a global equilibrium test is performed when an equilibrium is reached, at selected intervals. By default it is at every 10th Scheil step but this can be changed using the command [TEST\\_INTERVAL\\_FOR\\_GLOBAL](#). The global equilibrium test is also done every time the set of stable phases tries to change. This costs more computer time but the calculations are more robust.

Syntax	GLOBAL_MINIMIZATION
Prompt	Use global minimization? /Y/ By default this is set to <code>y</code> . This costs more computer time but the calculations are more robust.

## LIQUID\_PHASE\_NAME

 This command is for the SCHEIL module.

Use this command to use the liquid phase name from the database definitions or enter another phase name.

Syntax	LIQUID_PHASE_NAME
Prompt	Liquid phase name /Default/ Default selects the liquid phase from the database definitions, otherwise enter a phase name.

## SAVE\_FILE\_NAME

 This command is for the SCHEIL module.

Use this command to save the file. The files get the console number added to this name, e.g. 005.->Scheil\_006.POLY3 and Scheil\_segregation\_profile.txt).

Syntax	SAVE_FILE_NAME
Prompt	Name of files to be saved, POLY and segregation profiles.  Linux and Mac are case sensitive so ensure you enter .POLY3.

## SET\_NUMERICAL\_LIMITS



This command is available for the POLY, Scheil, and ED-EXP modules.

Change the criteria for convergence. This is to speed up a calculation in a complex system.

If there is any convergence problem in finding a stable solution at any stage during a calculation procedure enforced by COMPUTE\_EQUILIBRIUM, STEP\_WITH\_OPTIONS, MAP or ADVANCED\_OPTIONS), this message displays:

```
Convergence problems, increasing smallest site-fraction from 1.00E-30
to hardware precision 2.00E-14. You can restore using SET-NUMERICAL-
LIMITS
```

This implies that smallest site fraction in the current POLY workspace is automatically increased from the default value 1.00E-30 to the hardware-dependent precision (under Linux, as 2.00E-14). For other subsequent POLY module calculations, you can use this command to restore or reset the smallest site fraction to the previous or another preferred value, as well as to reset other numerical limits.

Syntax	SET_NUMERICAL_LIMITS
Prompt	<p>Maximum number of iterations /500/:</p> <p>By default, the program tries 500 iterations before it gives up. As some models give computation times of more than 1 CPU second/iteration, this number is also used to check the CPU time and the calculation stops if 500 CPU seconds/iterations are used.</p>
	<p>Required accuracy /1E-6/:</p> <p>This is a relative accuracy, and the program requires that the relative difference in each variable must be lower than this value before it has converged. A larger value normally means fewer iterations but less accurate solutions. The value should be at least one order of magnitude larger than the machine precision.</p>
	<p>Smallest fraction /1E-12/:</p> <p>This is the value assigned to constituents that are unstable. It is normally only in the gas phase you can find such low fractions.</p> <p>The default value for the smallest site-fractions is 1E-12 for all phases, except for IDEAL phase with one sublattice site (such as the GAS mixture phase in many databases) for which the default value is always as 1E-30 (unless you have used the SET_NUMERICAL_LIMITS command to reset an even-lower value (e.g. 1E-45, that is naturally enforced to all the phases in the system).</p>
	<p>Approximate driving force for metastable phases /Y/:</p> <p>Y is the default. Enter N to change the default as required and based on the options described below.</p> <p>This setting involves the convergence of metastable phases and affects their driving forces. It can also have an effect on when an equilibrium is considered successful.</p> <p>The default is to allow an equilibrium with metastable phases that have not converged, as long as the stable phases have converged. This is efficient but often causes approximate values of the</p>

Syntax	SET_NUMERICAL_LIMITS
	<p>driving forces for the metastable phases.</p> <p>If you change the default, it enforces metastable phases to converge. This gives accurate driving forces for metastable phases as well as stable phases. It can however take a slightly longer time, and if metastable phases do not converge it causes the equilibrium calculation to fail.</p>

## SKIP\_STEP\_CALCULATION

 This command is for the SCHEIL module.

Use the SKIP\_STEP\_CALCULATION command to skip the step (also called *one axis*) calculation that is by default performed before the Scheil calculation in Thermo-Calc. The step calculation is used to plot the equilibrium solidification curve, which is shown for comparison if the default axis variables are used. This is sometimes referred to as the *equilibrium line*.

If a comparison to equilibrium is not necessary, change the default setting and then only a Scheil calculation is performed. This results in a much faster calculation, which for a complex system may be preferable.

Syntax	SKIP_STEP_CALCULATION
Prompt	<p>Skip the step calculation before Scheil (equilibrium curve)? /N/:</p> <p>By default this is set to N. If a comparison to equilibrium is not necessary, enter Y and only a Scheil calculation is performed.</p>

## SOLUTE\_TRAPPING

 This command is for the SCHEIL module.

 [About the Scheil-Gulliver Solidification Simulations](#)

The SOLUTE\_TRAPPING command enables solute trapping. Once enabled, there are additional prompts and settings that are included and described with the [START\\_WIZARD](#) command.

Syntax	SOLUTE_TRAPPING
Prompt	<p>Consider solute trapping in primary phase? /N/</p> <p>The default (N) does not use solute trapping. If you choose Y, then solute trapping is used.</p> <div style="border: 2px solid blue; padding: 5px; margin-top: 10px;"> <p> When the USE_BACK_DIFFUSION command is enabled, the SOLUTE TRAPPING command is automatically disabled (set to N).</p> </div>

## START\_WIZARD

 This command is for the SCHEIL module.

 [About the Scheil-Gulliver Solidification Simulations](#)

Use this command to use the START\_WIZARD with step-by-step prompts to create a Scheil simulation.

You are prompted for settings including the major element or alloy, composition input in mass (weight) percent, the alloying elements, the temperature, fast diffusing components, and whether any phase should have a miscibility gap check. There are additional prompts included in the Start Wizard if USE BACK DIFFUSION or SOLUTE TRAPPING commands are set to Yes to enable back diffusion and solute trapping, respectively.



The Scheil with back diffusion feature is only available for systems with diffusion data, i.e. this model requires the use of a mobility database.

Use the other commands to make changes to settings that are not included in this wizard.

- EVALUATE\_SEGREGATION\_PROFILE
- GLOBAL\_MINIMIZATION
- LIQUID\_PHASE\_NAME
- SAVE\_FILE\_NAME
- STORED\_COMPOSITION\_TYPE
- TEMPERATURE\_STEP
- TERMINATION\_CRITERIA



If you also have a license for the Diffusion Module (DICTRA), you can import a previously calculated Scheil segregation profile into the software using the command INPUT\_SCHEIL\_PROFILE. Press F1 to search the online help.

Syntax	START_WIZARD
Prompt	<p>The following is a list of the prompts. Press &lt;Enter&gt; to accept the defaults that are shown between the forward slashes (/default/).</p> <p>Database /&lt;Default&gt;/:</p> <p>The available thermodynamic database and its version are available to enter (based on license).</p>

## Syntax

## START\_WIZARD

Enter ? for a list.

Major element or alloy:

The major element in the system, e.g. Fe for a steel.

Composition input in mass (weight) percent? /Y/:

Set whether to specify the Composition input in mass (weight) percent (the default) or in Mole percent. Enter Y for mass percent and N for mole percent.

1st alloying element:



You can specify all the alloying elements and their corresponding compositions on the same line when you are prompted to specify your first alloying element. For example, you can enter `Cr 5 Ni 1 Mo 0.5 C 0.01 N 0.02`.

You can directly specify mass or mole percent after the name (for example, enter `Cr 5`). If this is not specified, then you are prompted to enter it separately.

Mass (weight) percent /1/

2nd alloying element:

Specify the other alloying elements in the same way as you specify the first (for example, `c 1`).

Mass (weight) percent /1/

Next alloying element:

After you have specified your last alloying element, press <Enter> when requested to specify the next element. This ends the process of defining the bulk composition of the alloy system.

Temperature (C) /2000/:

Enter a value for the temperature in degrees C (Celsius) . This value should be sufficiently high so that the solidification simulation starts with the alloy system in the liquid single-phase region.

Decide which phases (if any) to reject at the `Reject phase(s)` prompt. Enter the name of the phases to reject or press <Enter> for `NONE`.

Decide whether to `Restore phase(s)`. You may want to restore a phase that you rejected when you ran the simulation earlier, or you may want to restore a phase that is rejected by default in your database. Enter the name of the phases to restore or press <Enter> for `NONE`.

Then you are asked to confirm whether these phases should be retained in this system:

OK? /Y/:

Press <Enter> to accept the default Y or type N and press <Enter>. Thermodynamic data about the alloy system you defined is retrieved from the database.

Should any phase have a miscibility gap check? /N/:

If you answer Y, you are prompted to specify the Phase with miscibility gap. Answer with a solution phase name(s) (e.g. FCC) as well as the Major constituents for sublattice (for each sublattice site) (for this example, `Cr` is entered for sublattice 1 and `C` for sublattice 2).

Press Enter to specify another phase.

Fast diffusing components: /None/:

Syntax	START_WIZARD
	Enter the elements that diffuse fast in solid phases and can assume to be in equilibrium in the whole phase. E.g. C in a steel.

## Back Diffusion and Solute Trapping Settings

The following prompts are included with the Start Wizard at various points in the set up if the:

- [USE\\_BACK\\_DIFFUSION](#) command is set to `Y` to enable back diffusion
- [SOLUTE\\_TRAPPING](#) command is set to `Y` to enable solute trapping

Syntax	START_WIZARD
Back Diffusion settings	<p>Mobility Database /&lt;Default&gt;/:</p> <p>The available mobility database and its version are available to enter (based on license). Enter ? for a list.</p> <p>You then are shown a list of what phases are available for the database and the defined system and have the opportunity to reject or restore phases at the next prompts.</p> <p>Reject phase(s) /NONE/:</p> <p>Restore phase(s): /NONE/:</p> <p>Then you are asked to confirm whether these phases should be retained in this system:</p> <p>OK? /Y/:</p> <p>Press &lt;Enter&gt; to accept the default <code>Y</code> or type <code>N</code> and press &lt;Enter&gt;.</p> <p>Temperature time dependence /LINEAR/:</p> <p>Specify the type of temperature time dependence. Press &lt;Enter&gt; to accept this default.</p> <p>An increased value moves the result from equilibrium toward a Scheil-Gulliver calculation.</p> <p> LINEAR is the only option and it corresponds to a constant cooling rate.</p> <p>Cooling rate (K/s) /.1/:</p> <p>Specify the cooling rate in Kelvin per second (K/s). An increased value moves the result from equilibrium toward a Scheil-Gulliver calculation.</p> <p>Expression for secondary dendrite arm spacing /CONSTANT/:</p> <p>Secondary dendrite arm spacing is the distance in meters (m) between two secondary dendrite arms. Specify the type of secondary dendrite arm spacing by entering <code>CONSTANT</code> (the default) or <code>POWER_LAW</code>.</p> <p>An increased value moves the result from equilibrium toward a Scheil-Gulliver calculation. Use</p> <ul style="list-style-type: none"> <li>• <code>CONSTANT</code> for a constant dendrite arm spacing. If you choose this, then enter a value for the Secondary dendrite arm spacing (m) /5E-05/: prompt.</li> <li>• <code>POWER_LAW</code> in the form <math>c * (\text{cooling rate})^{-n}</math> where <code>c</code> and <code>n</code> are entered in the next prompts. If you choose this, then enter a value for the Exponent <code>n</code> /.33/: and Scaling factor <code>c</code> /5E-05/:</li> </ul>

Syntax	START_WIZARD
	<p>Primary phase /AUTOMATIC/:</p> <p>The primary phase is the phase where the back diffusion takes place.</p> <p>If <code>Automatic</code> is selected (or kept as the default), the program tries to find the phase which gives the most back diffusion. To override this setting, choose a specific primary phase from the list. For back diffusion, only phases with diffusion data can be used as primary phases.</p>
Solute Trapping settings	<p>Expression for solidification speed /SCANNING_SPEED_AND_ANGLE/:</p> <p>SCANNING_SPEED_AND_ANGLE is currently the only available option. It is calculated as scanning speed * cos(alpha) where scanning speed and the angle alpha are given by the user.</p> <p>Decreasing the value moves the result toward a Scheil-Gulliver calculation.</p> <p>Scanning speed (m/s) /1/:</p> <p>Enter the Scanning speed to calculate the solidification speed, <math>V_s = V_{\text{scanning}} * \cos(\alpha)</math> m/s, where <math>V_{\text{scanning}}</math> is the user specified scanning speed.</p> <p>Alpha (degrees) /45/:</p> <p>Enter an Alpha value in degrees, which is the alpha angle, <math>\alpha</math>, between the solid/liquid boundary and scanning direction.</p> <p>Primary phase /AUTOMATIC/:</p> <p>If <code>Automatic</code> is selected (or kept as the default), the program tries to find the phase which gives the most solute trapping. To override this setting, choose a specific primary phase from the list. For solute trapping, only phases that dissolve all elements in the system can be used as primary phases.</p>

---

## TEMPERATURE\_STEP

 This command is for the SCHEIL module.

Use this command to define a temperature step.

Syntax	TEMPERATURE_STEP
Prompt	Set temperature step (C) /1/

## TERMINATION\_CRITERIA

 This command is for the SCHEIL module.

Use this command to define the termination criteria for the fraction of liquid or temperature.

Syntax	TERMINATION_CRITERIA
Prompts	Fraction of liquid or Temperature (F or T)? /F/ Fraction of liquid /.01/

## TEST\_INTERVAL\_FOR\_GLOBAL

 This command is for the SCHEIL module.

By default, global minimization is enabled (i.e. set to `Y` with the command [GLOBAL MINIMIZATION](#)).

Use the `TEST_INTERVAL_FOR_GLOBAL` command to change the interval at which a global equilibrium test is done, which by default is every 10th step as long as there are no changes in the set of stable phases. Every time the set of stable phases tries to change, a global equilibrium test is also done regardless of the test interval value chosen. A global equilibrium test means that the calculated equilibrium state obtained by the ordinary minimization calculation is tested against the Global Minimization Technique, and if the result is found unstable then the full global minimization calculation is performed instead.

Syntax	TEST_INTERVAL_FOR_GLOBAL
Prompt	Global test interval /10/: Global equilibrium test is done every Nth step, unless the set of stable phases changes. By default a test is done every 10th step. Enter a different number to change the interval.

## USE\_BACK\_DIFFUSION

 This command is for the SCHEIL module.



The Scheil with back diffusion feature is only available for systems with diffusion data, i.e. this model requires the use of a mobility database.



[About the Scheil-Gulliver Solidification Simulations](#)

The USE\_BACK\_DIFFUSION command enables back diffusion in the solid primary phase. Once enabled, there are additional prompts and settings that are included and described with the [START\\_WIZARD](#) command.

Syntax	USE_BACK_DIFFUSION
	<pre>Use back diffusion? /N/</pre> <p>The default (N) does not use back diffusion. If you choose Y, then back diffusion is used.</p>
Prompt	<div style="border: 1px solid blue; padding: 5px;">  <p>When the SOLUTE TRAPPING command is enabled, the USE_BACK_DIFFUSION command is automatically disabled (set to N).</p> </div>

## PARROT Module Commands

---

 [The PARROT Module](#) in the *Data Optimization User Guide*.

### In this section:

---

About the ENTER_PARAMETER Command .....	358
AMEND_PARAMETER .....	362
AMEND_SYMBOL .....	365
COMPILE_EXPERIMENTS .....	368
CONTINUE_OPTIMIZATION .....	369
CREATE_NEW_STORE_FILE .....	370
EDIT_EXPERIMENTS .....	371
ENTER_PARAMETER .....	372
LIST_ALL_VARIABLES .....	374
LIST_CONDITIONS .....	375
LIST_PARAMETER .....	376
LIST_PHASE_DATA .....	378
LIST_RESULT .....	379
LIST_STORE_FILE .....	384
LIST_SYMBOL_IN_GES .....	385
OPTIMIZE_VARIABLES .....	386
READ_PARROT_WORKSPACES .....	387
RECOVER_VARIABLES .....	388
REINITIATE .....	389
RESCALE_VARIABLES .....	390
SAVE_PARROT_WORKSPACES .....	391
SET_ALTERNATE_MODE .....	392

---

SET_EXTERNAL_PARAMETER .....	393
SET_FIX_VARIABLE .....	394
SET_OPTIMIZING_CONDITION .....	395
SET_OPTIMIZING_VARIABLE .....	397
SET_OUTPUT_LEVELS .....	398
SET_SCALED_VARIABLE .....	399
SET_STORE_FILE .....	400



## About the ENTER\_PARAMETER Command



For the PARROT Module, this command is the same and is described here. See [ENTER\\_PARAMETER](#) for the command details.

In the descriptions of the standard thermochemical properties and special physical properties for a phase, there are a number of parameters which may depend on the temperature and pressure. The expressions for these parameters can be given in a free form as a sum of terms with powers of T and P and may also include the natural logarithm and exponential function. This type of expression is called *TP-functions*. Identical parameters (in terms of parameter-names) are stored only once in the GIBBS workspaces.

The composition-dependence of the Gibbs energy is described in the GIBBS module by the internal data structure, which is created when the phase is entered (see [Gibbs Energy System \(GES\) Commands](#)). The Gibbs energy of a phase is always referred to one formula unit of the phase, i.e. the amount derived from the number of sites (i.e. the stoichiometric coefficient) for each sublattice. If vacancy is a constituent of a sublattice, the amount of matter per formula unit of the phase may vary with composition.

### How to Define a Parameter

A valid parameter should have the general form of:

```
<identifier>(<phase>, <constituent array>; <digit>) <xxx> <expression> <yyy>
<keyword Y or N> <zzz> !
```

The identifier must be followed by an opening parenthesis, a phase name, a comma and a constituent array. Optionally, the constituent array can be followed by a semicolon and a digit. The parameter name is terminated by a closing parenthesis. The parameter form is defined as:

- `<identifier>` is the parameter type;
- `<phase>` is the phase name (maximum 24 characters);
- `<constituent array>` is the specific constituent array in the phase;
- `<digit>` is the degree of composition-dependent interaction contribution (an integer number from 0 through 9), that is only for excess energy (L), Curie temperature (TC) and Bohr magneton number (BMAGN), as well as for volume-related parameters (V0 or VA or VC or VK); if it is valued as zero, or if it is for the standard Gibbs energy (G) for which the degree is always zero, it can be omitted;

- `<expression>` is the mathematical relation to describe the parameter;
- `<xxx>` and `<yyy>` are the low and high temperature limits respectively for the applicable temperature range of the parameter expression;
- `<keyword Y or N>` is the indicator on if there is continuation for the parameter expression or not;
- `<zzz>` is the reference index/number for the assessment of this parameter;
- The exclamation point `!` is used to indicate that the current parameter definition is ended.

## Parameter Name

The GES parameter name has a general form of:

```
<identifier>(<phase>,<constituent array>;<digit>)
```

Examples of parameter names:

- `G(GAS,C1O2)`: The Gibbs energy of formation of a CO<sub>2</sub> molecule in gas.
- `G(FCC,FE:VA)`: The Gibbs energy of formation of fcc Fe with interstitials.
- `L(LIQ,Fe,Cr;0)`: The regular solution parameter for Fe and Cr in liquid.
- `L(LIQ,Fe,Cr;1)`: The sub-regular solution parameter.
- `TC(BCC,Fe:Va)`: The Curie temperature of bcc Fe.
- `BMAGN(BCC,Fe:Va)`: The Bohr magneton number parameter of bcc Fe.

The parameter name consists of several parts. The first is a type-identifier and these can be used:

- `G`: Standard energy parameter (Gibbs energy of formation) or for interaction parameters;
- `L`: Excess energy parameter (Gibbs energy of interaction) always used for interaction parameters;
- `TC`: Curie temperature for magnetic ordering;
- `BMAGN` or `BM`: Bohr magneton number for magnetic ordering (or Born function  $\omega_{Pr,Tr}$  for aqueous solute species).
- `VO`: Molar volume at 298.15 K and 1 bar (a numeric value only);

- VA: Integrated thermal expansivity;  $\int_{298.15}^T \alpha(T) dT$
- VC: High-pressure fitting parameter;
- VK: Isothermal compressibility.



For more information about the high pressure volume, see X.-G. Lu, M. Selleby, B. Sundman, "Implementation of a new model for pressure dependence of condensed phases in Thermo-Calc", Calphad. 29, 49–55 (2005).



When necessary quantities as  $H$  (enthalpy),  $S$  (entropy),  $V$  (Volume),  $F$  (Helmholtz energy), etc., can be calculated from the Gibbs energy.

## Phase Name

Specifying the `PHASE NAME` in uppercase is recommended; however, if you prefer to write it as a mixture of uppercase and lowercase, it automatically converts all lowercase to uppercase, as the GIBBS module only recognizes uppercase phase names. It is important that if a phase bears a legal phase-type (among `G`, `A`, `Y`, `L`, `I`, `F` and `B`) in its phase definition (already by the `PHASE` keyword; such as `GAS:G`, `LIQUID:L`, `SLAG:L`, `IONIC-LIQ:Y`, `SPINEL:I`, `FCC_L12:F`, `HCP_D021:F`, `BCC_B2:B`, `AQUEOUS:A`), such a valid phase-type code should not be attached to the phase name when `ENTER_PARAMETER` is executed.

## Constituent Array

The constituent array consists of a list of constituent names. Interaction parameters have two or more constituents from the same sublattice separated by a comma. If the phase has sublattices, at least one constituent in each sublattice must be specified. The constituents in different sublattices must be given in sublattice order and are separated by a colon.

After the component array, a sub-index digit can be specified after a semicolon. This digit must be in the range 0 to 9. The interpretation of the sub-index depends on the excess energy model used for the phase. If no semicolon and digit are given, the sub-index value is assumed to be as zero.

The excess energy parameters, e.g. the regular/subregular (binary) parameter or ternary parameters, are multiplied with two or more fractions of the constituents from the same sublattice of the solution phase. These additional constituents must be given as interacting constituents (as the following prompt).



Solution phases with sublattices can have interacting constituents in each sublattice.

### ***Interaction Parameter***

An interaction parameter, which is used to describe the excess term of a quantity, must have two or more constituents that interact with each other on a specified sublattice site of the given phase. It is arbitrary which of these constituents is given as the first constituent and what is given as the interacting constituents. The software always sorts the constituents (in each sublattice) in alphabetical order when the parameter name is written as a prompt (for entering its parameter value) and when the parameter is listed (using the GIBBS commands [LIST\\_PARAMETER](#) or [LIST\\_PHASE\\_DATA](#)). This is important for all asymmetric interaction parameters where the sign of the interaction parameter must depend on the appearance order.

Use an asterisk\* to denote that the excess interaction parameter is independent of the constituents of a specific sublattice. For example,  $L(\text{FCC\_L12}, \text{AL}, \text{NI} : *)$  means that the interaction parameter is for the binary interaction between constituents AL and NI on the first sublattice in the FCC\_L12 solution phase, while it is independent of all constituents on the second sublattice. An interaction parameter in the list of constituents is always added to the Gibbs energy and the asterisk \* is calculated with the term of  $[1\sum y(\text{specified constituents})]$ , which implies that in an A-B binary system these  $L$  parameters are identical (but in higher-order systems, these are different):

- $L(\text{phase}, \text{A}, \text{B})$  is multiplied with  $X(\text{A}) \cdot X(\text{B})$
- $L(\text{phase}, \text{A}, *)$  is multiplied with  $X(\text{A}) \cdot (1 - X(\text{A}))$
- $L(\text{phase}, \text{B}, *)$  is multiplied with  $X(\text{B}) \cdot (1 - X(\text{B}))$

If you press <Enter> when you are asked for a parameter name or if you have improperly input the entire parameter name, you are asked for each of these items in the name.

## AMEND\_PARAMETER

-  This command is for the PARROT module.
-  There is also a Gibbs (GES) module command with the same name.

Interactively modify the TP-function(s) for a specific parameter for a phase with this command. This is useful in order to correct typing errors because the old function is made available for interactive editing on the terminal.

Syntax	AMEND_PARAMETER
Prompts	<p>Parameter: &lt;Parameter name&gt;</p> <p>Specify a correct parameter name. If a parameter name is not acceptable or &lt;Enter&gt; is pressed, the error message displays:</p> <pre>*** Error, please re-enter each part separately</pre> <p>The program prompts for separate input for each part for a parameter name.</p>
	<p>Identifier (/X/): &lt;G or L, or TC, or BM, or V0 or VA or VC or VK&gt;</p> <p>Specify one of these types of legal identifiers</p> <p> <a href="#">ENTER_PARAMETER</a></p>
	<p>Phase name (ABCD/): &lt;Phase name&gt;</p> <p>Specify the phase name.</p>
	<p>Constituent (in sublattice # /ABC/): &lt;Species name&gt;</p> <p>Specify the constituent name.</p>
	<p>Interacting constituent (In sublattice # /XYZ/): &lt;Species name&gt;</p> <p>Specify the interacting constituent name; if there is no interacting constituent, press &lt;Enter&gt;.</p>
	<p>Interacting constituent (in sublattice # /XYZ/): &lt;Species name&gt;</p> <p>If there is more than one interacting constituent, specify these; otherwise press &lt;Enter&gt;.</p>
	<p>Degree /#/ : &lt;Degree&gt;</p> <p>Specify a numerical number as the degree of the phase parameter.</p>
	<p>After the parameter name is specified correctly, the program lists its current definition (either preset in database or defined by the ENTER_PARAMETER command), such as:</p> <pre>L (PHASE2, AL, MG; 1) = 298.15&lt;T&lt;2000.00: +5000 2000.00&lt;T&lt;4500.00: +4500 4500.00&lt;T&lt;6000.00: +4000</pre> <p>You are prompted to change the parameter definition:</p>

Syntax	AMEND_PARAMETER
	<p>Do you want to change the number of ranges /No/:</p> <p>Enter Y to change the number of ranges for the chosen function, or change some of the temperature limits in the definition, then retype both the low/high temperature limits and functions</p> <p>If you do not want to change the number of ranges but want to change the function(s) in one or more ranges, press &lt;Enter&gt; and the whole definition of the chosen parameter in all ranges (if any) is listed such as:</p> <pre>DIFFERENT FUNCTIONS IN THESE RANGES 298.15&lt;T&lt;2000.00 2000.00&lt;T&lt;4500.00 4500.00&lt;T&lt;6000.00</pre> <p>This message prompts:</p> <p>Do you want to change range limits /No/:</p> <p>If there is more than one range, this question is prompted. Press &lt;Enter&gt;.</p>
	<p>Range number (0 to exit) /0/: &lt;Range number&gt;</p> <p>If the function of a parameter is different in two or more temperature ranges, you must specify the range of the function of which you want to amend. Or press &lt;Enter&gt; or type 0 to exit this command without making any change.</p>
	<p>Function</p> <p>The previous function is available for editing. The editing is performed within the general subroutine <code>FOOLED</code>, as described in <a href="#">AMEND_SYMBOL</a>. This routine prompts as follows:</p> <pre>1:+:&gt;</pre> <p>The prompt consists of the current position in the string and the character at that position between colons, (:).</p> <h3>Commands</h3> <p>These commands can be given:</p> <ul style="list-style-type: none"> <li>• Help: ?</li> <li>• Move CP to last or first character: &lt;+/-&gt; A</li> <li>• Delete characters from CP: &lt;+-#characters&gt; D</li> <li>• Exit: E</li> <li>• Find: &lt;#occurrences&gt; F&lt;string&gt;@</li> <li>• Insert: I&lt;string&gt;@</li> <li>• Move: &lt;+-#positions&gt; M</li> <li>• Restore string: R</li> <li>• Substitute: S&lt;OLD&gt;@&lt;NEW&gt;@</li> <li>• Type string: T</li> </ul> <p>Where CP denotes the current position in a string, # means number of, @ is a terminator of an</p>

Syntax	AMEND_PARAMETER
	<p>input or search string.</p> <div data-bbox="363 317 1398 459" style="border: 1px solid blue; padding: 5px;"> When the string is typed the character at the current position is replaced by an underscore <code>_</code>.</div> <p>To finish the editing of the current function, type <code>E</code> at the prompt.</p>
	<p>Range number (0 to exit) /0/: &lt;Range number&gt;</p> <p>Give a range number to edit that function, or press &lt;Enter&gt; or type 0 to exit this command.</p>

## AMEND\_SYMBOL



This command is for the PARROT module.



The same command is also available for the Gibbs (GES) module.

Interactively calculate the current values of a function or table (predefined by the switched database or previously entered with ENTER\_SYMBOL) at the current temperature and pressure condition, and also possible to modify the definitions of variables or functions.



The current values of the temperature and pressure used to calculate the functions or tables are the ones listed with LIST\_STATUS. There is no way to change the current temperature and pressure values interactively.

To modify the definitions of entered symbols (variable, functions or parameters), the performance of this command is slightly different one from another, as described below:

- For variables, the values can be changed.
- For functions the low and high temperature limits in various ranges (if any), as well the expressions, can be changed.
- For parameters the low and high temperature limits in various ranges (if any), as well the expressions, can be changed.

Syntax	AMEND_SYMBOL
Prompts	<p>Name: &lt;Symbol name&gt;</p> <p>Specify the name of an entered symbol.</p> <p>For tables and functions (or parameters which are treated as functions after these are entered by the ENTER_SYMBOL command with the <code>Parameter</code> keyword, but not by the ENTER_PARAMETER command), after the symbol name is specified here, the program automatically calculates the values under the current temperature and pressure conditions, and lists the current values, such as:</p> <pre>FUNCTION VALUE  2.52500000E+01 TABLE VALUE    1.56000000E+02</pre> <p>For variables and functions (or parameters), there are additional prompts and depend on the symbol type.</p> <p>For a variable, its current value is displayed, and you can change it to a new value:</p> <pre>Value /Current value/: &lt;New value&gt;</pre> <p>For a function (or a parameter entered as a symbol):</p> <pre>Do you want to change the number of ranges /No/:</pre>

Syntax	AMEND_SYMBOL
	<p>Enter Y to change the number of ranges for the function, or change some of the temperature limits in the definition, then retype both the low/high temperature limits and functions (see all the remaining details in the ENTER_SYMBOL command).</p> <p>If you do not want to change it, press &lt;Enter&gt;. The definition of the chosen function in all ranges (if any) is listed, for example:</p> <pre>DIFFERENT FUNCTIONS IN THESE RANGES 298.15&lt;T&lt;2000.00 2000.00&lt;T&lt;4500.00 4500.00&lt;T&lt;6000.00</pre> <p>and this message displays:</p> <pre>Do you want to change range limits /No/:</pre> <p>If a function is different in two or more temperature ranges, you must specify the range of the function of which you want to amend. Or press &lt;Enter&gt; or type 0 to exit this command without making any change.</p>
	<pre>Range number (0 to exit) /0/: &lt;Range number&gt; Function:</pre> <p>The previous function is available for editing. The editing is performed within the general subroutine FOOLED. This routine prompts as follows:</p> <pre>1:+:&gt;</pre> <p>The prompt consists of the current position in the string and the character at that position between colons (::).</p> <h3>Commands</h3> <p>These commands can be given:</p> <ul style="list-style-type: none"> <li>• Help: ?</li> <li>• Move CP to last or first character: &lt;+/-&gt; A</li> <li>• Delete characters from CP: &lt;+-#characters&gt; D</li> <li>• Exit: E</li> <li>• Find: &lt;#occurrences&gt; F&lt;string&gt;@</li> <li>• Insert: I&lt;string&gt;@</li> <li>• Move: &lt;+-#positions&gt; M</li> <li>• Restore string: R</li> <li>• Substitute: S&lt;OLD&gt;@&lt;NEW&gt;@</li> <li>• Type string: T</li> </ul> <p>Where CP denotes the current position in a string, # means number of, @ is a terminator of an input or search string.</p>

---

Syntax	AMEND_SYMBOL
	<div data-bbox="363 260 1390 401" style="border: 1px solid blue; padding: 5px;"> When the string is typed the character at the current position is replaced by an underscore _.</div> <p data-bbox="363 428 1040 457">To finish the editing of the current function, type <code>E</code> at the prompt.</p>

## COMPILE\_EXPERIMENTS

The descriptions of the experimental equilibria saved on a textual \*.POP file, given as commands in the POLY and ED\_EXP syntax, is compiled by this command into the PARROT structured data, i.e. stored into the current work \*.PAR file which is created with [CREATE\\_NEW\\_STORE\\_FILE](#), set by [SET\\_STORE\\_FILE](#), or opened with [READ\\_PARROT\\_WORKSPACES](#), all done before COMPILE\_EXPERIMENTS.

This command also lists the compilation details on screen or onto a listing file (\*.TXT).



If a syntax error in the commands is detected, the compilation is terminated. You need to correct the error in the \*.POP file (using a textual editor) and recompile it with the \*.PAR work (store) file.

After a successful compilation, the program translates all the experimental data points in the \*.POP file into a graphical experimental data file (\*.EXP) in the DATAPLOT syntax, which can be further edited and used as a graphical experimental data file (\*.EXP) for the purposes of imposing onto calculated/plotted phase diagrams and/or property diagrams for comparisons during the assessment process.



*DATAPLOT User Guide* included with this documentation set.

Syntax	COMPILE_EXPERIMENTS
	A window displays to open the original experimental data file (*.POP). Enter a *.POP <b>File name</b> box, and specify the working directory in the <b>Look in</b> box. Normally the default file-type in the <b>Files of type</b> field is the correct one for the experimental data format (i.e. *.POP file). Click <b>Open</b> or <b>Cancel</b> button as applicable.
Prompts	<p>Output to screen or file /Screen/: &lt;*.pop file name&gt;</p> <p>A list of the source code in the *.POP file and error messages during compilation is written on screen (by pressing &lt;Enter&gt;) or on a list *.TXT file under a specific file name which can later on be opened and edited by any basic textual editor.</p>
	<p>Initiate store file /Y/:</p> <p>After a successful compilation, the data in the POLY workspace is stored as a new block of equilibria on the current work (store) file. By default the store file is initiated before compilation and only the compiled block is stored on the current work file (consequently, the *.PAR file is updated).</p> <p>A window displays to generate/store the graphical experimental data file (*.EXP) in the DATAPLOT syntax. Enter a *.EXP file name in the <b>File name</b> field and specify the working directory when opening the *.EXP file in the <b>Look in</b> field. Normally the default file type in the <b>Files of type</b> field is the correct one for the graphical experimental data format (i.e. *.EXP file). Click <b>Open</b> or <b>Cancel</b> as applicable.</p> <p>To keep the current work file, enter N.</p>

## CONTINUE\_OPTIMIZATION

 This command is for the PARROT module.

This command continues the optimization using the same Hessian matrix.



In some cases it is not possible to attempt continuation. For example, if the optimization has already converged or if the optimizing parameters or the set of experimental equilibria used in the current optimization runs are changed. The program gives a warning message if you try to continue when you make these changes.

Syntax	CONTINUE_OPTIMIZATION
Prompts	<p>Are you sure? /N/: &lt;N or Y&gt;</p> <p>Prior to this question, a message displays to confirm if it is OK to continue:</p> <p style="padding-left: 40px;">It is safe to CONTINUE only after TOO MANY ITERATIONS and no change in variables and experiments ...</p> <p>You can accept the default (N) in order to cancel this special attempt.</p>
	<p>Number of iterations /N/: &lt;Integral number of iteration&gt;</p> <p>Specify an integral number for the optimization iteration. The specified iteration number is shown as the default number. The PARROT program tries this exact number of different sets of values of the optimizing variables without stopping the optimization even it would have converged earlier or should give up earlier.</p>

## CREATE\_NEW\_STORE\_FILE



This file is hardware dependent and cannot be read by any text editor.

Create a binary file to be used as a *work file* (also called *store file*) before any optimization can be done. The workspace used by the GIBBS, POLY and PARROT modules is stored automatically on the work file but not any experimental information. The binary work file has a default extension of `PAR` under MS-Windows or `par` under Linux/UNIX.

A work file that is created at a previous run can be used in the PARROT module with the other command SET-STORE-FILE.

Syntax	CREATE_NEW_STORE_FILE
Prompt	<*.par file name> A <b>Save</b> window displays. Enter a <b>File name</b> and specify the working directory in the <b>Save in</b> field. Normally the default file-type in the <b>Files of type</b> field is the correct one for the PARROT workspace format (i.e. PAR file). Click <b>Save</b> or <b>Cancel</b> as applicable.

## EDIT\_EXPERIMENTS

Initialize and make available the sub-module for editing experimental equilibria, i.e. the ED\_EXP module. All experimental equilibria compiled from the \*.POP file can be accessed in the ED\_EXP module.

The ED\_EXP module is similar to the normal POLY module, but some commands are special and some POLY commands are not available. In the ED\_EXP module, you can calculate each equilibrium separately or together, provide start values for equilibria which failed to converge, set weights, and modify the values of experiments or conditions.

Syntax	EDIT_EXPERIMENTS
	You are now in this submodule and can start using the commands.

- The first required command after entering the ED\_EXP module for the first time (from the PARROT module GOTO MODULE ED-EXP) should always be READ\_BLOCK in order to load the experimental data block from the current work \*PAR-file for editing.
- The READ\_BLOCK command must also be used prior to any other ED\_EXP command if the module is reinitiated, or if no experimental data block is not previously read from a work file compiled with a proper experimental data .POP file, or if you want to change to another data block for editing.
- If any change is made in the ED\_EXP module remember to use the command SAVE\_WORKSPACE before going BACK to the PARROT module.



Although the TABLE\_HEAD, COMMENT and FLUSH\_BUFFER commands are visible in the ED\_EXP module, these cannot be used here as these only work in a \*.POP file (or \*.DOP file for DICTRA).

## ENTER\_PARAMETER

 This command is for the PARROT module.

 There is also a Gibbs (GES) module command with the same name.

 [About the ENTER\\_PARAMETER Command](#)

Use this command to enter TP-function(s) for a specific parameter for a phase interactively with this command. If there is already a parameter expression defined (for this parameter), that is deleted and replaced with newly entered one.

The entered TP-Function(s) for the parameter can be changed later with the command AMEND\_PARAMETER.

Syntax	ENTER_PARAMETER
Prompts	<p>Parameter: &lt;Parameter name&gt;</p> <p>As explained in <a href="#">About the ENTER_PARAMETER Command</a>, specify a correct and complete Parameter Name, which should contain all the necessary parts of the general form:</p> <p>&lt;Identifier&gt;&lt;Phase&gt;,&lt;Constituent array&gt;;&lt;Digit&gt;</p> <p>If a parameter name is not acceptable or &lt;Enter&gt;, is pressed, a message displays:</p> <pre>*** Error, please re-enter each part separately</pre> <p>and you are prompted for input for each required part for a parameter name.</p>
Options	Description or Information
Identifier	<p>Identifier (/X/): &lt;G or L, or TC, or BM, or V0 or VA or VC or VK&gt;</p> <p>If this command is used one or more times, the previous value on this prompt is set as default. Press &lt;Enter&gt; for the same type identifier or specify a new type.</p>
Phase name	<p>Phase name (/ABCD/): &lt;Phase name&gt;</p> <p>Each parameter is valid for a specific phase only. The name of that phase must be supplied (maximum 24 characters). The name can be abbreviated.</p>
Constituent	<p>Constituent (in sublattice # /ABC/): &lt;Species name&gt;</p> <p>A parameter is identified by the constituents on a specified sublattice site of the given phase, the stoichiometric coefficients of which are multiplied with the parameter. The name of the constituent can be abbreviated.</p> <div style="border: 2px solid blue; padding: 10px; margin: 10px 0;">  It is the species name, not the stoichiometric formula required here. </div> <p>If this command is used one or more times, the previous value on this prompt is set as default. Accept it by pressing &lt;Enter&gt; if the constituent is the same, or specify a new species name.</p>

Syntax	ENTER_PARAMETER
Interacting constituent	<p>For phases with several sublattices, the program asks for one constituent in each sublattice.</p> <p>Interacting constituent (in sublattice # /XYZ/): &lt;Species name&gt;</p> <p>If this command is used one or more times, the previous value on this prompt is set as default. Press &lt;Enter&gt; to accept it if the constituent is the same, or specify a new species name.</p> <div data-bbox="410 436 1395 575" style="border: 2px solid blue; padding: 10px; margin: 10px 0;">  To cancel the default value of the interacting constituent type <code>NONE</code> or the name of another constituent. </div> <p>This question is repeated until all the interested interacting constituent(s) on a specific sublattice in the phase are specified, and finally an &lt;Enter&gt; is enforced.</p>
Degree	<p>Degree /#/ : &lt;Degree&gt;</p> <p>Degree is model-dependent. Specify an integer number (a value from 0 through 9) as the degree of composition-dependent interaction contribution for the phase parameter. This is valid for excess energy (<math>L</math>), Curie temperature (<math>T_C</math>) and Bohr magneton number (<math>B_{MAGN}</math>), as well as for volume-related parameters (<math>V_0</math>, <math>V_A</math>, <math>V_C</math> or <math>V_K</math>).</p> <p>For binary interaction parameters, the degree is usually the power in the Redlich-Kister expression. For ternary interaction parameters, it is usually the Hillert ternary index.</p> <p>For a standard <math>G</math> parameter for a pure component (end-member) its degree should be always 0 and this prompt should not display.</p>
Phase parameter	<p>After this prompt, the program echoes on the screen the full TP-Function of the phase parameter.</p> <p>Low temperature limit /298.15/: &lt;Lowest temperature limit in K&gt;</p> <p>Specify the lowest temperature limit (in Kelvin), or the lowest-pressure limit (in Pascal but entered as a negative number), for the current TP-Function.</p> <p>Function: &lt;Definition for a function&gt;</p> <p>A TP-Function consists of terms in T and P.</p> <p>&amp; &lt;Continuation of the definition for the current function&gt;</p> <p>Continuation of a TP-Function definition.</p> <p>High temperature limit /6000/: &lt;High temperature limit in K&gt;</p> <p>Specify the high temperature limit (in Kelvin), or the high-pressure limit (in Pascal; only if the <i>lowest limit</i> is entered as a negative number) for the current TP-Function.</p> <p>Any more ranges? /N/:</p> <p>Enter Y for more function(s) or N to end this command.</p>

## LIST\_ALL\_VARIABLES

 This command is for the PARROT module.

Create a list of the values and status of all variables in the PARROT workspace on screen or in a text file. After an optimization some statistical information is also written. The correlation matrix is written if that option is chosen.

 [SET\\_OUTPUT\\_LEVELS.](#)

Syntax	LIST_ALL_VARIABLES
Prompt	Output to screen or file /Screen/: <File name>
	A list of the current values and status, as well statistical information (if after optimization) and the correlation matrix (optionally), for all variables is written on screen (by pressing <Enter>) or on the list file under a specific file name which can later on be opened and edited by basic text editor.
	<p><b>Example</b></p> <p>The following is the listed variables for the example TCEX36, after the first optimization when having set all variables available for optimization.</p> <pre> == OPTIMIZING VARIABLES == AVAILABLE VARIABLES ARE V1 TO V00 VAR.   VALUE           START VALUE       SCALING FACTOR    REL.STAND.DEV V1     2.03729090E+04    2.03688352E+04    2.03688352E+04    3.41524152E+00 V2     -2.94286372E+01    -2.94286372E+01    -2.94286372E+01    2.37944771E+00 V11    -2.17373936E+04    -2.18095983E+04    -2.18095983E+04    3.97086303E-02 V12    1.52107184E+01    1.51660547E+01    1.51660547E+01    5.84552327E-02 V15    2.42082351E+04    2.45139169E+04    2.45139169E+04    5.00914471E+00 V16    -8.38723972E+00    -8.83460472E+00    -8.83460472E+00    1.60961318E+01 V17    3.08916533E+03    3.15802558E+03    3.15802558E+03    9.63738108E+00 V19    2.20292586E+04    2.21385673E+04    2.21385673E+04    4.14274011E+01  V20  - 7.04217974E+00    -7.22424882E+00    -7.22424882E+00    1.00561540E+02 NUMBER OF OPTIMIZING VARIABLES :    9 ALL OTHER VARIABLES ARE FIX WITH THE VALUE ZERO THE SUM OF SQUARES HAS CHANGED FROM    8.00002729E+04 TO    8.00002719E+04 DEGREES OF FREEDOM    45.  REDUCED SUM OF SQUARES    1.77778382E+03 </pre>

---

## LIST\_CONDITIONS

 This command is for the PARROT module.

Use this command to generate a list of the current values of optimization conditions, as well the current status of listing, either on screen or in a basic text file which can be opened and edited by a text editor.

Syntax	LIST_CONDITIONS
Prompt	Output to screen or file /Screen/: <File name> Press <Enter> to output to screen or on the list file under a specific file name.

## LIST\_PARAMETER

-  This command is for the PARROT module.
-  There is also a Gibbs (GES) module command with the same name.

List the TP-function(s) of a specific parameter for a phase. You must supply the name of the phase parameter:

The parameter name: <Identifier>(<Phase>,<Constituent array>;<Digit>)



For encrypted commercial databases, you may not be able to use this command to list any retrieved parameter.

Syntax	LIST_PARAMETER
	Parameter: <Parameter name> Specify a correct and complete parameter name, which should contain all the necessary parts of the general form: <identifier>(<phase>,<constituent array>;<digit>)
Prompts	 See <a href="#">About the ENTER_PARAMETER Command</a> to learn how to define the parameter name. If a parameter name is not acceptable or <Enter> is pressed, an error message displays: <pre>*** Error, please re-enter each part separately</pre> The program prompts for separate input for each required part for a parameter name.
	Identifier (/X/): <G or L, or TC, or BM, or V0 or VA or VC or VK> Specify one of these types of legal identifiers.
	Phase name (/ABCD/): <Phase name> Specify the phase name.
	Constituent (in sublattice # /ABC/): <Species name> Specify the constituent name on the specified sublattice site of the given phase.
	Interacting constituent (in sublattice # /XYZ/): <Species name> Specify the interacting constituent name on the specified sublattice site of the given phase; if there is no interacting constituent, press <Enter>.
	Interacting constituent (in sublattice # /XYZ/): <Species name> If there is more than one interacting constituent on the specified sublattice site of the given phase, specify them; otherwise press <Enter>.
	Degree /#/ : <Degree>

---

Syntax	LIST_PARAMETER
	Specify a numerical number as the degree for the phase parameter.

## LIST\_PHASE\_DATA



This command is for the PARROT module.

List all the phase from the Gibbs (GES) module for a specific phase. These are displayed on screen. The thermochemical parameters listed for each phase are always in SI units.

Syntax	LIST_PHASE_DATA
Prompt	Phase name: <Phase name> Specify a phase name.

## LIST\_RESULT

 This command is for the PARROT module.

List the result of the current optimization run on screen or in a specified file. The level of detail listed can be prescribed by the command SET\_OUTPUT\_LEVELS.

The LIST\_RESULT command has an option **G** for creation of an experimental data file with two columns, one for the experimental value and the other for the calculated value. It allows plotting a diagram and visualizing the fitting results. The plot is automatically plotted and you can scale it giving commands in the POST module. The option **D** lists all experiments, including those with fulfilled inequalities which are suppressed with the default option **C**.

Syntax	LIST_RESULT
Prompt	Full, Condensed, Detailed or Graphic format: /C/: <C or D or F or G>
	<p>Choose a format.</p> <ul style="list-style-type: none"> <li>The <b>F</b> (full) format is obsolete but retained for backward compatibility.</li> <li><b>C</b> (condensed) is the default.</li> <li><b>D</b> (detailed) lists all experiments, including those with fulfilled inequalities (not included with the default condensed file).</li> <li>The <b>G</b> (graphical) format creates an experimental data file containing two columns, one with the experimental value and the other with the calculated value. This allows plotting of a diagram where all symbols should be on the diagonal if the fit is perfect. The plot is automatically plotted and you can scale it giving commands in the POST module.</li> </ul> <p>Output to screen or file /Screen/: &lt;File name&gt;</p> <p>A list of the current optimization results are written on screen (by pressing &lt;Enter&gt;) or on the list file under a specific file name which can later on be opened and edited by basic text editor.</p>
	<p>The output normally consists of these parts:</p> <ul style="list-style-type: none"> <li>A title showing the data of the action;</li> <li>A message on successful optimization with the iteration number in the last optimization;</li> <li>A paragraph describing the optimization condition;</li> <li>A list of the latest set of optimized and fixed variables;</li> <li>A paragraph describing the optimization quality (some statistical information);</li> <li>A correlation matrix for all optimizing variables (optionally shown, pre-set by the SET_OUTPUT_LEVELS command);</li> <li>A list of all parameters (including their symbol names, status and current values or function expressions), and all phase descriptions (including phase name, model names, constituents, and phase's <b>G/L/TC/BM</b> expressions) predefined for each phase in the system (optionally shown, pre-set by the SET_OUTPUT_LEVELS command);</li> <li>A paragraph describing the alternate equilibria (and possible error during optimization);</li> </ul>

Syntax

LIST\_RESULT

- A detailed list on all the equilibrium points used in the current optimization. If the alternate-mode is used in the optimization, only an error value is shown. For normal-mode calculations, a list is shown for the experimental equilibrium numbers (first column) and corresponding original experimental data (second column, in the form `quantity = value`) which are used in optimization (i.e. non-zero weighted points). Such experimental data are by each point compared by the calculated value (third column) after the last optimization. Also listed are the experimental error (column 4), the difference between the calculated value and original experimental data (column 5), and the contribution to the sum of least square (column 6).

### Example

The following is the listed result after the first optimization based on the prescribed definitions in the TCEX36a.TCM and TCEX36b.TCM, but having set all variables available for optimization.

```
=====
                OUTPUT FROM P A R R O T.
*** SUCCESSFUL OPTIMIZATION. ***      NUMBER OF ITERATIONS:  10
== OPTIMIZING CONDITIONS ==
    RELATIVE STANDARD DEVIATIONS FOR EXPERIMENTS:  N
    MINIMUM SAVE ON FILE:  Y
    ERROR FOR INEQUALITIES =  1.00000000E+00
    RELATIVE STEP FOR CALCULATION OF DERIVATIVES =  1.00000000E-04
ARGUMENTS FOR SUBROUTINE VA05AD (HSL)
    MAXFUN =  1    DMAX =  1.00000000E+02    H =  1.00000000E-04    ACC = (INITIAL
SUM OF SQUARES) *  1.00000000E-03
== OPTIMIZING VARIABLES ==
AVAILABLE VARIABLES ARE V1  TO V00
VAR.   VALUE                START VALUE          SCALING FACTOR      REL.STAND.DEV
V1     2.03749463E+04        2.03729090E+04        2.03729090E+04      3.41455863E+00
V2     -2.94286372E+01      -2.94286372E+01      -2.94286372E+01      2.37944774E+00
V11    -2.17395673E+04      -2.17373936E+04      -2.17373936E+04      3.98405298E-02
V12     1.52107184E+01       1.52107184E+01       1.52107184E+01       5.82861832E-02
V15     2.42106560E+04       2.42082351E+04       2.42082351E+04       5.07239609E+00
V16    -8.38723972E+00      -8.38723972E+00      -8.38723972E+00      1.69546796E+01
V17     3.08947424E+03       3.08916533E+03       3.08916533E+03       9.85220694E+00
```

V19 2.20314615E+04 2.20292586E+04 2.20292586E+04 4.16329629E+01 V20  
 -7.04217974E+00 -7.04217974E+00 -7.04217974E+00 1.03161466E+02

NUMBER OF OPTIMIZING VARIABLES : 9

ALL OTHER VARIABLES ARE FIX WITH THE VALUE ZERO

THE SUM OF SQUARES HAS CHANGED FROM 8.00002719E+04 TO 8.00002709E+04

DEGREES OF FREEDOM 45. REDUCED SUM OF SQUARES 1.77778380E+03

Number of alternate equilibria 14

SYMBOL STATUS VALUE/FUNCTION

R 80000000 8.3145100E+00

RTLNP 20000000 +R\*T\*LN(1E-05\*P)

V1 48000000 2.0374946E+04

V2 48000000 -2.9428637E+01

V11 48000000 -2.1739567E+04

V12 48000000 1.5210718E+01

V15 48000000 2.4210656E+04

V16 48000000 -8.3872397E+00

V17 48000000 3.0894742E+03

21 V19 48000000 2.2031462E+04 22 V20 48000000 -7.0421797E+00

LIQUID

EXCESS MODEL IS REDLICH-KISTER\_MUGGIANU

CONSTITUENTS: A,B

$G(\text{LIQUID}, A; 0) - G(\text{BCC}, A; 0) = 500.00 < T < 2000.00: +14000 - 10 * T$

$G(\text{LIQUID}, B; 0) - G(\text{BCC}, B; 0) = 500.00 < T < 2000.00: +18000 - 12 * T$

$L(\text{LIQUID}, A, B; 0) = 500.00 < T < 2000.00: +V11 + V12 * T$   $L(\text{LIQUID}, A, B; 1) =$   
 $500.00 < T < 2000.00: +V13 + V14 * T$

A2B

2 SUBLATTICES, SITES 2: 1 CONSTITUENTS: A : B

$G(\text{A2B}, A; B; 0) - 2 G(\text{BCC}, A; 0) - G(\text{BCC}, B; 0) =$

$500.00 < T < 2000.00: +V1 + V2 * T + V3 * T * \ln(T)$

BCC

EXCESS MODEL IS REDLICH-KISTER\_MUGGIANU

CONSTITUENTS: A,B

$G(\text{BCC}, A; 0) - G(\text{BCC}, A; 0) = 500.00 < T < 2000.00: 0.0$

$G(\text{BCC}, B; 0) - G(\text{BCC}, B; 0) = 500.00 < T < 2000.00: 0.0$

$L(\text{BCC}, A, B; 0) = 500.00 < T < 2000.00: +V15 + V16 * T$

$L(\text{BCC}, A, B; 1) = 500.00 < T < 2000.00: +V17 + V18 * T$

FCC

```

EXCESS MODEL IS REDLICH-KISTER_MUGGIANU
CONSTITUENTS: A,B

      G(FCC,A;0)-G(BCC,A;0) =    500.00<T< 2000.00: 408
      G(FCC,B;0)-G(BCC,B;0) =    500.00<T< 2000.00: +3300-3*T
      L(FCC,A,B;0) =    500.00<T< 2000.00: +V19+V20*T
      L(FCC,A,B;1) =    500.00<T< 2000.00: +V21+V22*T

=====  BLOCK NUMBER  1

DEFINED CONSTANTS

      DX=2E-2, P0=101325, DH=500, DT=10

DEFINED FUNCTIONS AND VARIABLES%

      HTR=HM(LIQUID)-HM(A2B)

Alternate equilibrium calculation                0.4183
Alternate equilibrium calculation                0.1932
Alternate equilibrium calculation                0.1016
Alternate equilibrium calculation                1.4354E-03
Alternate equilibrium calculation                2.5063E-02
Alternate equilibrium calculation                8.3929E-03
Alternate equilibrium calculation                141.4
Alternate equilibrium calculation                141.4
Alternate equilibrium calculation                141.4
Alternate equilibrium calculation                141.4
Alternate equilibrium calculation                1.2532E-03
Alternate equilibrium calculation                5.4781E-04
Alternate equilibrium calculation                1.5404E-03
Alternate equilibrium calculation                1.2702E-03
ACR(B)=9.4E-1                0.9397        2.85E-02  -2.7745E-04  -9.7472E-03
ACR(B)=8.4E-1                0.8395        2.82E-02  -4.9038E-04  -1.7396E-02
ACR(B)=7.4E-1                0.7407        2.81E-02   7.3804E-04   2.6305E-02
ACR(B)=6.4E-1                0.6424        2.81E-02   2.3935E-03   8.5272E-02
ACR(B)=5.4E-1                0.5434        2.82E-02   3.4449E-03   0.1220
ACR(B)=4.4E-1                0.4428        2.85E-02   2.8265E-03   9.9024E-02
ACR(B)=3.4E-1                0.3394        2.90E-02  -5.8174E-04  -2.0040E-02
ACR(B)=2.3E-1                0.2320        2.97E-02   2.0260E-03   6.8208E-02
ACR(B)=1.2E-1                0.1194        3.06E-02  -6.4192E-04  -2.0981E-02
HMR(LIQUID)=-1964            -1957.        5.00E+02   7.439        1.4878E-02
HMR(LIQUID)=-3500            -3478.        5.00E+02   21.67        4.3338E-02

```

---

HMR (LIQUID)=-4588	-4565.	5.00E+02	22.69	4.5382E-02	
HMR (LIQUID)=-5239	-5217.	5.00E+02	21.50	4.3008E-02	
HMR (LIQUID)=-5454	-5435.	5.00E+02	19.11	3.8216E-02	
HMR (LIQUID)=-5233	-5217.	5.00E+02	15.50	3.1008E-02	
HMR (LIQUID)=-4575	-4565.	5.00E+02	9.691	1.9382E-02	
HMR (LIQUID)=-3481	-3478.	5.00E+02	2.669	5.3385E-03	118
HMR (LIQUID)=-1950	-1957.	5.00E+02	-6.561	-1.3122E-02	

## **LIST\_STORE\_FILE**

 This command is for the PARROT module.

Use the LIST\_STORE\_FILE command to display the name of the store file and its full path.

## LIST\_SYMBOL\_IN\_GES

 This command is for the PARROT module.

Lists TP-function(s) for the entered model parameters for phases in the system on screen. In many cases, the optimizing variables are parts of the TP-functions which in turn are entered in model parameters for various phases in the GIBBS workspace.

This command is a way to find out how the functions depend on the optimizing variables. In the PARROT module the list is only shown on screen, not written to any file (which the LIST\_SYMBOL command in the GIBBS module does).

Syntax	LIST_SYMBOL_IN_GES
Prompt	Name: <Symbol name> Specify a symbol name. <div style="border: 1px solid blue; padding: 5px;"> Only those symbols that match this name are listed. Or press &lt;Enter&gt; for a list of all available symbols entered in the system.</div>

## OPTIMIZE\_VARIABLES

 This command is for the PARROT module.

Perform variable optimization. All system-definition data needed for the optimization is read from the current work file (\*.PAR). The result of the optimization is automatically stored in the current work file.

Syntax	OPTIMIZE_VARIABLES
Prompt	Number of iteration /N/: <Integral number of iteration> Specify an integral number for the optimization iteration. The specified iteration number is shown as the default number. The PARROT module tries this number of different sets of values of the optimizing variables unless it has converged earlier, or has given up earlier.
	<div style="border: 1px solid blue; padding: 10px;"> It initially takes a small step in each variable to find the steepest slope. You can give zero iteration to calculate the error in all selected experiments, and then use LIST_RESULT to check how good (bad) the current fit is.</div>

## READ\_PARROT\_WORKSPACES

 This command is for the PARROT module.

 [Workspace Files](#)

Read the previous PARROT/GIBBS/POLY workspaces back to replace the current PARROT/GIBBS/POLY workspaces. It is useful if late changes made through various PARROT/GIBBS/POLY commands are not satisfactory.

Syntax	READ_PARROT_WORKSPACES
	The previous PARROT/GIBBS/POLY workspaces are always associated with the latest action to either open a work file by SET_STORE_FILE, or to create a work file by CREATE_NEW_STORE_FILE (either interactively in the PARROT module or through a MACRO_FILE_OPEN of an *SETUP.TCM file), or to update the work file by SAVE_PARROT_WORKSPACES.
	Unlike the READ commands in other modules (e.g. GIBBS or PLOY), this command in the PARROT module does not ask for the file name where to read a previously opened/created/updated PARROT/GIBBS/POLY workspaces, for the reason described above.
	 You cannot use this command if there is no work file opened or created already.

## **RECOVER\_VARIABLES**

The RECOVER\_VARIABLES command is used to set back the values of all variable to the start values.

---

## REINITIATE



This command should not be used unless you want to destroy the current PARROT workspace. However, this PARROT command does not reinitiate the GIBBS/POLY3 workspaces.



This command is for the PARROT module.

The REINITIATE command is used to reinitiate the workspace used by PARROT. All output and optimizing conditions are given their default values. All variables are set fixed with their value equal to zero.

## **RESCALE\_VARIABLES**

When using the RESCALE\_VARIABLES command, the current values of all the parameters are copied to their start values and the scaling factors for further optimization. Thus it should be done now and again, in particular if you think the optimization results are improved after the previous run(s), or if any variable changes more than a factor of 10.

## SAVE\_PARROT\_WORKSPACES

 This command is for the PARROT module.

If the latest changes made through various PARROT/GIBBS/POLY commands are as required, use this command to save the current workspace (i.e. data area) used by the PARROT program (also including the current GIBBS and POLY workspaces), onto the present work \*.PAR file which has already been opened by SET\_STORE\_FILE or been created by CREATE\_NEW\_STORE\_FILE (either interactively in the PARROT module or through a MACRO\_FILE\_OPEN of an \*SETUP.TCM file).



The PARROT/GIBBS/POLY workspaces are updated after each PARROT/GIBBS/POLY command. This means the current PARROT/GIBBS/POLY workspaces are always updated onto the latest work \*.PAR file that is associated with the latest SET\_STORE\_FILE or CREATE\_NEW\_STORE\_FILE command.



### Workspace Files

Syntax

SAVE\_PARROT\_WORKSPACE



Unlike the SAVE commands in other modules (e.g. GIBBS or PLOY), this command in the PARROT module does not ask for the file name where to save the current PARROT/GIBBS/POLY workspaces, for the reason described above. You cannot use this command if there is no work \*.PAR file opened or created already.

## SET\_ALTERNATE\_MODE

Turn the alternate mode on or off.

With the ALTERNATE mode is possible to include functions to be evaluated together with an alternate calculation and to select ALTERNATE mode for each experimental equilibrium.



The alternate mode is described in [About Alternate Mode](#). It should be used only to optimize start values of the model parameters in the beginning of the assessment.

Syntax	SET_ALTERNATE_MODE
Prompt	On /Y/ : By default alternate mode is ON.

## SET\_EXTERNAL\_PARAMETER

 This command is for the PARROT module.

Optimize model parameters of the so-called external models, which may not be as fully-implemented/integrated parts inside the GIBBS system and thus are independently defined within user-specified/written source codes. This command makes links between the PARROT optimizing variables and external model parameters.



In order to be able to build the specified external model and to use this SET\_EXTERNAL\_PARAMETER command, it requires you to first write codes for a preferred external model (as it is not within the standard GIBBS system of the Thermo-Calc software) and then provide it to a consultant at Thermo-Calc Software AB for the purpose of including the external model in a separate DLL that is interactively connected to the software. For more information, see the [Consultancy](#) page on our website.

Syntax	SET_EXTERNAL_PARAMETER
Prompt	External name: <Parameter name in the external model> Correctly specify the parameter name defined within a specified/written external model.
	As optimizing variable number: <n> Give the number of the PARROT optimizing variable, which should have a unique link to the specified parameter name of the external model. For example, @@ Link PARROT variables to CAM parameters: <pre>SET-EXTER-PAR GT (SLAG_A, O-2, SI+4, CA+2) 1 SET-EXTER-PAR ET (SLAG_A, O-2, SI+4, CA+2) 2 SET-EXTER-PAR EXT (SLAG_A, O-2, SI+4, CA+2) 3</pre>

## SET\_FIX\_VARIABLE

Prescribe a fixed value to a variable. The variable is considered as a constant at the optimization, usually after successful optimization runs for some specific variables (for some of the phases in the system). Such fixed variables can also be set back for further optimization run by using SET\_OPTIMIZING\_VARIABLE.

Syntax	SET_FIX_VARIABLE
Prompts	<p>Variable number: &lt;Variable number(s)&gt;</p> <p>Specify the number of the variable. It is possible to give a range by giving two numbers with a hyphen in between (no spaces allowed), e.g. 1. The parameters within the limits are set fixed to their current values (no question of values).</p>
	<p>Start value /xxxx.xxxxx/: &lt;A value to be fixed&gt;</p> <p>Specify a numerical value to be fixed for the selected variable (only when one variable number is specified at the previous prompt). The current value for the selected variable is shown as default.</p> <p>If a range of variable numbers is specified at the previous prompt, this question is not prompted, and the current values are used as fixed values.</p>

## SET\_OPTIMIZING\_CONDITION

 This command is for the PARROT module.

Specify the conditions for the optimization. The default values are chosen automatically by the PARROT program, and in most cases these should not be changed.



The optimization stops when the sum of errors decreases by this value, even though it might be possible to increase it further with a new OPTIMIZE command.

Syntax	SET_OPTIMIZING_CONDITION
	<p>Relative standard deviation for experiments? /N/:</p> <p>By default the standard deviation of the experimental determinations are absolute values. Enter Y if these are used as relative weighting factors.</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;"> <p> The estimated standard deviations of the optimized variables might be different for the two cases.</p> </div>
Prompt	<p>Min save on file? /Y/:</p> <p>Specify whether the program should minimize the transfer of data to and from the current work file during the optimization:</p> <ul style="list-style-type: none"> <li>• If the default (N) is kept, the workspaces are stored on the current work file after every iteration in the optimization procedure.</li> <li>• Enter Y to speed up the optimization. After optimization use the SAVE_PARROT_WORKSPACES command to update the progress onto the current work file.</li> </ul>
	<p>Relative step for calculation of derivatives: /1e-04/: &lt;xxx&gt;</p> <p>In the calculation of the correlation matrix for equilibria with inaccuracy in the independent state variables, some numerical derivatives might have to be calculated.</p> <p>Specify the relative step (xxx) for the calculation of these derivatives.</p>
	<p>Maxfun (va05ad): /100/: &lt;n&gt;</p> <p>The maximum number of iterations in the optimization. The same value set by the OPTIMIZE command.</p>
	<p>Dmax (va05ad): /100/: &lt;n&gt;</p> <p>An estimate of the maximum distance between the start and the final values of the variables. A smaller value makes the program vary the parameters with smaller factors.</p>

---

Syntax	SET_OPTIMIZING_CONDITION
	<pre>H (va05ad): /1e-04/: &lt;xxx&gt;</pre> <p>The step used in the scaled variables for calculating numerical derivatives during the optimization.</p>
	<pre>Acc/(initial sum of squares) (va05ad): /.001/: &lt;xxx&gt;</pre> <p>The break condition for the optimization. The accepted value is the difference between the true minimum and the calculated one.</p>

## SET\_OPTIMIZING\_VARIABLE

 This command is for the PARROT module.

Specify which variable value should be estimated at the optimization. Such a variable can have a value of zero or another start value, or is fixed by SET\_FIX\_VARIABLE prior to the previous optimization run.

For a good reference prior to this command, a list of all variables (which are used to define various parameters for phases in the optimizing system, as pre-entered in the \*SETUP.TCM file or interactively entered in the GIBBS module) are obtained by using LIST\_ALL\_VARIABLES.

Syntax	SET_OPTIMIZING_VARIABLE
Prompts	<p>Variable number: &lt;Variable number(s)&gt;</p> <p>Specify the number(s) for the variable(s) to be set. Any non-zero parameter within the range is allowed to be optimized. A parameter with a current value equal to zero must be specified explicitly here in order to be optimized.</p> <p>Specify a range by giving two numbers connected by a hyphen (no space allowed!), e.g. 2-8. Under this circumstance, no question of start values are asked.</p>
	<p>Start value /xxxx.xxxxx/: &lt;A guess value&gt;</p> <p>Specify a start guess of the optimum value (only when one variable number is specified at the previous prompt). The current numerical value for the selected variable is shown as default.</p> <div style="border: 2px solid red; padding: 10px; margin-top: 10px;"> <p> This guess is critical as the initial guess of all parameters must make it possible to calculate the selected equilibria. If a range of variable numbers is specified at the previous prompt, this question is not prompted, and the current values (as start values) are not changed.</p> </div>

## SET\_OUTPUT\_LEVELS

 This command is for the PARROT module.

Choose the type of information the PARROT module gives during the optimization procedure and when listing results.

Syntax	SET_OUTPUT_LEVELS
Prompts	<p>List increment /1/: &lt;Increment in iteration number&gt;</p> <p>Specify the increment in iteration number for which information is listed on screen during the optimization. By giving a larger number it shortens the list.</p>
	<p>List sum of squares: /Y/:</p> <p>Choose whether the sum of squares is listed on screen during the optimization procedure.</p>
	<p>List scaled variables: /Y/:</p> <p>Choose whether the scaled variable values are listed on the terminal during the optimization procedure.</p>
	<p>List weighted residuals: /N/:</p> <p>Choose whether the weighted residuals are listed on the terminal during the optimization procedure.</p>
	<p>List all parameters: /N/:</p> <p>Choose whether all parameters in the models are listed in LIST_RESULT.</p>
	<p>List correlation matrix: /N/:</p> <p>Choose whether the correlation matrix of the variables are listed with the commands LIST_RESULT and LIST_ALL_VARIABLES.</p>

## SET\_SCALED\_VARIABLE

This command is similar to SET\_OPTIMIZING\_VARIABLE in that it specifies start values for optimizing variables. The difference being it is functional for only one optimizing variable at one time. It also prescribes a minimum and maximum value for the variable. During the optimization runs, the variable value is limited within this min-max range.

Such a variable may have a value of zero or another start value, or may be fixed by SET\_FIX\_VARIABLE prior to the previous optimization run. For a good reference prior to using this command, a list of all variables (which are used to define various parameters for phases in the optimizing system, as entered in the \*SETUP.TCM file or interactively entered in the GIBBS module) is obtained by using LIST\_ALL\_VARIABLES.

Syntax	SET_SCALED_VARIABLE
Prompts	<p>Variable number: &lt;Variable number(s)&gt;</p> <p>Specify the number for an optimizing variable to be set. Any non-zero parameter is allowed to be optimized, and a parameter with a current value equal to zero must be specified explicitly here in order to be optimized.</p>
	<p>Start value /xxxx.xxxxx/: &lt;A guess value&gt;</p> <p>Specify a start guess of the optimum value. The current numerical value for the selected variable is shown as default.</p> <div style="border: 2px solid red; padding: 5px; margin-top: 10px;">  This guess is critical as the initial guess of all parameters must make it possible to calculate the selected equilibria.         </div>
	<p>Min value /xxxx.xxxxx/: &lt;A guess value&gt;</p> <p>Specify a minimum guess of the optimum value. It should be smaller than, at least equal to, the current start value for the selected variable.</p>
	<p>Max value /xxxx.xxxxx/: &lt;A guess value&gt;</p> <p>Specify a maximum guess of the optimum value. It should be larger than, at least equal to, the current start value for the selected variable.</p>

## SET\_STORE\_FILE

 This command is for the PARROT module.

Specify a store file (work file) to be used for compilation and optimization. The \*.PAR work file must be created (CREATE\_NEW\_STORE\_FILE) as a store file. The GIBBS, POLY and PARROT workspaces are read from the specified store file (with a default extension of PAR).

Syntax	SET_STORE_FILE
Prompt	Enter a *.PAR file name in the <b>File name</b> field and specify the working directory in the <b>Look in</b> field. Normally the default file-type in the <b>Files of type</b> field is the correct one for the PARROT workspace format (i.e. *.PAR file). Click <b>Open</b> or <b>Cancel</b> as applicable.

## EDIT\_EXPERIMENTS (ED\_EXP) Submodule Commands



EDIT\_EXPERIMENTS (ED-EXP) is a submodule of PARROT.

### In this section:

ADVANCED_OPTIONS .....	404
CHANGE_STATUS .....	405
COMMENT .....	410
COMPUTE_ALL_EQUILIBRIA .....	411
CREATE_NEW_EQUILIBRIUM .....	412
DEFINE_COMPONENTS .....	413
DELETE_SYMBOL .....	414
ENTER_SYMBOL .....	415
EVALUATE_FUNCTIONS .....	418
EXPERIMENT .....	419
EXPORT .....	421
FLUSH_BUFFER .....	422
GRAPHICS_PLOT .....	423
IMPORT .....	424
LABEL_DATA .....	425
LIST_ALL_EQUILIBRIA .....	426
LIST_CONDITIONS .....	427
LIST_EQUILIBRIUM .....	428
LIST_STATUS .....	429
LIST_SYMBOLS .....	430
MAKE_POP_FILE .....	431

---

READ_WORKSPACES .....	432
REINITIATE_MODULE .....	433
RESTORE_ALL_WEIGHTS .....	434
SAVE_WORKSPACES .....	435
SELECT_EQUILIBRIUM .....	436
SET_ALL_START_VALUES .....	437
SET_ALTERNATE_CONDITION .....	439
SET_CONDITION .....	442
SET_NUMERICAL_LIMITS .....	445
SET_REFERENCE_STATE .....	447
SET_START_CONSTITUTION .....	449
SET_START_VALUE .....	450
SET_WEIGHT .....	451
SHOW_VALUE .....	453
STORE_ALL_WEIGHTS .....	454
TABLE_HEAD, TABLE_VALUES and TABLE_END .....	455
TRANSFER_START_VALUES .....	457



---

## ADVANCED\_OPTIONS



This command is available with the POLY and ED-EXP modules.



[ADVANCED\\_OPTIONS](#) described for the POLY module.

Syntax	ADVANCED_OPTIONS
Prompt	Which option? /Step_and_Map/: <Option>

## CHANGE\_STATUS



This command is available with the POLY and ED-EXP modules.

In the POLY module, set the status for components, species and phases in the defined system for all the sequential calculations (single-point, stepping, and mapping) in equilibrium or local/partial equilibrium state. Each component, species and phase has a status. The default status is `ENTERED`.

The most important use is to calculate metastable equilibria and metastable phase diagrams by setting some phases (that would otherwise be stable) to the `SUSPENDED` or `DORMANT` phase-status. Another important applications is to calculate paraequilibria by setting some components to the `SPECIAL` component-status.

For a component and for a species, the status can be one of the following:

- `ENTERED`: the component(s) or species are included in the calculation. This is the default status.
- `SUSPENDED`: the component(s) or species are not considered in the calculation.
- `SPECIAL`: the specified component(s) are not included in summations for mole or mass fractions. It only works for component(s).



Only component(s) can have the status `SPECIAL`, which implies that these are not included in summations for mole or mass fractions.

For example, for the *u-fractions* or other normalized fractions, when one or more of the components are excluded from the summation, you must specify which component(s) should be excluded from the calculation of mole or mass fraction. This component status is particularly useful when calculating paraequilibrium states. Such component(s) are normally interstitial component, and must have the status `SPECIAL`. This is assigned by the `CHANGE_STATUS` command.

Syntax	CHANGE_STATUS
Prompts	<p>For phases, species or components? /Phases/: &lt;Keyword&gt;</p> <p><i>Keyword</i> = phase or species or components</p> <p>Phase name(s): &lt;Name(s) of the phase(s)&gt;</p> <p>For <i>phase</i> as the keyword, the names of the phases that have their status changed must be given (all on one line). A comma or space must be used as separator. The status to be assigned to the phases can also be given on the same line if preceded with an equal sign (=).</p>

Syntax	CHANGE_STATUS
	 <p>An asterisk, *, can be used to denote all phases. The special notations *S, i.e. a wildcard * directly followed by an S, sign, means all suspended phases. In the same way, *D means all dormant phases, and *E means all entered phases.</p>
	<p>Name(s): &lt;Name(s) of the Species or Component(s)&gt;</p> <p>For species or components as the keyword, the names of the species or components that have their status changed must be given (all on one line). A comma or space must be used as separator. Similarly to the case of phase as the keyword, the status to be assigned to the species or components can also be given on the same line if preceded with an equal sign =.</p>  <p>An asterisk, *, can be used to denote all species or components. The special notations *S, i.e. a wildcard * directly followed by an S, sign, means all suspended species or components. In the same way, *E means all entered species or components.</p>
	<p>Status /Entered/: &lt;New status&gt;</p> <p>The new status to be assigned must be given.</p> <p>For species, the values ENTERED or SUSPENDED can be used.</p>  <p>For the POLY module, using the Entered or Suspended status here has the same result as using the commands <a href="#">MAKE_COMPONENT_ENTERED</a> and <a href="#">MAKE_COMPONENT_SUSPENDED</a>. The exception is that any existing conditions that depend on a suspended component are removed when using the MAKE_COMPONENT_SUSPENDED command.</p> <p>For components, the status ENTERED, SUSPENDED or SPECIAL can be given. SPECIAL means that this component is excluded from sums for mole fractions and mass fractions, which is useful when calculating the <i>ufractions</i> or other normalized fractions of system components.</p> <p>For phases, the status ENTERED, SUSPENDED, DORMANT or FIXED can be given. DORMANT means the same as suspended but the driving force is calculated. FIXED means that it is a condition that the phase is stable at a certain amount.</p> <p>For example, for the <i>ufractions</i>, when one or more of the components are excluded from the summation, you must specify which component should be excluded from the calculation of the mole fraction. This component must have the status SPECIAL. This is assigned by the CHANGE_STATUS command: <code>Change_Status comp C=special.</code></p>
	<p>Start value, number of mole formula units /0/: &lt;Initial amount&gt;</p> <p>For ENTERED phases, an initial amount of the phase can be given. Normally, 0 is given if the phase is not likely to be stable, and 0.5 or 1 or any positive number if the phase could be stable, but such an initial amount is only used as the rough starting estimation in the equilibrium calculations.</p>
	<p>Number of mole formula units /0/: &lt;Equilibrium amount&gt;</p>

Syntax

## CHANGE\_STATUS

For FIXED phases, the *equilibrium amount* of the phase [always using an initial estimation being the  $NPF(\text{phase})$  value which it is the normalized mole number of components (per mole formula unit) of the specific status-fixed phase] must be given. If the equilibrium amount is zero, then the phase is at its stability limit.

### Note when specifying a FIXED phase status

Special attention should be paid when specifying a FIXED phase status in equilibrium calculations (for single points, stepping or mapping calculations), as described below.

The phase amount variables,  $NP(\text{phase})$ ,  $BP(\text{phase})$  and  $VP(\text{phase})$ , as well as all their M/W/V-suffixed quantities, should not be used as conditions. Instead, use the CHANGE\_STATUS command to set a relevant condition, e.g. `CHANGE_STATUS phase <phase>=fix <amount>` where the fixed <amount> is roughly the same as the F-suffixed quantity  $NPF(\text{phase})$ .

The  $NPF(\text{phase})$  quantity is the normalized mole number of components (per mole formula unit) of the specific phase in the defined system, which unlike other F-suffixed state variables [e.g.  $GF(\text{phase})$ ,  $HF(\text{phase})$  and  $DGF(\text{phase})$ ] cannot be directly applied in any POLY command, implying that it cannot be directly evaluated or listed/shown. If intended to be shown such a normalized phase amount value in an equilibrium state, you should use a properly-entered symbol (function or variable), for example:  $NPF_{abc} = NP(abc)/NA$  or  $NPF_{abc} = NPM(abc)/NA*N$ .  $N$  is the total system size (in mole). The  $NA$  value is a quantity that is phase-dependent (and sometimes also equilibrium-dependent for ionic solution phases), and is the total atomic number in a mole-formula-unit of the specific phase  $abc$  (excluding interstitial component and, of course, vacancy). For example, the SIGMA, FCC, BCC and LIQUID phases (among others) in a defined Fe-Cr-Ni-C-N-O system (retrieved from a specific database) may be modeled by certain models, and their  $NA$  values must be evaluated in different ways, as described below:

```
LIQUID (C,Cr,CrO3/2,Fe,FeO,FeO3/2,N,Ni,NiO)1 NA = 1
FCC_A1 (Cr,Fe,Ni)1(Va,C,N,O)1 NA = 1
BCC_A2 (Cr,Fe,Ni)1(Va,C,N,O)3 NA = 1
SIGMA (Fe,Ni)8(Cr)4(Cr,Fe,Ni)18 NA = 30
```

If in the same Fe-Cr-Ni-C-N-O system the liquid solution phase is modeled by the Two-Sublattice Ionic Liquid Model, i.e.:

```
IONIC_LIQ (Cr+3,Fe+2,Ni+2)p(VA,C,N,O-2,FeO3/2)q,7
```

then the evaluation of its  $NA$  value becomes even more complicated:

$$NA = p + q*y_{C2} + q*y_{N2} + q*y_{O2e} + q*y_{FeO2} / 2$$

where the stoichiometric coefficients  $p$  and  $q$  are also dependent upon the real equilibrium state (rather than having fixed values in the system). Similar situations occur for other (solid) phases which are described by a multiple sublattice model with ionic constituents, such as SPINEL and HALITE phases in some databases.

There is no strange thing when using a zero value [i.e. 0] in a FIXED phase-status, since it means the specified phase is stable in equilibrium state but has a zero-amount of mass in the equilibrium calculations; in other words, on a phase diagram, the specific phase is on a zero-fraction line (ZFL), i.e. it starts becoming stable on one side of a corresponding phase-boundary line or unstable on the other side of the same boundary. It is often and efficient to do so when calculating e.g. solidus equilibrium states.

However, when a non-zero value [it must always be positive; e.g. 1 or 0.5 or 0.3 or 1.5] is to be specified in a FIXED phase-status, it is unnecessarily the exactly same stable amount of the specific

Syntax	CHANGE_STATUS
	<p>FIXED-status phase in a calculated equilibrium state any longer; instead, the &lt;equilibrium amount&gt; value is the NPF(phase) value that is only roughly used as the estimated starting-value of the FIXED-status phase in the equilibrium calculations.</p> <p>Therefore, a FIXED-status for a liquid phase being unity does not necessarily imply that it is a liquidus equilibrium state (where the liquid phase is in equilibrium with some solid phases but the liquid phase takes all the mass in the defined system). A unity value for setting the liquid phase status in calculating liquidus equilibrium state can only be used when the liquid mixture phase is predefined as a single-sublattice solution phase (such as metallic liquid phase in multicomponent alloy systems) and the total system size as one mole (i.e. N=1).</p> <p>When a phase is described by a solution model in which two or more sublattices are considered and these sublattice sites may also have different stoichiometric coefficients [meaning that the mixture phase could have more than one atom in formula [NA&gt;1; see some examples above], the unity value should not be used when setting the FIXED status for the phase; instead, you should use an appropriate value that ranges from 0 to a NPF(phase) value that equals to or is smaller than 1/NA (if the total system size N=1) or 1/NA*N (if N differs from unity). For this reason, if a multicomponent system bears an IONIC_LIQUID phase that is described by the Two-Sublattice Ionic Liquid Model (or any other multiple-sublattice ionic solution phases), it is difficult to use a proper NPF(ION_LIQ) value in setting its FIXED phase-status, because that should be less than (or equal to) the complex value of</p> $N/[p + q*y_{C2} + q*y_{N2} + q*y_{O22} + q*y_{FeO2}^{3/2}].$

## Examples

For example, to obtain the metallic fraction in a system with carbon as an interstitial component, you can set the component status for carbon as `SPECIAL`:

```
Change_Status comp C=special
```



The `SUSPENDED` status for components and species does not always work as expected.

For a phase, it may have one of these statuses:

- **ENTERED**: the phase(s) are included in the equilibrium calculations and these are stable if that minimizes the total Gibbs energy in the defined system. This is the default status for all phases already retrieved from the chosen database(s). An ENTERED phase-status is always associated with an initially-estimated amount [in mole number; normally, as 0 if the phase is not likely to be stable, and as 0.5 or 1 or any positive number if the phase could be stable] but it is only used as the rough starting value in the equilibrium calculations.

- **SUSPENDED:** the phase(s) are not considered in the equilibrium calculations.
- **DORMANT:** the phase(s) are not considered in the equilibrium calculations but their driving forces for precipitation are calculated.
- **FIXED:** it is an equilibrium condition that the status-fixed phase must be stable, and be in equilibrium at a specified amount [always using an initial estimation being the  $NPF(\text{phase})$  value which it is the normalized mole number of components (per mole formula unit) of the specific status-fixed phase]. See more descriptions at the end of this command.

---

## COMMENT



This command is for the ED-EXP module.

The COMMENT command is only used in the \*.POP/\*.DOP files (so can be included with experimental data files) to write brief comments or descriptive information on a newly created equilibrium (experimental) data point or a set of points given in a table.

A comment can have a maximum of about 60 characters written on the same line.

## COMPUTE\_ALL\_EQUILIBRIA



This command is for the ED-EXP module.

In ED\_EXP and PARROT modules, each experiment is treated as an individual equilibrium with some measured values. These are created with CREATE\_NEW\_EQUILIBRIUM, and stored in an experimental data \*.POP file and then compiled and saved in the POLY workspace of a PARROT work \*.PAR file.



In the DICTRA module, you work with a .DOP file instead of a .POP file.

With COMPUTE-ALL-EQUILIBRIA, all equilibria from the current to the last experimental points are calculated. If an equilibrium calculation fails, the calculation is stopped at that equilibrium. Equilibria with weight zero is skipped.



A current or present experimental point means the latest selected or calculated one. Once used the current point turns to the last point in the data block; in such a case you first use SELECT\_EQUILIBRIUM so that the current point is switched to a desired one.

For this command there is always a list output on screen, which consists of six columns for all experimental points available in the current data block:

- The first column is the *equilibrium identifier* (a number) assigned by this command,
- the second the *data label* assigned with LABEL\_DATA,
- the third the *number of iterations*,
- the fourth the *current weight*, and
- the fifth the *current temperature*.
- In the sixth column the *fixed stable phases* are listed together with any *comment text* given after a COMMENT in the \*.POP or \*.DOP file.

If the weight is zero for an equilibrium columns 3-5 are replaced by the text <unused>. If the alternate mode is used for some experimental points, the listing is slightly different for such points. The 3-4 columns are displayed with \*alt\*, instead. If an alternate calculation is failed at one experimental point, the point is automatically assigned with a zero weight, and a warning message is shown above the data line (with all six columns).

## CREATE\_NEW\_EQUILIBRIUM

 This command is available with the POLY and ED-EXP modules.

During data-assessments using the ED\_EXP module (a submodule of PARROT), you can, in the POLY module, create several equilibria with different sets of conditions and phases (but normally with the same set of components). By default, there is one equilibrium. To keep the set of conditions and phase for this equilibrium, create another one using this command, and use another set of conditions for that. Two equilibria may be useful to calculate easily the enthalpy difference between two states. In the PARROT module, the experimental information is stored as a sequence of equilibria.

Syntax	CREATE_NEW_EQUILIBRIUM
Prompts	Equilibrium number /2/: <A new equilibrium number>  Each equilibrium in the POLY workspace is identified by a unique integer number. Such an equilibrium number can be recalled with SELECT_EQUILIBRIUM.
	Initiation code /2/:  When an equilibrium is created, you can choose to <b>ENTER</b> all components and phases (initiation code 2), <b>ENTER</b> the components only (initiation code 1) or <b>SUSPEND</b> everything (initiation code 0). No other values are legal.
	 The entered components and phases can later be changed with CHANGE_STATUS.

## DEFINE\_COMPONENTS

 This command is available with the POLY and ED-EXP modules.

Change the set of components. By default, the elements are used as components. The set of components can be important because some conditions are set using components, for example, the amounts, activities or chemical potentials.

For example, in the system Fe-Si-O, you can define FEO, FE2O3 and SIO2 as components, thus replacing the default FE, SI and O.



This implies a command REINITIATE\_MODULE and it should be given as the first command in the POLY module.

Syntax	DEFINE_COMPONENTS
Prompt	<p>Give all new components / existing components/: &lt;New components&gt;</p> <p>The <i>new components</i> must be specified all on one line. These replace the existing component definitions.</p> <div data-bbox="370 1045 418 1108" data-label="Image"> </div> <p>The number of components cannot be changed with this command. Use CHANGE_STATUS instead.</p>
	<p>To keep some existing components, it is recommended that you also enter these on the line. Otherwise, the added new components may not be defined correctly. This is especially important if some components are built out of several elements.</p> <div data-bbox="370 1318 418 1381" data-label="Image"> </div> <p>The new components do not have to be present as species.</p>

## DELETE\_SYMBOL



This command is available with the POLY and ED-EXP modules.

Use the DELETE\_SYMBOL command to remove symbols i.e. constants, variables, functions or tables.

Syntax	DELETE_SYMBOL
Prompt	Name: <Name of a symbol> Specify the name of the symbol to be deleted. Only one symbol can be deleted each time.

## ENTER\_SYMBOL

-  This command is for the ED-EXP module.
-  The GES, POLY and POST modules also have a command with the same name.

Symbols are a useful feature modules to define quantities that are convenient. Symbols can be constants, variables, functions, or tables.

Syntax	ENTER_SYMBOL
Prompt	<p>Constant, Variable, Function or Table? /Function/: &lt;Keyword&gt;</p> <p>The <b>Keyword</b> can be specified as <b>CONSTANT, VARIABLE, FUNCTION, or TABLE.</b></p> <ul style="list-style-type: none"> <li><b>CONSTANTS</b> can only be entered once and is a means of using a name for a numeric value. For example, the value of 1 atmosphere in Pascal can be denoted by <code>P0</code> after the command <code>ENTER_CONSTANT P0=101325</code>. Defined constants can be used as values in condition assignments, for example, <code>SET-COND P=P0</code>.</li> <li><b>FUNCTIONS</b> are expressions of state variables or other functions. These expressions are saved, and whenever a function value is requested all functions are evaluated. The reason for this is that they may depend on each other.</li> <li><b>VARIABLES</b> are similar to functions because they can also be expressions of state variables. However, contrary to functions, they are only evaluated when they are entered or if they are explicitly named in an <code>EVALUATE_FUNCTIONS</code> command. It is possible to enter a variable with a new expression any time. This expression is evaluated directly and its value stored as the value of the variable. Defined variables can be used as values in the <code>SET_CONDITION</code> command.</li> <li><b>TABLES</b> are used for listing results from the <code>STEP</code> or <code>MAP</code> commands. A table consists of a list of any number of state variables, functions, or variables. Defined tables can also be used in the <code>POST</code> (post-processor) module.</li> </ul> <div style="border: 2px solid #00AEEF; padding: 10px; margin-top: 10px;"> <p> There is a special connection between tables and variables. If a variable is used in a table, it is evaluated for each line of the table in the <code>TABULATE</code> command or when the table is used in a plot.</p> </div>
	<p>Name: &lt;Name of the symbol&gt;</p> <div style="border: 2px solid black; padding: 10px; margin-top: 10px;"> <p> Each symbol has a unique name that must start with a letter and can have maximum 8 characters. Legal characters include letters (either UPPER or lower case), digits and underscore <code>_</code>. Any other special character, such as parentheses ( and ), plus <code>+</code>, minus <code>-</code>, slash <code>/</code> or <code>\</code>, full stop <code>.</code>, are illegal for symbol names.</p> </div> <p>You can enter the symbol name and the value or function on the same line; these must be separated</p>

Syntax	ENTER_SYMBOL
	<p>with an equal sign =, for example, <code>TC=T-273.15</code> or <code>T_C=T273.15</code> which stands for a definition of temperature in Celsius. Otherwise, these questions are asked.</p> <div style="border: 2px solid blue; padding: 10px; margin: 10px 0;">  For different types of symbols (constant, function, variable, or table), the questions have different prompts. </div>
	<p>Function: &lt;Definition for a function or variable&gt;</p> <p><i>Functions</i> and <i>variables</i> are evaluated from an expression of state variables or other functions, constants, or variables. The expression is a FORTRAN-like expression and operators +, -, *, = and ** can be used (** only with integer powers).</p> <p>Unary functions like LOG, LOG10, EXP, SIN, COS, and ERF can also be used. An expression can be continued on more than one line. An expression should be terminated by a semicolon (;) or an empty line (press &lt;Enter&gt; at the next prompt).</p> <p>Examples of function expressions:</p> <ul style="list-style-type: none"> <li>• GM(LIQUID): The Gibbs energy of liquid per mole component</li> <li>• H.T/4.184: The heat capacity of the system in calories</li> <li>• ACR(CR)/X(FCC,CR): The activity coefficient for Cr in FCC</li> <li>• T-273.15: The temperature in Celsius</li> </ul> <p>&amp;: &lt;Continuation of the definition for the symbol&gt;</p> <p>The ampersand &amp; means that you can continue to write the function on the new line if one line is not enough for the function. If you finish the function press &lt;Enter&gt; again.</p>
	<p>Value: &lt;Value for a constant&gt;</p> <p>A <i>constant</i> can only be assigned a numeric value once.</p>
	<p>Value or expression: &lt;Value of expression for a variable&gt;</p> <p>A <i>variable</i> can be assigned a numeric value or an expression. An expression is evaluated immediately and discarded. Only the numeric value is kept. This gives a possibility to save a value between calculations with different conditions because all state variables and functions are evaluated for the new conditions.</p>
	<p>Variable(s): &lt;Variable(s) in a table&gt;</p> <p>A <i>table</i> consists of a list of state variables or functions. One way to obtain results from a STEP command is through a table.</p> <p>Example:</p> <pre>ENTER TABLE K=T,X(LIQ,C),X(LIQ,CR),ACR(C)</pre> <p>Which means that the table called K contains four columns, i.e. the temperature, the mole fractions of C and Cr in the LIQUID phase, and the activity of C.</p> <p>To show the temperature in Celsius in a table, give the command ENTER_FUNCTION TC=T-273; and then use the symbol TC in the table.</p> <p>&amp; &lt;Continuation of the definition for the table&gt;</p>

Syntax

ENTER\_SYMBOL

The ampersand & means that you can continue to write the table on the new line if one line is not enough for the table. If you finish the table press <Enter> again.

## EVALUATE\_FUNCTIONS



This command is available with the POLY and ED-EXP modules.

The value of one or more or all entered functions or variables are evaluated and listed.

Syntax	EVALUATE_FUNCTIONS
Prompt	<p>Name(s): &lt;Name(s) of defined function(s)&gt;</p> <p>The names of one or more entered functions or variables must be specified. By typing a wildcard *, all functions and variables are evaluated.</p> <div data-bbox="354 625 1396 739" style="border: 1px solid blue; padding: 5px;"> Variables are evaluated only if these are explicitly named.</div>

## EXPERIMENT



This command is for the ED-EXP module.



This command is used in original experimental data files (\*.POP or \*.DOP), but can also be given interactively inside the ED-EXP module, to change the value or uncertainty of an experiment or to add more experimental information.



In the DICTRA module, you work with a .DOP file instead of a .POP file.

An experiment usually consists of two parts with a colon (:) separating them:

- as a quantity relation, and
- as the uncertainty of the value for the quantity,

The quantity relation can be a normal POLY module condition or an inequality (which is similar to condition but the relation between the quantity and given value is not in equality, i.e. < or >). The uncertainty can be expressed as an absolute value or relative to the value (x%).



An experiment that uses an inequality gives zero contribution to the sum of errors in the optimization procedure of PARROT if the value is on the right side. If the value is on the wrong side, the value after the colon determines how steeply the error increases with the value.

### Syntax

### EXPERIMENT

The experiment must be typed after the command. Several experiments can be given on the same line.

The syntax of EXPERIMENT is similar to a CONDITION. Usually, a state variable set equal to a value with a given uncertainty is specified after the colon (:). For example,  $T=1273.15:5$ .

An inequality, < or >, can also be used. For example:

$ACR(C) < -0.01:0.001$ ,  $W(BCC, AG) > 0.05:10\%$

## Additional Information About the Experiment Command

A typical experiment added interactively specifies that a phase should not be stable in a certain experiment, because a phase may appear in a range of composition or of temperature where it has never been observed during the optimization. A phase is unstable if its driving force is negative, and you can add experimental information to enforce that. For example, you can suppress the HCP phase in an existing experimental point:

```
CHANGE-STATUS HCP=DORMANT
EXPERIMENT DGM(HCP)<-.001:.0001
```



DGM is the POLY variable for the driving force that is already divided by  $R_T$ , and it is dimensionless. Only dormant phases can have driving forces larger than zero, as this indicates that these would be stable if allowed to form. The experiment in the above case tries to modify the model parameters to make the HCP phase unstable.

More general examples of experiment definitions:

```
EXPERIMENT X(LIQ,PB)=.10:.01 X(FCC,PB)=0.02:.01
EXPERIMENT ACR(PB)=0.8:5%
EXPERIMENT T=700:5
```

The first experiment above describes a tie-line where the experimentally measured mole fraction of Pb is 0.1 in the liquid phase and 0.02 in the FCC phase, and its uncertainty is 0.01 for both measurements. The second experiment is that the activity of Pb should be 0.8 with an uncertainty of 5 percent.



The reference state of the component Pb must be set with the command SET\_REFERENCE\_STATE. The final one is that the temperature should be 700 K with an uncertainty of 5 degrees.

Experiments that are functions of states variable(s) must be described as the defined functions. For example:

```
ENTER_FUNCTION HTR=HM(LIQUID)-HM(FCC);
EXPERIMENT HTR=7000:1000
```

## EXPORT

 This command is for the ED-EXP module.

Transfer/export a calculated value from an equilibrium to an optimizing variable in the PARROT workspace. Thus the value can be used, for example, in the Gibbs energy description of a phase. This variable should not be optimized (i.e. it should be set as a fixed variable in PARROT).

Syntax	EXPORT
Prompt	<p>Function name: &lt;Function name&gt;#&lt;n&gt;</p> <p>Specify the name of the function, the value of which should be transferred to a <math>v</math> variable. The number (n) of the <math>v</math> variable must be given after the function name, separated by hash character #.</p> <div style="border: 2px solid red; padding: 5px; margin: 10px 0;"> <p> The function name is UPPER/lowercase sensitive, and it should normally be given in UPPER case.</p> </div>
	<p><b>Example</b></p> <pre>Enter_function strngy=gm(fcc).x(cu); Export strngy#6</pre> <p>This transfers the value of the partial derivative of the Gibbs energy of the FCC phase with respect to the mole fraction of Cu to the optimizing variable 6 (i.e. <math>v_6</math>).</p>

## FLUSH\_BUFFER

 This command is for the ED-EXP module.

This command is only used in the \*.POP file (i.e. it can also be used in the experimental data files). It is needed if the number of experiments require more space than can be fitted into one buffer.

 In the DICTRA module, you work with a .DOP file instead of a .POP file.

When the PARROT program compiles experiments, it informs if any FLUSH commands are needed. FLUSH\_BUFFER terminates the current block, saves it to the work file and starts a new block. With the READ command in the EDIT-EXP module, you can select the block to edit.

Syntax	FLUSH_BUFFER
	After a FLUSH_BUFFER command, the workspace is reinitiated and all functions or constants must be entered again in the *.POP file. You can take advantage of this; for example, you can use the FLUSH command to have blocks with different components in one *.POP file.

## GRAPHICS\_PLOT



This command is for the ED-EXP module.

This command can help create an experimental data file from the \*.POP file. Several data sets and different symbols are allowed.



In the DICTRA module, you work with a .DOP file instead of a .POP file.

Syntax	GRAPHICS_PLOT
	<Dataset#> <X value> <Y value> <Symbol index>

## IMPORT

 This command is for the ED-EXP module.

Transfer/import the value of one of the optimizing variables to a constant. Normally, it is directly used inside an experimental data (\*.POP) file. It is the opposite of the EXPORT command.

 In the DICTRA module, you work with a .DOP file instead of a .POP file.

The IMPORT command is useful if several experiments are done at the same activity of a component, but this activity is not known. The activity should then be optimized and all equilibria with the same activity should use the same value. In this case the variable must be set as an optimizing variable, and an initial guess of the activity should be given. During the optimization, the PARROT program tries to find the activity that gives the best fit.

 If an approximate value of the activity is known, it can be supplied as an experiment.

Syntax	IMPORT
Prompt	<p>Constant name: &lt;Constant name&gt;#&lt;n&gt;</p> <p>The value of the <math>v</math> variable must be assigned a symbolic constant. The number (<math>n</math>) of the <math>v</math> variable must be given after the constant name, separated by hash character #.</p> <div style="border: 1px solid red; padding: 5px; margin-top: 10px;"> <p> The constant name is UPPER/lowercase sensitive, and it should normally be given in UPPER case.</p> </div>
	<p><b>Example</b></p> <pre>Enter_constant acu=0.1 Import acu#2</pre> <p>This transfers the value of the optimizing variable 2 (i.e. <math>v_2</math>) to the constant ACU.</p>

## LABEL\_DATA

 This command is for the ED-EXP module.

Add a label to the experimental equilibrium point, either as a single point or several points given individually or in a table. The label is maximum four characters and must start with the letter **A**. Normally, it is directly used inside an experimental data file (\*.POP).

 In the DICTRA module, you work with a .DOP file instead of a .POP file.

Several experimental equilibria can be given the same label, and the label can be used with SET\_WEIGHT to assign the same weight to all equilibria with the same label.

Syntax	LABEL_DATA
Prompt	Label? /A1/: a<bcd> Specify a textual label (as maximum characters and must start with the letter <b>A</b> ) for the current experimental equilibrium point(s).

---

## LIST\_ALL\_EQUILIBRIA



This command is for the ED-EXP module.

Syntax	LIST_ALL_EQUILIBRIA
	List all the details (including equilibrium indicators, labels, iteration numbers, weight, and fixed phases or comments) of all equilibrium points from the current one to the last one on screen. You can always use SELECT_EQUILIBRIUM to select a specific point as the current equilibrium point for such a list.

## LIST\_CONDITIONS



This command is available with the POLY and ED-EXP modules.

All the conditions that are set by the SET\_CONDITION command and the command-sequence CHANGE\_STATUS PHASE =FIXED <0 or 1 or alike>, are listed. The current conditions are also listed by the LIST\_EQUILIBRIUM command. The degree of freedom in the defined system is also shown.

- If this is zero, you can perform a COMPUTE\_EQUILIBRIUM command.
- If it is larger than zero, some more conditions are required, and you must further set additional ones, using SET\_CONDITION or CHANGE\_STATUS.
- If it is negative, then too many conditions are defined and the unnecessary ones conditions need to be removed using the SET\_CONDITION command (with a value of NONE for the to-be-deleted condition) or the CHANGE\_STATUS command (i.e. changing a FIXED status of a phase to another type of phase status, ENTERED or DORMANT or SUSPENDED).

Syntax	LIST_CONDITIONS
Example	<pre>P=100000, T=800, N(NI)=1E-1, N=1 FIXED PHASES FCC_A1=1 LIQUID=0 DEGREE OF FREEDO 0</pre>

## LIST\_EQUILIBRIUM



This command is available with the POLY and ED-EXP modules.

The result (always in SI units) from the last calculated equilibrium is listed on screen or in a text file.



You can also execute this command if no calculation is made or if the calculation fails. Make sure to interpret the results accordingly.

Syntax	LIST_EQUILIBRIUM
Prompt	<p>Output to screen or file /Screen/: &lt;File name&gt;</p> <p>The name of the text file where the list of the calculation results shall be written.</p>
	<p>Options /VWCS/: &lt;Option(s)&gt;</p> <p>Select the output units and formats by optionally specifying a combination of these letters.</p> <ul style="list-style-type: none"> <li>• <i>Fraction order</i>: <b>V</b> means VALUE ORDER; <b>A</b> means ALPHABETICAL ORDER.</li> <li>• <i>Fraction type</i>: <b>W</b> means MASS FRACTION; <b>X</b> means MOLE FRACTION.</li> <li>• <i>Composition</i>: <b>C</b> means only COMPOSITION; <b>N</b> means CONSTITUTION and COMPOSITION.</li> <li>• <i>Phase</i>: <b>S</b> means including only STABLE PHASES; <b>P</b> means including ALL NON-SUSPENDED PHASES.</li> </ul>
	<p>Default options are <b>VWCS</b>. If the output fraction type should be in mole fraction (rather than mass fraction), then give <b>VXCS</b> or type <b>X</b> (implying that in this case the options V, C and S are accepted as the listing manners for fraction order, composition and phase).</p> <p>If accepting all the default options, or if accepting all the altered options that had already changed when using this command previously, you can type <b>L_E,,,</b> or <b>LIST_EQUILIBRIUM ,,,</b></p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;">  <p>LIST_EQUILIBRIUM displays ORD (ordered) or DISORD (disordered) (when available). See <a href="#">CHANGE_LEGEND</a> for details.</p> </div>

## LIST\_STATUS



This command is for the ED-EXP module.



There is also a POLY module command with the same name.

List the status of components, species or phases.

Syntax	LIST_STATUS
Prompt	<p>Specify what to list:</p> <p>Option /CPS/ :&lt;keyword(s)&gt;</p> <p>keyword = C or P or S, or any combination</p> <ul style="list-style-type: none"> <li>• C: list component status</li> <li>• P: list phase status</li> <li>• S: list species status</li> <li>• CS: list both components and species</li> </ul> <p>Default is CPS.</p>
	<p>The results depend on the key word specified in the options for CHANGE_STATUS, a table with the current status of phases or species or components, or the combinations, is shown:</p> <ul style="list-style-type: none"> <li>• For components, the statuses and reference states are listed.</li> <li>• For species, only the status is listed.</li> <li>• For ENTERED and FIXED phases, the status, driving forces and equilibrium amounts are listed.</li> </ul>
	<div style="border: 2px solid blue; padding: 10px;"> <p> The metastable phases are listed in descending order of stability. To avoid long outputs only 10 metastable phases (in ENTERED status) are listed by lines, while all other less stable phases are merged onto one line. For DORMANT phases, their phase names and driving forces are listed. For SUSPENDED phases, only the phase names are listed.</p> </div>

## LIST\_SYMBOLS

>\_ This command is for the ED-EXP module.

>\_ There are also POLY and POST module commands with the same name.

Use this command to list the definitions for all constants, functions, variables and tables. In order to find the value of a function or variable, use SHOW\_VALUE or EVALUATE\_FUNCTIONS. A table is tabulated with the TABULATE command.

Syntax	LIST_SYMBOLS
	The defined variables are listed up together with the defined functions, but variable names are followed by a percentage sign %.

## MAKE\_POP\_FILE

 This command is for the ED-EXP module.

 In the DICTRA module, you work with a .DOP file instead of a .POP file.

Use this command to make interactive changes and additions to the original experimental data file (the \*.POP or \*.DOP file). For example, to document or transfer the current or final experimental data to a \*.POP file, this command will write it to a new \*.POP file or overwrite previous versions.



Check the output carefully for errors. Tables in original \*.POP files are written as individual experimental points. The output can be shown on screen or be saved as a new \*.POP file where to write or overwrite the edited experimental information in a simple text format.

Syntax	MAKE_POP_FILE
Prompt	Output to screen or file /Screen/: <POP/DOP-file name> Specify a name of the new *.POP file where to write or overwrite the edited experimental information in a simple text format. If an existing *.POP file name is specified, it is overwritten.

## READ\_WORKSPACES



This command must be given each time the ED\_EXP module is entered, unless it is already given once in ED\_EXP and does not use LIST\_RESULT or OPTIMIZE\_VARIABLES in PARROT in between.

Syntax	READ_WORKSPACES
Prompt	Block number /#/: <n> The number of a data block that should be edited must be given. If there is no FLUSH_BUFFER command in the original POP file, then there is only one data block with the number 1.
	<div data-bbox="370 730 418 787"></div> <pre>READ_WORKSPACE = READ</pre> <p>This command is equivalent to READ_WORKSPACES in the POLY module, but it only reads the POLY workspace from the work file set by SET_STORE_FILE in the PARROT module.</p>

---

## REINITIATE\_MODULE

 This command is for the ED-EXP module.

The REINITIATE\_MODULE command makes sure that the entire ED-EXP module is reinitiated to the original state.

Syntax	REINITIATE_MODULE
	All the defined components, defined conditions, changed status, entered symbols, defined independent axis-variables, calculated starting equilibrium-points, added initial equilibrium points, stepped/mapped equilibrium data, and so forth, are removed completely. The saved file name is restored to the default.

## RESTORE\_ALL\_WEIGHTS

 This command is for the ED-EXP module.

Restore a specific weight-set (previously saved by STORE\_ALL\_WEIGHTS) and assign all the different experimental points in the current data block with the previously-set weights in the sequential assessments.

Syntax	RESTORE_ALL_WEIGHTS
Prompt	Set number (0 for list) /0/: <weight-set number> The default value 0 is for a list on screen.

## SAVE\_WORKSPACES

 This command is for the ED-EXP module.

In the PARROT workspace this command saves the current status of the ED\_EXP submodule before you switch back to the PARROT module. This is required when you want to save equilibria changes, including changes to conditions, experimental data, and/or computed equilibria results, as well as any newly added equilibria.

With this command all the changes made in the ED\_EXP submodule are saved in the PARROT workspace. This means you do not need to execute SAVE\_PARROT\_WORKSPACES in the PARROT module.

However, when in the PARROT module, additional changes can only be saved with the SAVE\_PARROT\_WORKSPACES command even if they have been made prior to executing SAVE\_WORKSPACES in ED\_EXP.

When you exit the ED\_EXP (or Thermo-Calc), the next time the file is opened it is in the same state as when you executed the SAVE\_WORKSPACES command. However, if you use OPTIMIZE\_VARIABLES the computed results are automatically updated.

Syntax	SAVE_WORKSPACES
	<File name>
Prompt	 Sometimes good starting values (SET_START_VALUE) are required to successfully get or compute an equilibrium correctly. In this case, it is important to save the computed equilibrium in the PARROT workspace. It is important to remember that the computed results can be ruined by improper optimizations. For this reason, it is suggested you save the .PAR file with a different name to back up the PARROT workspace.

## SELECT\_EQUILIBRIUM



This command is available with the POLY and ED-EXP modules.

If you create more than one initial equilibrium (during data-assessments using the PARROT/ED\_EXP modules), you can switch between them using this command.

Syntax	SELECT_EQUILIBRIUM
Prompt	<p>Number /Next/: &lt;Choice on equilibrium&gt;</p> <p>Options: FIRST, LAST, NEXT, PREVIOUS or PRESENT.</p> <p>Most commands affect only the PRESENT equilibrium. However, the REINITIATE_MODULE and DEFINE_COMPONENTS commands remove all the stored equilibria.</p>

## SET\_ALL\_START\_VALUES



This command is available for the POLY and ED-EXP modules.

Set start values, e.g. if the calculation fails or if you have a miscibility gap or ordering. If temperature and pressure are not conditions, you are asked for values of them. Then for each phase prompt on if it should be stable and on its constitution.

Syntax	SET_ALL_START_VALUES
	<p>T /XXXX/: &lt;Temperature in K&gt;</p> <p>If the temperature is not a condition, supply a guess of its final value (in K).</p>
	<p>P /100000/: &lt;Pressure in Pa&gt;</p> <p>If the pressure is not a condition supply a guess of its final value (in Pa).</p>
Prompts	<p>Automatic start values for phase constitutions? /N/: &lt;Y, N or F&gt;</p> <ul style="list-style-type: none"> <li>For N, enter an initial amount of each entered phase and the major constituents or site fraction of each constituent.</li> <li>For Y, the program automatically sets the start values for phase constitutions in all possible phases.</li> <li>F is for force. In some cases the calculation fails because impossible conditions are set, e.g. <math>w(C)=1.5</math> [<math>w(C)</math> is mass fraction and must thus be less than unity. The program tries to fulfill this condition by putting maximum amount of carbon in all phases, but can fail anyway. When you detect the error and set <math>w(C)</math> to 0.015, the calculation may still fail because it could start from the previous values. To get back to 'fresh' start values, choose F.</li> </ul>
	<p>Should &lt;phase&gt; be stable /N/: &lt;Y/1 or N/2&gt;</p> <p>A guess if this phase should be stable or not is requested. All entered phases are prompted for this question and next two.</p> <div style="border: 2px solid red; padding: 10px; margin: 10px 0;"> <p> You cannot have more phases stable than you have components, but at least one (which dissolves all constituents) must be set as stable. For backward compatibility, this question can be answered by 1 (for Yes) or 0 (for No).</p> </div> <p>The phase name may have a hash sign # followed by a digit, e.g. BCC_A2#2. For phases with miscibility gaps, there should be two phases with the same name but with different numbers after the hash sign.</p>
	<p>Major constituent(s): &lt;Name of major constituent(s) in the phase&gt;</p> <p>The constituent with the largest fraction in the phase should be specified. If there are more than one constituent with a large fraction, give them all on the same line. If the default major constituents should be used answer with an asterisk (*). By giving \$ the constitution is not changed. If there should be no major constituent give NONE and or if the major constituent(s) are improperly specified, you are asked for individual fractions in the phase.</p>

Syntax	SET_ALL_START_VALUES
	<pre data-bbox="358 258 948 281">Y(&lt;phase&gt;,&lt;constituent&gt;) /.XXXXXXXXX/: &lt;.YYYYY&gt;</pre> <p data-bbox="358 302 1398 359">The current value (.XXXXXXXXX) is default. You can accept the default by pressing &lt;Enter&gt; or give a new value (.YYYYY).</p> <p data-bbox="358 380 1398 495">The phase name or constituent name may have a hash sign # followed by a digit, e.g. Y(BCC_A2#2,FE), Y(BCC_A2#2,C#2). For phases with miscibility gaps, there should be two phases with the same name but with different numbers after the hash sign. For phases with sublattices, the constituents in sublattice 2 or higher is also be suffixed with a hash sign # followed by a digit.</p>

## SET\_ALTERNATE\_CONDITION

 This command is for the ED-EXP module.



A special command used in the experimental data \*.POP file but also possible in the ED\_EXP module. It is used only when you have specified that the alternate mode is set in the PARROT module.



In the DICTRA module, you work with a .DOP file instead of a .POP file.

The command syntax is almost the same as for the POLY command SET\_CONDITION while the uncertainty should also be specified.



The alternate condition (including the normally POLY module condition plus uncertainty; see below) must be given explicitly, but can be given on the same line or on separate lines with each one started with the command.

<b>Syntax</b>	<b>SET_ALTERNATE_CONDITION</b>
<b>Prompt</b>	State variable expression: <State variable name or linear expression> Give either a state variable or a linear expression of state variables.
<b>Options</b>	<b>Description and Information</b>
<b>State variables</b>	<p>Some of the state variables that can be used in conditions are:</p> <ul style="list-style-type: none"> <li>• <b>T</b>: temperature in the system (in K)</li> <li>• <b>P</b>: pressure in the system (in Pascal)</li> <li>• <b>N</b>: system size (mole number in moles)</li> <li>• <b>B</b>: system size (mass in grams)</li> <li>• <b>N (&lt;component&gt;)</b>: mole number of a component in the system</li> <li>• <b>X (&lt;component&gt;)</b>: mole fraction of a component in the system</li> <li>• <b>W (&lt;component&gt;)</b>: weight fraction of a component in the system</li> <li>• For the SET_CONDITION command this is <b>ACR (&lt;component&gt;)</b>: activity of a component in the system. For the SET-ALTERNATIVE_CONDITION command, this is <b>ACR (&lt;phase&gt;,&lt;component&gt;)</b> (activity of a component in a phase)</li> <li>• For the SET_CONDITION command this is <b>MUR (&lt;component&gt;)</b>: chemical potential of a component in the system. For the SET-ALTERNATIVE_CONDITION command, this is <b>MUR (&lt;phase&gt;,&lt;component&gt;)</b> (chemical potential of a component in a phase)</li> </ul>

Syntax	SET_ALTERNATE_CONDITION
	<ul style="list-style-type: none"> <li>• <math>X(\langle\text{phase}\rangle, \langle\text{component}\rangle)</math>: mole fraction of a component in a phase</li> <li>• <math>W(\langle\text{phase}\rangle, \langle\text{component}\rangle)</math>: weight fraction of a component in a phase</li> <li>• <math>ACR(\langle\text{species}\rangle, \langle\text{phase}\rangle)</math>: activity of a species referred to a phase</li> <li>• <math>MUR(\langle\text{species}\rangle, \langle\text{phase}\rangle)</math>: chemical potential of a species referred to a phase</li> <li>• <math>H</math>: enthalpy in the system (in J)</li> <li>• <math>HM(\langle\text{phase}\rangle)</math>: enthalpy of a phase (in J/mol)</li> </ul>
Additional prompts	<p>Factor: &lt;A factor for the state variable, or a continuation&gt;</p> <p>This question means that the previous question was not answered. The program is then expecting a single state variable or a complete state variable expression, or the numeric factor in an expression with only one state variable. In a state variable expression a state variable may be preceded by a constant factor. An example of this is:</p> <pre>2*MUR (FE) +3*MUR (O) ==-35000</pre> <p>This means that it should be a condition that two times the chemical potential of FE plus three times the chemical potential of O should be -35000 J/mol.</p>
	<p>State variable: &lt;A specified state variable, or a continuation&gt;</p> <p>This question is prompted if a single state variable name has not given in either the prompt <code>State variable expression</code> or <code>Factor</code>, or a state variable expression is given but the expression is incomplete, for example, <code>T-</code> or <code>2*MUR (FE) +</code>, for which the program is then expecting a continuation of the unfinished expression. You need to specify a state variable or a complete state variable expression, or complete the unfinished state variable expression. If a numeric factor is given before this prompt, only one state variable can be specified; otherwise, the program only takes the first state variable to complete the expression (i.e. the factor times the state variable).</p>
	<p>Value /X/: &lt;A numeric value, a constant or a variable&gt;</p> <p>The value of the condition. This can be a numeric value, a constant or a variable. A suggestion is given as the default value. The special value <code>NONE</code> means that the condition is removed.</p>
	<p>Uncertainty /None/: &lt;Uncertainty of the condition&gt;</p> <p>The uncertainty of the condition. This can be a numeric value, a constant or a variable. The default value <code>NONE</code> means that the uncertainty for the value specified above is zero. The uncertainty can either be expressed as an absolute value or relative to the value of the condition in percent (x%).</p>
Notes	<p>There are more state variables that can be used in conditions. For more information, type an <code>INFO STATE_VARIABLES</code> command in the <code>POLY</code> module. A condition is normally a value of a single state variable with its value.</p> <p><b>Example</b></p> <pre>T=1273.15 P=1E5 X (C) =.002 W (CR) =1.5 ACR (CR) =0.85 X (FCC, C) =.001 H=-250000</pre>

## Syntax

## SET\_ALTERNATE\_CONDITION

```
HM (BCC) = -225000
```

A condition can also be a value of a linear expression involving more than one state variable.

**Example**

```
X (LIQ, S) - X (PYRR, S) = 0
```

This means that it is a condition that the mole fraction of *s* should be the same in the LIQUID and PYRRHOTITE phases. In practice that should be the congruent melting point.



After the equal sign only a numeric value is allowed.

## SET\_CONDITION

 This command is available with the POLY and ED-EXP modules.

Specify the equilibrium conditions for the calculation. All kinds of state variables, as well as most of the  $M/W/V/F$ -suffixed state variables (for normalization) and  $R$ -suffixed state variables (with respect to chosen reference states), can be used as conditions. In addition to temperature and pressure conditions, a multicomponent system can have mixed types of conditions. You should repeat this command for a multicomponent system until the degree of freedom in the defined system becomes zero. When a FIXED phase status is used on a specific phase (use CHANGE\_STATUS), it is equivalent to one condition (implying that particular phase is stable in the defined system).

Syntax	SET_CONDITION
	<p>Each condition must be given explicitly, but can be given on the same line (separated by a comma (,) or an empty space) or on separate lines with each one started with the command.</p> <p><b>Example</b></p> <pre>SET_COND T=1273, P=1E5, W(C)=.0015, X(LIQ,Cr)=.22, ACR(N)=.2</pre> <p>or</p> <pre>SET_COND T=1273, P=1E5 SET_COND W(C)=.0015, X(LIQ,Cr)=.22 ACR(N)=.2</pre> <p>In this example, the temperature is set to 1273 K, the pressure to 1 bar (1E5 Pascal), the mass (weight) fraction of C to 0.0015 and the mole fraction of Cr to 0.22 and the activity of N to 0.2.</p>
Prompt	<p>State variable expression: &lt;State variable name or linear expression&gt; or a linear expression of state variables.</p> <p>Some of the state variables that can be used in conditions are:</p> <ul style="list-style-type: none"> <li>• T: temperature in the system (in K)</li> <li>• P: pressure in the system (in Pascal)</li> <li>• N: system size (mole number in moles)</li> <li>• B: system size (mass in grams)</li> <li>• N(&lt;component&gt;): mole number of a component in the system</li> <li>• X(&lt;component&gt;): mole fraction of a component in the system</li> <li>• W(&lt;component&gt;): weight fraction of a component in the system</li> <li>• For the SET_CONDITION command this is ACR(&lt;component&gt;): activity of a component in the system. For the SET-ALTERNATIVE_CONDITION command, this is ACR(&lt;phase&gt;,&lt;component&gt;) (activity of a component in a phase)</li> <li>• For the SET_CONDITION command this is MUR(&lt;component&gt;): chemical potential of a component in the system. For the SET-ALTERNATIVE_CONDITION command, this is MUR(&lt;phase&gt;,&lt;component&gt;) (chemical potential of a component in a phase)</li> </ul>

Syntax	SET_CONDITION
	<ul style="list-style-type: none"> <li>• <math>X(\langle \text{phase} \rangle, \langle \text{component} \rangle)</math>: mole fraction of a component in a phase</li> <li>• <math>W(\langle \text{phase} \rangle, \langle \text{component} \rangle)</math>: weight fraction of a component in a phase</li> <li>• <math>ACR(\langle \text{species} \rangle, \langle \text{phase} \rangle)</math>: activity of a species referred to a phase</li> <li>• <math>MUR(\langle \text{species} \rangle, \langle \text{phase} \rangle)</math>: chemical potential of a species referred to a phase</li> <li>• <math>H</math>: enthalpy in the system (in J)</li> <li>• <math>HM(\langle \text{phase} \rangle)</math>: enthalpy of a phase (in J/mol)</li> </ul> <p>There are many state variables that can be used in conditions. For more information, enter an <code>INFO STATE_VARIABLES</code> command. A condition is normally a value of a single state variable with its value.</p> <p><b>Example</b></p> <pre>T=1273.15 P=1E5 X(C)=.002 W(CR)=0.5 ACR(CR)=0.85 X(FCC,C)=.001 H=-250000 HM(BCC)=-225000</pre> <p>A condition can also be a value of a linear expression involving more than one state variable. For example,</p> <pre>X(LIQ,S)-X(PYRR,S)=0</pre> <p>This means that it is a condition that the mole fraction of <code>s</code> component should be the same in the LIQUID and PYRRHOTITE phases. In practice, that should be the congruent melting point.</p> <div style="border: 2px solid blue; border-radius: 10px; padding: 10px; margin-top: 10px;">  After the equal sign only a numeric value is allowed in the expression. </div>
	<p>Factor: &lt;A factor for the state variable, or a continuation&gt;</p> <p>This question means you did not answer the previous question. The program is then expecting a single state variable or a complete state variable expression, or the numeric factor in an expression with only one state variable. In a state variable expression a state variable may be preceded by a constant factor. An example of this is:</p> <pre>2*MUR(Fe)+3*MUR(O)=-35000</pre> <p>This means that it should be a condition that two times the chemical potential of <code>Fe</code> plus three times the chemical potential of <code>O</code> should be -35000 J/mol.</p>
	<p>State variable: &lt;A specified state variable, or a continuation&gt;</p> <p>This question is prompted if a single state variable name has not given in either the prompt <code>State variable expression</code> or <code>Factor</code>, or a state variable expression is given but the expression is incomplete, for example, <code>T-</code> or <code>2*MUR(Fe)+</code>, for which the program is then expecting a continuation of the unfinished expression. You need to specify a state variable or a complete state variable expression, or complete the unfinished state variable expression. If a numeric factor is given before</p>

---

Syntax	<b>SET_CONDITION</b>
	this prompt, only one state variable can be specified; otherwise, the program only takes the first state variable to complete the expression (i.e. the factor times the state variable).
	Value /X/: <A numeric value, a constant or a variable>  The value of the condition. This can be a numeric value, a constant or a variable. A suggestion is given as the default value. The special value <code>NONE</code> means that the condition is removed; for example <code>T=NONE</code> takes away the temperature condition.

## SET\_NUMERICAL\_LIMITS

 This command is available for the POLY, Scheil, and ED-EXP modules.

Change the criteria for convergence. This is to speed up a calculation in a complex system.

If there is any convergence problem in finding a stable solution at any stage during a calculation procedure enforced by COMPUTE\_EQUILIBRIUM, STEP\_WITH\_OPTIONS, MAP or ADVANCED\_OPTIONS), this message displays:

```
Convergence problems, increasing smallest site-fraction from 1.00E-30
to hardware precision 2.00E-14. You can restore using SET-NUMERICAL-
LIMITS
```

This implies that smallest site fraction in the current POLY workspace is automatically increased from the default value 1.00E-30 to the hardware-dependent precision (under Linux, as 2.00E-14). For other subsequent POLY module calculations, you can use this command to restore or reset the smallest site fraction to the previous or another preferred value, as well as to reset other numerical limits.

Syntax	SET_NUMERICAL_LIMITS
Prompt	<p>Maximum number of iterations /500/:</p> <p>By default, the program tries 500 iterations before it gives up. As some models give computation times of more than 1 CPU second/iteration, this number is also used to check the CPU time and the calculation stops if 500 CPU seconds/iterations are used.</p>
	<p>Required accuracy /1E-6/:</p> <p>This is a relative accuracy, and the program requires that the relative difference in each variable must be lower than this value before it has converged. A larger value normally means fewer iterations but less accurate solutions. The value should be at least one order of magnitude larger than the machine precision.</p>
	<p>Smallest fraction /1E-12/:</p> <p>This is the value assigned to constituents that are unstable. It is normally only in the gas phase you can find such low fractions.</p> <p>The default value for the smallest site-fractions is 1E-12 for all phases, except for IDEAL phase with one sublattice site (such as the GAS mixture phase in many databases) for which the default value is always as 1E-30 (unless you have used the SET_NUMERICAL_LIMITS command to reset an even-lower value (e.g. 1E-45, that is naturally enforced to all the phases in the system).</p>
	<p>Approximate driving force for metastable phases /Y/:</p> <p>Y is the default. Enter N to change the default as required and based on the options described below.</p> <p>This setting involves the convergence of metastable phases and affects their driving forces. It can also have an effect on when an equilibrium is considered successful.</p> <p>The default is to allow an equilibrium with metastable phases that have not converged, as long as the stable phases have converged. This is efficient but often causes approximate values of the</p>

---

Syntax	SET_NUMERICAL_LIMITS
	<p>driving forces for the metastable phases.</p> <p>If you change the default, it enforces metastable phases to converge. This gives accurate driving forces for metastable phases as well as stable phases. It can however take a slightly longer time, and if metastable phases do not converge it causes the equilibrium calculation to fail.</p>

## SET\_REFERENCE\_STATE



This command is available for the POLY, ED-EXP and POST modules.

The reference state for a component is important when calculating activities, chemical potentials, and enthalpies and is determined by the database being used. For each component the data must be referred to a selected phase, temperature, and pressure, i.e. the reference state.



All data in all phases where this component dissolves must use the same reference state. However, different databases can use different reference states for the same element/component. It is important to be careful when combining data obtained from different databases.

By default, activities, chemical potentials, and so forth are computed relative to the reference state used by the database. If the reference state in the database is not suitable for your purposes, use this command to set the reference state for a component using `SER`, i.e. the Stable Element Reference (which is usually set as default for a major component in alloys dominated by the component). In such cases, the temperature and pressure for the reference state is not needed.

In order to specify conditions in the specific reference state, you can append an `R` to the state variables. Thus, `AC` is the activity (of a system component or of a species in a phase) with respect to the default reference state, and `ACR` is the activity with respect to the selected reference state; `MU` is the chemical potential of a system component with respect to the default reference state, and `MUR` stands for the chemical potential with respect to the selected reference state. The `AC` and `ACR` variables, for both components in a system and species in a phase, can be specified in natural logarithm, e.g. `LNAC(Fe)`, `LNACR(C)`, `LNAC(O2,GAS)`, `LNACR(O2,GAS)`.

For the POST module, and after calculating with the `STEP_WITH_OPTIONS` or `MAP` commands, the reference state for a component can also be changed to plot various properties of the component in the entire system or in a specific phase. Then you can set the diagram axes as the chemical potential or activity quantities with the `R` suffix, i.e. `MUR(comp)`, `MUR(comp,ph)`, `ACR(comp)`, `ACR(comp,ph)` or the common logarithms [e.g. `LNACR(comp,ph)`].

Syntax	SET_REFERENCE_STATE
Prompts	Component: <Name of the component> The name of the component must be given.
	Reference phase: <Name of a phase used as the new reference state>

Syntax	SET_REFERENCE_STATE
	<p>Enter the Name of a phase for the system.</p> <div data-bbox="363 317 1398 520" style="border: 1px solid blue; padding: 10px; margin: 10px 0;">  <p>For a phase to be usable as a reference for a component, the component needs to have the same composition as an end member of the phase. The reference state is an end member of a phase. The selection of the end member associated with the reference state is only performed once this command is executed.</p> </div> <p>If a component has the same composition as several end members of the chosen reference phase, then the end member that is selected at the specified temperature and pressure will have the lowest Gibbs energy.</p> <p>The following are example using temperature. Pressure is used in the same way:</p> <ul style="list-style-type: none"> <li>• In the Fe-C system, BCC can be a reference state for Fe but not for C since BCC can exist as pure Fe but cannot exist as pure C.</li> <li>• If GAS is chosen as the reference phase of component O at such a high temperature that O1 has the lowest energy, then O1 remains the reference state even at calculations performed at lower temperatures where O2 has a lower energy than O1.</li> <li>• Setting the reference state for component O as gas (one sublattice) with constituents O1, O2 and O3 results in O2 being the reference state if, at the present temperature, pure O2 has the lowest energy of O1, O2 and O3. If the reference state is set above a critical higher temperature, then O1 has the lowest energy and consequently becomes the reference state.</li> </ul>
	<p>Temperature /*/: &lt;Temperature for the reference state&gt;</p> <p>Select the Temperature (in K) for the reference state. The wildcard value * means the current temperature is used at the time of evaluation of the reference energy for the calculation.</p> <div data-bbox="363 1194 1398 1369" style="border: 1px solid blue; padding: 10px; margin: 10px 0;">  <p>The temperature set with this command does not affect the temperature used when evaluating the energy of the reference state during the end member selection process.</p> </div>
	<p>Pressure /1E5/: &lt;Pressure for the reference state&gt;</p> <p>Select the Pressure (in Pa) for the reference state. The wildcard value * means the current pressure is used for evaluating the reference energy at the time of calculation.</p>
	<p><b>Examples</b></p> <pre>S-R-S Fe SER S-R-S Cr FCC * 100000 S-R-S H2O AQUEOUS * 100000 S-R-S ZE REF_ELECTRODE * 100000</pre>

## SET\_START\_CONSTITUTION



This command is available with the POLY and ED-EXP modules.

This command is similar to SET\_ALL\_START\_VALUES, but is used for an individual phase that may have a wrong constitution.

Syntax	SET_START_CONSTITUTION
Prompts	<p>Phase name: &lt;Name of a phase, and possible major constituent(s)&gt;</p> <p>Specify the name of a phase for which the constitution shall be set.</p> <p>If there is a major constituent of the phase, you must specify this on the same line as the phase name. By giving an*, select the default major constituents. A \$ means keeping the same constitution, and NONE means to give individual site-fractions.</p>
	<p>Y(&lt;Phase&gt;#&lt;Composition_set&gt;,&lt;Constituent&gt;#&lt;Sublattice&gt;) /xxx/: &lt;SF&gt;</p> <p>The site fraction (SF) of the constituent shall be given. The default value xxx is the last calculated one.</p>

## SET\_START\_VALUE



This command is available with the POLY and ED-EXP modules.



This command is not often required as the automatic start values can be handled for most problems.

Set a start value of an individual variable.

Syntax	SET_START_VALUE
Prompts	State variable: <Name of a state variable> The state variable must be given.
	Value Specify the value to be assigned the state variable as start value.

## SET\_WEIGHT

 This command is for the ED-EXP module.

Each experimental value has an uncertainty associated with it, specified by the value after the colon (:) with the command EXPERIMENT. During an optimization the absolute difference between the experimental and calculated values gives a contribution to the sum of error.

With the SET\_WEIGHT command, you can change the scale of such a contribution (the uncertainty) for a single experimental point, or the contributions (all uncertainties) for a set of equilibria. The default weight is always unity. A value smaller than one means that the experiments should have less weight. A value larger than one that these should have higher weight. The special value zero means that the set of equilibria should be ignored in editing in ED\_EXP (as is given) and in OPTIMIZE\_VARIABLES in PARROT.



The weight may be needed to obtain a balance between different kinds of experiments. For example, if there are only five experimental values of the composition of a phase diagram but 500 experimental values of activities or enthalpies, then the five composition points may have to be given higher weight than unity, otherwise these are not well described by the optimizing procedure.

Syntax	SET_WEIGHT
	<pre>Value /1/: &lt;Weight value&gt;</pre> <p>Specify a weight of the experiments in the specified equilibria (asked in the next prompt). The contribution to the sum of errors of these experiments are multiplied by this weight.</p>
Prompts	<div style="border: 1px solid blue; padding: 5px;">  The weight is squared, thus use 0.7 to make the error half (0.49) as big, and 1.4 to make it twice (1.96) as large.         </div>
	<pre>Equilibria (range) or label(s) /Present/:&lt;Selection&gt;</pre> <p>Define the equilibria. These are available in the read data block from the current work file and are given the above-defined weight.</p> <p>The selection may be given as a range but the equilibrium numbers must then be separated only by a minus sign.</p>

Syntax	SET_WEIGHT
	<div data-bbox="363 260 1393 401" style="border: 1px solid blue; padding: 5px;"> The start and end equilibrium numbers must be available in the read data block read from the current work file.</div> <p data-bbox="363 432 1393 489">For example, <code>5-101</code>, means that all equilibria starting from the identifier 5 until the identifier 101. Some of numbers between these two identifiers may not exist in the data block.</p> <p data-bbox="363 510 1393 567">The selection may also be given as one or more labels (which should be separated only by a space). All equilibria with the specified label(s) in the read data block are given the above-defined weight.</p> <p data-bbox="363 588 1393 674">By pressing &lt;Enter&gt; to accept the default value <code>present</code>, the above defined weight is only assigned to the current or present experimental point (which is selected by the <code>POLY SELECT_EQUILIBRIUM</code> command).</p> <div data-bbox="363 705 1393 846" style="border: 1px solid blue; padding: 5px;"> If is used but no specific equilibrium point is selected afterwards, the last point in the read data block remains the <code>present</code> point.</div>

## SHOW\_VALUE



This command is available for the POLY and ED-EXP modules.

Show the current value of any state variable, function or variable.

Syntax	SHOW_VALUE
Prompt	<p>State variable or symbol: &lt;Name(s) of state variable(s) or symbol(s)&gt;</p> <p>A single or several state variables or symbols (entered functions or variables) can be specified.</p>
	<div style="border: 1px solid #00AEEF; padding: 5px; margin-bottom: 10px;">  The wildcard * can be used to denote all phases or all components.         </div> <p>You can use the dollar-sign \$ to denote all stable phases. Thus, the command SHOW W(*,*) lists the mass fraction of all phases, whereas SHOW W(\$, *) lists the mass fraction of all stable phases.</p> <div style="border: 1px solid #00AEEF; padding: 5px; margin-bottom: 10px;">  If you SHOW a function, all functions are evaluated using the current values of the state variables. But if you SHOW a variable, it retains its value from the time it was ENTERED or last EVALUATED.         </div> <p>Therefore to save a value from one calculation to another, ENTER it to a variable. This is frequently done in the PARROT module to transfer values between equilibria.</p>

## STORE\_ALL\_WEIGHTS

Store a new weight-set that is the current situation of specified weights for all the different experimental points read from the currently-selected experimental data block in an assessment. This is useful when you want to try various weight-settings for some experimental points or for all the points, and then compare the assessment results from different weight-sets. Such a saved weight-set can be recalled and restored for all experimental data point with RESTORE\_ALL\_WEIGHTS sequentially.

Syntax	STORE_ALL_WEIGHTS
Prompt	Command line: <Weight-set name> Give a weight-set name (comment line texts) for the current weight set for all experimental points.

## TABLE\_HEAD, TABLE\_VALUES and TABLE\_END

These commands are used only in the \*.POP file. It represents a convenient way to enter many experimental measurements of the same type in a table format.



In the DICTRA module, you work with a .DOP file instead of a .POP file.



These commands should always be used together and in a sequence, meaning that a table should always start with the TABLE-HEAD command, then TABLE-VALUES, and finish with TABLE-END. Between TABLE\_HEAD and TABLE-VALUES also add definition lines (e.g. on phase status, reference states, conditions, experiments, labels, etc.) for the experimental measurements of the current information type.

Syntax	TABLE_HEAD TABLE_VALUES TABLE_END
Prompt	
	<p>After the TABLE-HEAD command, there must be an equilibrium description similar to a single experimental equilibrium but with some special notation. Then, there is always a TABLE-VALUES command, after which the actual data is given in columns. At the end of each table, there must be a TABLE-END command.</p> <p>The TABLE-HEAD command must be followed by a numeric value. This is used to generate unique numeric identifiers for all the equilibria in the table. The numeric identifier is incremented by one for each experimental (equilibrium) point in the table.</p>
	<p><b>Example</b></p> <p>An example of the use of a table for enthalpy measurements in the liquid is given here. Much more elaborate tables can be used</p> <pre> \$ Enthalpies of mixing, Topor and Kleppa, Met Trans 1984 TABLE-HEAD 1 CREATE-NEW @@ 1 CHANGE-STATUS PHASE LIQ=FIX 1 SET-REFERENCE-STATE AU LIQ * 1E5 SET-REFERENCE-STATE CU LIQ * 1E5 SET-CONDITION P=1E5 T=1379 X(LIQ,AU)=@1 LABEL ALH EXPER HMR=@2:5% TABLE-VALUES 0.0115 -322 </pre>

Syntax	<p>TABLE_HEAD</p> <p>TABLE_VALUES</p> <p>TABLE_END</p>
	<pre> 0.0563 -1520 0.8499 -2976 0.9002 -2114 TABLE-END </pre>
	<p>The equilibrium description between TABLE-HEAD and TABLE-VALUES is similar as for a single experiment, except for these details:</p> <ul style="list-style-type: none"> <li>• @@ Used to automatically generate a series of identifying numbers instead of creating only one with the command CREATE_NEW_EQUILIBRIUM. The program automatically generates the identifying numbers starting after the number given after the TABLE-HEAD command (in this case it is 1).</li> <li>• @1 values in the table are specified by an @ sign followed by a column number. In the above case, the mole fraction of Au in the liquid is in column 1 (see the line for SET_CONDITION). The columns may have other information than values, and you can use phase names or any text. If the text contains spaces or special characters, it must be surrounded by double quotes in the table, e.g. "ABC DEF&amp;ghi".</li> <li>• For the <i>syntax checker</i> all commands describing the equilibrium must be in UPPER case after the TABLE_HEAD command.</li> <li>• After the TABLE-VALUES command, there should be one line for each experimental measurement of the same type giving the values or texts that should be copied into the places of the condition or experiment defined by @1, t, etc. You can have columns that are not used but there must be exactly the same number of columns of each line, otherwise the syntax checker gives an error message in the PARROT module.</li> </ul>

## TRANSFER\_START\_VALUES

 This command is for the ED-EXP module.

In the ED\_EXP module it is time consuming to calculate all equilibria when the optimization is sensitive to start values of the composition of the phases. Usually in one data block there are several experimental points with the same kind of equilibrium and each must have its start value set. When you manage to calculate equilibria of such experimental points, this command is useful to transfer the site fractions from a successfully-calculated equilibria to the present experimental point.

Syntax	TRANSFER_START_VALUES
Prompt	From equilibrium /Previous/: <Equilibrium number> The equilibrium number is the numeric identifier for the equilibrium from which the start values should be copied to the present experimental point. <i>Previous</i> is the default if you press <Enter>.

---

## Tabulation Reaction (TAB) Commands

---

**In this section:**

---

ENTER_FUNCTION .....	459
ENTER_REACTION .....	461
LIST_SUBSTANCES .....	463
SET_ENERGY_UNIT .....	466
SET_PLOT_FORMAT .....	467
SWITCH_DATABASE .....	468
TABULATE .....	470
TABULATE_DERIVATIVES .....	471
TABULATE_REACTION .....	476
TABULATE_SUBSTANCE .....	479

## ENTER\_FUNCTION



This command is available with the GES and TAB modules.

Used to define a new thermodynamic function to be included in the sixth column of tabular output. The last entered function is listed in the 6th column of all tables, and can be plotted (if choosing the Plot Column value 6) in all graphs, for all subsequently tabulated substances or reactions.

Syntax	ENTER_FUNCTION
Prompt	Name: <Name> Function: <Definition>
	A limited number of state variables, G, H, S, T, P, V and H298 can be used in the function definition. For example, $-(G-H)/T$ can be defined as a function which is identical to S for a substance or $\Delta S$ for a reaction, $G+T*S-P*V$ can be defined as a function which is U (internal energy) for a substance or $\Delta U$ (internal energy change) for a reaction.
Options	Name of the function (maximum 8 characters) and Definition of the function. A long function can be typed over several lines. A function should be terminated by a semicolon (;) or an empty line.

### Example

For the pure substance Fe (using the SSUB database), by entering a function as  $G+T*S-P*V$ , the following table is obtained by typing <Enter> in the prompt Output file /Screen/:, and the figure by typing 6 in the prompt Plot column /2/:

```
TAB: ENTER-FUNCTION
Name: InEnergy
Function: G+T*S-P*V;
TAB: TABULATE_SUBSTANCE
Substance (phase): Fe
Pressure /100000/: <Enter>
Low temperature limit /298.15/: <Enter>
High temperature limit /2000/: <Enter>
Step in temperature /100/: <Enter>
Output file /try1/: try2
Graphical output? /Y/: <Enter>
Plot column? /2/: 6
```

This gives the following output:

O U T P U T F R O M T H E R M O - C A L C

Column 6: InEnergy G+T\*S-P\*V

Phase : FE\_S Pressure : 100000.00

Specie: FE

\*\*\*\*\*

T (K)	Cp (Joule/K)	H (Joule)	S (Joule/K)	G (Joule)	InEnergy
298.15	2.48446E+01	2.17972E-06	2.72800E+01	-8.13353E+03	2.17972E-06
300.00	2.48905E+01	4.60049E+01	2.74338E+01	-8.18414E+03	4.60049E+01
400.00	2.71299E+01	2.64957E+03	3.49085E+01	-1.13138E+04	2.64957E+03
500.00	2.93561E+01	5.47211E+03	4.11976E+01	-1.51267E+04	5.47211E+03
600.00	3.19293E+01	8.53245E+03	4.67701E+01	-1.95296E+04	8.53245E+03
700.00	3.50985E+01	1.18777E+04	5.19207E+01	-2.44667E+04	1.18777E+04
800.00	3.92042E+01	1.55830E+04	5.68623E+01	-2.99068E+04	1.55830E+04
900.00	4.49645E+01	1.97726E+04	6.17903E+01	-3.58387E+04	1.97726E+04
1000.00	5.42147E+01	2.46891E+04	6.69619E+01	-4.22728E+04	2.46891E+04
1100.00	4.55851E+01	2.99025E+04	7.19412E+01	-4.92328E+04	2.99025E+04 \$
Stable phase is FE_S2					
1200.00	3.40840E+01	3.51037E+04	7.64466E+01	-5.66322E+04	3.51037E+04
1300.00	3.49398E+01	3.85549E+04	7.92086E+01	-6.44162E+04	3.85549E+04
1400.00	3.57994E+01	4.20918E+04	8.18293E+01	-7.24692E+04	4.20918E+04
1500.00	3.66636E+01	4.57149E+04	8.43287E+01	-8.07780E+04	4.57149E+04
1600.00	3.75330E+01	4.94247E+04	8.67226E+01	-8.93314E+04	4.94247E+04 \$
Stable phase is FE_S3					
1700.00	4.05217E+01	5.41173E+04	8.95609E+01	-9.81363E+04	5.41173E+04
1800.00	4.12595E+01	5.82055E+04	9.18975E+01	-1.07210E+05	5.82055E+04 \$
Stable phase is FE_L					
1900.00	4.60000E+01	7.74165E+04	1.02377E+02	-1.17099E+05	7.74165E+04
2000.00	4.60000E+01	8.20165E+04	1.04736E+02	-1.27456E+05	8.20165E+04

## ENTER\_REACTION



This command is for the TABULATE\_REACTION (TAB) module.

This is the same as TABULATE\_REACTION in that both the chemical reaction can be specified and the thermodynamic property changes for the reaction can be generated as a table, and plotted as a graph or saved as an experimental file (with an extension of .EXP). The thermodynamic data are automatically retrieved from the SSUB database (by default), or the current database set by SWITCH\_DATABASE (thus there is no need to use the TDB module before this command). You can select any other database by using the SWITCH\_DATABASE command in the TAB module. The rules to specify a reaction are the same as TABULATE\_REACTION.

Syntax	ENTER_REACTION
Prompts	<p>Same reaction? /Y/:</p> <p>This displays if there is at least one reaction already defined either by ENTER_REACTION or TABULATE_REACTION. For Y the next prompt for defining reaction is not asked.</p>
	<p>Reaction: &lt;Chemical reaction equation&gt; &amp; &lt;RA+RB=PC+PD&gt;</p> <p>A long reaction can be typed over several lines. A reaction should be terminated by a semicolon (;) or an empty line. A list showing the used database, defined elements, getting-data sequence, references, etc.:</p> <p>Pressure /100000/: &lt;Pressure of interest, in pa&gt;</p> <p>Low temperature limit /298.15/: &lt;T-low, in K&gt;</p> <p>High temperature limit /2000/: &lt;T-high, in K&gt; Step in temperature /100/: &lt;T-step&gt;</p>
	<p>Output file /Screen: &lt;Return for screen, or type a file name&gt;</p> <p>If you type Return for screen a list of thermodynamic property changes for the defined/chosen reaction is shown up as a table (which is demonstrated in the example outputs given below), and the command is terminated.</p> <p>If typing a file name, the table is displayed and saved as an *.EXP or an *.TAB file. Then prompts:</p>
	<p>Graphical output /Y/:</p> <p>For N a table is output to the screen and this table is saved as a basic text file with the default extension .TAB under the current working directory. In this case, the graph cannot be plotted.</p> <p>By default a table is created with all thermodynamic functions as normal (which is displayed on screen) and generates a graph with temperature on the X-axis and a chosen property on a certain column in the table on the Y-axis (which is plotted on screen and saved as an *.EXP file). You are also prompted for the plot column.</p>
	<p>Plot column? /2/: &lt;1 or 2 or 3 or 4 or 5 or 6&gt;</p> <p>Specify which property (as column number) to be plotted as the Y-axis (versus temperature as X-axis) as a graph on the screen. Simultaneously, all tabulated properties and the Y-axis setting (i.e. the plotted column) for the graph is written into an *.EXP file using the DATAPLOT format. The default column 2 is heat capacity, 3 enthalpy, 4 entropy and 5 Gibbs energy, and the additional</p>

---

Syntax	<b>ENTER_REACTION</b>
	column 6 is a user-entered function. The table displays on the screen. The plot then displays on the screen, followed by a <code>POST</code> prompt. The POST module (postprocessor) automatically opens, and all types of the POST-module commands to refine the plotted diagram may be used. Possibilities include scaling the X/Y-axis, changing the X/Y-axis texts, etc. The command <code>BACK</code> or <code>EXIT</code> at the <code>POST</code> prompt always takes you back to the TAB module.

## LIST\_SUBSTANCES



This command is for the TABULATE\_REACTION (TAB) module.

List all species in the current database with a certain set of elements. This is useful to specify species as pure substances in the command [TABULATE\\_SUBSTANCE](#).

Syntax	LIST_SUBSTANCES
Prompts	<p>With elements /*/ &lt;*&gt; or a set of elements&gt;</p> <p>The elements building up various species must be specified. The wildcard * means all elements in the current database.</p> <p>If some elements are specified, it also prompts for how to list species with such elements.</p>
	<p>Exclusively with those elements /Y/:</p> <p>All species (not phases) in the current database are searched. Keep the default (Y) if you do not want other elements included in the search except for those specified. Enter N to list all species containing at least one of the specified elements.</p>

### Example

For the SSOL database, there are different lists, as shown below:

```
TAB: l-sub
With elements /*/: <Enter>
VA                AG                AL
AM                AS                AU
B                 BA                BE BI BR C
CA                CD                CE
CL                CO                CR
CS                CU                DY
ER                EU                F
FE                GA                GD
GE                H                 HF
HG                HO                I
IN                IR                K
LA                LI                LU
MG                MN                MO
NA                NB
```

ND	NI	NP
OS	P	
PA	PB	PD
PR	PT	PU
RB	RE	RH
RU	S	SB
SC	SE	SI
SM	SN	SR
TA	TB	TC
TE	TH	TI
TL	TM	U
V	W	Y
YB	ZN	ZR
C1	C2	C3
C4	C5	C6
C7	N2	O2
P1	P2	P4
NA/+1	K/+1	CL/-1
H2O	H1/+	LI1/+
CS1/+	RB1/+	O1H1/-
F1/-	BR1/-	I1/-
H1/+1	H1O1/-1	SI1O2 CA1O1 TAB:

TAB: l-sub

With elements /\*/: fe cr ni c n o

Exclusively with those elements? /Y/: <Enter>

C	CR	FE
N	NI	O
C	C2	C3
C	C5	C6
C	N2	O2

TAB:l-sub fe cr ni c n o

Exclusively with those elements? /Y/: n

C	CR	FE N	NI O
C1	C2	C3	
C4	C5	C6	
C7	N2	O2	

H2O

O1H1/-

H1O1/-1 SI1O2 CA1O1 TAB:

---

## SET\_ENERGY\_UNIT

Set the energy unit as calories or Joule, in all the subsequent outputs (tables, graphs, and files) in the current calculation operation.

Syntax	SET_ENERGY_UNIT
Prompt	Energy unit (C=Cal, J=Joule) /J/: <Unit>

## SET\_PLOT\_FORMAT



This command is for the TABULATE\_REACTION (TAB) module.

Set the plotting format while the result is plotted on screen or the file is saved as EXP file [using DATAPLOT graphical format] and TAB file (as a basic text file and as displayed on screen).



This command is different from SET\_PLOT\_FORMAT in the POST module, but is identical to SET\_PLOT\_ENVIRONMENT in the SYS (system utility) monitor.

Syntax	SET_PLOT_FORMAT
Options	<p>The default set <code>Unit</code> is 1 (Windows) or 9 (Linux) for graphical presentation on screen, or all other units for outputs as *.EXP and *.TAB files.</p> <p>Also set the format of the graphical output to different graphical devices. The default unit 22 (1 for legacy Windows, and 9 for legacy Linux) may be altered with SET_PLOT_ENVIRONMENT in the SYS monitor or by the TC.INI file.</p>

## SWITCH\_DATABASE



This command is for the TABULATE\_REACTION (TAB) module.

By default, the TAB module always automatically retrieves thermodynamic data for substances or reactant/product substances in a defined reaction, or for pure phases or ideal solution phases, from the SSUB Substances database. However, you can select thermodynamic data for complex non-ideal solution phases from another database, which is possible with this command.

Syntax	SWITCH_DATABASE
Prompt	<p>Database /XYZ/: &lt;new database name&gt;</p> <p>Specify a New database name. XYZ stands for the default SSUB database or the current database which is switched on.</p> <p>If an appropriate database name is not given and the USER option is not used, a list displays of available databases.</p> <p>Use one of these databases:</p> <ul style="list-style-type: none"> <li>• PURE = SGTE Pure Elements Database</li> <li>• SSUB = SGTE Substances Database</li> <li>• USER = user defined database</li> </ul>

This command switches (or changes) from the current database to a new one, and reinitializes the entire TAB module for defining a substance or reaction and the GIBBS workspace for storing retrieved data. All the directly connected databases as predefined by the original database initiation file `tc_initd.tdb` are listed by pressing the <Enter> key without giving any argument. You can also supply your own database by giving the argument USER and then the database name and, if it is not located in the current working directory, its correct path.

This command (when required to use another database for subsequent tabulations) must be used before executing the TAB commands TABULATE\_SUBSTANCE for substances, and ENTER\_REACTION or TABULATE\_REACTION for reactions.

When a predefined database or an appropriate USER database is switched on, the DATA command GET\_DATA is automatically executed.

**DATABASE NAME:** The new database is specified by typing the abbreviation given in the front of one of the available predefined databases. For convenience when switching/appending databases you can add them to the predefined database list in the database initiation file `tc_initd.tdb` of your installed software.

---

When this command is used in a macro (\*.TCM) file, if the `USER` option is selected, the database setup file name (\*setup.TDB) containing the setup definitions of the user database, and its correct path, must be provided.

If a `USER` database name or its path is not given on the same line of the `SWITCH_DATABASE` command, or if it is incomplete or incorrect, an Open window displays to specify the filename of the database setup file name (\*setup.TDB) of the to-be-switched `USER` database.



Unlike in the TDB module, if a user database is chosen, the Gibbs energy system is also reinitialized and only data from the user databases are retrieved for tabulations.

After this command, those commands to define systems (in terms of either elements or species), to reject/restore phases or species, retrieve data, as well as to append data from additional database(s), can be done.

## TABULATE



This command is for the TABULATE\_REACTION (TAB) module.

Gives the values of an already-defined table (with various properties varied with the stepping axis variable) from equilibria calculated with a STEP command. You can tabulate a table even if the table is entered after the step calculation.

Syntax	TABULATE
Prompts	Name: <Name of a defined table> The name of the table must be given. The table must be entered.
	Output to Screen or file /Screen/: <File name> The table can be listed on the screen (by pressing <Enter>), on a file (the file name must be specified here).

## TABULATE\_DERIVATIVES



This command is for the TABULATE\_REACTION (TAB) module.



This command is mainly for system debugging purposes and for experienced users. It automatically calculates all partial derivatives for a pure phase or solution phase at a given composition.



This is not the same as the chemical potential.

Before using this command, the system must be defined and thermodynamic data retrieved from an appropriate solution database in the TDB module. The phase name can be given in either upper, lower or mixed cases, and can be abbreviated until it is unique, e.g. fcc, BCC, cem, Liq, etc. as the SSOL database is used for the tabulations, or GAS, FE-S, wustite, Fe2O3-hem, etc. as the SSUB database is used. The module then optionally goes through a specific sublattice or all the possible sublattices for the specified phase, as well as through the whole framework of a currently defined system (including all the defined elements/species, as well as the default defined vacancy and electron if necessary for the current database), when it prompts for inputs of site fractions for various species possibly occupying such sublattices. Therefore, thermodynamic properties can be tabulated for a pure phase, or for a solution phase as a pure end-member (with a composition definition for the corresponding non-interacting species on all necessary sublattices in the end-member), or for a solution phase as a real solution (with a composition definition for related interacting species on all necessary sublattices in the solution).

Syntax	TABULATE_DERIVATIVES
Prompts	<p>Phase name /XXXX/: &lt;Name of the phase&gt;</p> <p>Specify the phase name of a pure phase or a solution phase.</p> <p>XXXX is the last phase the TAB module accounted for. It automatically prompts for other options and necessary inputs for the definition of the composition of the specified phase.</p> <p>For a pure phase (such as Fe-S, Wustite, and Fe2O3-Hematite), no further composition definition is needed. For a solution phase with one sublattice (such as AQUEOUS solution, GAS mixture and SLAG solution), it requests <math>n-1</math> site-fraction inputs for the phase constituents (if there are <math>n</math> species in the defined phase within the whole framework of currently defined system, including all the defined elements/species, as well as the default defined vacancy and electron if necessary for the current database); the <math>n</math>th species is automatically assigned as the rest.</p>

Syntax

TABULATE\_DERIVATIVES



The sum of input site fractions must not exceed unity.

### Example Prompts

For example, these prompts and inputs can be seen for a LIQUID solution phase (from the SSOL database) within the Fe-Cr-Ni-C-N-O framework (Note that by default, the unprompted Ni species are assigned as the rest):

```
FRACTION OF CONSTITUENT (RETURN FOR PROMPT): <RETURN> C /1/: .05
CR /1/: .1
FE /1/: < RETURN>
SUM OF FRACTIONS EXCEED UNITY, PLEASE REENTER
FE /1/: .8
N /1/: .005
```

For a solution phase with two or more sublattices (such as FCC alloy solution, and ION\_LIQ ionic liquid solution), it first asks for which sublattice the site-fractions of constituents shall be specified: the default value 0 for all possible sublattices, and a given positive number for a specific sublattice (of course, this number must be reasonable for the currently specified phase, i.e. it must be smaller than the total sublattice number of the phase). Then, for the given sublattice or for all sublattices, it prompts for all the necessary inputs of site-fractions for the possible constituents on each sublattice n-1 times if there are n species on such a sublattice in the defined phase; the nth species on each of the sublattices is automatically assigned as the rest. For example, the following prompts and inputs can be seen for an FCC solution phase (from the SSOL database) within the Fe-Cr-Ni-C-N-O framework (Note that by default, the unprompted O species on its sublattice 1 and VA on its sublattice 2 are assigned as the rest):

```
SPECIFY SUBLATTICE (0 FOR ALL) /0/: <RETURN>
FRACTIONS IN SUBLATTICE 1
CR /1/: .1
FE /1/: .8
NI /1/: .0995
FRACTIONS IN SUBLATTICE 2
C /1/: .05
N /1/: .05
```

```
Temperature /2000/: <Temperature of interest, in K>
```

Specify the temperature of interest in K.

```
Pressure /100000/: <Pressure of interest, in pa>
```

Specify the pressure of interest in Pa.

## Example Output

For the FCC phase in the Fe-Cr-Ni-C-N-O system (using the SSOL database), the following table is obtained at a specified composition:

```
TAB: TAB_DER
Phase name /BCC/: FCC
SPECIFY SUBLATTICE (0 FOR ALL) /0/: <Enter>
FRACTIONS IN SUBLATTICE 1
CR /1/: .1
FE /1/: .8
NI /1/: .3
SUM OF FRACTIONS EXCEED UNITY, PLEASE REENTER
NI /0/: .0995
FRACTIONS IN SUBLATTICE 2
C /1/: .05
N /1/: .05
Temperature /1800/: <Enter>
Pressure /100000/: <Enter>
Gibbs energy: ..... -1.27432533E+05
Helmholz energy: ..... -1.27433205E+05
Enthalpy: ..... 5.95773994E+04
Internal energy: ..... 5.95767279E+04
Entropy: ..... 1.03894407E+02
Molar volume: ..... 6.71473258E-06
Thermal expansivity: ..... 7.63886416E-05
Isothermal compressibility: ..... 6.02925387E-12
Heat capacity at constant pressure: 4.33555074E+01
First partial derivative with respect to CR in sublattice 1 of Gibbs energy:
..... -1.26034739E+05 of enthalpy: .....
4.63000206E+04 of entropy: ..... 9.57415334E+01 of
volume: ..... 6.87203263E-06
Second partial derivative of Gibbs energy with respect to also
CR in 1: 1.54392858E+05
FE in 1: -1.53353158E+04
NI in 1: -1.71750366E+04
O in 1: 0.00000000E+00
```

---

```
C                in 2: -1.82016870E+05
in 2: -3.73062665E+05
VA                in 2: -9.36260878E+04
first partial derivative with respect to FE in sublattice 1      of Gibbs energy:
..... -1.02869265E+05
      of enthalpy: ..... 6.11738912E+04      of entropy:
..... 9.11350866E+01      of volume: .....
7.53441165E-06
Second partial derivative of Gibbs energy with respect to also
FE                in 1: 1.82508696E+04
NI                in 1: -3.07043434E+03
in 1: 0.00000000E+00
C                in 2: -1.36027071E+05
in 2: -2.13007485E+05
VA                in 2: -1.11741180E+05
First partial derivative with respect to NI in sublattice 1      of Gibbs energy:
..... -1.32427029E+05      of enthalpy: .....
5.21563580E+04      of entropy: ..... 1.02546326E+02      of
volume: ..... 0.00000000E+00
      Second partial derivative of Gibbs energy with respect to also
NI                in 1: 1.48390257E+05
in 1: 0.00000000E+00
C                in 2: -7.70620431E+04
in 2: -1.61551726E+05
VA                in 2: -1.12772206E+05
First partial derivative with respect to O in sublattice 1      of Gibbs energy:
..... -2.62929308E+05      of enthalpy: .....
5.02555370E+04      of entropy: ..... 1.73991581E+02      of
volume: ..... 0.00000000E+00
Second partial derivative of Gibbs energy with respect to also
in 1: 2.99322360E+07
C                in 2: 0.00000000E+00
N                in 2: 0.00000000E+00
VA                in 2: -1.82377137E+05
First partial derivative with respect to C in sublattice 2      of Gibbs energy:
..... -1.59508417E+05      of enthalpy: .....
1.21269096E+05      of entropy: ..... 1.55987507E+02      of
volume: ..... 1.06885187E-05
Second partial derivative of Gibbs energy with respect to also
C                in 2: 2.99322360E+05
N                in 2: -1.90144000E+04
```

---

---

VA in 2: -2.78465070E+04

First partial derivative with respect to N in sublattice 2 of Gibbs energy:  
..... -2.39671400E+05 of enthalpy: .....  
4.04456947E+04 of entropy: ..... 1.55620608E+02 of  
volume: ..... 0.00000000E+00

Second partial derivative of Gibbs energy with respect to also

N in 2: 2.99322360E+05

VA in 2: -3.30383665E+04

First partial derivative with respect to VA in sublattice 2 of Gibbs energy:  
..... -9.52042686E+04 of enthalpy: .....  
5.37142294E+04 of entropy: ..... 8.27324989E+01 of  
volume: ..... 6.86700739E-06

Second partial derivative of Gibbs energy with respect to also

VA in 2: 1.66290200E+04

## TABULATE\_REACTION



This command is for the TABULATE\_REACTION (TAB) module.

Tabulate thermodynamic property changes for a chemical reaction as a table, or variation of one chosen property can be plotted against temperature as a graph, or saved as an experimental file (with an extension of .EXP). The thermodynamic data are automatically retrieved from the SSUB5 database (by default), or the current database set by the SWITCH\_DATABASE command (thus there is no need to use the DATA module before this command). You can select any other database by using SWITCH\_DATABASE.

It is important to remember the rules to specify a reaction:

- The reaction is specified, by giving the reactants and products, such as  $\text{Ga}+\text{S}=\text{GaS}$ ,  $5\text{Al}+2\text{Fe}=\text{Al}_5\text{Fe}_2$ .
- Under Linux environments, the notation for a chemical formula treats upper and lower cases as the same (such as  $\text{ga}+\text{s}=\text{GaS}$ ,  $5\text{Al}+2\text{Fe}=\text{al}_5\text{fe}_2$ ); but under Windows, the TAB module only accepts upper cases, e.g.  $\text{GA}+\text{S}=\text{GAS}$ ,  $5\text{AL}+2\text{FE}=\text{AL}_5\text{FE}_2$ .
- Elements with a single letter symbol must be followed by a stoichiometry factor, even if it is 1; thus, the symbol  $\text{CO}$  is taken as cobalt, carbon monoxide must be given as  $\text{C1O1}$  (or  $\text{O1C1}$ ).
- All the reactants and products must be defined as species-stoichiometries in the currently-used database; in other words, the reaction definition only accepts already-defined species-stoichiometries as reactants and products. For example, if a database does not have a species defined with a stoichiometry of  $\text{Al}_5\text{Fe}_2$  {e.g. the Al and Fe (but not  $\text{Al}_5\text{Fe}_2$ ) may be defined as species, and the  $\text{Al}_5\text{Fe}_2$  phase may be defined as a two-sublattice stoichiometric phase  $[\text{Al}]_5[\text{Fe}]_2$ , rather than a single sublattice stoichiometric phase  $[\text{Al}_5\text{Fe}_2]$ , then the reaction  $5\text{Al}+2\text{Fe}=\text{Al}_5\text{Fe}_2$  cannot be tabulated.
- Make sure that the Gibbs free energy expression for a stoichiometric phase is really corresponding to the species-stoichiometry. For example, if a database has a definition of Function  $\text{GAL}_5\text{FE}_2$  for 1/7 of the  $\text{Al}_5\text{Fe}_2$  formula, then the relation  $\text{G}(\text{Al}_5\text{Fe}_2)=7*\text{GAL}_5\text{FE}_2$  must be defined; otherwise, the tabulated thermodynamic properties for the reaction  $5\text{Al}+2\text{Fe}=\text{Al}_5\text{Fe}_2$  is wrong.

Syntax	TABULATE_REACTION
Prompts	Same reaction? /Y/:
	This prompt displays if there is at least one reaction already defined either with the commands ENTER_REACTION or TABULATE_REACTION. The following prompts only display if N is entered.
	Reaction: <Chemical reaction equation> & <RA+RB=PC+PD>

Syntax	TABULATE_REACTION
	<p>A long reaction can be typed over several lines. A reaction should be terminated by a semicolon (;) or an empty line. A list showing the used database, defined elements, getting-data sequence, references, etc.</p> <pre>Pressure /100000/: &lt;pressure of interest, in pa&gt; Low temperature limit /298.15/: &lt;T-low, in K&gt; Specify the starting temperature K. High temperature limit /298.15/: &lt;T-high, in K&gt; Specify the ending temperature K. Step in temperature /100/: &lt;T-step&gt; Specify the temperature step for the tabulation.</pre>
	<p>Output file /Screen/: &lt;Return for screen, or type a file name&gt;</p> <p>If you type <code>Return for screen</code>, a list of thermodynamic property changes for the defined/chosen reaction is shown as a table (which is demonstrated in the example outputs given below), and the command is terminated.</p> <p>If typing a file name, the table is both displayed on screen and saved as an *.EXP or a *.TAB file, and the program prompts:</p>
	<p>Graphical output /Y/:</p> <ul style="list-style-type: none"> <li>For N a table is output to the screen, and the same table is saved as a basic text file with the default extension .TAB under the current working directory. In this case, the graph cannot be plotted.</li> <li>For Y a table is created with all thermodynamic functions as normal (which is displayed on screen) and generates a graph with temperature on the X-axis and a chosen property on a certain column in the table on the Y-axis (which is plotted on screen and saved as an *.EXP file), and it also asks which column is to be plotted on the resulting graph:</li> </ul>
	<p>Plot column? /2/: &lt;1 or 2 or 3 or 4 or 5 or 6&gt;</p> <p>Specify which property (as column number) to be plotted as the Y-axis (versus temperature as X-axis) as a graph on the screen. Simultaneously, all tabulated properties and the Y-axis setting (i.e. the plotted column) for the graph is written into an *.EXP file using the DATAPLOT format. The default column 2 is heat capacity, 3 enthalpy, 4 entropy and 5 Gibbs energy, and the additional column 6 is a user-entered function. The table displays on the screen. The plot then displays on the screen, followed by a POST: prompt. The POST module automatically opens, and all types of the POST module commands to refine the plotted diagram may be used. Possibilities include scaling the X/Y-axis, changing the X/Y-axis texts, etc. The command BACK or EXIT at the POST: prompt always takes you back to the TAB module.</p>

## Example

For the reaction  $Ga+S=GaS$  (using the SSUB database), the following table is obtained by typing `<Enter>` in the prompt `Output file /SCREEN/`, and the figure by typing `2` in the prompt `Plot column /2/`:

```

O U T P U T   F R O M   T H E R M O - C A L C
Reaction:  S+GA=GA1S1
stable as S_S
GA stable as GA_S
```

---

GA1S1 stable as GA1S1\_S

Delta-Cp (K)	Delta-H (Joule/K)	Delta-S (Joule)	Delta-G (Joule/K)	Delta-G (Joule)
298.15	-2.79489E+00	-2.09200E+05	-1.50580E+01	-2.04710E+05
300.00	-2.87516E+00	-2.09205E+05	-1.50755E+01	-2.04683E+05
302.	---- GA becomes GA_L ,delta-H = 5589.80			
367.	---- S becomes S_S2 ,delta-H = 401.00			
389.	---- S becomes S_L ,delta-H = 1721.00			
400.00	-1.22278E+01	-2.17521E+05	-4.07488E+01	-2.01222E+05
500.00	-1.57686E+01	-2.19323E+05	-4.47280E+01	-1.96959E+05
600.00	-1.02518E+01	-2.20583E+05	-4.70410E+01	-1.92359E+05
700.00	-6.58954E+00	-2.21430E+05	-4.83534E+01	-1.87583E+05
800.00	-4.46246E+00	-2.21963E+05	-4.90686E+01	-1.82709E+05
900.00	-2.82671E+00	-2.22328E+05	-4.94995E+01	-1.77778E+05
1000.00	-1.21787E+00	-2.22530E+05	-4.97138E+01	-1.72816E+05
1100.00	3.71702E-01	-2.22572E+05	-4.97552E+01	-1.67841E+05

\*\*\*\*\*

## TABULATE\_SUBSTANCE



This command is for the TABULATE\_REACTION (TAB) module.

Tabulate thermodynamic properties of a substance (with a given chemical formula but it is not certain in which phase/state it may form) or a pure stoichiometric phase (getting data from a specific substance database such as SSUB, or from a specific solution database such as SSOL and TCFE), or of a solution phase with a fixed composition (getting data from a specific solution database such as SSOL and TCFE), under a constant pressure and various temperatures.

In case of a pure stoichiometric phase or a solution phase with a fixed composition, you must already define the system and get thermodynamic data from an appropriate solution database in the TDB module before using this TAB command. The phase name can be given either upper or lower or mixed cases, and can be abbreviated until it is unique, e.g. *fcc*, *BCC*, *Liq*, *cem*, *Al5Fe2*, etc. as the SSOL database is used for the tabulations, or *GAS*, *FE-S*, *wustite*, *Fe2O3-hem*, etc. as the SSUB database is used. The module then optionally goes through a specific sublattice or all the possible sublattices for the specified phase, as well as through the whole framework of a currently defined system (including all the defined elements/species, as well as the default defined vacancy and electron if necessary for the current database), when it prompts for inputs of site fractions for various species possibly occupying such sublattices. Therefore, you can tabulate thermodynamic properties for a pure phase, or for a solution phase as a pure end-member (with a composition definition for the corresponding non-interacting species on all necessary sublattices in the end-member), or for a solution phase as a real solution (with a composition definition for related interacting species on all necessary sublattices in the solution).

Moreover, the variation for one of the tabulated properties for a substance, a pure stoichiometric phase, or a solution phase with a fixed composition can be plotted against temperature as a graph or saved as an experimental file (with an extension of .EXP).

Syntax	TABULATE_SUBSTANCE
Prompt	<p>Substance (phase): &lt;Name of the species or phase&gt;</p> <p>In case of a substance, give its chemical formula, e.g. Fe, H2, C1H6, FeC1, CaC1O3, MgSiO3, etc. When the TABULATE_SUBSTANCE &lt;SUBSTANCE&gt; command sequence is used for the first time in the TAB module, it always uses the SSUB Substances database as the default. If SSUB is not available and a specific database has not been switched on yet, it may prompt to specify an appropriate substance or solution database (e.g. typing SSUB&lt;version number&gt;; SSOL&lt;version number&gt;; TCFE&lt;version number&gt; etc.). Prior to this command, you can also use SWITCH_DATABASE to set the current database. If a solution database is set as the current database, only the neutral species that are valid substance standing by themselves as phases can be tabulated.</p>

## Syntax

## TABULATE\_SUBSTANCE



Before other prompts, a list of the used database, defined elements, getting-data sequence, references, etc. is displayed on screen. From such information, you know what the TAB module is performing.

In case of a pure stoichiometric phase or a solution phase, give its phase name, e.g. FCC, CEMENTITE, LIQUID, SLAG, AQUEOUS, GAS, Al<sub>5</sub>Fe<sub>2</sub>, Fe-S, Wustite, Fe<sub>2</sub>O<sub>3</sub>-Hematite, etc.



**REMEMBER:** It is important that you have already defined the system and gotten thermodynamic data from an appropriate solution database in the TDB module before using this TAB\_SUB command. It automatically prompts for other options and necessary inputs for the definition of the composition of the specified phase.

For a pure stoichiometric phase defined with one single sublattice (such as Fe-S, Wustite, and Fe<sub>2</sub>O<sub>3</sub>-Hematite), no further composition definition is needed.

For a solution phase defined with one sublattice (such as AQUEOUS solution, GAS mixture and SLAG solution), it requests  $n-1$  site-fraction inputs for the phase constituents (if there are  $n$  species in the defined phase within the whole framework of currently defined system, including all the defined elements/species, as well as the default defined vacancy and electron if necessary for the current database); the  $n$ th species is automatically assigned as the rest.



The sum of input site fractions must not exceed unity. For example, the following prompts and inputs can be seen for a LIQUID solution phase (from the SSOL database) within the Fe-Cr-Ni-C-N-O framework (Note that by default, the unprompted Ni species is assigned as the rest):

```
Fraction of constituent (return for prompt): <Return>
```

```
C /1/: .05
CR /1/: .1
FE /1/: <RETURN>
```

```
Sum of fractions exceed unity, please reenter
```

```
FE /1/: .8
N /1/: .005
```

For a pure stoichiometric phase defined with two or more sublattices (such as Al<sub>5</sub>Fe<sub>2</sub> inter-metallic stoichiometric phase) and for a solution phase defined with two or more sublattices (such as FCC alloy solution, and ION\_LIQ ionic liquid solution, Al<sub>5</sub>Fe<sub>4</sub> inter-metallic solution phase), it first asks for which sublattice the site-fractions of constituents shall be specified: the default value 0 for all possible sublattices, and a given positive number for a specific sublattice (this number must be reasonable for the currently specified phase, i.e. it must be smaller than the total sublattice number of the phase). Then, for the given sublattice or for all sublattices, it prompts for all the necessary inputs of site-fractions for the possible constituents on each sublattice  $n-1$  times if there are  $n$  species on such a sublattice in the defined phase; the  $n$ th species on each of the sublattices are automatically assigned as the rest. For example, these prompts and inputs can be seen for an FCC

Syntax	TABULATE_SUBSTANCE
	<p>solution phase (from the SSOL database) within the Fe-Cr-Ni-C-N-O framework (Note that by default, the unprompted <code>O</code> species on its sublattice 1 and <code>VA</code> on its sublattice 2 is assigned as the rest):</p>
	<pre>Specify sublattice (0 for all) /0/:   FRACTIONS IN SUBLATTICE          1   CR /1/: .1   FE /1/: .8   NI /1/: .0995   FRACTIONS IN SUBLATTICE          2   C /1/: .05   N /1/: .05</pre>
	<pre>Pressure /100000/: &lt;Pressure of interest, in pa&gt;</pre> <p>Specify the constant pressure condition in Pa.</p>
	<pre>Low temperature limit /298.15/: &lt;T-low, in K&gt;</pre> <p>Specify the starting temperature K.</p>
	<pre>High temperature limit /298.15/: &lt;T-high, in K&gt;</pre> <p>Specify the ending temperature K.</p>
	<pre>Step in temperature /100/: &lt;T-step&gt;</pre> <p>Specify the temperature step for the tabulation.</p>
	<pre>Output file /Screen/: &lt;Return for screen, or type a file name&gt;</pre> <p>If you type <code>Return for screen</code> a list of basic thermodynamic functions for the chosen substance or for the specified pure or solution phase with the fixed composition is shown up as a table (which is demonstrated in the example outputs given below), and the command is terminated.</p> <p>If typing a file name, the table is both displayed on screen and saved as an <code>*.EXP</code> or a <code>*.TAB</code> file, and the program further prompts:</p>
	<pre>Graphical output /Y/:</pre> <ul style="list-style-type: none"> <li>For <code>N</code> a table is output to the screen, and the same table is saved as a basic text file with the default extension <code>.TAB</code> under the current working directory. In this case, the graph cannot be plotted.</li> <li>For <code>Y</code> a table is created with all the thermodynamic functions as normal (which is displayed on screen) as well as generates a graph with temperature on the X-axis and a chosen property on a certain column in the table on the Y-axis (which is plotted on screen and saved as an <code>*.EXP</code> file), and it also ask which column is to be plotted on the resulting graph:</li> </ul>
	<pre>Plot column ? /2/: &lt;1 or 2 or 3 or 4 or 5 or 6&gt;</pre> <p>Specify which property (as column number) to be plotted as the Y-axis (versus temperature as X-axis) as a graph on the screen. Simultaneously, all tabulated properties and the Y-axis setting (i.e. the plotted column) for the graph is written into an <code>*.EXP</code> file using the DATAPLOT format. The default column 2 is heat capacity, 3 enthalpy, 4 entropy and 5 Gibbs energy, and the additional column 6 is a user-entered function. The table displays on the screen. The plot then displays on the screen, followed by a <code>POST:</code> prompt. The <code>POST</code> module (postprocessor) automatically opens, and all types of the <code>POST</code>-module commands to refine the plotted diagram may be used. Possibilities include scaling the X/Y-axis, changing the X/Y-axis texts, etc. The command <code>BACK</code> or <code>EXIT</code> at the <code>POST:</code> prompt always takes you back to the <code>TAB</code> module.</p>

## Example 1

For the FCC alloy phase as a non-ideal solution with a fixed composition

```
[Fe0.80,Cr0.10,Ni0.0995,O0.005][C0.05,N0.05,VA0.90]
```

The following table is obtained by typing `Return` for screen at the prompt `Output file /SCREEN/:`

```

O U T P U T   F R O M   T H E R M O - C A L C
Phase : FCC                               Pressure :      100000.00
Specie: CR1/--2
*****
      T      Cp      H      S      G
      (K)    (Joule/K)  (Joule)  (Joule/K)  (Joule)
*****
298.15  2.70517E+01  6.23824E+03  4.40241E+01  -6.88755E+03
300.00  2.70889E+01  6.28832E+03  4.41916E+01  -6.96915E+03
400.00  2.87304E+01  9.08420E+03  5.22235E+01  -1.18052E+04
500.00  2.99904E+01  1.20222E+04  5.87742E+01  -1.73649E+04
600.00  3.10889E+01  1.50770E+04  6.43408E+01  -2.35275E+04
700.00  3.21116E+01  1.82375E+04  6.92106E+01  -3.02100E+04
800.00  3.30994E+01  2.14982E+04  7.35633E+01  -3.73524E+04
900.00  3.40742E+01  2.48569E+04  7.75182E+01  -4.49094E+04
1000.00  3.50483E+01  2.83130E+04  8.11586E+01  -5.28456E+04
1100.00  3.60268E+01  3.18667E+04  8.45449E+01  -6.11327E+04
1200.00  3.70143E+01  3.55187E+04  8.77219E+01  -6.97476E+04
1300.00  3.80149E+01  3.92700E+04  9.07241E+01  -7.86713E+04
1400.00  3.90311E+01  4.31222E+04  9.35784E+01  -8.78875E+04
1500.00  4.00649E+01  4.70768E+04  9.63064E+01  -9.73827E+04
1600.00  4.11174E+01  5.11358E+04  9.89256E+01  -1.07145E+05
1700.00  4.21896E+01  5.53010E+04  1.01450E+02  -1.17165E+05
1800.00  4.33555E+01  5.95774E+04  1.03894E+02  -1.27433E+05
1900.00  4.58528E+01  6.40379E+04  1.06306E+02  -1.37943E+05
2000.00  4.75402E+01  6.87138E+04  1.08704E+02  -1.48694E+05

```

## Example 2



For a phase, as either an end-member or real solution, the listed species name is irrelevant.

For the H2 species as a pure substance (using the SSUB5 database), the following table is obtained by typing `Return` for `screen` at the prompt `Output file /SCREEN/`, whilst the figure is generated by typing `5` at the prompt `Plot column /2/`:

```

O U T P U T F R O M T H E R M O - C A L C
Phase : GAS                      Pressure :      100000.00
Specie: H2
*****
H          S          G          T          Cp
(K)      (Joule/K)    (Joule)    (Joule/K)    (Joule)
*****
2.88369E+01  3.17684E-06  1.30680E+02  -3.89622E+04  298.15
300.00  2.88473E+01  5.33580E+01  1.30858E+02  -3.92042E+04
400.00  2.91591E+01  2.95686E+03  1.39209E+02  -5.27268E+04
500.00  2.92650E+01  5.87874E+03  1.45729E+02  -6.69856E+04
600.00  2.93441E+01  8.80908E+03  1.51071E+02  -8.18336E+04
700.00  2.94579E+01  1.17488E+04  1.55602E+02  -9.71730E+04
800.00  2.96320E+01  1.47027E+04  1.59547E+02  -1.12935E+05
900.00  2.98786E+01  1.76776E+04  1.63050E+02  -1.29068E+05
1000.00  3.02043E+01  2.06810E+04  1.66214E+02  -1.45533E+05
1100.00  3.05319E+01  2.37171E+04  1.69108E+02  -1.62302E+05
1200.00  3.09281E+01  2.67897E+04  1.71781E+02  -1.79348E+05
1300.00  3.13615E+01  2.99040E+04  1.74274E+02  -1.96652E+05
1400.00  3.18115E+01  3.30625E+04  1.76614E+02  -2.14197E+05
1500.00  3.22641E+01  3.62664E+04  1.78824E+02  -2.31970E+05
1600.00  3.27094E+01  3.95151E+04  1.80921E+02  -2.49958E+05
1700.00  3.31406E+01  4.28078E+04  1.82917E+02  -2.68151E+05
1800.00  3.35526E+01  4.61426E+04  1.84823E+02  -2.86539E+05
1900.00  3.39415E+01  4.95175E+04  1.86648E+02  -3.05113E+05
2000.00  3.43045E+01  5.29300E+04  1.88398E+02  -3.23866E+05

```

---

## REACTOR (REACTOR\_SIMULATOR) Commands

---

### In this section:

---

CREATE_DIVIDERS .....	485
CREATE_PIPES .....	486
CREATE_STAGE_BOXES .....	487
EXECUTE_POLY3_COMMAND .....	489
LIST_RECORDS .....	490
READ_WORKSPACE .....	491
SAVE_WORKSPACES .....	492
START_SIMULATION .....	493

## CREATE\_DIVIDERS



This command is for the REACTOR\_SIMULATOR (REACTOR) module.

Define the segment boundaries and the ways on how to distribute phases between various segments and to split the feeds of heat and mass.

Syntax	CREATE_DIVIDERS
Prompts	<p>Number of dividers /4/: &lt;The number of dividers&gt;</p> <p>Specify the number of dividers (boundaries between various segments) in the reactor. It is normally the segment number plus one: for example, if the stage box number is defined as 4, the divider number should be defined as 5.</p>
	<p>Number of outputs for divider # /3/: &lt;The number of outputs&gt;</p> <p>Specify the number of outputs for a specific divider #.</p>
	<p>Percent of input to output 1 /100/: &lt;Percent of input&gt;</p> <p>Define the percentage of input to the output 1 in the current divider #.</p>
	<p>Percent of input to output ## /100/: &lt;Percent(s) of input(s)&gt;</p> <p>Define the percentages of inputs to the outputs ## in the current divider #. It is repeated till the last output is specified.</p>

## CREATE\_PIPES



This command is for the REACTOR\_SIMULATOR (REACTOR) module.

Define the pipes between various segments and the ways on how the mass and heat transport between the surroundings (with the record number as 0) and the reactor, among various segments in the reactor, and along various dividers (segment boundaries in the reactor).

Syntax	CREATE_PIPES
Prompts	<p>From record: &lt;A record number&gt;</p> <p>Specify a record number. If there are four segments in the reactor, these records are available:</p> <ul style="list-style-type: none"> <li>• 0 surroundings</li> <li>• 1-4 stage boxes (segments)</li> <li>• 5-9 dividers (segment boundaries)</li> </ul>
	<p>Pipe to record: &lt;Another record number&gt;</p> <p>Specify another record number to which the mass or heat flows from the current record (specified above). Before this prompt, a message shown on the screen indicates what kind of mass or heat flow (according to the definitions of stage boxes and dividers, which have already been created prior to this command) should be specified in the pipe. This is repeated for all inputs and/or outputs for mass and/or heat in current record.</p>

## CREATE\_STAGE\_BOXES



This command is for the REACTOR\_SIMULATOR (REACTOR) module.

Create several reaction stages as boxes (segments), where the you define the feed in either mass input and initial temperature or heat input from surroundings, as well as the mass out and initial temperature in each of the segments.

Syntax	CREATE_STAGE_BOXES
Prompts	<p>Number of stage boxes /4/: &lt;The number of stage boxes&gt;</p> <p>Specify the number of stage boxes in the reactor. Then define the feed of mass and/or energy (from surroundings to) the reactor.</p>
	<p>Give feed to system: &lt;Feed&gt;</p> <p>One feed (of mass or heat) to the reactor must be specified here. Two types of feeds can be used:</p> <ul style="list-style-type: none"> <li>As input amount of heat (enthalpy): H=Value.</li> <li>As input amount of specific species or element and its initial temperature: N (SPECIES)=Value, Input temperature (in one line, or the input temperature in the next prompt).</li> </ul>
	<p>Input temperature /298.15/: &lt;Input temperature&gt;</p>
	<p>Give feed to system: &lt;Feed&gt;</p> <p>Specify other feeds (mass or energy) to the reactor. Press &lt;Enter&gt; to finish the feed inputs, and then:</p> <p>Give for stage box #</p> <p>Now start to define the type of stage box (segment is specifically numbered as #), and to specify a guess of initial temperature and an output for each phase in the segment.</p>
	<p>Type of box/Equilibrium/: &lt;Type of box&gt;</p> <p>Only the EQUILIBRIUM type is allowed. Press &lt;Enter&gt; to accept it.</p>
	<p>Is the stage heat controlled? /Y/: &lt;Yes or No&gt;</p> <p>Y means the current segment is heat controlled.</p>
	<p>Initial guess of temperature? /1000/: &lt;Initial temperature&gt;</p> <p>Give a guess of initial temperature (in K) for the current segment; press &lt;Enter&gt; to accept the default.</p>
	<p>Give initial amount: &lt;Initial amount of mass or heat&gt;</p> <p>The initial amount of mass and/or heat in the current segment can be specified here. Two types of initial amount can be used:</p> <ul style="list-style-type: none"> <li>As initial amount of heat (enthalpy): H=Value.</li> <li>As initial amount of specific species or element and its initial temperature: N (SPECIES)=Value, Input temperature (in one line, or the input temperature in the next</li> </ul>

Syntax	CREATE_STAGE_BOXES
	prompt).
	Input temperature /298.15/: <Input temperature>
	<p>Give initial amount: &lt;Initial amount of mass or heat&gt;</p> <p>Specify initial amount (mass or energy) in the current segment. Press &lt;Enter&gt; to finish the initial amount inputs, and then this message displays:</p> <p>Each phase may have a separate output, give these</p> <p>Now start to define the output for each phase.</p>
	<p>Phase name /Rest/: &lt;Name of phase&gt;</p> <p>Specify a phase that has a separate output, e.g. GAS; by pressing &lt;Enter&gt; to accept that all REST phases do not have separate outputs. Then the program asks for further definitions of all the other segments (one by one), repeating some of the above questions, till the last segment in the reactor is defined.</p>

## EXECUTE\_POLY3\_COMMAND



This command is for the REACTOR\_SIMULATOR (REACTOR) module.

Syntax	EXECUTE_POLY3_COMMAND
	To directly use all kinds of POLY commands inside the REACTOR module. Any legal POLY commands can be entered.
	 <a href="#">POLY Module Commands</a>

---

## LIST\_RECORDS



This command is for the REACTOR\_SIMULATOR (REACTOR) module.

Syntax	LIST_RECORDS
	Lists all the already-created records for the reactor (including stage boxes, dividers, pipes, etc.).

## READ\_WORKSPACE



This command is for the REACTOR\_SIMULATOR (REACTOR) module.

The REACTOR workspace (along with the related POLY and GIBBS workspaces) and the simulated results from the SIMULATE command can be read by this command from a binary file where they must be saved with SAVE\_WORKSPACE. The \*.RCT file is not printable.

Syntax	READ_WORKSPACE
Options	<p><b>File name:</b> The name of a saved *.RCT file where the REACTOR and POLY/GIBBS workspaces shall be read from must be specified. You do not need to type the extension if it is the default .RCT, otherwise type the whole *.RCT file name.</p> <p>An <b>Open</b> window displays if a file name is not given after the command or its path is incorrect, so that the path (in the <b>Look in</b> field) and <b>File name</b> can be specified. The <b>Files of type</b> (i.e. RCT) cannot be changed.</p>

## SAVE\_WORKSPACES



This command is for the REACTOR\_SIMULATOR (REACTOR) module.



The POLY module and Gibbs (GES) module also have a command with the same name.

The workspaces are saved to a file with this command. In the REACTOR workspace, all definitions of multi-stage steady-state reactor and its distribution coefficients are saved, together with the GIBBS and POLY workspaces.

After using this command, you can always return to the state when you issued the command by giving a READ command.

After saving the workspaces to a file, you can exit the software and at a later time READ the file and continue from the saved state.



START\_SIMULATION automatically saves on the work file with the most lately specified name. To avoid destroying the simulated results, do not SAVE after a START\_SIMULATE command is enforced, similar to that you should not use SAVE after a STEP or MAP command. You may append several results by START\_SIMULATE without destroying the previous results but SAVE erases them all.

Syntax	SAVE_WORKSPACES
	File name  A file name must be specified. The default extension of the RCT file is .RCT, while or any other extension as required.
	A <b>Save</b> window displays if a file name is not given after the command, so that the path (in the <b>Save in</b> field) and <b>File name</b> can be specified. The <b>Files of type</b> (i.e. RCT) cannot be changed. If there is already a file with the same file name under the directory a warning message displays. Click <b>OK</b> to overwrite the current REACTOR (and POLY3/GIBBS) workspaces onto the existing *.RCT file. Click <b>Cancel</b> to return to the REACTOR module. You can use the SAVE command with an unspecified name (i.e. through SAVE , , , command-sequence) to save the REACTOR (and POLY3/GIBBS) workspaces.

## START\_SIMULATION



This command is for the REACTOR\_SIMULATOR (REACTOR) module.

Start the reactor simulation, and list the results for all iterations. The output for each iteration consists of the conditions set in each segment, and you can also select some state variables to list. After each loop, the temperatures in all segments are listed.

Syntax	START_SIMULATION
Prompts	<p>Max number of loops /10/: &lt;The max number of loops&gt;</p> <p>Give the max number of loops. The reactor simulation repeats until this loop number is reached.</p>
	<p>Output to screen or file /Screen/: &lt;File name or return&gt;</p> <p>Decide on either listing simulation results on the screen (by pressing &lt;Enter&gt;) or saving them on an experimental file (with the default extension .EXP).</p> <div style="border: 2px solid blue; padding: 5px; margin: 10px 0;">  The conditions in each segment are not saved in the file, only on the screen.         </div>
	<p>Output variables /T BP(\$)/: &lt;Variable name(s)&gt;</p> <p>Specify the variable name(s) that is also listed on the screen together with the conditions in each segment, or in the output experimental file.</p>

### Example

For a four-stage reactor, the output at a certain iteration may look like this:

```
H=-991745.1, P=100000, N(C)=2.8468936, N(N)=6.78399966E-4, N(O)=3.8171615,
N(SI)=1.77031
DEGREE OF FREEDOM 0
BP(GAS)=43.464341, BP(C1SI1_BETA)=25.175981, BP(C_S)=8.1588099,
BP(QUARTZ_S3)=68.193896
1.78803E+03
BP(GAS)=61.971213, BP(C1SI1_BETA)=38.144346, BP(QUARTZ_L)=66.230717 2.06569E+03
BP(GAS)=67.66262, BP(C1SI1_BETA)=35.480983, BP(QUARTZ_L)=58.219292
2.08691E+03
BP(GAS)=71.23129, BP(SI_L)=22.471787
2.20831E+03
H=-924751,87, P=100000, N(C)=2.0594558, N(N)=5.17999978E-4, N(O)=4.0142358,
```

---

```
N(SI)=2.7551438
DEGREE OF FREEDOM 0
BP(GAS)=43.464341, BP(C1SI1_BETA)=25.175981, BP(C_S)=8.1588098,
BP(QUARTZ_S3)=68.193896
1.78803E+03
BP(GAS)=61.971213, BP(C1SI1_BETA)=38.144346, BP(QUARTZ_L)=66.230717 2.06569E+03
BP(GAS)=67.66262, BP(C1SI1_BETA)=35.480983, BP(QUARTZ_L)=58.219292
2.08691E+03
BP(GAS)=71.23129, BP(SI_L)=22.471787
2.20831E+03
H=-813239., P=100000, N(C)=1.6592668, N(N)=3.59999988E-4, N(O)=3.7549293,
N(SI)=2.896635
DEGREE OF FREEDOM 0
BP(GAS)=43.464341, BP(C1SI1_BETA)=25.175981, BP(C_S)=8.1588098,
BP(QUARTZ_S3)=68.193896
1.78803E+03
BP(GAS)=61.971213, BP(C1SI1_BETA)=38.144346, BP(QUARTZ_L)=66.230717 2.06569E+03
BP(GAS)=67.66262, BP(C1SI1_BETA)=35.480983, BP(QUARTZ_L)=58.219292
2.08691E+03
BP(GAS)=71.23129, BP(SI_L)=22.471787
2.20831E+03
H=7374.6403, P=100000, N(C)=8.84911857E-1, N(N)=1.99999996E-4, N(O)=1.9379203, N
(SI)=1.853872
DEGREE OF FREEDOM 0
BP(GAS)=43.464341, BP(C1SI1_BETA)=25.175982, BP(C_S)=8.1588097,
BP(QUARTZ_S3)=68.193896
1.78803E+03
BP(GAS)=61.971213, BP(C1SI1_BETA)=38.144346, BP(QUARTZ_L)=66.230717 2.06569E+03
BP(GAS)=67.66262, BP(C1SI1_BETA)=35.480983, BP(QUARTZ_L)=58.219292
2.08691E+03
BP(GAS)=71.23129, BP(SI_L)=22.471787
2.20831E+03
10 1788.03 2065.69 2086.91 2208.31 0.00 0.00 0.00
```

---

## POST Module Commands

---

The POST module (post-processor) is available from the these modules: POLY, TAB, BIN, TERN, POT, SCHEIL and POURBAIX.



[Using the POST Module](#)

### In this section:

---

ADD_LABEL_TEXT .....	498
APPEND_EXPERIMENTAL_DATA .....	501
CHANGE_LEGEND .....	502
DUMP_DIAGRAM .....	503
ENTER_SYMBOL .....	504
FIND_LINE .....	507
LIST_DATA_TABLE .....	508
LIST_PLOT_SETTINGS .....	509
LIST_SYMBOLS .....	510
MAKE_EXPERIMENTAL_DATAFILE .....	511
MODIFY_LABEL_TEXT .....	512
PLOT_DIAGRAM .....	513
PRINT_DIAGRAM .....	514
QUICK_EXPERIMENTAL_PLOT .....	515
REINITIATE_PLOT_SETTINGS .....	516
RESTORE_PHASE_IN_PLOT .....	517
SELECT_PLOT .....	518
SET_AXIS_LENGTH .....	519
SET_AXIS_PLOT_STATUS .....	520
SET_AXIS_TEXT_STATUS .....	521
SET_AXIS_TYPE .....	522

---

SET_COLOR .....	523
SET_CORNER_TEXT .....	525
SET_DIAGRAM_AXIS .....	526
SET_DIAGRAM_TYPE .....	528
SET_FONT .....	529
SET_INTERACTIVE_MODE .....	530
SET_LABEL_CURVE_OPTION .....	531
SET_PLOT_FORMAT .....	532
SET_PLOT_OPTIONS .....	533
SET_PLOT_SIZE .....	535
SET_PREFIX_SCALING .....	536
SET_RASTER_STATUS .....	537
SET_REFERENCE_STATE .....	538
SET_SCALING_STATUS .....	540
SET_TIC_TYPE .....	541
SET_TIELINE_STATUS .....	542
SET_TITLE .....	543
SET_TRUE_MANUAL_SCALING .....	544
SUSPEND_PHASE_IN_PLOT .....	545
TABULATE .....	546

---



## ADD\_LABEL\_TEXT

Add a text to an area in a phase diagram or a property diagram. The label is written starting from the specified coordinates. Optionally, you can also let the program automatically add the text by first calculating an equilibrium state at the specified coordinates and then making a label out of the stable phase names.

The label texts and coordinates, either added by this command or modified with `MODIFY_LABEL_TEXT` (or manually edited in the **Console Results** window), are stored in the workspace, and these may later be saved onto an \*.EXP file (with `MAKE_EXPERIMENTAL_DATAFILE`).



In some cases such an optional calculation for automatic phase region labeling may not work. It is not possible to automatically add labels for certain sets of axes, like entropy, enthalpy, pH, Eh, etc.



### Plot Labels

Syntax	ADD_LABEL_TEXT
Prompts	<p>Give X coordinate in axis units: &lt;Value of the X coordinate&gt;</p> <p>Specify the value of the X coordinate where the label is to start.</p>
	<p>Give Y coordinate in axis units: &lt;Value of the X coordinate&gt;</p> <div style="border: 1px solid red; padding: 5px; margin: 10px 0;"> <p> The labels disappear if any axis is changed after wards.</p> </div> <p>Specify the value of the Y coordinate where the label is to start.</p>
	<p>Automatic phase labels? /Y/:</p> <p>By default the program automatically calculates the equilibrium at the given coordinates and generates a label with the names of the stable phases. However, the automatic calculation works only for phase diagrams that are mapped with two axes in the POLY module, and sometimes it may not work correctly especially for complex heterogeneous interaction systems; in such cases, a message displays.</p>
	<p>Text: &lt;Text for the label&gt;</p> <p>If you answer N enter text at the prompt. You can enter simple text, use LaTeX, rotate the text, as well as adjust the font size.</p> <p><b>LaTeX:</b> You can use LaTeX commands to enter text and symbols. First enter the command followed by the expression or string, for example to display <math>A_{12}O_3</math> with subscripts:</p> <pre>\latex Mole-fraction Al_2O_3</pre>

Syntax	ADD_LABEL_TEXT
	<p data-bbox="381 275 435 321"></p> <p data-bbox="480 260 1235 317">For a list of supported LaTeX commands, see: <a href="https://www2.ph.ed.ac.uk/snuggletex/documentation/math-mode.html">https://www2.ph.ed.ac.uk/snuggletex/documentation/math-mode.html</a>.</p> <div data-bbox="363 365 1395 569" style="border: 1px solid red; padding: 10px;"> <p data-bbox="381 449 435 495"></p> <p data-bbox="480 409 1357 527">By default when using the LaTeX command, you are in math mode and Thermo-Calc automatically interprets the entered text as math. The above link uses the dollar sign (\$) in all the examples but this is NOT required. For example, if you enter <math>E=mc^2</math> this is displayed including the \$ instead of the equation.</p> </div> <div data-bbox="363 600 1395 741" style="border: 1px solid blue; padding: 10px;"> <p data-bbox="381 638 435 695"></p> <p data-bbox="480 638 1344 699">To add a mixture of plain text and equation text, you can exit math mode using the string <code>\textsf</code> followed by the text to display enclosed in curly brackets.</p> </div> <p data-bbox="444 768 1325 800">For example <code>\latex \textsf{Mole-fraction }Al_2O_3</code> displays this sans serif label</p> <div data-bbox="448 821 769 863" style="border: 1px solid gray; padding: 5px; display: inline-block;">Mole-fraction Al<sub>2</sub>O<sub>3</sub></div> <div data-bbox="363 905 1395 1014" style="border: 1px solid blue; padding: 10px;"> <p data-bbox="381 926 435 982"></p> <p data-bbox="480 940 1325 972">White space has to be entered in the text context and not outside in math mode.</p> </div> <p data-bbox="358 1041 1344 1129"><b>Rotate the text:</b> If you want to rotate the label, start a text string with the rotation command (which is case sensitive and must be in lower case letters) then enter the number of degrees to rotate the label. For example, to rotate the label 45 degrees enter:</p> <pre data-bbox="358 1146 509 1171">\rotation 45</pre> <p data-bbox="358 1188 1284 1220">To rotate the text and use LaTeX, the rotation command must be used first. For example:</p> <pre data-bbox="358 1236 862 1262">\rotation 45 \latex Mole-fraction Al_2O_3</pre> <p data-bbox="358 1278 1146 1310">Rotates the text 45° and includes subscripts in the title Mole-fraction Al<sub>2</sub>O<sub>3</sub>.</p>
	<p data-bbox="358 1335 826 1360">Text size: <code>/.35/:</code> &lt;Size for the label&gt;</p> <p data-bbox="358 1377 1373 1465">A smaller size of the text may be necessary to fit the label text into the diagram. A default size is given using the latest size in adding label texts, or .35 if the command is called for the first time. A recommended size is e.g. 0.25.</p> <div data-bbox="363 1497 1395 1640" style="border: 1px solid blue; padding: 10px;"> <p data-bbox="381 1539 435 1596"></p> <p data-bbox="480 1539 1308 1600">These changes to plot labels can also be manually edited in the <b>Console Results</b> window.</p> </div>
	<p data-bbox="358 1692 521 1730"><b>Examples</b></p> <p data-bbox="358 1751 776 1782">To produce a label with the unit Celsius:</p> <pre data-bbox="358 1797 850 1822">\textsf{Temperature []^\circ C}\textsf{[]}</pre>

---

Syntax	ADD_LABEL_TEXT
	Text with white space, suitable for an axis label: <code>\textsf{Text with space} Al_2O_3</code>

## APPEND\_EXPERIMENTAL\_DATA

Add experimental data and text onto a calculated diagram. This is done by placing the experimental data and text on a formatted textual file (always having an extension of EXP) that is prepared outside of Thermo-Calc software (Console Mode and Graphical Mode), according to the syntax of the DATAPLOT Graphical Language.

The picture generated from the data is superimposed on the ordinary graphical output from the postprocessor. Such an experimental data file (\*.EXP) may be created by an ordinary textual editor.

Another important usage of this command is to superimpose plots from several independent calculations. For this purpose, there is a command MAKE\_EXPERIMENTAL\_DATAFILE, which dumps a calculated diagram on an EXP file according to the DATAPLOT syntax. With the aid of a normal text editor, many such files may be merged together. Remember that you can only have one prologue section on each such EXP file.



*DATAPLOT User Guide* included with this documentation set.

Syntax	APPEND_EXPERIMENTAL_DATA
Prompts	<p>Use experimental data (Y or N) /N/:</p> <p>By default, no experimental data are plotted. Enter Y to use the data from an experimental data file in the next plot. An <b>Open</b> window displays to specify the name of the file with the experimental data.</p>
	<p>Prologue number: /0/: &lt;Prologue number&gt;</p> <p>Select which prologue to use. In a prologue, you can, for example, give the scaling of an axis, the axis texts, and so on.</p> <div style="border: 2px solid blue; padding: 10px; margin-top: 10px;">  <p>By answering -1, a list of all prologues on the file is given; and by 0, no experimental data are appended.</p> </div>
	<p>Dataset number(s) /-1/: &lt;Dataset number(s)&gt;</p> <p>Select from which dataset(s) data should be read. Several datasets may be given separated with commas or spaces. By answering -1, a list of all datasets on the file is given; and by 0, no experimental data are appended.</p>

## CHANGE\_LEGEND

When there is a legend in the diagram you can append constitution and/or ordering descriptions to the phase name.



LIST\_EQUILIBRIUM also displays `ORD` (ordered) or `DISORD` (disordered) (when available).



In some cases the legend only shows the phase name for a line when `SET_LABEL_CURVE_OPTION` is set to `F`.

Syntax	CHANGE_LEGEND
Prompt	<p>Set suffix for phase names in legend? (All, None, Constitution, Ordering)/None/:</p> <ul style="list-style-type: none"> <li>• <code>None</code>: the legend includes the standard information about the phases.</li> <li>• <code>All</code>: the Constitution description and the Ordering description (when there is ordering) are also included.</li> <li>• <code>Constitution</code>: it is added to the standard description.</li> <li>• <code>Ordering</code> (when applicable): it is added to the standard description.</li> </ul>
	<p> For examples and more details about what these options show, see <a href="#">About Legend Styles</a> and <a href="#">Legend and Label Styles</a> where it is described for Graphical Mode.</p>

## DUMP\_DIAGRAM

This is an alternative command used to create output. First use the PLOT\_DIAGRAM command to send output to the Console Results window. Then you can save it to a file either using this command or right-click the diagram in the Console Results window and select Save as.



Similar to [PRINT\\_DIAGRAM](#), it does not matter what graphic device/printer is set up. The command can convert the diagram shown in the Console Results window to the selected format.

Syntax	DUMP_DIAGRAM
Prompt	Output format/: (jpg,ps,pdf,svg,emf,gif,png) /png/
	<p>Enter a format or press &lt;Enter&gt; to accept the default (png).</p> <p>If you choose to enter the parameters and file name in one line (i.e. when you are entering the command), the <b>Save</b> window does not open as described below. For example, if you enter <code>dump_diagram png test</code> at the prompt, an image file called <b>test.png</b> is output to the same folder where the macro file is stored. If a file with the specified name already exists, you are prompted to overwrite the old file. You can alternatively add <b>Y</b> or <b>N</b> to the command string to specify whether to overwrite the file or not e.g. <code>dump_diagram png test y</code>.</p> <p>A <b>Save</b> window opens if you have not specified a file name. Select a directory path and enter a <b>File name</b>. If a file with the specified name already exists, then you are prompted whether to overwrite the old file or not. The third argument of the command (Y) allows you to specify whether to overwrite the old file or not (N).</p>



After the output is done, the temporarily set graphic device for conversion and printing is switched back to the one set by the command SET\_PLOT\_FORMAT, so that you can save the graph in file(s) using PLOT\_DIAGRAM.

## ENTER\_SYMBOL



This command is for the POST module.



There are also a GES and POLY module commands with the same name.

Symbols are a useful feature of the POLY and POST modules to define quantities that are convenient. Symbols can be constants, variables, functions or tables.

Functions or tables (with defined functions as values) can be entered in the POST module after a stepping or mapping calculation, for purposes of plotting such entered functions or tables as axis variables.

Within the POLY module, symbols are normally defined prior to an equilibrium calculation (enforced by a C\_E command), stepping calculation (enforced by the command STEP\_WITH\_OPTIONS) or mapping calculation (enforced by MAP). These can be entered after an equilibrium calculation; however, for defined functions, variable or tables, it requires using EVALUATE\_FUNCTIONS before showing the corresponding values in the calculated equilibrium state.

The symbols entered in the POST module are not saved in the currently-loaded POLY3 workspaces. Therefore, if you want to apply such symbols in other similar calculations for the same defined system, you must use the ENTER\_SYMBOL command prior to the STEPPING or MAPPING calculation in the POLY module.



See [example 44](#) for an example of using variables and functions.

Symbols are a useful feature modules to define quantities that are convenient. Symbols can be constants, variables, functions, or tables.

Syntax	ENTER_SYMBOL
Prompt	<p>Constant, Variable, Function or Table? /Function/: &lt;Keyword&gt;</p> <p>The <i>Keyword</i> can be specified as CONSTANT, VARIABLE, FUNCTION, or TABLE.</p> <ul style="list-style-type: none"> <li>CONSTANTS can only be entered once and is a means of using a name for a numeric value. For example, the value of 1 atmosphere in Pascal can be denoted by P0 after the command ENTER_CONSTANT P0=101325. Defined constants can be used as values in condition assignments, for example, SET-COND P=P0.</li> <li>FUNCTIONS are expressions of state variables or other functions. These expressions are saved, and whenever a function value is requested all functions are evaluated. The reason for this is that they may depend on each other.</li> <li>VARIABLES are similar to functions because they can also be expressions of state variables. However, contrary to functions, they are only evaluated when they are entered or if they are explicitly named in an EVALUATE_FUNCTIONS command. It is possible to enter a variable</li> </ul>

Syntax	ENTER_SYMBOL
	<p>with a new expression any time. This expression is evaluated directly and its value stored as the value of the variable. Defined variables can be used as values in the SET_CONDITION command.</p> <ul style="list-style-type: none"> <li>• TABLES are used for listing results from the STEP or MAP commands. A table consists of a list of any number of state variables, functions, or variables. Defined tables can also be used in the POST (post-processor) module.</li> </ul> <div data-bbox="354 495 1395 667" style="border: 1px solid blue; padding: 10px; margin-top: 10px;">  There is a special connection between tables and variables. If a variable is used in a table, it is evaluated for each line of the table in the TABULATE command or when the table is used in a plot. </div>
	<p>Name: &lt;Name of the symbol&gt;</p> <div data-bbox="354 743 1395 947" style="border: 1px solid black; padding: 10px; margin-top: 10px;">  Each symbol has a unique name that must start with a letter and can have maximum 8 characters. Legal characters include letters (either UPPER or lower case), digits and underscore <code>_</code>. Any other special character, such as parentheses ( and ), plus +, minus -, slash / or \, full stop (.), are illegal for symbol names. </div> <p>You can enter the symbol name and the value or function on the same line; these must be separated with an equal sign =, for example, <code>TC=T-273.15</code> or <code>T_C=T273.15</code> which stands for a definition of temperature in Celsius. Otherwise, these questions are asked.</p> <div data-bbox="354 1094 1395 1234" style="border: 1px solid blue; padding: 10px; margin-top: 10px;">  For different types of symbols (constant, function, variable, or table), the questions have different prompts. </div>
	<p>Function: &lt;Definition for a function or variable&gt;</p> <p><i>Functions</i> and <i>variables</i> are evaluated from an expression of state variables or other functions, constants, or variables. The expression is a FORTRAN-like expression and operators +, -, *, = and ** can be used (** only with integer powers).</p> <p>Unary functions like LOG, LOG10, EXP, SIN, COS, and ERF can also be used. An expression can be continued on more than one line. An expression should be terminated by a semicolon (;) or an empty line (press &lt;Enter&gt; at the next prompt).</p> <p>Examples of function expressions:</p> <ul style="list-style-type: none"> <li>• <code>GM(LIQUID)</code>: The Gibbs energy of liquid per mole component</li> <li>• <code>H.T/4.184</code>: The heat capacity of the system in calories</li> <li>• <code>ACR(CR)/X(FCC,CR)</code>: The activity coefficient for Cr in FCC</li> <li>• <code>T-273.15</code>: The temperature in Celsius</li> </ul> <p>&amp;: &lt;Continuation of the definition for the symbol&gt;</p> <p>The ampersand &amp; means that you can continue to write the function on the new line if one line is</p>

Syntax	ENTER_SYMBOL
	not enough for the function. If you finish the function press <Enter> again.
	Value: <Value for a constant> A <i>constant</i> can only be assigned a numeric value once.
	Value or expression: <Value of expression for a variable> A <i>variable</i> can be assigned a numeric value or an expression. An expression is evaluated immediately and discarded. Only the numeric value is kept. This gives a possibility to save a value between calculations with different conditions because all state variables and functions are evaluated for the new conditions.
	Variable(s): <Variable(s) in a table> A <i>table</i> consists of a list of state variables or functions. One way to obtain results from a STEP command is through a table. Example: <pre>ENTER TABLE K=T,X(LIQ,C),X(LIQ,CR),ACR(C)</pre> Which means that the table called K contains four columns, i.e. the temperature, the mole fractions of C and Cr in the LIQUID phase, and the activity of C. To show the temperature in Celsius in a table, give the command ENTER_FUNCTION TC=T-273; and then use the symbol TC in the table. & <Continuation of the definition for the table> The ampersand & means that you can continue to write the table on the new line if one line is not enough for the table. If you finish the table press <Enter> again.

## FIND\_LINE

Identifies various property curves on a plotted property diagram (after a stepping calculation); it also works well in find stable phase regions on a phase diagram (after a mapping calculation).

Syntax	FIND_LINE
Prompts	X coordinate /.484623611/: <Value of the X coordinate> Specify the value of the X coordinate where you want to find the details of property curve (or phase region).
	Y coordinate /.5153737962/: <Value of the Y coordinate> Specify the value of the Y coordinate where you want to find the details of property curve (or phase region). The POST module displays the identified property (for a property curve on a property diagram), or the identified stable phase-assemblage (for a phase region on a phase diagram).

## LIST\_DATA\_TABLE

 This is a POST module command.

This command is used for listing various properties (i.e. the Y-axis variable on a property diagram) that change with an independent variable (i.e. the X-axis variable on a property diagram), which have already been defined in the POST module (and often are plotted on the **Console Results** window), but only after a STEP calculation.

The listing output can be either on screen or in a spreadsheet file in the MS Excel format (under specified file name with the default file extension of XLS), which can be further edited or used for graphical handling by opening the file (using the MS-Excel program) outside of the Thermo-Calc software system.

Syntax	LIST_DATA_TABLE
Prompt	<p>Output to screen or file /Screen/: &lt;File name or return &gt;</p> <p>Specify a file name if you want to save on a spreadsheet file (in the MS Excel format, with the default extension of XLS) on which the already-defined properties on a property diagram (after a STEP calculation) is saved, or press &lt;Enter&gt; if want to see a simple textual table on screen (for the already-defined properties that change alone with an independent variable).</p> <div style="border: 2px solid blue; padding: 5px; margin-top: 10px;"> <p> This command only works for a property diagram after a stepping calculation but not for any phase diagram after a mapping calculation.</p> </div>

---

## LIST\_PLOT\_SETTINGS



This is a POST module command.

Syntax	LIST_PLOT_SETTINGS
	Lists the present values of most parameters specifying the type of diagram to be plotted.

## LIST\_SYMBOLS

- >\_ This command is for the POST module.
- >\_ There is also a POLY module command with the same name.

Use this command to list the definitions for all constants, functions, variables and tables. In order to find the value of a function or variable, use SHOW\_VALUE or EVALUATE\_FUNCTIONS. A table is tabulated with the TABULATE command.

Syntax	LIST_SYMBOLS
	The defined variables are listed up together with the defined functions, but variable names are followed by a percentage sign %.

## MAKE\_EXPERIMENTAL\_DATAFILE

Write the graphical information onto screen or a formatted text file (always having an extension of `.EXP`) using the DATAPLOT graphical format. In order to merge two or more diagrams from separate calculations, you can write them out on files with this command, and then add/edit them together with a normal text editor.



There is another type of experimental data file that is generated in Thermo-Calc when working with the PARROT module. For more information see [Experimental Data Files \(\\*.POP, \\*.DOP\)](#).

Syntax	MAKE_EXPERIMENTAL_DATAFILE
Prompt	Output to screen or file /Screen/: <Name of a file or return> Specify the name of the desired file. Press <Enter> for a list. Otherwise enter a file name where the graphical information is written. The default file extension is <code>EXP</code> (for Windows) or <code>exp</code> (for Linux).

## MODIFY\_LABEL\_TEXT

Move a label created with the command ADD\_LABEL\_TEXT to another position, or replace its text with another one.

The label texts and the coordinates, either added by ADD\_LABEL\_TEXT command or modified by this command or manually edited on the Console Results window, are stored in the workspace, and can be saved onto an \*.EXP file (through MAKE\_EXPERIMENTAL\_DATAFILE).

Syntax	MODIFY_LABEL_TEXT
Prompts	<p>Which label to modify? /#/: &lt;Number of the label&gt;</p> <p>Before this prompt, all the labels created by the ADD_LABEL_TEXT command is listed with an identifying number. Specify the Number of the label you want to modify. The default one (#) is the last added label.</p>
	<p>New X coordinate: /xxx/: &lt;New X position&gt;</p> <p>Specify the New X position. The previous X coordinate is displayed. Press &lt;Enter&gt; to accept.</p>
	<p>New Y coordinate: /yyy/: &lt;New Y position&gt;</p> <p>Specify the New Y position. The previous Y coordinate is displayed. Press &lt;Enter&gt; to accept.</p>
	<p>New text /ABCDEFGH.../: &lt;New labeling text&gt;</p> <p>Specify the New labeling text. The previous text is displayed. Press &lt;Enter&gt; to accept.</p> <div style="border: 2px solid red; padding: 5px; margin-top: 10px;">  The new label text must not be longer than the previous text.         </div>

## PLOT\_DIAGRAM

After you use the SET\_PLOT\_FORMAT command to define the default graphic device (the printer) you can use a PLOT\_DIAGRAM command to either save the calculation output to a file or display the output in the Console Results window.



The diagram axes must be set first.

The functionality of this command is shared in two ways:

- in PLOT\_DIAGRAM for normal screen display and file saving and
- *Windows OS*: If the device is not defined, you can alternatively use PRINT\_DIAGRAM for a hard copy.

Syntax	PLOT_DIAGRAM
Prompt	Output to screen or file /Screen/: <Name of a file or return for screen> Specify the name of the file
	<div style="border: 2px solid #00AEEF; padding: 5px; margin-bottom: 10px;">  For Linux users, the prompt is PLOTFILE /SCREEN/).         </div> <p>A file with a proper extension for the selected graphic format (set by SET_PLOT_FORMAT) is saved under the current working directory. If no extension is given in the file name, the default extension e.g. ps for Postscript portrait/landscape modes, P7 or p7 for HPGL landscape, P8 or p8 for HPGL portrait, etc.) is automatically given to the file name.</p> <p>If the default device is set (SET_PLOT_FORMAT), press &lt;Enter&gt; to plot the diagram on the screen in the Console Results window or give a file name to save it.</p>

---

## PRINT\_DIAGRAM

### [Saving and Printing the Output in Console Mode](#)

First use PLOT\_DIAGRAM to create a diagram that displays in the **Console Results** window. Then after entering the PRINT\_DIAGRAM command, the **Print** window opens where you can choose additional settings for a hard copy of the diagram.



PRINT\_DIAGRAM functions in the same way as in the **Console Results** window where you can right-click a diagram and choose **Print**.

## QUICK\_EXPERIMENTAL\_PLOT

This is similar to APPEND\_EXPERIMENTAL\_DATA but can be used when there is no graphical information to be plotted in the POLY3 workspace. It defines a pair of axes, sets the axis labels to X and Y, and scales both X- and Y-axes between 0.0 and 1.0 unless a prologue is read from the DATAPLOT (\*.EXP) data file.



[Plotting Experimental Data Files](#) in the *DATPLOT User Guide* included with this documentation set.

Syntax	QUICK_EXPERIMENTAL_PLOT
Prompts	<p>Prologue number: /0/: &lt;Prologue number&gt;</p> <p>An <b>Open</b> window displays. Select the prologue to use. In a prologue, you can, for example, give the scaling of an axis, the axis texts, and so on. By answering -1, a list of all prologues on the file is given; and with 0, no experimental data are appended.</p>
	<p>Dataset number(s) /-1/: &lt;Dataset number(s)&gt;</p> <p>Select from which dataset(s) data should be read. Several datasets may be given separated with commas or spaces. By answering -1, a list of all datasets on the file is given; and with 0, no experimental data are appended.</p>

## **REINITIATE\_PLOT\_SETTINGS**

The REINITIATE\_PLOT\_SETTINGS command gives default values to all parameters describing the diagram. All plot settings made in the POST module are deleted, and it returns to the initial settings when the POST module is entered.

---

## RESTORE\_PHASE\_IN\_PLOT

Restores a phase previously suspended from the plot with SUSPEND\_PHASE\_IN\_PLOT. It works only after at least one phase is suspended.

Syntax	RESTORE_PHASE_IN_PLOT
Prompt	Phase name: <Name of a phase> Give the name of the phase to be restored.

## SELECT\_PLOT

Select a plot (diagram) within the current **Results** tab in the **Console Results** window.

Syntax	SELECT_PLOT
Prompt	<p>Plot &lt;N/New&gt;</p> <p>Where <b>N</b> is a positive integer. Enter the number of the plot that is selected in the <b>Console Results</b> window (in the currently selected <b>Results</b> tab). If a plot with that number does not exist, it is created.</p> <p>The default parameter value is <b>NEW</b>. If this is entered, then a new plot is created. Create a new plot can also be done by pressing &lt;Ctrl+Shift+T&gt;.</p>

## SET\_AXIS\_LENGTH

Change the real length of an axis in inches. The default number of tic-marks on an axis is 10 when the relative length is 1. The number and units per tic-mark must be a multiple of 1, 2 or 5 to obtain a reasonable scaling of an axis.

Syntax	SET_AXIS_LENGTH
Prompts	Axis (X, Y or Z): <Name of an axis> Specify which axis to set the length.
	Axis length /11.5/: <New relative axis length> Specify the new real axis length in inches. The relative length 1 corresponds to 10 tic-marks on the axis.

---

## SET\_AXIS\_PLOT\_STATUS

Specify whether the axis title texts and axis label texts are to be plotted or not on a diagram. It does not work for the axis lines and tic-marks. If you want to plot a diagram without tic-marks, use SET\_TIC\_TYPE first.

This can be used to merge different diagrams on a pen-plotter or to obtain the diagram faster. The default status is that all the set axes are to be plotted.

Syntax	SET_AXIS_PLOT_STATUS
Prompt	Axis plot (Y or N) /Y/: (Y or N)

## SET\_AXIS\_TEXT\_STATUS

Change the axis text from the automatic text given by the axis specification to a text.



### ADD\_LABEL\_TEXT

Syntax	SET_AXIS_TEXT_STATUS
Prompts	Axis (X, Y or Z): <Name of an axis> Specify which axis text status to change.
	Automatic axis text (Y or N) /N/: Specify if automatic axis text is to be used.
	Axis text: <Enter axis text> You can enter simple text, use LaTeX, rotate the text, as well as adjust the font size.

## SET\_AXIS\_TYPE

Change the axis type among linear, logarithmic and inverse axis.

Syntax	SET_AXIS_TYPE
Prompts	Axis (X, Y or Z): <Name of an axis> Specify which axis to change the axis type.
	Axis type /Linear/: <New axis type> Specify which new axis type to set. Select <code>Linear</code> (default), <code>Logarithmic</code> or <code>Inverse</code> . Only the three first characters are relevant.

## SET\_COLOR

Some devices support colours or line thickness. With this command, select different colors or line types on some types of information on a diagram.



This command can fail if a chosen device does not support colors or line thickness.

This command gives four sequential prompts (i.e. text and axis, invariant equilibria, tie-lines, and all other lines), with the keyword and default option (both listed below). Depending on if the switched-on graphic device (by SET\_PLOT\_FORMAT) supports colors or line types, the POST automatically switches on the appropriate Keyword.

Syntax	SET_COLOR
Prompt	<pre>Text and axis keyword /Default option/ &lt;Return or new option&gt; Invariant equilibria keyword /Default option/: &lt;Return or new option&gt; Tie-lines color keyword /Default option/: &lt;Return or new option&gt; Keyword of all other lines /Default option/: &lt;Return or new option&gt; This is usually for the solubility lines.</pre>
	<div style="border: 1px solid blue; padding: 10px;">  <p>The <code>Keyword</code> is either <code>Color</code> or <code>LineStyle</code>, depending on if color or line type is supported by the switched-on graphic device (by SET_PLOT_FORMAT); the POST module <i>automatically</i> switches on the appropriate <code>Keyword</code>.</p> </div>

### Color Options

The options for `Color` are:

BACKGROUND	BLUE	PURPLE	GRAY
FOREGROUND (default)	YELLOW	GOLD4	ORANGERED3
RED (default for invariant equilibria)	MAGENTA	TURQUOISE4	MAROON
GREEN (default for tie lines)	CYAN	PINK	PLUM

---

	SIENNA	OLIVEDRAB	SEAGREEN
	ORANGE1	CORAL1	USERDEF

### ***LineStyle Options***

The options for `LineStyle` are:

INVISIBLE	VERY_THICK (default for invariant equilibria)
NORMAL (default)	THIN (default for tie lines)
THICK	VERY_THIN
DASHED	DOTTED

## SET\_CORNER\_TEXT

Adds text to the corners on a plotted diagram. Normally, you can write such texts as a subtitle.

The primary database (not the appended ones) used in calculations and the calculated conditions are automatically plotted at the upper-left corner as a part of the header on all plotted diagram. Unless the plotting option WRITE CONDITIONS is switched off by the command SET\_PLOT\_OPTIONS, the calculated conditions are always written on plotted diagrams; and only when the plotting option PLOT HEADER is switched off by the command SET\_PLOT\_OPTIONS, the used database also always appear. Under such circumstance, avoid writing texts at the UPPER\_LEFT corners.

Syntax	SET_CORNER_TEXT
Prompt	Corner/Lower_Left/: <Return or new option> Choose an option: <ul style="list-style-type: none"> <li>• LOWER_LEFT</li> <li>• UPPER_LEFT</li> <li>• UPPER_RIGHT</li> <li>• LOWER_RIGHT</li> <li>• TOP_OF_TRIANGLE</li> </ul>
	Text: <Texts as a subtitle or a note> Write the texts that are to be written on the specified diagram corner.

## SET\_DIAGRAM\_AXIS

Specify an axis variable for a plot or a listing. To be able to plot a diagram, at least two axis variables (X and Y) must be specified. In Console Mode, three axes (X, Y, and Z) can be specified for a diagram.



The axis variables in the plot can be different from the number of axis used for the mapping.

If you want to plot a phase diagram with tie-lines in the plane after a MAP calculation, then the composition axis must be `mole-fraction` or `weight-fraction`, or `mole-percent` or `weight-percent` of one specific component. You must not use the stable variable `x(comp)` or `w(comp)` even if that is what you used when calculating the diagram, because that only gives one side of the two-phase region. It may be sufficient to indicate that mole-fraction is the same as the stable variable `x(*, comp)`, and weight-fraction is the same as the stable variable `w(*, comp)`.

If you plot a phase diagram where the tie-lines are not in the plane after a MAP calculation, then `mole-fraction` and `x(comp)` are identical.

When you plot a property diagram after a STEP calculation and want a composition axis variable, you should normally use `x(comp)` and not `mole-fraction`.

The axis must have exactly the same number of columns, or one axis must have one column. In the first case the columns are plotted matching them one by one; in the latter all columns on one axis is plotted against the single column. For example, you can have the temperature on one axis and the amount of each phase on the other. The amount of each phase is the state variable `NP(*)` in mole fraction or `BPW(*)` for mass fractions of phases.

Automatic diagram axis: The POST module can set automatic diagram axis identical to those used in the MAP command.

If the state variable `x(comp)` is used in a MAP calculation, then mole-fraction of the component is used as diagram axis; if a potential or some other state variables are used, then that is used as a diagram axis.

After a STEP calculation, automatic diagram axis is not set as there is only one axis used in the calculation.

Syntax	SET_DIAGRAM_AXIS
	Axis (X, Y or Z): <Axis name>
	Specify which axis (X or Y or Z) to set with a variable.

Syntax	SET_DIAGRAM_AXIS
Prompts	<p>Variable type: &lt;Variable name&gt;</p> <p>Specify a variable for the chosen axis here.</p> <p>The valid variables are:</p> <ul style="list-style-type: none"> <li>• TEMPERATURE-CELSIUS as temperature in °C</li> <li>• TEMPERATURE as temperature in K</li> <li>• PRESSURE as pressure in Pa</li> <li>• ACTIVITY for a component</li> <li>• LNACTIVITY for a component [ln(ACR(component))]</li> <li>• MOLE-FRACTION for a component</li> <li>• MOLE-PERCENT for a component</li> <li>• WEIGHT-FRACTION for a component</li> <li>• WEIGHT-PERCENT for a component</li> <li>• ENTHALPY as enthalpy in J/mol</li> <li>• ENTROPY as entropy in J/mol/K</li> <li>• NONE to clear the axis</li> <li>• Any valid state variable including those with wildcards, e.g. NP(*) or x(*,component). Due to the use of wildcard * in such a state variable, the program asks for a column number.</li> <li>• Any entered function or variable.</li> <li>• Any entered table.</li> </ul>
	<p>If you specify an entered table as the variable, then you are given these prompts:</p> <p>For component: &lt;Component name&gt;</p> <p>When an activity, mole or weight fraction or percent of a component shall be plotted, the component name must be supplied here.</p>
	<p>Column number: &lt;Column number(s)&gt;</p> <p>Specify the column number(s) in the chosen table that are to be plotted onto the earlier specified axis. For example:</p> <ul style="list-style-type: none"> <li>• 1 for column 1</li> <li>• 2, 3 for columns 2 and 3</li> <li>• 2, 3 &gt;5 for column 2,3 and all columns above column 5</li> <li>• * for all columns</li> </ul>

## SET\_DIAGRAM\_TYPE

Choose the diagram type as perpendicular plot or triangular plot (Gibbs triangle, especially for ternary systems). The default is with a perpendicular axes.

For phase diagrams of ternary or pseudo-ternary systems, it is usually need to plot isothermal sections as triangular grams. If desired, all lines outside the region limited by a line joining the end points of the X- and Y-axis is removed.

Syntax	SET_DIAGRAM_TYPE
Prompts	<p>Triangular diagram (Y or N) /N/:</p> <p>The default is to set a perpendicular axis. Perpendicular diagrams have (almost) the same scaling on both axes.</p> <p>Create tetrahedron WRML file (Y or N) /N/:</p> <div style="border: 2px solid blue; padding: 5px; margin: 10px 0;">  The 3D plotting in Console Mode does not work. Enter Y to specify a triangular plot instead.         </div>
	<p>Plot 3rd axis (Y or N) /N/:</p> <p>If a triangular plot is selected, enter Y to specify a 3rd axis and to connect (and plot) the end points of the X- and Y-axes.</p>
	<p>Clip along 3rd axis (Y or N) /N/:</p> <p>If a triangular plot is selected, enter Y so that all lines outside the region limited by a line joining the end points of the X- and Y-axis are removed.</p>

## SET\_FONT

Select the font to use for labels and numbers when plotting the diagram under the currently selected graphic device (SET\_PLOT\_FORMAT). For some devices there may be other fonts available and these are selected by SET\_PLOT\_FORMAT.

Syntax	SET_FONT
Prompts	Select font number /1/: <#> Specify an available font number # as the default font for the current graphic device, or accept the font number 1 by pressing <Enter>. By typing a question mark ? here, the program lists the available fonts under the currently selected graphic device.
	Font size /.34/: <##> The size of the chosen font is specified. A value around 0.34 is recommended.

## **SET\_INTERACTIVE\_MODE**



In the POLY module this is called SET\_INTERACTIVE.

Use the SET\_INTERACTIVE\_MODE command to use it in demonstration or macro files to stop the execution of the command file and pass over input focus to the keyboard.

## SET\_LABEL\_CURVE\_OPTION

Identify the curves drawn in the post-processor by marking each curve with a digit and then list the meaning of these digits beside the plot.



The number of digits can be quite large in some cases. If so, use SET\_FONT and reduce the font size; usually, 0.2 is sufficient.

Syntax	SET_LABEL_CURVE_OPTION
Prompt	Curve label option (A, B, C, D, E, F OR N) /A/: <Option>
	<p>The options:</p> <ul style="list-style-type: none"> <li>• A: List stable phases along line</li> <li>• B: As A but curves with same fixed phase have same number</li> <li>• C: List axis quantities</li> <li>• D: As C but curves with same quantities have same number</li> <li>• E: As B with changing colors</li> <li>• F: As D with changing colors</li> <li>• N: No labels</li> </ul> <p>This question is rather cryptic but usually the option B or E is good for phase diagrams (after a MAP command) and option D or F for property diagrams (after a STEP command). Those interested can try out the option A and C by themselves.</p> <div style="border: 1px solid blue; padding: 10px; margin: 10px 0;">  <p>Option B or E lists the fixed phases along each curve, whereas option D or F gives the axis variables used along each curve. E provides varied colors for different stable-phase curves on phase diagrams (after a MAP command); F provides varied colors for different property curves on property diagrams (after a STEP command). The option N (NONE) disables all curve labels and lists.</p> </div> <p>If, for example, you use T (temperature in K) as the variable axis in a STEP command and then plot the amount of stable phases (with SET_DIAGRAM_AXIS Y NP (*)) on the Y-axis versus T-C on the X-axis, then the list may have lines like:</p> <pre>1: T-273.15, NP(LIQUID) 2: T-273.15, NP(FCC_A1)</pre> <p>This means that for curve 1 the X-axis is T-273.15 (the same for all curves, of course) and NP (LIQUID) on the Y-axis. Curve 2 has the same X-axis but NP (FCC_A1) on the Y-axis.</p>

## SET\_PLOT\_FORMAT

Adjust the format of the graphical output to another graphical device. This default can be changed with the [SET\\_PLOT\\_ENVIRONMENT](#) command in the SYSTEM MONITOR.

Syntax	SET_PLOT_FORMAT
Prompt	<p>Graphic device number /#/: &lt;Number of the device&gt;</p> <p>Specify a graphic device number. Depending on the hardware, different plot formats (graphic devices) may be available. These are listed online by giving a question mark ?.</p> <p>For some formats there can be additional sub-prompts (Y or N) about the available fonts being used for the chosen format. Also enter Y for further specifications, e.g. font type and size.</p>

## SET\_PLOT\_OPTIONS

You are prompted for and can toggle on (Y) or off (N) the plotting of some options on all subsequently generated diagrams.

For the `Always initiate POST on re-entering` prompt, the default is Y which implies that the POST module automatically reinitiates whenever the POLY3 workspace is changed upon re-entering. If answered N the same diagram axis variables are kept even if the workspaces in the POLY module are read from or written to a \*.POLY3 file. The N answer is useful when there are several sets of \*.POLY3 files with the same calculation but for different sets of model parameters.

For the `Always solid line` prompt, the default answer is N which means that the POST module plots solid lines for stable phases but dashed lines for metastable phases. This is normally for plotting property diagram after a STEPPING calculation; however, when there is a metastable extrapolation in a phase region on a phase diagram after a MAPPING calculation, this is also enforced. When the alternative answer Y is chosen, all the lines are always plotted in solid lines.

Syntax	SET_PLOT_OPTIONS
Prompts	<p>Plot header /Y/:</p> <p>Toggle the plotting of the header text above a diagram.</p>
	<p>Plot logo /Y/:</p> <p>Toggle the plotting of the Thermo-Calc logo at the lower-left corner of a diagram.</p>
	<p>Plot footer /Y/:</p> <p>Toggle the plotting of the footer identifier text at the right margin of a diagram.</p>
	<p>White-Contoured-PS-CHARS /Y/:</p> <p>By default, the PostScript characters are written as a white contoured status.</p>
	<p>Plot remote exponent(s) /Y/:</p> <p>By default the remote exponent on the axis is plotted.</p>
	<p>Plot symbols at node points /0/: &lt;#&gt;</p> <p>This makes it possible to plot symbols at the node points on plotted lines on a diagram.</p>
	<p>Symbol size /.1/: &lt;.#&gt;</p> <p>Set the size of symbols which are plotted at the node points.</p>
	<p>Write condition? /Y/:</p> <p>By default, the initial equilibrium conditions for the current calculations are automatically plotted at the upper-left corner as a part of the header. If you enter N, the calculated conditions are not included on a plotted diagram.</p>
	<p>Write database name? /Y/:</p>

Syntax	SET_PLOT_OPTIONS
	<p>By default, the primary database (not appended ones) used in the current calculations are plotted at the upper-left corner as a part of the header. If you enter N, the database name is not included on a plotted diagram.</p>
	<p>Always initiate POST on re-entering: /Y/:</p> <p>By default the POST module automatically reinitiates whenever the POLY3 workspaces are changed upon re-entering.</p> <p>If you enter N, the same diagram axis variables are kept even if the workspaces in the POLY module is read from or written to a *.POLY3 file. This can be useful when there are several sets of *.POLY3 files with the same calculation but for different set of model parameters.</p>
	<p>Always solid line: /N/:</p> <p>By default the POST module plots solid lines for stable phases and dashed lines for metastable phases. This is normally for plotting property diagrams after a STEPPING calculation. However, when there is a metastable extrapolation in a phase region on a phase diagram after a MAPPING calculation, this is also enforced. Enter Y to always plot solid lines in this case, for example.</p>

## SET\_PLOT\_SIZE

Change the size of the diagram by specifying a global plot size (as a relative scaling factor). The default value of the relative scaling factor is 1, while the real size of the plotted diagram depends on what output device is chosen with SET\_PLOT\_FORMAT. The default plot size (corresponding to the default global plot size 1) is adjusted to the chosen device.

Syntax	SET_PLOT_SIZE
Prompt	Global plot size /1/: <Relative scaling factor> Enter the relative scaling factor as a numerical number (e.g. 0.5, 0.8, 1.0, 1.5, etc.). By pressing <Enter>, the default relative scaling factor (1) is accepted.

## SET\_PREFIX\_SCALING

Specify on which axis is to have a prefix scaling then set the scaling of remote exponents with certain powers.

Syntax	SET_PREFIX_SCALING
Prompt	Axis (X, Y or Z): <Name of an axis> Specify which axis is to have a prefix scaling.
	Use prefix scaling /Y/: (Y or N or #) <ul style="list-style-type: none"><li>• NO switches off its action.</li><li>• YES arranges the prefix scaling so that the axis scaling is done with the remote exponents being powers of three, i.e. ..., -6, -3, 0, 3, 6, .... The default is 3.</li><li>• # is an integer number that sets the remote exponents as being powers of # (as the powers for the remote exponents).</li></ul>

---

## SET\_RASTER\_STATUS

Plot a raster (i.e. with gridlines on both axis directions) in the diagram.

Syntax	SET_RASTER_STATUS
Prompt	Raster plot (Y or N) /N/: By default no raster is plotted. Enter Y to plot using a raster plot.

## SET\_REFERENCE\_STATE



This command is available for the POLY, ED-EXP and POST modules.

The reference state for a component is important when calculating activities, chemical potentials, and enthalpies and is determined by the database being used. For each component the data must be referred to a selected phase, temperature, and pressure, i.e. the reference state.



All data in all phases where this component dissolves must use the same reference state. However, different databases can use different reference states for the same element/component. It is important to be careful when combining data obtained from different databases.

By default, activities, chemical potentials, and so forth are computed relative to the reference state used by the database. If the reference state in the database is not suitable for your purposes, use this command to set the reference state for a component using `SER`, i.e. the Stable Element Reference (which is usually set as default for a major component in alloys dominated by the component). In such cases, the temperature and pressure for the reference state is not needed.

In order to specify conditions in the specific reference state, you can append an `R` to the state variables. Thus, `AC` is the activity (of a system component or of a species in a phase) with respect to the default reference state, and `ACR` is the activity with respect to the selected reference state; `MU` is the chemical potential of a system component with respect to the default reference state, and `MUR` stands for the chemical potential with respect to the selected reference state. The `AC` and `ACR` variables, for both components in a system and species in a phase, can be specified in natural logarithm, e.g. `LNAC(Fe)`, `LNACR(C)`, `LNAC(O2,GAS)`, `LNACR(O2,GAS)`.

For the POST module, and after calculating with the `STEP_WITH_OPTIONS` or `MAP` commands, the reference state for a component can also be changed to plot various properties of the component in the entire system or in a specific phase. Then you can set the diagram axes as the chemical potential or activity quantities with the `R` suffix, i.e. `MUR(comp)`, `MUR(comp,ph)`, `ACR(comp)`, `ACR(comp,ph)` or the common logarithms [e.g. `LNACR(comp,ph)`].

Syntax	SET_REFERENCE_STATE
Prompts	Component: <Name of the component> The name of the component must be given.
	Reference phase: <Name of a phase used as the new reference state>

Syntax	SET_REFERENCE_STATE
	<p>Enter the <code>Name</code> of a phase for the system.</p> <div data-bbox="363 317 1398 520" style="border: 1px solid blue; padding: 10px; margin: 10px 0;">  <p>For a phase to be usable as a reference for a component, the component needs to have the same composition as an end member of the phase. The reference state is an end member of a phase. The selection of the end member associated with the reference state is only performed once this command is executed.</p> </div> <p>If a component has the same composition as several end members of the chosen reference phase, then the end member that is selected at the specified temperature and pressure will have the lowest Gibbs energy.</p> <p>The following are example using temperature. Pressure is used in the same way:</p> <ul style="list-style-type: none"> <li>• In the Fe-C system, BCC can be a reference state for Fe but not for C since BCC can exist as pure Fe but cannot exist as pure C.</li> <li>• If GAS is chosen as the reference phase of component O at such a high temperature that O1 has the lowest energy, then O1 remains the reference state even at calculations performed at lower temperatures where O2 has a lower energy than O1.</li> <li>• Setting the reference state for component O as gas (one sublattice) with constituents O1, O2 and O3 results in O2 being the reference state if, at the present temperature, pure O2 has the lowest energy of O1, O2 and O3. If the reference state is set above a critical higher temperature, then O1 has the lowest energy and consequently becomes the reference state.</li> </ul>
	<p>Temperature <code>/*/:</code> &lt;Temperature for the reference state&gt;</p> <p>Select the Temperature (in K) for the reference state. The wildcard value <code>*</code> means the current temperature is used at the time of evaluation of the reference energy for the calculation.</p> <div data-bbox="363 1194 1398 1373" style="border: 1px solid blue; padding: 10px; margin: 10px 0;">  <p>The temperature set with this command does not affect the temperature used when evaluating the energy of the reference state during the end member selection process.</p> </div>
	<p>Pressure <code>/1E5/:</code> &lt;Pressure for the reference state&gt;</p> <p>Select the Pressure (in Pa) for the reference state. The wildcard value <code>*</code> means the current pressure is used for evaluating the reference energy at the time of calculation.</p>
	<p><b>Examples</b></p> <pre> S-R-S Fe SER S-R-S Cr FCC * 100000 S-R-S H2O AQUEOUS * 100000 S-R-S ZE REF_ELECTRODE * 100000 </pre>

## SET\_SCALING\_STATUS

When an axis variable is selected by SET\_DIAGRAM\_AXIS, the scaling status for the axis is always set to automatic scaling. By this command, choose between manual or automatic scaling on a specified axis. If manual scaling is chosen, specify a minimum and a maximum value. Manual scaling can be used to magnify interesting parts of a diagram.

Syntax	SET_SCALING_STATUS
Prompts	Axis (X, Y or Z): <Name of an axis> Specify which axis that you want to set the scaling status.
	Automatic scaling (Y or N) /N/: The default is manual scaling. Enter Y for automatic scaling (N). For manual scaling specify the Min and Max values.
	Min value: <Minimum value> The minimum value at the start point of the specified axis.
	Max value: <Maximum value> The maximum value at the end point of the specified axis.

## SET\_TIC\_TYPE

Change the drawing of the axis tics. You may alter the placement of tic marks on the axis, e.g. inside or outside or no tic marks. The size of tic marks cannot be altered by this command, but these are adjusted based on the selected graphic device and the defined relative scaling factor (global plot size).

Syntax	SET_TIC_TYPE
Prompt	TIC Type /1/: <1 or -1 or 0> The tic type 1 is default, i.e. the tics are drawn outside the diagram. -1 means that the tics are inside the diagram and 0 means no tics.

## SET\_TIELINE\_STATUS

If the tie-lines are in the plane of the calculation, you can select to plot the tie-lines in two-phase fields with this command.

Syntax	SET_TIELINE_STATUS
Prompt	Plotting every tie-line no. /0/: <Number of tie-lines> The tie-lines plotted are not equally spaced by the graphics software. Instead, you can select to plot a subset of the calculated tie-lines, i.e. every one (1), every second (2), every three (3), etc. By accepting the default value 0, no tie-line is plotted.

## SET\_TITLE

Specify a title that displays on all listed tables and plotted diagrams from the POST module.



[ADD\\_LABEL\\_TEXT](#)

Syntax	SET_TITLE
Prompt	<p>Title</p> <p>Enter the title you want to appear on all tabulation or graphic outputs.</p> <div style="border: 1px solid blue; padding: 5px;"><p> There is a maximum length of about 60 characters for the title. You can enter simple text, use LaTeX, rotate the text, as well as adjust the font size.</p></div>

## SET\_TRUE\_MANUAL\_SCALING

The tic-marks on the axes are normally placed in even intervals using the whole axis length. The scaling routine adjusts the given minimum and maximum values slightly to accomplish this. If such an automatic adjustment behaviour is unwanted, it can be avoided by using this command (followed by a message showing the TRUE MANUAL SCALING set for a specific axis).

The command works like a toggle. To reset the automatic scaling behaviour, repeat the command a second time for that specific axis (followed by a message saying the SEMI MANUAL SCALING set for the specific axis).

Syntax	SET_TRUE_MANUAL_SCALING
Prompt	Axis (X, Y or Z): <Name of an axis> Specify which axis (X or Y or Z) you want to toggle between automatic adjustments or avoid the adjustment of the given maximum and minimum values.

## SUSPEND\_PHASE\_IN\_PLOT

Specify that lines originating from the presence of a certain phase shall not be plotted on a diagram. If you want to bring the suspended phase back to the diagram, use RESTORE\_PHASE\_IN\_PLOT.

Syntax	SUSPEND_PHASE_IN_PLOT
Prompt	Phase name: <Name of a phase> Specify the name of the phase to be suspended from the diagram.

## TABULATE



This command is available for a table entered in either the POLY or POST module.

For any entered table, this command gives a table of values from equilibria calculated with a STEP command.



Unlike the TAB module's command (also called TABULATE) it is not possible to plot columns from the tabulated tables. For this use normal POST commands.

Syntax	TABULATE
Prompts	Name: <Name of a table entered in either POLY or POST> Specify a table name that is entered in either the POLY or POST module.
	Output on screen or file /Screen/: Specify a file name if you want to save on a file the table values along the defined STEP calculation, or press <Enter> if want to see the table values displayed.

---

# Technical Notes

---

**In this section:**

---

Technical Note: Pipe or Redirect Output Data from Console Mode .....	548
--	-----

## Technical Note: Pipe or Redirect Output Data from Console Mode

Audience:	Advanced Programmers and Users
OS:	Windows, Linux, or macOS
Application:	For users who want to pipe or redirect data from Thermo-Calc running a macro in Console Mode. For example, you can start the application with the Thermo-Calc.bat file and the entire application closes when you enter the EXIT command in the Console prompt.
Command examples:	<p><b>WINDOWS</b></p> <pre>Thermo-Calc.bat macro.tcm &gt; output.log</pre> <p><b>LINUX</b></p> <pre>./Thermo-Calc.sh batch macro.tcm &gt; output.log</pre> <p><b>MACOS</b></p> <pre>./Thermo-Calc.command batch macro.tcm &gt; output.log</pre>
Command line definitions	
>	Redirection command. The arrow indicates the flow of data. For example in <code>./Thermo-Calc.sh macro.tcm &gt; output.log</code> , the contents of <code>macro.tcm</code> is used as input to the <code>Thermo-Calc.sh</code> script file and the resulting output (standard out) is then written to <code>output.log</code> instead of to the screen.

# Console Mode Examples

---

## In this section:

---

Thermo-Calc Console Mode Examples .....	550
Binary Phase Diagram Examples .....	552
Property Diagram Examples .....	556
Ternary Phase Diagram Examples .....	561
Multicomponent Phase Diagram Examples .....	565
Examples Calculating Thermodynamic and Related Quantities .....	567
Single Equilibrium Calculation Examples .....	571
Potential Diagram Example .....	573
Examples of Scheil-Gulliver Calculations .....	574
Example Calculating Chemical Ordering .....	576
Examples of Gibbs Energy System (GES) .....	577
Example Calculating the T0 Temperature .....	582
Examples Calculating Paraequilibrium .....	583
Example Using Functions and Variables .....	584
Example Using PARROT and EDIT_EXPERIMENT .....	585
Pourbaix Diagram Examples .....	587
Pourbaix Examples TCEX40A, 40B, 40C, and 40E .....	590
POURBAIX_DIAGRAM Commands for Options 1 to 4 .....	594
Example Using Tabulate Reaction (TAB) .....	607
Example Using REACTOR .....	608
Examples with Thermophysical Properties .....	609
Application Examples .....	612

## Thermo-Calc Console Mode Examples

There are many examples to examine and to help you learn Thermo-Calc in Console Mode. Some of the examples in the table below are included with specific sections, whereas some of the general examples are included (repeated) in more than one section.

Each topic has a brief description of the example and includes some or all of the commands used in the macros.



[About the Examples](#)



[Opening Console Mode Macro Files](#)



All the macro files and the outputs are also combined into one file that you can search and read. This is included with your installation as a PDF (go to **Help > Manuals folder > All Thermo-Calc Documentation** and open *Thermo-Calc Console Mode Example Macros*).

Topic	TCEX Examples included
<a href="#">Binary Phase Diagram Examples</a>	1, 4, 13, 16, 17, 26, 31, 34, 50
<a href="#">Property Diagram Examples</a>	2, 8, 9, 11, 12, 22, 23, 28, 29, 32, 41, 49, 50
<a href="#">Ternary Phase Diagram Examples</a>	3, 5, 14, 17, 19A, 19B, 21, 33, 37, 42, 43
<a href="#">Multicomponent Phase Diagram Examples</a>	6, 10, 23, 27
<a href="#">Examples Calculating Thermodynamic and Related Quantities</a>	2, 13, 14, 18, 22, 26, 34, 39, 51
<a href="#">Single Equilibrium Calculation Examples</a>	7, 18, 20
<a href="#">Potential Diagram Example</a>	35
<a href="#">Examples of Scheil-Gulliver Calculations</a>	15, 30A, 30B, 48, 54, 57
<a href="#">Example Calculating Chemical Ordering</a>	16
<a href="#">Examples of Gibbs Energy System (GES)</a>	31, 38, 39, 49, 52
<a href="#">Example Calculating the T0 Temperature</a>	23

<i>Topic</i>	<i>TCEX Examples included</i>
<a href="#">Examples Calculating Paraequilibrium</a>	23, 42, 43
<a href="#">Example Using Functions and Variables</a>	44
<a href="#">Example Using PARROT and EDIT_EXPERIMENT</a>	36A (consists of various parts).
<a href="#">Pourbaix Diagram Examples</a>	40, 40A, B, C, and E, 53
<a href="#">Example Using Tabulate Reaction (TAB)</a>	12
<a href="#">Example Using REACTOR</a>	24
<a href="#">Examples with Thermophysical Properties</a>	55 - Viscosity 56 - Surface Tension 58 and 59 - Molar Volume 60 - Electrical Resistivity and Thermal Conductivity
<a href="#">Application Examples</a>	9 - Dew Point 10 - Continuous casting, preventing Cr <sub>2</sub> O <sub>3</sub> clogging 11 - Gas oxidation of Cu <sub>2</sub> S 20 - Adiabatic decompression in Geological system 24 - Silicon arc furnace 27 - Chemical Vapour Deposition 28 - Materials design of a stainless steel 29 - Gas speciation 41 - Solubility product 44 - Proof strength calculated from empirical functions

---

## Binary Phase Diagram Examples

---

### ***tcex01 - Calculating the Binary Fe-C Phase Diagram***

This example calculates the binary Fe-C phase diagram using the Binary module. The TCBIN database is used and included with your installation.

Some of the main commands used:

- [LIST\\_EQUILIBRIUM](#)
- [LIST\\_STATUS](#)
- [LIST\\_CONDITIONS](#)
- [CHANGE\\_STATUS](#)
- [COMPUTE\\_EQUILIBRIUM](#)
- [SET\\_DIAGRAM\\_AXIS](#)
- [SET\\_TIELINE\\_STATUS](#)
- [PLOT\\_DIAGRAM](#)
- [SET\\_SCALING\\_STATUS](#)
- [SET\\_LABEL\\_CURVE\\_OPTION](#)
- [ADD\\_LABEL\\_TEXT](#)
- [MODIFY\\_LABEL\\_TEXT](#)

### ***tcex04 - Calculating the Fe-Cr Miscibility Gap***

This example calculates the miscibility gap in the Fe-Cr system. The FEDEMO database is used and included with your installation.

These commands are used:

- [SWITCH\\_DATABASE](#)
- [GET\\_DATA](#)
- [LIST\\_EQUILIBRIUM](#)
- [SET\\_AXIS\\_VARIABLE](#)
- [DEFINE\\_SYSTEM](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)

- [SET\\_CONDITION](#)
- [SAVE\\_WORKSPACES](#)
- [REJECT](#)
- [LIST\\_STATUS](#)
- [COMPUTE\\_EQUILIBRIUM](#)
- [SET\\_TIELINE\\_STATUS](#)

### ***tcex13 - Calculating the Binary Al-Ti Phase Diagram and its G Curve***

This example calculates an Al-Ti binary phase diagram and G curve using the BINARY module. The TCBIN database is used and included with your installation.

### ***tcex16 - Second Order Transition of the Al-Fe System***

This example calculates the second order transition line in the BCC field of the Al-Fe system.



This uses the SSOL database. A license is required to run the example.

These commands are of note and used in this example:

- [REJECT](#)
- [RESTORE](#)
- [ADVANCED\\_OPTIONS](#)
- [SET\\_SCALING\\_STATUS](#)
- [DEFINE\\_SYSTEM](#)
- [MAKE\\_EXPERIMENTAL\\_DATAFILE](#)
- [LIST\\_EQUILIBRIUM](#)
- [APPEND\\_EXPERIMENTAL\\_DATA](#)
- [STEP\\_WITH\\_OPTIONS](#)

---

## ***tcex17 - The Pseudo-binary System - CaO-SiO<sub>2</sub>***

This example calculates the pseudo-binary system CaO-SiO<sub>2</sub> using the OXDEMO database, which is included with your installation.

The oxide demo (OXDEMO) database can be used both for pseudo-binary systems like the one in this case, CaO-SiO<sub>2</sub>, or for full ternary systems like Ca-Fe-O.



For a more advanced example in Graphical Mode using the TCS Metal Oxide Solutions Database (TCOX), see [T\\_17: Al<sub>2</sub>O<sub>3</sub>-MgO Phase Diagram](#).

These commands are of note and used in this example:

- [LIST\\_SYSTEM](#)
- [REJECT](#) and [RESTORE](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [LIST\\_STATUS](#)
- [DEFINE\\_COMPONENTS](#)
- [SET\\_CONDITION](#)
- [COMPUTE\\_EQUILIBRIUM](#) and [LIST\\_EQUILIBRIUM](#)

## ***tcex26 - The As-Ga Phase Diagram: Plotting the Partial Pressures of a Gas Species***

This is an example of plotting the partial pressures of a gas species along the solubility lines in the As-Ga phase diagram. The calculation makes it possible to monitor the input gases to a process of depositing solid As-Ga. The data is taken from the PG35 Binary Semi-Conductors database.

These commands are of note and used in this example:

- [REJECT](#)
- [RESTORE](#)
- [SET\\_DIAGRAM\\_AXIS](#)
- [SET\\_SCALING\\_STATUS](#)
- [SET\\_AXIS\\_TEXT\\_STATUS](#)
- [SET\\_LABEL\\_CURVE\\_OPTION](#)

### ***tcex31 - Using GES to Calculate CVM***

Using the Gibbs Energy System (GES) module, this example calculates the CVM and compares it with the sublattices of a fictitious A B system. You also learn how to overlay diagrams from two calculations.



This example uses some GES commands that are not yet supported by GES6. Therefore, we enforce the use of GES5.

These GES commands are of note and used in this example:

- [ENTER\\_ELEMENT](#)
- [ENTER\\_SPECIES](#)
- [ENTER\\_SYMBOL](#)
- [ENTER\\_PHASE](#)
- [ENTER\\_PARAMETER](#)
- [LIST\\_PHASE\\_DATA](#)

### ***tcex34 - The Al-Zn Phase Diagram and its G Curve***

This example uses the BINARY module to calculate the phase diagram and G curves in the Al-Zn system. It uses the TCBIN database included with your installation.

## Property Diagram Examples

### ***tcex02 - Plotting Thermodynamic Functions***

This example shows how to plot thermodynamic functions in unary, binary, and ternary systems. It also involves working with partial derivatives and partial quantities.



This uses the SSOL database. A license is required to run the example.

These commands are used and may be of interest:

<a href="#">GOTO_MODULE</a>	<a href="#">GET_DATA</a>	<a href="#">SET_CONDITION</a>
<a href="#">SWITCH_DATABASE</a>	<a href="#">AMEND_PHASE_DESCRIPTION</a>	<a href="#">LIST_EQUILIBRIUM</a>
<a href="#">DEFINE_SYSTEM</a>	<a href="#">COMPUTE_EQUILIBRIUM</a>	<a href="#">SET_AXIS_VARIABLE</a>
<a href="#">SAVE_WORKSPACES</a>	<a href="#">STEP_WITH_OPTIONS</a>	<a href="#">SET_DIAGRAM_AXIS</a>
<a href="#">ENTER_SYMBOL</a>	<a href="#">PLOT_DIAGRAM</a>	<a href="#">SET_LABEL_CURVE_OPTION</a>
<a href="#">REJECT</a>	<a href="#">RESTORE</a>	<a href="#">LIST_SYSTEM</a>
<a href="#">LIST_STATUS</a>	<a href="#">CHANGE_STATUS</a>	<a href="#">SHOW_VALUE</a>
<a href="#">SET_INTERACTIVE</a>		

### ***tcex08 - Property Diagrams for a High Speed Steel***

This example shows how to calculate property diagrams for a high speed steel i.e. phase fraction plots, activity vs temperature, and so forth.



This uses the TCFE database. A license is required to run the example.

These commands are of note and used in this example:

- [DEFINE\\_DIAGRAM](#)
- [DEFINE\\_ELEMENTS](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [REINITIATE\\_PLOT\\_SETTINGS](#)

### ***tcex09 - Calculating a Dew Point with POLY3***

This is an example calculating the dew point in the POLY3 module. The SUBDEMO database is used and included with your installation.

The STEP\_WITH\_OPTIONS and MAP commands are of note and used in this example.

### ***tcex11 - Oxidation of Cu<sub>2</sub>S with an H<sub>2</sub>O/O<sub>2</sub> Gas Mixture***

This example demonstrates the oxidation of Cu<sub>2</sub>S with an H<sub>2</sub>O/O<sub>2</sub> gas mixture. Thermo-Calc is used to find the optimum O/H ratio (i.e. oxygen potential) as certain oxygen potential values can desulfurize Cu<sub>2</sub>S without forming copper oxides.

In Thermo-Calc, the problem reduces to perform equilibria calculations in a Cu-S-H-O system. The amounts of the components should be kept to correct ratio corresponding to Cu<sub>2</sub>S and H<sub>2</sub>O using a command SET\_INPUT\_AMOUNTS in POLY3. Initially, O/H = 0.5 is given. Optimum O/H ratio is calculated with the desired calculation conditions. For example, to simulate one phase disappearing, you can FIX the phase with zero amount.

The *tcex11.TDB* file is used to access data and this is included with your installation.

These commands are of also of note and used in this example:

- [SET\\_NUMERICAL\\_LIMITS](#)
- [SET\\_REFERENCE\\_STATE](#)
- [SHOW\\_VALUE](#)

### ***tcex12 - Tabulation of Thermodynamic Data for Reactions***

This example shows a number of independent cases using the TABULATE\_REACTION (TAB) module to tabulate thermodynamic data for reactions.



This uses the SSUB database. A license is required to run the example.

These commands are of note and used in this example:

- [TABULATE\\_REACTION](#)
- [DEFINE\\_SPECIES](#)
- [ENTER\\_FUNCTION](#)
- [QUICK\\_EXPERIMENTAL\\_PLOT](#)
- [SET\\_SCALING\\_STATUS](#)
- [LIST\\_SUBSTANCES](#)
- [TABULATE\\_SUBSTANCE](#)

### ***tcex22 - Calculating an Adiabatic Flame Temperature***

This example examines a heat balance when  $C_3H_8$  is burned in oxygen by calculating the adiabatic flame temperature.



This uses the SSUB database. A license is required to run the example.

### ***tcex23 - Calculating a Paraequilibrium and the T0 Temperature in a Low Alloyed Steel***

This example calculates a paraequilibrium and the T0 temperature in a low alloyed steel.



This uses the TCFE database. A license is required to run the example.

### ***tcex28 - Pitting Resistance Equivalence (PRE) for a Duplex Stainless Steel***

This example calculates the Pitting Resistance Equivalence (PRE) for a duplex stainless steel.



This uses the TCFE database. A license is required to run the example.

These commands are of note and used in this example:

- [REJECT](#)
- [RESTORE](#)
- [DEFINE\\_ELEMENTS](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [EVALUATE\\_FUNCTIONS](#)
- [SET\\_START\\_CONSTITUTION](#)

### ***tcex29 - Calculating the Speciation of a Gas***

This example shows how to calculate the speciation of a gas.



This uses the SSUB database. A license is required to run the example.

These commands are of note and used in this example:

- [SET\\_INPUT\\_AMOUNTS](#)
- [LIST\\_CONDITIONS](#)
- [SET\\_CONDITION](#)

### ***tcex32 - Calculating Oxide Layers on Steel***

This example calculates oxide layers on a steel.



This uses the TCOX database. A license is required to run the example.

These commands are of note and used in this example:

- [SET\\_START\\_CONSTITUTION](#)
- [SET\\_REFERENCE\\_STATE](#)
- [APPEND\\_DATABASE](#)

### ***tcex41 - Calculating a Solubility Product***

Calculation of a solubility product.



This uses the TCFE database. A license is required to run the example.

These commands are of note and used in this example:

- [DEFINE\\_ELEMENTS](#)
- [REINITIATE\\_PLOT\\_SETTINGS](#)
- [SET\\_DIAGRAM\\_AXIS](#)

### ***tcex49 - Quasichemical Model Using GES***

This example shows how to enter parameters for a FACT quasichemical liquid model and how to calculate the sulfur activity using the GIBBS\_ENERGY\_SYSTEM (GES) module commands.



This example uses some GES commands that are not yet supported by GES6. Therefore, the example uses GES5.

These commands are of note and used in this example:

- [ENTER\\_ELEMENT](#)
- [ENTER\\_SYMBOL](#)
- [AMEND\\_ELEMENT\\_DATA](#)
- [ENTER\\_PARAMETER](#)
- [ENTER\\_SPECIES](#)
- [SET\\_REFERENCE\\_STATE](#)
- [SHOW\\_VALUE](#)

---

## Ternary Phase Diagram Examples

---

### ***tcex03 - Calculating an Isothermal Section***

This example calculates an isothermal section using the Ternary module. The FEDEMO database is used and included with your installation.

These commands are used:

- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [ADD\\_LABEL\\_TEXT](#)
- [PLOT\\_DIAGRAM](#)
- [SET\\_DIAGRAM\\_AXIS](#); [SET\\_DIAGRAM\\_TYPE](#); and [SET\\_AXIS\\_TYPE](#)
- [SET\\_REFERENCE\\_STATE](#)
- [CHANGE\\_STATUS](#)
- [SET\\_SCALING\\_STATUS](#)

### ***tcex05 - Calculating a Vertical Section in the Al-Cu-Si System***

This example calculates a vertical section in the Al-Cu-Si system and of a vertical section from Al to 10% Cu<sub>2</sub>Si. The ALDEMO database is used and included with your installation.

### ***tcex14 - Variations in Solidification of an Al-Mg-Si Alloy***

This example calculates the heat and heat capacity variations during solidification of an Al-Mg-Si alloy. The ALDEMO database is used and included with your installation.

These commands are of note and used in this example:

- [LIST\\_SYSTEM](#)
- [LIST\\_CONDITIONS](#)
- [SET\\_AXIS\\_TEXT\\_STATUS](#)

### ***tcex17 - The Pseudo-binary System - CaO-SiO<sub>2</sub>***

This example calculates the pseudo-binary system CaO-SiO<sub>2</sub> using the OXDEMO database, which is included with your installation.

The oxide demo (OXDEMO) database can be used both for pseudo-binary systems like the one in this case, CaO-SiO<sub>2</sub>, or for full ternary systems like Ca-Fe-O.



For a more advanced example in Graphical Mode using the TCS Metal Oxide Solutions Database (TCOX), see [T\\_17: Al<sub>2</sub>O<sub>3</sub>-MgO Phase Diagram](#).

These commands are of note and used in this example:

- [LIST\\_SYSTEM](#)
- [REJECT](#) and [RESTORE](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [LIST\\_STATUS](#)
- [DEFINE\\_COMPONENTS](#)
- [SET\\_CONDITION](#)
- [COMPUTE\\_EQUILIBRIUM](#) and [LIST\\_EQUILIBRIUM](#)

### ***tcex19a and tcex19b - Mapping of Univariant Equilibria with the Liquid in Al-Cu-Si***

- Part A. Step-by-step calculation using the POLY3 module.
- Part B. Using the TERNARY module, you can get the information on invariant reactions, such as temperature and compositions.

The database used with this example is included with the file named tcex19\_cost2.TDB. It is in the same folder as this example.

These commands are of note and used in these examples:

- [DEFINE\\_SYSTEM](#)
- [GET\\_DATA](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [SET\\_CONDITION](#) and [LIST\\_CONDITIONS](#)
- [ADVANCED\\_OPTIONS](#)
- [SAVE\\_WORKSPACES](#) and [READ\\_WORKSPACES](#)
- [APPEND\\_EXPERIMENTAL\\_DATA](#)
- [MAKE\\_EXPERIMENTAL\\_DATAFILE](#)

### ***tcex21 - Calculating a Ternary Isotherm in Fe-Cr-Ni***

This example calculates a ternary isotherm in Fe-Cr-Ni with a user-defined database included with the example in the file named *tcex21.TDB*.

These commands are of note and used in this example:

- [DEFINE\\_SYSTEM](#)
- [GET\\_DATA](#)
- [SET\\_CONDITION](#)
- [SET\\_DIAGRAM\\_TYPE](#)

### ***tcex33 - An Fe-Cr-C Isopleth***

This example is a benchmark calculation of an isopleth in the Fe-Cr-C system. The FEDEMO database is used and included with your installation.

### ***tcex37 - Calculating an Isothermal Section***

This example shows how to use command lines to enter and calculate an isothermal section using the POLY3 module. The FEDEMO database is used and included with your installation.

### ***tcex42 - Paraequilibrium Calculation - Formation of Para-pearlite - Isopleth***



This uses the TCFE database. A license is required to run the example.

This example uses an Fe-Mn-C system at 2.5%Mn Mass u-fraction to show a paraequilibrium calculation where there is formation of para-pearlite. It is an isopleth calculation and shows the STEP\_WITH\_OPTIONS command using the Paraequilibrium option.

### ***tcex43 - Paraequilibrium Calculation - Formation of Para-pearlite - Isothermal***



This uses the TCFE database. A license is required to run the example.

This example uses an Fe-Mn-C system at 700 °C to show a paraequilibrium calculation where there is formation of para-pearlite. It is an isothermal calculation and shows the STEP\_WITH\_OPTIONS command using the Paraequilibrium option.

## Multicomponent Phase Diagram Examples

---

### ***tcex06 - An Isopleth in Low Alloyed Fe-Mn-Si-Cr-Ni-C Steel***

This example calculates a multicomponent phase diagram using the `DEFINE_MATERIAL` command in `POLY` and the `TCFE` steel database. The material contains 1.5 %Cr + 0.4 %Mn + 3.5 %Ni + 0.3 %Si and 1 %C (by weight). These conditions and the temperature are set by the `DEFINE_MATERIAL` command.

After the first equilibrium results, a phase diagram is calculated with one axis variable as temperature and the other as the carbon content.



This uses the `TCFE` database. A license is required to run the example.

### ***tcex10 - Preventing Cr<sub>2</sub>O<sub>3</sub> Clogging in a Continuous Casting Process***

The example calculates an equilibrium with suspended or dormant phases and shows how to avoid Cr-oxide clogging in a continuous casting process.



This uses the `TCOX` database. A license is required to run the example.

The background of this example is that a manufacturer wanted to increase the Cr content of a material from 18 to 25 weight percent. They then had trouble in the continuous casting of this material because solid  $\text{Cr}_2\text{O}_3$  was formed.

Using Thermo-Calc to calculate the equilibria in the steel/slag system, a simple correction was found: modify the Mn or Si content to decrease the oxygen potential.

In Thermo-Calc, you can `FIX` a phase with zero amount to simulate how to avoid forming this phase. You then release one of the conditions, usually one of the compositions, and this composition is determined by the equilibrium calculation.

These commands are of note and used in this example:

- [`DEFINE\_SYSTEM`](#)
- [`CHANGE\_STATUS`](#)
- [`COMPUTE\_EQUILIBRIUM`](#)
- [`DEFINE\_SYSTEM`](#)

- [DEFINE\\_SYSTEM](#)
- [LIST\\_CONDITIONS](#)
- [LIST\\_EQUILIBRIUM](#)
- [LIST\\_STATUS](#)
- [SET\\_CONDITION](#)
- [SET\\_NUMERICAL\\_LIMITS](#)
- [SET\\_REFERENCE\\_STATE](#)
- [SHOW\\_VALUE](#)

### ***tcex23 - Calculating a Paraequilibrium and the T0 Temperature in a Low Alloyed Steel***

This example calculates a paraequilibrium and the T0 temperature in a low alloyed steel.



This uses the TCFE database. A license is required to run the example.

### ***tcex27 - Calculating Chemical Vapor Depositions (CVD)***

This example calculates the chemical vapor deposition (CVD).



This uses the SSUB database. A license is required to run the example.

These commands are of note and used in this example:

- [DEFINE\\_COMPONENTS](#)
- [SET\\_CONDITION](#)
- [ADD\\_INITIAL\\_EQUILIBRIUM](#)
- [LIST\\_AXIS\\_VARIABLE](#)
- [LIST\\_INITIAL\\_EQUILIBRIA](#)

## Examples Calculating Thermodynamic and Related Quantities

### ***tcex02 - Plotting Thermodynamic Functions***

This example shows how to plot thermodynamic functions in unary, binary, and ternary systems. It also involves working with partial derivatives and partial quantities.



This uses the SSOL database. A license is required to run the example.

These commands are used and may be of interest:

<a href="#">GOTO_MODULE</a>	<a href="#">GET_DATA</a>	<a href="#">SET_CONDITION</a>
<a href="#">SWITCH_DATABASE</a>	<a href="#">AMEND_PHASE_DESCRIPTION</a>	<a href="#">LIST_EQUILIBRIUM</a>
<a href="#">DEFINE_SYSTEM</a>	<a href="#">COMPUTE_EQUILIBRIUM</a>	<a href="#">SET_AXIS_VARIABLE</a>
<a href="#">SAVE_WORKSPACES</a>	<a href="#">STEP_WITH_OPTIONS</a>	<a href="#">SET_DIAGRAM_AXIS</a>
<a href="#">ENTER_SYMBOL</a>	<a href="#">PLOT_DIAGRAM</a>	<a href="#">SET_LABEL_CURVE_OPTION</a>
<a href="#">REJECT</a>	<a href="#">RESTORE</a>	<a href="#">LIST_SYSTEM</a>
<a href="#">LIST_STATUS</a>	<a href="#">CHANGE_STATUS</a>	<a href="#">SHOW_VALUE</a>
<a href="#">SET_INTERACTIVE</a>		

### ***tcex13 - Calculating the Binary Al-Ti Phase Diagram and its G Curve***

This example calculates an Al-Ti binary phase diagram and G curve using the BINARY module. The TCBIN database is used and included with your installation.

### ***tcex14 - Variations in Solidification of an Al-Mg-Si Alloy***

This example calculates the heat and heat capacity variations during solidification of an Al-Mg-Si alloy. The ALDEMO database is used and included with your installation.

These commands are of note and used in this example:

- [LIST\\_SYSTEM](#)
- [LIST\\_CONDITIONS](#)
- [SET\\_AXIS\\_TEXT\\_STATUS](#)

### ***tcex18 - A3 Temperature of a Steel***

This example calculates the A3 temperature of a steel and examines the influence of each alloying element on this temperature.

A3 temperature is the temperature where ferrite starts to form from austenite. You can easily read A3 from an Fe-C phase diagram. However, for complex multicomponent steels no simple diagram can be used. Using POLY, it is easy to find out the influence of each alloying element on the A3 temperature. This information is useful if you want to modify the compositions of a steel but keep A3 unchanged.



This uses the TCFE database. A license is required to run the example.

These commands are of note and used in this example:

- [DEFINE\\_ELEMENTS](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [COMPUTE\\_TRANSITION](#)
- [DEFINE\\_MATERIAL](#)
- [ENTER\\_SYMBOL](#)
- [SHOW\\_VALUE](#)

### ***tcex22 - Calculating an Adiabatic Flame Temperature***

This example examines a heat balance when  $C_3H_8$  is burned in oxygen by calculating the adiabatic flame temperature.



This uses the SSUB database. A license is required to run the example.

---

### ***tcex26 - The As-Ga Phase Diagram: Plotting the Partial Pressures of a Gas Species***

This is an example of plotting the partial pressures of a gas species along the solubility lines in the As-Ga phase diagram. The calculation makes it possible to monitor the input gases to a process of depositing solid As-Ga. The data is taken from the PG35 Binary Semi-Conductors database.

These commands are of note and used in this example:

- [REJECT](#)
- [RESTORE](#)
- [SET\\_DIAGRAM\\_AXIS](#)
- [SET\\_SCALING\\_STATUS](#)
- [SET\\_AXIS\\_TEXT\\_STATUS](#)
- [SET\\_LABEL\\_CURVE\\_OPTION](#)

### ***tcex34 - The Al-Zn Phase Diagram and its G Curve***

This example uses the BINARY module to calculate the phase diagram and G curves in the Al-Zn system. It uses the TCBIN database included with your installation.

### ***tcex39 - Calculating Reversible Carnot Cycles of a Heat Engine***

This example shows how to calculate the reversible Carnot cycle of a heat engine using one mole of an ideal gas with two fictitious species A and A2. The GES, POLY3, and POST modules are used, as well as the SUBDEMO database, which is included with your installation.

One application of the Second Law is to the efficiencies of heat engines, pumps and refrigerators. Whenever there is a difference of temperature, work can be produced - the principle of heat engines. The Gibbs energy also enables the prediction of the maximum work that a process may achieve. The goal of this example is to help relate the results to different thermodynamic quantities.

These commands are of note and used in this example:

- [REINITIATE](#)
- [ENTER\\_PARAMETER](#)

- [ENTER\\_SPECIES](#)
- [SET\\_PLOT\\_FORMAT](#)
- [ENTER\\_PHASE](#)

### ***tcex51 - Calculation of Molar Volume, Thermal Expansivity, and Density***

This example uses the POLY3 module to calculate the molar volume, thermal expansivity and density of the FCC\_A1, BCC\_A2 LIQUID, and liquid phases of C-Fe. The FEDMO database is used and this example is available to all users.

## Single Equilibrium Calculation Examples

---

### ***tcex07 - Single Equilibria in Low Alloyed Fe-Mn-Si-Cr-Ni-C Steel***

This example shows how to calculate a single equilibria in a low alloyed Fe-Mn-Si-Cr-Ni-C steel. There are two common ways to perform a single equilibrium calculation:

- Get data from a database, then in POLY use SET\_CONDITION and COMPUTE\_EQILIBRIUM.
- Go directly to POLY and use DEFINE\_MATERIAL.

The COMPUTE\_TRANSITION command is also used to determine the temperature or composition where one phase forms or disappears. It is the same as the CHANGE\_STATUS → SET\_CONDITION → COMPUTE\_EQILIBRIUM sequence of commands.

A STEP\_WITH\_OPTIONS command using the option *normal* is also used in addition to the other commands that should be becoming more familiar as you work through the examples.



This uses the TCFE database. A license is required to run the example.

### ***tcex18 - A3 Temperature of a Steel***

This example calculates the A3 temperature of a steel and examines the influence of each alloying element on this temperature.

A3 temperature is the temperature where ferrite starts to form from austenite. You can easily read A3 from an Fe-C phase diagram. However, for complex multicomponent steels no simple diagram can be used. Using POLY, it is easy to find out the influence of each alloying element on the A3 temperature. This information is useful if you want to modify the compositions of a steel but keep A3 unchanged.



This uses the TCFE database. A license is required to run the example.

These commands are of note and used in this example:

- [DEFINE\\_ELEMENTS](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [COMPUTE\\_TRANSITION](#)
- [DEFINE\\_MATERIAL](#)
- [ENTER\\_SYMBOL](#)
- [SHOW\\_VALUE](#)

### ***tcex20 - Adiabatic Decompression in a Geological System***

This example calculates the adiabatic decompression in a geological system using the geochemical database. The database is included with the file named PGEO.TDB, which is in the same folder as this example.

These commands are of note and used in this example:

- [DEFINE\\_SYSTEM](#)
- [DEFINE\\_COMPONENTS](#)
- [SET\\_INPUT\\_AMOUNTS](#)
- [LIST\\_CONDITIONS](#)
- [SET\\_CONDITION](#)
- [SHOW\\_VALUE](#)

---

## Potential Diagram Example

---

### ***tcex35 - Calculating a Potential Diagram***

This example uses the POTENTIAL module to calculate a potential diagram. Note that there are no commands for this module, you just follow the prompts. The SUBDEMO database is used and included with your installation.

---

## Examples of Scheil-Gulliver Calculations

---



[About the Scheil-Gulliver Solidification Simulations](#)

### ***tcex15 - Simulating the Solidification of a Cr-Ni Alloy***

This is an example of a solidification simulation of a Cr-Ni alloy. There is no back diffusion in the solid, i.e. this is a classic Scheil-Gulliver simulation. The FEDMO database is used and included with your installation.

These commands are of note and used in this example:

- [START\\_WIZARD](#)
- [COMPUTE\\_TRANSITION](#)
- [ADD\\_INITIAL\\_EQUILIBRIUM](#)
- [ENTER\\_SYMBOL](#)
- [ADVANCED\\_OPTIONS](#)
- [STEP\\_WITH\\_OPTIONS](#)
- [MAKE\\_EXPERIMENTAL\\_DATAFILE](#)
- [CHANGE\\_STATUS](#)

### ***tcex30a and 30b - Scheil Calculation for an Al-4Mg-2Si-2Cu Alloy***



[The Equilibrium Calculation Module \(POLY3\)](#) and [About the Scheil-Gulliver Solidification Simulations](#)

These examples show two ways to do a Scheil calculation for an Al-4Mg-2Si-2Cu alloy. In part A, you use the POLY3 module and the STEP\_WITH\_OPTIONS command with an EVALUATE option. Then in part B you use the SCHEIL module commands to do the same thing. Data from the COST2 database is used and included with the example in the file named *tcex30\_cost2.TDB*.

### ***tcex48 - Scheil Solidification with C Back Diffusion***

The FEDEMO database is used and the example is available to all users.

This is an example of Scheil solidification with simulated back diffusion of carbon in solid phases. The results are compared between a simple Scheil and equilibrium calculation. First perform a Partial Equilibrium Scheil by assigning carbon as a fast diffusing element. Plot solidification curve and a microsegregation diagram and save to file (so called .exp files). The "back diffusion" of C in solid phases is not from any calculation involving kinetics, but comes from the equilibration of chemical potential of C at each step in temperature, thus allowing each already solidified part of the system to change its C concentration.



See tcex54 for an example that calculates "real" back diffusion. This model requires a kinetic (mobility) database in addition to the thermodynamic one. Also see [About the Scheil-Gulliver Solidification Simulations](#) for details about the different Scheil options.

### ***tcex54 - Scheil Solidification of an Al-2.1Cu-1Si Alloy including Back Diffusion***

This is an example of a solidification simulation of a Al-2.1Cu-1Si alloy. The back diffusion option in the Scheil module is used, i.e. there is back diffusion in the primary phase. The example uses both the thermodynamic ALDEMO and mobility MALDEMO demo databases included with your installation.

### ***tcex57 - Scheil Solidification with Solute Trapping***

This is an example of a solidification simulation of a Al-7.5Si-0.2Cu system. The solute trapping option in the Scheil module is used. The example uses the thermodynamic ALDEMO demo database included with your installation.



See the Graphical Mode example [T\\_13: Scheil Solidification with Solute Trapping](#) for an additional comparison to Classic Scheil.

---

## Example Calculating Chemical Ordering

---

### *tcex16 - Second Order Transition of the Al-Fe System*

This example calculates the second order transition line in the BCC field of the Al-Fe system.



This uses the SSOL database. A license is required to run the example.

These commands are of note and used in this example:

- [REJECT](#)
- [RESTORE](#)
- [ADVANCED\\_OPTIONS](#)
- [SET\\_SCALING\\_STATUS](#)
- [DEFINE\\_SYSTEM](#)
- [MAKE\\_EXPERIMENTAL\\_DATAFILE](#)
- [LIST\\_EQUILIBRIUM](#)
- [APPEND\\_EXPERIMENTAL\\_DATA](#)
- [STEP\\_WITH\\_OPTIONS](#)

---

## Examples of Gibbs Energy System (GES)

---

### ***tcex31 - Using GES to Calculate CVM***

Using the Gibbs Energy System (GES) module, this example calculates the CVM and compares it with the sublattices of a fictitious A B system. You also learn how to overlay diagrams from two calculations.



This example uses some GES commands that are not yet supported by GES6. Therefore, we enforce the use of GES5.

These GES commands are of note and used in this example:

- [ENTER\\_ELEMENT](#)
- [ENTER\\_SPECIES](#)
- [ENTER\\_SYMBOL](#)
- [ENTER\\_PHASE](#)
- [ENTER\\_PARAMETER](#)
- [LIST\\_PHASE\\_DATA](#)

### ***tcex38 - Calculating the Morral “Rose”***

This example uses the Gibbs energy system (GES) module to calculate the Morral rose, which are miscibility gaps.

These commands are of note and used in this example:

- [REINITIATE](#)
- [ENTER\\_ELEMENT](#)
- [AMEND\\_ELEMENT\\_DATA](#)
- [ENTER\\_PARAMETER](#)
- [ENTER\\_PHASE](#)

### ***tcex39 - Calculating Reversible Carnot Cycles of a Heat Engine***

This example shows how to calculate the reversible Carnot cycle of a heat engine using one mole of an ideal gas with two fictitious species A and A2. The GES, POLY3, and POST modules are used, as well as the SUBDEMO database, which is included with your installation.

One application of the Second Law is to the efficiencies of heat engines, pumps and refrigerators. Whenever there is a difference of temperature, work can be produced - the principle of heat engines. The Gibbs energy also enables the prediction of the maximum work that a process may achieve. The goal of this example is to help relate the results to different thermodynamic quantities.

These commands are of note and used in this example:

- [REINITIATE](#)
- [ENTER\\_PARAMETER](#)
- [ENTER\\_SPECIES](#)
- [SET\\_PLOT\\_FORMAT](#)
- [ENTER\\_PHASE](#)

### ***tcex49 - Quasichemical Model Using GES***

This example shows how to enter parameters for a FACT quasichemical liquid model and how to calculate the sulfur activity using the GIBBS\_ENERGY\_SYSTEM (GES) module commands.



This example uses some GES commands that are not yet supported by GES6. Therefore, the example uses GES5.

These commands are of note and used in this example:

- [ENTER\\_ELEMENT](#)
- [ENTER\\_SYMBOL](#)
- [AMEND\\_ELEMENT\\_DATA](#)
- [ENTER\\_PARAMETER](#)

- [ENTER\\_SPECIES](#)
- [SET\\_REFERENCE\\_STATE](#)
- [SHOW\\_VALUE](#)

## ***tcex52 - Changing the Excess Models for Interaction Parameters in a Solution Phase***



### [Solution Models in the GIBBS \(GES\) Module](#)



This example uses some GES commands that are not yet supported by GES6. Therefore, the example uses GES5.

This example shows how to change the excess models for binary and ternary interactions in a solution phase, either

- through direct interactive amendments of phase descriptions within the GIBBS\_ENERGY\_SYSTEM (GES) module, or
- enforced by specific type-definitions given in a database file retrieved by the DATABASE\_RETRIEVAL module (also called the TDB module).

For binary excess models, the example changes it from the default *R-K model* to a *Mixed-Excess-Model*. The phase has to be a substitutional phase in this case.

For ternary extrapolation models, the example changes it from the default *R-K-M model* to a *Toop\_Kohler model*.

### *Notes Related to the Example*

The following is included in the macro file as part of the instructions but may be of use to new users:

### **Amending the binary excess model**

The default binary excess model is the Redlich-Kister Model for all the three associated binary interaction pairs (A-B, A-C and B-C) in the substitutional LIQUID solution phase (without sublattice) that consists of three elements (A, B and C).

---

Before changing this default binary excess model for the ternary LIQUID solution phase, one must have already entered the G parameters (for standard Gibbs energies of all pure end-members) and L parameters (for binary R-K excess interaction energies), as shown in the example.

In this particular example, we want to change from the default R-K binary excess model to the Mixed-Excess-Model (with three different binary excess models, namely Legendre, Polynom and Redlich-Kister models, applied to the A-B, A-C and B-C binaries, respectively), as demonstrated below:

For the A-B interaction, the Legendre binary excess model should be used (rather than the default Redlich-Kister Model), with the first species (i.e. A) as the independent constituent and the second species (i.e. B) as the dependent constituent, while the L parameters for the A-B interaction shall remain the same as those handled by the R-K model.

For the A-C interaction, the Polynom binary excess model should be used (rather than the default Redlich-Kister Model), with the second species (i.e. C) as the independent constituent and the first species (i.e. A) as the dependent constituent, while the L parameters for the A-C interaction shall remain the same as those handled by the R-K model.

### Amending the ternary extrapolation model

The default ternary excess model is the Redlich-Kister\_Muggianu Model (i.e., the MUGGIANU\_RESTOR method for ternary extrapolation based on binary parameters) for the associated ternary interaction terms; when no ternary L parameter is entered for that, such a default Redlich-Kister\_Muggianu Model is thus to be used for extrapolation from binary excess energies to ternary interactions in the substitutional LIQUID solution phase (without sublattice) that consists of three elements (A, B and C).

However, in this particular example as illustrated in the following, we shall change from this default R-K-M ternary excess model to the TOOP-KOHLER method for the ternary extrapolation method, with the species C as the Toop constituent, while the species A and B as the Kohler constituents (entering A and B, or B and A, as the basis constituent and first interacting constituent). This will implicitly enforce that, during the ternary extrapolation, only the A-B binary interaction parameters are utilized in accordance with the Kohler ternary extrapolation formula for A-B-C ternary interaction, while any other binary interaction parameters used in line with the Toop-Kohler ternary extrapolation formula (for the A-C-B and B-C-A ternary interactions). This makes the extrapolated ternary excess interaction terms different from those handled either by the default MUGGIANU\_RESTOR method or by the alternative KOHLER-ALL method.

Note that only when all the relevant binary excess energies in a ternary system are treated by the default Redlich-Kister Model (i.e., the Mixed-Excess-Model should have not been used), the MUGGIANU\_RESTOR method for ternary extrapolations is equivalent to the Redlich-Kister\_Muggianu Model, or the KOHLER-ALL method to the Redlich-Kister\_Kohler Model.

---

## Example Calculating the T0 Temperature

---

### ***tcex23 - Calculating a Paraequilibrium and the T0 Temperature in a Low Alloyed Steel***

This example calculates a paraequilibrium and the T0 temperature in a low alloyed steel.



This uses the TCFE database. A license is required to run the example.

## Examples Calculating Paraequilibrium

---

### ***tcex23 - Calculating a Paraequilibrium and the T0 Temperature in a Low Alloyed Steel***

This example calculates a paraequilibrium and the T0 temperature in a low alloyed steel.



This uses the TCFE database. A license is required to run the example.

### ***tcex42 - Paraequilibrium Calculation - Formation of Para-pearlite - Isopleth***



This uses the TCFE database. A license is required to run the example.

This example uses an Fe-Mn-C system at 2.5%Mn Mass u-fraction to show a paraequilibrium calculation where there is formation of para-pearlite. It is an isopleth calculation and shows the STEP\_WITH\_OPTIONS command using the Paraequilibrium option.

### ***tcex43 - Paraequilibrium Calculation - Formation of Para-pearlite - Isothermal***



This uses the TCFE database. A license is required to run the example.

This example uses an Fe-Mn-C system at 700 °C to show a paraequilibrium calculation where there is formation of para-pearlite. It is an isothermal calculation and shows the STEP\_WITH\_OPTIONS command using the Paraequilibrium option.

---

## Example Using Functions and Variables

---

### ***tcex44 - Exploring Variables and Functions***

This example uses variables and functions to predict properties e.g. proof strength for an austenitic stainless steel at elevated temperatures (20-550 °C). The example was created using an expression from [2000Eli].



This uses the TCFE database. A license is required to run the example.



In Graphical Mode there is a Yield Strength Property Model that does not require user input of empirical functions. See [About the Yield Strength Property Model](#).

### REFERENCE

[2000Eli] J. Eliasson, R. Sandström, Proof strength values for austenitic stainless steels at elevated temperatures. Steel Res. 71, 249–254 (2000).

## Example Using PARROT and EDIT\_EXPERIMENT

### *tcex36a - Using PARROT and EDIT EXPERIMENTS*



The example is divided into parts corresponding to the files in the **tcex36a** folder. It is recommended you open *tcex.36.readme* before continuing with this advanced exercise.



Also see the *Data Optimization User Guide*.

Each problem has "selected" experimental data. The example uses a fictitious binary A-B system where element A is BCC until it melts. Element B is BCC up to 1100 K and FCC above. There is a compound A2B stable in a limited temperature range.

- The creation of the "setup" file is described (*tcex36a.TCM*) where you enter the elements and phases into the Gibbs Energy System (GES) module
- The creation of the experimental data file is described (*tcex 36b.TCM*). When you run the TCEX36a macro, the contents of this macro file is automatically launched and the experimental data file created.
- The actual run in the PARROT module in order to obtain the result (*tcex 36cpd.TCM*)

There are many commands used in this advanced example (listed in alphabetical order):

<a href="#">ADVANCED_OPTIONS</a>	<a href="#">AMEND_ELEMENT_DATA</a>	<a href="#">AMEND_PHASE_DESCRIPTION</a>
<a href="#">APPEND_EXPERIMENTAL_DATA</a>	<a href="#">COMPILE_EXPERIMENTS</a>	
<a href="#">CONTINUE_OPTIMIZATION</a>	<a href="#">CREATE_NEW_EQUILIBRIUM</a>	<a href="#">CREATE_NEW_STORE_FILE</a>
<a href="#">DEFINE_COMPONENTS</a>	<a href="#">EDIT_EXPERIMENTS</a>	<a href="#">ENTER_PARAMETER</a>
<a href="#">ENTER_PHASE</a>	<a href="#">LABEL_DATA</a>	<a href="#">LIST_ALL_VARIABLES</a>
<a href="#">LIST_PHASE_DATA</a>	<a href="#">LIST_RESULT</a>	<a href="#">MACRO_FILE_OPEN</a>
<a href="#">OPTIMIZE_VARIABLES</a>	<a href="#">QUICK_EXPERIMENTAL_PLOT</a>	<a href="#">READ_WORKSPACES</a>
<a href="#">RESCALE_VARIABLES</a>	<a href="#">SAVE_PARROT_WORKSPACES</a>	<a href="#">SAVE_WORKSPACES</a>

---

<a href="#"><u>SELECT_EQUILIBRIUM</u></a>	<a href="#"><u>SET_ALL_START_VALUES</u></a>	<a href="#"><u>SET_ALTERNATE_CONDITION</u></a>
<a href="#"><u>SET_ALTERNATE_MODE</u></a>	<a href="#"><u>SET_FIX_VARIABLE</u></a>	<a href="#"><u>SET_OPTIMIZING_VARIABLE</u></a>
<a href="#"><u>SET_STORE_FILE</u></a>	<a href="#"><u>SET_WEIGHT</u></a>	

## Pourbaix Diagram Examples

---

### ***tcex40***

This is the first in a series of examples to demonstrate the advanced POURBAIX module calculations and plotting. This example automatically calculates and plots a Pourbaix diagram for 0.001 m Fe in a 0.1 m NaCl aqueous solution at 25 °C and 1 bar. Other diagrams, along various phase boundaries for the same interactions resulted from the same Pourbaix module calculation, are also plotted.



It is recommended you read the text file *TCEX40-README.txt*, which is located in the **Extended** subfolder.



Also see the *Pourbaix Diagrams Educational Material Guide*.

A so-called Pourbaix diagram is actually a phase diagram with independently-varied acidity (pH) and electropotential (Eh), for a heterogeneous interaction system at a certain bulk composition (that is by default always set as 1 kg of water dissolving a specified amount of metals and other solutes), under a defined temperature and pressure condition.

### *The PAQ2 Database*

The example uses the PAQ2 (TCS Public Aqueous Solution Database using the SIT aqueous solution model) included with your installation. The database contains an AQUEOUS solution phase and REF\_ELECTRODE phase (as a reference for electron in aqueous electrolyte systems), as well as some data for various solid phases (solution or stoichiometric) and gaseous mixture phase. The PAQ2 is specially designed for these example calculations of the so-called Pourbaix diagrams (i.e., pH-Eh plots) and other types of phase diagrams or property diagrams in some aqueous-bearing multicomponent heterogeneous interaction systems within the limited chemical framework of Fe-Co-Cr-Ni-C-N-S-H<sub>2</sub>O-NaCl, via the Single-Database Option in the advanced POURBAIX module or through the normal TDB-GES-POLY-POST routine.

### *Additional Examples in this Series*

Additional examples (TCEX40A, TCEX40B, TCEX40C, and TCEX40E) are provided to demonstrate options and features in the Pourbaix module and to show more advanced applications of the TCAQ and AQS aqueous solution databases [in the Multiple-Database

Option, i.e., with appended data from the SSOL and SSUB databases]. See [Pourbaix Examples TCEX40A, 40B, 40C, and 40E](#).



Example TCEX40D is deprecated as of version 2022a.

## ***tcex53 - Pourbaix Diagram Calculations for a Heterogeneous Interaction System***



Also see the *Pourbaix Diagrams Educational Material Guide*.

This example uses the PAQ2 database to calculate some Pourbaix diagrams. The DATABASE\_RETRIEVAL (TDB), GIBBS\_ENERGY\_SYSTEM (GES), POLY3, and POST modules are used for the Fe-X-H<sub>2</sub>O-NaCl heterogeneous interaction system, where X = Cr-Ni-Co.



The initial bulk composition of Fe-based alloy in this calculation is just preliminarily assigned, in which the BCC\_A2 and/or FCC\_A1 solution phase(s) are considered as of primarily interest. For practical calculations, one shall have more precise inputs for the initial bulk compositions of alloys.

### ***Step 1: Single-Point Calculations for H<sub>2</sub>O-NaCl system***

To demonstrate how to define the molality of NaCl in an aqueous-bearing heterogeneous interaction system.

### ***Step 2: Single-Point Calculations for Fe-X (X = Cr-Ni-Co) system***

To demonstrate how to define the initial amount of alloy in an aqueous-bearing heterogeneous interaction system. In this example, we are interested in only the BCC\_A2 and FCC\_A1 phases in the Fe-based alloy.

### *Step 3: Single-Point Calculations for Fe-Cr-Ni-Co + H<sub>2</sub>O-NaCl system*

### *Step 4: Pourbaix Diagram Mapping for Fe-Cr-Ni-Co + H<sub>2</sub>O-NaCl system - define the variables*

From the same mapping calculations you arrive at by the end of the example, you can plot more diagrams using a variety of X-Y axis variables.



For a list of pre-defined symbols, go to the end of the tcex53.TCM macro file (open it in a text editor).

## Pourbaix Examples TCEX40A, 40B, 40C, and 40E

---

These are the extended examples that are described in [Pourbaix Diagram Examples](#).

### TCEX 40A

This example shows you how to use Option 1 where you **start a completely new calculation**. It uses a single database. The Pourbaix diagram for 0.001 m Fe in a 0.1 m NaCl solution at 25 °C and 1 bar is automatically calculated and plotted.



A more advanced version of 40A can be found in TCEX40E, which also uses Option 1 but with multiple databases. However, licenses are required for three commercial Thermo-Calc databases.



See [POURBAIX\\_DIAGRAM Commands for Options 1 to 4](#).

A so-called Pourbaix diagram is actually a phase diagram with independently-varied electropotential (Eh) and acidity (pH), for an heterogeneous interaction system at a certain bulk composition (that is by default always set as 1 kg of water solving a specified amount of metals and other solutes), under defined temperature and pressure conditions. Some property diagrams, along various phase boundaries for the same interactions resulting from the same Pourbaix module calculation, are also plotted, without leaving Option 1.



At the end of this example, a new POLY3 file called *POURBAIX.POLY3* is automatically saved. The following examples (TCEX40B and TCEX40C) use this same POLY3 file for the calculations. Therefore it is important to make copies of this file and rename these to TCEX40A.POLY3, TCEX40B.POLY3, and TCEX40C.POLY3. This must be done outside of the Thermo-Calc software and after the TCEX40A calculation and plotting is complete. This is so that the required POLY3 file structure is not lost.



The reason you do not save from within Thermo-Calc is because of how the file is saved. When the command sequence (GO POLY3>SAVE TCEX40A.POLY3>Y) is used, the POLY3 workspace of the POURBAIX calculation results in the POLY3 module monitor and all the initial equilibrium points and the mapping/stepping calculation results are lost as well as parts of the POLY3 file structure (e.g., some definitions of the previously defined POURBAIX-type calculation system).



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.

## TCEX 40B

This example shows you how to use Option 2 where you **Open an old (existing) file and plot other property diagrams**.

Example 40A calculated and plotted the Pourbaix diagram for 0.001 m Fe in a 0.1 m NaCl solution at 25 °C and 1 bar. Example 40B reads the original POURBAIX.POLY3 file, and uses the calculated results to plot property diagrams for the same heterogeneous interaction system.



For example 40B, the POLY3 file is not modified. However, it is still recommended that a copy of the original POLY3 file is used so you do not lose the file structure and calculated results.



If you have a POLY3 file generated after choosing Option 4, that file can also be used for Option 2, and vice versa.

## TCEX 40C

This example shows you how to use Option 3 where you **Open an old file and make another Pourbaix calculation**. The previously calculated (and copied) POURBAIX.POLY3 file from TCEX40A is edited to make another Pourbaix calculation and plot.

The same chemical system is used (Fe-H<sub>2</sub>O-NaCl) but with different P-T-X (pressure-temperature-composition) conditions and without the Gaseous mixture phase. The new conditions are 0.01 m Fe in a 1 m NaCl solution at 200 °C and 1 kbar.



In this example, the GASEous mixture phase for the Pourbaix diagram is not considered.



The POLY3 file is modified and rewritten with the new conditions. As previously recommended, make a copy of the original POLY3 file and rename it (e.g. TCEX40C.POLY3) so you do not lose the file structure and new calculated results.



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.

After the default Pourbaix diagram is generated and refined, you can also use Option 3 to plot property diagrams along different phase boundaries for the same system. These settings are possible from within Option 3 and you do not need to go back to Option 2.

## **TCEX 40E**

This example shows you how to use Option 1 where you **start a completely new calculation**. It uses multiple databases (TCAQ, SSUB, and SSOL). The Pourbaix diagram for 0.001 m Fe<sub>90</sub>Co<sub>5</sub>Ni<sub>5</sub> alloy (9E-3 m Fe + 5E-4 m Co + 5E-4 m Ni) in an aqueous solution with 2E-2 m H<sub>2</sub>S and 3 m NaCl at 25 °C and 1 bar is automatically calculated and plotted.

You need licenses for three Thermo-Calc databases, i.e.

- A: AQS (TGG Aqueous Solution Database; HKF model) or TCAQ (TCS Aqueous Solution Database; SIT model);
- B: SSUB (SGTE Pure Substances Database)
- C: SSOL (SGTE Alloy Solutions Database) or TCFE (TCS Steels/Fe-Alloys Database) or similar.

The TCAQ database contains only an AQUEOUS solution phase (using the SIT model) and REF\_ELECTRODE phase (as a reference state for electron in aqueous electrolyte systems). The SSUB database is appended for the GASEous mixture phase and some pure solid phases (so-called secondary phases formed during aqueous-involving interaction processes), and the SSOL database is appended for various solid solution phases.

The Pourbaix module inside the software also accepts the AQS2 aqueous solution database (using the complete Revised HKF model) in the Multiple Database Option, in addition to the TCAQ2 aqueous solution database (using the SIT model).

## POURBAIX\_DIAGRAM Commands for Options 1 to 4

### Option 1: Start a Completely New Pourbaix Diagram Calculation

Option 1 is used to make a completely new Pourbaix diagram calculation and automatically plot a pH-Eh diagram. i.e., define a new chemical system; specify the T-P-X conditions; calculate the initial equilibria; perform the pH-Eh mapping calculation; plot pH-Eh and various property diagrams.



Many of the commands in Option 1 are also repeated for Options 2 to 4. The details for each command are included with this section unless it is specific to one of the other option choices. The order that the commands display on screen is also different and based on what is entered at the prompts. These prompts are also available in other commands and when applicable a link to that command is included.

```
Consider the GAS phase in calculating a Pourbaix diagram? /Y/
```

You can choose to ignore the GAS mixture phase on a calculated/plotted Pourbaix diagram. However, such a plot is actually not a complete Pourbaix diagram because the thermodynamically-stable phase boundaries between the AQUEOUS solution phase and GAS mixture phase will not be calculated.

- If you enter `Y` (the default), GAS mixture phase is always considered.
- If you enter `N` the GAS mixture phase is completely ignored.

```
Use single database? /Y/
```

If you keep the default you are prompted for a database.

```
Combined database: /PAQ2/
```

The default is PAQ2, a database included with all installations.

The following information is provided on screen if you chose `Y` at the first prompt, Need any instruction on the POURBAIX module? /N/.

## Define a Chemical System and its Initial Bulk Composition

Normally a Pourbaix diagram and its related equilibrium property in a heterogeneous interaction system are calculated under a certain bulk composition, which is usually 1 kg of water with defined amounts of dissolving solute substances. The solutes may either be completely dissolved into the aqueous solution, or be partially dissolved and simultaneously form some solid phases.

### CHEMICAL SYSTEM (ELEMENTS)

- Default defined elements (solvent H<sub>2</sub>O): H and O and specially assigned ZE (electron) and VA(vacancy).
- Prompt specified elements (solutes ELEM): Fe Ni Co Cr C N S Na Cl

### INITIAL BULK COMPOSITION

- Default defined composition (solvent): 1.0 kg of H<sub>2</sub>O
- Prompt specified composition (solutes): x mole of ELEM
- Mass of Water (weight) = 1 kg



When entering solutes in chemical formulas, the first element's letter must be in UPPER case, and second in lower case, for example NaCl, CO<sub>2</sub>, CoCl<sub>3</sub>, Fe<sub>0.93</sub>S, NaSO<sub>4-1</sub>, and H<sub>2</sub>SO<sub>4</sub>.

First solute

Enter a solute, e.g. Fe.

Molality of <first solute> [mol/kg] /.001/

Enter a molality for the first solute, e.g. Fe.

Second solute

Enter a second solute and its molality on the same line, e.g. NaCl .1.

Next solute

You can continue to enter solutes. Press <Enter> to skip to the next prompt.

The following information is provided on screen if you chose Y at the first prompt, Need any instruction on the POURBAIX module? /N/.

---

## Defined Chemical System and Initial Bulk Composition

Solutes have been split up into chemical elements and their mole numbers. If confirmed, the Pourbaix module will, in further steps, count the initial bulk composition in terms of chemical elements.

```
Confirm defined system and initial bulk composition? /Y/
```

If you keep the default (Y) thermodynamic data is retrieved from the database. Then it lists the database's phases for the defined system.

You can then reject or restore phases as required with the next prompts.

```
Reject phase(s) /NONE/
```

```
Restore phase(s): /NONE/
```

Then the program lists the phases retained in the system and you prompted whether this is OK or not.

```
OK? /Y/
```

If you answer Y, then a list of references for the assessed data is displayed on the screen. The next prompt is related to miscibility gap checks, global minimization, saving the information to a POLY3 file, and setting numerical limits.

```
Should any phase have a miscibility gap check? /N/
```

The database usually creates two or more "composition sets" for phases that can have miscibility gaps. However, for some phases this is not done automatically, for example the miscibility gap in the bcc phase in Fe-Cr is usually ignored. But if it is important to include a Cr-rich bcc phase specify this here. It costs some computation time and may make subsequent MAP or STEP processes more difficult to converge. To keep the default (not to have any such phase) press <Enter>. Otherwise answer Y and follow the additional prompts.

```
Enforce Global Minimization Technique in calculations? /N/
```

If you choose Y, the Global Minimization is done when possible (depending on the conditions setting), and a Global Minimization test is performed when an equilibrium is reached. This costs more computer time but the calculations are more robust.

```
Save all functions, variables and tables in POLY3 file? /Y/
```

Choose Y if you want to save all the defined symbols (functions, variables and tables) into the POURBAIX.POLY3 file, for further normal POLY3 calculations or normal POST post-processing. Choosing yes may result in a large POLY3 file.

```
Set numerical limits? /N/
```



[SET NUMERICAL LIMITS](#)

---

The Pourbaix module has a default setting for numerical limits.

- Maximum number of iterations: 20000
- Required accuracy: 1E-6
- Smallest fraction: 1E-15
- Approximate driving force for metastable phases: Y

If you kept the defaults for the above prompts, the program lists the default and pre-defined calculation conditions and you are prompted to confirm.

```
Confirm defined conditions? /Y/
```

Now Thermo-Calc calculates the start points. This can take a few moments. Near the end of the calculation there is a list of the default axis variables. The maximum pH limit is calculated precisely and determined automatically by the Pourbaix module as a function of the temperature-pressure conditions and initial bulk compositions of the current defined interaction system. You are prompted to accept these settings.

```
Accept the default settings for two mapping variables? /Y/
```

The program now lists the initial equilibria for mapping and continues to calculate the mapping. Then it sets the automatic diagram axes and starts to plot the diagram. But first there are several prompts to guide you.

```
Any missing phase boundary you could possibly think of? /N/
```

Sometimes the Pourbaix module may miss some phase boundaries during the mapping process. If you answer `Y` you can add additional starting point(s) into the calculation. You are now prompted to specify a pH-Eh coordinate pair, for instance (4.0, -0.15), which is close to the expected phase boundary. However, such additional pH-Eh point should not be outside the AQUEOUS/GAS(O<sub>2</sub>) or AQUEOUS/GAS(H<sub>2</sub>) boundaries; otherwise, the module will fail in finding any new starting point.

The next set of prompts relate to plotting.

```
Change the pH/Eh steps for smoother curves? /N/
```

You can modify the mapping steps of X/Y axes for smoother curves; sometimes it may help finding missing phase boundaries by decreasing the pH and Eh increments.

```
Zoom in? /N/
```

Change the scaling status for both X and Y axes. The program then asks for minimum and maximum values for the axes. In some cases values are suggested.

```
Change Curve-Label Option for the diagram? /N/
```

Change the Curve-Label option to one of the following:

- A: List stable phases along phase boundaries
- B: As A, but curves with same fix phase have same number
- C: List axis quantities
- D: As C, but curves with same quantities have same number
- E: As B, but curves with changing colours
- F: As D, but curves with changing colours
- N: No labels

Usually, Option E or B is good for phase diagrams such as pH-Eh plots, and F or D is good for property diagrams like pH-RC(Fe+2).

### SET\_LABEL\_CURVE\_OPTION

Add Label-Texts onto the Pourbaix diagram? /N/

Add label text to an area in the diagram. The label is written starting from the specified coordinates. Optionally, let the program automatically add the text by calculating an equilibrium at some specific coordinates and also make a label as the stable phase names in the area. However, such an automatic calculation may fail, especially for a Pourbaix diagram where either pH or Eh is not an intensive variables.

In most cases, the program is able to write the label if the coordinates and texts are specified.

Change the Subtitle of the diagram? /N/

This is the same as the POST module command SET\_CORNER\_TEXT → UPPER\_RIGHT <SUBTITLE>. The default subtitle is automatically assigned by the POURBAIX module and is based on the defined conditions in the system (temperature, pressure and initial bulk compositions).

If you choose `y` then the existing subtitle is displayed and you are prompted to enter a new subtitle or to have None.

New Subtitle /NONE/

You can change the subtitle for all the diagrams in the same POURBAIX run (i.e., within the same Option). The new subtitle that can be altered later at the same option, when further modifying the present graph or when refining any sequentially defined plot. By entering text (empty space at any position is allowed), the new subtitle is defined. By typing OLD, the existing subtitle is kept.

By pressing <Enter> (i.e., accepting None), the subtitle is deleted from all sequential plots, which can be set back later at the same option, too.

Change Axis-Texts? /N/

---

Change the axis text for both X and Y axes.

Further Refine the diagram in POST Module? /N/

With this option, you can further change the appearance the plot and use any command in the normal POST Module.



### [POST Module Commands](#)

Hard copy of the diagram? /N/

## **Recommended Responses**

- Hard copy of the diagram? /N/: Y
- CURRENT DEVICE: MS-Windows
- GRAPHIC DEVICE NUMBER /1/: 5
- NEW DEVICE: Postscript portrait mode
- USE POSTSCRIPT FONTS /YES/: Y
- SELECT FONTNUMBER /9/: 27
- NEW FONT: Times-Bold
- FONT SIZE /.35833/: .32
- PLOTFILE : /defaultname/: yourfilename

Save X-Y coordinates of curve on text file? /N/

Write all the current graphic information (generated and modified in the above procedure) on a text file using the DATAPLOT format (see the *DATAPLOT User Guide* or search the online help). This also makes it possible to merge two or more diagrams from separate but similar calculations, or put some experimental data points onto a calculated diagram. You do this by first writing them onto text files with this option and then edit them together (using a general text editor). The program asks for a file name. Accept the default (usually POURBAIX) or use another name. If an extension is not given, the written file is given an extension of \*.EXP.

Modify the diagram? /N/

Further modify the appearance of the created plot. This brings you back to the above prompts for changing various features.

Any more diagram? /N/

Make many other plots using the same POURBAIX module calculation file.

## Option 2: Open an Existing File and Plot Other Property Diagrams

### Copy and Rename the POLY3 file

At the end of a Pourbaix module calculation, a new POLY3 file is automatically saved.



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.



If you are using a POLY3 file more than once (as is done in the extended examples), it is recommended that you copy and rename the previously-generated POLY3 file. This is so that the required POLY3 file structure and Pourbaix results are not lost. Do this outside of Thermo-Calc i.e. do not save from within Thermo-Calc.



The reason you do not save from within Thermo-Calc is because of how the file is saved. When the command sequence (GO POLY3>SAVE <FILE NAME>.POLY3>Y) is used, the POLY3 workspace of the POURBAIX calculation results in the POLY3 module monitor and all the initial equilibrium points and the mapping/stepping calculation results are lost as well as parts of the POLY3 file structure (e.g., some definitions of the previously defined POURBAIX-type calculation system).

Open an existing POLY3 file created by the Pourbaix module (from a previous Pourbaix calculation Option 1 or 3 or a previous Stepping calculation Option 4), and selectively plotting other property diagrams. i.e., open the old GES and POLY3 workspaces; plot pH-Eh or various property diagrams.

File name /ABCDEF/

The name of a previously saved POLY3 file where the POLY3 and GES5 workspaces is to be read must be specified. You do not need to enter the file extension if it is the default .POLY3 name. The default file name (e.g. ABCDEF) is the most recent specified POLY3-file name or RESULT if there is no workspace already saved in the run. You can press <Enter> or type a specific file name to read the POLY3 and GES5 workspaces from the file.

For Windows, and at the prompt, an **Open file** window displays if a file name is not given or the file path is incorrect. The file type (i.e. POLY3) cannot be changed. Click **Open** or **Cancel** to continue with the selected file.

When the file opens, it confirms on screen what kind of calculation is contained in the POLY3 file. For example: *The loaded POLY3 file is of POURBAIX-MAPPING Calculation.*

On screen it also lists the system components and the default and predefined calculation conditions to enable you to review these.

Next you are given a set of properties that are available to be selected as axis variables for making other diagrams from the same calculation.

Abbreviation	Definition
pH	Acidity
Eh	Electronic potential (V)
Ah	Electronic affinity (kJ)
pe	Electronic activity [ $\log_{10} A_{\text{Cre}}$ ]
IS	Ionic strength
TM	Total concentration
Aw	Activity of water
Oc	Osmotic coefficient
MF (AQsp)	Mole fractions of aqueous species
ML (AQsp)	Molalities of aqueous species
AI (AQsp)	Activities of aqueous species
RC (AQsp)	Activity coefficients of Aq species

Where AQsp is the name of a specific aqueous species, and "\*" can be used as a wild sign for all AQSp. It is not necessary to give a complete aqueous species name. However, the entered AQsp name must be unique. If neither AQsp nor \* is given, all species are searched.



The rest of the prompts are the same as with [Option 1](#) and relate to plotting. Enter ? at any prompt for details.

## Option 3: Open an Existing File and Make Another Pourbaix Calculation

### Copy and Rename the POLY3 file

At the end of a Pourbaix module calculation, a new POLY3 file is automatically saved.



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.



If you are using a POLY3 file more than once (as is done in the extended examples), it is recommended that you copy and rename the previously-generated POLY3 file. This is so that the required POLY3 file structure and Pourbaix results are not lost. Do this outside of Thermo-Calc i.e. do not save from within Thermo-Calc.



The reason you do not save from within Thermo-Calc is because of how the file is saved. When the command sequence (GO POLY3>SAVE <FILE NAME>.POLY3>Y) is used, the POLY3 workspace of the POURBAIX calculation results in the POLY3 module monitor and all the initial equilibrium points and the mapping/stepping calculation results are lost as well as parts of the POLY3 file structure (e.g., some definitions of the previously defined POURBAIX-type calculation system).

Open an existing POLY3 file created by the Pourbaix module and make another Pourbaix diagram calculation i.e. open the old GES and POLY3 workspaces, adopt the defined chemical system, modify the T-P-X conditions, calculate the initial equilibria, perform the pH-Eh mapping calculation and plot pH-Eh or other property diagrams.



The rest of the prompts are the same as with [Option 1](#) or [Option 2](#). Enter ? at any prompt for details.

```
Modify Label-Texts on the Pourbaix diagram? /N/
```

Once some labels are written by the Add-Label-Text command onto the diagram, you can also modify the labels (changing their positions and text contents). The text-size cannot be changed. The program asks which label and you enter its new X/Y coordinates with the new text. The new text must not be longer than the old one.

```
Modify Label-Curve-Option on the Pourbaix diagram? /N/
```

Because some phase-region labels have been written onto the diagram, the Label-Curve-Option might need to be modified, normally to the Option N.

### **Important Information About Option 3**

The default definitions of the Eh and pH quantities in the advanced Pourbaix module (and in the ordinary TDB-GES-POLY calculation routines) should ALWAYS be as below:

$$\begin{aligned} \text{Eh} &= \text{MUR}(\text{ZE}) / \text{RNF} \\ \text{pH} &= -\log_{10}[\text{AI}(\text{H}+1, \text{AQUEOUS})] \\ &= -\log_{10}[\text{ACR}(\text{H}+1, \text{AQUEOUS}) * \text{AH}_2\text{O}] \end{aligned}$$

where RNF is the Faraday constant (96485.309 C/mol), and AH<sub>2</sub>O is the molecular weight of H<sub>2</sub>O (55.508435 g). MUR(ZE) is the electrochemical potential (ECP; in the unit of J/mol; w.r.t. the standard hydrogen electrode). ACR(H+1,AQUEOUS) is the site-fraction-based activity of the H+1 aqueous species in AQUEOUS solution phase, but AI(H+1,AQUEOUS) [that equals ACR(H+1,AQUEOUS)\*AH<sub>2</sub>O] is the molality-based activity of the H+1 aqueous species that should be used for defining the pH quantity.

Within an aqueous-bearing heterogeneous interaction system, the fundamental system-components must be H<sub>2</sub>O, H+1 and ZE, which are corresponding to the basic elements O & H and the hypothetical electron (ZE) in the aqueous solution phase. For the additional chemical elements in the system, their corresponding system-components shall be defined as in their element forms (such as Fe, Cr, Mn, Ni, Na, Cl, S) or (for some) in their molecular forms (e.g., NaCl, H<sub>2</sub>S). The reference state for the H<sub>2</sub>O component must always be defined as the solvent species H<sub>2</sub>O in the AQUEOUS solution phase under the current temperature (\*) and 100000 Pascal (i.e., 1 bar). The reference states for the H+1 and ZE components are by default set as their SER.

Various conventional properties of aqueous solute species I are converted in the following manners:

$$\begin{aligned} \text{ML} &= \text{Y}(\text{AQUEOUS}, \text{I}) * \text{AH}_2\text{O} / \text{YH}_2\text{O} \\ \text{RC} &= \text{ACR}(\text{I}, \text{AQUEOUS}) * \text{YH}_2\text{O} / \text{Y}(\text{AQUEOUS}, \text{I}) \\ \text{AI} &= \text{RC} * \text{ML} \\ &= \text{ACR}(\text{I}, \text{AQUEOUS}) * \text{AH}_2\text{O} \end{aligned}$$

where YH<sub>2</sub>O [i.e., Y(AQUEOUS,H<sub>2</sub>O)] and Y(AQUEOUS,I) are the site-fractions of solvent H<sub>2</sub>O and solute species I.

LIST THE DEFINED AXIS-VARIABLES:

$$\begin{aligned} \text{Axis No 1: LNACR}(\text{H}+1) & \text{Min: } -34.532525 \text{ Max: } 2.3025851 \text{ Inc: } 0.8 \\ \text{Axis No 2: MUR}(\text{ZE}) & \text{Min: } -150000 \text{ Max: } 200000 \text{ Inc: } 7718.85 \end{aligned}$$



The default settings (listed above) for two mapping variables [in terms of  $\ln\text{ACR}$  (H+1) and  $\text{MUR}(\text{ZE})$ , and their minimum/maximum values and increment steps] are covering the following pH-Eh ranges/steps:

pH: 0.00 -> 14.00, at a step of 0.35

Eh: -1.55 -> 2.07, at a step of 0.08 (V)

The maximum pH limit has been calculated precisely and determined automatically by the Pourbaix module, as a function of the temperature-pressure conditions and initial bulk compositions of the current defined interaction system.

## Option 4: Open an Existing File and Make another Stepping Calculation

Open an existing POLY3 file created by POURBAIX Module (from a previous POURBAIX calculation Option 1 or 3 or a previous STEPPING calculation Option 4), and make a normal STEPPING calculation. i.e., open the old GES and POLY3 workspaces; adopt the defined chemical system; specify one of the T-P-X conditions as the stepping variable; calculate the initial equilibria; perform the stepping calculation; plot various property diagrams.



Many prompts are the same as with [Option 1](#) or [Option 2](#). Enter ? at any prompt for details.

### Copy and Rename the POLY3 file

At the end of a Pourbaix module calculation, a new POLY3 file is automatically saved.



A reminder that Linux and Mac are case sensitive; when applicable ensure you enter file extensions with capital letters.



If you are using a POLY3 file more than once (as is done in the extended examples), it is recommended that you copy and rename the previously-generated POLY3 file. This is so that the required POLY3 file structure and Pourbaix results are not lost. Do this outside of Thermo-Calc i.e. do not save from within Thermo-Calc.



The reason you do not save from within Thermo-Calc is because of how the file is saved. When the command sequence (GO POLY3>SAVE <FILE NAME>.POLY3>Y) is used, the POLY3 workspace of the POURBAIX calculation results in the POLY3 module monitor and all the initial equilibrium points and the mapping/stepping calculation results are lost as well as parts of the POLY3 file structure (e.g., some definitions of the previously defined POURBAIX-type calculation system).



The stepping calculation modifies the POLY3 file but this can be used for other Stepping calculations inside the Pourbaix module (e.g. repeating Option 4) or in normal POLY3 module.

## Stepping Parameters

The following parameters are suitable for a STEPPING calculation for the defined aqueous-bearing system. Choose one of them as the stepping variable:

```
T Temperature (C)
P Pressure (bar)
pH Acidity
Eh Electronic Potential (V)
N(Comp) Mole of One Component
```

where "Comp" is the name of a specific component as listed below. One should not use any key component (H2O or H+1 or ZE) as such a stepping variable, because of that their compositional definitions and variations are determined by the AQUEOUS solution phase in the POURBAIX Module.

An interacting component defined as an element (such as Fe or Na or Cl) can be selected as a stepping variable.

However, if you want to make a stepping calculation with a non-element chemical formula (such as NaCl or CaS2) as the stepping variable, one must have made some necessary modifications in the POLY3 Module, before reaching this point, by conducting some POLY3 commands as given below (as an example for the system Fe-Na-Cl-H-O):

```
POLY_3: def-component H2O H+1 ZE Fe NaCl Cl
POLY_3: set-ref-state H2O AQUEOUS * 1E5
POLY_3: set-ref-state ZE REF_ELEC * 1E5
POLY_3: change-status phase REF-ELEC=SUSPEND
POLY_3: set-cond b(H2O)=1000 n(H+1)=0 n(ZE)=0
POLY_3: set-cond P=1E5 T=298.15**
```

---

```
POLY_3: set-cond n(Fe)=1E-3 n(Cl)=0 n(NaCl)=3 **
```

**\*\* Here the P-T-n(Fe) conditions can be set as what you wanted, while the initial n(NaCl) value must be set as between its minimum and maximum values that you are interested in. Later on, it will be prompted for specifying the n(NaCl) Min-Max-Step values of the stepping calculation.**

Stepping variable: /T/

The chosen stepping variable must be one of the suitable parameters listed above.

Minimum temperature (C) /15/

Maximum temperature (C) /35/

Increment in temperature (C) /1/

The AQUEOUS solution phase under the defined bulk composition and pressure condition is stable only between 0 and 99° C. However, you can make a stepping calculation outside this temperature range, where the AQUEOUS solution phase become metastable. In this case, the resulting POLY3 file is possibly able to be opened by any other option within the Pourbaix module.

---

## Example Using Tabulate Reaction (TAB)

---

### *tcex12 - Tabulation of Thermodynamic Data for Reactions*

This example shows a number of independent cases using the TABULATE\_REACTION (TAB) module to tabulate thermodynamic data for reactions.



This uses the SSUB database. A license is required to run the example.

These commands are of note and used in this example:

- [TABULATE\\_REACTION](#)
- [DEFINE\\_SPECIES](#)
- [ENTER\\_FUNCTION](#)
- [QUICK\\_EXPERIMENTAL\\_PLOT](#)
- [SET\\_SCALING\\_STATUS](#)
- [LIST\\_SUBSTANCES](#)
- [TABULATE\\_SUBSTANCE](#)

---

## Example Using REACTOR

---

### ***tcex24 - Simulation of the Silicon Arc Furnace***

This is a simple reactor model with output of gases at the top and output of condensed phases at the bottom. The gas phase from one segment flows to higher segments, 80% reacts in the first above, 15% in the second above and 5% in the third above.

The condensed phases flow downwards and all of it goes to the next lowest segment. Heat can be added at any module. The only way to specify the initial state of the reactants added to the reactor is to specify the heat content.



This uses the SSUB database. A license is required to run the example.

These commands are of note and used in this example:

- [DEFINE\\_SPECIES](#)
- [CREATE\\_STAGE\\_BOXES](#)
- [CREATE\\_DIVIDERS](#)
- [CREATE\\_PIPES](#)
- [START\\_SIMULATION](#)

---

## Examples with Thermophysical Properties

---

### ***tcex55 - Plotting Viscosity of Cr-Ni at 1873 K***

This is an example of including viscosity in the calculations.

Using the FEDEMO database, which is the free demonstration version of the TCS Steel and Fe-alloys Database (TCFE), the viscosity of metallic liquids is plotted at 1873 K for Cu-Ni and compared to experimental data from [2005Sat].

#### ***Reference***

[2005Sat] Y. Sato, K. Sugisawa, D. Aoki, T. Yamamura, "Viscosities of Fe–Ni, Fe–Co and Ni–Co binary melts," Meas. Sci. Technol. 16, 363–371 (2005).

The example includes an experimental data (\*.exp) file called `Cr-Ni_1873K.exp`.



The same example is also provided in Graphical Mode as **T\_12\_Viscosity\_in\_Cr-Ni.tcu**.

### ***tcex56 - Plotting Surface Tension of Cu-Zr at 1373 K***

This is an example of including surface tension in the calculations.

Using the ALDEMO database, which is the free demonstration version of the TCS Al-based Alloy Database (TCAL), the surface tension of liquid metallic is plotted at 1373 K for Cu-Zr and compared to experimental data from [2005Kra].

#### ***Reference***

[2005Kra] V. P. Krasovskyy, Y. V. Naidich, N. A. Krasovskaya, "Surface tension and density of copper–Zirconium alloys in contact with fluoride refractories," J. Mater. Sci. 40, 2367–2369 (2005).

The example includes an experimental data (\*.exp) file called `Cr-Zr_ALDEMO.exp`.



The same example is also provided in Graphical Mode as **T\_11\_Surface\_tension\_in\_Cu-Zr.tcu**.

## ***tcex58 and tcex59 - Molar Volume***

There are two examples that include molar volume in the calculations and use the ALDEMO database, which is the free demonstration version of the TCS AI-based Alloy Database (TCAL).

The examples calculate volumetric thermal expansion coefficients of the L12-type Al<sub>3</sub>Sc (tcex58) and the L12-type Al<sub>3</sub>Zr (tcex59), respectively. Both Al<sub>3</sub>Zr and Al<sub>3</sub>Sc are modeled as the same phase (named as AL3SC, since Al<sub>3</sub>Sc is stable while Al<sub>3</sub>Zr is metastable).

The examples include an experimental data (.exp) file called Al3X.exp, which is the same for both examples to read the data from [2015Sah].



The same example, combining both tcex58 and tcex59 is also provided in Graphical Mode as **T\_15\_Molar\_Volume\_for\_Al3Sc-Al3Zr.tcu**.

## **Reference**

[2015Sah] S. Saha, T. Z. Todorova, J. W. Zwanziger, Temperature dependent lattice misfit and coherency of Al<sub>3</sub>X (X=Sc, Zr, Ti and Nb) particles in an Al matrix. Acta Mater. 89, 109–115 (2015).

## ***tcex60 - Electrical Resistivity and Thermal Conductivity***

This example calculates the electrical resistivity (ELRS) and thermal conductivity (THCD) of pure Cu. It makes a step calculation over a wide temperature range, covering both the FCC\_A1 state and the liquid state. It plots thermal conductivity of the system (both FCC\_A1 and liquid) and that of a single phase (taking FCC\_A1 as an example), respectively. Also plots electrical resistivity of the system and that of FCC\_A1.

The example is available to all users as it uses the ALDEMO database, which is the free demonstration version of the TCS AI-based Alloy Database (TCAL).

The example includes two experimental data (\*.exp) files called 1972Ho\_cu\_thcd.exp and 1981Ho\_cu\_elrs.exp.



The same example is also provided in Graphical Mode as **T\_16\_Electrical\_Resistivity\_Thermal\_Conductivity\_Cu.tcu**

## References

[1972Ho] C. Y. Ho, R. W. Powell, P. E. Liley, Thermal Conductivity of the Elements. J. Phys. Chem. Ref. Data. 1, 279–421 (1972).

[1981Ho] C. Y. Ho, M. W. Ackerman, K. Y. Wu, T. N. Havill, R. H. Bogaard, R. A. Matula, S. G. Oh, and H. M. James. Electrical resistivity of ten selected binary alloy systems. CINDAS report 59, for Office of standard reference data National Bureau of Standards, Department of Commerce (1981).

## ***More Information about the Thermophysical Properties***

The thermophysical properties are progressively being added to the Thermo-Calc databases starting with Thermo-Calc software version 2020a.



You can learn more about the models by searching the help (press F1 when in Thermo-Calc).



You can find information on our website about the [properties that can be calculated](#) with Thermo-Calc and the Add-on Modules.

## Application Examples

---

### ***tcex09 - Calculating a Dew Point with POLY3***

This is an example calculating the dew point in the POLY3 module. The SUBDEMO database is used and included with your installation.

The STEP\_WITH\_OPTIONS and MAP commands are of note and used in this example.

### ***tcex10 - Preventing Cr<sub>2</sub>O<sub>3</sub> Clogging in a Continuous Casting Process***

The example calculates an equilibrium with suspended or dormant phases and shows how to avoid Cr-oxide clogging in a continuous casting process.



This uses the TCOX database. A license is required to run the example.

The background of this example is that a manufacturer wanted to increase the Cr content of a material from 18 to 25 weight percent. They then had trouble in the continuous casting of this material because solid Cr<sub>2</sub>O<sub>3</sub> was formed.

Using Thermo-Calc to calculate the equilibria in the steel/slag system, a simple correction was found: modify the Mn or Si content to decrease the oxygen potential.

In Thermo-Calc, you can FIX a phase with zero amount to simulate how to avoid forming this phase. You then release one of the conditions, usually one of the compositions, and this composition is determined by the equilibrium calculation.

These commands are of note and used in this example:

- [DEFINE\\_SYSTEM](#)
- [CHANGE\\_STATUS](#)
- [COMPUTE\\_EQUILIBRIUM](#)
- [DEFINE\\_SYSTEM](#)
- [DEFINE\\_SYSTEM](#)
- [LIST\\_CONDITIONS](#)
- [LIST\\_EQUILIBRIUM](#)
- [LIST\\_STATUS](#)
- [SET\\_CONDITION](#)

- [SET\\_NUMERICAL\\_LIMITS](#)
- [SET\\_REFERENCE\\_STATE](#)
- [SHOW\\_VALUE](#)

### ***tcex11 - Oxidation of Cu<sub>2</sub>S with an H<sub>2</sub>O/O<sub>2</sub> Gas Mixture***

This example demonstrates the oxidation of Cu<sub>2</sub>S with an H<sub>2</sub>O/O<sub>2</sub> gas mixture. Thermo-Calc is used to find the optimum O/H ratio (i.e. oxygen potential) as certain oxygen potential values can desulphurize Cu<sub>2</sub>S without forming copper oxides.

In Thermo-Calc, the problem reduces to perform equilibria calculations in a Cu-S-H-O system. The amounts of the components should be kept to correct ratio corresponding to Cu<sub>2</sub>S and H<sub>2</sub>O using a command SET\_INPUT\_AMOUNTS in POLY3. Initially, O/H = 0.5 is given. Optimum O/H ratio is calculated with the desired calculation conditions. For example, to simulate one phase disappearing, you can FIX the phase with zero amount.

The *tcex11.TDB* file is used to access data and this is included with your installation.

These commands are of also of note and used in this example:

- [SET\\_NUMERICAL\\_LIMITS](#)
- [SET\\_REFERENCE\\_STATE](#)
- [SHOW\\_VALUE](#)

### ***tcex20 - Adiabatic Decompression in a Geological System***

This example calculates the adiabatic decompression in a geological system using the geochemical database. The database is included with the file named PGEO.TDB, which is in the same folder as this example.

These commands are of note and used in this example:

- [DEFINE\\_SYSTEM](#)
- [DEFINE\\_COMPONENTS](#)
- [SET\\_INPUT\\_AMOUNTS](#)
- [LIST\\_CONDITIONS](#)
- [SET\\_CONDITION](#)
- [SHOW\\_VALUE](#)

## ***tcex24 - Simulation of the Silicon Arc Furnace***

This is a simple reactor model with output of gases at the top and output of condensed phases at the bottom. The gas phase from one segment flows to higher segments, 80% reacts in the first above, 15% in the second above and 5% in the third above.

The condensed phases flow downwards and all of it goes to the next lowest segment. Heat can be added at any module. The only way to specify the initial state of the reactants added to the reactor is to specify the heat content.



This uses the SSUB database. A license is required to run the example.

These commands are of note and used in this example:

- [DEFINE\\_SPECIES](#)
- [CREATE\\_STAGE\\_BOXES](#)
- [CREATE\\_DIVIDERS](#)
- [CREATE\\_PIPES](#)
- [START\\_SIMULATION](#)

## ***tcex27 - Calculating Chemical Vapor Depositions (CVD)***

This example calculates the chemical vapor deposition (CVD).



This uses the SSUB database. A license is required to run the example.

These commands are of note and used in this example:

- [DEFINE\\_COMPONENTS](#)
- [SET\\_CONDITION](#)
- [ADD\\_INITIAL\\_EQUILIBRIUM](#)
- [LIST\\_AXIS\\_VARIABLE](#)
- [LIST\\_INITIAL\\_EQUILIBRIA](#)

## ***tcex28 - Pitting Resistance Equivalence (PRE) for a Duplex Stainless Steel***

This example calculates the Pitting Resistance Equivalence (PRE) for a duplex stainless steel.



This uses the TCFE database. A license is required to run the example.

These commands are of note and used in this example:

- [REJECT](#)
- [RESTORE](#)
- [DEFINE\\_ELEMENTS](#)
- [AMEND\\_PHASE\\_DESCRIPTION](#)
- [EVALUATE\\_FUNCTIONS](#)
- [SET\\_START\\_CONSTITUTION](#)

## ***tcex29 - Calculating the Speciation of a Gas***

This example shows how to calculate the speciation of a gas.



This uses the SSUB database. A license is required to run the example.

These commands are of note and used in this example:

- [SET\\_INPUT\\_AMOUNTS](#)
- [LIST\\_CONDITIONS](#)
- [SET\\_CONDITION](#)

# **Data Optimization User Guide**

**Thermo-Calc Version 2024a**



---

# Introduction

---

**In this section:**

---

Introduction to Data Optimization .....	3
Data Optimization .....	4
The Least-Squares Method .....	6
The CALPHAD Approach .....	7

## Introduction to Data Optimization

---

This guide describes the basic idea behind data optimization, contains brief descriptions of the modules and file types that you work with while doing your optimization, as well as rough outline of overarching workflow that you typically follow when working with optimization.

This guide assumes that you are familiar with using Thermo-Calc Console Mode. It also assumes that you understand the principles of thermodynamics and are familiar with the theoretical foundations of computational thermodynamics.



Press F1 to search the help for information about all the commands available in the Thermo-Calc Console Mode, including PARROT and ED\_EXP. Note that the commands listed under PARROT and ED\_EXP are only commands that are specific to PARROT and/or ED\_EXP. These modules also contain some commands that are also available in POLY.



Thermo-Calc Console Mode [example tcex36](#) is a detailed example of assessment and optimization in Thermo-Calc.



More about data optimization and PARROT, including a video about how to use PARROT, is available on the Thermo-Calc Software [website](#).

## Data Optimization

---

In a thermodynamic database, each phase in a system is characterized using a mathematical model of its Gibbs energy. The Gibbs energy of a phase depends on various state variables. It can be defined individually even in a heterogeneous system with many stable phases since the properties of one phase are completely independent of the properties of the other phases in the system. In most alloy systems, the thermodynamic properties of a phase can be modeled by expressing how the Gibbs energy depends on temperature and composition. State variables such as pressure, volume or entropy can also serve as parameters in the Gibbs energy expression. The Gibbs energy of the whole system is the sum of the products of the Gibbs energy of each phase multiplied by the amount of that phase.

How the Gibbs energy of the phase varies with various state variables is determined by a mathematical model and how various adjustable parameters of the model is set. By optimizing these parameters, you can calculate the thermodynamic properties of a system under various conditions (by calculating phase diagrams for example). The thermodynamic properties themselves are functions of temperature, pressure or composition. They include, among other properties, standard enthalpy of formation, entropy, heat capacity, molar volume, thermal expansivity, compressibility and Curie temperature for magnetic transformations.

Data optimization is about adjusting the model parameters so that calculated equilibria fit well with experimental data. An important part of a data optimization is therefore collecting and assessing available experimental and theoretical information about phase equilibria and thermochemical properties of a system.

When you optimize the parameters of the model of a phase, you put what is called optimizing variables into the Gibbs energy expressions of those parameters. During the optimization, the values of these variables are varied in order to find a fit between the calculated equilibria and the experimental data that you base the optimization on. Once you have settled on a Gibbs energy expression for a certain parameter of a phase of the element, then you must stick to this expression in all future optimizations involving the element. If you do not do this, then you must re-optimize all previous systems that involve this element when the expression is changed.

The Thermo-Calc software allows you to optimize parameters that characterize not only binary systems, but also ternary system and systems of even higher orders. You can even optimize parameters that characterize systems of different orders at the same time.



More about data optimization and PARROT, including a video about how to use PARROT, is available on the Thermo-Calc Software [website](#).

---

## The Least-Squares Method

---

Data optimization in Thermo-Calc is based on the least-squares method for fitting values calculated on the basis of a model with observed quantities. The software is accordingly trying to find the optimizing variable values that lead the minimized sum of the squares of the differences between the calculated values and the observed quantities (that is, of the errors or residuals).

The least-squares method works best under the following conditions:

- The observed quantities have a Gaussian probability distribution.
- The observed quantities are only subject to random errors.
- The different observations (experiments) are uncorrelated
- The standard deviation of each observation can be estimated.
- The number of observations is large.
- The models used give precise predictions.

Of course, these conditions are usually not all met in a normal thermodynamic assessment. But even in non-ideal conditions, there is no known method that works better than the least-squares method.

## The CALPHAD Approach

---



CALPHAD is originally an abbreviation for *CAL*culat*ion of PH*ase *Diagrams*, but was later expanded to refer to *computer coupling of phase diagrams and thermochemistry*. More about the CALPHAD methodology, including some of its history, is available on the Thermo-Calc Software [website](#).

The data optimization functionality in Thermo-Calc works according to the CALPHAD approach to computational thermodynamics. This approach builds on the development of models that accurately represent thermodynamic properties for various phases. These models enable you to predict the thermodynamic properties of multicomponent systems-based data concerning binary and ternary subsystems. The predictions can take many factors into accounts, such as, for example, crystallography, type of bonding, order-disorder transitions and magnetic properties.

### References

For in-depth information about CALPHAD as well as data optimization, see *Computational Thermodynamics – The Calphad Method* by Hans Leo Lukas, Suzana G. Fries and Bo Sundman (Cambridge University Press, 2007). CALPHAD was given its first general description in *Computer Calculations of Phase Diagrams* by Larry Kaufman and Harold Bernstein (Academic Press, 1970). The book contains some case studies of optimizations using Thermo-Calc.

Information about the concept of ‘lattice stability’ (which is important for the development of multicomponent thermodynamic databases) can also be found in the paper *Hume-Rothery and Calphad Thermodynamics* by Larry Kaufman, published in *Calphad and Alloy Thermodynamics*, edited by Patrice E.A. Turchi, Antonios Gonis, Robert Shull (The Minerals, Metals & Materials Society, 2002), pp. 3-19.

---

# Working with Data Optimization in Thermo-Calc

---

**In this section:**

---

Data Optimization in Thermo-Calc .....	9
Data Optimization Workflow .....	10
The PARROT Module .....	12
The EDIT_EXPERIMENTS Module .....	13
Data Optimization in Other Modules .....	14
Optimization File Types .....	15
Optimization Workspaces .....	16
Method for Optimization and Simulation .....	17

---

## Data Optimization in Thermo-Calc

---



More about data optimization and PARROT, including a video about how to use PARROT, is available on the Thermo-Calc Software [website](#).

The actual performance of the optimization of a system is primarily done in the PARROT module, and in its sub-module ED\_EXP. The POLY and POST modules are used to calculate and plot property or phase diagrams based on the optimized variables. This allows you to visually assess how good a fit you have achieved between your calculated results and your experimental data.

The end result of a successful optimization is typically an updated Thermo-Calc database file (with filename extension \*.TDB) or a new database file.

## Data Optimization Workflow

---

When you carry out an optimization using Thermo-Calc, you typically follow a workflow as outlined below.

1. Collect experimental data about your system from various sources such as journal articles and reports.
2. Create a POP file in which you enter the experimental data you have collected.
3. Create a setup macro file in which you define your system and the variables you want to optimize. A useful general procedure is to first find a minimum set of variables that allows you to calculate most of the experiments.
4. Run the setup macro file.
5. Compile your POP file. This file often contains errors, which need to be corrected, and the file recompiled several times before the compilation is completed without errors.
6. In the ED\_EXP module, check that the experiments in your POP file reach equilibrium with reasonable results. If any experiments do not reach equilibrium with reasonable results, then try changing the starting values on equilibrium conditions (such as composition) to see whether the equilibrium can be computed after all. If this does not work, then you can temporarily exclude the experiment from the optimization. At a later stage in the optimization, when the optimizing variables have different values, you can check whether the experiments can be computed with reasonable results.
7. Back in the PARROT module, run the first optimization cycle and evaluate the feedback in the console. You can also plot diagrams in the POLY module that allow you to visually inspect the fit between calculated optimization results and the experimental data. If you are not satisfied with the fit, then enter the ED\_EXP module again and adjust the weights of the experiments in a way that is likely to improve the fit and run another optimization cycle. Reiterate until you are satisfied with the fit.
8. Once you get the optimization stable and smooth with the minimum set of variables chosen in step 3 above, try using different sets of variables to see whether improvements are possible.
9. When you are satisfied with the fit between calculated results and experimental data with your final set of optimizing variables, update your setup file and POP file. In the setup file, enter the calculated optimizing variable values as the variables' start values. In the POP file, enter the final weights of the experiments. With these files

updated, you can easily regenerate the parameter values of your optimized system. Having the files updated also makes it easier to optimize the system again in light of new data or new theoretical models.

10. Finally, update the database with your calculated results or create a new database with information about the system that you have optimized.

## The PARROT Module

---



More about data optimization and PARROT, including a video about how to use PARROT, is available on the Thermo-Calc Software [website](#).

The PARROT module consists of a comprehensive subroutine package for data evaluation of thermodynamic model parameters from experimental data. The module has 99 variables that you can use for optimization and the module can handle a thousand experimental measurements in an optimization. However, there are limits on the simultaneous numbers of variables and experiments. These limits are listed at each optimization.



The PARROT module is part of the Thermo-Calc software. The module is not included in the software development kits (SDKs).

The experimental data is entered and edited in a sub-module of PARROT: [The EDIT EXPERIMENTS Module](#).

---

## The EDIT\_EXPERIMENTS Module

---

The EDIT\_EXPERIMENTS module, or ED\_EXP for short, is used for entering, evaluating, and editing experimental data. To enter the module, go to [The PARROT Module](#) and then issue the command [EDIT\\_EXPERIMENTS](#).

Since ED\_EXP uses the POLY module for doing equilibrium calculations of experimental data points, many POLY commands are also available in the ED\_EXP module. However, the commands may work somewhat differently in the two modules. There are also unique ED\_EXP commands which are not found in POLY.

When you execute ED\_EXP commands, the PARROT workspace is modified. However, note that you must always first use [READ\\_WORKSPACES](#) in ED\_EXP before doing anything in the module. The command loads the experimental data that you are going to work with during the optimization. Furthermore, before leaving ED\_EXP you must use [SAVE\\_WORKSPACES](#) or the results of your work in the ED\_EXP module is lost.

---

## Data Optimization in Other Modules

---

All kinds of thermodynamic data, calculated equilibrium states or dynamic parameters are transferred back and forth between the PARROT and the GIBBS module as well between these and the POLY module. Whenever an optimization run is performed, PARROT calls the GIBBS module for stored system definition data and model parameters.

In Thermo-Calc, the GIBBS module (the Gibbs Energy System) handles the models of the various phases that can form in a multicomponent system. It stores thermodynamic data and performs various Gibbs energy equilibrium calculations. It contains subroutines to analytically calculate the first and second partial derivatives of integral Gibbs energy with respect to any set of variables. Many thermodynamic models for various types of substances are implemented in the module. Some commands that are available in the PARROT module are also available in the GIBBS module.

In the GIBBS module, parameters of the Gibbs energy models are referred to as *TP-functions*.

PARROT calls the POLY module for equilibrium calculations. The POLY module is used for calculating and storing complex heterogeneous equilibria.

## Optimization File Types

These are the important files you work with when optimizing.

<i>File type</i>	<i>Description</i>
POP file	The POP file (Parrot OPTimization file) is the file that contains all the experimental data that you use for optimizing Gibbs energy values for your system. The POP file is a plain text file that contains ED_EXP commands. By default, the file has the filename extension *.POP in Thermo-Calc.
Setup file	The setup file is a Thermo-Calc macro file (*.TCM). This file typically contains commands that define your system, opens the PARROT workspace (and associated GIBBS and POLY workspaces) and sets optimizing variables.
Other macro files	It is often useful to have other macro files than the setup file at hand while you are doing your optimization. For example, it is convenient to have a macro file that automatically plots phase diagrams or other diagrams based on the latest values of your optimizing variables. An optimization process typically involves many optimization cycles, so this operation needs to be done many times during a typical optimization.
Workspace files	The PARROT module has its own workspace with dynamic memories, similar to the workspaces of POLY and GIBBS. It is stored and updated in a PARROT workspace file with the filename extension *.PAR. The workspace file is a binary file whose format is hardware dependent. This means that the format is unique for each type of CPU. Hence, a workspace file saved on a computer with one type of CPU cannot be used on a computer with another type of CPU.
EXP-file	<p>The EXP-file is a plain text file with information in the form of DATAPLOT Graphical Language commands. These commands can specify some or all of the data points in your POP file. This allows you to plot these data points on top of your plotted optimization results (with the POST commands <a href="#">APPEND_EXPERIMENTAL_DATA</a> or <a href="#">QUICK_EXPERIMENTAL_PLOT</a>). You can then visually assess the fit between these results and the experimental data. The PARROT workspace is not influenced in any way by the creation or use of EXP-files.</p> <p> <i>DATPLOT User Guide</i> included with this documentation set.</p>

---

## Optimization Workspaces

---

When writing the setup file, you enter a [CREATE\\_NEW\\_STORE\\_FILE](#) command which creates a PARROT workspace file. Alternatively, select an existing PARROT workspace file with [SET\\_STORE\\_FILE](#). The workspace file is automatically updated and saved with the latest optimization results (after each use of the [OPTIMIZE\\_VARIABLES](#) command). You can also explicitly instruct Thermo-Calc to save the workspace using [SAVE\\_PARROT\\_WORKSPACES](#). If you want to get rid of your latest changes and return to the state of your workspace when it was last saved, use [READ\\_PARROT\\_WORKSPACES](#).

When the PARROT workspace is saved, parts of the POLY and GIBBS workspaces are also saved to the PARROT workspace file, along with the PARROT workspace itself. Using the POLY command [SAVE\\_WORKSPACES](#) creates a new POLY3 file, nothing is saved to the PARROT workspace file.

If you do any POLY calculations based on data in the current workspace file, then a \*.POLY3 file is created, which contains a copy of the current set of parameters. If you continue on your optimization and read the old \*.POLY3 file, then the new set of parameters is overwritten with the old set. It is therefore recommended that you never read a \*.POLY3 or \*.GES5 file while you are doing an optimization.

---

## Method for Optimization and Simulation

---

The PARROT module typically uses ordinary POLY minimization for equilibrium calculations. The global minimization technique that is used in POLY cannot be used because it automatically creates new composition sets which corrupt the PARROT workspace. Furthermore, the equilibrium definitions that you use as input for your optimization must each be attributed to a specific phase. Some of these definitions may specify some local or partial equilibrium state rather than a global minimum. However, it is possible to use the global minimization technique for some equilibrium calculations by using the ED\_EXP command [ADVANCED\\_OPTIONS](#) with the `Toggle_Alternate` option.

Optimization in PARROT is performed on the basis of a ‘maximum likelihood’ principle. According to this principle, the best fit between various calculated results and all the input of experimental data is found where the sum of the square of the weighted residuals is at its minimum.

Typically, the results of an optimization is considered to be better the fewer optimizing variables that are needed to get the same level of fit between computed results and experimental data. If you can get almost the same fit using eight instead of twelve parameters, then this should be considered to be a significant improvement. When more parameters are used, the values of the individual parameters tend to become unrealistic. This is particularly true when it comes to temperature-dependent parameters. However, relatively small differences in the number of parameters are often insignificant. For example, it does not typically matter whether you used 24 or 25 parameters to reach a certain degree of fit.

It is often difficult to compare the quality of different optimizations based on the number of optimizing variables that have been optimized. Since you may rely more or less heavily on different pieces of available experimental data (and do so with good reason), it is possible that two different users could end up with very different numbers of parameters for the same system. For example, suppose you optimize the Fe-Al system to incorporate it into an Al database, while someone else optimizes the same system to incorporate it into an Fe database. In this situation, the two of you would probably make different judgements about which experimental information your calculated results must have a good fit with.

---

## Creating the POP File

---

Before starting an optimization in Thermo-Calc, you must collect relevant experimental data about your system that is available in the research literature. This data is then entered in the POP file. The experimental data points and measurements are entered in this plain text file in the form of ED\_EXP commands.

The syntax of the POP file is in principle independent of the models used for describing the phases in the system.

### In this section:

---

Experimental Information .....	19
POP File Syntax .....	22
Entering Experimental Information .....	24
Using Stability Conditions .....	35
Grouping Equilibria with Labels .....	36
Dividing the POP File into Several Blocks .....	37

## Experimental Information

---

An optimization of thermodynamic model parameters should be based on a range of reliable experimental results that is as wide as possible, as well as on empirical or semi-empirical correlations and theoretical considerations. During an optimization in Thermo-Calc, many kinds of thermochemical and thermophysical data can be mixed and used together.

The information needed for an optimization represent measurements of thermodynamic parameters in the system at equilibrium. These can be measurements of any of the thermodynamic quantities that can be set as conditions in Thermo-Calc. The information may represent measurements of activities or enthalpies in single-phase regions for example, or solubilities or transformation temperatures in a multi-phase region.

For each equilibrium that is used in the optimization, at least one quantity must have been measured and you also need to know the conditions that must be set to compute the system's equilibrium state. For example, consider a binary system in a single-phase region. Suppose that you have measured the temperature, pressure, composition and the chemical potential. Three of these quantities are necessary to specify the equilibrium state and the fourth can be used as experimental information to model the phase.

If you have a two-phase region and know the temperature and pressure conditions, as well as which the stable phases are, then you could use the compositions of one or both phases as experimental data. The former information about the conditions is sufficient to determine the equilibrium. The experimental information can then be used to optimize the parameters that are used to characterise and model the system.



For examples how to enter various kinds of experimental data in the POP file, see [Experiments and Experimental Data](#).

### ***Conflicting or Missing Information***

You are likely to sometimes have inconsistent or conflicting experimental information. In principle, all available experimental data should still be entered into the POP file, unless you have good reasons to exclude some information. For example, if you have reason to believe that the samples used in some experiments were not pure, then these experiments could be excluded from the POP file.

For some systems, you have very little experimental data to go on. When this is the case, do not try to fit the available data uncritically with high accuracy without considering possible errors. You can look for data in systems with similar elements to get an idea about what the

---

reasonable estimates of data that is lacking could be. Or you can use calculations from first principles or semi-empirical methods to get reasonable estimates that can be used as experimental data.

### ***Invariant Equilibria Information***

The most valuable experimental information for an optimization is information about stable invariant equilibria. An invariant equilibrium is such that neither pressure, temperature nor composition can be changed without there being a change in the set of phases present in the system. It is recommended that all information about invariant equilibria for a system is included in the POP file. These equilibria need not all be based on explicit experimental measurements of thermodynamic properties.

Reasonable estimates from available experimental data are often useful to include in the POP file. Such estimates, which should be based on the available experimental data, helps Thermo-Calc find a good set of start values on the optimization variables. However, at the end of the assessment, these estimated equilibria should be excluded and you should complete the optimization based only on the experimental information that is based on measurements.

With some experience from phase diagram evaluation, it is possible to make reasonable estimates of metastable invariant equilibria. Such estimates helps reduce the number of phases that are assessed simultaneously during the optimization. For example, you may assume that a certain intermediate phase does not form in a system. Then extrapolate the liquidus curves below the stable three-phase equilibria and finally, estimate temperatures and compositions of metastable three-phase equilibria between two other phases and the liquid.

Another useful technique is to extrapolate a liquidus line from a peritectic equilibrium to estimate the congruent melting temperature of a compound. This estimate may be more useful than the information about the peritectic equilibrium itself as the equilibrium involves only two phases.

### ***Equilibria in Higher-Order Systems Information***

When you optimize a binary system, the available experimental information can often be described equally well by very different sets of parameters. Information about ternary and higher order systems that include the components of the binary system are then often useful for optimizing the binary system (the same is true of information about quaternary system with respect to optimizing ternary systems). Often, it is only when you extrapolate the optimized parameters to a higher-order system that you can judge which of the sets of model parameters that is best. Sometimes information from several ternary systems is

required to make a reliable judgement about which set of parameters best describes a particular binary system.

### ***Heat Capacity Information***

If you want to optimize your system based on data about heat capacities, then the corresponding parameters should first be optimized separately before you perform the general optimization of your system. The parameters should be kept fixed during the general optimization. This is because it is normally not possible to optimize heat capacity data together with other data in a meaningful way. If you do this, then it is very likely that the parameters related to the heat capacity are badly determined.

## POP File Syntax

---

It is recommended that you specify an experimental equilibrium as close as possible to actual experimental conditions. Ideally, there is information about the set of stable phases, temperature, pressure and some or all compositions for the equilibrium.

In addition, it is recommended that you keep the POP file well-organised and extensively commented. This is especially important if someone else reassesses the system when new experimental information is available.

If you have too many equilibria defined in your file, or if you define equilibria with different sets of components in the same file, then you must divide the POP file in *blocks* (using the [FLUSH\\_BUFFER](#) command). You know whether you have defined too many equilibria for one block when you compile the file. A too large number of equilibria leads to an error message when the file is compiled.

If you use the [DEFINE\\_COMPONENTS](#) command, then it must be the first command in the POP file (or if you have several blocks, [DEFINE\\_COMPONENTS](#) can also be put as the first command after [FLUSH\\_BUFFER](#)). This is because the command automatically reinitiates the current workspace, so the effect of any commands placed before it are not saved.



The last command in the POP file should always be [SAVE\\_WORKSPACES](#).

### **Legal Commands in a POP File**

The following commands can be used in a POP file.

- [ADVANCED\\_OPTIONS](#)
- [CHANGE\\_STATUS](#)
- [COMMENT](#)
- [CREATE\\_NEW\\_EQUILIBRIUM](#)
- [DEFINE\\_COMPONENTS](#)
- [ENTER\\_SYMBOL](#)
- [EVALUATE\\_FUNCTIONS](#)
- [EXPERIMENT](#)
- [EXPORT](#)

- [FLUSH\\_BUFFER](#)
- [IMPORT](#)
- [LABEL\\_DATA](#)
- [SAVE\\_WORKSPACES](#)
- [SET\\_ALL\\_START\\_VALUES](#)
- [SET\\_ALTERNATE\\_CONDITION](#)
- [SET\\_CONDITION](#)
- [SET\\_NUMERICAL\\_LIMITS](#)
- [SET\\_REFERENCE\\_STATE](#)
- [SET\\_START\\_VALUE](#)
- [SET\\_WEIGHT](#)
- [TABLE\\_HEAD, TABLE\\_VALUES and TABLE\\_END](#)

---

## Entering Experimental Information

---

A POP file in Thermo-Calc (\*.POP) mainly consists of descriptions of equilibria, each equilibrium describing a data point that is based on an experiment from the existing literature on your system. An equilibrium is specified using ED\_EXP commands.

The following is an example of a specification of an equilibrium with a single stable phase in the Au-Cu system. Equilibria with a single stable phase are often from experiments where enthalpies of mixing or chemical potentials have been measured.

```
CREATE_NEW_EQUILIBRIUM 1 1
CHANGE_STATUS PHASE LIQUID=FIX 1
SET_CONDITION T=1379 P=1E5 X(LIQUID,AU)=0.0563
SET_REFERENCE_STATE AU LIQ * 1E5
SET_REFERENCE_STATE CU LIQ * 1E5
EXPERIMENT HMR=-1520:200
COMMENT Measurement by Topor and Kleppa, Met trans 1984
```

---

## Creating a New Equilibrium

The first command in an equilibrium specification is always [CREATE\\_NEW\\_EQUILIBRIUM](#). This command takes two integers as arguments. The first integer should specify a unique identifier that can be used later to refer to the equilibrium in question. The second integer is an initialisation code 0, 1 or 2:

- Code 1: This means that all components are entered but all phases are suspended. This code is appropriate in most cases.
- Code 0: This means that all components and phases in the equilibrium are suspended initially. This code is needed if you are using experimental data about systems of different orders simultaneously, such as data about both binary and ternary systems for example.
- Code 2: This means that all components and phases are initially entered.

For example, the first command in the example above creates an equilibrium data point with the unique identifier 1. All components of the system are entered but all phases are suspended:

```
CREATE_NEW_EQUILIBRIUM 1 1
```

## Setting Equilibrium Conditions and Uncertainty

All the equilibrium-related commands that are placed in between two [CREATE\\_NEW\\_EQUILIBRIUM](#) commands are interpreted as specifying the conditions of the equilibrium created by the first command. The values of these conditions should be based on the conditions specified in the experiment that you base the equilibrium definition on.

In the example, the [CHANGE\\_STATUS](#) command specifies that the equilibrium is a single-phase equilibrium with the liquid phase.

```
CHANGE_STATUS PHASE LIQUID=FIX 1
```

Furthermore, the temperature is 1379 K, the pressure is 1 bar (1E5 Pa) and the mole fraction of Au is 0.0563.

```
SET_CONDITION T=1379 P=1E5 X(LIQUID,AU)=0.0563
```

Sometimes you have reason to think that some information about an equilibrium condition may not be accurate. If this is the case, then you specify the degree of uncertainty for a condition with a colon after the condition quantity, directly followed by either an absolute value or a percentage. For example, if you have reason to doubt the reliability of the Au mole fraction measurement, then you could write:

```
SET_CONDITION T=1379 P=1E5 X(LIQUID,AU)=0.0563:10%
```

This indicates that you think the Au mole fraction is between 0.0507 and 0.0619. PARROT calculates the equilibrium twice, once for 0.0507 and once for 0.0619, and then computes the change of each experimental value between these two equilibria. This change is used to modify the uncertainty specified for the experimental value.



### [Entering the Experimental Value and its Uncertainty](#)

The possibility of specifying uncertainties on conditions may be useful when you have experimental data about a ternary system. In a binary system, a tie-line is determined by the two phases, the temperature and the pressure. Measurements of the compositions of both phases can then be used as experimental data. To specify the tie-line in a ternary two-phase equilibrium one of the four compositions must also be set as a condition. If the measurement of this composition has the same uncertainty as the other compositions, then you can assign this same degree of uncertainty both to the composition selected as a condition and the compositions selected as experimental data. Alternatively, the sample (overall) composition could be used as a condition and all four phase compositions could be used as experiment values. These phases must have the status *entered* (not fixed) since the relative amount of each is unknown.

## Setting Reference States

If you want to specify a non-default reference state for one or several of your system components, then you do this with the [SET\\_REFERENCE\\_STATE](#) command. In the following example, the reference states for Au and Cu are set to the liquid phase at current temperature (\*) and a pressure of 1E5 Pa.

```
SET_REFERENCE_STATE AU LIQ * 1E5
SET_REFERENCE_STATE CU LIQ * 1E5
```



In order for these reference states to be taken into account, the <sub>R</sub> suffix must be used when the measurement of the enthalpy per mole of the system is entered for an experiment.

### The Reference State Suffix R



#### [Suffixes](#)

You can use the reference state suffix <sub>R</sub> for some thermodynamic variables to calculate their value with respect to a reference state that you have previously set for a system component with the SET\_REFERENCE\_STATE command in POLY (or in response-driven modules such as the POURBAIX module). The value of energy-related variables that are used with the <sub>R</sub> suffix depends on the reference states of all the components in the defined system.



It is possible to use an <sub>R</sub> suffix on all compositional extensive state variables as well, but the value of the state variable is always the same, with or without the suffix.

If the reference state for a system component is the default reference state (the stable reference state (SER) which is defined in a Thermo-Calc database), then  $MUR(\text{comp}) = MU(\text{comp})$ ,  $ACR(\text{comp}) = AC(\text{comp})$  and  $LNACR(\text{comp}) = LNAC(\text{comp})$ .

In the case of some thermodynamic variables, you can also use the <sub>R</sub> suffix to express chemical potentials and activities of species relative to some single-substitutional-lattice solution phases (such as aqueous solution, gaseous mixture, metallic liquid solution, slag mixture or MO solid solution). These state variables are  $MU(\text{sp,ph})$ ,  $MUR(\text{sp,ph})$ ,  $AC(\text{sp,ph})$ ,  $ACR(\text{sp,ph})$ ,  $LNAC(\text{sp,ph})$  and  $LNACR(\text{sp,ph})$ .

The reference states and standard states of various solution species are pre-defined for some solution phases in some databases. For all solution species in any solution model in any database, it is always the case that  $MUR(sp,ph) = MU(sp,ph)$ ,  $ACR(sp,ph) = AC(sp,ph)$  and  $LNACR(sp,ph) = LNAC(sp,ph)$ .

---

## Entering the Experimental Value and its Uncertainty

The `EXPERIMENT` command is used to specify the quantity that the calculated results should be fitted to. When specifying this quantity, you both specify the measured quantity itself and your estimation of the uncertainty of this value.

In the example, the `EXPERIMENT` command specifies the enthalpy per mole of the system, where this value is specified with respect to non-default reference states that you have set in the equilibrium specification:

```
EXPERIMENT HMR=-1520:200
```

The value before the colon is the quantity of the variable. The value after the colon is an assessment of the uncertainty of this quantity. A higher value means a greater uncertainty. In the example, the uncertainty has been specified with an absolute value but it could also be specified in percent of the quantity:

```
EXPERIMENT HMR=-1520:13%
```

Several experiments can be specified after the experiment command. Besides assigning values to conditions, it is also possible to specify that a certain quantity is greater than or less than a certain value. For instance, the following command would specify that the activity of the C-component is less than 0.01 and the mass fraction of the BCC phase of Ag is greater than 0.05:

```
EXPERIMENT ACR(C) < 0.01:0.001 W(BCC, AG) > 0.05:10%
```

Note that an experimental quantity can typically also be treated as an equilibrium condition, and vice versa. Which measured quantities that you treat as conditions and which quantities you treat as experimental data should be based on the accuracy of the different measurements. In most cases, the quantity that is based on the experimental technique with the lowest accuracy should be used as the experimental value.

## Entering Many Equilibria as a Table

When one thermodynamic variable (e.g. heat capacity) has been measured as a function of another (e.g. temperature), then experimental data is often presented as a table. By using the `TABLE_HEAD`, `TABLE_VALUES` and `TABLE_END` commands, you can enter such experimental information into the POP file in the form of a table. These three commands should always be used together and must be entered in the right order. See [TABLE\\_HEAD, TABLE\\_VALUES and TABLE\\_END](#).

The `TABLE_HEAD` command is followed by a number. This number identifies the table as a whole, but also uniquely identifies the first equilibrium that is included in the table. This numeric identifier is incremented by one for each additional equilibrium in the table. Each row in the table represents one equilibrium. Thus, the equilibrium defined by the first row in the table above (on the line immediately below `TABLE_VALUE`) is uniquely identified with the number 1, the equilibrium defined by the second row by number 2, and so on until the last equilibrium on the sixth row, which is identified by the number 6.

Since the unique identifiers of the equilibria in the table are automatically generated from the unique numerical identifier that you give to the table, the unique numerical identifier that is normally given to the [CREATE\\_NEW\\_EQUILIBRIUM](#) command is replaced by two at signs (@@).

At least one of the equilibria conditions specified with the [SET\\_CONDITION](#) command must be a condition whose value is set by the numbers in one of the table columns. To assign the numbers in a column to a condition, use the syntax `SET_CONDITION <condition>=@<column number>`. For instance, in the example above, the mole fraction of  $X(\text{ALO}_3/2)$  is for each equilibrium given by the number in the first column of the table.

At least one of the measured variables that are entered with the experiment command must be a variable whose value is set by the numbers in a table column. To assign the numbers in a column to a measured variable, use the syntax `EXPERIMENT <variable>=@<column number>`. In the example above, the measured temperature for each equilibrium is given by the number in the second table column.

The actual table should be entered between `TABLE_VALUES` and `TABLE_END`. Separate the columns with tabs or any number of spaces. The columns do not have to contain numbers, but can also contain other types of values, such as phase names for example. If a column contains text with spaces or special characters, then the text must be surrounded by double quotes ("ABC DEF&ghi", for example). The table may have columns that are not used, but each row must have exactly the same number of columns.

The following block of commands exemplifies how a table can be entered into a POP file:

```
TABLE_HEAD 1
CREATE_NEW_EQUILIBRIUM @@ 1
```

```
CHANGE_STATUS PHASE TETR=FIX 0
CHANGE_STATUS PHASE ION=ENT 1
SET_CONDITION P=101325 N=1 ACR(O2,GAS)=.21 X(ALO3/2)=@1
EXPERIMENT T=@2:5
TABLE_VALUE
0.46082476
0.57142380
0.66822276
0.7496 2176
0.50382426
0.63132249
TABLE_END
```

---

## Setting Site Fraction Start Values

In some systems, you have to set non-default composition start values on some experiments in order to get them to converge properly during optimization. This may be important if your system has a miscibility gap for example, in which case several equilibria with different compositions exist at certain temperatures. In such cases, you have to make sure it is specified in the POP file which side of the miscibility gap the equilibrium is on. The calculated equilibrium could otherwise end up on the wrong side, resulting in a large error.

In the POP file, you can set the site fraction of a constituent on an equilibrium with [SET\\_START\\_VALUE](#).

For example, the following command sets the composition start value of VA constituent of the FCC\_A1 phase to 0.99:

```
SET_START_VALUE Y(FCC_A1,VA#2)=0.99
```

Setting the mole fraction for a component in a phase is not as reliable as setting the site fraction.

Additional equilibria entered after the last SET\_START\_VALUE command are automatically given a similar start value if the following command appears previously in the POP file:

```
SET_ALL_START_VALUES Y
```

If start values are not set automatically for phase constituents, then you may have to set the composition start value for each equilibrium (or table of equilibria) separately.

---

## Commenting About an Equilibrium

The COMMENT command inserts a comment about the equilibrium that is saved to the PARROT workspace file when the POP file is compiled. The comment must be entered on one line.

Comments can also be entered immediately after a dollar sign (\$), but these are not saved to the PARROT workspace. Instead, these comments are ignored by Thermo-Calc when the POP file is compiled.

---

## Simultaneous Use of Binary and Ternary Experiments

You can use experimental information about binary and ternary systems (as well as systems of even higher order) in the same POP file and optimization. To do this, you must inform the PARROT module that not all components of the ternary system (or system of even higher order) should be considered. This allows you, for example, to use experimental information about a binary equilibrium that exists within a ternary system.

Use [CHANGE\\_STATUS](#) with the keyword component and which components that should be entered. You should also give the initialization code 0 (rather than 1) to the [CREATE\\_NEW\\_EQUILIBRIUM](#) command. This code indicates that all components must be entered.

For example, the following set of ED\_EXP commands characterises a binary (A-B) three-phase equilibrium (FCC-BCC-LIQ) in a ternary system (A-B-C). It can be used in a POP file that otherwise only contain experimental information about the ternary system (A-B-C):

```
CREATE_NEW_EQUILIBRIUM 1 0
CHANGE_STATUS COMPONENT A B = ENTERED
CHANGE_STATUS PHASE FCC BCC LIQ=FIX 1
SET_CONDITION P=1E5
EXPERIMENT T=1177:10
COMMENT from A-B
```

## Using Stability Conditions

---

Early in the optimization when parameter values are not so good it is often useful to set the driving force for precipitation of a phase (per mole of components) to make sure that it appears where it should. The DGM condition is very useful for making sure that phases appear where they should. It can be removed when the optimization becomes stable.

The DGM condition is also useful for suppressing phases that appear where they should not appear. Do this by setting the value and the uncertainty to something reasonable. For example, the driving force for precipitation of the BCC phase could be set as follows:

```
DGM(BCC) <-0.1:0.1
```

This produces an error even before BCC becomes stable and the weight of the experiment in question can be adjusted as needed. If you specify that the driving force for the BCC phase should be below 0 (DGM(BCC)<0:1E-4 for example), then the optimization converges extremely slowly (if at all). With such a sharp error condition, calculating the equilibrium is like finding the minimum point on a lawn that gently slopes towards a rock wall. The optimization keeps bouncing into the wall.

Another useful stability condition is the phase stability function, abbreviated QF. This can be used to specify that a phase is outside the miscibility gap for a solution phase. If QF(phase) is negative, then the phase is inside the miscibility gap; if QF(phase) is positive, then it is outside the miscibility gap.

---

## Grouping Equilibria with Labels

---

If you have several equilibria describing experiments that you want to be able to treat collectively, you can give all of the equilibria one and the same label.

Use LABEL to give an equilibrium a label. The label must start with the letter A and can only be up to four characters long. For example, you can insert the following command in several equilibria specifications:

```
LABEL ALH
```

During the optimization, you could use this label to, for example, set all the equilibria that have it to the same weight in the ED\_EXP module.

---

## Dividing the POP File into Several Blocks

---

For two different reasons, you may have to divide your POP file into different blocks:

- PARROT uses a buffer for storing compiled experimental data. If there is not enough memory in this buffer for storing all the experiments from the POP file, then the POP file can be divided into several blocks. When PARROT encounters the end of a POP file block, it saves the experimental data that has been compiled to PARROT workspace, clears out the memory buffer and then proceeds to read the next block in the POP file. This goes on until the experimental data from all the blocks have been saved to the PARROT workspace.
- If you want to put equilibria with different sets of components in one and the same POP file, then you must divide the file up into blocks. Each block should only contain equilibria that all have the same set of components.

To divide the equilibria in the POP file into different blocks, enter the [FLUSH\\_BUFFER](#) command between two equilibria. This command marks the beginning of new block. When the command is encountered during the compilation of the POP file, the compilation of the current block is terminated, the equilibria saved to the PARROT workspace file and a new block is initiated. That the buffer is reinitiated means that all functions and constants that were entered in the previous block must be entered again in new block in order to be used.

When editing your experimental data in the ED\_EXP module, you can select which block to load and edit using READ\_WORKSPACES <block number>. The block before the first instance of FLUSH\_BUFFER is block number 1, the block after the first instance and before the second instance is block number 2, and so on.

---

# The Setup File

---

**In this section:**

---

Creating the Setup File .....	39
Defining the System .....	40
Entering the Optimizing Variables .....	41
Initializing Optimizing Variables in PARROT .....	45
Creating the Workspace File .....	46
Compiling the POP File .....	47
Returning Control to the Console .....	48

---

## Creating the Setup File

---

The second step is to create the setup file. This is an ordinary Thermo-Calc macro file (\*.TCM). Typically, your setup file should contain the following:

- A system definition.
- A section where you enter model parameters and optimizing variables.
- A GO PARROT command.
- A section where initial values are assigned to the optimizing variables for the first optimization cycle.
- A CREATE\_NEW\_STORE\_FILE <filename.par> which creates the PARROT workspace file onto which the results of your optimization are continuously saved and updated. By default this workspace file has the same filename as the setup file, but with the \*.PAR filename extension.
- A COMPILE\_EXPERIMENTS <filename.pop> command. Since you often encounter syntax errors when you first try to compile a POP file, it may be useful to execute this command from the console rather than as part of the setup macro file.
- Normally, you also have a [SET\\_INTERACTIVE](#) at the end of the setup file. This command returns control over Thermo-Calc to the console.

It is possible to interactively execute all these commands one at a time in the console. It may be instructive to enter the commands that would normally go into the setup file interactively while using the on-line help while you are learning to use the PARROT module.

## Defining the System

---

If you have an existing database with information about your system, then you can write the system definition part of your setup file in two different ways.

- If you have an unencrypted user database, then you can define your system by entering DATA module commands directly in the setup file. Use the DATA commands that you normally use to define a system when doing equilibrium calculations in Thermo-Calc Console Mode.
- Define your system by entering DATA module commands directly in the console, then generate a macro file with the GIBBS command `LIST_DATA <name of setup file>.tcm p`. This command saves all the data that has been loaded in the GIBBS workspace as a macro file. This file can then be modified as required into an optimization setup file.

## Creating Additional Composition Sets of a Phase

If you need several composition sets of a phase, then these should be created with the GIBBS command `AMEND_PHASE_DESCRIPTION <phase name> composition_sets <composition set number>` in the setup file.



If you add composition sets of a phase after the POP file has been compiled, the data structure in the workspace is corrupted and you have to recompile the POP file.

---

## Entering the Optimizing Variables

---

When you have retrieved the data about your system for the setup file, you must manually enter the model parameters that you want to optimize. This is done using the GIBBS command [ENTER\\_PARAMETER](#), but when you specify the Gibbs energy expression for the parameter, you enter variables that can be given various values into the expression. It is the values of these variables (and thus, the Gibbs energy of the parameter) that you are optimizing.

How you in detail specify the optimizing variables in your set up file depends on which phase you are trying to optimize and what thermodynamic model that is used in the optimization.



[Thermodynamic Models](#)

## Entering Optimizing Variables in Parameter Specifications

There are ninety-nine predefined variables in the PARROT module which you can use in your optimization. These variables are referred to as  $v_1, v_2, v_3, \dots, v_{98}$  and  $v_{99}$ . When the GIBBS workspace is initialized, all optimizing variables are set to a fixed numerical value of zero. When you do an optimization, there is always some variable for which you want to evaluate the best value, but you can assign fixed values to optimizing variables too. It is often useful to enter all the variables that you can conceivably want to optimize, and then you can interactively select and change which variable to focus at various stages of the optimization.

The setup file excerpts that follow show how optimizing variables are entered. However, before the optimizing variables are introduced in this example, the following command enters an ionic liquid solution phase (a phase for which you have some experimental information):

```
ENTER_PHASE IONIC_LIQ Y,
AL+3,ZR+4 ; ALO2-1,O-2 ; N N
```

This `ENTER_PHASE` command enters a phase called IONIC\_LIQ into the GIBBS workspace. This phase is of the type ionic liquid solution, which is specified by type code `Y`. Such a phase is modeled by the *Ionic Two Sublattice Liquid Model*. The next two arguments specify the two sublattices, in this case the cations AL+3,ZR+4 and the anions ALO2-1,O-2. Components that interact in a sublattice are separated by a comma. The first `N` at the end means that no additional constituents are added to the phase. The second `N` means that a list of possible parameters for the phase should not be shown in the console.

There are model parameters that describe the phase which you do not want to optimize, in which case the Gibbs energy expression of the parameter does not contain any optimizing variables.

### Example 1

For example, the following parameters of the IONIC\_LIQ phase do not contain any optimizing variables:

```
ENTER_PARAMETER G(IONIC_LIQ,AL+3:ALO2-1;0) 298.15 +2*GAL2O3_L; 6000 N
ENTER_PARAMETER G(IONIC_LIQ,AL+3:O-2;0) 298.15 +GAL2O3_L+400000; 6000
N
ENTER_PARAMETER G(IONIC_LIQ,ZR+4:O-2;0) 298.15 +2*GZRO2L; 6000 N
```

These parameters define the Gibbs energy expressions for three constituents in the constituent array, namely AL+3:ALO2-1, AL+3:O-2 and ZR+4:O-2. The names that occur in these expressions—that is, GAL2O3\_L and GZRO2L—are the names of complex functions of temperature that are entered in the GIBBS workspace with the [ENTER\\_SYMBOL](#) command.

The numbers that frame the Gibbs energy expressions specify the temperature range within which the expression should be applied. In the above, the expressions are applied in a temperature range from 298.15 K to 6000 K.

## Example 2

In this example, suppose that you want to optimize the model parameter for the ZR+4:ALO2-1 combination, then you could enter the variables V1 and V2 in the Gibbs energy expression for that parameter in the following way:

```
ENTER_PARAMETER G(IONIC_LIQ,ZR+4:ALO2-1;0) 298.15 +GZRO2L +2*GAL2O3_L
+V1+V2*T; 6000 N
```

You normally have different optimizing variables in different parameters, but in some cases you may want several parameters to vary together, in which case you can use the same variable in the Gibbs energy expressions of several parameters.

---

## Entering Interaction Parameters

For some systems, you have to consider interaction parameters in order to take excess Gibbs energy into account and correctly model the phase. By default, the software uses a Redlich-Kister expression for the excess Gibbs energy. The coefficient is entered after the constituent array when you use the [ENTER\\_PARAMETER](#) command. In the last section, all the parameters were entered with a coefficient of zero (0).

To take account of the excess Gibbs energy, you might enter the following:

```
ENTER_PARAMETER L(IONIC_LIQ,ZR+4:ALO2-1,O-2;0) 298.15 +V11+V12*T;  
6000 N  
  
ENTER_PARAMETER L(IONIC_LIQ,ZR+4:ALO2-1,O-2;1) 298.15 +V13+V14*T;  
6000 N  
  
ENTER_PARAMETER L(IONIC_LIQ,ZR+4:ALO2-1,O-2;2) 298.15 +V15+V16*T;  
6000 N
```

Parameters with interaction coefficients are referred to as *interaction parameters*. The regular parameter has a coefficient of 0. The subregular parameter has a coefficient of 1. It is possible to interaction coefficients up to a value of 9, but you are advised not to use more than three coefficients (that is, with coefficient values of 0, 1 and 2).

## Initializing Optimizing Variables in PARROT

---

Variables that have been entered into Gibbs energy expressions for model parameters in the setup file have to be initialized and given start values before they are optimized.

You have to estimate what start values to give to the optimizing variables based on your knowledge of the system or of relevantly similar systems. There are no general rules for what start values your optimizing variables should have. If you are not able to make any informed estimation about what start values to use, then you may not be able to find a good fit between your experimental data and your calculated results. In this case, it is often best to start the optimization in the so-called alternate mode. In alternate mode, the PARROT module helps you find some reasonable start values for your optimizing variables.



### About Alternate Mode

Even if you are starting your optimization in alternate mode, you still have to initialize the optimizing variables though (you can all give them a start value of 0).

To initialize an optimizing variable and to give it a start value, use either [SET\\_OPTIMIZING\\_VARIABLE](#) or [SET\\_SCALED\\_VARIABLE](#). Using SET-[OPTIMIZING\\_VARIABLE](#) results in a scaling factor equal to the current value of the parameter unless the current value is zero, in which case the scaling factor is 1000. This is typically appropriate for enthalpic parameters, but not necessarily for other parameters. The scaling factor is the factor by which the optimizing variable value is varied during the optimization.

For example, if you want to optimize the variables V1, V2, V11, V13 and V17, then you could enter the following in the setup file:

```
SET_OPTIMIZING_VARIABLE 1 +43000
SET_OPTIMIZING_VARIABLE 2 -35.2
SET_OPTIMIZING_VARIABLE 11 +72000
SET_OPTIMIZING_VARIABLE 13 +27000
SET_OPTIMIZING_VARIABLE 17 -100000
```

The value given to each optimizing variable (+43000 for variable V1 for example) is also that variable's initial scaling factor. If you want to set the initial scaling factor of a variable to a value that is lower or higher than its start value, then use SET-[SCALED\\_VARIABLE](#) <start value><scaling factor><min value><max value>.

## Creating the Workspace File

---

In the setup file, after the system definition and the entered optimizing variables, use the PARROT command [CREATE\\_NEW\\_STORE\\_FILE](#) to create and save a PARROT workspace file for the system defined in the setup file. For example, the following command saves all current data on system definitions (elements, species, components, phases), symbol definitions (constants, variables, functions, tables) and parameters that are stored in the workspace used by GIBBS, POLY and PARROT into a file called AL2O3-ZRO2.PAR:

```
CREATE_NEW_STORE_FILE AL2O3-ZRO2
```



The experimental information in the POP file is not saved to this PARROT workspace file.

---

## Compiling the POP File

---

At the end of the setup file, use the command `COMPILE_EXPERIMENTS` followed by the name of your POP file to compile the data in that file and save it to the current PARROT workspace file. Note that you must first have opened or created a new PARROT workspace file before you compile your POP file. The other argument to the command can typically be given default values with comma signs (,):

```
COMPILE_EXPERIMENTS AL2O3-ZRO2.POP,,,
```

If Thermo-Calc encounters errors in the syntax of the POP file, then error messages are displayed in the console. Normally, the compilation also stops when syntax errors are encountered. If this happens, then you must correct the errors and recompile the file. It is useful to have two console windows open when you do this. That way, you can use one console for editing the experimental data and another console for compiling and reading any further error messages.

## Returning Control to the Console

---

Put the command [SET\\_INTERACTIVE](#) at the end of the setup file if you want to return control to the console when the commands in the setup file have all been executed.

---

# Optimizing in PARROT

---

**In this section:**

---

Optimizing the System in PARROT .....	50
Changes that Require POP File Recompilation .....	51
Optimization Workflow .....	52
Ensuring Computation of all Equilibria .....	53
Setting Weights .....	55
Optimizing and Evaluating Results .....	56
Continuing the Optimization and Resetting Variables .....	58
Updating the Set of Optimizing Variables .....	60
Reducing the Number of Optimizing Variables .....	62
Plotting Intermediate Results .....	63

---

## Optimizing the System in PARROT

---

When you have created a POP file, a setup macro file, and (optionally) EXP files, you should start doing optimization runs interactively in the console. This process can be divided into a number of steps (such as running optimization, evaluating optimization results, modifying weights and models and adding new information), but usually, you have to go through many optimization cycles before achieving a satisfactory fit between calculation results and experimental data.

Typically, it is not obvious when an optimization is finished. You must exercise your judgment about when the fit you have achieved is good enough.



It is possible to interactively change almost everything in the initial setup macro file and in the POP file. For example, you can add more parameters to be optimized and modify or add more experimental information.

---

## Changes that Require POP File Recompilation

---

Some changes that you do interactively in the PARROT module or the ED\_EXP sub-module destroys the data structure in the workspace. For example, if you add more composition sets to a phase, then the number of phases actually change and the links break between the experimental data and the thermodynamic models. Note that if the global minimization technique is used in POLY, then new composition sets that corrupts the PARROT workspace may be created automatically. If this happens, then you have to compile the POP file again.

If you recompile the POP file, then the experimental equilibria are calculated with the default start values.

## Optimization Workflow

---

Normally, the workflow during an optimization is the following:

1. In the PARROT module, run the setup macro file using [MACRO\\_FILE\\_OPEN](#).
2. Go to ED\_EXP with [EDIT\\_EXPERIMENTS](#), use [READ\\_WORKSPACES](#) to load your experimental data and then execute .
3. If an equilibrium cannot be computed, then this either means that the start values (site fractions, for example) were inappropriate or that the equilibrium cannot be calculated with the current set of parameters. If the latter, then the equilibrium may have to be excluded from the optimization. However, you should make an effort to exclude the possibility that the reason is just that you do not have the appropriate start values for the equilibrium.
4. When each equilibrium is either successfully computed or excluded from the calculation, go back to the PARROT module and run the [OPTIMIZE\\_VARIABLES](#) command. Give the number of optimization iterations you want the Thermo-Calc program to carry out as an argument to the command. This number specifies how many different sets of values for the optimizing variables that PARROT tries to fit with the experimental data. It is recommended that you run 0 iterations first to see the discrepancy between the experimental data and the initial calculated results. The current workspace file is automatically updated with the last set of optimized variables and calculated results.
5. List and evaluate the result of the optimization using [LIST\\_RESULT](#). Typically, what is of interest in the results are the relative standard deviation of the optimizing variables and the error values in the list of calculated experimental data results. If the relative standard deviation for an optimizing variable is very large, this means that the variable doesn't have any effect on the computed results and that the variable should not be used (you can make PARROT ignore the variable by setting its value to 0 with [SET\\_FIX\\_VARIABLE](#)). If the relative standard deviation is close to zero for a variable, then this might indicate that you need to use more variables. When it comes to errors in the calculated equilibrium results, these can be decreased by lowering the weight that PARROT gives the experiment in the calculation (the more weight, the more the experiment affects the calculated results).
6. Use [OPTIMIZE\\_VARIABLES](#) or [CONTINUE\\_OPTIMIZATION](#) to carry out a new optimization cycle. Go back to step 5 above and continue to do optimization cycles until you judge that there is a good enough fit between your calculated results and your experimental data. It is recommended that you do not only rely on feedback from [LIST\\_RESULT](#) but also frequently plot various diagrams and compare them visually to some of your experimental data.

## Ensuring Computation of all Equilibria

---

When you have run the setup macro file, you should start by trying to compute all the equilibria that were created when your POP file was compiled. To do this, you must first go to the ED\_EXP submodule, and load the experimental data from the current PARROT workspace file with the READ\_WORKSPACES <block #> command (normally, the block number is 1).

When you have loaded all equilibria, you should try to compute them all using the ED\_EXP command . A list of calculated equilibrium results are shown in the console, but the software is not able to reach an equilibrium for an experiment, then the program cancels the calculations and displays an error message.

The following list shows an example output where the first two experiments in the POP file have been successfully computed, but an equilibrium is not reached for the third experiment. No other equilibria are computed, since the calculations are aborted when the error occurs.

```
Eq  Lab  Iter  Weight  Temp   Exp  Fix phases or comments
  1  A1   22    1.     2575.3  IONIC_ FLUORI TETR
  2  A1   18    1.     2156.9  IONIC_ CORUND TETR
*** ERROR  1614 IN QTHISS
*** CONDITIONS CAN NOT BE FULLFILLED
```

When this happens, try to find out why the equilibrium calculation fail. Are the condition start values that you have given the equilibrium inappropriate for example? If you have a similar equilibrium that has already been successfully computed, then you can select the equilibrium that does not work, use the ED\_EXP command [SET\\_ALL\\_START\\_VALUES](#) and accept all the suggestions for the site fractions. Alternatively, use TRANSFER\_START\_VALUE <equilibrium number> to make the currently selected equilibrium inherit the start values of the indicated equilibrium. Run [COMPUTE\\_EQILIBRIUM](#) again to see if the equilibrium can be computed with the inherited start value.

If you cannot find an explanation of why the equilibrium cannot be computed, then it may be that the experiment is faulty. If so, then this experiment should be excluded from the optimization by being given a weight of 0. You should then run COMPUTE\_EQILIBRIUM again to make sure that all equilibria can be computed. However, experiments that fail to reach equilibria could succeed in doing so if the optimizing variables have other values. It is therefore sometimes worth trying to include experiments later in the optimization process (when the optimizing variables are relatively close to their final values), even if they had to be excluded at its start. In some cases, it might also be appropriate to change the error tolerance for in an experiment (using the ED\_EXP command [EXPERIMENT](#)).



You must save the PARROT workspace with [SAVE\\_WORKSPACES](#) before you exit the ED\_EXP module.

---

## Setting Weights

---

To change the weight of an experiment, first enter the ED\_EXP module from PARROT using [EDIT\\_EXPERIMENTS](#). Then use the [SET\\_WEIGHT](#) command to set the weight for a specific equilibrium (equilibrium number 3 for example), for a range of equilibria (the equilibria 10–19) or all the equilibria that have a certain label (such as all the equilibria with the label A1). To save the new weight(s), use [SAVE\\_WORKSPACES](#).

By default, all the experimental information in your POP file is treated equally: each experiment has a weight of 1. However, experiments that you excluded because they did not reach equilibrium now have a weight of 0. The higher the weight of an experiment is, the more Thermo-Calc tries to ensure that the computed result fits the condition measured in that experiment.

The contribution of an experiment's error to the sum of errors is multiplied by the square of the weight set for that experiment. In other words, if you want to cut the error of an experiment with weight 1.0 by half, then you should use a weight of 0.7. This makes the experiment's error 0.49. Setting the weight to 1.4 instead makes the error twice as large (1.96).

Note that if you use [READ\\_WORKSPACES](#) after you have changed weights on the experiments, your changes are lost unless you have saved those changes. In the ED\_EXP module, you can get a list of all the experiments in the console if you first read the experimental data from the workspace file (with [READ\\_WORKSPACES](#)) and then use [LIST\\_ALL\\_EQUILIBRIA](#).

## Optimizing and Evaluating Results

When you have ensured that all the experimental data points can reach equilibrium, go back to the PARROT module and use `OPTIMIZE VARIABLES <number of iterations> with 0` iterations. Present the result of this initial optimization by using `LIST_RESULT`.

When evaluating the optimization results, the most important information is found in the list of experiments at the end of the listed results. The rightmost column shows the difference between the value of the experimental data and the value computed by Thermo-Calc in the last optimization cycle.

The following shows part of list of experiments in the output of a `LIST_RESULT` command:

106	ACR(B)=0.34	0.3127	2.89E-02	-2.7282E-02	-0.9444	
107	ACR(B)=0.23	0.2085	2.89E-02	-2.1522E-02	-0.7450	
108	ACR(B)=0.12	0.1042	2.89E-02	-1.5761E-02	-0.5455	
110	HMR(LIQUID)=-1964	3.6380E-12	5.00E+02	1964.	3.928	
111	HMR(LIQUID)=-3500	0.000	5.00E+02	3500.	7.000	*
112	HMR(LIQUID)=-4588	0.000	5.00E+02	4588.	9.176	*
113	HMR(LIQUID)=-5239	7.2760E-12	5.00E+02	5239.	10.48	*
114	HMR(LIQUID)=-5454	-3.6380E-12	5.00E+02	5454.	10.91	*
115	HMR(LIQUID)=-5233	1.8190E-12	5.00E+02	5233.	10.47	*
116	HMR(LIQUID)=-4575	-1.8190E-12	5.00E+02	4575.	9.150	*
117	HMR(LIQUID)=-3481	-1.8190E-12	5.00E+02	3481.	6.962	*
118	HMR(LIQUID)=-1950	0.000	5.00E+02	1950.	3.900	

### PARROT:

The experiments for which the optimization has not found a good fit are marked with an asterisk (\*) or a hash sign (#) in the rightmost column. An asterisk in this column indicates that the experiment has an error that is larger than the error tolerance and a hash sign indicates that the error is much larger. The error tolerance depends on the degree of accuracy set for that particular piece of experimental information. The error values are presented in the next to rightmost column.

Note that if you are optimizing in *alternate mode*, then you usually do not need to change the weights for the experiments. Instead, you can simply run `OPTIMIZE_VARIABLES` again, specifying, say, thirty iterations, evaluate the results again, and continue in that way until you are satisfied with the results.



[About Alternate Mode](#)

## **The Critical Set of Experiments**

The set of weighted experiments that you end up with as a result of your optimization is referred to as the *critical set* of experiments. When you work to determine this set and the weights of the experiments, the following factors should be reflected in both the selection of experiments and the weights of those experiments:

- The reliability of the experimental technique.
- The extent of agreement/disagreement between independent measurements of the same quantity.
- The extent of agreement/disagreement between data obtained with different experimental methods.

## Continuing the Optimization and Resetting Variables

To continue the optimization after you have used [OPTIMIZE\\_VARIABLES](#), you can either use this command and enter <Number of iterations> again or [CONTINUE\\_OPTIMIZATION](#) <Number of iterations>.

If you use [CONTINUE\\_OPTIMIZATION](#), then PARROT continues the optimization using the same Hessian matrix. If the optimization is going well but you run out of iterations, then continuing with the same Hessian matrix may be useful.

As before, use [LIST\\_RESULT](#) to inspect the fit between the calculated and the experimental values for your experiments. If necessary, adjust the weights of the experiments again in the ED\_EXP module. You may even discover that you have to exclude additional experiments from the optimization.

Besides looking at the rightmost column in the list of experiments (that is produced when you use [LIST\\_RESULT](#)), it is also useful to look at the *sum of squares* of the errors of all the variables. This is shown in the output of [OPTIMIZE\\_VARIABLES](#) and [CONTINUE\\_OPTIMIZATION](#). This sum should be as low as possible.

The following shows part of the output that you get when using [LIST\\_RESULT](#):

```

== OPTIMIZING VARIABLES ==

AVAILABLE VARIABLES ARE V1 TO V00

VAR.   VALUE           START VALUE      SCALING FACTOR   REL.STAND.DEV
V1     6.94641846E+05    4.30000000E+04   4.30000000E+04   1.42299558E+00
V2     -3.32619842E+02   -3.52000000E+01  -3.52000000E+01  7.91369573E-01
V11    8.37422071E+04    7.20000000E+04   7.20000000E+04   2.02212692E-01
V13    2.65253341E+04    2.70000000E+04   2.70000000E+04   1.45201828E-01
V17    -3.66637428E+05   -1.00000000E+05  -1.00000000E+05  8.04279315E-01

NUMBER OF OPTIMIZING VARIABLES :    5
ALL OTHER VARIABLES ARE FIX WITH THE VALUE ZERO
THE SUM OF SQUARES HAS CHANGED FROM  3.50196879E+05 TO  3.49096720E+05
DEGREES OF FREEDOM  14.  REDUCED SUM OF SQUARES  2.49354800E+04

```

In the example you can see that the sum of squares have decreased slightly. (The final sum of squares that you get after an optimization run is also presented directly in the output from [OPTIMIZE\\_VARIABLES](#) and [CONTINUE\\_OPTIMIZATION](#).) As long as the sum of squares figure is decreasing, the optimization is working as it should.

When your calculated results have been improving for a while, that is, when the sum of squares has been decreasing, and you are confident that you are making definitive progress, then you should use the PARROT command [RESCALE\\_VARIABLES](#) to reset the starting values of the optimizing variables to their current values. When you do this, the start values and

---

scaling factors of the optimizing variables (shown in the table's third column in the preceding screen shot), are all set to the current values (shown in the table's second column).

If your optimization is not going well, then you may want to reset the values of your optimization variables to their current start values. To do this, use [RECOVER\\_VARIABLES](#).

After rescaling the variables, continue the optimization using OPTIMIZE\_VARIABLES again. Cycle through calls of the commands OPTIMIZE\_VARIABLES/CONTINUE\_OPTIMIZATION, CONTINUE\_OPTIMIZATION, RESCALE\_VARIABLES and OPTIMIZE\_VARIABLES again until your set of optimizing variables does not seem to improve anymore. Even when it seems to you that you have reached this point, it is recommended that you use OPTIMIZE\_VARIABLES/CONTINUE\_OPTIMIZATION a few extra times to make sure that the variable values cannot be further improved, that is, that the sum of squares of the errors of all the variables cannot be reduced further. If the final solution is repeatedly calculated after the same number of iterations and reaching the same result, then it is generally reasonable to accept the current variable values.

## Updating the Set of Optimizing Variables

Besides evaluating the fit between the calculated results and the experimental data, you should also ensure that you have the right number of optimizing variables. If you have too many or too few optimizing variables, then you may not be able to use your model to accurately extrapolate thermodynamic properties in a wide range of temperature, pressure and composition conditions.

With more optimization parameters, the sum of errors usually decrease, but the parameters also tend to become less precisely determined. A measure of this precision is the Relative Standard Deviation (RSD) of each optimizing variable. However, the RSD is only significant if you have used [RESCALE VARIABLES](#) followed by an optimization which converges and the values of the variables do not change much. The RSD tells you how much the parameter can be changed in either a positive or in a negative direction without changing the reduced sum of errors with more than one unit.

A large RSD value thus means that the parameter has not been determined well.

```
== OPTIMIZING VARIABLES ==
```

```
AVAILABLE VARIABLES ARE V1 TO V00
```

VAR.	VALUE	START VALUE	SCALING FACTOR	REL. STAND. DEV
V1	2.01820948E+04	2.01874462E+04	2.01874462E+04	2.60519742E-02
V2	-2.90936164E+01	-2.90969472E+01	-2.90969472E+01	1.41576516E-02
V11	-2.18127453E+04			
V12	1.55559524E+01			
V15	2.36701148E+04	2.37258719E+04	2.37258719E+04	1.00528931E-01
V16	-7.56541897E+00	-7.64417807E+00	-7.64417807E+00	3.17218507E-01
V17	3.00342429E+03	3.02274918E+03	3.02274918E+03	2.60675550E-01
V19	2.20133190E+04	2.14534347E+04	2.14534347E+04	6.92923849E-01
V20	-6.72498276E+00	-6.31334932E+00	-6.31334932E+00	1.79242248E+00

```
NUMBER OF OPTIMIZING VARIABLES : 7
```

```
ALL OTHER VARIABLES ARE FIX WITH THE VALUE ZERO
```

```
THE SUM OF SQUARES HAS CHANGED FROM 1.51360340E-01 TO 1.44420287E-01
```

```
DEGREES OF FREEDOM 18. REDUCED SUM OF SQUARES 8.02334928E-03
```

If such a significant RSD has a value larger than 1, then this typically means that you are using too many optimizing variables. However, the RSD values also depend on the weighting of the experiments, so it is sometimes possible to reduce the RSD by changing weights.

The RSD should not be too low either. If one or more of them is very close to 0, then this suggests that you are trying to optimize the system with too few optimizing variables. You can add optimizing variables interactively using [ENTER PARAMETER](#) in GIBBS. However, it is

---

recommended that you add a sufficient number of variables in the setup file from the start so that do not need to do this. You can fix variables to a value of 0 until you discover that they are needed.

Besides the RSD, you should also keep an eye on the variable values themselves (shown in the VALUE column, next to the leftmost column in the table shown in the preceding screenshot). If the value of a variable is in the order of  $1E5$  or higher and this variable has been put in the temperature-independent part of an interaction parameter's Gibbs energy expression, then you may not have the right weights on your experiments or you are using too many optimizing variables. This is also typically the case if a variable with a value in the order of 10 or more (per mole atoms) is in the temperature-dependent part of the Gibbs energy expression of an interaction parameter. Such a high parameter value may lead to inverted miscibility gaps or re-stabilisation of a phase at high temperature. It is important to keep an eye on this since the temperature-independent and the temperature-dependent interaction parameter often varies together. If the temperature-dependent interaction parameter cannot be reliably determined, you may have to fix its value.

Note that even if you have found a good enough fit between calculated and experimental values after an optimization run, a parameter value may suddenly start to change by several orders of magnitude when you use [OPTIMIZE\\_VARIABLES](#) is used again. If this happens, then careful reconsideration of the weighting of the experiments of the set of optimizing variables is required. However, when the optimization has converged and the parameter values remain stable after repeated use of `OPTIMIZE_VARIABLES`, this should usually not happen.

---

## Reducing the Number of Optimizing Variables

---

If you have one or more optimizing variables with RSD larger than 1, then you should remove one or more of the variables by setting them to a value of 0, or alternately, set them to a reliably estimated value (estimated based on, for example, semi-empirical methods).

To fix an optimizing variable to certain value, use the PARROT command SET\_FIX\_VARIABLE <variable number> <value>.

---

## Plotting Intermediate Results

---

It is typically not sufficient to look at the plain text output of [LIST\\_RESULT](#) to reliably judge how the optimization is progressing. Plotting a phase or property diagram of the system you are optimizing using the POLY and POST modules is often very useful.

It is recommended that you create a macro file for plotting the diagram since you probably want to repeatedly plot the diagram as you progress with the optimization. This macro file can then be called directly from the PARROT module with the [MACRO\\_FILE\\_OPEN](#) command.

---

## Superimposing Experimental Data from EXP Files

If you create one or several EXP files based on the experiments in your POP file, you can visually compare the fit your plot and the experimental data you are optimizing the system against. An EXP-file contains data points specified in the DATAPLOT format. You could make an EXP-file for each type of data that you are using in the optimization.

Use the POST command [APPEND\\_EXPERIMENTAL\\_DATA](#) to superimpose the data points in an EXP-file on a phase or property diagram that you have plotted. For example, to put the experimental data points from a file entitled DP.EXP onto a plotted diagram, you could enter the following:

```
APPEND_EXPERIMENTAL_DATA Y DP.EXP 0 1
```

Such a command call tells Thermo-Calc to superimpose the data points in dataset 1 of DP.EXP onto the plot in the Console Results window, but not to impose any information about axis scaling, labels and other information which is normally found in the prologue of the DP.EXP file.

The POST command [QUICK\\_EXPERIMENTAL\\_PLOT](#) can also be used to superimpose data points on a plotted phase or property diagram.



*DATPLOT User Guide* included with this documentation set.

---

## Finalizing the Optimization

---

**In this section:**

---

Finishing the Optimization .....	66
Rounding off Optimizing Variable Values .....	67
Updating the Database File .....	68
Creating a Database File .....	69
Updating the Setup File and the POP file .....	70

## Finishing the Optimization

---

When to stop optimizing is a matter of judgment. It is rarely the case that you finish optimizing a system with the feeling that it cannot be improved.

The final step is to update your files with the final results. Besides updating your database file (or alternatively, creating a new database file), it is advised that you also update your setup and POP files with the final weights and optimizing variable values.

---

## Rounding off Optimizing Variable Values

---

When you save the optimizing variable values, it is important that you round off the values correctly. When you have a metallic system, it is usually best to keep whatever number of decimal digits that you need to make less than one J/mol difference at 1000 K. When you have an aqueous system or a system that you are optimizing based on data from very different temperature ranges, then this is not always the only appropriate guideline. However, rounding off the values in such a way that it gives a difference that is larger than one J/mol may lead to differences in the phase diagram.

Another approach you can take when rounding off variable values is by progressively fixing the value of more and more of the variables. This allows you to round-off all but one of the variables values so that each only has a few significant digits. Variables whose values have been rounded off in this way are easier to handle compared with those that are rounded off so that they continue to have many significant (non-zero) digits.

To round off the variable values according to this second approach, first use [SET\\_FIX\\_VARIABLE](#) to set the variable(s) with the highest RSD to a rounded off value (such as, say, 0.4). When you then re-optimize using [OPTIMIZE\\_VARIABLES](#), the sum of errors changes. However, you should get almost exactly the same sum of errors as before after you rescale the variable values using [RESCALE\\_VARIABLES](#). (If the sum of errors is different, then this means that the variable with the highest RSD was not rounded off in a good way.) If you successfully round off the first variable value, then continue to round of the optimizing variable which now has the highest RSD. Continue doing this until you have fixed all but one of the optimizing variables to their rounded-off values. The final sum of errors that you get after having fixed the variable values should not deviate significantly from the initial sum of errors you had before starting to round-off the variable values.

## Updating the Database File

---



### About the Database Files

To update the database file (with filename extension TDB) open it in a text editor and enter or update each of the parameters that you have optimized.

For example, suppose you have the following command in your setup file:

```
ENTER_PARAMETER L(IONIC_LIQ,AL+3,ZR+4:O-2;0) 298.15 +V17; 6000 N
```

If you then end up with a value for optimizing variable V17 of -100000, then you should enter the following under the IONIC\_LIQ phase in the database file:

```
PARAM L(IONIC_LIQ,AL+3,ZR+4:O-2;0) 298.15 -100000; 6000 N REF !
```



If you often need to round-off the optimizing variable value in some way, see [Rounding off Optimizing Variable Values](#).

When you have updated or entered the parameters that you optimized, save the database file.

---

## Creating a Database File

---

In some circumstances you may want to save all the information about your system into a new database file. To do this, go to the GIBBS module and use `LIST_DATA <Filename> N`. This creates a new user database file that contains all the information about the system that is in the workspace.

The N-argument tells the [LIST\\_DATA](#) command to save the output into database file rather than some other type of file, such as a macro file for example. Note that the output database file that you get from `LIST_DATA` have to undergo some manual editing before it can be used as proper database file.

---

## Updating the Setup File and the POP file

---

It may be useful for you to be able to recreate your optimization from scratch. It is therefore recommended that you do the following when you are done with the optimization:

- Update your POP file so that it contains the weights of your critical set of experiments. Use [SET\\_WEIGHT](#) to set the weights directly in the POP file. You may also have to use [SET\\_START\\_VALUE](#) for some conditions in the POP file. Otherwise, you may not be able to compute the equilibria with .
- Copy the final values of your optimizing variables and set them as start values of the variables in the setup macro file (with [SET\\_FIX\\_VARIABLE](#)).



It is important that you round off the values of the optimizing variables correctly. See [Rounding off Optimizing Variable Values](#). When you have done this, you can run the setup macro file to recreate the final result of the optimization.

---

# Alternate Mode

---

**In this section:**

---

About Alternate Mode .....	72
Optimizing in Alternate Mode .....	73
Preparing the POP File for Alternate Mode .....	74
Examples of SET_ALTERNATE_CONDITION .....	75

## About Alternate Mode

---

To turn Alternate Mode ON or OFF, use the PARROT command [SET\\_ALTERNATE\\_MODE](#).

When you have one or more multi-phase equilibria (that is, with the fixed phase-status) in your POP file and the optimizing values lack appropriate starting values (such as the default value of 0 for example), then Thermo-Calc may not be able to compute the equilibria for the experiments. In these cases, a common tangent cannot be found, that is, a state where the chemical potentials for all components are the same in all phases cannot be found. A common tangent may not exist or it be found at the wrong composition or temperature given the initial set of model parameters.

If you are not able to estimate reasonable starting values for the optimizing variables, then this problem can be overcome by initially running your optimization in alternate mode. The alternate mode in PARROT module should help you find reasonable starting values for the optimizing variables. Note that you are only supposed to use alternate mode in the beginning of the optimization. Once you have achieved a reasonable fit between calculated results and experimental data, turn off alternate mode and calculate all equilibria again in the normal mode.

In alternate mode, PARROT modifies multi-phase equilibria so that the chemical potentials for each stable phases is calculated separately. For example, consider the following experimental equilibrium:

```
CREATE_NEW_EQUILIBRIUM 1 1
CHANGE_STATUS PHASE LIQ FCC=FIX 1
SET_CONDITION T=1000 P=1E5
EXPERIMENT X(LIQ,B)=.2:.01 X(FCC,B)=.1
```

When PARROT optimizes in alternate mode, the computation of this equilibrium is done by way of the following two calculations:

```
CHANGE_STATUS PHASE LIQ=FIX 1
SET_CONDITION T=1000 P=1E5 X(LIQ,B)=.2
CHANGE_STATUS PHASE FCC=FIX 1
SET_CONDITION T=1000 P=1E5 X(FCC,B)=.1
```

The difference between the chemical potentials in every pair of phases is calculated and PARROT tries to adjust the parameters of the phases so that these differences are minimized. In other words, PARROT tries to make the chemical potentials equal in all phases. For PARROT to be able to do this, some additional information about the experimental multi-phase equilibria is required, at least for all invariant equilibria.

---

## Optimizing in Alternate Mode

---

When you are optimizing in the alternate mode, you usually do not need to enter the ED\_EXP module and manually change the weights of experiments that the computed results have not been fitted well with. Instead, after using [OPTIMIZE\\_VARIABLES](#) initially with no iterations, and listing the result with [LIST\\_RESULT](#), simply run OPTIMIZE\_VARIABLES again specifying, say, thirty iterations. Continue to optimize until your optimization seems to be going in the right direction, then set the optimizing variables' starting values to their current values use [RESCALE\\_VARIABLES](#) before you turn alternate mode off by answering N at the [SET\\_ALTERNATE\\_MODE](#) prompt.

---

## Preparing the POP File for Alternate Mode

---

Information in your POP file that concerns equilibria with ENTERED or DORMANT phases is ignored when you optimize in alternate mode.

To use the alternate mode, ensure that there is enough information about each phase in a given equilibrium. In addition to the temperature and pressure of the system, you should specify the composition of all phases in a given equilibrium. This is done with, [SET\\_CONDITION](#), [EXPERIMENT](#), or [SET\\_ALTERNATE\\_CONDITION](#).

If both compositions of a binary tie-line have been measured, then you could specify this in the POP file in the following way:

```
CREATE_NEW_EQUILIBRIUM 1 1
CHANGE_STATUS PHASE FCC BCC=FIX 1
SET_CONDITION P=1E5 T=1000
EXPERIMENT X(BCC,B)=.2:.01 X(FCC,B)=.3:.01
```

This equilibrium could be calculated with alternate mode without any modification. The compositions specified with the experiment command is used as condition when the thermodynamic properties of each phase are calculated.

## Examples of SET\_ALTERNATE\_CONDITION

---

The following topics demonstrate how [SET\\_ALTERNATE\\_CONDITION](#) is used to prepare different kinds of equilibria for optimization in alternate mode.

### ***How to Specify the Composition of One Side of a Binary Tie-Line***

If only one side of the tie-line has been measured, then you must provide an estimate of the composition of the other phase. You can make such an estimate with SET\_ALTERNATE\_CONDITION. This command has no effect unless alternate mode is turned on. The equilibrium described could thus be specified in the following way:

```
CREATE_NEW_EQUILIBRIUM 1 1
CHANGE_STATUS PHASE FCC BCC=FIX 1
SET_CONDITION P=1E5 T=1000
EXPERIMENT X(BCC,B)=.2:.01
SET_ALTERNATE_CONDITION X(FCC,B)=.3
```

When the alternate mode is turned on, the composition given by the SET\_ALTERNATE\_CONDITION command is used for the FCC phase. The properties of the BCC phase is calculated using the composition specified by experiment as a condition.

### ***How to Specify the Compositions of a Three-Phase Equilibrium***

If you have a three-phase equilibrium in your POP file, then one of its compositions can be specified as experimentally determined, while two can be provided as alternate conditions:

```
CREATE_NEW_EQUILIBRIUM 1 1
CHANGE_STATUS PHASE FCC BCC LIQ=FIX 1
SET_CONDITION P=1E5
EXPERIMENT T=912:5 X(LIQ,B)=0.2:.02
SET_ALTERNATE_CONDITION X(FCC,B)=0.1 X(BCC,B)=.4
```

### ***How to Specify the Composition of a Stoichiometric Phase***

When you specify the composition of stoichiometric phase with SET\_ALTERNATE\_CONDITION, the value must be given with at least seven decimal digits, as in the following example:

```
CREATE_NEW_EQUILIBRIUM 1 1
```

```

CHANGE_STATUS PHASE LIQ A2B=FIX 1
SET_CONDITION P=1E5 X(LIQ,B)=0.2
EXPERIMENT T=992:5
SET_ALTERNATE_CONDITION X(A2B,B)=.6666667

```

## Entering Other Experiments in the POP with Alternate Mode

The following table shows some additional examples of how to enter different kinds of experiments in the POP file when you are using the alternate mode.

Example	ED_EXP commands
Two-phase equilibrium: The melting temperature of an Au-Cu alloy	<pre> CHANGE_STATUS PHASE LIQUID FCC=FIX 1 SET_CONDITION X(FCC,CU)=0.14 P=1E5 EXPERIMENT T=970:2 SET_ALTERNATE_CONDITION X(LIQUID,CU)=0.16 </pre>
Invariant equilibrium experiment: A three-phase equilibrium in a binary system	<pre> CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC BCC LIQUID=FIX 1 SET_CONDITION P=1E5 EXPERIMENT T=912:5 SET_ALTERNATE_CONDITION X(FCC,B)=0.1SET_ALTERNATE_CONDITION X(BCC,B)=0.4SET_ALTERNATE_CONDITION X(LIQ,B)=0.2 </pre>
Ternary system, with two compositions measured (both with uncertainty 0.02)	<pre> CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC BCC=FIX 1 SET_CONDITION T=1273 P=1E5 SET_CONDITION X(FCC,B)=0.1:0.02 EXPERIMENT X(FCC,C)=0.12:.02 SET_ALTERNATE_CONDITION X(BCC,B)=0.17 SET_ALTERNATE_CONDITION X(BCC,C)=0.07 </pre>

---

# Troubleshooting

---

During an optimization, you are likely to run into many different problems and challenges. This topic gives some general guidelines that can help avoid common mistakes and problems.

**In this section:**

---

Useful Guidelines .....	78
Excluding and Including the Correct Equilibria .....	80
Conflicting Data .....	81

## Useful Guidelines

---

The following rules of thumb are generally good to adhere to when you are planning and executing an optimization:

- Do not use data from thermochemical tables of unknown origin.
- Use experimentally determined properties; avoid converted quantities.
- Carefully estimate the accuracy of the experiments.
- Correct systematic errors (by adjusting a temperature scale for example).
- Use negative information such as, for example, the information that a phase should not be stable in a composition or temperature region.
- With a hundred activity measurements in a system but only ten composition points from the phase diagram, it is often appropriate to decrease the weights on the activity experiments.
- If you initially exclude any intermediate phases to only optimize the liquid and the terminal or end-member phases for the pure components, then it is often useful to compute a metastable phase diagram with just these phases. In the metastable phase diagram, the metastable solubility lines should not have any strange kinks or turns. Such a diagram is often also useful to compute at later stages in the optimization.
- When you have fitted the liquid phase and some solution phases reasonably well, then you can fix the optimizing variables that characterise these phases. You can then go on and optimize the variables for the intermediate phases.
- Phases with miscibility gaps are always difficult to optimize. Try to keep control of the miscibility gaps by using real or estimated experimental information. (Note that it is not possible to calculate the top of a miscibility gap as a single equilibrium.)
- Phases with order/disorder transformation are often difficult to optimize. It may be difficult to find the right start values on the optimizing variables. The TABULATION module can be useful sometimes. Properties for specified site fractions can be calculated in the TABULATION module. This is not possible in the POLY module since this module only calculates for equilibrium site fractions. The ordered state may also disappear during the optimization of these phases. If this happens, you may be able to add an experiment that controls the state of order.
- Only use the alternate mode to find an initial set of model parameters that can then be made it possible to calculate the experimental equilibria in the normal mode.

- When you have achieved a satisfactory fit between calculated results and experimental data with an appropriate number of optimization variables, perform a couple of final optimization runs to make sure that your results are stable.
- Make sure that the values of the optimizing variables keep within a reasonable range. If a variable starts to change several orders of magnitudes, then you must consider whether you have the right weights on the experiments and whether you are using too many optimizing variables.

---

## Excluding and Including the Correct Equilibria

---

Use as few experiments as possible initially to get a reasonable overall fit. It is generally best to first focus on the invariant equilibria, as well as metastable states that can be estimated by excluding some phases. Crucially, any experiments for which the computed results are clearly faulty should be excluded from the final critical set, while all important invariant equilibria are calculated and included in this critical set.

If you cannot compute some invariant equilibria with intermediate phases, then it may be best to first exclude those phases from the optimization and only optimize the liquid and the most important solution phases. After you have obtained reasonable results for these phases, you can enter the intermediate phases back in again. With the variables for the optimized liquid and solid phases set fixed, you can finally optimize the intermediate phases.

---

## Conflicting Data

---

It is important that you do not include conflicting sets of data at the same time during an optimization. After all, if they are genuinely conflicting, then at least one of the sets must be incorrect. If you have conflicting sets of data, then only use one of these sets at a time together with the rest of the data. Often, the optimization makes it clear which dataset coheres best with the other information that is available about the system.

Sometimes, conflicting data cannot be detected directly. For example, you may have activity data which is inconsistent with solubility data from the phase diagram. This would be indicated by large errors in the fit when both the activity data and the solubility data are included. To find these inconsistencies, let alone correct the errors, you may thus have to optimize with some datasets excluded.

---

## Experiments and Experimental Data

---

These topics contain examples of different kinds of equilibria in different kinds of systems. They are intended to help you figure out how to use various kinds of experimental data when creating POP files. The topics that follow contain examples how to enter various kinds of experimental data into a POP file.

### In this section:

---

Phase Diagram Data .....	83
Data for Individual Compounds .....	84
Single Phase Mixing Enthalpies or Partial Enthalpies .....	86
Enthalpies of Formation and Enthalpies of Reactions .....	87
Chemical Potentials via EMF Measurements .....	88
Driving Force for Metastable Phases .....	89
Pressure-Temperature-Volume EOS Data .....	90
Crystal Structure, Lattice Parameters, and Site-Occupancy .....	91
Magnetism and Other Atomistic Properties .....	92
Data About Systems of Different Orders .....	93

## Phase Diagram Data

P-T-X phase diagram data for binary and ternary systems, or systems of even higher order can be used in the POP file. This includes data of the following types:

- Data about the temperature- and composition-dependence of various properties (obtained from differential thermal analysis (DTA) measurements for example), including enthalpy, lattice parameter, dilatometric length, electronic conductivity and magnetic susceptibility;
- Data about reaction types (invariant/monovariant) and phase relations obtained from qualitative or quantitative metallographic studies;
- Data about phase amounts or compositions obtained from microprobe measurements (using X-ray spectroscopy or transmission electron microscopy (TEM));
- Data about the positions or directions of two-phase tie-lines and three-phase equilibria for ternary systems.

The following table shows a POP file excerpt that demonstrates how some kinds of phase diagram data can be entered.

Example	ED_EXP commands
Solidus (melting temperature) of an Au-Cu alloy	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE LIQUID FCC=FIX 1 SET_CONDITION X(FCC,CU)=0.14 P=1E5 EXPERIMENT T=970:2</pre>
Three-phase equilibrium in a binary A-B system	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC BCC LIQUID=FIX 1 SET_CONDITION P=1E5 EXPERIMENT T=912:5</pre>
Congruent transformation in a binary A-B system	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE BCC LIQUID=FIX 1 SET_CONDITION P=1E5 X(BCC,B)-X(LIQ,B)=0 EXPERIMENT T=1213:10</pre>
Tie-line in a ternary A-B-C system	<p>An uncertainty factor is set not only for the experimental measurement, but also for the X(FCC,B) condition. The factor is 0.02.</p> <pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC BCC=FIX 1 SET_CONDITION T=1273 P=1E5 X(FCC,B)=0.1:0.02 EXPERIMENT X(FCC,C)=0.12:.02</pre>

## Data for Individual Compounds

You can use data about individual components in the POP file. Such data includes activity, heat capacity, heat content, entropy, Gibbs energy of phase transformations, heat of transition, and heat of melting.

Example	ED_EXP commands
Carbon activity data in the fcc phase	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC_A1=FIX 1 CHANGE_STATUS PHASE GRAPHITE=D SET_REFERENCE_STATE C GRAPHITE,,,, SET_CONDITION P=101325 T=1273 X(MN)=0.03 SET_CONDITION X(C)=0.03 EXPERIMENT ACR(C)=0.29:5%</pre>
Ln(activity coefficient for O)	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE IONIC_LIQ=FIX 1 CHANGE_STATUS PHASE GAS=D SET_CONDITION X(O)=0.02 T=1523 P=101325 SET_REFERENCE_STATE O GAS,,,, ENTER_SYMBOL FUNCTION LNFO=LOG(ACR(O)/X(O)); EXPERIMENT LNFO=-1.5:0.01</pre>
Entropy at 298.15 K of Cu <sub>2</sub> O	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE CU2O=FIX 1 CHANGE_STATUS PHASE CUO=FIX 0 SET_CONDITION P=101325 T=298.15 EXPERIMENT S=92.36:1</pre>
Gibbs energy of formation of NiAl <sub>2</sub> O <sub>4</sub>	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE SPINEL=ENT 1 CHANGE_STATUS PHASE FCC_O2GAS=DORM SET_REFERENCE_STATE NI FCC,,,, SET_REFERENCE_STATE AL FCC,,,, SET_REFERENCE_STATE O O2GAS,,,, SET_CONDITION P=101325 T=1000 N(NI)=1 N(AL)=2 SET_CONDITION N(O)=4 EXPERIMENT GM=-298911:5%</pre>
Heat capacity of MgFe <sub>2</sub> O <sub>4</sub>	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE SPINEL=ENT 1 SET_CONDITION P=101325 N(Fe)=2 N(Mg)=1 N(O)=4</pre>

Example	ED_EXP commands
	<pre>SET_CONDITION T=800 ENTER_SYMBOL FUNCTION CP=H.T; EXPERIMENT CP=207:5%</pre>
Heat content (H-H <sub>298</sub> ) for MnFe <sub>2</sub> O <sub>4</sub>	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE SPINEL=ENT 1 SET_CONDITION P=101325 T=298.15 N(MN)=1 N(Fe)=2 SET_CONDITION N(O)=4 ENTER_SYMBOL VAR H298=H; ENTER_SYMBOL FUN HMNFE2O4=H-H298; CREATE_NEW_EQUILIBRIUM 2 1 CHANGE_STATUS PHASE SPINEL=ENT 1 SET_CONDITION P=101325 N(MN)=1 N(Fe)=2 N(O)=4 SET_CONDITION T=400 EXPERIMENT HMNFE2O4=16610:5%</pre>

## Single Phase Mixing Enthalpies or Partial Enthalpies

You can use calorimetric data for mixing in the POP file. This data could be the result of, for example, drop calorimetry or scanning calorimetry. This includes data about, for example, enthalpy of mixing of liquids, series of mixing-enthalpy and about partial enthalpy. Information about enthalpies of mixing can also be derived from theoretical principles using special quasi-random structures (SQSs).

Example	ED_EXP commands
Enthalpy of mixing in the liquid state of an Au-Cu alloy	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE LIQUID=FIX 1 SET_CONDITION T=1379 P=1E5 SET_CONDITION X(LIQUID,AU)=0.0563 SET_REFERENCE_STATE AU LIQ * 1E5 SET_REFERENCE_STATE CU LIQ * 1E5 EXPERIMENT HMR=-1520:200</pre>

## Enthalpies of Formation and Enthalpies of Reactions

You can use calorimetric data for phase transformations in the POP file. This data could be the result of, for example, direct-reaction calorimetry, solution calorimetry or combustion calorimetry. Information about enthalpies of formation can also be derived from theoretical principles.

Example	ED_EXP commands
Enthalpy of melting of the eutectic in the Al <sub>2</sub> O <sub>3</sub> -ZrO <sub>2</sub> binary (J/g)	<pre> CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE CORUNDUM ZRO2_TETR=FIX 0 CHANGE_STATUS PHASE IONIC_LIQ=ENT 1 SET_CONDITION P=101325 N=1 LNAC(O)=-80 SET_REFERENCE_STATE ZRO2 TETR,,,,, SET_REFERENCE_STATE AL2O3 CORUND,,,,, EXPERIMENT HWR=1080:90           </pre>
Enthalpy of formation of o-CaZrO <sub>3</sub> from the component oxides	<pre> CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE ZRO2_MONO HALITE CAZRO3_ O=ENT 1 SET_CONDITION T=298.15 P=101325 N(CAO)=.5 SET_CONDITION N(ZRO2)=.5 LNAC(O)=-20 SET_REFERENCE_STATE CAO HALITE,,,, SET_REFERENCE_STATE ZRO2 ZRO2_MONO,,,, EXPERIMENT HMR(CAZRO3_O)=-15960:5%           </pre>

## Chemical Potentials via EMF Measurements

You can use chemical potentials and activities/activity coefficients from electromagnetic field (EMF) measurements in the POP file.

Example	ED_EXP commands
Emf data on Cu(s)/Cu <sub>2</sub> O(s)	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC CU2O=FIX 1 CHANGE_STATUS PHASE O2GAS=DORM SET_CONDITION P=101325 T=1000 SET_REFERENCE_STATE O O2GAS,, 100000 EXPERIMENT MUR(O)=-95387:1000</pre>

## Driving Force for Metastable Phases

You can use chemical driving forces for metastable phases in the POP file.

Example	ED_EXP commands
To prevent BCC_A2 to be stable	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC_A1 M7C3 M23C6=FIX 1 CHANGE_STATUS PHASE BCC_A2=D SET_CONDITION P=101325 T=1473 EXPERIMENT DGM(BCC)&lt;-0.1:0.1</pre>
To promote CEMENTITE to be stable	<pre>CHANGE_STATUS PHASE BCC M7C3 GRAPHITE=FIX 1 CHANGE_STATUS PHASE CEMENTITE=DORMANT SET_CONDITION P=P0 T=973 EXPERIMENT DGM(CEM)&gt;0.1:0.01</pre>

## Pressure-Temperature-Volume EOS Data

You can use Pressure-Temperature-Volume Equation of State (EOS) data in the POP file. Such data includes, for example, molar volume, density, thermal expansion and bulk modulus/compressibility.

Example	ED_EXP commands
Volume of liquid Pt	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE LIQUID=ENT 1 SET_CONDITION P=101325 N=1 T=1873 EXPERIMENT VM=1.00758E-5:5%</pre>
Thermal expansion	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC_A1=ENT 1 SET_CONDITION P=101325 N=1 T=400 ENTER_SYMBOL FUNCTION ALPHA=VM.T/VM/3; EXPERIMENT ALPHA=2.35E-5:5%</pre>

## Crystal Structure, Lattice Parameters, and Site-Occupancy

You use information about defects, ordering or site occupancy, as well as information about crystal structures in the form of lattice parameter values. This information may come from measurements made by X-ray, neutron diffraction, Mössbauer spectroscopy or perturbed angular-correlation.

Example	ED_EXP commands
Lattice parameter for fcc	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE FCC_A1=ENT 1 SET_CONDITION P=101325 N=1 T=298.15 SET_CONDITION X(CR)=0.05 ENTER_SYMBOL FUNCTION LPFCC=((4*VM/6.02214179E23)** (1/3))*1E10; EXPERIMENT LPFCC=4.02:5%</pre>
Lattice parameter for bcc	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE BCC_A2=ENT 1 SET_CONDITION P=101325 N=1 T=298.15 SET_CONDITION X(AL)=0.08 ENTER_SYMBOL FUNCTION LPBCC=((2*VM/6.02214179E23)** (1/3))*1E10; EXPERIMENT LPBCC=2.90:5%</pre>
Site-occupancy, degree of inversion for spinel	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE SPINEL=ENT 1 SET_CONDITION P=101325 N(MG)=1 N(AL)=2 N(O)=4 SET_CONDITION T=1073 EXPERIMENT Y(SPINEL,AL+3#1)=0.31:5%</pre>

## Magnetism and Other Atomistic Properties

You can use data about atomistic properties such as magnetism and Curie temperatures in the POP file.

Example	ED_EXP commands
Curie temperature of $\text{MnxFe}_{3-x}\text{O}_4$ , $x=1.26$	<pre>CREATE_NEW_EQUILIBRIUM 1 1 CHANGE_STATUS PHASE SPINEL=ENT 1 SET_CONDITION T=520 P=101325 N(O)=4 SET_CONDITION N(Fe)+N(Mn)=3 N(Mn)=1.26 EXPERIMENT TC(SPINEL)=520:5</pre>

## Data About Systems of Different Orders

You might want to use data about systems of different orders in the same optimization, such as data about a binary equilibrium with components A and B in a ternary system with components A, B and C, together with data about equilibria in the binary system with components A and B. To do this, you must first suspend all components in the ternary system (A, B and C), and then set the two components you want binary information on (A and B) to status `ENTERED`.

Suspending all components is done by giving the initialization code 0 to the [CREATE\\_NEW\\_EQUILIBRIUM](#) command. The components about which you want to use binary information must then be entered again using the [CHANGE\\_STATUS](#) command.

Example	ED_EXP commands
A binary (A-B) three-phase equilibrium (FCC-BCC-LIQ) in a ternary system (A-B-C)	<pre>CREATE_NEW_EQUILIBRIUM 1 0 CHANGE_STATUS COMPONENT A B = ENTERED CHANGE_STATUS PHASE FCC BCC LIQ=FIX 1 SET_CONDITION P=1E5 EXPERIMENT T=1177:10</pre>

---

# Thermodynamic Models

---

The most common models used in assessments are described briefly in these topics. Examples of how to write the phase definitions in the setup-file are also given for each of these models. In addition to the models described here, a large number of thermodynamic models for various phases in different states have been implemented.

**In this section:**

---

Gas .....	95
Compound Energy Formalism .....	96
Substitutional Liquid .....	97
Ionic Two-Sublattice Liquid Model .....	98
Models for Ordered Phases .....	99
4SL Model for FCC, HCP, and BCC .....	100
2SL Model for FCC, HCP, and BCC .....	102
Magnetic Ordering .....	103
Molar Volumes and High Pressure .....	104
Excess Models .....	105
General Einstein Model .....	106

## Gas

---

The gaseous mixture phase is usually treated as a substitutional phase without sublattice. In the gaseous mixture phase, there are usually molecules formed, and the number of constituents of the gas phase is often much larger than the number of elements. These complex constituents have to be defined as species. The ideal gas model may handle a gas phase under low pressures (and low temperatures). This implies that the P-V-T relations and thermodynamic properties of pure gaseous species are calculated as for ideal gas, and that there is interaction between gaseous species in the mixture. In a defined system, there may only exist one gaseous mixture phase. Type code `G` is used for the gas phase:

```
ENTER_PHASE GAS G, 1 H,H1O1,H1O2,H2,H2O1,H2O2,O,O2,O3; N N
```

## Compound Energy Formalism

---

If the atoms are sufficiently different in size, electronegativity or charge, they may prefer different types of sites in the lattice of crystalline solids. In some cases, a solute atom may even occupy interstitial sites between the normal lattice sites. All such phenomena are treated in Thermo-Calc by the sublattice concept. The Compound Energy Formalism (CEF) has proved to be the most general formalism for many different types of solution phases. It is therefore by default applied to various solid solution phases in the Thermo-Calc software package. CEF can also take ionic constraints (charged cation/anion species) into account. For a comprehensive description on the CEF, please see Hillert (2001), Frisk and Selleby (2001), Sundman and Ågren (1981).

Since the CEF is used by default, no special notation is needed in the phase definition, except for ionic phases, where type code `I` is given.

Examples of phase definitions in a setup-file using the CEF:

```
ENTER_PHASE SPINEL I, 4 1 2 2 4
    FE+2,FE+3; FE+2,FE+3,VA; FE+2,VA; O-2; N N
ENTER_PHASE SIGMA, 3 10 4 16
    AL,CO,CR,FE,NB; AL,CO,CR,FE,NB; AL,CO,CR,FE,NB; N N
ENTER_PHASE FCC_A1, 2 1 1
    CR,FE,MN; C,N,VA; N N
ENTER_PHASE BCC_A2, 2 1 3
    CR,FE,MN,VA; C,N,VA; N N
```

---

## Substitutional Liquid

---

The liquid phase is normally (but not always) modeled as a substitutional phase with no sublattice, which is occupied by two or more constituents. The constituents are normally the elements, but molecules or associates can also be used. Type code `L` is used for the liquid phase.

```
ENTER_PHASE LIQUID L, 1
    AL,C,CR,FE,MO,NI,SI ; N N
ENTER_PHASE LIQUID L, 1
    C,FE,FEO,FEO3/2,NI,NIO ; N N
```

## Ionic Two-Sublattice Liquid Model

---

Within the framework of the CEF, the ionic two-sublattice liquid (I2SL) model was developed to be used when there is a tendency for ionization in the liquid, which happens in liquid oxides and sulphides for example. The same model can be used both for metallic and oxide melts. At low levels of oxygen, the model becomes equivalent to a substitutional solution model between metallic atoms. Two sublattices are assumed, one containing charged cations and one containing charged anions, neutrals and vacancies. The complication for an ionic liquid is that the numbers of sites on the cation sublattice and anion sublattice vary with the composition to preserve electroneutrality. For a comprehensive description on the I2SL model, see [1985Hil; 1991Sun]. In these references, the possible excess parameters are also discussed. Type code *Y* is used for the I2SL model.

Examples of phase definitions in a setup-file:

```
ENTER_PHASE LIQUID Y,  
    AL+3,CR+2,FE+2,NI+2; VA; N N  
ENTER_PHASE IONIC_LIQ Y,  
    CA+2,MG+2; O-2; N N  
ENTER_PHASE IONIC_LIQ Y,  
    AL+3,CA+2,FE+2,MG+2,SI+4;  
    ALO2-1,O-2,SIO4-4,VA,FE03/2,SIO2; N N
```

## References

- [1985Hil] M. Hillert, B. Jansson, B. Sundman, J. Ågren, A two-sublattice model for molten solutions with different tendency for ionization. *Metall. Trans. A.* 16, 261–266 (1985).
- [1991Sun] B. Sundman, Modification of the two-sublattice model for liquids. *Calphad.* 15, 109–119 (1991).

## Models for Ordered Phases

---

Some solution phases have chemical order-disorder transformations: for example, the disordered FCC\_A1 phase can transform to the ordered L1<sub>2</sub> or L1<sub>0</sub> structure where the atoms of different kinds occupy different sublattices. In some cases, it may be advantageous and easy to describe such ordered structures as completely different phases; but in some other cases, like the B2-ordered BCC structure in Al-Fe, the disordered and ordered structures must be described as one single phase because the transformation is of second order along a line in temperature and composition.

The ordering can easily be described by the CEF. For instance, an ordered FCC phase with only L1<sub>2</sub> or L1<sub>0</sub> ordering can be efficiently handled by the so-called Two Substitutional-Sublattice Ordering Model (2SL model), while an ordered FCC with both L1<sub>2</sub> and L1<sub>0</sub> ordering must be described with four sublattices using the so-called Four Substitutional-Sublattice Ordering Model (4SL model). Please note that in both these models, all the substitutional constituents must enter into these sublattices. In addition, you may also have one sublattice for interstitials.

Thermo-Calc supports a feature of splitting the parameters of a chemically ordered phase onto two different phase descriptions. This is very convenient. In this way, you can have one phase for all parameters describing the disordered state (the configuration independent state) and in the other phase only those parameters needed to describe the remaining part (the configuration dependent part), that is, the Gibbs energy contribution due to the ordering transformation. This has been implemented in such a way that the contribution from the configuration dependent part is zero when the solution phase is disordered. The optimization of the disordered phase is independent of the order-disorder transformation; the ordering energy is an add-on to the disordered part. This is particularly useful in higher order systems, as an ordered phase can dissolve several elements that have no particular contribution to the actual ordering. The phases Gibbs energy add together by giving this command in the setup-file: `AMEND_PHASE_DESCRIPTION <ordered phase> disordered_part <disordered phase>`

Since PARROT does not automatically create new composition sets, it is necessary to create composition sets for both the ordered and disordered phases in the setup-file: `Amend_phase_description <phase> composition_sets <new highest set number>,,,,`

Example of phase definitions for the corresponding disordered and ordered parts in a setup-file:

```
ENTER_PHASE FCC_A1, 2 1 1
      AL,CR,FE,NI; VA; N N
ENTER_PHASE FCC_L12, 3 .75 .25 1
      AL,CR,FE,NI ; AL,CR,FE,NI; VA; N N
AMEND_PHASE_DESCRIPTION FCC_L12 DIS_PART FCC_A1
```

## 4SL Model for FCC, HCP, and BCC

Ordered FCC, HCP and BCC solution phases handled by the 4SL model requires four sublattices for substitutional ordering and can additionally have an interstitial sublattice. A 4SL model is especially useful for modelling different kinds of ordered phases that are based on the same disordered phase, such as L1<sub>0</sub> and L1<sub>2</sub> based on FCC\_A1, and B2, D0<sub>3</sub>, L2<sub>1</sub> and B32 based on BCC\_A2 etc.

For ordered FCC or HCP phases, these four substitutional sublattices represent four corners of the regular tetrahedron on these lattices, all of which are the nearest neighbours. These corners are equivalent lattice points, thus all G parameters for each end-member with the same elements, but distributed on different sites, must be identical. It should be emphasized that the end-member energy here represents the ordering energy rather than the formation energy of the compound.

The constraints on the parameters in the 4SL model can be derived based on the symmetry of the lattice:

$$G_{A:B:B:B} = G_{B:A:B:B} = G_{B:B:A:B} = G_{B:B:B:A}$$

$$G_{A:A:B:B} = G_{A:B:A:B} = G_{A:B:B:A} = G_{B:A:A:B} = G_{B:A:B:A} = G_{B:B:A:A}$$

$$G_{B:A:A:A} = G_{A:B:A:A} = G_{A:A:B:A} = G_{A:A:A:B}$$

$$L_{A,B:*,*:*} = L_{*,A,B:*,*} = L_{*,*,A,B:*} = L_{*,*,*:,A,B}$$

The asterisk \* means that the interaction parameter is independent of the occupation of that sublattice. For the disordered phase to be completely disordered, i.e. that all site fractions are equal on all four sublattices, all constraints must be correct.

For ordered BCC phases, the situation is a bit more complicated, as the four sublattice ordering phase represents an irregular tetrahedron with two pairs of sites that are next nearest neighbors. Thus, for an A-B binary solution phase, with A located on two sublattice sites and B on the other two, the end-member described by G<sub>A:A:B:B</sub> has four nearest neighbour bonds between A and B atoms, whereas the end-member described by G<sub>A:B:A:B</sub> has two nearest neighbour bonds between A and B atoms and two next nearest neighbour bonds. Many parameters thus have a relation:

$$G_{A:B:B:B} = G_{B:A:B:B} = G_{B:B:A:B} = G_{B:B:B:A}$$

$$G_{B:A:A:A} = G_{A:B:A:A} = G_{A:A:B:A} = G_{A:A:A:B}$$

$$G_{A:A:B:B} = G_{B:B:A:A}$$

$$G_{A:B:A:B} = G_{A:B:B:A} = G_{B:A:A:B} = G_{B:A:B:A}$$

---

$$L_{A,B:*.:.} = L_{*:A,B:*.} = L_{*:.*:A,B:} = L_{*:.*:*.} A, B$$

The  $G_{A:B:B:B}$  term represents D0<sub>3</sub>,  $G_{A:A:B:B}$  B2 and the  $G_{A:B:A:B}$  B32 ordering. There are also two kinds of reciprocal interaction parameters:

$$L_{A,B:A,B:*.} ; 0 \dots 9$$

$$L_{A,B:*.} A, B:*.} ; 0 \dots 9$$

Kusoffsky et al. (2001) have shown the influence of different parameters available in the 4SL model applied to fcc ordering. The possibility to use the 4SL model to BCC ordering has been studied by Sundman et al. (2009) in the Al-Fe system.

---

## 2SL Model for FCC, HCP, and BCC

---

The reason for using a 2SL model instead of a 4SL model is that the calculations are faster, but on the other hand it can only model one kind of ordered phase(s). For instance, it is not possible to model FCC with both  $L1_0$  and  $L2_1$ , or BCC with  $DO_3$  and B2, etc.

In order to get the fully disordered phase to form, constraints are needed as for the 4SL model. For the symmetric phases, e.g. B2 and  $L1_0$ , the relation between parameters are:

$$G_{A:B} = G_{B:A}$$

$$L_{A,B:*} = L_{*:A,B}$$

For a 2SL asymmetric model including, but not limited to,  $L1_2$ , more constraints are needed. This model has many complicated relations between the parameters. The relation between the parameters for the 2SL  $L1_2$  model can be derived from a 4SL model. The conversion from 4SL model parameters to 2SL parameters for  $L1_2$  can be found in Dupin et al. (2001) and for  $L1_0$  in Yuan et al. (2012). Higher-order systems with an  $L1_2$  phase modeled with two sublattices require a lot of ternary and some quaternary interaction parameters in order to make the disordered state stable. These parameters have been derived by e.g. Dupin (1995), also found in Kusoffsky et al. (2001).

Many publications exist on how to model order/disorder transformations using a two-sublattice model, see e.g. Dupin, Ansara (1999), De Keyzer et al. (2009).

## Magnetic Ordering

---

The magnetic contribution to the Gibbs energy is given by a model proposed by Inden (1975) and adapted by Hillert and Jarl (1978), and is described by:

$$G=RT\ln(b+1) f(t)$$

where  $t=T/T_C$ .

The values of the Curie temperature,  $T_C$ , and Bohr magneton number,  $b$ , at a certain composition are then used in an optimization procedure to calculate the contribution to Gibbs energy due to magnetic ordering. Curie temperature and Bohr magneton number are given in the command [ENTER\\_PARAMETER](#) for phases that undergo a magnetic transition:

```
AMEND_PHASE_DESCRIPTION BCC_A2 MAGNETIC -1.0 0.4
ENTER_PARAMETER TC(BCC_A2,FE:VA;0) 298.15 +V1; 6000 N
ENTER_PARAMETER BMAGN(BCC_A2,FE:VA;0) 298.15 +V2; 6000 N
```

The arguments -1 and 0.4 specify the values of parameters in the magnetic model: -1 is the antiferromagnetic factor and 0.4 is the structure factor. Usually the antiferromagnetic factor =-1 for bcc and =-3 for fcc (and other non-bcc phases). The structure factor corresponds to the short range order fraction of the enthalpy. This factor is 0.4 for bcc and 0.28 for non-bcc phases.

## Molar Volumes and High Pressure

---

Including volumes in a thermodynamic database enable calculations of volume changes, volume fraction, lattice mismatch in cubic structures and phase diagrams at increased pressure. Pressure independent volumes can be used up to about 1 GPa. At pressures above 1 GPa the pressure dependence of the volumes must be included by using a proper equation of state (EOS). An EOS has been developed by Lu et al. (2005), based on the EOS from Jacobs and Oonk (2000). Four parameters are used in the implementation of the model in Thermo-Calc:

- $v_0$ : Volume at 1 bar and at reference temperature  $T_0$
- $v_A$ : Integrated thermal expansivity
- $v_K$ : Isothermal compressibility at 1 bar
- $v_C$ : Parameter to fit high pressure data

All parameters can be composition dependent in the same way as all other model parameters in Thermo-Calc. Experimental information about the temperature, pressure and composition dependence of the volume can be assessed using these model parameters. To evaluate  $v_0$ ,  $v_A$  and  $v_K$  experimental data at 1 bar only are needed.

Examples on modeling molar volume in a setup-file:

```
ENTER_PARAMETER V0 (BCC_A2, FE:VA;0) 298.15 +V1; 6000 N
ENTER_PARAMETER VA (BCC_A2, FE:VA;0) 298.15 +V2*T+V3*T**2; 6000 N
ENTER_PARAMETER VC (BCC_A2, FE:VA;0) 298.15 +V4+V5*T; 6000 N
ENTER_PARAMETER VK (BCC_A2, FE:VA;0) 298.15 +V6+V7*T; 6000 N
ENTER_PARAMETER V0 (FCC_A1, CR, FE:VA;0) 298.15 +V8; 6000 N
ENTER_PARAMETER VA (FCC_A1, CR, FE:VA;0) 298.15 +V9*(T-298.15); 6000 N
```

## Excess Models

---

Unless specifically amended, the excess Gibbs energy terms are modeled by the Redlich-Kister-Muggianu Model. Other excess models for the Gibbs energy are also implemented in Thermo-Calc, such as e.g. Kohler and Toop-Kohler extrapolation models.

Thermo-Calc is not restricted to binary interaction parameters. Ternary, quaternary and higher-order interaction parameters can also be used if there is enough information to evaluate these from experimental data. The ternary interaction parameters are identified by the degrees of the so-called ternary L parameter (i.e. 0, 1 and 2) in the command [ENTER PARAMETER](#). If the parameter is composition-independent, you only need a single interaction parameter with a degree of zero, i.e. only the  $^0L$  term. If there is a ternary composition-dependent interaction parameter, you must enter all three parameters (0,1 and 2). If such a ternary interaction parameter should be zero, then assign it a function which is zero. If the three interaction parameters have the same value, then that is identical to having a composition-independent ternary parameter.

```
ENTER_PARAMETER L(LIQUID,A,B,C;0) 298.15 +1000; 6000 N
ENTER_PARAMETER L(LIQUID,A,B,C;1) 298.15 +1000; 6000 N
ENTER_PARAMETER L(LIQUID,A,B,C;2) 298.15 +1000; 6000 N
```

is identical to

```
ENTER_PARAMETER L(LIQUID,A,B,C;0) 298.15 +1000; 6000 N
```

This also means that

```
ENTER_PARAMETER L(LIQUID,A,B,C;0) 298.15 +1000; 6000 N
ENTER_PARAMETER L(LIQUID,A,B,C;1) 298.15 +ZERO#; 6000 N
ENTER_PARAMETER L(LIQUID,A,B,C;2) 298.15 +ZERO#; 6000 N
```

is NOT identical to

```
ENTER_PARAMETER L(LIQUID,A,B,C;0) 298.15 +1000; 6000 N
```

## General Einstein Model

An Einstein model implemented in a general way with multiple Einstein temperatures and accompanying weight functions.

It describes the vibrational contribution to Cp at low and intermediate temperatures.

The model is implemented as:

$$G_{Einstein} = f \times n \times R \times \left( \frac{3}{2} \Theta + 3T \times \ln(1 - e^{-\frac{\Theta}{T}}) \right)$$

Where

- $\Theta$  is the Einstein temperature.  $\ln(\Theta)$  is entered as the database parameters LNTHEA1, LNTHEA2, LNTHEA3, LNTHEA4, and LNTHEA5.
- $f$  is a weight function, entered as the database parameters THETA1, THETA2, THETA3, THETA4, and THETA5.
- $n$  is the number of atoms per mole formula unit
- $R$  is the universal gas constant
- $T$  is the temperature

Examples of parameters for this model:

```
PARAMETER LNTHEA1 (GRAPHITE,C;0) 298.15 +LN(1953); 6000 N REF0 !
PARAMETER THETA1 (GRAPHITE,C;0) 298.15 +0.484786; 6000 N REF0 !
PARAMETER LNTHEA2 (GRAPHITE,C;0) 298.15 +LN(947); 6000 N REF0 !
PARAMETER THETA2 (GRAPHITE,C;0) 298.15 +0.349135; 6000 N REF0 !
PARAMETER LNTHEA3 (GRAPHITE,C;0) 298.15 +LN(448); 6000 N REF0 !
PARAMETER THETA3 (GRAPHITE,C;0) 298.15 +0.121463; 6000 N REF0 !
PARAMETER LNTHEA4 (GRAPHITE,C;0) 298.15 +LN(193); 6000 N REF0 !
PARAMETER THETA4 (GRAPHITE,C;0) 298.15 +0.0387523; 6000 N REF0 !
PARAMETER LNTHEA5 (GRAPHITE,C;0) 298.15 +LN(65); 6000 N REF0 !
PARAMETER THETA5 (GRAPHITE,C;0) 298.15 +0.0058637; 6000 N REF0 !
```



The Einstein function also can be used with the GEIN(THETA) function.



For more information about this see [FUNCTION](#) and [PARAMETER](#) in the *Database Manager User Guide*.

## **References**

- [2001Che] Q. Chen, B. Sundman, Modeling of thermodynamic properties for BCC, FCC, liquid, and amorphous iron. *J. Phase Equilibria*. 22, 631–644 (2001).
- [2018Big] S. Bigdeli, Q. Chen, M. Selleby, A New Description of Pure C in Developing the Third Generation of Calphad Databases. *J. Phase Equilibria Diffus.* 39, 832–840 (2018).

# **Database Manager User Guide**

**Thermo-Calc Version 2024a**



---

# Introduction to the Database Manager Guide

---

**In this section:**

---

Thermodynamic and Kinetic/Mobility Databases .....	3
Initialization of the TDB Module .....	4

---

## Thermodynamic and Kinetic/Mobility Databases

---

This guide describes how to define and construct a thermodynamic or kinetic database. It includes details about the data structures and formats of the Thermo-Calc databases as well as the Diffusion Module (DICTRA) database-extensions.

The Thermo-Calc databases are created and maintained in the Thermo-Calc Database Format (TDB) which is an international standard for the CALPHAD-type thermodynamic calculations and kinetic simulations. This document gives a full description of the TDB format (with the Diffusion Module (DICTRA) database extension) and is a comprehensive guide about how to create and manage a Thermo-Calc (and Diffusion Module (DICTRA)) database.

Usually, a Thermo-Calc or Diffusion Module (DICTRA) database is constructed as a single text file with an extension of TDB, for example, `PAQ2setup.TDB`. For some databases (especially large databases used with older versions of Thermo-Calc or Diffusion Module (DICTRA)) a database may consist of one primary-definition file (`***setup.TDB`) and several additional-definition TDB files (such as `***param.TDB`, `***funct.TDB`, `***refer.TDB`, etc. These are built for a variety of defined parameters, functions, references and so forth, and are linked to the `***setup.TDB` file through the correct use of the [TYPE\\_DEFINITION](#) and/or [FTP\\_FILE](#) commands inside the `***setup.TDB` file.



With Thermo-Calc, the Add-on Modules and Thermo-Calc Software Development Kits (SDKs), you can code everything into a single `***setup.TDB` file for a database at any scale.

## Initialization of the TDB Module

---

The following explains how a database initialization file is constructed to work with the so-called TDB (database) module. When you start, the TDB module looks for a file that has information about the available standard databases. The database initialization file is called `tc_initd.tdb`. By default it is located in the data subdirectory of the Thermo-Calc installation directory.



See [Database Installation Folder](#) in the *Thermo-Calc User Guide*.

The location can be overridden by:

- A command line parameter `-t` followed by the path to the directory containing the `tc_initd.tdb` file.
- The environment variable `TC_DATABASE_DIRECTORY` pointing to the directory containing the `tc_initd.tdb` file. This overrides both the default value and a directory given with the `-t` command line argument.

Each database defined in `tc_initd.tdb` represents a database that can be used within Thermo-Calc. Each entry has three parts:

1. The database short-name (e.g. `FEDEMO`, maximum seven characters, no white space, no commas).
2. The path to the database setup file (max 78 characters, no commas, can contain white space if it is a quoted absolute path).
3. A short description of the database (max 60 characters, can contain white space). The entry must end with an exclamation mark (!).

The paths can be relative to the `tc_initd.tdb` file or absolute. For platform independence, the recommendation is to always use forward slashes (/) in the paths, but on Windows backslash (\) also works.

You can add comment lines in the database initialization file. Comments must start with a `$` sign.

### Examples

The following are examples of an initialization file defining two databases:

```
SSOL8 SSOL8\SSOL8SETUP.TDC  SGTE Solutions Database version 8.0 !
MYDB1 ../../../../MYDATABASES/ABC.TDB My database 1.0 !
```

In the initialization file you can also add databases with absolute file names with white space, if the path is surrounded by quotation marks ("):

```
TEST "G:\My Drive\My Folder\Test.TDB" My test database v0.1!
```



The use of absolute paths should be used with caution as these are not portable.

---

# The Database Definition File and Keywords

---

**In this section:**

---

Database Definition File Syntax .....	9
ELEMENT .....	11
SPECIES .....	12
PHASE .....	14
CONSTITUENT .....	20
ADD_CONSTITUENT .....	22
COMPOUND_PHASE .....	23
ALLOTROPIC_PHASE .....	24
TEMPERATURE_LIMITS .....	25
DEFINE_SYSTEM_DEFAULT .....	26
DEFAULT_COMMAND .....	27
DATABASE_INFORMATION .....	28
TYPE_DEFINITION .....	29
FTP_FILE .....	33
FUNCTION .....	34
PARAMETER .....	38
OPTIONS .....	48
TABLE .....	49
ASSESSED_SYSTEMS .....	50
REFERENCE_FILE .....	54
LIST_OF_REFERENCE .....	56
ADD_REFERENCE .....	57
CASE and ENDCASE .....	59
VERSION_DATA .....	60

---





## Database Definition File Syntax

---

The database definition (\*.TDB) file (normally named `***setup.TDB`) consists of a set of keyword codes each followed by one or several parameters (arguments).

- A complete keyword entry must end with an exclamation mark (!).
- A single keyword entry can be up to 2000 characters long. However, the maximum length of a line in an \*.TDB file is 78 characters--it may be necessary to continue the keyword parameter (arguments) on several lines. The ! must be at the end of the last line.
- It is recommended to always have at least one empty space at the beginning of each continuation line for the keyword parameters (arguments); otherwise, the DATA module can misunderstand the parameters (or arguments), or issue an error message.
- The keyword and its various parameters (arguments) are separated by a space or a comma.
- A dollar sign (\$) in the first position of the line indicates that the line is a comment line, which is ignored by the DATA module.

When the database is selected the DATA module only reads the database definition file (`***setup.TDB`) once, from beginning to end. The DATA module checks continuously when reading the definition file. This implies that (nearly) everything must be declared or defined before it is used in any other way.

For example, if the GRAPHITE phase is to be included in the database definition, the element C (carbon) and the phase GRAPHITE must be defined before declaring that carbon dissolves in graphite. This definition order is necessary to build the internal data structure acceptable by the DATA module (during its consistency checking).

This topic gives a description of the available keywords and the appropriate arguments. A basic knowledge of the Gibbs Energy System (GIBBS) module is assumed.

This syntax is used:

```
KEYWORD [arg.1]*# [arg.2]*## {optional arg.3}!
```

The keywords are written in full length but can be abbreviated as long as the abbreviation is unique. A keyword may have syntax consisting of several arguments and optional arguments. The number, # or ##, in the notation, [...]\*# or [...]\*##, indicates an argument with a maximum length of # ASCII characters.



Arguments within square brackets [...] must always be given, but are optional when enclosed in curly brackets {...}.

## ELEMENT

---

```
ELEMENT [element name]*2 [ref. state]*24 [mass] [H298] [S298] !
```

The *element name* (maximum two characters) is the one found in the periodic chart but there are no naming restrictions. However, the GIBBS module only recognizes UPPER-case element names (if the `Upper Case Mode` is selected by the GIBBS command [REINITIATE](#)), which means that lower-case (if defined in a database) is automatically converted to UPPER-case by the DATA/GIBBS module.

The elements are automatically entered as species using the same names of the elements. If, for example, the species corresponding to FE needs to be named FE1, you can define the species as FE1, which results in an element named FE and a species named FE1. Vacancies (VA) and electrons (denoted either as /- in gaseous, liquid or solid phases, or ZE in an aqueous solution phase), need to be entered as special elements for correct handling by the DATA module.

The *reference (ref.) state* (maximum 24 characters) is the stable phase (at 298.15 K and 1 bar) that should contain this element and be used as the reference state for all element thermodynamic data. The *mass*, given in gram per mole, is used in various calculation programs and should always be given the correct value. *H298* and *S298* denote the enthalpy and entropy difference between 0 and 298.15 K for the element in SI units. If these are unknown, the values can be set to zero. All this information (reference state, H298 and S298) define the SER (Stable Element Reference state).

### Examples

```
ELEMENT /- ELECTRON_GAS      0.0  0.0  0.0  !
ELEMENT VA VACUUM           0.0  0.0  0.0  !
ELEMENT ZE UNIT_CHARGE      0.0000000001  0.0  0.0  !
ELEMENT AL FCC_A1           26.98154  4577.296  28.3215  !
ELEMENT C GRAPHITE          12.011  1054.0    5.74  !
ELEMENT FE BCC_A2           55.847  4489    27.28  !
ELEMENT O 1/2_MOLE_O2 (G)   15.9994  4341    102.5158  !
ELEMENT TI HCP_A3           47.88  4810    30.648  !
ELEMENT ZR HCP_A3           91.224  5566.27  39.181  !
ELEMENT ZY DUMMY            1        1        1  !
```

## SPECIES

```
SPECIES [species name]*24 [stoichiometric formula] !
```

This keyword defines species in the data structure. Every *species name* (maximum 24 characters) must be unique. The species are built from the predefined set of elements in the stoichiometric formula. If an undefined element is referenced, DATA displays an error message and the data structure is probably damaged.

The species names do not have to be the same as the stoichiometry formula, although in general this is recommended. The elements are automatically entered as species using the same names of the elements.



You can define a species name as a mixture of UPPER-case and lower-case letters in a database, but the DATA module automatically converts all lower-case to UPPER-case because the GIBBS module only recognizes UPPER-case species names.

When naming the species in a database, use special characters (such as +, -, \_, / and .) in species names. Avoid using other special characters (such as ( and )).

The *stoichiometric formula* is written with a simplified chemical notation, in which the chemical elements should always be given in UPPER-case and in any preferred order, and the stoichiometric coefficients are written in either real numerical factors or integer digits.



It is important that the numerical factor of 1 is not left out. Subgroups are not allowed in a stoichiometry formula; however, while specifying the stoichiometry formula for a specific species in a database, you can specify it in a way that some elements (always together with the corresponding partial stoichiometric coefficients) are repeated (as in the examples below).

### Examples

```
SPECIES AL2O3      AL2O3 !
SPECIES Silica    SI1O2 !
SPECIES NaSb_6OH  NA1SB1O6H6 !
```

---

```
SPECIES FE+2          FE/+2 !
SPECIES SB-3          SB/-3 !
SPECIES AlCl2/3       AL.33333CL.666667 !
SPECIES AL1CL1H2O2    AL1CL1H2O2 !
SPECIES AlCl3_3H2O    AL1CL3H6O3 !
SPECIES AlO2H2Cl.H6O3 AL1O2H2CL1H6O3 !
SPECIES AlCl2-OH.3H2O AL1CL2O1H1H6O3 !
SPECIES AlCl2OH.3Water AL1O1H1CL2H6O3 !
```

## PHASE

```
PHASE [phase name]*24 [data-type code]*8 [numb. subl.] [sites in
subl. 1] [sites in subl. 2] etc... {auxiliary text string} !
```

This keyword defines a phase and its properties (except for what species are allowed to enter it and for its parameters).

The *phase name* (maximum 24 characters) must be unique; otherwise the DATA module sees it as an attempt to redefine a previously defined phase. This causes DATA to display an error message and ignore the rest of the line. A phase name can be suffixed by an underscore ( \_ ) and letters to identify the physical state(s) or structure type(s) of the phase.

### Examples of Recommended Suffixes

Suffix	Definition
ABC_S	The ABC phase in solid state.
ABC_S2	The ABC phase in solid state 2.
ABC_S3	The ABC phase in solid state 3.
ABC_LT	The ABC phase in solid state at low temperatures.
ABC_HT	The ABC phase in solid state at high temperatures.
ABC_L	The ABC phase in liquid state.
ABC_LIQ	The ABC phase in liquid state.
FCC_A1	The FCC phase in disordered structure type A1.
FCC_L12	The FCC phase in ordered structure type L12.

The *phase name* can also be attached with a colon sign ( : ) and a letter for a legal GIBBS phase-type code (e.g. IONIC\_LIQ:Y and GAS:G).

## GIBBS Phase-type Codes

Code	Definition
G	Bit set for a gaseous mixture phase.
A	Bit set for an aqueous solution phase.
Y	Bit set for an ionic liquid solution phase (specially treated by the Ionic Two-Sublattice Liquid Model).
L	Bit set for a liquid solution phase (but not A (aqueous) or Y (ionic liquid)).
I	Bit set for a phase with charged species (but not G (gaseous), A (aqueous) or Y (ionic liquid)).
F	Bit set for an ordered FCC or HCP solution phase using the <i>Four Substitutional-Sublattice Ordering Model</i> (additionally, such a phase can also have interstitial sublattices).
B	Bit set for an ordered BCC solution phase using the <i>Four Substitutional-Sublattice Ordering Model</i> (additionally, such a phase can also have interstitial sublattices).



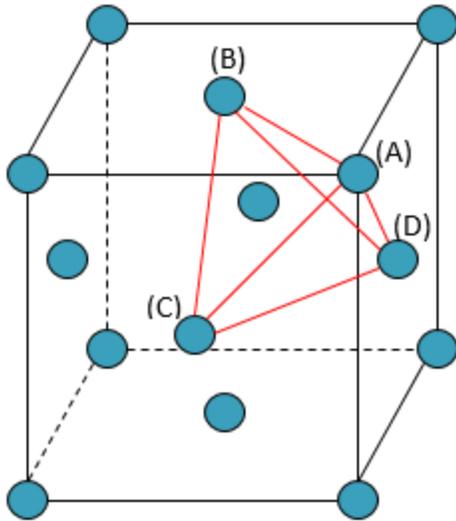
Other invalid characters (e.g.  $\text{M}$  or  $\text{P}$ ) are eventually treated, together with the colon (:), as a part of a phase name.

A  $\text{G}$  phase (gaseous mixture) or an  $\text{A}$  phase (aqueous solution) is usually treated as a substitutional phase without sublattice, and an  $\text{L}$  phase (ordinary liquid solution) is normally (but not always) modeled as a substitutional phase without sublattice, too.

For ordered FCC or HCP phases, these four substitutional sublattices represent four corners of the regular tetrahedron on these lattices, all of which are the nearest neighbours, as shown.

### FCC Unit Cell Example

An FCC unit cell with the lattice positions indicated that correspond to the G (FCC,A:B:C:D) end member. All lattice positions are equivalent for a four substitutional-sublattice ordering model.



A *Normal 4-Sublattice Model* requires that all the  $G$  parameters for each of the end-members with the same elements but distributed on different sites be given separately. However, as these corners are identical lattice points, the phase-type option  $F$  means that the  $G$  parameters need be given only once. The possible permutations are handled automatically.

## Additional Clarification

An  $A-B$  binary solution phase (with the element  $A$  located on one sublattice site and  $B$  on the other three sublattice sites) treated by the Normal 4-Sublattice Model has to have four  $G$  parameters for four end-members, i.e.

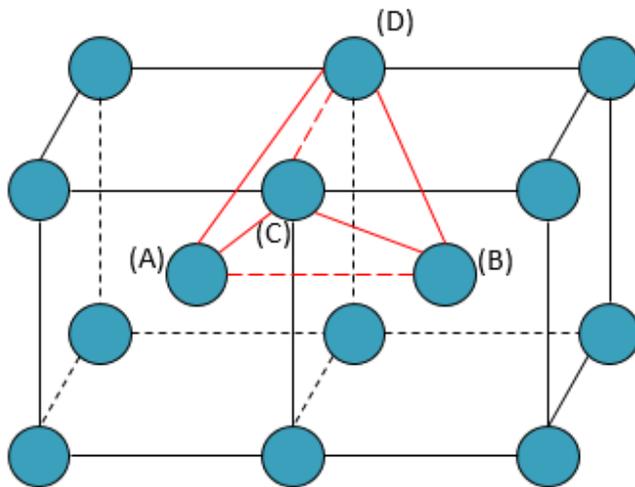
- $G(\text{phase}, A:B:B:B)$
- $G(\text{phase}, B:A:B:B)$
- $G(\text{phase}, B:B:A:B)$ , and
- $G(\text{phase}, B:B:B:A)$

This is because in the general case these  $G$  parameters can be different from each other. But for the FCC and HCP orderings, they are identical and thus all  $G$  parameters of such end-members need to be given only once, and the possible permutations are then automatically handled by the GIBBS module. Also, only one of the identical permutations is listed; in this example,  $G(\text{phase}, A:B:B:B)$  where it is alphabetically the first in the list of permutations. This significantly simplifies the usage of this model (*Four Substitutional-Sublattice Ordering Model*) in multicomponent alloys.

For ordered BCC phases, the phase-type option  $B$  means the same thing but it is more complicated since the 4-substitutional-sublattice ordering phase represents an irregular tetrahedron with two pairs of sites that are next nearest neighbours as shown:

### **BCC Unit Cell Example**

Two BCC unit cells with the lattice positions indicated that correspond to the  $G$  (BCC,A:B:C:D,0) end member. Lattice positions (A) and (B) are equivalent, as are lattice positions (C) and (B) for a four substitutional-sublattice ordering model.



For an end member described by the parameter  $G$  (phase, A:B:C:D) A and B are next nearest neighbours, as are C and D. And the nearest neighbours of A (or B) are C and D. Thus, for an A-B binary solution phase (with the element A located on two sublattice sites and B on two sublattice sites) treated by the Normal 4-Sublattice Model, the end-member described by the  $G$  (phase, A:A:B:B) term has four nearest neighbour bonds between A and B atoms, whereas the end-member described by the  $G$  (phase, A:B:A:B) term has two nearest neighbour bonds between A and B atoms and three next nearest neighbour bonds.

The first end-member (described by the  $G$  (phase, A:A:B:B) term) represents B2-ordering and the second (described by the  $G$  (phase, A:B:A:B) term) stands for B32-ordering. There are two permutations of the  $G$  (phase, A:A:B:B) term and four permutations of the  $G$  (phase, A:B:A:B) term, automatically conducted in the *Four Substitutional-Sublattice Ordering Model*. If you enter the unary, binary, ternary and quaternary parameters you are dealing with 1, 6, 21, 55 parameters for BCC:B, 1, 5, 15, 35 parameters for FCC:F and 1, 16, 81, 256 parameters for phases without F/B.

An additional feature with the phase-type options **F** and **B** is that a composition set that represents the solution phase has a suffix (indicating what ordering the phase has) that is automatically added to its phase name in some listings of equilibrium calculations (when performing either single-point or stepping or mapping calculations, and when plotting the calculated property diagrams or phase diagrams).

Such suffix indications can be:

<i>Solution Phase</i>	<i>Suffix for Disordered Phase</i>	<i>Suffix for Ordered Phase</i>
FCC PHASE	FCC_A1	FCC_L12
		FCC_L10
BCC PHASE	BCC_A2	BCC_B2
		BCC_B32
		BCC_D03
		BCC_L21
HCP PHASE	HCP_A3	HCP_D019
		HCP_B19



If you want to convert an existing database TDB file to use the F/B feature, add the phase-type code to the corresponding phase name in the PHASE and CONSTITUENT commands in the TDB-file, then when running the LIST\_DATA command in the Gibbs (GES) module. The created database file is in this less verbose format.

The *data-type code* consists of 1 to 8 characters where each character must stand for an action, which is to be coupled to this phase. The keyword TYPE\_DEFINITION, described below, must be used in the current database to specify what action should be taken by DATA for each character code.

The data entries [numb. subl.] [sites in subl. 1] [sites in subl. 2] etc., specify the total number of sublattices (always as an integer digit) and the sites (i.e. stoichiometric coefficients) of each of the sublattices (given in either integer digits or real numerical factors) for the phase.

Optionally, an *auxiliary text string* (maximum 10,000 characters) can be given after the last [sites in sublattice #] but before the exclamation mark !. This string displays in connection with the phase name in some listings within the DATA module.

### Examples

```
PHASE GAS:G % 1 1.0 !
PHASE LIQUID:L %ZCDQ 2 1.0 1.0
> Metallic liquid solution, modeled by CEF Model. !
PHASE IONIC-LIQ:Y %ZCDQ 2 1.0 1.0
> Ionic liquid solution, modeled by Ionic Two-Sublattice Model. !
PHASE SPINEL:I %ZA 4 1 2 2 4
> Complex Spinel Solution, by CEF model with ionic constraints. !
PHASE M23C6 % 3 20.0 3.0 6.0 !
PHASE FCC_A1 %&A 2 1 1
> Disordered FCC phase; also as MX carbides/nitrides. !
PHASE FCC_L10 %&AX 3 0.75 0.25 1
> Ordered FCC phase, modeled by 2-Sublattice Model for Ordering. !
PHASE FCC_L12:F %&AX 5 0.25 0.25 0.25 0.25 1.0
> Ordered FCC phase, modeled by 4-Sublattice Model for Ordering. !
PHASE AQUEOUS:A %HIJMR 1 1.0
> Aqueous Solution: using the Complete Revised HKF Model. !
```

## CONSTITUENT

---

```
CONSTITUENT [phase name]*24 [constituent description]*2000 !
```

This keyword (and the [ADD\\_CONSTITUENT](#) keyword for large solution phase) defines the phase-constitution as a list of constituents (for a substitutional phase with no sublattice) or of constituent arrays (for a sublattice phase).

The *phase name* (maximum 24 characters) must be a predefined phase (i.e. already through the [PHASE](#) keyword).



It is important that if a phase bears a legal phase-type (among G, A, Y, L, I, F and B) in its phase definition (already by the PHASE keyword; such as GAS:G, SLAG:L, LIQUID:L, IONIC\_LIQ:Y, SPINEL:I, FCC\_L12:F, HCP\_D021:F, BCC\_B2:B, AQUEOUS:A), such a valid phase-type code must also always be attached to the phase name in the CONSTITUENT keyword (and the ADD\_CONSTITUENT keyword).



Specifying the phase name in UPPER-case is recommended. You can define a phase name as a mixture of UPPER-case and lower-case letters in a database, but the DATA module automatically converts all lower-case to UPPER-case because the GIBBS module only recognizes UPPER-case phase names.

The *constituent description* (maximum 2000 characters) is a list of the species that enter a phase. The list starts with a colon (:), indicating the start of the sub-list of species for the first sublattice, and different sublattices are separated by colons. The complete sequence ends with a final colon.

Optionally, each sublattice may specify which species are considered to be major constituents. This is done by adding a percent sign (%) directly to the species name. The start values on the site fractions of the major constituents should sum to 0.99 on a specific sublattice. Thus, the minor constituents (i.e. those without a %) add up to 0.01. A maximum of 2000 characters can be coded in the constituent description, continuing in sequent lines. If the phase has a constituent description longer than 2000 characters, the rest can be coded in one or several ADD\_CONSTITUENT keywords.

---

## Examples

```
CONSTITUENT BCC_A2 :FE
CONSTITUENT IONIC-LIQ:Y :FE+2 : SB-3: !
CONSTITUENT M23C6 :CR FE :FE CR W MO : C: !
CONSTITUENT AQUEOUS:A :H2O% AG+1 AGF AGCL AGCL2-1 AGI3-2 AGSO4-1
AGC2H4+1
    AGN2H6+1 AGC2N2-1 AGC2H4NO2 AL+3 ALF3 ALO2-1, ... : !
CONSTITUENT SPINEL:I : AL+3% CR+3 FE+2% FE+3 MG+2% NI+2
    : AL+3% CA+2 CR+3 FE+2 FE+3 MG+2% NI+2 VA
    : FE+2 MG+2 VA%
    : N-3 O-2% :!
```

## ADD\_CONSTITUENT

---

```
ADD_CONSTITUENT [phase name]*24 [constituent description]*2000 !
```

This keyword adds more constituents to a phase that already has some constituents. Its syntax is the same as for the [CONSTITUENT](#) keyword. This keyword can be used several times, if the phase is very large, e.g. a gaseous mixture or a complex aqueous solution. This is useful when there are so many constituents in a phase that the 2000 characters available for the constituent description list is not enough.



Constituents are not necessary on all sublattices. In the second example below, no addition is made to the first sublattice.

### Examples

```
ADD_CONSTITUENT GAS :S1 S2 S3 ... : !
ADD_CONSTITUENT IM-PHASE : :CR:W ... : !
ADD_CONSTITUENT AQUEOUS:A :CUCL+1 CUCL2 CUCL2-1 CUCL3-2 CUOH+1 CUO2H2
CUO3H3-1
      CUO4H4-2 CU2OH+3 CU2O2H2+2 CU3O4H4+2 NIO2H2 NIO3H3-1 NIO4H4-2
NI2OH+3
      NI4O4H4+4 ZNOH+1 ZNO2H2 ZNO3H3-1 ZNO4H4-2 ... : !
```

---

## COMPOUND\_PHASE

---

```
COMPOUND_PHASE [phase name]*24 [data-type code]*8 [constituent] !
```

The keyword is a compact way to simultaneously define a species, a *compound phase* (maximum 24 characters) and its phase-constituent. It is useful for stoichiometric phases with constant compositions. The species name and stoichiometric formula must be identical, i.e. being the given *constituent*. The phase has this species as its only constituent. This keyword allows the database definition file for a large substance database to be more compact; it is a combination of the [SPECIES](#), [PHASE](#) and [CONSTITUENT](#) keywords.

### Examples

```
COMPOUND_PHASE AL2O3 % AL2O3 !  
COMPOUND_PHASE MAGNETITE %MF FE3O4 !  
COMPOUND_PHASE QUARTZ % SIO2 !
```

---

## ALLOTROPIC\_PHASE

---

```
ALLOTROPIC_PHASE [phase name]*24 [data-type code]*8 [constituent] !
```

This keyword does the same as the [COMPOUND\\_PHASE](#) keyword for entering an allotropic phase (maximum 24 characters), but does not enter the constituent as a species to the data structure. Use this if the species is already defined.

### **Examples**

```
ALLOTROPIC_PHASE BETHA-AL2O3 % AL2O3 !  
ALLOTROPIC_PHASE CRISTOBALITE % SiO2 !  
ALLOTROPIC_PHASE TRIDYMITTE % SiO2 !
```

---

## TEMPERATURE\_LIMITS

---

```
TEMPERATURE_LIMITS [lower limit] [upper limit] !
```

This keyword sets the default upper and lower temperature limits used by the GIBBS module for Gibbs energy parameters and functions. It can be used only once in one database definition file and all its sequential files.

### **Example**

```
TEMPERATURE_LIMITS 500.0 1800.0 !
```

## DEFINE\_SYSTEM\_DEFAULT

---

```
DEFINE_SYSTEM_DEFAULT [keyword] {G-ref. type index} !
```

This keyword sets the default value to ELEMENT or SPECIES in the DATA command [DEFINE\\_SYSTEM](#).

For a substance database, it can be appropriate to have ELEMENT as a default value whereas a large solution database can benefit from having SPECIES as a default value. A proper default value is useful for beginners. An advanced user is more likely to use the DATA commands DEFINE\_ELEMENT and DEFINE\_SPECIES to override the default value.

*{G-ref. type index}* is an integer indicating the reference state type for an element when entering and listing data in the GIBBS module. The following lists legal numbers and the corresponding meaning (the reference state type for an element):

<i>Number</i>	<i>Definition</i>
1	symbol: G
2	symbol: H298
3	symbol: H0

For example:

```
DEFINE_SYSTEM_DEFAULT element 2 !
```

---

## DEFAULT\_COMMAND

---

```
DEFAULT_COMMAND [secondary keyword and parameters] !
```

This keyword specifies commands to be executed by the DATA module at database initialization. The syntax of the available command is currently not the same as the user available DATA commands but the actions are similar. The available *secondary keyword and parameters* in syntax are:

```
DEFINE_SYSTEM_ELEMENT [element names]
DEFINE_SYSTEM_SPECIES [species names]
DEFINE_SYSTEM_CONSTITUENT [phase] [sublattice] [species]
REJECT_SYSTEM_ELEMENT [element names]
REJECT_SYSTEM_SPECIES [species names]
REJECT_SYSTEM_CONSTITUENT [phase] [sublattice] [species]
REJECT_PHASE [phase names]
RESTORE_PHASE [phase names]
```

### Examples

```
DEFAULT_COMMAND DEFINE_SYSTEM_ELEMENT FE VA !
DEFAULT_COMMAND REJECT_SYSTEM_CONSTITUENT LIQUID 2 C !
DEFAULT_COMMAND REJECT_PHASE LIQUID !
DEFAULT_COMMAND RESTOR_PHASE GAS !
```

## DATABASE\_INFORMATION

---

```
DATABASE_INFORMATION [text]*10000 !
```

This keyword defines a text for the detailed description of the current database. The text can be listed with the DATA command DATABASE\_INFORMATION. An apostrophe (') can be used in the text to indicate a new line; and two apostrophes, ( ' ' ) can be used in the text to indicate a new line plus an empty line.



The continuous *text* length (each line with max 78 characters) is 10,000 characters.

### Example

```
DATABASE_INFORMATION This is the XXX-Alloy Solution Database '  
    in the A-B-C-D-..... System. '  
    Developed by TCS, released in May 2001. ''  
... more ... !
```

---

## TYPE\_DEFINITION

---

```
TYPE_DEFINITION [data-type code]*1 [secondary keyword with  
parameters] !
```

This keyword couples phases to an action performed by the DATA module when the DATA command [GET\\_DATA](#) is executed.

### **Secondary Keywords**

The secondary keywords and associated parameters in syntax are:

```
SEQ [filename]  
RND# [filename]  
GES [valid GIBBS command with parameters]  
POLY3 [valid POLY command with parameters]  
TDB [valid DATA command with parameters]  
IF [conditional statement] THEN [keyword with parameters]  
AFTER [valid GIBBS command with parameters]
```

The secondary keyword `SEQ` specifies a sequential file that stores parameters belonging to the phases using the associated data type code (which is defined by this keyword). A special case where the filename is given as an asterisk (\*) implies that the database definition file also acts as a sequential data storage file. This case makes it possible to have a single file for a small database, which is especially suited for personal databases.

The secondary keyword `RND` should be concatenated with a positive integer # to indicate the type of the random file. Currently, there are these types of random files:

- `RND0`, the default, is used for complete Gibbs energy expressions (G0 parameters), where the search field is the unabbreviated parameter name.
- `RND1` is designated for functions, where the function name is used as the search field.
- `RND2` is reserved for binary interaction parameters, where its search field is also the unabbreviated parameter name without any interaction order notation.



Ternary and higher order interaction parameters must be specified on a sequential file. Moreover, the internal structures of these random files are subject to changes with different versions of DATA, and with implementations of DATA on various computer systems. For more information, see the FORTRAN program TDBSORT, which is available from Thermo-Calc Software AB.

The secondary keywords `GES`, `POLY3`, or `TDB` specifies a modification of, or addition to, phases having the associated data type code, such as magnetic contribution, another excess model, or any other valid GIBBS/POLY/DATA command that applies to a certain phase. By implementing this as a call to the interactive GIBBS/POLY/DATA module, flexibility is achieved. If a new type of addition is implemented in a GIBBS/POLY/DATA module, it can be immediately used in the database definition file without reprogramming the DATA module.



In several examples below, the use of the at (@) sign indicates any phase to which the relevant type definition (e.g. `A`, `B`, `4`, or `E`) applies.

The secondary keywords `IF` and `THEN` allow specification of a conditional statement structured with respect to the phase constitution that controls the execution of a following type-definition (keyword with parameters) string. See the last four examples.

The secondary keyword `AFTER` is similar to the `GES` keyword except the defined GIBBS command is executed after all parameters are entered. The reason for this is that the command has no effect unless there is a parameter. Following the `AFTER` keyword, a GIBBS command must be given and it is executed after entering the parameters of the phase.

The data-type code (always as one string) can be any normal or special character, e.g. `0`, `5`, `A`, `F`, `M`, `%`, `&`, and so forth, and is referred in the definition keywords [PHASE](#), [COMPOUND](#), [PHASE](#) and [ALLOTROPIC\\_PHASE](#) for various phases.

A phase can have several `ADDITIONAL` parts of different types (that are enforced by certain `TYPE_DEFINITIONS` which call the GIBBS command `AMEND_PHASE_DESCRIPTION` for describing various contributions to Gibbs energy). However, the DATA/GIBBS module automatically deletes any early-defined `ADDITIONAL` part(s) of the same type.

The DATA module can selectively retrieve functions that are necessary for a defined system from a database that has functions stored in its `setup` file or `SEQ` sequential function file, while all other functions irrelevant for the defined system are ignored and are not saved in associated GIBBS and POLY workspaces.

## Examples

```

TYPE_DEF % SEQ TCPATH\DATA\[DATABASE]\PARAMETERS.TDB !
TYPE_DEF I SEQ TCPATH\DATA\[DATABASE]\INTERACTION-PARAMS.TDB !
TYPE_DEF G RND0 TCPATH\DATA\[DATABASE]\GZERO-PARAMS.TDB !
TYPE_DEF F RND1 TCPATH\DATA\[DATABASE]\FUNCTIONS.TDB !
TYPE_DEF & RND2 TCPATH\DATA\[DATABASE]\BINARY-INTERACTIONS.TDB !
TYPE_DEF A GES AM_PH_DES @ MAGNETIC -1 0.40 !
TYPE_DEF B GES AM_PH_DES @ MAGNETIC -3 0.28 !
TYPE_DEF 4 GES AM_PH_DES @ EXCESS_MODEL REDLICH-KISTER_KOHLER !
TYPE_DEF 5 GES AM_PH_DES AQUEOUS EXCESS_MODEL HKF !
TYPE_DEF 6 GES AM_PH_DES AQUEOUS HKF_ELECTROSTATIC !
TYPE_DEF 7 GES AM_PH_DES AQUEOUS STATUS 02084000,,, !
TYPE_DEF 8 GES AM_PH_DES AQUEOUS MAJOR_CONST 1 H2O !
TYPE_DEF E AFTER AM_PH_DES LIQUID EXCESS MIXED-EXCESS
                A B LEGENDRE    C A POLYNOM ,,,, !
TYPE_DEF T AFTER AM_PH_DES LIQUID TERN-EXT TOOP-KOHLER B A C ,,,, !
TYPE_DEF Q AFTER AM_PH_DES LIQUID TERN-EXT KOHLER FE CR NI !
TYPE_DEF C IF (PD AND PT AND SN) THEN TDB RESTORE_PHASE BCT_A5 !
TYPE_DEF D IF (PD AND (PT OR SN)) THEN TDB REJECT_PHASE BCC_A2 !
TYPE_DEF E IF ((NB OR TI OR V) AND (C OR N)) THEN
                GES AM_PH_DES @ COMP_SET ,, CR NB TI V: C N: !
TYPE_DEF F IF (ALO3/2 OR CRO3/2 OR FEO OR MNO OR SIO2) THEN
                GES AM_PH_DES LIQUID COMP_SET ,, ALN%,ALO3/2%,CRO3/2%,
                FEO%,FEO3/2%,MNO3/2%,MNS%,SIO2%,TIO2% : !
TYPE_DEF R GES AM_PH_DES FE_LIQUID FRACTION_LIMITS Fe 0 0.6
                Ag 0 0.01 Al 0 0.05 Ca 0 0.05 Co 0 0.01 Cr 0 0.01
                Cu 0 0.02 Mg 0 0.05 Mn 0 0.05 Mo 0 0.05 Nb 0 0.05
                Ni 0 0.05 Pb 0 0.05 Si 0 0.10 Sn 0 0.02 Ti 0 0.05
                U 0 0.01 V 0 0.02 W 0 0.02 Zr 0 0.03
                B 0 0.01 C 0 0.01 H 0 0.01 N 0 0.01 O 0 0.01
                P 0 0.01 S 0 0.01 !

```

## Ordered Phase Restores Disorder Contribution

When an ordered phase with disordered contribution is restored, the disordered contribution is automatically restored as well. You can also restore the disordered phase *without* restoring the ordered phase.



However the DIS\_PART TYPE\_DEFINITION must always be set on the ordered phase. It is important that the disordered phase is declared *before* the ordered phase. Otherwise it causes an error when DIS\_PART TYPE\_DEFINITION is executed and then either the ordered or disordered phase is rejected.

### Example

```
TYPE_DEFINITION & GES A_P_D FCC_A1 MAGNETIC -3.0 2.80000E-01 !
PHASE FCC_A1 %& 2 1 1 !
    CONSTITUENT FCC_A1 :CR,NI% : C%,VA : !
$ THIS PHASE HAS A DISORDERED CONTRIBUTION FROM FCC_A1
TYPE_DEFINITION ' GES AMEND_PHASE_DESCRIPTION FCC_L12 DIS_PART FCC_
A1,,, !
TYPE_DEFINITION ( GES A_P_D FCC_L12 MAGNETIC -3.0 2.80000E-01 !
PHASE FCC_L12 %'( 3 .75 .25 1 !
    CONSTITUENT FCC_L12 :CR,NI% : CR,NI% : C,VA% : !
```

For example a GES command such as the following only affects phases defined above or at the phase with this TYPE\_DEFINITION in the TDB file:

```
TYPE_DEFINITION ( GES A_P_D FCC_L12 DIS_PART FCC_A1,,, !
```

For example a TDB command such as the following only affects phases defined *below* the phase with this TYPE\_DEFINITION in the TDB file:

```
TYPE_DEFINITION ( TDB RESTORE_PHASE FCC_A1 !
```

## FTP\_FILE

---

```
FTP_FILE [filename] !
```

FTP\_FILE is a special function random file and the function names correspond to the record numbers where these record names and the functions are stored. The FTP\_FILE decreases search time for the associated database in the DATA module. The file is used for large substance databases along with a SEQ sequential or RND0 random file for storage of GO parameters referring the functions named FxxxxT that are stored in the FTP file. The integer number xxxx is a search code used by DATA when such files are processed.



No modification of this file type is allowed.

### Example

```
FTP_FILE TCPATH\DATA\[DATABASE]\FTP-FILE.DATA !
```

## FUNCTION

---

```

FUNCTION [function name]*8 [lowest temp. limit]
[expression 2]; [upper temp. limit 2] Y
[expression 1]; [upper temp. limit 1] Y
[expression 3]; [upper temp. limit 2] Y
..... ; ..... Y
[expression n-1]; [upper temp. limit n-1] Y
[expression n]; [upper temp. limit n] N {Ref. Index} !

```

GIBBS can use predefined functions in the expression (TP-Function) of a Gibbs energy parameter or in other functions. This is often used when several parameters (or functions) have a common sub-expression, such as for metastable modifications of elements. This keyword can appear in both files for database definition and sequential storage, but not in FTP files. A valid *function name* can have up to 8 characters.

A function always starts with a lowest temperature limit of its applicability, followed by one or more (up to 10) expressions (TP-Functions) that are coded as mathematical relations of constants, functions of stable variables (T and P) and other entered functions (normally with a # suffix, e.g. +3\*GHSERAL#).



### ENTER PARAMETER

The expression is a FORTRAN-like expression and operators +, -, \*, = and \*\* can be used (\*\* only with integer powers). Unary-functions LN or LOG (both for natural logarithm) and EXP (for exponential) can also be used. There is also the GEIN function available for the Einstein Model.

$$\text{GEIN}(\text{THETA}) = 1.5 * \text{R} * \text{THETA} + 3 * \text{R} * \text{T} * \text{LN}(1 - \text{EXP}(-\text{THETA}/\text{T}))$$


### General Einstein Model

Each expression (TP-Function) should end with a semicolon (;), and be followed by its upper applicable temperature limit and a continuation indicator (Y to continue with the next expression or N to end the function's expression). If there is no continuation after a specific expression (TP-Function), the reference index can optionally be given after the N indicator.

A complete/valid function entry can be written in several continuation lines if the function's expression (TP-Function) is too long or if there is more than one applicable expression (TP-Function), as the maximum length of each line is 78 characters.



It is recommended to always have at least one empty space at the beginning of each continuation line. Otherwise, the DATA module may misunderstand the expression or issue some error messages when reading the function entry. Avoid entering functions like the one below.

```
FUNCTION GHSERXY 298.15
-1000+1058*T-38.9*T*LOG(T)+GFUNXY#; 6000 N !
```

Such a function is read by the DATA module as

```
1000+1058*T-38.9*T*LOG(T)+GFUNXY#
```

rather than

```
-1000+1058*T-38.9*T*LOG(T)+GFUNXY#
```

This is because the DATA module concatenates all lines and removes extra spaces before trying to enter the function in the GIBBS workspace. Thus, the - sign is taken as a delimiter between 298.15 and 1000, and the function incorrectly becomes:

```
FUNCTION GHSERXY 298.15 1000+1058*T-38.9*T*LOG(T)+GFUNXY#; 6000 N !
```



Avoid this mistake by giving at least one empty space as the first character of a new line, such as

```
FUNCTION GHSERXY 298.15
-1000+1058*T-38.9*T*LOG(T)+GFUNXY#; 6000 N !
```

which is read correctly as

```
FUNCTION GHSERXY 298.15 -1000+1058*T-38.9*T*LOG(T)+GFUNXY#; 6000 N !
```

The lowest-temperature limit (in Kelvin) for the applicability of the (first) TP-Function in a function is normally set by default as 298.15 K, in most cases. However, you can set another limit when it is applicable (according to experimental data and assessments).

An upper-temperature limit (in Kelvin; followed by a Y or N sign) for the applicability of each TP-Function in a function must be given after the semicolon (;) immediately following the specific TP-Function. The highest-temperature limit (in Kelvin) for the applicability of the current function is always followed by the N sign. If a negative number is given as the

lowest-temperature limit, it assumes there are breakpoints in pressure for this function. In these cases, it is interpreted as the lowest-pressure limit (in Pascal), and the other limits in the current function is also taken as pressure limit values (in Pascal).

The temperature/pressure limits for the functions are checked during calculations. An indicator is set if the actual temperature/pressure condition is below the lowest temperature/pressure limit or above the highest temperature/pressure limit. In these cases, an extrapolation is done using the TP-Function valid in the nearest temperature/pressure range.

The optional reference index *{Ref. Index}* is an integer number indicating where to find the particular function in a special reference file. The references are listed when doing the GET\_DATA command in the DATA module. They can also be listed in the GIBBS module with the command LIST\_DATA with the option R.



For accounting for the reference indices, also see the keyword [REFERENCE\\_FILE](#).

The reference index field can also be an abbreviation (such as REF:250, REF\_002, or REF-SGTE) which denotes the original reference. In this case, the reference cannot be obtained when issuing the DATA command GET\_DATA or the GIBBS command LIST\_DATA (with the option R).

However, the references directly coded in the database definition file (\*\*setup.TDB) starting with a letter can be shown when issuing the DATA command GET\_DATA or the GIBBS command LIST\_DATA (with the option N or R). Normally, such references must be located after the LIST\_OF\_REFERENCE keyword. It is recommended to use reference code names such as REF001, REF018, etc. The reference list, which is generated by the GIBBS command [LIST\\_DATA](#) <file> with the N or R option, is also possible to be directly read by the DATA module.

The DATA module can selectively retrieve functions which are necessary for a defined system from a database that has functions stored in its setup file or SEQ sequential function file, while all other functions irrelevant for the defined system are ignored and are not saved in associated GIBBS and POLY workspaces. Previously, this can only be done for large databases that have functions stored in RND1 random or FTP function files.

## Examples

```
FUNCTION GFREE 298.15 1000+GFUNXY#; 6000 N !
FUNCTION GFUNXY 298.15 -1000+200*T+30*T*LOG(T); 6000 N 505 !
FUNCTION G0_CAO 298.15 -663538.11+352.67749*T-57.7533*T*LN(T)
+5.3895E-03*T**2-8.879385E-07*T**3+575530*T**(-1);
```

---

```
1400.00 Y -625196.99+78.896993*T-20.40145*T*LN(T)
        -1.112923E-02*T**2+5.1896733E-07*T**3-6917350*T**(-1);
2900.00 Y -499226.55-490.37695*T+51.95912*T*LN(T)
        -2.961051E-02*T**2+1.4033905E-06*T**3-48114685*T**(-1);
3172.00 Y -587711.89+375.04117-62.76*T*LN(T);
6000.00 N REF020 !
```

## PARAMETER

```

PARAMETER <parameter name> [lowest temp. limit]
[expression 1]; [upper temp. limit 1] Y
[expression 2]; [upper temp. limit 2] Y
[expression 3]; [upper temp. limit 2] Y
..... ; ..... Y
[expression n-1]; [upper temp. limit n-1] Y
[expression n]; [upper temp. limit n] N {Ref. Index} !

```

This keyword can appear in both files for database definition and sequential storage, but not in FTP files. After the keyword, a valid *<parameter name>* in some cases should be given.

It is used to define standard Gibbs energies (i.e. the G parameters for Gibbs energy of formations) of all valid end-members of various stoichiometric and solution phases, and excess Gibbs energies (i.e. the L parameters for Gibbs energy of interactions) of all binary, ternary, quaternary or higher-order interactions in various solution phases; both standard Gibbs energies and excess energies can also have parameters for contributions from PT-dependent volume variations (i.e. the V0, VA, VC and VK parameters for molar volume, thermal expansivity, bulk modulus, isothermal compressibility and high-pressure fitting parameter), magnetic ordering (i.e. the TC and BM parameters for Curie temperature and Bohr magneton number) and hypothetical electrostatic interactions (i.e. BM parameter for Born functions  $\omega_{Pr,Tr}$  of aqueous solute species).

The general form of a parameter is:

```

<identifier>(<phase>, <constituent array>; <digit>) <xxx>
<expression> <yyy> <keyword Y or N> <zzz> !

```

### General Form

Name	Description
identifier	The parameter type.
phase	The phase name (maximum 24 characters).
constituent array	The specific constituent array in the phase.

Name	Description
digit	The degree of composition-dependent interaction contribution (an integer number from 0 to 9), that is only for excess energy (L), Curie temperature (TC) and Bohr magneton number (BMAGN), as well as for volume-related parameters (V0 or VA or VB or VC or VK); if it is valued as zero, or if it is for the standard Gibbs energy (G) for which the degree is always zero, it can be omitted.
expression	The mathematical relation to describe the parameter.
xxx and yyy	The low and high temperature limits respectively for the applicable temperature range of the parameter expression
keyword Y or N	The indicator on if there is continuation for the parameter expression or not
zzz	The reference index/number for the assessment of this parameter;
!	The exclamation mark is used to indicate that the current parameter definition is ended

## GIBBS Parameter Name

The GIBBS parameter name has a general form of:

`<identifier>(<phase>,<constituent array>;<digit>)`

### Type-Identifier and Definition

The GIBBS parameter name consists of several parts. The first is a *type-identifier*. The following type-identifiers are legal:

Type-Identifier	Definition
G	Standard energy parameter (Gibbs energy of formation)
L	Excess energy parameter (Gibbs energy of interaction)
TC	Curie temperature for magnetic ordering
BMAGN or BM	Bohr magneton number for magnetic ordering (or Born function $\omega$ Pr,Tr for aqueous solute species).
LNTHETA1, LNTHETA2, LNTHETA3, LNTHETA4, and LNTHETA5	$\ln(\Theta)$ where $\Theta$ is the Einstein temperature. Used by the general Einstein model and together with a corresponding THETAf parameter with the same number. See <a href="#">General Einstein Model</a> in the <i>Database Optimization User Guide</i> .

Type-Identifier	Definition
THETA1, THETA2, THETA3, THETA4, and THETA5	A weight function used together with a corresponding LNTHETA parameter with the same number. Used by the general Einstein model. See <a href="#">General Einstein Model</a> in the <i>Database Optimization User Guide</i> .
V0	Molar volume at 298.15 K and 1 bar (a numeric value only)
VA	Integrated thermal expansivity $\int_{298.15}^T \alpha(T) dT$
VC	High-pressure fitting parameter
VK	Isothermal compressibility
VISC	$RT \cdot \ln(\eta)$ where $\eta$ is the dynamic viscosity
ELRS	Electric resistivity
THCD	Thermal conductivity
SIGM	Surface tension of a liquid end-member
XI	Surface tension dampening factor/interaction for one constituent in a two-constituent combination. See the examples for how to use these parameters

## Examples

Parameter	Definition
G (GAS, C1O2)	The Gibbs energy of formation of a CO2 molecule in gas.
G (FCC, Fe:VA)	The Gibbs energy of formation of fcc Fe with interstitials.
L (LIQ, Fe, Cr; 0)	The regular solution parameter for Fe and Cr in liquid.
L (LIQ, Fe, Cr; 1)	The sub-regular solution parameter.
TC (BCC, Fe:Va)	The Curie temperature of bcc Fe.
BMAGN (BCC, Fe:Va)	The Bohr magneton number parameter of bcc Fe.
SIGM (LIQUID, AL; 0)	Surface tension of Al in LIQUID. Constant value or linear temperature dependence.

Parameter	Definition
XI (LIQUID,AL,CU;0)	Surface tension dampening factor for Al in the Al-Cu combination in LIQUID. Constant value.
XI (LIQUID,AL,CU;1)	Surface tension dampening factor for Cu in the Al-Cu combination in LIQUID. Constant value.

You can also use G for interaction parameters; and on output list (performed by the GIBBS command LIST\_PARAMETER or LIST\_PHASE\_DATA) the type-identifier L is always used for interaction parameters. Note that the type-identifier BM is also used for Born functions  $\omega_{Pr,Tr}$  of aqueous solute species.

The identifier must be followed by an opening parenthesis, a phase name, a comma and a constituent array. Optionally, the constituent array can be followed by a semicolon and a digit. The parameter name is terminated by a closing parenthesis.



It is important that if a phase bears a legal phase-type (among G, A, Y, L, I, F and B) in its phase definition (already by the PHASE keyword; such as GAS:G, LIQUID:L, SLAG:L, IONIC\_LIQ:Y, SPINEL:I, FCC\_L12:F, HCP\_D021:F, BCC\_B2:B, AQUEOUS:A), such a valid phase-type code should not be attached to the phase name in the PARAMETER keyword.



Specifying the phase name in UPPER-case is recommended. You can define a phase name as a mixture of UPPER-case and lower-case letters in a database, but the DATA module automatically converts all lower-case to UPPER-case because the GIBBS module only recognizes UPPER-case phase names.

The *constituent array* consists of a list of constituent names. Interaction parameters have two or more constituents from the same sublattice separated by a comma. If the phase has sublattices, at least one constituent in each sublattice must be specified. The exception is if the phase is ionic liquid (specified with the "phase bit" :Y), then neutral endmember and interaction parameters are written as if the first sublattice does not exist, e.g. PARAM G (IONIC\_LIQ,FeO3/2;0), i.e. the constituent array does not contain any constituents from the first (cation) sublattice.



For more ionic liquid specific parameters, see [Parameters for the Two Sublattice Ionic Liquid Model](#).

The constituents in different sublattices must be given in sublattice order and are separated by a colon.

After the component array, a sub-index digit can be specified after a semicolon. This digit must be in the range 0 to 9. The interpretation of the sub-index depends on the excess energy model used for the phase. If no semicolon and digit are given, the sub-index value is assumed to be as zero.

The excess energy parameters, e.g. the regular/subregular (binary) parameter or ternary parameters, are multiplied with two or more fractions of the constituents from the same sublattice of the solution phase. These additional constituents must be given as interacting constituents.



Be careful about the sign of odd terms, for example,  $L(\text{BCC}, \text{B}, \text{A:VA};1)$  is treated as  $L(\text{BCC}, \text{A}, \text{B:VA};1)$ , i.e. it is always put into alphabetical order.



Solution phases with sublattices may have interacting constituents in each sublattice.

You can use an asterisk (\*) to denote that the excess interaction parameter is independent of the constituents of a specific sublattice. For example,  $L(\text{FCC\_L12}, \text{AL}, \text{NI};*)$  means that the interaction parameter is for the binary interaction between constituents AL and NI on the first sublattice in the FCC\_L12 solution phase, while it is independent of all constituents on the second sublattice. A interaction parameter in the list of constituents is always added to the Gibbs energy and the asterisk (\*) is calculated with the term of  $[1-\sum y(\text{specified constituents})]$ , which implies that in an A-B binary system the following three L parameters are identical (but in higher-order systems, they are different):

- $L(\text{phase}, \text{A}, \text{B})$  is multiplied with  $X(\text{A}) * X(\text{B})$
- $L(\text{phase}, \text{A}, *)$  is multiplied with  $X(\text{A}) * (1-X(\text{A}))$
- $L(\text{phase}, \text{B}, *)$  is multiplied with  $X(\text{B}) * (1-X(\text{B}))$

A parameter always starts with a lowest temperature limit of its applicability, followed by one or more (up to 10) expressions (TP-Functions) coded as mathematical relations of constants, functions of stable variables (T and P) and entered functions (normally with a # suffix, e.g. +3\*GSERAL#).

The expression is a FORTRAN-like expression and operators +, -, \*, = and \*\* can be used (\*\* only with integer powers). Unary-functions LN or LOG (both for natural logarithm) and EXP (for exponential) can also be used.

There is also the Einstein function GEIN available:

$$\text{GEIN}(\text{THETA}) = 1.5 * R * \text{THETA} + 3 * R * T * \text{LN}(1 - \text{EXP}(-\text{THETA}/T))$$



### General Einstein Model

Each expression (TP-Function) should ends with a semicolon (;) and be followed by its upper applicable temperature limit and a continuation indicator (Y to continue with the next expression, or N to end the parameter's expression). If there is no continuation after a specific expression (TP-Function), the reference index can be optionally given after the N indicator.

A complete/valid parameter entry can be written in several continuation lines if the parameter's expression (TP-Function) is too long or if there is more than one applicable expression (TP-Function), as the maximum length of each line is 78 characters.



It is recommended to always have at least one empty space at the beginning of each continuation line. Avoid entering parameters such as:

```
PARAMETR G(LIQUID,A,B) 298.15
-2000+4568*T+2*GFUNAB#; 6000 N !
```

Such a parameter is read by the DATA module as 2000+4568\*T+2\*GFUNAB#, rather than as -2000 +4568\*T+2\*GFUNAB#.



Avoid this mistake by giving at least one empty space as the first character of a new line, such as

```
PARAMETR G(LIQUID,A,B) 298.15
-2000+4568*T+2*GFUNAB#; 6000 N !
```

The lowest-temperature limit (in Kelvin) for the applicability of the (first) TP-Function in a parameter is normally set by default as 298.15 K, in most cases; however, you can set another limit when it is applicable (according to experimental data and assessments). An upper-temperature limit (in Kelvin; followed by a Y or N sign) for the applicability of each TP-Function in a parameter must be given after the semicolon (;) immediately following the specific TP-Function; and the highest-temperature limit (in Kelvin) for the applicability of the

current parameter is always followed by the N sign. If a negative number is given as the lowest-temperature limit, it is assumed that there are breakpoints in pressure for this parameter. In such cases, it is interpreted as the lowest-pressure limit (in Pascal), and the other limits in the current parameter are also taken as pressure limit values (in Pascal).

The temperature/pressure limits for the parameters are checked during calculations. An indicator is set if the actual temperature/pressure condition is below the lowest temperature/pressure limit or above the highest temperature/pressure limit. In these cases, an extrapolation is done using the TP-Function valid in the nearest temperature/pressure range.

The optional reference index *{Ref. Index}* is an integer number indicating where to find the particular parameter in a special reference file. The references are listed when doing the GET\_DATA command in the DATA module. These can also be listed in the GIBBS module with the command LIST\_DATA and the option R or N.



For accounting the reference indices, also see the keyword [REFERENCE\\_FILE](#).

The reference index field can also be an abbreviation (such as REF:250, REF\_002, or REF-SGTE) denoting the original reference. In this case, the reference cannot be obtained when issuing the DATA command GET\_DATA or the GIBBS command LIST\_DATA (with the option R or N).

However, the references directly coded in the database definition file (\*\*setup.TDB) that starts with a letter can be shown when issuing the DATA command GET\_DATA or the GIBBS command LIST\_DATA (with the option R or N). Normally, such references must be located after the LIST\_OF\_REFERENCE keyword. It is recommended to use reference code names such as REF001, REF018, etc. The reference list, which is generated by the GIBBS command LIST\_DATA <file> with the N option, is thus also possible to be directly read by the DATA module.

## Examples

```
PARAMETER G(BCC,FE:VA) 298.15 1000+200*T+...; 6000 N 91DIN !
PARAMETER TC(BCC,FE:VA) 298.15 +1043; 6000 N 91DIN !
PARAMETER BMAGN(BCC,FE:VA) 298.15 +2.22; 6000 N 91DIN !
PARAMETER G(SIGMA,FE:CR:CR;0) 298.15 1000+200*T+...; 6000 N 101 !
PARAMETER G(LIQUID,AL;0) 298.15 +11005.553-11.840873*T
+7.9401E-20*T**7+GHSERAL#;
933.60 Y +10481.974-11.252014*T+1.234264E+28*T**(-9)+GHSERAL#;
2900.00 N REF:283 !
```

```

PARAMETER G(BCC_A2,PB:C) 298.15 UN_ASS#; 300 N REF:0 !
PARAMETER G(BCC_A2,NI:C;0) 298.15 +GHSERNI#+3*GHSERCC#
                                +400000-100*T; 6000 N REF071 !
PARAMETER G(BCC_A2,MN:VA) 298.15 +GMNBCC#; 6000 N REF285 !
PARAMETER G(PHASE,A:B) 298.15 +3*GEIN(800); 6000 N 101 !
PARAMETER BM(AQUEOUS,OH-1) 298.15 +Z0002PW0#; 1600 N 155 !
PARAMETER L(BCC,FE,CO:VA;0) 298.15 1000+200*T+...; 6000 N !
PARAMETER L(BCC,FE,CO:VA;1) 298.15 1000+200*T+...; 6000 N !
PARAMETER L(BCC,FE,CO:VA;2) 298.15 1000+200*T+...; 6000 N !
PARAM TC(BCC_A2,CO,MO:VA;0) 298.15 -3700; 6000 N R454 !
PARAM TC(BCC_A2,CO,MO:VA;1) 298.15 +2300; 6000 N R454 !
PARAM BMAGN(BCC_A2,CO,MO:VA;0) 298.15 -3.445; 6000 N R454 !
PARAM V0(BCC_A2,CR,FE:VA;0) 298.15 +ZERO#; 6000 N REF06V !
PARAM V0(BCC_A2,CR,FE:VA;1) 298.15 -1.10524097E-7; 6000 N REF06V !
PARAM V0(BCC_A2,CR,FE:VA;2) 298.15 +1.40024130E-7; 6000 N REF06V !
PARAM VA(BCC_A2,CR,FE:VA;0) 298.15 -6.49444634E-6*DELTAT#; 6000 N
REF06V !
PARAM VA(BCC_A2,CR,FE:VA;1) 298.15 +2.91269321E-5*DELTAT#; 6000 N
REF06V !

```

## Parameters for the Two Sublattice Ionic Liquid Model

For the two sublattice ionic liquid model (specified with the "phase bit" :Y in the phase declaration), there are some additional requirements on the allowed parameters. While in the regular case all combinations of a phase's constituents are allowed as end-members, there are some constraints on the allowed parameters for ionic liquid.

In the ionic liquid model, the vacancy has been given an "induced" charge  $-Q$ , where  $Q$  is the average charge of the first (cation) sublattice.

All allowed parameters are listed below. The following notations are used:

- C= Cation
- C1= Cation 1 if more than one. Similar for anions and neutrals.
- A= Anion
- VA= Vacancy
- N= Neutral
- D= Anion or vacancy or neutral (i.e. anything on the second sublattice)

The phase description is:

$$(C_1, C_2 \dots C_i) P(A_1, A_2 \dots A_i, VA, N_1, N_2 \dots N_i) Q$$

In the ionic liquid model, the Vacancy has been given an “induced” charge  $-Q$ , where  $Q$  is the average charge of the first (cation) sublattice.

Allowed end-members, assuming the phase name is IONIC\_LIQ:

$$G(\text{IONIC\_LIQ}, C:A; 0)$$

$$G(\text{IONIC\_LIQ}, C:VA; 0)$$

$$G(\text{IONIC\_LIQ}, N; 0)$$

(The last line is an exception to the case that the constituent array must contain constituents from all sublattices)

This leaves one not allowed end-member:

$$G(\text{IONIC\_LIQ}, C:N; 0)$$

Allowed binary interaction parameters, where  $i$  is the interaction order in a Redlich-Kister expansion.

$$G(\text{IONIC\_LIQ}, C_1, C_2:A; i)$$

$$G(\text{IONIC\_LIQ}, C_1, C_2:VA; i)$$

$$G(\text{IONIC\_LIQ}, C:VA, N; i)$$

$$G(\text{IONIC\_LIQ}, N_1, N_2; i)$$

$$G(\text{IONIC\_LIQ}, C:A_1, A_2; i)$$

$$G(\text{IONIC\_LIQ}, C:A, VA; i)$$

$$G(\text{IONIC\_LIQ}, C:A, N; i)$$

No other binary interactions are allowed.

Allowed ternary interaction parameters. The interaction is composition independent if there is one parameter with  $i = 0$ . It is composition dependent if there are three parameters with  $i = 0, 1$  and  $2$ . There can only be one or three parameters for one combination of constituents

$$G(\text{IONIC\_LIQ}, C_1, C_2, C_3:A; i)$$

$$G(\text{IONIC\_LIQ}, C_1, C_2, C_3:VA; i)$$

$$G(\text{IONIC\_LIQ}, N_1, N_2, N_3; i)$$

$$G(\text{IONIC\_LIQ}, C_1, C_2:VA, N; i)$$

$$G(\text{IONIC\_LIQ}, C:VA, N_1, N_2; i)$$

$$G(\text{IONIC\_LIQ}, C:A, D_1, D_2; i)$$

$$G(\text{IONIC\_LIQ}, C_1, C_2:A, D; i)$$

No other ternary interactions are allowed.

---

## OPTIONS

---

```
OPTIONS /[alloy name]([composition limitations for all alloying
elements]) !
```

This keyword defines an alloy in a database. An alloy has a name, a major component and a number of alloying elements. The purpose for defining an alloy is to be able to tell you about applicable composition limits of the current database in applications to that particular type of alloy. It is possible to have several alloys in the same database. The alloys are given after the keyword in the database.

The alloy name must be preceded by a slash (/) and terminated by the opening parenthesis, with no spaces are allowed in between.

- The alloy name is maximum 8 characters.
- After the parenthesis, follows the major element and a parenthesis with its minimum mass and minimum mole percent given inside.
- Then, the alloying element names, each with its maximum mass and mole percent are given within parenthesis.
- There must a space between definitions for each alloying element.
- The alloy definition is terminated by a closing parenthesis, and the whole OPTIONS keyword by the exclamation mark (!).

### **Example**

```
OPTIONS /SSteel (Fe (60,60) CR (30,30) NI (15,15) SI (1,1) N (.1,1)) !
```

## TABLE

---



It is recommended to always have at least one empty space at the beginning of each continuation line. Otherwise, the DATA module may issue some error messages when reading the table entry.

```
TABLE [name]*8 [start temp] [end temp] [delta temp] [table values] !
```

This keyword can appear in both files for database definition and sequential storage, but not in FTP files. It makes a table of Gibbs energy as a function of temperature where the values are given from the start temperature to the end temperature, at a step of the delta temperature.

### **Example**

```
TABLE DEMENTAB 1000.0 1500.0 100.0 -2912.9008 -2834.2416 -2755.5824  
-2677.7600 -2600.7744 -2524.2072 !
```

## ASSESSED\_SYSTEMS

```
ASSESSED_SYSTEM [descriptions on special treatments for specific
assessed systems]*8000 !
```

This keyword can be included in the database definition file (the `***setup.TDB` file). A maximum of 8000 characters after the keyword (to describe some special options when the DATA, GIBBS and POLY models deal with the existing systems with assessed data) is allowed until the exclamation mark (!).

Several ASSESSED\_SYSTEMS keywords can be used in the same TDB file in order to have more lists of assessed systems. There is no limit to the number of the entries in the same TDB file.

The assessed systems in the database, and the special treatment options, are typed after the keyword. The elements (always in UPPER CASE) in each assessed system must be in alphabetical order and be separated by a hyphen, such as `C-FE` for the Fe-C binary system, and `C-CR-FE` for the Fe-Cr-C ternary system. A space must be between each assessed system. Information on assessed binary, ternary or higher-order systems may also be given in this way.



A ternary system like C-CR-FE does not imply that the binary C-CR, C-FE and CR-FE are assessed. There is no way to indicate partially assessed systems.

- There is a field to give some descriptive information for a specific system, with various options about how to:
- Reject or restore phase(s) from the current database in the DATA module;
- Set major constituent(s) in the first composition set and to set a second composition set for a specific phase available in the current database in the GIBBS module; *and*
- Calculate this specific system in the POLY module.
- This is the facility used by the BIN (binary phase diagrams) and TERN (ternary phase diagrams) modules in the Thermo-Calc software/database package.

The descriptive information must immediately follow the specific system name, and must be enclosed within parenthesis ( and ), and the left parenthesis must follow directly after the system, such as :

```
AL-NI (TDB +L12 ;G5 C-S:L12/NI:AL:VA ;P3 STP:.8/1200/1 STP:.2/600/1)
```

The syntax TDB means that the commands to the DATA module proceed, and +L12 in the example means that the phase called L12 should be restored (it has been rejected by default).

The directive ;G5 means that the following are commands to the GIBBS module. C\_S: means creating a second composition set, after the colon follows the phase name and after the slash the major constituents.

After the directive ;P3 follows commands to the POLY module. STP: means setting a start point with the value of the X-axis first (composition for the second element in a binary system), the slash separates the Y-axis value (temperature), and possibly one or more directions (-1, 1, -2 or 2).

A summary of the allowed syntax is:

TDB accepts

- +phase and -phase for restore/reject.
- ;G5 accepts
- MAJ:phase/constituent-array for major constituents of the first composition set.
- C\_S:phase/constituent-array for a second composition set.
- ;P3 accepts
- TMM:l<sub>t</sub>/h<sub>t</sub> for the low-/high-temperature limits (l<sub>t</sub> and h<sub>t</sub>; for instance TMM:500/4000) suitable for calculating phase diagrams and property diagrams of a binary system.
- \* for a default start point which is set as:
- for a binary system: at the composition  $X(2^{\text{nd}} \text{ element}) = .1234$ , temperature  $T = 1100$  K and with the default directions; or
- for a ternary system: at the compositions  $X(2^{\text{nd}} \text{ element}) = .1234$  and  $X(3^{\text{rd}} \text{ element}) = .1234$  and with the default directions.
- STP:x/t/d1/d2/d3 for a specific start point in a binary system which is set as at the composition  $X(2^{\text{nd}} \text{ element}) = x$  and temperature  $T = t$  (in K), and with the directions d1, d2 and/or d3.
- STP:x1/x2/d1/d2/d3 for a specific start point in a ternary system which is set as at the compositions  $X(2^{\text{nd}} \text{ element}) = x1$  and  $X(3^{\text{rd}} \text{ element}) = x2$  and with the directions d1, d2 and/or d3.
- The direction(s) can be defined as -1, 1, -2 or 2. If no direction is specified, all default directions are used (meaning no ADD command is enforced in the POLY module).

- If only one start point is specified, the direction(s) may be omitted; if more than one start points are specified, at least one direction for each start point must be given for all start points.

Each entry for a specific binary or ternary sub-system can be written in one or several lines (each line with 78 characters).

### Other Examples

```

ASSESSED_SYSTEMS
AL-NI (TDB +L12 +BCC_B2 ;G5 C_S:L12/NI:AL:VA
      ;P3 STP:.8/1000/1 STP:.45/700/1 STP:.7/700/1)
AL-PB (TDB -HCP -BCC
      ;G5 MAJ:LIQ/AL MAJ:FCC/AL:VA C-S:LIQ/PB C-S:FCC/PB:VA ;P3 *)
CR-FE (;G5 C-S:BCC/CR:VA ;P3 STP:.6/1200/1/-2/2)
AG-CU (;G5 MAJ:FCC/AG:VA C_S:FCC/CU:VA ;P3 STP:.3/1000)
C-NB (;P3 STP:.9/1100/1)
C-SI (;P3 *)
CO-CR (;G5 MAJ:FCC/CO:VA C_S:FCC/CR:VA ;P3 STP:.1/1100)
CR-FE (TDB -HCP ;G5 C_S:BCC/CR:VA ;P3 STP:.6/1200/1/-2/2)
CR-NI (;P3 *)
CR-W (;G5 MAJ:BCC/W:VA C_S:BCC/CR:VA
      ;P3 TMM:500/4000 STP:.3/700/1 STP:.3/1800/2 )
CU-FE (TDB -HCP ;G5 MAJ:LIQ/CU MAJ:FCC/FE:VA C_S:FCC/CU:VA
      ;P3 STP:.9/1400)
FE-N (TDB +FE4N ;P3 *)
FE-O (TDB -LIQUID +IONIC ;G5 C_S:ION_LIQ/FE+2:O-2 MAJ:ION_LIQ/FE+2:VA
      ;P3 STP:.2/2000/1 )
FE-S (TDB -LIQUID +IONIC ;G5 C_S:ION_LIQ/FE+2:S MAJ:ION_LIQ/FE+2:S-2
      ;P3 *)
AL-MG-SI (;P3 *)
C-CR-FE ;G5 MAJ:BCC/FE:VA C_S:BCC/CR:VA ;P3 *) !

```



The semicolon (;) is a part of the ;G5 and ;P3 directives. A long descriptive information can be written in more than one line, such as for the AL-NI, AL-PB , CU-FE and FE-O systems shown above.

The directive ;P3 \* is needed if the default start point should be used. If there is no P3 directive, the BIN or TERN module generates some 20 different start points in order to cover all possible compositions and temperatures (for a binary system) or all possible compositions (for a ternary system under any specific temperature).

## REFERENCE\_FILE

---

REFERENCE\_FILE [file name] !

This keyword takes a reference file name as its argument. This reference file (that must be edited in a restrictive way and be saved as a blocked file with a fixed line-length for each line in the file, as described below) contains a complete list of the references for the various parameters (and sometimes functions) in the database. The file must have a fixed record structure: each reference entry with one or several records, and each record with exactly 78 characters written in one single blocked line; and no empty space is allowed at the beginning of all lines in the file. If there are more than one record entered for a reference entry, all the continuation lines must start with an ampersand (&). The line number of the first record for a specific reference entry is then accounted as the unique integer for that specific reference, which is referred when a parameter or function calls this integer as the optional *{Ref. Index}*.



Keywords [PARAMETER](#) or [FUNCTION](#) have information about specifying a reference index.

### Example

```

/-1<G>                T.C.R.A.S. Class 1
AG1.64TE1             THERMODATA 01/93
&28/01/93
&SILVER 1.64-TELLURIDE. Solid Standard State.
AG1                   HULTGREN SELECTED VAL.          SGTE **
&AT.WEIGHT 107.870,STANDARD STATE:CODATA KEY VALUE.MPT=1234.93K. &--
U.D. 30/10/85.
AG1<G>                T.C.R.A.S Class: 1
AG1/+1<G>             T.C.R.A.S Class: 1
AG1BR1               N.P.L.                          SGTE **
&Tfusion uncertain and heat vaporization estimated.
AG1BR1<G>             THERMODATA 01/93
&28/01/93
&Gaseous Standard State.
AG1BR103             BARIN & KNACKE.SUPPL.REF:62,* SGTE **
&AGO3BR SILVER OXYTRIBROMIDE

```

In the example, the unique integers of related references (for assessed elements, species, phases, interactions, etc.) are:

1/-1<G>

2AG1.64TE1

5 AG1

8AG1<G>

9AG1/+1<G>

10AG1BR1

12AG1BR1<G>

15AG1BR1O3

## LIST\_OF\_REFERENCE

---

```
LIST_OF_REFERENCE
NUMBER SOURCE
  [REFxxx] '[Detailed reference]'
.....
..... !
```

This keyword starts a reference list that is directly coded in the database definition file (`***setup.TDB`). Its argument begins on the following line, and normally has an explanation line (NUMBER SOURCE) that is followed by various reference codes. Each reference code may occupy one or more lines (each line with maximum 78 characters), but must have a reference code name (that starts with a letter) and the detailed reference information (that is written within two single-quotation marks, `' '`). It is recommended to use reference code names such as `REF001`, `REF018`, etc. A maximum of 400,000 characters after the keyword is allowed until the exclamation mark (!).

Such a reference list can be shown when issuing the DATA command [GET\\_DATA](#) or the GIBBS command [LIST\\_DATA](#) (with the `R` option). The reference list, which is generated by the GIBBS command `LIST_DATA <file>` with the `N` option, has this structure, and is thus possible to be directly read by the DATA module.

### Example

```
LIST_OF_REFERENCES
NUMBER SOURCE
REF283 'Alan Dinsdale, SGTE Data for Pure Elements,
      Calphad Vol 15(1991) p 317-425,
      also in NPL Report DMA(A)195 Rev. August 1990'
REF224 'P-Y Chevalier, Thermochemica Acta, 130 (1988) p 33-41; AG-SI'
!
```

## ADD\_REFERENCE

---

```
ADD_REFERENCE
  [REFxxx] '[Detailed reference]'
  .....
  ..... !
```

Use this keyword to have an unlimited number of references. The use of this is the same as that of [LIST\\_OF\\_REFERENCE](#), and is used after it. Several `ADD_REFERENCE` sections can be used in a single database if there are too many references.

It is recommended to always start with a so-called *empty reference* (such as the `DUMP0` reference in the following example) as the first entry in each of the `ADD_REFERENCE` sections to make sure that all references are listed appropriately as retrieving data by issuing the [GET\\_DATA](#) command.

### Example

```
ADD_REFERENCE
  DUMP0 'Empty reference 0'
  REF4 'J-O Andersson, B. Sundman, CALPHAD Vol 11, (1987), p 83-92
      TRITA 0270 (1986); CR-FE'
  REF5 'J-O Andersson, Met. Trans A, Vol 19A, (1988) p 627-636
      TRITA 0207 (1986); C-CR-FE'
  REFS 'Pingfang Shi (2006), TCS PTERN Public Ternary Alloys
      Database,
      v1.2; Modified L0(BCC,Fe,C) and L0(BCC,Cr,C) parameters
      at high temperatures.'
  ... ..
  !
ADD_REFERENCE
  DUMP1 'Empty reference 1'
  REF275 'A. Fernandez Guillermet, Z. Metallkde. Vol 79(1988) p.524-
      536,
      TRITA-MAC 362 (1988); C-CO-NI AND C-CO-FE-NI'
  REF393 'K. Frisk, Metall. Trans. Vol 21A (1990) p 2477-2488,
      TRITA 0409 (1989); CR-FE-N'
```

REF1096 'P. Gustafson, Metall. Trans. 19A(1988) p 2547-2554,  
TRITA-MAC 348, (1987); C-CR-FE-W'

... ..

!

## CASE and ENDCASE

---

```

CASE [ELEMENT/SPECIE/PHASE] !
  IF (boolean algebra on element, species or phase names) THEN
    [GIBBS/POLY/DATA command] !
  ENDCASE !

```

This keyword takes as its argument, a definition on which type of the following Boolean algebra operates. A simple Boolean algebra using `AND` and `OR` with a maximum of four levels of parentheses works. The `CASE` construction must end with the `ENDCASE` keyword. This makes it possible to have additional GIBBS or POLY commands executed depending on the user selection of elements, species or phases. The DATA commands that can be given as secondary keyword to [DEFAULT\\_COMMAND](#) can also be executed.

### Example 1

```

CASE ELEMENT !
  IF ((CR OR TI OR V) AND N)
  THEN GES AM_PH_DES @ C_S ,, CR MO TI V:C N: !
  ENDCASE !

```

### Example 2

```

CASE ELEMENT !
  IF(O) THEN TDB DEFINE_SYSTEM_ELEMENT /- !
  ENDCASE !

```

### Example 3

```

CASE ELEMENT !
  IF(AL AND FE)
  THEN TDB DEF_SYS_ELEMENT VA !
  ENDCASE !

```

---

## VERSION\_DATA

---

```
VERSION_DATE [string]*78 !
```

The string is denoted as the version/revision date and database manager of the database. Nowadays, this directive is used mainly for the purpose of keeping the development and revision history mostly updated.

### **Example**

```
VERSION_DATE Last update and adjustment: Database Manager, 2015-09-25  
!
```

---

## Working with the Mobility Databases

---

**In this section:**

---

Extensions to Database Definition File Syntax .....	62
PARAMETER .....	63
DIFFUSION .....	64
ZEROVOLUME_SPECIES .....	65

---

## Extensions to Database Definition File Syntax

---

Software packages for simulation of diffusional phase transformations, such as the Diffusion Module (DICTRA) and Precipitation Module (TC-PRISMA), need both thermodynamic data and kinetic data (i.e. diffusivities or mobilities). Naturally, the handling and storage of kinetic data also benefits from the use of some kind of database management. Thus, the TDB database definition file syntax has been extended to incorporate some new keywords needed for storing kinetic data - [PARAMETER](#), [DIFFUSION](#), and [ZEROVOLUME\\_SPECIES](#).

## PARAMETER

```

PARAMETER [special GIBBS parameter name] [low temp. limit]
[expression 1]; [upper temp. limit 1] Y
[expression 2]; [upper temp. limit 2] Y
[expression 3]; [upper temp. limit 2] Y
..... ; ..... Y
[expression n-1]; [upper temp. limit n-1] Y
[expression n]; [upper temp. limit n] N {Ref. Index} !

```

This keyword allows you to enter all types of normal GIBBS parameters for thermodynamic data as well as five special extensions suitable for kinetic data used in the Diffusion Module (DICTRA).

Valid extensions to special GIBBS parameter names are:

- MQ: Activation enthalpy for mobility equation.
- MF: Pre-exponential factor for mobility equation.
- DQ: Activation enthalpy for diffusivity equation.
- DF: Pre-exponential factor for diffusivity equation.
- VS: Volume counted per mole of volume carrying species.

### Example



See the TDB file for DICTRA example *exa4: Carburization of binary Fe-C alloy: Comparison to an analytical erf solution*. Press F1 in Thermo-Calc to search the help for more information.

```

~
$===== D I F F U S I O N   D A T A =====
$
  ZEROVOLUME_SPECIES C VA !

| PARA DQ(FCC_A1&C,*:*) , , -148000+32.8*T+R*T*LOG(4.53E-7) ; , , N!
| PARA DQ(FCC_A1&FE,*:*) , , -286000+R*T*LOG(0.700E-4) ; , , N!

```

## DIFFUSION

```
DIFFUSION [model keyword] [phase name] [additional parameter(s)] !
```

This keyword specifies the type of diffusion model to use for a phase if the default model is not desired. The default model calculates the full diffusion matrix. A diffusivity is calculated from the different mobilities and the thermodynamic factors. The former ones are calculated as:

$$M = \exp(\sum MF/RT) \exp(\sum MQ/RT) / RT$$

where  $\sum$  stands for a weighted summation of the different MF's and MQ's plus possibly a Redlich-Kister term.

### Valid Model Keywords

Keyword	Description
NONE	No diffusion in this phase
DILUTE	Constitution list of dependent species in each sublattice must be given as an additional parameter. Only the diagonal terms in the diffusion matrix are calculated. $D = \exp(\sum DF/RT) \exp(\sum DQ/RT)$ .
SIMPLE	Constitution list of dependent species in each sublattice must be given as additional parameter. Only the diagonal terms in the diffusion matrix are calculated. $D = \sum DF + \sum DQ$ .
MAGNETIC	The so-called ALPHA and ALPHA2 parameters must be given as additional parameters. ALPHA is for the substitutional magnetic model and ALPHA2 for the interstitial one. By appending an & sign and a species name after the alpha keyword one can supply individual values for the different species. The full diffusion matrix is calculated.

### Examples

```
DIFFUSION NONE SIGMA !
```

```
DIFFUSION DILUTE CEMENTITE : FE : C : !
```

```
DIFFUSION MAGNETIC BCC_A2 ALPHA=0.3 ALPHA2&C=1.8 ALPHA2&N=0.6 !
```

---

## ZEROVOLUME\_SPECIES

---

```
ZEROVOLUME_SPECIES [list of species] !
```

In the Diffusion Module (DICTRA), the assumption that the volume is carried by the substitutional elements only is applied. The interstitial elements are assumed to have zero molar volumes. This keyword uses a list of which species are to be considered as zero volume ones for an argument.



There is no underscore (\_) between *Zero* and *Volume*.

### **Example**

```
ZEROVOLUME_SPECIES VA C N !
```

---

## Database Definition File Examples

---

**In this section:**

---

Example 1: A Steel Database .....	67
Example 2: A Custom Database for the Sb-Sn Binary System .....	69
Example 3: A Public Database for the Fe-Cr-C Ternary System .....	73

## Example 1: A Steel Database

---

```

TEMP-LIM 500.0 2000.0 !
$
$ELEMENT, NAME, REF.STATE, ATOMIC-MASS, H0, S0 !
ELEMENT VA VACUUM          0.0    0.0 0.0 !
ELEMENT C  GRAPHITE        12.011 0.0 0.0 !
ELEMENT V  BCC              50.9415 0.0 0.0 !
ELEMENT CR BCC-PARAMAGNETIC 51.996  0.0 0.0 !
ELEMENT FE FCC-PARAMAGNETIC 55.847  0.0 0.0 !
ELEMENT NI FCC-PARAMAGNETIC 58.69   0.0 0.0 !
ELEMENT MO BCC              95.94   0.0 0.0 !
ELEMENT W  BCC              183.85  0.0 0.0 !
$
$PHASE, NAME, TYPE, NR-OF-SUBL, SITES-IN-EACH-SUBL. !
PHASE BCC      B1M  2 1.0 3.0 !
PHASE FCC      F2M  2 1.0 1.0 !
PHASE HCP      0    2 2.0 1.0 !
PHASE LIQUID   3    2 1.0 1.0 !
PHASE CEMENTITE 4    2 3.0 1.0 !
PHASE M23C6    4    2 23.0 6.0 !
PHASE M7C3     4    2 7.0 3.0 !
PHASE M6C      4    4 2.0 2.0 2.0 1.0 !
PHASE SIGMA    0    3 10.0 4.0 16.0 !
PHASE MU-PHASE 0    3 7.0 2.0 4.0 !
PHASE R-PHASE  0    3 27.0 14.0 12.0 !
PHASE GRAPHITE 4    1 1.0 !
$
$CONSTITUENT, PHASE-NAME : CONSTITUENTS !
CONSTITUENT BCC :V CR FE NI MO W:VA C: !
CONSTITUENT FCC :V CR FE NI MO W:VA C: !
CONSTITUENT HCP :CR FE NI:VA C N: !
CONSTITUENT LIQUID :C V CR FE NI MO W VA:VA C: !
CONSTITUENT CEMENTITE :CR FE:C: !
CONSTITUENT M23C6 :CR FE:C: !

```

---

```
CONSTITUENT M7C3 :CR FE:C: !
CONSTITUENT M6C :FE:W:FE W:C: !
CONSTITUENT SIGMA :FE:V CR MO:FE V CR MO: !
CONSTITUENT MU-PHASE :FE:MO W:FE MO W: !
CONSTITUENT R-PHASE :FE:MO :FE MO: !
CONSTITUENT GRAPHITE :C: !
$
$TYPE_DEFINITIONS:
TYPE-DEFINITION 0 SEQ TCPATH\DATA\METDATA\TC-THEREST.TDB !
TYPE-DEFINITION 1 SEQ TCPATH\DATA\METDATA\TC-BCC.TDB !
TYPE-DEFINITION 2 SEQ TCPATH\DATA\METDATA\TC-FCC.TDB !
TYPE-DEFINITION 3 SEQ TCPATH\DATA\METDATA\TC-LIQUID.TDB !
TYPE-DEFINITION 4 SEQ TCPATH\DATA\METDATA\TC-CARBIDES.TDB !
TYPE-DEFINITION M SEQ TCPATH\DATA\METDATA\TC-CURIE-BOHR.TDB !
TYPE-DEFINITION B GES AM-PH BCC MAGNETIC -1 .4 !
TYPE-DEFINITION F GES AM-PH FCC MAGNETIC -3 .28 !
$
$DEFAULT_COMMANDS:
DEFAULT-COMMAND DEF_ELEMENT VA !
DEFAULT-COMMAND REJ_SYS-CONST LIQUID 1 VA !
$DATABASE_INFORMATION:
DATABASE-INFO The following binary and ternary systems are available: '
    FE-CR-NI by Hertzman'
    FE-MO      Fernandez'
    FE-CR-C    Andersson'
    FE-W-C     Gustafson'
    FE-W       Andersson & Gustafson' !
```

## Example 2: A Custom Database for the Sb-Sn Binary System

---

```

$
$ELEMENT, NAME, REF.STATE, ATOMIC-MASS, H0, S0 !
ELEM VA VACUUM          0.0  0.0 0.0 !
ELEM MG HCP (A3)        24.305 0.0 0.0 !
ELEM SB RHOMBOHEDRAL (A7) 121.75  0.0 0.0 !
ELEM SN BCT (A5)        118.69  0.0 0.0 !
ELEM /- ELECTRON-GAS    0      0   0 !
$
$SPECIES, NAME, STOICHIOMETRIC-FORMULA !
SPECIE MG1 MG1!
SPECIE MG2 MG2!
SPECIE MG2+ MG/+2!
SPECIE SB1 SB1!
SPECIE SB2 SB2!
SPECIE SB4 SB4!
SPECIE SB3- SB/-3!
SPECIE SB5- SB/-5!
SPECIE SN1 SN1!
SPECIE SN4- SN/-4!
$
$PHASE, NAME, TYPE, NR-OF-SUBL, SITES-IN-EACH-SUBL. !
PHASE BCT          Z 1 1.0!
PHASE HCP          Z 1 1.0!
PHASE RHOMBO      Z 1 1.0!
PHASE GAS:G       Z 1 1.0!
PHASE LIQUID:L    Z 1 1.0!
PHASE IONICLIQ:Y Z 2 1 1!
PHASE SPLIQ:Y     Z 2 1 1!
PHASE BMG3SB2:I   Z 2 3 2!
PHASE AMG3SB2:I   Z 2 3 2!
PHASE MG2SN:I     Z 2 2 1!
PHASE SBSN        Z 2 1 1!

```

---

```
PHASE SB2SN3      Z 2 2 3!
$
$CONSTITUENT, PHASE-NAME : CONSTITUENTS !
CONSTITUENT RHOMBO :SB SN:!
CONSTITUENT HCP :MG SN:!
CONSTITUENT BCT :SB SN:!
CONSTITUENT GAS:G :MG1 MG2 SB1 SB2 SB4 SN1:
  > Gas phase, using the Ideal EOS and Mixing Model. !
CONSTITUENT LIQUID:L :SB SN:!
CONSTITUENT IONICLIQ:Y :MG2+:SB SB3- SN SN4- VA:
  > This is the Ionic Liquid Solution Phase. !
CONSTITUENT SPLIQ:Y :MG2+:SB SB3- SN SN4- VA:!
CONSTITUENT BMG3SB2:I :MG2+:SB3- SB5- VA SN4-:!
CONSTITUENT AMG3SB2:I :MG2+:SB3- VA SN4-:!
CONSTITUENT MG2SN:I :MG2+ VA:SB3- SN4-:!
CONSTITUENT SBSN :SB SN:SB SN:!
CONSTITUENT SB2SN3 :SB:SN:!
$
$DEFAULT_COMMANDS:
DEFAULT-COM DEF-ELEM VA /-!
DEFAULT-COM REJ-PHASE LIQUID!
DEFAULT-COM REJ-PHASE SPLIQ!
$
$TYPE_DEFINITIONS:
TYPE-DEFINITION Z SEQ * !
$
$DATABASE_INFORMATION:
DATABASE_INFO The Sb-Sn system with isentropic temperatures!
$
$VERSION_DATE:
VERSION_DATE Last update 1986-05-18 11:39:49 !
$
$
$ HERE COMES THE THERMODYNAMIC DATA (expressed in functions &
parameters):
$
```

---

---

```
FUNCTION MGLIQUID 298.15 -4630.90976+192.994374*T-34.0888057*T*LOG(T)
    -36544605.6*T**(-2); 6000 N!
$
FUNCTION MGSOLID 298.15 -8367.34+143.677876*T-26.1849785*T*LOG(T)
    +4.858E-4*T**2-1.393669E-6*T**3+78950*T**(-1);
    923.00 Y -13804.4772 +202.909445*T-34.0888057*T*LOG(T)
    -3.65446056E7*T**(-2) +1.06753982E28*T**(-9); 6000 N!
$
FUNCTION SBLIQUID 298.15 9071.98+146.800*T-31.38*T*LOG(T)
    -2.441646E8*T**(-2); 6000 N!
$
.....
..... <more>
$
FUNCTION LFCT 298.15 -17325.6+5.03600*T; 6000 N!
FUNCTION GFCTSBSN 298.15 LFCT+SBSOLID+SNSOLID+2948.291+3721.286;
    6000 N!
FUNCTION ISB 298.15 15000; 6000 N!
FUNCTION ISN 298.15 47199.9-95.6270*T; 6000 N!
$
.....
..... <more>
$
PARAMETER G(RHOMBO,SB;0) 298.15 SBSOLID; 6000 N!
PARAMETER G(RHOMBO,SN;0) 298.15 2035+SNSOLID; 6000 N!
$
PARAMETER G(HCP,MG;0) 298.15 MGSOLID; 6000 N!
PARAMETER G(HCP,SN;0) 298.15 32000+SNSOLID; 6000 N!
PARAMETER G(HCP,MG,SN;0) 298.15 -69566-9.23183*T; 6000 N!
$
PARAMETER G(BCT,SN;0) 298.15 SNSOLID; 6000 N!
PARAMETER G(BCT,SB;0) 298.15 1000+SBSOLID; 6000 N!
PARAMETER G(BCT,SB,SN;0) 298.15 0.5*ISB+0.5*ISN; 6000 N!
PARAMETER G(BCT,SB,SN;1) 298.15 0.5*ISB-0.5*ISN; 6000 N!
$
PARAMETER G(IONICLIQ,MG2+:SB3-;0) 298.15 -204389-4.98506*T
```

---

```
-2.75637E9*T**(-2)+3*MGLIQUID+2*SBLIQUID; 6000 N!  
PARAMETER G(IONICLIQ,MG2+:SN4-;0) 298.15 -98639.5+881.073*T  
-174.523*T*LOG(T)-1.79808E9*T**(-2); 6000 N!  
PARAMETER G(IONICLIQ,MG2+:SB;0) 298.15 SBLIQUID; 6000 N!  
$  
.....  
..... <more>  
$
```

## Example 3: A Public Database for the Fe-Cr-C Ternary System

---

```
$
$ Revision history:
$   Created as PDEMO by Pingfang Shi on 2004-10-05
$   Renamed to DFeCrC by Pingfang Shi on 2006-10-25
$
$ FURTHER MODIFICATIONS:
$ =====
$ pfs: /20041005 (PDEMO)
$   * Retrieved all definitions from PTERN for the Fe-Cr-C ternary !
$
$ pfs: /20061025 (DFeCrC)
$   * Ignore ELEMENT /- definition.
$   * Modify the L(BCC_A2,FE:C,VA;0) [and L(BCC_A2,CR:C,VA;0)]
$     parameters, in order to avoid BCC-appearance (Fe-C)
$     at temperatures higher than 3900 K.
$       $L(BCC_A2,CR:C,VA;0) 298.15 -190*T;          6000 N REF1 !
$       L(BCC_A2,CR:C,VA;0) 298.15 -190*T; 3000 Y 0; 6000 N REF1 !
$       $L(BCC_A2,FE:C,VA;0) 298.15 -190*T;          6000 N REF3 !
$       L(BCC_A2,FE:C,VA;0) 298.15 -190*T; 3900 Y 0; 6000 N REF3 !
$   * Add Cr into HCP_A3 phase, and
$     add all necessary G0/L parameters (according to SSOL2/TCFE3):
$       G(HCP_A3,CR:VA;0)
$       TC(HCP_A3,CR:VA;0)
$       BMAGN(HCP_A3,CR:VA;0)
$       G(HCP_A3,FE:C;0)
$       G(HCP_A3,CR:C;0)
$       L(HCP_A3,CR:C,VA;0)
$       L(HCP_A3,CR,FE:VA;0)
$   * Add "TMM:300/3000" limits to all three binary joins in the
$     ASSESSED_SYSTEM section.
$   * Change the reference indices, e.g., 101 to REF1, in order to
$     always get the reference list correctly.
```







```

$-----
$
FUNCT GHSERCC  298.15 -17368.441+170.73*T-24.3*T*LN(T)
                -4.723E-04*T**2+2562600*T**(-1)-2.643E+08*T**(-2)
                +1.2E+10*T**(-3); 6000 N!
FUNCT GHSERCR  298.15 -8856.94+157.48*T-26.908*T*LN(T)
                +.00189435*T**2-1.47721E-06*T**3+139250*T**(-1); 2180 Y
                -34869.344+344.18*T-50*T*LN(T)-2.88526E+32*T**(-9); 6000 N!
FUNCT GHSERFE  298.15 +1225.7+124.134*T-23.5143*T*LN(T)
                -.00439752*T**2-5.8927E-08*T**3+77359*T**(-1); 1811.00 Y
                -25383.581+299.31255*T-46*T*LN(T)+2.29603E+31*T**(-9); 6000 N!

FUNCT GFELIQ   298.15 +GHSERFE#+12040.17-6.55843*T
                -3.6751551E-21*T**7; 1811 Y
                -10839.7+291.302*T-46*T*LN(T); 6000 N!
FUNCT GFEFCC   298.15 +GHSERFE#-1462.4+8.282*T
                -1.15*T*LN(T)+6.4E-04*T**2; 1811 Y
                -27098.266+300.25256*T-46*T*LN(T)+2.78854E+31*T**(-9); 6000 N!
FUNCT GFCEM    298.15 +GPCEM1#-10745+706.04*T-120.6*T*LN(T); 6000 N!
FUNCT GFEM23C6 298.15 +7.666667*GFCEM#-1.666667*GHSERCC#
                +66920-40*T; 6000 N!

FUNCT GCRFCC   298.15 +GHSERCR#+7284+.163*T; 6000 N!
FUNCT GCRM23C6 298.15 -521983+3622.24*T-620.965*T*LN(T)
                -.126431*T**2; 6000 N!
FUNCT GCRM3C2  298.15 -100823.8+530.66989*T-89.6694*T*LN(T)
                -.0301188*T**2; 6000 N!
FUNCT GCRM7C3  298.15 -201690+1103.128*T-190.177*T*LN(T)
                -.0578207*T**2; 6000 N!

FUNCT GPCLIQ   298.15 +YCLIQ#*EXP(ZCLIQ#); 6000 N!
FUNCT ACLIQ    298.15 +2.32E-05*T+2.85E-09*T**2; 6000 N!
FUNCT BCLIQ    298.15 +1+3.2E-10*P; 6000 N!
FUNCT CCLIQ    298.15 1.6E-10; 6000 N!
FUNCT DCLIQ    298.15 +1*LN(BCLIQ#); 6000 N!
FUNCT ECLIQ    298.15 +1*LN(CCLIQ#); 6000 N!

```

---

FUNCT	VCLIQ	298.15 +7.626E-06*EXP (ACLIQ#);	6000 N!
FUNCT	XCLIQ	298.15 +1*EXP (.5*DCLIQ#)-1;	6000 N!
FUNCT	YCLIQ	298.15 +VCLIQ#*EXP (-ECLIQ#);	6000 N!
FUNCT	ZCLIQ	298.15 +1*LN (XCLIQ#);	6000 N!
FUNCT	GPCGRA	298.15 +YCGRA#*EXP (ZCGRA#);	6000 N!
FUNCT	ACGRA	298.15 +2.32E-05*T+2.85E-09*T**2;	6000 N!
FUNCT	BCGRA	298.15 +1+3.6E-10*P;	6000 N!
FUNCT	CCGRA	298.15 3.3E-10;	6000 N!
FUNCT	DCGRA	298.15 +1*LN (BCGRA#);	6000 N!
FUNCT	ECGRA	298.15 +1*LN (CCGRA#);	6000 N!
FUNCT	VCGRA	298.15 +5.259E-06*EXP (ACGRA#);	6000 N!
FUNCT	XCGRA	298.15 +1*EXP (.9166667*DCGRA#)-1;	6000 N!
FUNCT	YCGRA	298.15 +VCGRA#*EXP (-ECGRA#);	6000 N!
FUNCT	ZCGRA	298.15 +1*LN (XCGRA#);	6000 N!
FUNCT	GPCFCC	298.15 +YCFCC#*EXP (ZFEFCC#);	6000 N!
FUNCT	ACFCC	298.15 +1.44E-04*T;	6000 N!
FUNCT	VCFCC	298.15 +1.031E-05*EXP (ACFCC#);	6000 N!
FUNCT	YCFCC	298.15 +VCFCC#*EXP (-EFEFCC#);	6000 N!
FUNCT	GPCRLIQ	298.15 +YCRLIQ#*EXP (ZCRLIQ#);	6000 N!
FUNCT	ACRLIQ	298.15 +1.7E-05*T+9.2E-09*T**2;	6000 N!
FUNCT	BCRLIQ	298.15 +1+4.65E-11*P;	6000 N!
FUNCT	CCRLIQ	298.15 3.72E-11;	6000 N!
FUNCT	DCRLIQ	298.15 +1*LN (BCRLIQ#);	6000 N!
FUNCT	ECRLIQ	298.15 +1*LN (CCRLIQ#);	6000 N!
FUNCT	VCRLIQ	298.15 +7.653E-06*EXP (ACRLIQ#);	6000 N!
FUNCT	XCRLIQ	298.15 +1*EXP (.8*DCRLIQ#)-1;	6000 N!
FUNCT	YCRLIQ	298.15 +VCRLIQ#*EXP (-ECRLIQ#);	6000 N!
FUNCT	ZCRLIQ	298.15 +1*LN (XCRLIQ#);	6000 N!
FUNCT	GPCRBCC	298.15 +YCRBCC#*EXP (ZCRBCC#);	6000 N!
FUNCT	ACRBCC	298.15 +1.7E-05*T+9.2E-09*T**2;	6000 N!
FUNCT	BCRBCC	298.15 +1+2.6E-11*P;	6000 N!
FUNCT	CCRBCC	298.15 2.08E-11;	6000 N!

---

---

FUNCT	DCRBCC	298.15	+1*LN(BCRBCC#);	6000	N!
FUNCT	ECRBCC	298.15	+1*LN(CCRBCC#);	6000	N!
FUNCT	VCRBCC	298.15	+7.188E-06*EXP(ACRBCC#);	6000	N!
FUNCT	XCRBCC	298.15	+1*EXP(.8*DCRBCC#)-1;	6000	N!
FUNCT	YCRBCC	298.15	+VCRBCC#*EXP(-ECRBCC#);	6000	N!
FUNCT	ZCRBCC	298.15	+1*LN(XCRBCC#);	6000	N!
FUNCT	GFELIQ	298.15	+YFELIQ#*EXP(ZFELIQ#);	6000	N!
FUNCT	AFELIQ	298.15	+1.135E-04*T;	6000	N!
FUNCT	BFELIQ	298.15	+1+4.98009787E-12*P+3.20078924E-14*T*P;	6000	N!
FUNCT	CFELIQ	298.15	+4.22534787E-12+2.71569924E-14*T;	6000	N!
FUNCT	DFELIQ	298.15	+1*LN(BFELIQ#);	6000	N!
FUNCT	EFELIQ	298.15	+1*LN(CFELIQ#);	6000	N!
FUNCT	VFELIQ	298.15	+6.46677E-06*EXP(AFELIQ#);	6000	N!
FUNCT	XFELIQ	298.15	+1*EXP(.8484467*DFELIQ#)-1;	6000	N!
FUNCT	YFELIQ	298.15	+VFELIQ#*EXP(-EFELIQ#);	6000	N!
FUNCT	ZFELIQ	298.15	+1*LN(XFELIQ#);	6000	N!
FUNCT	GFEFCC	298.15	+YFEFCC#*EXP(ZFEFCC#);	6000	N!
FUNCT	AFEFCC	298.15	+7.3097E-05*T;	6000	N!
FUNCT	BFEFCC	298.15	+1+3.25236341E-11*P+3.36607808E-16*T*P;	6000	N!
FUNCT	CFEFCC	298.15	+2.62285341E-11+2.71455808E-16*T;	6000	N!
FUNCT	DFEFCC	298.15	+1*LN(BFEFCC#);	6000	N!
FUNCT	EFEFCC	298.15	+1*LN(CFEFCC#);	6000	N!
FUNCT	VFEFCC	298.15	+6.688726E-06*EXP(AFEFCC#);	6000	N!
FUNCT	XFEFCC	298.15	+1*EXP(.8064454*DFEFCC#)-1;	6000	N!
FUNCT	YFEFCC	298.15	+VFEFCC#*EXP(-EFEFCC#);	6000	N!
FUNCT	ZFEFCC	298.15	+1*LN(XFEFCC#);	6000	N!
FUNCT	GFEBCC	298.15	+YFEBCC#*EXP(ZFEBCC#);	6000	N!
FUNCT	AFEBCC	298.15	+2.3987E-05*T+1.2845E-08*T**2;	6000	N!
FUNCT	BFEBCC	298.15	+1+2.80599565E-11*P+3.06481523E-16*T*P;	6000	N!
FUNCT	CFEBCC	298.15	+2.20949565E-11+2.41329523E-16*T;	6000	N!
FUNCT	DFEBCC	298.15	+1*LN(BFEBCC#);	6000	N!
FUNCT	EFEBCC	298.15	+1*LN(CFEBCC#);	6000	N!
FUNCT	VFEBCC	298.15	+7.042095E-06*EXP(AFEBCC#);	6000	N!

---



```

+117369-24.63*T; 6000 N REF0 !
PARAM G(LIQUID,CR;0) 298.15 +GHSERCR#+GPCRLIQ#
+24339.955-11.420225*T+2.37615E-21*T**7; 2180 Y
+GHSERCR#+GPCRLIQ#
+18409.36-8.563683*T+2.88526E+32*T**(-9); 6000 N REF0 !
PARAM G(LIQUID,FE;0) 298.15 +GFELIQ#+GPFELIQ#; 6000 N REF0 !

PARAM L(LIQUID,C,CR;0) 298.15 -90526-25.9116*T; 6000 N REF1 !
PARAM L(LIQUID,C,CR;1) 298.15 +80000; 6000 N REF1 !
PARAM L(LIQUID,C,CR;2) 298.15 +80000; 6000 N REF1 !
PARAM L(LIQUID,C,FE;0) 298.15 -124320+28.5*T; 6000 N REF3 !
PARAM L(LIQUID,C,FE;1) 298.15 +19300; 6000 N REF3 !
PARAM L(LIQUID,C,FE;2) 298.15 +49260-19*T; 6000 N REF3 !
PARAM L(LIQUID,CR,FE;0) 298.15 -14550+6.65*T; 6000 N REF4 !
PARAM L(LIQUID,C,CR,FE;0) 298.15 -496063; 6000 N REF2 !
PARAM L(LIQUID,C,CR,FE;1) 298.15 +57990; 6000 N REF2 !
PARAM L(LIQUID,C,CR,FE;2) 298.15 +61404; 6000 N REF2 !

$***** FCC_A1 (Solution) *****
$PHASE FCC_A1 %BC 2 1 1 !
$ Note the C TYPE_DEF for 2nd FCC composition-set (MC) is not necessary
$ for the (Fe,Cr)-C system.
PHASE FCC_A1 %B 2 1 1 !
CONST FCC_A1 : CR,FE%
: C,VA% : !

PARAM G(FCC_A1,CR:VA;0) 298.15 +GCRFCC#+GPCRBCC#; 6000 N REF0 !
PARAM TC(FCC_A1,CR:VA;0) 298.15 -1109; 6000 N REF0 !
PARAM BMAGN(FCC_A1,CR:VA;0) 298.15 -2.46; 6000 N REF0 !
PARAM G(FCC_A1,FE:VA;0) 298.15 +GFEFCC#+GPFEFCC#; 6000 N REF0 !
PARAM TC(FCC_A1,FE:VA;0) 298.15 -201; 6000 N REF0 !
PARAM BMAGN(FCC_A1,FE:VA;0) 298.15 -2.1; 6000 N REF0 !

PARAM G(FCC_A1,CR:C;0) 298.15 +GHSERCR#+GHSERCC#
+1200-1.94*T; 6000 N REF2 !
PARAM G(FCC_A1,FE:C;0) 298.15 +GFEFCC#+GHSERCC#+GPCFCC#

```

```

+77207-15.877*T; 6000 N REF3 !
PARAM TC (FCC_A1,FE:C;0) 298.15 -201; 6000 N REF3 !
PARAM BMAGN (FCC_A1,FE:C;0) 298.15 -2.1; 6000 N REF3 !

PARAM L (FCC_A1,CR:C,VA;0) 298.15 -11977+6.8194*T; 6000 N REF2 !
PARAM L (FCC_A1,FE:C,VA;0) 298.15 -34671; 6000 N REF3 !
PARAM L (FCC_A1,CR,FE:VA;0) 298.15 +10833-7.477*T; 6000 N REF4 !
PARAM L (FCC_A1,CR,FE:VA;1) 298.15 +1410; 6000 N REF4 !
PARAM L (FCC_A1,CR,FE:C;0) 298.15 -74319+3.2353*T; 6000 N REF2 !

$***** BCC_A2 (Solution) *****
PHASE BCC_A2 %A 2 1 3 !
CONST BCC_A2 : CR%,FE%
: C,VA% : !

PARAM G (BCC_A2,CR:VA;0) 298.15 +GHSERCR#+GPCRBCC#; 6000 N REF0 !
PARAM TC (BCC_A2,CR:VA;0) 298.15 -311.5; 6000 N REF0 !
PARAM BMAGN (BCC_A2,CR:VA;0) 298.15 -.01; 6000 N REF0 !
PARAM G (BCC_A2,FE:VA;0) 298.15 +GHSEFFE#+GPFEBCC#; 6000 N REF0 !
PARAM TC (BCC_A2,FE:VA;0) 298.15 1043; 6000 N REF0 !
PARAM BMAGN (BCC_A2,FE:VA;0) 298.15 2.22; 6000 N REF0 !

PARAM G (BCC_A2,CR:C;0) 298.15 +GHSERCR#+3*GHSECC#+GPCRBCC#+3*GPCGRA#
+416000; 6000 N REF1 !
PARAM TC (BCC_A2,CR:C;0) 298.15 -311.5; 6000 N REF1 !
PARAM BMAGN (BCC_A2,CR:C;0) 298.15 -.008; 6000 N REF1 !
PARAM G (BCC_A2,FE:C;0) 298.15 +GHSEFFE#+3*GHSECC#+GPFEBCC#+3*GPCGRA#
+322050+75.667*T; 6000 N REF3 !
PARAM TC (BCC_A2,FE:C;0) 298.15 1043; 6000 N REF3 !
PARAM BMAGN (BCC_A2,FE:C;0) 298.15 2.22; 6000 N REF3 !

$ PF-20061025: Modify the L(BCC_A2,FE:C,VA;0) [and L(BCC_A2,CR:C,VA;0)]
$ parametera, in order to avoid BCC-appearance (Fe-C)
$ at temperatures higher than 3900 K.
$PARAM L (BCC_A2,CR:C,VA;0) 298.15 -190*T; 6000 N REF1 !
PARAM L (BCC_A2,CR:C,VA;0) 298.15 -190*T; 3000 Y 0; 6000 N REFS !

```

```

$PARAM L(BCC_A2,FE:C,VA;0) 298.15 -190*T;          6000 N REF3 !
PARAM L(BCC_A2,FE:C,VA;0) 298.15 -190*T; 3900 Y 0; 6000 N REFS !
$
PARAM L(BCC_A2,CR,FE:VA;0) 298.15 +20500-9.68*T;    6000 N REF4 !
PARAM TC(BCC_A2,CR,FE:VA;0)      298.15 1650; 6000 N REF4 !
PARAM TC(BCC_A2,CR,FE:VA;1)      298.15  550; 6000 N REF4 !
PARAM BMAGN(BCC_A2,CR,FE:VA;0)   298.15 -.85; 6000 N REF4 !

PARAM L(BCC_A2,CR,FE:C;0) 298.15 -1250000+667.7*T; 6000 N REF2 !
PARAM TC(BCC_A2,CR,FE:C;0)      298.15 1650; 6000 N REF5 !
PARAM TC(BCC_A2,CR,FE:C;1)      298.15  550; 6000 N REF5 !
PARAM BMAGN(BCC_A2,CR,FE:C;0)   298.15 -.85; 6000 N REF5 !

$***** HCP_A3 (Solution) *****
$PHASE HCP_A3 %BC 2 1 .5 !
$ Note the C TYPE_DEF for 2nd HCP composition-set (M2C) is not necessary
$   for the (Fe,Cr)-C system.
PHASE HCP_A3 %B 2 1 .5 !
CONST HCP_A3 : CR,FE,
              : VA%,C : !

PARAM G(HCP_A3,CR:VA;0) 298.15 +GHSERCR#+4438;          6000 N REF1 !
PARAM TC(HCP_A3,CR:VA;0) 298.15 -1109; 6000 N REF1 !
PARAM BMAGN(HCP_A3,CR:VA;0) 298.15 -2.46; 6000 N REF1 !
PARAM G(HCP_A3,FE:VA;0) 298.15 +GHSEFFE#+GPFHEHCP#
                        -3705.78+12.591*T-1.15*T*LN(T)+6.4E-04*T**2; 1811 Y
                        +GHSEFFE#+GPFHEHCP#
                        -3957.199+5.24951*T+4.9251E+30*T**(-9); 6000 N REF0 !
PARAM G(HCP_A3,CR:C;0) 298.15 +GHSERCR#+.5*GHSECC#
                        -18504+9.4173*T-2.4997*T*LN(T)+.001386*T**2; 6000 N REF1 !
PARAM G(HCP_A3,FE:C;0) 298.15 +GFEFCC#+.5*GHSECC#+GPCFCC#
                        +52905-11.9075*T; 6000 N REF3 !

PARAM L(HCP_A3,CR:C,VA;0) 298.15 +4165; 6000 N REF1 !
PARAM L(HCP_A3,FE:C,VA;0) 298.15 -22126; 6000 N REF3 !
$PARAM L(HCP_A3,FE:C,VA;0) 298.15 -17335; 6000 N TCFE3 !

```

---

```
PARAM L(HCP_A3,CR,FE:VA;0) 298.15 +10833-7.477*T; 6000 N REF4 !

$***** SIGMA (Solution) *****
PHASE SIGMA % 3 8 4 18 !
CONST SIGMA : FE
              : CR
              : CR,FE : !

PARAM G(SIGMA,FE:CR:CR;0) 298.15 +8*GFEFCC#+22*GHSERCR#
                          +92300-95.96*T+GPSIG1#; 6000 N REF4 !
PARAM G(SIGMA,FE:CR:FE;0) 298.15 +8*GFEFCC#+4*GHSERCR#+18*GHSERFE#
                          +117300-95.96*T+GPSIG2#; 6000 N REF4 !

$***** CEMENTITE (Solution) *****
PHASE CEMENTITE % 2 3 1 !
CONST CEMENTITE : CR,FE%
                  : C : !

PARAM G(CEMENTITE,CR:C;0) 298.15 +3*GHSERCR#+GHSERCC#
                          -48000-9.2888*T; 6000 N REF2 !
PARAM G(CEMENTITE,FE:C;0) 298.15 +GFECCEM#; 6000 N REF3 !

PARAM L(CEMENTITE,CR,FE:C;0) 298.15 +25278-17.5*T; 6000 N REF2 !

$***** M3C2 (Compound) *****
PHASE M3C2 % 2 3 2 !
CONST M3C2 : CR
            : C : !

PARAM G(M3C2,CR:C;0) 298.15 +GCRM3C2#; 6000 N REF2 !

$***** M7C3 (Solution) *****
PHASE M7C3 % 2 7 3 !
CONST M7C3 : CR%,FE
            : C : !
```

---





---

# Database Checker

---

**In this section:**

---

About the Database Checker .....	88
Using the Database Checker .....	90

## About the Database Checker

---

The Thermo-Calc software package includes a program to check that the syntax of unencrypted database files is correct. The program applies the syntax rules set out in the *Database Manager User Guide* and reports errors and issues warnings. This program is intended for advanced users who develop and manage databases.



Thermo-Calc accepts some deviations from these syntax rules. Although warnings can be ignored, errors must be corrected. Reported errors must be corrected in the TDB file before the database can be loaded by the application. Sometimes the warnings can also reveal the need for additional corrections as shown in the example below.

For example, a warning is generated if a phase name is detected that is also an abbreviation of another phase name in the database. Although allowed, it is not recommended as this can (in some cases) lead to unexpected results because abbreviations are also used when entering commands in the Console and included in macro files. The following shows why in this case an abbreviation is not recommended.

Imagine that a macro file uses the abbreviation `ABC` to refer to a phase in a database that is named `ABCDE`. As long as there is no other phase in the database with the same exact name or initial abbreviation, it works. But if a new phase is added to the database and is actually named `ABC` then the macro can in error refer to the wrong phase. The macro still runs but now refers to the phase `ABC` instead of `ABCDE` because `ABC` is an exact match, *which takes precedence over abbreviations*. The Database Checker displays a warning because this situation can only occur when one phase name is an abbreviation of another.

With respect to using this tool, the executable `DatabaseChecker` file is found in the Thermo-Calc home directory.

The program can also be launched by selecting **Tools** →  **Database Checker** from the main menu in Thermo-Calc.



Large amounts of RAM can be required when checking large databases.

The Database Checker application is standalone and available as indicated from the **Tools** menu in Thermo-Calc. However, as required, you can run the `DatabaseChecker.exe` file as a stand-alone program from the file Thermo-Calc installation directory.

-  [Installation Default Directory: Windows](#), [Installation Default Directory: macOS](#), or [Installation Default Directory: Linux](#)
-  [Using the Database Checker](#)

### *Custom Databases: GES5 and GES6 Calculation Engine Version*

As of Thermo-Calc version 2019b, the default calculation engine is set to GES6. For custom database users, you may sometimes see messages in the log indicating that additional preparation is being done by Thermo-Calc prior to a calculation. The preparation is carried out automatically; it is usually fast and only needed the first time a custom database is used after a modification. While databases with warnings are tolerated, databases that contain errors can only be loaded by GES6 after the errors are corrected.

The Database Checker tool uses the same parser as GES6 and can be used to verify the consistency and correctness of databases during development, before the databases are actually used.

-  [Database Checker](#)



To globally set the default from GES6 back to GES5, go to the **Options** window [Global General Settings](#) tab and click **Version 5** for the *Preferred Gibbs Energy System* setting. The [SET\\_GES\\_VERSION](#) Console Mode command is used to change GES versions for a single Thermo-Calc session.



Contact Thermo-Calc Software AB at [info@thermocalc.com](mailto:info@thermocalc.com) for more information about the changes or if you encounter any issues you think are related to the new version of the calculation engine.

## Using the Database Checker

---

### [About the Database Checker](#)

Once you have installed the Thermo-Calc and are familiar with the TDB database file concept you can start working with the file to ensure it does not contain errors that prevent it from working with Thermo-Calc.

### **Example of Working with the Database Checker**

The following shows a couple of the features available when using an unencrypted TDB database file included with Thermo-Calc Console Mode example *tcex11 - Oxidation of Cu<sub>2</sub>S with an H<sub>2</sub>O/O<sub>2</sub> Gas Mixture*.

### [Thermo-Calc Console Mode Examples](#)

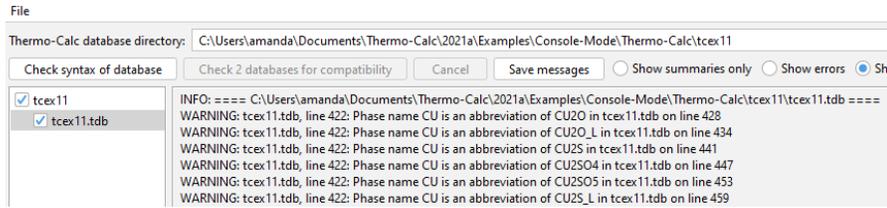
Open a TDB file. This example uses `tcex11.TDB`, which is the user-defined database referred to from the Thermo-Calc macro file `tcex11.TCM`.

1. In Thermo-Calc, from the menu select **Tools→Database Checker**.
2. In the Thermo-Calc Database Checker window, click the **File**  button and navigate to a directory with TDB files or a file with a `.TDB` extension.



If two databases are selected at the same time, these can be tested for compatibility i.e., whether the databases can be appended. The tool generates error messages for database entities e.g., phases, that are not compatible but this does not necessarily mean that the databases cannot be appended, only that the erroneous phases cannot be appended and must be rejected from the system.

3. Click **Select directory or file**. With the file loaded, click **Check syntax of database**. Choose how to show the information: **Show summaries only**, **Show errors**, **Show errors and warnings**, or **Show all messages**. The Database Checker lists the information in the window.



4. Click **Save messages** as needed to save a `.txt` file and use this to help improve your database.

---

# TDB Editor

---

**In this section:**

---

About the TDB Editor .....	93
Installing VSCode and the TDB Editor .....	94
Using the TDB Editor .....	97

## About the TDB Editor

---



In order to use this tool, it is important to understand how to work with the standard TDB database format. For example, see [Thermodynamic and Kinetic/Mobility Databases](#) in the *Database Manager User Guide*, which is part of the *Thermo-Calc Documentation Set*, or press F1 in Thermo-Calc and search for *TDB file* in the help.

The TDB Editor is an editing tool for those who develop their own custom databases for use with Thermo-Calc or other applications. The tool speeds up the database editing process by providing immediate feedback through syntax coloring, syntax checking, and by allowing formatting and easy navigation to items of interest in the TDB file.

The advantage to using the TDB Editor compared to a basic text editor includes features such as syntax coloring and checking, formatting of TDB files, and generally easier navigation.

The TDB Editor has two components (see [Installing VSCode and the TDB Editor](#) for details):

- As an installed extension to the open source code editor Microsoft Visual Studio Code (VSCode), which has the user interface for the TDB Editor. VSCode is available for free.
- A server application that runs invisibly in the background and provides the VSCode extension with services specific to the TDB format, such as syntax checking, formatting and the location within the document of various items, such as functions and references.



The syntax checker uses the same text parsing engine as the Database Checker tool and the Thermo-Calc application itself. This means that an edited TDB file with no errors shown in the TDB Editor, is also accepted by the Thermo-Calc application. For more information see [About the Database Checker](#).

---

## Installing VSCode and the TDB Editor

---



### [About the TDB Editor](#)

To use the TDB Editor, you install the Visual Studio Code (VSCode) and then in VSCode you install the TDB Editor extension, which is included with your Thermo-Calc installation.



In order to access the most-up-to-date version of the VSCode plugin, it must be updated every time a *new* version of Thermo-Calc is installed.

## System Requirements

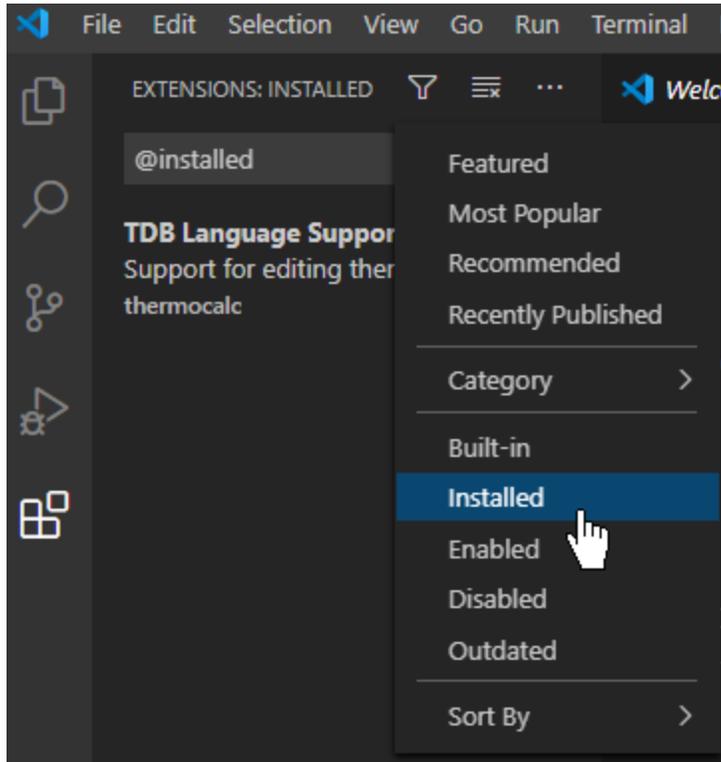
The same system requirements apply to the TDB Editor as to the Thermo-Calc installation but the TDB Editor is only available for Windows and Linux platforms.

A full list of [system requirements](#) and program availability is found on the Thermo-Calc website. You can also see the [roadmap for platform support](#) to use for planning purposes.

## How to Install the TDB Editor Extension

1. Navigate to the Thermo-Calc distribution directory based on your operating system. Locate the file `tdb-language-support.vsix`.
  - a. On Windows, the file is typically located under `C:\Program Files\Thermo-Calc-<version>`.
  - b. On Linux, the file is typically located under `<user>/Thermo-Calc/<version>`.
  - c. On macOS, the file is typically located under `/Applications/Thermo-Calc-<version>`.
2. Copy the `tdb-language-support.vsix` file to your desktop.
3. Install VSCode for your platform from <https://code.visualstudio.com/download>.
4. In VSCode, press `Ctrl+Shift+X` (or from the menu, select **View→Extensions**). The **Extensions** panel opens.

5. In the upper right corner of the **Extensions** panel, click the three dots (...) **Views and More Actions** button. Select **Install from VSIX** and select the tdb-language-support.vsix file on your desktop.
6. Locate **TDB Language Support** in the Extensions panel. For example, click **Filter Extensions → Installed** and the file displays in the panel.



7. Click **TDB Language Support** to see the README file, which includes current version information, known bugs, and limitations.

## READ ME Section Included with the Extension



After installing the TDB Editor, there is a README section available on the front page of the extension in VSCode that has the most recent information, some of which is listed below.

- The syntax checker cannot resolve references to TDB files with partial database information and gives errors in such cases. Only databases contained in single files can be handled. For syntax checking of multi-file databases, use the stand-alone Database Checker tool. See [About the Database Checker](#).
- If you encounter warning messages that do not match the content in the TDB Editor, a possible solution is to save the file to disk (press Ctrl+S on the keyboard), which reloads the file and restarts the syntax checker.
- Very large files can sometimes cause issues with the language support functionality although the actual editing should be unaffected by this.

## Using the TDB Editor

### [? About the TDB Editor](#)

Once you have installed the TDB Editor and are familiar with the TDB database file concept you can start working with it to edit an existing file or create a new TDB file.

### How to Create a TDB File

To create a new TDB file, save an empty text file with a `.TDB` extension.

### How to Open a TDB File

In VSCode, from the menu select **File**→**Open** and navigate to a file with a `.TDB` extension.

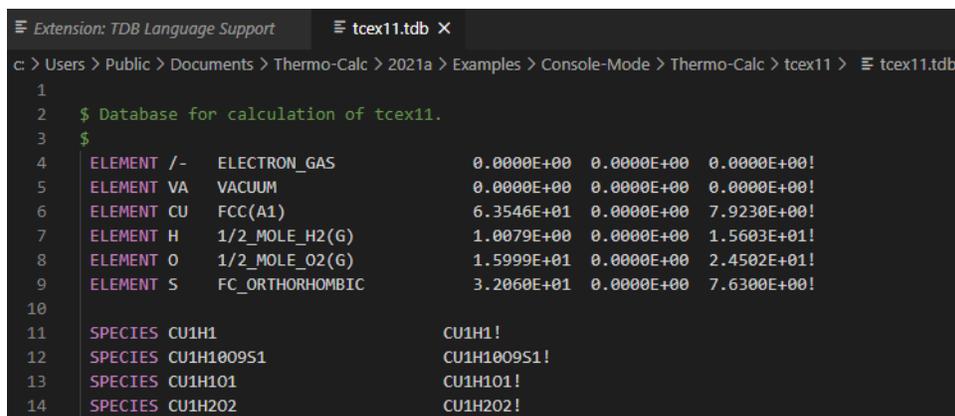
### Example of Working with the TDB Editor

The following shows a couple of the features available when using an unencrypted TDB database file included with Thermo-Calc Console Mode example *tcex11 - Oxidation of Cu<sub>2</sub>S with an H<sub>2</sub>O/O<sub>2</sub> Gas Mixture*.

### [? Thermo-Calc Console Mode Examples](#)

Open a TDB file. This example uses `tcex11.TDB`, which is the user-defined database referred to from the Thermo-Calc macro file `tcex11.TCM`.

The unencrypted database opens in the TDB Editor where you can quickly see the color syntax of the commands making it easy to navigate.

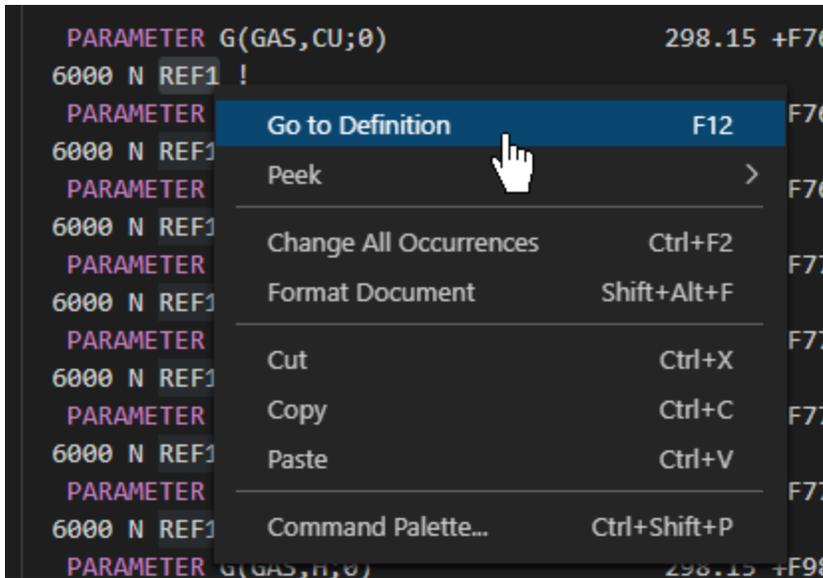


```

c > Users > Public > Documents > Thermo-Calc > 2021a > Examples > Console-Mode > Thermo-Calc > tcex11 > tcex11.tdb
1
2 $ Database for calculation of tcex11.
3 $
4 ELEMENT /- ELECTRON_GAS 0.0000E+00 0.0000E+00 0.0000E+00!
5 ELEMENT VA VACUUM 0.0000E+00 0.0000E+00 0.0000E+00!
6 ELEMENT CU FCC(A1) 6.3546E+01 0.0000E+00 7.9230E+00!
7 ELEMENT H 1/2_MOLE_H2(G) 1.0079E+00 0.0000E+00 1.5603E+01!
8 ELEMENT O 1/2_MOLE_O2(G) 1.5999E+01 0.0000E+00 2.4502E+01!
9 ELEMENT S FC_ORTHORHOMBIC 3.2060E+01 0.0000E+00 7.6300E+00!
10
11 SPECIES CU1H1 CU1H1!
12 SPECIES CU1H1009S1 CU1H1009S1!
13 SPECIES CU1H101 CU1H101!
14 SPECIES CU1H202 CU1H202!

```

Where there are definitions or references included with the file, you can also navigate quickly to these using **Go To Definition** (press F12 or right-click to select from the menu).

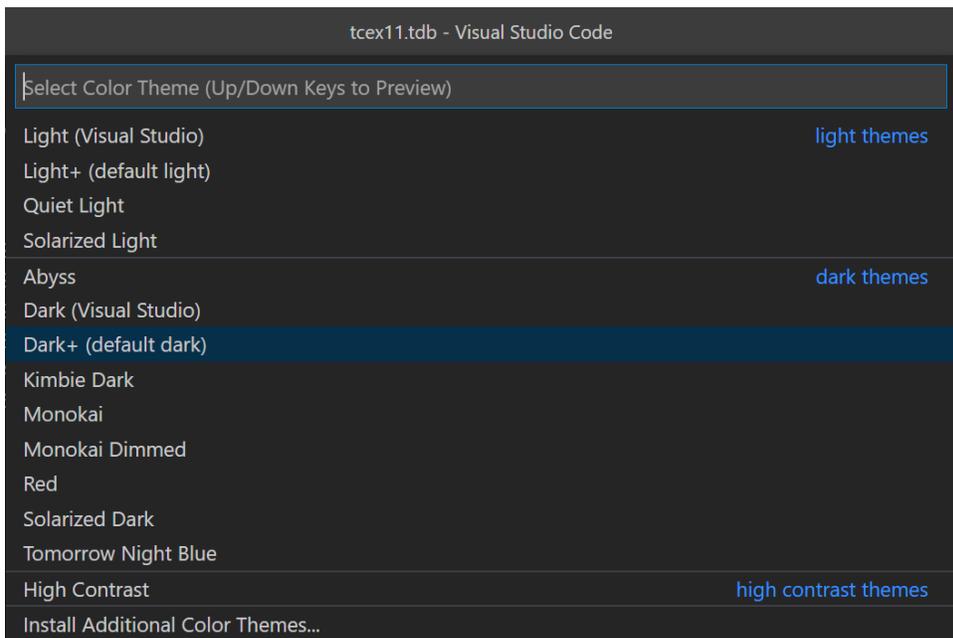


There are many other VSCode features available that you can learn more about from the Visual Studio Code Help menu. One such item of particular use is to be able to customize the color scheme.

## Customize the Color

VSCode comes with a number of pre-defined color themes and additional themes can be downloaded as extensions.

To select a color theme, in VSCode press `Ctrl+K` then `Ctrl+T`.



A preview of each theme is shown. Use the arrow keys to try each and select the theme that best suits your preferences.

# **DATAPLOT User Guide**

**Thermo-Calc Version 2024a**



---

# Introduction to the DATAPLOT User Guide

---

**In this section:**

---

Using this Guide .....	3
About the DATAPLOT Graphical Language .....	4
Plotting Experimental Data Files .....	5

---

## Using this Guide

---

The *DATAPLOT User Guide* gives all the details on the graphical language DATAPLOT that a phase diagram or property diagram calculated by Thermo-Calc or a kinetic profile simulated by the Diffusion Module (DICTRA) can be defined.

This guide

- Discusses the important features of the DATAPLOT language.
- Describes the commands to define PROLOGUE and DATASET.
- Several examples and the standard codes for formatting are provided.

One topic introduces the method of formatting DIGLIB symbols in LaTeX documents, for the purpose of necessarily/appropriately referring to the corresponding LaTeX symbols (closest to those DIGLIB symbols which have been plotted on a diagram using the DATAPLOT Graphical Language) in the texts of LaTeX documents for publications/reports.

## About the DATAPLOT Graphical Language

---

In order to obtain graphical output of any numerical data and informative strings, a graphical language called DATAPLOT was developed in connection with the graphical software DIGLIB. Using this graphical language, you can store information in a normal text file (\*.EXP), which can be plotted as graphical symbols, lines, texts or Greek letters on any plot device support by DIGLIB.

You can generate and plot DATAPLOT (\*.EXP) files together with various calculation and/or experimental results from, for example, phase and property diagrams calculated with the Thermo-Calc software composition profiles or other diagrams simulated by the Diffusion Module (DICTRA).

An EXP file automatically generated by the POST-processor (using the [MAKE EXPERIMENTAL DATAFILE](#) command) in the Thermo-Calc software is a DATAPLOT (\*.EXP) file and may contain all types of legal DATAPLOT commands and parameters. With a text editor, you can modify or add some DATAPLOT commands and related parameters in an existing EXP file. This is useful when appending experimental information to calculated/simulated plots, and when specifying user-defined texts, symbols, colors, fonts, filled patterns, diagram types, diagram sizes, symbol/character sizes, titles, special characters, etc.



There is another type of experimental data file that is generated in Thermo-Calc when working with the PARROT module. For more information see [Experimental Data Files \(\\*.POP, \\*.DOP\)](#).



See the [Tips and Tricks: How to Plot Experimental Data](#) blog post on our website for useful guidance about this topic.

## DATAPLOT Examples

To help you prepare your EXP (experimental) files to use for your plotted diagrams generated by Thermo-Calc simulations, there are examples of EXP files using the DATAPLOT graphical language standard, which are normally independent of software versions.



[Examples, Manuals, and Materials Folders](#)

## Plotting Experimental Data Files

---

You can read and plot experimental data files in both Console Mode and Graphical Mode.



See the [Tips and Tricks: How to Plot Experimental Data](#) blog post on our website for useful guidance about this topic.

### Graphical Mode

In Graphical Mode, the Experimental File Reader activity can only be created from the top My Project node. This activity allows you to read an experimental data file (\*.EXP). This type of file has information specifying a plotted diagram, written in the DATAPLOT graphical language. From this activity node you can plot the EXP file or save a copy of the file to another location.

#### ***How to Plot an EXP file in Experimental File Reader***

1. Right-click  **My Project** and from the **Create New Activity** submenu choose **Experimental File Reader**.
2. In the **Configuration** window in the *EXP file* field, enter the name of the EXP file to plot. Or click the **Select Exp file**  button to navigate to a location on the computer to add the dataset.
3. Once the .EXP file is added, under **Dataset Selection** you can choose to include or exclude specific datasets. Use the **Select/Deslect all datasets** checkbox as required.
4. Right-click the **Experimental File Reader** node and from the **Create New Successor** submenu choose **Plot Renderer**.

If the source EXP file is changed, you can also click the **Reload the selected EXP file from disk**  button to refresh the data. If you want to save or export the loaded file, click the **Save EXP file**  button, then navigate to where you want to save it and give it a file name.

5. In the **Configuration** window click **Perform** or right-click the **Plot Renderer** node and select **Perform**.

---

## Console Mode

In Console Mode you use the QUICK EXPERIMENTAL PLOT command found in the POLY-3 POST module (as well as in the Diffusion Module (DICTRA) POST module) to plot the .EXP files. This command can be used when there is no graphical information to be plotted in the POLY3 workspace. It defines a pair of axes, sets the axis labels to X and Y, and scales both X- and Y-axes between 0.0 and 1.0 unless a prologue is read from the DATAPLOT (\*.EXP) data file.

### ***How to Plot an EXP file in POST***

1. Go to the POLY-3 Module: At the SYS prompt type `Go Poly`.
2. Go to the POST module: At the POLY\_3 prompt type `Post`.
3. At the POST prompt, type `Quick`.
4. Enter Prolog and Dataset numbers (if applicable). See [QUICK EXPERIMENTAL PLOT](#).
5. AT the POST prompt, type `Plot`.

---

## Using DATAPLOT Graphical Language

---

**In this section:**

---

DATAPLOT File Structure .....	8
DATAPLOT Language Syntax .....	9
Coordinate Systems .....	10
Graphical Operation Codes .....	11
Tables or Blocks .....	13
Drawing a Polygon .....	14
Drawing an Analytical Function .....	15
Painting of an Enclosed Area .....	16
Writing a Text .....	17
Plotting a Symbol .....	18
Other Commands .....	19
Interactive Plotting .....	20
Formatting DIGLIB Symbols in LaTeX Documents .....	21
LaTeX Formatting Codes .....	23

---

## DATAPLOT File Structure

---

A DATAPLOT file is a normal text file with the extension `EXP` that can be created with a text editor or by a program. The file must contain one or more `DATASETS` and possibly also one or more `PROLOGUES`. Each `PROLOGUE/DATASET` is an entity that can be individually selected for plotting.

A `PROLOGUE/DATASET` is identified by a unique positive number in the file. A `PROLOGUE` normally contains various `DATASET` commands for defining information about axis scaling, axis text, axis length, title and so on. A `PROLOGUE` is terminated by another `PROLOGUE` or by the first `DATASET`. This means that all `PROLOGUES` must be placed at the beginning of the file, before the first `DATASET`.

A `DATASET` normally contains `DATASET` commands that are associated to separate data points, as well as with one or more `BLOCKS` of data (calculated or experimental). A `DATASET` is terminated by another `DATASET` command or the end of file.

## DATAPLOT Language Syntax

---

The DATAPLOT language consists of commands with or without parameters:

```
COMMAND {parameter(s)}
```

The basic graphical command consists of an X/Y coordinate pair and a Graphical Operation Code (GOC). With other commands, the interpretation of this basic command can be modified. There are separate commands to draw a polygon or a function and ways to get texts in different fonts.

To edit a DATAPLOT file, the graphical commands can be abbreviated.



A command (with parameters) must not exceed 80 characters. If it is too long (normally as writing necessary codes in a command's parameters for a complex expression), two or more lines can be edited.

## Coordinate Systems

---

The DATAPLOT language accepts coordinates in three different coordinate systems, which are called *world*, *virtual* and *normalized*.

- The *world* coordinates are selected to represent any kind of data and be of “any” magnitude.
- The *virtual* coordinate system uses centimeters as units. However, the actual size of one unit is dependent on the implementation of the device driver in DIGLIB. It is not recommended to use this coordinate system if different output devices are used for preliminary and final plots.
- The *normalized* coordinate system goes from zero to one. When plotting, you interactively scale each axis by selecting the minimum and maximum word coordinates on the axis. In the normalized coordinate system, the minimum axis value is represented by zero and the maximum by one. DIGLIB draws a square between the four points (of the X and Y axes) that are determined by the coordinates zero and one in the normalized coordinates.



It is also possible to draw triangular plots as described below. However, in most places, it only references square diagrams. All data points within the minimum and maximum word coordinates are plotted inside this square. DIGLIB also writes tic marks and corresponding word values at such tic marks.

It may be convenient to use normalized coordinates to draw boxes and texts. You can give normalized coordinates outside zero and one if you want to write texts outside the area enclosed by the square. To ensure proper operation outside the normalized box, the clipping must be turned off.

## Graphical Operation Codes

The Graphical Operation Code (GOC) determines how the coordinates are interpreted and what is done at the point determined by the coordinates. For an individual data point, its GOC codes must be given. For each data [BLOCK](#), you define a default GOC that is used for the whole BLOCK, unless a GOC is explicitly defined for a specific point.

- If *s* is omitted, no symbol is plotted. A number selecting the symbol can optionally follow the character *s*. The DIGLIB software determines which symbol the number represents. See [Plotting a Symbol](#).
- If any of the other characters are omitted, the default defined for the BLOCK is used. Absolute values imply the current coordinates; relative values mean that these are added to the current coordinates. The GOC must be written without any space between various characters, but the order for the characters is irrelevant.
- If no GOC is defined for a BLOCK, the system default is *MWA*, (Move Word Absolute), i.e., move to the given point that is interpreted as absolute word coordinates. *GOC=DNA* means draw a line from the current point to the new one interpreting the coordinates as normalized. After each draw or move operation, the so-called current point is the new point.
- The [TEXT](#) following the *'* command is interpreted and expanded by the DIGLIB software, if it includes any *^* operator or [STRING](#) names using the *~* descriptor. To get PostScript outputs, the TEXT must be edited by the LaTeX Text Formatting Program instead (see [LaTeX Formatting Codes](#)).

### GOC Character Definitions

The GOC is a combination of the following characters:

Character	Meaning
W, V or N	To use world (W), virtual (V) or normalized (N) coordinates. W is the default.
A or R	XY are absolute (A) or relative (R) values. A is the default.
M or D	To perform a move (M) or draw (D) operation to XY. M is the default.
B	To apply “soft” splines on a line drawn between the coordinate pairs (used only on BLOCK data).

---

<i>Character</i>	<i>Meaning</i>
s	To plot default symbol at XY.
S#	To change default symbol to No # symbol and plot it at XY.
'TEXT	To plot <code>TEXT</code> at XY. It must be the last operation in a GOC code. The <code>TEXT</code> following the <code>'</code> command is interpreted and expanded by the DIGLIB software, if it includes any <code>^</code> operator or <code>STRING</code> names using the <code>~</code> descriptor.

## Tables or Blocks

---

In many cases, you have (calculated or experimental) data in the form of tables, and want to plot one or several columns as X-axis and one or more columns as Y-axis. A single table can contain many sets of data records of the same kind of information for a specific data [BLOCK](#). There can be many data BLOCKS in a [DATASET](#).

By enclosing these tables in a data BLOCK, you can select which column(s) is the X-axis and which is the Y-axis. For example,

```
BLOCK X=C1; Y1=C3; Y2=C2; GOC=C4,DAW
```

Where the X-axis values are in column 1, the Y-axis values are in columns 2 and 3, and any GOC codes are in column 4. The default GOC for this BLOCK is Draw Absolute Word. The GOC code inside the table is only necessary if the default GOC is not applicable.

A more elaborate use of the table is shown in this example:

```
BLOCK X=C3; Y=1E3/(C3+273); GOC=C8,MAWS1
```

Where the X-axis values are in column 3, the Y-axis values are in column 1 with some additional calculations applied. Any GOC is in column 8, and the default GOC is Move Absolute Word and plot the symbol no. 1.

Columns in a table must be separated by one or more space characters and do not have to be justified.

It is possible to have tables with mixed text and numbers, but each word followed by a space is counted as one column. The columns used for plotting also must be numerical. An example of a legal line in a table is

```
298.15 This_is_the_second_column 11.5 This_is_the_fourth_column
```



A line in the table must not exceed 80 characters. A BLOCK must be terminated by a line with the [BLOCKEND](#) command.



See the [Tips and Tricks: How to Plot Experimental Data](#) blog post on our website for useful guidance about this topic.

---

## Drawing a Polygon

---

Normally, each point is written on a separate line. But in order to draw a line in a more compact way, use the command [DRAWLINE](#). DRAWLINE is followed by a couple of X/Y number pairs. The X/Y pairs must be separated by a space, and there must be a comma sign between the X and Y values. DRAWLINE makes a move operation to the first pair of X/Y coordinates, and then draws a line among all pairs up to the last one. All pairs must fit on one line of 80 characters, but there can be several consequent DRAWLINE commands.



### [Example 2 – Draw Polygons and Symbols](#)

---

## Drawing an Analytical Function

---

Use a [BLOCK](#) command to set an axis to a function. It is not necessary to use a value from any column in order to compute the function value to be plotted. Use the [FUNCTION](#) command to plot a function with an even increment of the independent variable.



[Example 4 – Draw Curves Defined by Functions](#)

## Painting of an Enclosed Area

---

Use the [PAINT](#) command to paint or fill an area in a specified pattern in the plot. Available patterns are determined by the DIGLIB software. A related command to PAINT is [PCFUNCTION](#).

## Writing a Text

---

Use the [TEXT](#) command to write a text at the current point. You can write a text at any X/Y pair by appending a single quote followed by the text on the same line. For example:

```
1.1 1.0 NAM'This is a text
```

writes `This is a text` at the normalized coordinates (1.1, 1.0).

Use the command [FONT](#) to select the font used for the text, and [CHARSIZE](#) to select the size of the characters. Set this size of the symbols with [SYMBOLSIZE](#).

If a text or a single character should be of a different font than all the other text, or to use subscripts or superscripts in a text, use the ^ operators or [STRING](#) command to create the text. The command [STRING](#) stores the text in a specified variable that includes all text-formatting information defined by the DIGLIB software.



[Example 3 – Using Strings and Various Line Types](#) and [Example 5 – Use Included Files for Predefined Symbols](#).

However, if a graphical output is done on a PostScript device using the PostScript hardcopy fonts, special text formatting codes as [LaTeX Formatting Codes](#) should be used and the [STRING](#) formatting syntax is then not valid.

---

## Plotting a Symbol

---

As described in [Graphical Operation Codes](#), a GOC code in a data BLOCK may contain an `s` option to plot a symbol for an X/Y pair or the same symbols for the data BLOCK. A number selecting the symbol can optionally follow the character `s`. You can plot a symbol at any current X/Y position by appending a quote specified by the symbol number in the GOC code (e.g., `1.1 1.0 MANS5' This is a text`).

You can also insert DIGLIB symbols into LaTeX-edited documents.



[Example 1 – Draw Lines and Symbols](#), [Example 2 – Draw Polygons and Symbols](#) and [Example 8 - DIGLIB Symbols](#)

---

## Other Commands

---

### *General*

- When plotting symbols representing various experimental data, it is important that the symbols are centered around the coordinate values.
- The dollar sign \$ as the first character of a line stands for a comment character, which means the whole line is ignored when plotting.

### *ATTRIBUTE*

Use the [ATTRIBUTE](#) command to change the coordinates where text is displayed. The default is the lower left corner of the first character in the text.

### *CLIP*

Use the [CLIP](#) command to change the default where all data outside the normalized coordinates zero and one are not plotted.

### *COLOR*

If you are using a color device, use the [COLOR](#) command to change color of the lines. On some black and white devices, colors are simulated with different width and dashing of the lines.

### *INCLUDE*

Use the [INCLUDE](#) command to create libraries with texts and include these in similar plots.

### *LINETYPE*

Use the [LINETYPE](#) command to define line types as solid, long dashed, short dashed or dotted.

## Interactive Plotting

---

The DATAPLOT file is read into the POST module in the workspace with [APPEND EXPERIMENTAL DATA](#) or [QUICK EXPERIMENTAL PLOT](#). These commands ask for the name of the DATAPLOT file and also which PROLOGUE(S) and DATASET(S) are to be plotted.



Press F1 and search the Help for details.

By giving the PROLOGUE/DATASET number as -1, you get a list of the available PROLOGUES/DATASETS in the file. Note that if `DATASET 0` is in a DATAPLOT file, its data is always used even if other DATASETS are chosen.

## Formatting DIGLIB Symbols in LaTeX Documents

---

When using the LaTeX editor, you can use the DIGLIB symbols in texts and in figures. This then refers to the corresponding LaTeX symbols (closest to those DIGLIB symbols which are plotted on a plot using the DATAPLOT Graphical Language) in the texts of LaTeX documents for publications/reports.

This topic shows how to generate some DIGLIB symbols in texts, through the attached LaTeX source file and its converted jpg file.



### [Example 8 - DIGLIB Symbols](#)

#### ***DIGLIB\_Sym.tex***

```

\documentclass[dvips,12pt]{article}
\textwidth 165mm
\textheight 225mm
\oddsidemargin 1mm
\evensidemargin 1mm
\topmargin 1mm
%%\usepackage{amssymb}
%% next replace amssymb and to get udtimes
\usepackage[utopia]{mathdesign}
\usepackage{rotating}
\usepackage[latin1]{inputenc}
\usepackage{graphics}
\usepackage{graphicx,subfigure}% with figures
%\usepackage[draft]{graphicx}% without figures
\usepackage{subfigure}% with figures
\topmargin 1mm
\oddsidemargin 1mm
\evensidemargin 1mm
\begin{document}
{\Large \bf Diglib symbols and their corresponding LaTeX symbols}
\vspace{5mm}

```

The information below gives the closest corresponding LaTeX symbol. All symbols (except +) must be generated in math mode. Most of these require the package `amssymb`, i.e., you need a directive `usepackage{amssymb}` in the preamble. Two of the symbols require the more extensive `mathdesign` that can be included with `usepackage[utopia]{mathdesign}`.

```

\vspace{5mm}
{\Large
\begin{tabular}{llll}
Diglib & Latex & Latex name & Note\\
1&  $\vartriangle$ &  $\backslash\vartriangle$  & amssymb \\
2&  $\square$ &  $\backslash\square$  & \\
3& {\Huge  $\diamond$ } &  $\backslash\diamond$  & size  $\backslash$Large$  \\
4&  $\udtimes$ &  $\backslash\udtimes$  & mathdesign \\
5&  $\triangledown$ &  $\backslash\triangledown$  & amssymb \\ 6 & + & normal + \\
7&  $\ast$ &  $\backslash\ast$  & amssymb \\
8&  $\times$ &  $\backslash\times$  & \\
9& {\Huge  $\circ$ } &  $\backslash\circ$  & size  $\backslash$Large$  \\
10& {\Huge  $\star$ } &  $\backslash\star$  & size  $\backslash$Large, amssymb \\
11&  $\curlyvee$ &  $\backslash\curlyvee$  & amssymb \\
12&  $\Join$ &  $\backslash\Join$  & \\
13&& - & nothing similar, overlapping  $\$ > < \$$  \\
14&& - & nothing similar, 10-edged star \\
15&  $\maltese$ &  $\backslash\maltese$  & mathdesign \\
16&& - & nothing similar, a pentagon \\
17&  $\curlywedge$ &  $\backslash\curlywedge$  & amssymb \\
\end{tabular}}
\end{document}$ 
```

## LaTeX Formatting Codes



For a list of supported LaTeX commands, see:

<https://www2.ph.ed.ac.uk/snuggletex/documentation/math-mode.html>.



By default when using the LaTeX command, you are in math mode and Thermo-Calc automatically interprets the entered text as math. The above link uses the dollar sign (\$) in all the examples but this is NOT required. For example, if you enter  $E=mc^2$  this is displayed including the \$ instead of the equation.



As of Thermo-Calc 2015a the LTEXT text formatting is replaced by LaTeX. For existing users who may have plots and files that use the LTEXT format, and especially when using Console Mode, the following examples show the command changes from LTEXT and its LaTeX equivalent:

### LTEXT vs LaTeX Commands for Labels

<i>Symbol or text format</i>	<i>LTEXT</i>	<i>LaTeX</i>
Subscripts in an axis text	S-A-TEXT Y N Mole-fraction Al <sup>DO2</sup> O <sup>DO3</sup>	S-A-TEXT Y N \latex \textsf{Mole-fraction } Al_ 20_3
Label with subscript	add .13 .15 n c-ZrO <sup>DO2</sup>	add .13 .15 n \latex c-ZrO_2
Greek symbol	ADD .05 .50 N <sup>GRd</sup>	ADD .05 .50 N \latex \delta

---

## PROLOGUE Commands

---

**In this section:**

---

PROLOG .....	25
DIAGRAM_TYPE .....	26
TIC_TYPE .....	27
TITLE .....	28
XLENGTH .....	29
XSCALE .....	30
XTEXT .....	31
XTYPE .....	32
YLENGTH .....	33
YSCALE .....	34
YTEXT .....	35
YTYPE .....	36

## PROLOG

---

PROLOG is the beginning of a number of consequent lines of user-defined diagram layout commands. The lines are displayed with the `optional text` when using the `-1` option in the POST-processor as prompted for the PROLOG number with [APPEND\\_EXPERIMENTAL\\_DATA](#) or [QUICK\\_EXPERIMENTAL\\_PLOT](#).



[Examples of DATAPLOT Files and Outputs](#)



See the [Tips and Tricks: How to Plot Experimental Data](#) blog post on our website for useful guidance about this topic.

Syntax	
	<code>PROLOGUE # optional text</code>
	# is an unsigned integer identifying the PROLOG.

## DIAGRAM\_TYPE

---

DIAGRAM\_TYPE sets the diagram type to square (the default) or triangular.



### [Example 6 – Triangular Diagrams for Ternary Systems](#)

**Syntax**

```
DIAGRAM_TYPE type plot_3rd_axis clip_along_third-axis
```

`type` is a character string reading SQUARE (default) or TRIANGULAR. If `type` reads TRIANGULAR, then two additional parameters are given: `plot_3rd_axis` and `clip_along_third-axis` that are character strings reading YES or NO.

## TIC\_TYPE

---

TIC\_TYPE sets the relative length of the tic marks.

Syntax	TIC_TYPE #
	# is a real number. Default value is 1. Negative number gives tics on the inside of the diagram frame. 0 gives no tics.

---

## TITLE

---

TITLE sets the title text string to be printed above the diagram.

Syntax	TITLE text
	text is an arbitrary text string that may contain text-formatting codes

---

## XLENGTH

---

XLENGTH sets the X-axis length to approximately # centimeters.

Syntax	
	XLENGTH #
	# is a positive real number (the approximate X-axis length in centimeters).

## XSCALE

---

XSCALE sets the scaling in word coordinates of the X-axis.

Syntax	
	<code>XSCALE min max</code>
	<code>min</code> and <code>max</code> are real numbers.

## XTEXT

---

XTEXT sets the X-axis text.

Syntax	
	<code>XTEXT text</code>
	<code>text</code> is an arbitrary text string that may contain text-formatting codes.

## XTYPE

---

XTYPE sets the X-axis type as linear (default), logarithmic or inverse.

Syntax	XTYPE type
	type is a character string reading LIN, LOG or INV.

---

## YLENGTH

---

YLENGTH sets the Y-axis length to approximately # centimeters.

Syntax	
	Y LENGTH #
	# is a positive real number (the approximate Y-axis length in centimeters).

## YSCALE

---

YSCALE sets the scaling in word coordinates of the Y-axis.

Syntax	YSCALE min max
	min and max are real numbers.

## YTEXT

---

YTEXT sets the Y-axis text.

Syntax	
	<code>Y TEXT text</code>
	<code>text</code> is an arbitrary text string that may contain text-formatting codes.

## YTYPE

---

YTYPE sets the Y-axis type as linear (default), logarithmic or inverse.

Syntax	Y TYPE type
	type is a character string reading LIN, LOG or INV.

---

# DATASET Commands

---

**In this section:**

---

ATTRIBUTE .....	38
BLOCK .....	39
BLOCKEND .....	40
CHARSIZE .....	41
CLIP .....	42
COLOR .....	43
DATASET .....	45
DATAPPOINT .....	46
DRAWLINE .....	47
FONT .....	48
FUNCTION .....	49
GLOBALSIZE .....	50
INCLUDE .....	51
LINETYPE .....	52
PAINT .....	53
PCFUNCTION .....	54
SYMBOLSIZE .....	55
STRING .....	56
TEXT .....	58

## ATTRIBUTE

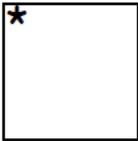
---

ATTRIBUTE specifies where the current XY position is in the character or symbol plotbox.



[Example 2 – Draw Polygons and Symbols](#), [Example 3 – Using Strings and Various Line Types](#), and [Example 5 – Use Included Files for Predefined Symbols](#)

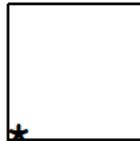
Syntax	ATTRIBUTE
Prompts	attribute
	attribute may be TOP, CENTER or BOTTOM. ATTRIBUTE CENTER is default for symbols; ATTRIBUTE BOTTOM is default for characters.



TOP



CENTER



BOTTOM

## BLOCK

BLOCK defines how the following numeric data block shall be interpreted. The definitions of X and Y coordinates may also be expressed as a function of the column values, making it possible to perform transformations.

<b>Syntax</b>	<code>BLOCK X&amp;=C#; ...; Y&amp;=C#; ...; GOC=C#,@@@...</code>
	&' are optional unsigned integers that make it possible to plot several (maximum 9) X- or Y- axis columns. #' are unsigned integers identifying the column numbers. The column number # in "GOC=C#" is the location of any possible GOC codes in the current data BLOCK; @@@ stands for the default Graphical Operation Code (GOC) for the current BLOCK. The GOC code inside the current table is only necessary if the current default GOC is not applicable.

### GOC Characters

The GOC is a combination of the following characters:

<i>Character</i>	<i>Meaning</i>
W, V or N	To use world (W), virtual (V) or normalized (N) coordinates. W is the default.
A or R	XY are absolute (A) or relative (R) values. A is the default.
M or D	To perform a move (M) or draw (D) operation to XY. M is the default.
B	To apply "soft" splines on a line drawn between the coordinate pairs (used only on BLOCK data).
S	To plot default symbol at XY.
S#	To change default symbol to No # symbol and plot it at XY.
'TEXT	To plot TEXT at XY. It must be the last operation in a GOC code. The TEXT following the ' command is interpreted and expanded by the DIGLIB software, if it includes any ^ operator or STRING names using the ~ descriptor.

## **BLOCKEND**

---

Use BLOCKEND to terminate the local definition of the graphical operation code defined by the earlier BLOCK command.

## CHARSIZE

---

CHARSIZE redefines the default size of the characters in the plot. The character size has an initial default value, which may vary with the current font setting and the output device (the plot format).



### [Examples of DATAPLOT Files and Outputs](#)

Syntax	CHARSIZE \$\$
	\$\$ must be an unsigned real.

## CLIP

---

CLIP turns clipping on or off. If it is OFF, it allows output outside the ordinary plot area defined by normalized coordinates zero and one.



[Example 2 – Draw Polygons and Symbols](#) and [Example 9 - DIGLIB Fonts and Lines](#)

Syntax	CLIP CLP
	clp is a character string reading ON or OFF.

## COLOR

Use COLOR to redefine the current color setting. There are 22 legal color codes (from 0 to 21).



### Example 7 – Color Codes

#### Syntax

COLOR code

code is an unsigned integer number (from 0 to 21) identifying the color, or a character string specifying the color.

### **DIGLIB Color Codes**

<i>Code</i>	<i>Color</i>	<i>Equivalent Character</i>
0	Background	Invisible
1	Black	Normal
2	Red	Very_Thick
3	Green	Thin
4	Blue	Thick
5	Yellow	Very_Thin
6	Magenta	Dashed
7	Cyan	Dotted
8	Purple	
9	Gold	
10	Turquoise	
11	Pink	
12	Gray	

---

<i>Code</i>	<i>Color</i>	<i>Equivalent Character</i>
13	Orange-red	
14	Maroon	
15	Plum	
16	Sea green	
17	Olive	
18	Sienna	
19	Orange	
20	Coral	
21	UserDef	
22 and higher	Any digit larger than 21 has the color effect as 1 (black).	

## DATASET

---

DATASET indicates the beginning of a number of consequent lines comprising a set of user-defined data. The DATASET lines are displayed on the terminal along with the `optional text` when using the `-1` option in the POST-processor as prompted for the DATASET number in the [APPEND\\_EXPERIMENTAL\\_DATA](#) or [QUICK\\_EXPERIMENTAL\\_PLOT](#) command.



See the [Tips and Tricks: How to Plot Experimental Data](#) blog post on our website for useful guidance about this topic.

### Syntax

```
DATASET # optional text
```

# is an unsigned integer identifying this set of data.

## DATAPOINT

---

DATAPOINT is not a DATASET command, but the basic DATAPLOT command (see synopsis below) performs an action at the current point determined by the specified X/Y-coordinates. A DATASET may contain various data points, in addition to one or more data [BLOCKs](#). Such data points are separated and independent on each other.

Syntax	X Y GOC
	X and Y are unsigned real numbers identifying the X/Y-coordinates for the current data point. GOC stands for Graphical Operation Code (GOC) for the current point. Legal GOC characters.
	<b>Example</b>
	<pre>0.7 0.95 N'Example 6 0.5 0.08 MNA'E^FS18^SQ(^SK^FS10A+5#8*C#^FS10 -!a^FS18)^FS11+B^DIA#b#S</pre>

## DRAWLINE

---

DRAWLINE draws a line starting at  $(x_1, y_1)$  to  $(x_n, y_n)$  through  $(x_2, y_2) \rightarrow (x_{n-1}, y_{n-1})$ .



[Example 1 – Draw Lines and Symbols](#) and [Example 2 – Draw Polygons and Symbols](#)

### Syntax

```
DRAWLINE x1,y1 x2,y2 . . . . xn,yn
```

$x$  and  $y$  may be reals or integers of any value. This is the same as connecting all points in a table:

x1	y1	M
x2	y2	D
. . .	. . .	D
xn	yn	D

## FONT

---

FONT redefines the default font setting in the POST-processor.



### [Example 9 - DIGLIB Fonts and Lines](#)

Syntax	FONT #
	# is an unsigned integer.

## Font Numbers

<i>Number</i>	<i>Font</i>
1	Bold
2	Normal
3	Italic
5	Bold and Italic
10	Monotype Normal
11	Monotype Italic
12	Monotype Bold
13	Monotype Bold and Italic
14	Narrow Normal
15	Narrow Italic

# FUNCTION

---

FUNCTION defines and plots a user-defined function.



## Example 4 – Draw Curves Defined by Functions

### Syntax

```
FUNCTION Y=f(X); start end number_of_XY_pairs; GOC;  
FUNCTION X=f(Y); start end number_of_XY_pairs; GOC;
```

$f(X)$  or  $f(Y)$  are legal mathematical functions of  $X$  or  $Y$ , understandable by a FORTRAN program.  $start$  and  $end$  are unsigned real numbers, and  $number\_of\_XY\_pairs$  an unsigned integer.  $GOC$  is a legal graphical operation code as defined with [BLOCK](#).

## GLOBALSIZE

---

GLOBALSIZE redefines the default global size of the plot. The global size has an initial default value.

Syntax	GLOBALSIZE \$\$
	\$\$ must be an unsigned real.

## INCLUDE

---

INCLUDE adds a file into the current input stream.



### Example 5 – Use Included Files for Predefined Symbols.

Syntax	
	<code>INCLUDE filename</code>
	<code>filename</code> is a legal filename (with its correct path) for the operation system.

## LINETYPE

---

LINETYPE redefines the current linetype in the plot.



[Example 3 – Using Strings and Various Line Types](#), [Example 8 - DIGLIB Symbols](#) and [Example 9 - DIGLIB Fonts and Lines](#)

Syntax	LINETYPE #
	<p># must be an unsigned integer. Legal linetypes are (for both normal graphical outputs and for PostScript formats):</p> <ul style="list-style-type: none"> <li>• Solid (default)</li> <li>• Long dashed</li> <li>• Short dashed</li> <li>• Dotted</li> </ul>

### Line Types

Number	Type
1	Straight line
2	Dashed line
3	Dotted line
4	Dash-Dot line

## PAINT

---

PAINT paints the area enclosed by the current path in the current pattern. The current path starts at the last MOVETO given and includes all subsequent DRAWS. This command only works for the PostScript format (as graphical files or on printed hardcopy).



### PCFUNCTION

#### Syntax

PAINT <code> <video> <mode>

<> denotes optional parameters. To set a new current pattern, supply any or all of the optional parameters.

- <code> is a single letter 0-9, A-Z or a-t (if <code>=t, also supply a space and a number in the range 0.00-1.00). The default is 0.
- <video> is a string reading NORMAL (the default) or INVERSE.
- <mode> is a string reading TRANSPARENT (the default) or OPAQUE.

## PCFUNCTION

---

PCFUNCTION appends a user-defined function to the current path. It is used with [PAINT](#).

Syntax	<pre>PCFUNCTION Y=f(X); start end number_of_XY_pairs; GOC; PCFUNCTION X=f(Y); start end number_of_XY_pairs; GOC;</pre>
	<p><math>f(X)</math> or <math>f(Y)</math> are legal mathematical functions of X or Y, understandable by a FORTRAN program. <i>start</i> and <i>end</i> are unsigned real numbers, and <i>number_of_XY_pairs</i> an unsigned integer. <i>GOC</i> is a legal graphical operation code as defined with <a href="#">BLOCK</a>.</p>

## SYMBOLSIZE

---

SYMBOLSIZE redefines the current symbol size setting. The symbol size has an initial default value.



[Example 3 – Using Strings and Various Line Types](#), [Example 7 – Color Codes](#) and [Example 8 - DIGLIB Symbols](#)

Syntax	SYMBOLSIZE \$\$
	\$\$ must be an unsigned real.

## STRING

STRING defines a string containing a text and operation codes (e.g., to change the default font settings).



[Example 3 – Using Strings and Various Line Types](#) and [Example 5 – Use Included Files for Predefined Symbols](#)

### Syntax

```
STRING name text_with_each_character_in_^S#^G^F#^U#^D#^R#^L#^N
```

name is a valid alphanumeric name (variable) to represent the text including all text formatting codes. It is highly recommended that all the characters in the name are in CAPITAL CASE (e.g. ACA2SO4); otherwise, the defined STRING can be incorrectly plotted.

```
text_with_each_character_in_^S#^G^F#^U#^D#^R#^L#^N
```

means the text is coded with each of its characters (and/or numeric numbers) that are formatted with various operators (^S#, ^G, ^F#, ^U#, ^D#, ^R#, ^L# and/or ^N). # are unsigned integers. ^ is the caret character and does not mean a control character. Between an # (in an operator) and a numeric number (as a part of the text), there must always be a comma sign (,) or the number is not plotted in the text, because the # with the number is interpreted as another incorrect #. In some of the commands, #=0 resets the option to previous (or default) value.

### String Operation Codes in the DIGLIB Software

For the PostScript hardcopy fonts, these operation codes are not valid; see [LaTeX Formatting Codes](#):

Operator	Operation
^S#	Set character size to size #
^G	Set font to Greek
^F#	Set font to font number #
^U#	Move up # units
^D#	Move down # units
^R#	Move right # units

---

Operator	Operation
$\wedge L \#$	Move left # units
$\wedge N$	Do not move, remain at current

### Examples

```
STRING Alpha1 ^Ga^F0^D0^S8,1^S0^U0
```

is set to  $\alpha_1$

```
STRING M23C6 M^D0^S8,23^S0^U0C^D0^S8,6^S0^U0
```

is set to  $M_{23}C_6$

```
STRING ACA2CO3 ^Ga^F0^D0^S8Ca^D0^S4,2^S0^U0^S8CO^D0^S4,3^S0^U0
```

is set to  $Ca_2CO_3$

## TEXT

---

TEXT outputs, at the current position, the text following the keyword TEXT or the text in the string `string_name` that is defined with [STRING](#).

Syntax	<code>TEXT text or ~string_name</code>
	It is possible to mix ^ operators and previously defined string names using the ~ operator.

## Examples of DATAPLOT Files and Outputs

---

Once your Thermo-Calc software is installed you can open the Examples and PDF documentation (*Manuals*) from the main menu:

- To open the Manuals folder select **Help → Manuals Folder**. Note for Linux openSUSE users: This link is not functional. To access the folder, navigate to the default location based on your installation type
- To open the Examples based on whether you are in Console Mode or Graphical Mode, select **File → Examples Files** or **Help → Examples Files**.

### In this section:

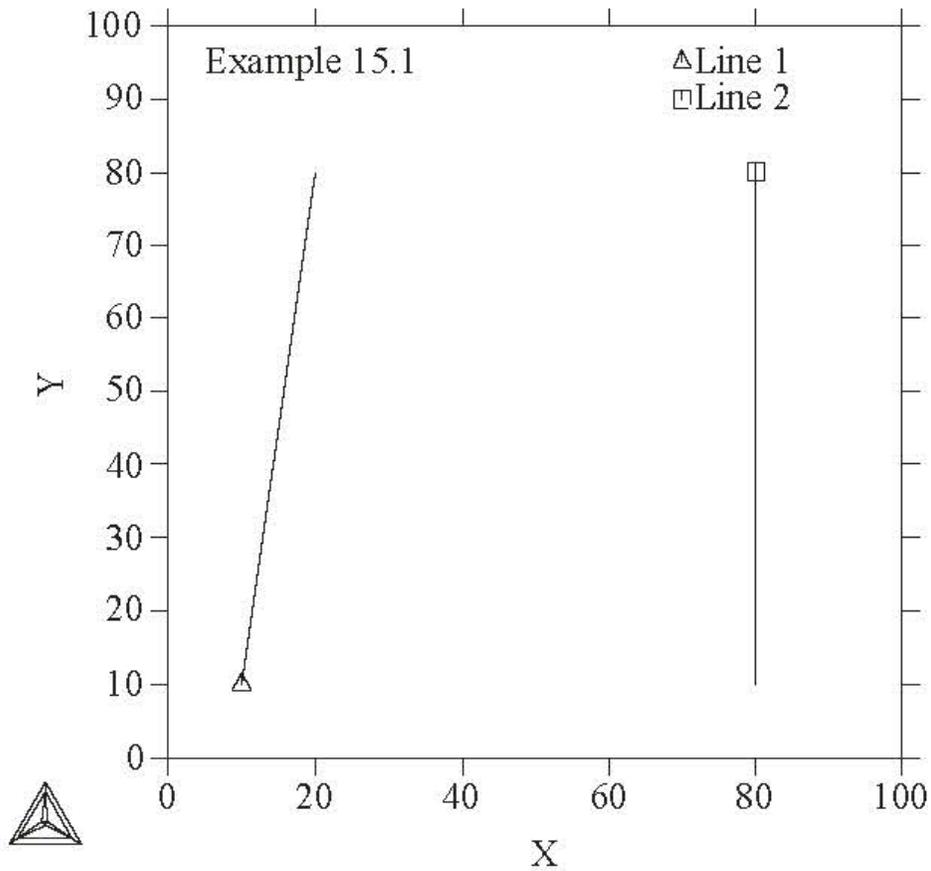
---

Example 1 – Draw Lines and Symbols .....	60
Example 2 – Draw Polygons and Symbols .....	62
Example 3 – Using Strings and Various Line Types .....	64
Example 4 – Draw Curves Defined by Functions .....	67
Example 5 – Use Included Files for Predefined Symbols .....	69
Example 6 – Triangular Diagrams for Ternary Systems .....	72
Example 7 – Color Codes .....	78
Example 8 - DIGLIB Symbols .....	83
Example 9 - DIGLIB Fonts and Lines .....	91

## Example 1 – Draw Lines and Symbols

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file `DATAPLOT 1 - Draw Lines and Symbols.EXP` available with your installation. To open the folder: From the Thermo-Calc menu, select **Help → Manuals Folder**.

### Output Example



### Macro Text

```
DATAPLOT Example 1
PROLOG 1 EXAMPLE 1 0<X<100, 0<Y<100
XSCALE 0.00000 100
YSCALE 0.00000 100
XTYPE LINEAR
```

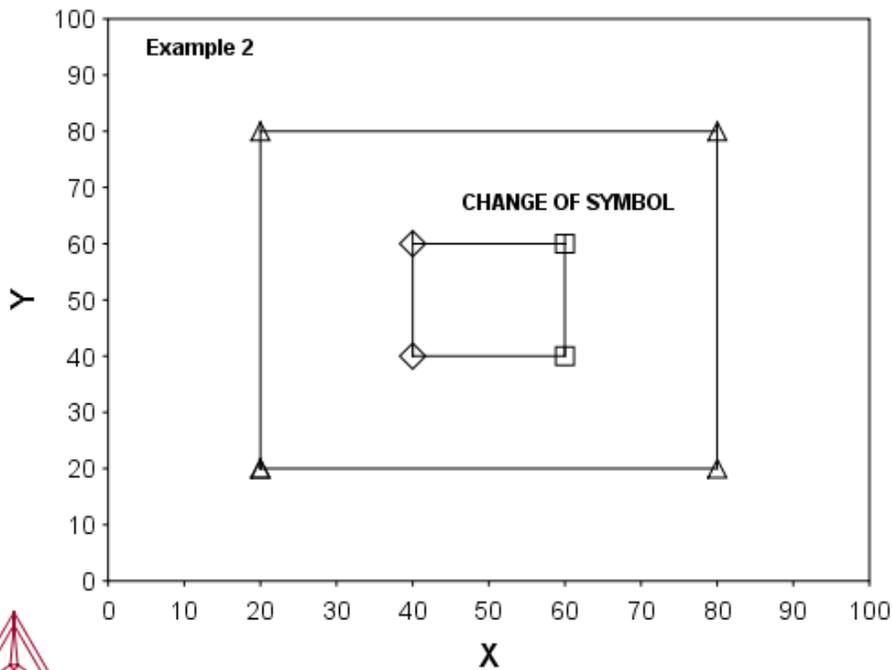
---

```
YTYPE LINEAR
XLENGTH 11.5000
YLENGTH 11.5000
TITLE EXAMPLE 1
XTEXT X
YTEXT Y
DATASET 1 Two lines started with two symbols
ATTRIBUTE CENTER
0.05 0.95 N'Example 1
0.7 0.95 NS'Line 1
0.7 0.90 NS2'Line 2
10 10 S1
20 80 D
80 80 S2
80 10 D
50 60
```

## Example 2 – Draw Polygons and Symbols

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file `DATAPLOT 2 - Draw Polygons and Symbols.EXP` available with your installation. To open the folder: From the Thermo-Calc menu, select **Help** → **Manuals Folder**.

### Output Example



### Macro Text

```
$DATAPLOT Example 2
PROLOG 2 EXAMPLE 2 0<X<100, 0<Y<100
XSCALE 0.00000 100
YSCALE 0.00000 100
XTYPE LINEAR
YTYPE LINEAR
XLENGTH 11.5000
YLENGTH 11.5000
TITLE EXAMPLE 2
XTEXT X
```

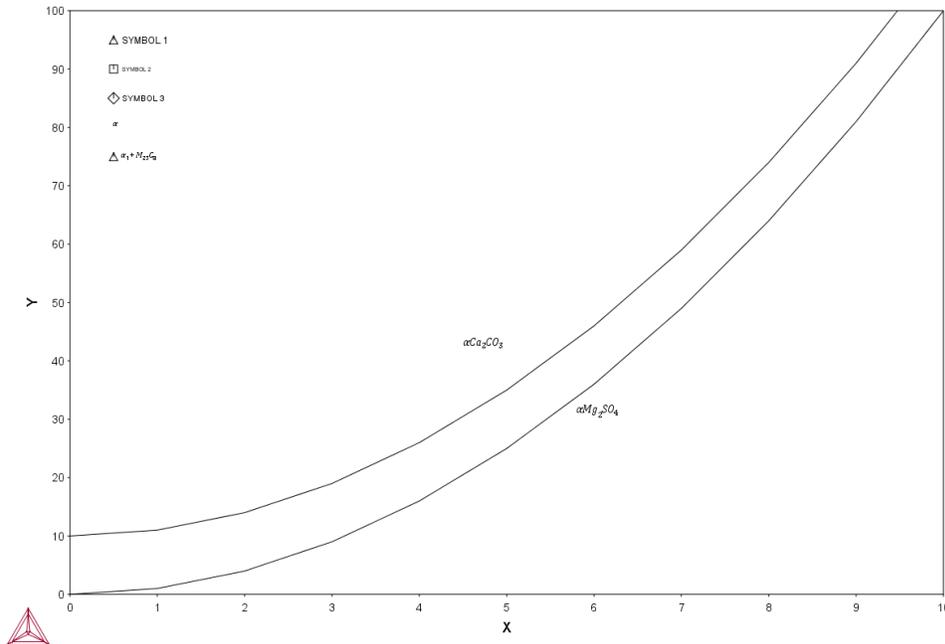
---

```
YTEXT Y
DATASET 2 Two ploygons with three types of symbols
ATTRIBUTE CENTER
CLIP OFF
0.05 0.95 N'Example 2
1.1 0.95 NS1'SYMBOL 1
1.1 0.90 NS2'SYMBOL 2
1.1 0.85 NS3'SYMBOL 3
BLOCK X=C1; Y=C2; GOC=C3,DSWA
40 40 M
40 60
60 60 S2'CHANGE OF SYMBOL
60 40
40 40 S0
BLOCKEND
BLOCK X=C1*100; Y=C2*100; GOC=C3,DSWA
0.2 0.2 MS1
0.2 0.8
0.8 0.8
0.8 0.2
0.2 0.2
BLOCKEND
$Draw an additional polygon:
DRAWLINE 20,10 30,15 40,30 50,80
DRAWLINE 50,80 60,60 90,30
```

## Example 3 – Using Strings and Various Line Types

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file DATAPLOT 3 – Using String and Line Types.EXP available with your installation. To open the folder: From the Thermo-Calc menu, select **Help** → **Manuals Folder**.

### Output Example



### Macro Text

```
$DATAPLOT Example 3
PROLOG 3 EXAMPLE 3 0<X<10, 0<Y<100
XSCALE 0.00000 10
YSCALE 0.00000 100
XTYPE LINEAR
YTYPE LINEAR
XLENGTH 11.5000
YLENGTH 11.5000
TITLE EXAMPLE 3
XTEXT X
YTEXT Y
DATASET 3 Draw curves; plot formatted texts and symbols
```

---

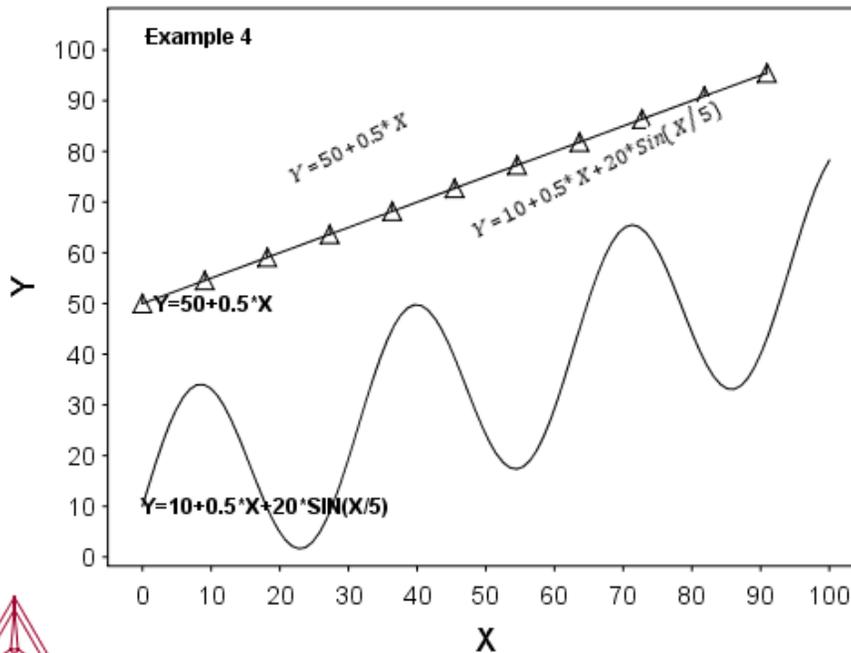
```
$Define some strings:
STRING BCC \latex \alpha
STRING BCC1 \latex \alpha_1 + M_23C_8
STRING M23C6 M_23C_6
STRING ACA2CO3 \latex \alpha Ca_2CO_3
STRING AMG2SO4 \latex \alpha Mg_2SO_4
ATTRIBUTE CENTER
CLIP OFF
FONT 2
0.05 0.95 NS1'SYMBOL 1
CHARSIZE 0.2
0.05 0.90 NS2' SYMBOL 2
CHARSIZE 0.3
0.05 0.85 NS3'SYMBOL 3
0.05 0.80 N' ~BCC
0.05 0.75 NS1'~BCC1
SYMBOLSIZE 0.4
CHARSIZE 0.4
0.45 0.42 N'~ACA2CO3
0.58 0.30 N'~AMG2SO4
CLIP ON
LINETYPE 1
BLOCK X=C1; Y=C1*C1; GOC=C2, DWA
0 M
1
2
3
4
5
6
7
8
9
10
BLOCKEND
```

```
LINETYPE 2  
BLOCK X=C1; Y=C1*C1+10; GOC=C2, DWA  
0 M  
1  
2  
3  
4  
5  
6  
7  
8  
9  
10  
BLOCKEND
```

## Example 4 – Draw Curves Defined by Functions

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file `DATAPLOT 4 - Curves and Functions.EXP` available with your installation. To open the folder: From the Thermo-Calc menu, select **Help → Manuals Folder**.

### Output Example



### Macro Text

```
$DATAPLOT Example 4
PROLOG 4 EXAMPLE 4 0<X<100, 0<Y<100
XSCALE 0.00000 100
YSCALE 0.00000 100
XTYPE LINEAR
YTYPE LINEAR
XLENGTH 11.5000
YLENGTH 11.5000
TITLE EXAMPLE 4
XTEXT X
```

---

```
YTEXT Y
DATASET 4 Plot two functions as lines:
ATTRIBUTE CENTER
0.05 0.95 N'Example 4
$ Draw two lines defined by FUNCTIONS:
FUNCTION Y=10+0.5*X+20*Sin(X/5); 0 100 100; DWA;
FUNCTION Y=50+0.5*X; 0 100 10; DS1WA;
$ Write functions beside the lines:
$ Note the real rotation angle (27 degree) can be seen
$ only on the PostScript hardcopy
0.25 0.68 N'^RO27Y=50+0.5*X
0.25 0.45 N'^RO27Y=10+0.5*X+20*Sin(X/5)
```

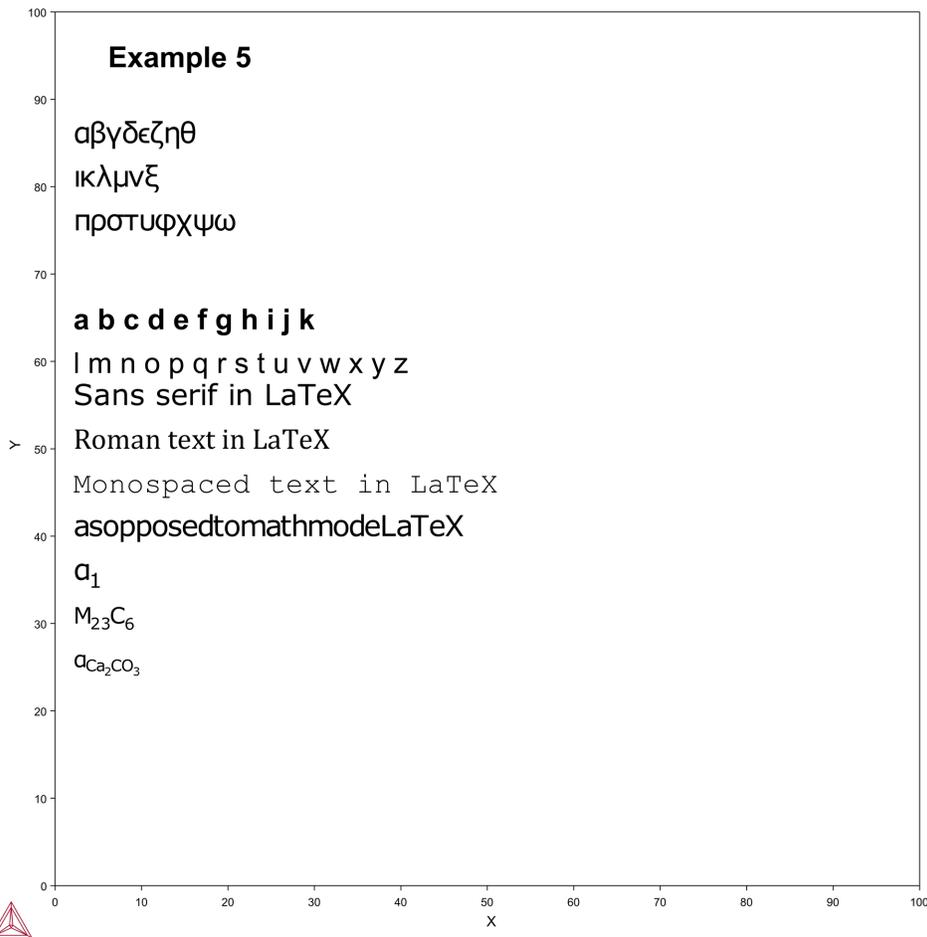
## Example 5 – Use Included Files for Predefined Symbols

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file `DATAPLOT 5 - Predefined Symbols.EXP` available with your installation. To open the folder: From the Thermo-Calc menu, select **Help** → **Manuals Folder**.

### Output Example



In the plot result, the hard-to-read LaTeX line is intended to highlight how to type readable text using LaTeX.



---

## Macro Text

```
$DATAPLOT Example 5 Symbols
PROLOG 5 EXAMPLE 5 0<X<100, 0<Y<100
XSCALE 0.00000 100
YSCALE 0.00000 100
XTYPE LINEAR
YTYPE LINEAR
XLENGTH 11.5000
YLENGTH 11.5000
TITLE EXAMPLE 5
XTEXT X
YTEXT Y
DATASET 5 Write equations and strings with different properties
STRING TEST1A \latex \alpha \beta \gamma \delta \epsilon \zeta \eta \theta
STRING TEST1B \latex \iota \kappa \lambda \mu \nu \xi \omicron
STRING TEST1C \latex \pi \rho \sigma \tau \upsilon \phi \chi \psi \omega
STRING TEST2A a b c d e f g h i j k
STRING TEST2B l m n o p q r s t u v w x y z
STRING BCC1 \latex \alpha_1
STRING ACA2CO3 \latex \alpha_{Ca_2CO_3}
ATTRIBUTE CENTER
CHARSIZE 1
0.05 0.95 N'Example 5
CLIP OFF
0.01 0.85 N'~TEST1A
0.01 0.80 N'~TEST1B
0.01 0.75 N'~TEST1C
FONT 1
0.01 0.65 N'~TEST2A
FONT 2
0.01 0.60 N'~TEST2B
0.01 0.55 N'\latex \textsf{Sans serif in LaTeX}
0.01 0.50 N'\latex \textrm{Roman text in LaTeX}
0.01 0.45 N'\latex \texttt{Monospaced text in LaTeX}
```

0.01 0.40 N'\latex as opposed to math mode LaTeX

0.01 0.35 N'~BCC1

CHARSIZE 0.8

0.01 0.30 N'\latex M\_{23}C\_6

0.01 0.25 N'~ACA2CO3

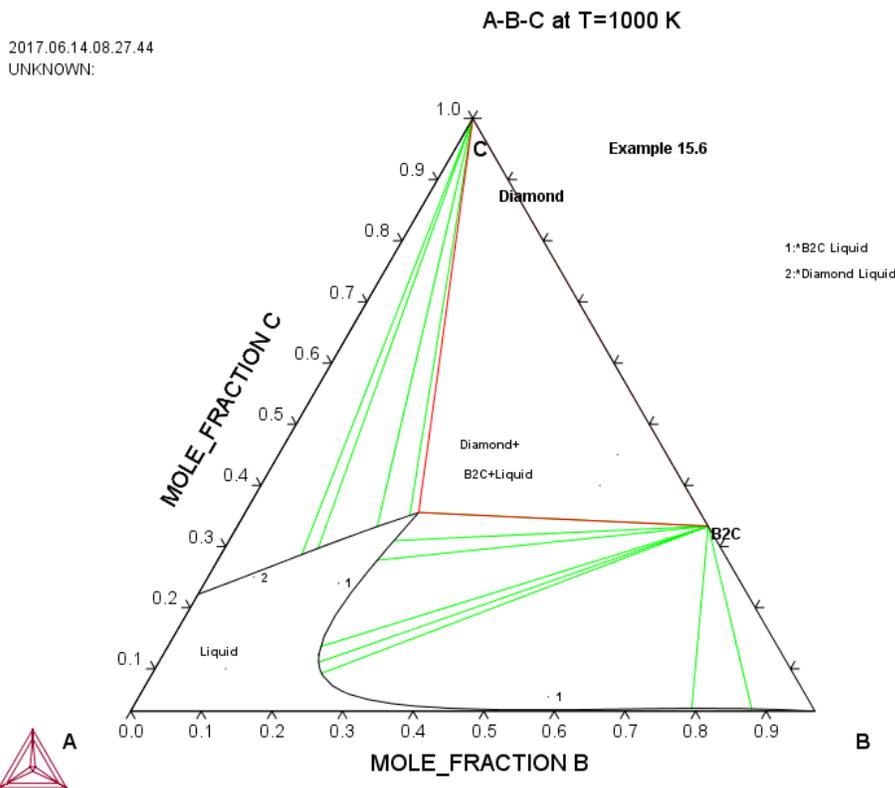
## Example 6 – Triangular Diagrams for Ternary Systems

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file `DATAPLOT 6 - Ternary Plots.EXP` available with your installation. To open the folder: From the Thermo-Calc menu, select **Help → Manuals Folder**.

### Output Example



In the DATAPLOT language, this type of diagram is only available for Console Mode.



### Macro Text

```
$DATAPLOT Example 6 - Ternary Phase Diagram
```

```
PROLOG 6 EXAMPLE 6 0<X<0.969224, 0<Y<1.00000
```

```
XSCALE 0.00000 0.969224
```

```
YSCALE 0.307492E-01 1.00000
```

---

```
XTYPE LINEAR
YTYPE LINEAR
XLENGTH 11.5000
YLENGTH 11.5000
TITLE A-B-C at T=1000 K
XTEXT MOLE_FRACTION B
YTEXT MOLE_FRACTION C
DIAGRAM_TYPE TRIANGULAR YES YES
DATASET 6 Plot a ternary phase diagram
CLIP OFF
0.80 0.90 N'Example 6
0.86 0.31 N'B2C
0.52 0.97 N'Diamond
CHARSIZE 0.25
1.4E+01 1.10E+01 MVA'1:*B2C Liquid
1.4E+01 1.05E+01 MVA'2:*Diamond Liquid
0.10 0.10 N'Liquid
0.48 0.45 N'Diamond+
0.48 0.40 N' B2C+Liquid
5.80E-01 5.40E-02 MWA' 1
1.90E-01 2.40E-01 MWA' 1
0.65E-01 2.50E-01 MWA' 2
CHARSIZE 0.45
-0.10 -0.05 N'A
1.06 -0.05 N'B
0.48 1.05 N'C
CHARSIZE 0.35
```

---

---

\$\$ Calculated A-B-C Phase Equilibrium Data:

\$ PHASE REGION FOR:

\$F0 LIQUID

\$E DIAMOND\_A4

\$F0 B2C

\$ INVARIANT EQUILIBRIUM

COLOR 2

BLOCK X=C1; Y=C2; GOC=C3,WAD;

2.455855989E-01 3.5568857193E-01 M

0.0000000000E+00 9.9999523163E-01

2.455855989E-01 3.5568857193E-01 M

6.666668653E-01 3.3333334327E-01

0.0000000000E+00 9.9999523163E-01 M

6.666668653E-01 3.3333334327E-01

COLOR 1

BLOCKEND

\$ PHASE REGION FOR:

\$E LIQUID

\$F0 B2C

BLOCK X=C1; Y=C2; GOC=C3,WAD;

\$ PLOTTED COLUMNS ARE : X(LIQUID,B) and X(LIQUID,C)

2.2030337155E-01 1.2340000272E-01 M

2.2632879019E-01 1.1058768630E-01

2.3371633887E-01 9.9345825613E-02

2.4253317714E-01 8.9345827699E-02

2.6429468393E-01 7.2744041681E-02

2.8429466486E-01 6.2814079225E-02

---

2.9617273808E-01 5.8319382370E-02  
3.2811737061E-01 4.9470417202E-02  
3.6353862286E-01 4.3130427599E-02  
3.9895987511E-01 3.8979098201E-02  
4.5209178329E-01 3.5266116261E-02  
5.2293431759E-01 3.3152002841E-02  
6.1148744822E-01 3.3077053726E-02  
6.4690870047E-01 3.3490389585E-02  
6.8232995272E-01 3.4017231315E-02  
7.3546189070E-01 3.4814555198E-02  
7.5317251682E-01 3.5033416003E-02  
8.0630439520E-01 3.5373892635E-02  
8.5943627357E-01 3.4983776510E-02  
9.1256815195E-01 3.3575300127E-02  
9.6747112274E-01 3.0857827514E-02  
9.6922445297E-01 3.0749246478E-02  
2.2030337155E-01 1.2340000272E-01 M  
2.1294665337E-01 1.5308913589E-01  
2.1171525121E-01 1.8851040304E-01  
2.1532440186E-01 2.2393165529E-01  
2.2180187702E-01 2.5935292244E-01  
2.2992117703E-01 2.9477417469E-01  
2.3888295889E-01 3.3019542694E-01  
2.4555855989E-01 3.5568857193E-01  
\$ PLOTTED COLUMNS ARE : X(B2C,B) and X(B2C,C)  
6.666668653E-01 3.3333334327E-01 M  
6.666668653E-01 3.3333334327E-01

---

---

\$ TIELINES

COLOR 3

6.6666668653E-01 3.3333334327E-01 M

8.7775242329E-01 3.4625384957E-02

6.6666668653E-01 3.3333334327E-01 M

7.9250496626E-01 3.5342670977E-02

6.6666668653E-01 3.3333334327E-01 M

2.4555824697E-01 3.5568737984E-01

6.6666668653E-01 3.3333334327E-01 M

2.3944084346E-01 9.2542596161E-02

6.6666668653E-01 3.3333334327E-01 M

2.3359020054E-01 3.0954307318E-01

6.6666668653E-01 3.3333334327E-01 M

2.2585247457E-01 2.7766343951E-01

6.6666668653E-01 3.3333334327E-01 M

2.1618695557E-01 1.3621240854E-01

2.2632879019E-01 1.1058768630E-01 M

6.6666668653E-01 3.3333334327E-01

COLOR 1

BLOCKEND

\$ PHASE REGION FOR:

\$F0 DIAMOND\_A4

\$E B2C

BLOCK X=C1; Y=C2; GOC=C3,WAD;

\$ PLOTTED COLUMNS ARE : X(DIAMOND\_A4,B) and X(DIAMOND\_A4,C)

0.0000000000E+00 9.9999523163E-01 M

0.0000000000E+00 9.999976158E-01

---

BLOCKEND

\$ PHASE REGION FOR:

\$E LIQUID

\$F0 DIAMOND\_A4

BLOCK X=C1; Y=C2; GOC=C3,WAD;

\$ PLOTTED COLUMNS ARE : X(LIQUID,B) and X(LIQUID,C)

2.4555855989E-01 3.5568857193E-01 M

2.0263540745E-01 3.3483061194E-01

1.6243904829E-01 3.1375369430E-01

1.2243904918E-01 2.9153180122E-01

8.2439050078E-02 2.6854267716E-01

4.2439054698E-02 2.4548016489E-01

2.4390530307E-03 2.2313812375E-01

2.4999999937E-07 2.2181616724E-01

\$ TIELINES

COLOR 3

0.0000000000E+00 9.9999511242E-01 M

2.3529186845E-01 3.5085919499E-01

0.0000000000E+00 9.9999469519E-01 M

1.9826586545E-01 3.3260950446E-01

0.0000000000E+00 9.9999409914E-01 M

1.3240024447E-01 2.9716023803E-01

0.0000000000E+00 9.9999397993E-01 M

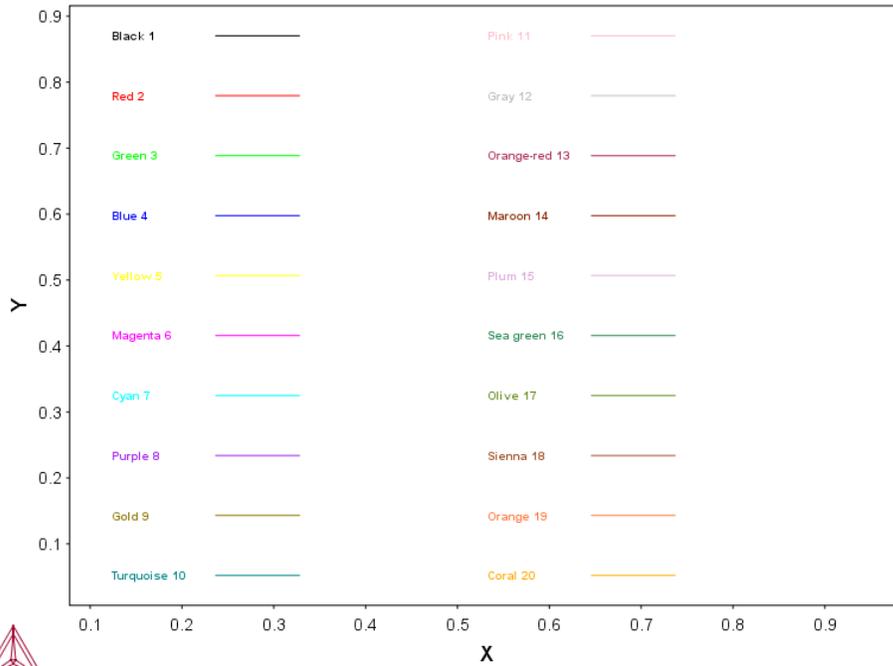
1.1439975351E-01 2.8695335984E-01

BLOCKEND

## Example 7 – Color Codes

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file `DATAPLOT 7 - Color Codes.EXP` available with your installation. To open the folder: From the Thermo-Calc menu, select **Help → Manuals Folder**.

### Output Example



### Macro Text

```

$DATAPLOT Example 7 - DIGLIB Colors
PROLOG 1 DIGLIB Colors
XSCALE 0.0 1.0
YSCALE 0.0 1.0
XTYPE LINEAR
YTYPE LINEAR
XLENGTH 15.5000
YLENGTH 11.5000
TITLE DIGLIB Colors
XTEXT X
YTEXT Y

```

---

```
DATASET 1 DIGLIB Colors:
$ DIGLIB Colors:
CHARSIZE 0.3
SYMBOLSIZE 0.5
COLOR 1
0.05 0.95 N' Black 1
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.95 M
0.275 0.95 D
BLOCKEND
COLOR 2
0.05 0.85 N' Red 2
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.85 M
0.275 0.85 D
BLOCKEND
COLOR 3
0.05 0.75 N' Green 3
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.75 M
0.275 0.75 D
BLOCKEND
COLOR 4
0.05 0.65 N' Blue 4
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.65 M
0.275 0.65 D
BLOCKEND
COLOR 5
0.05 0.55 N' Yellow 5
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.55 M
0.275 0.55 D
BLOCKEND
COLOR 6
```

---

```
0.05 0.45 N' Magenta 6
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.45 M
0.275 0.45 D
BLOCKEND
COLOR 7
0.05 0.35 N' Cyan 7
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.35 M
0.275 0.35 D
BLOCKEND
COLOR 8
0.05 0.25 N' Purple 8
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.25 M
0.275 0.25 D
BLOCKEND
COLOR 9
0.05 0.15 N' Gold 9
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.15 M
0.275 0.15 D
BLOCKEND
COLOR 10
0.05 0.05 N' Turquoise 10
BLOCK X=C1; Y=C2; GOC=C3,N
0.175 0.05 M
0.275 0.05 D
BLOCKEND
COLOR 11
0.5 0.95 N' Pink 11
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.95 M
0.725 0.95 D
BLOCKEND
```

---

---

```
COLOR 12
0.5 0.85 N' Gray 12
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.85 M
0.725 0.85 D
BLOCKEND
COLOR 13
0.5 0.75 N' Orange-red 13
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.75 M
0.725 0.75 D
BLOCKEND
COLOR 14
0.5 0.65 N' Maroon 14
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.65 M
0.725 0.65 D
BLOCKEND
COLOR 15
0.5 0.55 N' Plum 15
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.55 M
0.725 0.55 D
BLOCKEND
COLOR 16
0.5 0.45 N' Sea green 16
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.45 M
0.725 0.45 D
BLOCKEND
COLOR 17
0.5 0.35 N' Olive 17
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.35 M
0.725 0.35 D
```

---

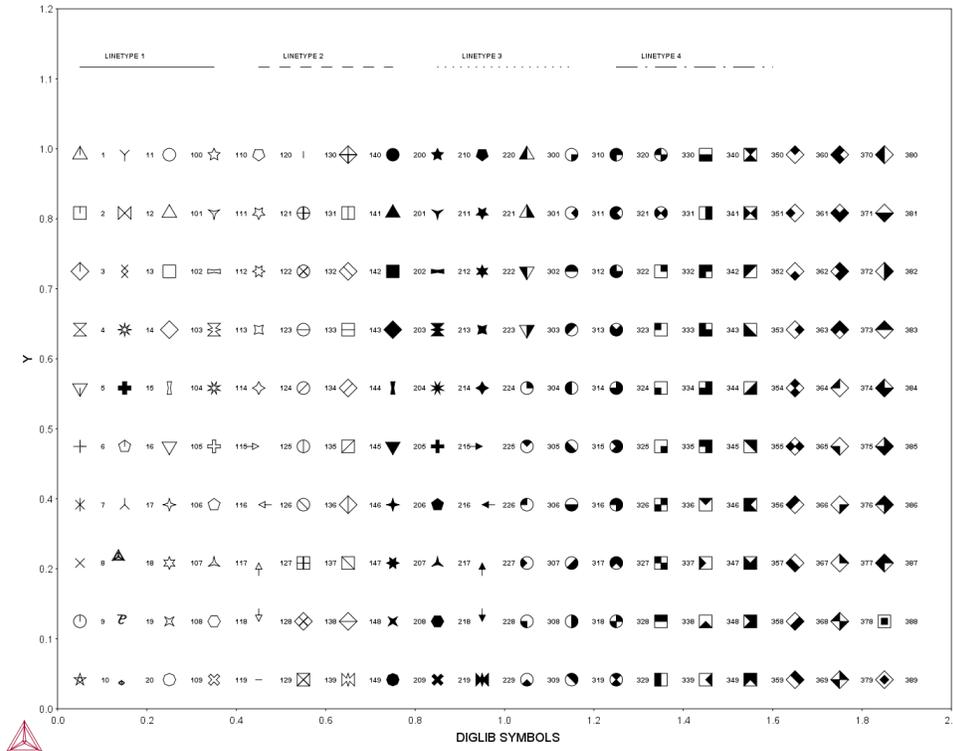
---

```
BLOCKEND
COLOR 18
0.5 0.25 N' Sienna 18
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.25 M
0.725 0.25 D
BLOCKEND
COLOR 19
0.5 0.15 N' Orange 19
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.15 M
0.725 0.15 D
BLOCKEND
COLOR 20
0.5 0.05 N' Coral 20
BLOCK X=C1; Y=C2; GOC=C3,N
0.625 0.05 M
0.725 0.05 D
BLOCKEND
```

## Example 8 - DIGLIB Symbols

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file `DATAPLOT 8 - DIGLIB Symbols.EXP` available with your installation. To open the folder: From the Thermo-Calc menu, select **Help** → **Manuals Folder**.

### Output Example



A default symbol is the current symbol in the run of the software (it is usually the No 1 symbol if the POST-processor is switched on for the first time). S (i.e., # is not specified) means that the current symbol is plotted. S0 (i.e., #=0) means that no symbol is plotted.

### Macro Text

```
$DATAPLOT DIGLIB Symbols 2
PROLOG 1 DIGLIB Symbols 2 0<X<2.0, 0<Y<1.2
XSCALE 0.0 2.0
YSCALE 0.0 1.2
XTYPE LINEAR
YTYPE LINEAR
XLENGTH 15.5000
```

---

```
YLENGTH 11.5000
TITLE DIGLIB Symbols
XTEXT DIGLIB SYMBOLS
DATASET 1 Various DIGLIB Symbols:
$ DIGLIB Symbols:
CHARSIZE 0.3
SYMBOLSIZE 0.5
0.05 0.95 WS1' 1
0.05 0.85 WS2' 2
0.05 0.75 WS3' 3
0.05 0.65 WS4' 4
0.05 0.55 WS5' 5
0.05 0.45 WS6' 6
0.05 0.35 WS7' 7
0.05 0.25 WS8' 8
0.05 0.15 WS9' 9
0.05 0.05 WS10' 10
0.15 0.95 WS11' 11
0.15 0.85 WS12' 12
0.15 0.75 WS13' 13
0.15 0.65 WS14' 14
0.15 0.55 WS15' 15
0.15 0.45 WS16' 16
0.15 0.35 WS17' 17
0.15 0.25 WS18' 18
0.15 0.15 WS19' 19
0.15 0.05 WS20' 20
0.25 0.95 WS100' 100
0.25 0.85 WS101' 101
0.25 0.75 WS102' 102
0.25 0.65 WS103' 103
0.25 0.55 WS104' 104
0.25 0.45 WS105' 105
0.25 0.35 WS106' 106
0.25 0.25 WS107' 107
```

---

0.25	0.15	WS108'	108
0.25	0.05	WS109'	109
0.35	0.95	WS110'	110
0.35	0.85	WS111'	111
0.35	0.75	WS112'	112
0.35	0.65	WS113'	113
0.35	0.55	WS114'	114
0.35	0.45	WS115'	115
0.35	0.35	WS116'	116
0.35	0.25	WS117'	117
0.35	0.15	WS118'	118
0.35	0.05	WS119'	119
0.45	0.95	WS120'	120
0.45	0.85	WS121'	121
0.45	0.75	WS122'	122
0.45	0.65	WS123'	123
0.45	0.55	WS124'	124
0.45	0.45	WS125'	125
0.45	0.35	WS126'	126
0.45	0.25	WS127'	127
0.45	0.15	WS128'	128
0.45	0.05	WS129'	129
0.55	0.95	WS130'	130
0.55	0.85	WS131'	131
0.55	0.75	WS132'	132
0.55	0.65	WS133'	133
0.55	0.55	WS134'	134
0.55	0.45	WS135'	135
0.55	0.35	WS136'	136
0.55	0.25	WS137'	137
0.55	0.15	WS138'	138
0.55	0.05	WS139'	139
0.65	0.95	WS140'	140
0.65	0.85	WS141'	141
0.65	0.75	WS142'	142

---

---

0.65	0.65	WS143'	143
0.65	0.55	WS144'	144
0.65	0.45	WS145'	145
0.65	0.35	WS146'	146
0.65	0.25	WS147'	147
0.65	0.15	WS148'	148
0.65	0.05	WS149'	149
0.75	0.95	WS200'	200
0.75	0.85	WS201'	201
0.75	0.75	WS202'	202
0.75	0.65	WS203'	203
0.75	0.55	WS204'	204
0.75	0.45	WS205'	205
0.75	0.35	WS206'	206
0.75	0.25	WS207'	207
0.75	0.15	WS208'	208
0.75	0.05	WS209'	209
0.85	0.95	WS210'	210
0.85	0.85	WS211'	211
0.85	0.75	WS212'	212
0.85	0.65	WS213'	213
0.85	0.55	WS214'	214
0.85	0.45	WS215'	215
0.85	0.35	WS216'	216
0.85	0.25	WS217'	217
0.85	0.15	WS218'	218
0.85	0.05	WS219'	219
0.95	0.95	WS220'	220
0.95	0.85	WS221'	221
0.95	0.75	WS222'	222
0.95	0.65	WS223'	223
0.95	0.55	WS224'	224
0.95	0.45	WS225'	225
0.95	0.35	WS226'	226
0.95	0.25	WS227'	227

---

---

0.95	0.15	WS228'	228
0.95	0.05	WS229'	229
1.05	0.95	WS300'	300
1.05	0.85	WS301'	301
1.05	0.75	WS302'	302
1.05	0.65	WS303'	303
1.05	0.55	WS304'	304
1.05	0.45	WS305'	305
1.05	0.35	WS306'	306
1.05	0.25	WS307'	307
1.05	0.15	WS308'	308
1.05	0.05	WS309'	309
1.15	0.95	WS310'	310
1.15	0.85	WS311'	311
1.15	0.75	WS312'	312
1.15	0.65	WS313'	313
1.15	0.55	WS314'	314
1.15	0.45	WS315'	315
1.15	0.35	WS316'	316
1.15	0.25	WS317'	317
1.15	0.15	WS318'	318
1.15	0.05	WS319'	319
1.25	0.95	WS320'	320
1.25	0.85	WS321'	321
1.25	0.75	WS322'	322
1.25	0.65	WS323'	323
1.25	0.55	WS324'	324
1.25	0.45	WS325'	325
1.25	0.35	WS326'	326
1.25	0.25	WS327'	327
1.25	0.15	WS328'	328
1.25	0.05	WS329'	329
1.35	0.95	WS330'	330
1.35	0.85	WS331'	331
1.35	0.75	WS332'	332

---

---

1.35 0.65 WS333' 333  
1.35 0.55 WS334' 334  
1.35 0.45 WS335' 335  
1.35 0.35 WS336' 336  
1.35 0.25 WS337' 337  
1.35 0.15 WS338' 338  
1.35 0.05 WS339' 339  
1.45 0.95 WS340' 340  
1.45 0.85 WS341' 341  
1.45 0.75 WS342' 342  
1.45 0.65 WS343' 343  
1.45 0.55 WS344' 344  
1.45 0.45 WS345' 345  
1.45 0.35 WS346' 346  
1.45 0.25 WS347' 347  
1.45 0.15 WS348' 348  
1.45 0.05 WS349' 349  
1.55 0.95 WS350' 350  
1.55 0.85 WS351' 351  
1.55 0.75 WS352' 352  
1.55 0.65 WS353' 353  
1.55 0.55 WS354' 354  
1.55 0.45 WS355' 355  
1.55 0.35 WS356' 356  
1.55 0.25 WS357' 357  
1.55 0.15 WS358' 358  
1.55 0.05 WS359' 359  
1.65 0.95 WS360' 360  
1.65 0.85 WS361' 361  
1.65 0.75 WS362' 362  
1.65 0.65 WS363' 363  
1.65 0.55 WS364' 364  
1.65 0.45 WS365' 365  
1.65 0.35 WS366' 366  
1.65 0.25 WS367' 367

---

---

```
1.65 0.15 WS368' 368
1.65 0.05 WS369' 369
1.75 0.95 WS370' 370
1.75 0.85 WS371' 371
1.75 0.75 WS372' 372
1.75 0.65 WS373' 373
1.75 0.55 WS374' 374
1.75 0.45 WS375' 375
1.75 0.35 WS376' 376
1.75 0.25 WS377' 377
1.75 0.15 WS378' 378
1.75 0.05 WS379' 379
1.85 0.95 WS380' 380
1.85 0.85 WS381' 381
1.85 0.75 WS382' 382
1.85 0.65 WS383' 383
1.85 0.55 WS384' 384
1.85 0.45 WS385' 385
1.85 0.35 WS386' 386
1.85 0.25 WS387' 387
1.85 0.15 WS388' 388
1.85 0.05 WS389' 389
```

```
$ DIGLIB Line Types:
```

```
LINETYPE 1
```

```
BLOCK X=C1; Y=C2; GOC=C3,DWA
```

```
0.05 1.10 M
```

```
0.35 1.10
```

```
BLOCKEND
```

```
LINETYPE 2
```

```
BLOCK X=C1; Y=C2; GOC=C3,DWA
```

```
0.45 1.10 M
```

```
0.75 1.10
```

```
BLOCKEND
```

```
LINETYPE 3
```

```
BLOCK X=C1; Y=C2; GOC=C3,DWA
```

0.85 1.10 M

1.15 1.10

BLOCKEND

LINETYPE 4

BLOCK X=C1; Y=C2; GOC=C3,DWA

1.25 1.10 M

1.60 1.10

BLOCKEND

0.10 1.12 W' LINETYPE 1

0.50 1.12 W' LINETYPE 2

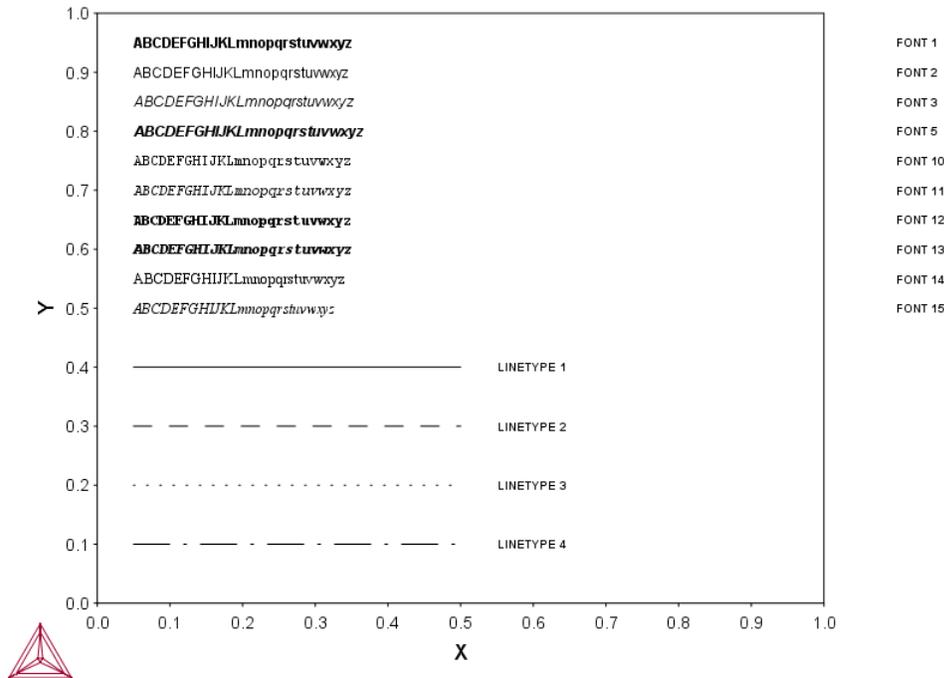
0.90 1.12 W' LINETYPE 3

1.30 1.12 W' LINETYPE 4

## Example 9 - DIGLIB Fonts and Lines

The macro text below produces the example output. You can copy the text into your own EXP file, or use the example file `DATAPLOT 9 - DIGLIB Fonts.EXP` available with your installation. To open the folder: From the Thermo-Calc menu, select **Help** → **Manuels Folder**.

### Output Example



### Macro Text

```
$DATAPLOT Example 9 - DIGLIB Fonts
PROLOG 1 DIGLIB Fonts & Line Types 0<X<1, 0<Y<1
XSCALE 0.0 1.0
YSCALE 0.0 1.0
XTYPE LINEAR
YTYPE LINEAR
XLENGTH 11.5000
YLENGTH 11.5000
TITLE DIGLIB Fonts
XTEXT X
YTEXT Y
```

---

DATASET 1 Various DIGLIB Fonts and Line Types:

\$ DIGLIB Fonts:

CHARSIZE .35

FONT 1

0.05 0.95 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 2

0.05 0.90 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 3

0.05 0.85 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 5

0.05 0.80 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 10

0.05 0.75 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 11

0.05 0.70 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 12

0.05 0.65 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 13

0.05 0.60 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 14

0.05 0.55 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 15

0.05 0.50 N'ABCDEFGHIJKLMnopqrstuvwxyz

FONT 1

CLIP OFF

CHARSIZE .30

1.10 0.95 N'FONT 1

1.10 0.90 N'FONT 2

1.10 0.85 N'FONT 3

1.10 0.80 N'FONT 5

1.10 0.75 N'FONT 10

1.10 0.70 N'FONT 11

1.10 0.65 N'FONT 12

1.10 0.60 N'FONT 13

1.10 0.55 N'FONT 14

---

```
1.10 0.50 N'FONT 15
CLIP ON
$ DIGLIB Line Types:
LINETYPE 1
BLOCK X=C1; Y=C2; GOC=C3,DWA
0.05 0.40 M
0.50 0.40
BLOCKEND
$DRAWLINE 0.05,0.40 0.5,0.40
LINETYPE 2
BLOCK X=C1; Y=C2; GOC=C3,DWA
0.05 0.30 M
0.50 0.30
BLOCKEND
$DRAWLINE 0.05,0.30 0.5,0.30
LINETYPE 3
BLOCK X=C1; Y=C2; GOC=C3,DWA
0.05 0.20 M
0.50 0.20
BLOCKEND
$DRAWLINE 0.05,0.20 0.5,0.20
LINETYPE 4
BLOCK X=C1; Y=C2; GOC=C3,DWA
0.05 0.10 M
0.50 0.10
BLOCKEND
$DRAWLINE 0.05,0.10 0.5,0.10
0.55 0.40 N' LINETYPE 1
0.55 0.30 N' LINETYPE 2
0.55 0.20 N' LINETYPE 3
0.55 0.10 N' LINETYPE 4
```

# Pourbaix Diagrams

Educational Material



---

# Introduction to Pourbaix Diagrams

---

**In this section:**

---

About Pourbaix Diagrams in Thermo-Calc .....	3
Pourbaix Diagrams .....	5
Working with Pourbaix Diagrams .....	7
Key Concepts .....	9
Basic Settings and Definitions .....	11

## About Pourbaix Diagrams in Thermo-Calc

---

This guide is for Thermo-Calc software users who know the basics of thermodynamics and want to know more about [Pourbaix Diagrams](#).

You are introduced to the concept of a Pourbaix diagram and shown how to interpret such diagrams through a series of examples.

Materials corrosion occurs almost everywhere. It may lead to serious material damages, unexpected application failures, tremendous economic costs and environmental degradations. Consequently, scientists and engineers must often conduct expensive and time-consuming corrosion experiments as part of failure analyses, risk evaluations, quality improvements and application enhancements.

Under certain conditions, when a metal or alloy is exposed to an aqueous solution with a concentration of inorganic/organic mixture, corrosion phenomena occur at a corresponding degree. During corrosion, some metallic phases dissolve, the metal or alloy surface gets damaged and some secondary solid phases form at the solid-liquid interfaces (such as oxides, hydroxides, silicates, sulphides, sulphates, carbonates, nitrates, phosphates, borates, or halides). Such corrosive chemical or electrochemical reactions can be studied by means of the so-called Pourbaix diagrams if the reactions reach their equilibrium states [1973/1974Pou; 2003Cra; 2011Ver; 2011Tho].

### GES5 and GES6 Calculation Engine Version

As of Thermo-Calc version 2019b, the default calculation engine is set to GES6. For POURBAIX module users, you need to use GES5. When you try to enter this module, you are instructed to switch to GES5 using the SET\_GES\_VERSION command.



[About the Gibbs Energy System \(GES\) Module](#)



To globally set the default from GES6 back to GES5, go to the **Options** window [Global General Settings](#) tab and click **Version 5** for the *Preferred Gibbs Energy System* setting. The [SET\\_GES\\_VERSION](#) Console Mode command is used to change GES versions for a single Thermo-Calc session.



Contact Thermo-Calc Software AB at [info@thermocalc.com](mailto:info@thermocalc.com) for more information about the changes or if you encounter any issues you think are related to the new version of the calculation engine.



GES6 is a re-write and new implementation of the Gibbs Energy System module of Thermo-Calc and it is the default engine. The previous version, GES5, is still available and will continue to work in the background of the program where necessary. The main purpose of GES6 is to support faster development of new features. However, not all GES5 functionality is implemented in GES6. In such cases (except for the POURBAIX module), GES6 falls back silently and uses functionality from GES5 in the background.

## References

- [1973Pou] M. Pourbaix, Lectures on Electrochemical Corrosion (Plenum Press, New York, 1973).
- [1974Pou] M. Pourbaix, Atlas of Electrochemical Equilibria in Aqueous Solutions (National Association of Corrosion Engineers, Houston, TX, ed. 2, 1974).
- [2003Cra] S. D. Cramer, B. S. Covino Jr., Eds., Corrosion: Fundamentals, Testing, and Protection: ASM Handbook, Volume 13A (ASM International, 2003).
- [2011Ver] E. D. Verink Jr., "Simplified procedure for constructing Pourbaix diagrams", in Uhlig's Corrosion Handbook, R. W. Revie, Ed. (John Wiley & Sons, Inc., Hoboken, NJ, USA, ed. 3, 2011), pp. 93–101.
- [2011Thom] W. Thompson, M. H. Kaye, C. W. Bale, A. Pelton, "Pourbaix diagrams for multielement systems" in Uhlig's Corrosion Handbook, Revie, Ed. (John Wiley & Sons, Hoboken, NJ, USA, 2011), pp. 103–109.

## Pourbaix Diagrams

---

Marcel Pourbaix applied thermodynamics to predict materials corrosion resistance. He determined the phase stability relations in terms of varied pH and Eh values for an interaction system of metal and pure water or dilute aqueous solution. He presented the stability regions of metal and secondary phases (such as metal-oxides/hydroxides) on a pH-Eh diagram, which is now known as a Pourbaix diagram.

A Pourbaix diagram is a kind of phase diagram that shows the stability boundaries for a metal-aqueous interaction system. The phase boundaries are shown as a function of pH (acidity) and Eh (standard hydrogen electronic potential). An aqueous solution phase is always present in such a system. At a given pH and Eh, a metal may lose its stability to a soluble or corrosive aqueous solution, or be in equilibrium with either the aqueous solution (insoluble/immune) or with a secondary-phase file that has formed (consisting of oxides, hydroxides, sulphides or other solids). In the latter case, further dissolution of the passive or protective metal is prevented.

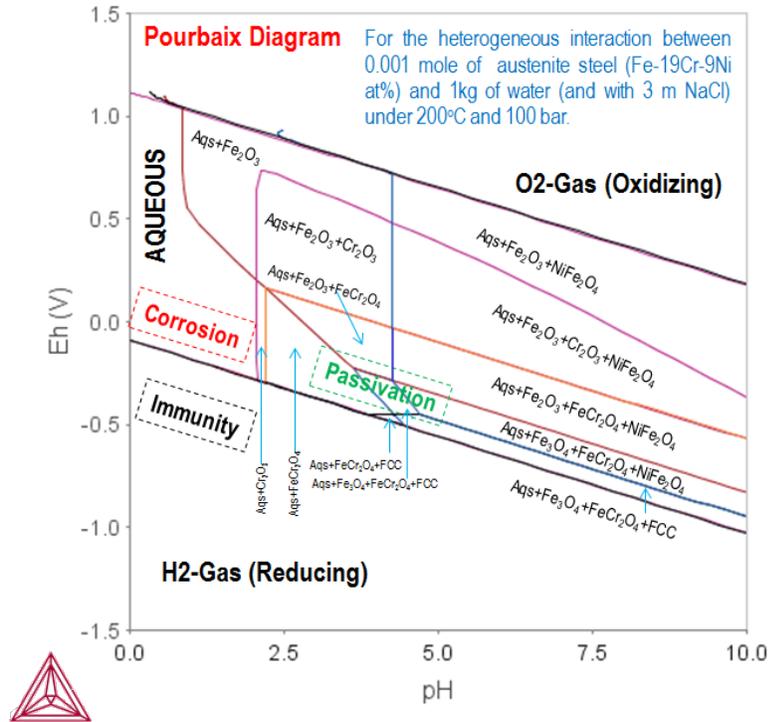
The speciation and partition in the aqueous solution and the interacting phases depend not only on pH and Eh, but also on other factors such as the bulk composition, temperature and pressure in the system. The interacting phases may be gas mixtures, stoichiometric solids or solid solutions.

A Pourbaix diagram is divided in regions of *immunity*, *corrosion* and *passivity*. These regions provides information about the stability of a particular metal or alloy in a specific aqueous electrochemical environment under certain pH, Eh, pressure and temperature conditions.

- The immunity region is the region in which there is no metal dissolution.
- The corrosion region is the region in which there is active metal dissolution.
- The passivation region is the region in which a protective metal-oxide film that prevents metal dissolution is formed.

The following shows these three regions in a Pourbaix diagram for the heterogeneous interaction between 0.001 mole of austenite steel (Fe-19Cr-9Ni [at%]) and 1 kg of water (with 3 m NaCl), at 200° C and 100 bar:

## Pourbaix Diagrams in Multicomponent Systems



Depending on the system's bulk chemical composition, secondary phases may form during the transformation process. These can not only be oxides, but also be, for example, hydroxides, sulphides, sulphates, carbonates, nitrates, phosphates, borates, silicates, hydrous silicates and halides. Information about these secondary phases may help one understand the passivation behaviours in corrosion processes. For example, it may be important to understand whether any secondary phases are formed in different pH-Eh regions during hydrolysis, oxidation, reduction or other reaction processes.

In a Pourbaix diagram, the predominant ion boundaries are often represented by dashed and straight lines imposed on a traditional Pourbaix diagram for a simple interaction system (Pourbaix, 1973, 1974; Cramer and Covino, 2003; Verink, 2011; Thompson et al., 2011). Such a predominance boundary is normally an equal molality (or equal activity) line for a pair of two most-concentrated aqueous solute species (charged or neutral) involving a common metal. A line is then simply determined by a reaction constant of its corresponding electrochemical reaction in the homogeneous aqueous solution phase. However, this is not the case in Thermo-Calc.



You cannot get any information about predominant ion boundaries from a Pourbaix diagram in Thermo-Calc.

## Working with Pourbaix Diagrams

With the Thermo-Calc software package, Pourbaix diagrams can be calculated over a wide range of pressure, temperature and composition conditions for complex heterogeneous interactions between multicomponent primary alloy solution or compound phases, concentrated aqueous solutions (with dissolved inorganic and organic substances), complicated secondary solids and oxidizing/reducing gaseous mixtures. In the Console Mode, diagrams can be calculated and plotted either using the response-driven POURBAIX module or directly using the DATA, GIBBS, POLY and POST modules.



This command or module is not available with GES6.



GES6 is a re-write and new implementation of the Gibbs Energy System module of Thermo-Calc and it is the default engine. The previous version, GES5, is still available and will continue to work in the background of the program where necessary. The main purpose of GES6 is to support faster development of new features. However, not all GES5 functionality is implemented in GES6. In such cases (except for the POURBAIX module), GES6 falls back silently and uses functionality from GES5 in the background.

It is possible in Thermo-Calc to set aqueous properties other than pH and Eh as axis variables. For example, the following properties can also be set as axis variables:  $A_h$  (electronic affinity),  $p_e$  (electronic activity  $\log_{10}ACRe$ ),  $I_S$  (ionic strength),  $T_M$  (total aqueous concentration, in molality),  $O_C$  (Osmotic coefficient), activity or activity coefficient of solvent H<sub>2</sub>O, and activities or activity coefficients or concentrations in m (molality) of charged or neutral solute species.

Beside a concentrated aqueous solution phase handled by a proper thermodynamic model named SIT, HKF or PITZ, it is also possible to consider complex alloy solution phases or compound phases, as well as secondary solid phases and gaseous mixtures, which are treated by specific solution models.

Most frequently, a Pourbaix diagram is calculated for a specific alloyed phase with the initial alloying composition, such as the FCC solution phase in an austenite steel. However, it is possible to calculate a Pourbaix diagram for two co-existing phases, such as the FCC+BCC solution phases in a duplex steel for example, or even for more than two co-existing phases in a steel/alloy.



Each point on the line in a pH-Eh diagram in Thermo-Calc represents a certain minimum state of Gibbs energy. When a line is crossed, a phase transformation should occur. This means that a Pourbaix diagram in Thermo-Calc does not provide any information about any predominant ion boundary.

---

## Key Concepts

---

The following are two key concepts important for understanding how aqueous solutions behave and how an aqueous-bearing heterogeneous interaction system is properly defined and calculated in Thermo-Calc.

### Effective Interaction Ratio

A pH-Eh plot is always related to a certain amount of initial alloys or other condensed materials that has effectively reacted with an aqueous solution in the system. This is the amount of condensed material that is fully in equilibrium with the defined aqueous solution phase. The amount is specified relative to an aqueous solution that is normally comprised of 1 kg of water with certain specified solute concentrations at certain temperature and pressure conditions. (This is why a calculated Pourbaix diagram is typically presented for an initial amount of the interacting metal or alloy at a certain level, such as  $10^{-6}$ ,  $10^{-3}$ , 0.1 or 1 mole of metal or alloy.) It is called the *Effective Interaction Ratio* (between the initial alloy or alloyed phases and the initial aqueous solution) and it is expressed in terms of molality (mol/kg).

The Effective Interaction Ratio is important for two reasons. First, the ratio has implications for kinetic or dynamic effects such as chemical reaction mechanism and kinetics, fluid flow dynamics, surface area and interaction time. Secondly, the ratio, being expressed as  $10^{-6}$  mole of metal (or alloy), is the solubility limit that can be detected for cathodic corrosion protection by immunity.

One should always be careful when setting initial amounts and compositions in the original condensed materials, as well as when setting the initial concentrations of dissolved solutes in the original aqueous solution phase. It is often useful to make a series of calculations for different levels of initial amount of the interacting metal/alloy while the conditions are fixed for other settings (such as initial aqueous concentration, pressure, temperature, pH, and Eh).

### Solubility

When a heterogeneous equilibrium has a dissolving solution or mixture phase and a stoichiometric or solution phase, then the concept of solubility becomes important. A solubility of a phase (the solute) is its property of dissolving in the solvent phase. This concept concerns the constitution of a phase and is applied where one or several of the constituents are dominant (which is highly concentrated and dissolving) while there are only small amounts of the other remaining species (which are less concentrated and dissolved). The dissolving solution or mixture phase can be liquid, gas, aqueous or solid, as long as it has dissolving capacity. The stoichiometric or solution phase has some constituents which tend to be dissolved into the dissolving solution or mixture phase.

---

## **Key Points about Solubility**

- Under certain temperature, pressure and composition conditions, a Fe- or Cr-dominant BCC phase can dissolve certain amounts of Ni and C from a carbide phase such as  $M_23C_6$ ,  $M_7C_3$  or  $M_3C$ . The Ni and C elements in the carbides have the solubility defined relative to the BCC phase.
- An aqueous solution phase is always dominant in the solvent water, that is,  $H_2O$ . Under specific temperature, pressure and aqueous composition conditions, any other element (such as Fe and C) or substance (such as a pure  $SO_2$  gas, stoichiometric phase  $Cu_2S$  and solution phase  $(Fe,Ni)_1(O,Va)_1$ ) have a certain solubility limit in the defined aqueous solution.
- Under certain temperature and pressure conditions, and under given certain concentrations of other dissolved species in the mixture, an  $O_2$ -dominant gaseous mixture phase can dissolve certain amounts of  $Fe^{+2}$  or  $Fe^{+3}$  species from magnetite ( $Fe_3O_4$ ). The magnetite solid has solubility defined relative to the gaseous mixture under the given conditions.
- Under certain temperature and pressure condition, under specific concentrations of other dissolved species in the liquid phase, a Fe-dominant liquid mixture phase can dissolve certain amounts of, for example, Cr and O. The Cr and O components have the solubility defined in the liquid mixture.

## Basic Settings and Definitions

In a homogeneous aqueous solution or an aqueous-bearing heterogeneous interaction system, the most essential definition is for system-components there must be  $H_2O$ ,  $H+1$  and  $Z_E$  (electron) plus those for elements dissolved in aqueous solution (such as  $Na$ ,  $Cl$ ,  $S$ ) and associated in interacting metals/alloys (such as  $Fe$ ,  $Cr$ ,  $Mn$ ,  $Mg$ ,  $Ni$ ,  $Al$ ,  $Si$ ,  $Zn$ ). Three fundamental phases in a system are the `AQUEOUS` solution, the `GAS` mixture and the `REF_ELECTRODE`. The `REF_ELECTRODE` phase is used for setting the reference state for electrostatic potential in the system and for calculating the Eh condition (defined as  $MUR(Z_E) / RNF$ ). Other phases should be appropriately selected and retrieved from critically-assessed databases that cover not only the target phases (solution or stoichiometric) in interacting metals/alloys but also the secondary phases (solution or stoichiometric). The two public databases PAQ and PAQS are specially designed as single-database choices that cover all kinds of phases necessary for calculations. However, these databases are each limited to a framework of a very small number of elements. When it comes to simulations of complex multicomponent systems with a wide variety of elements and phases, the thermodynamic data must be selected and retrieved from several databases: `AQUEOUS` solution and `REF_ELECTRODE` phases must be retrieved from TCAQ or AQS; alloy phases (such as `FCC_A1`, `BCC_A2`, `HCP_A3`, `CEMENTITE`) from alloy solution databases (such as SSOL for general alloy phases, TCFE for steel/Fe-alloy phases, TCAL for Al-based alloy phases; TCMG for Mg-based alloy phases, and TCNI for Ni-based superalloy phases); gaseous mixture phase and secondary phases from specific substance or solution databases (such as SSUB for `GAS` phase and various solid compound phases or TCOX for oxide solution phases). Note that the `REF_ELECTRODE` phase should always be suspended in equilibrium calculations, while `GAS` phase could be set as `ENTERED`, `SUSPENDED` or `DORMANT`, depending on the purpose of the calculation.

When defining an interaction system, the initial condition for the  $H_2O$  component is always set as 1 kg of water. The initial composition conditions for dissolving and interacting elements are normally defined in moles (such as  $n(Fe) = 0.009$ ,  $n(Cr) = 5E-4$ ,  $n(Ni) = 3E-4$ ,  $n(Mn) = 5E-5$ ,  $n(S) = 5E-5$ ,  $n(Na) = 3$ ,  $n(Cl) = 3$ ). This makes it straightforward and convenient to count various related aqueous solution properties based on molality. The initial conditions for the  $H+1$  and  $Z_E$  components can be given as molar compositions (such as  $n(H+1) = 0$ ,  $n(Z_E) = 0$ ) or their activities or potentials (such as  $\ln ACR(H+1) = -9.21$ ,  $MUR(Z_E) = 8400$ ).

The pH and Eh properties of the aqueous solution in the interacting system are always defined in the following way:

$$pH = -\log_{10}[ACR(H+1, AQUEOUS) * AH_2O] \quad Eh = MUR(Z_E) / RNF$$

The symbol  $AH_2O$  is the molecular weight of solvent  $H_2O$  (equals 55.508435) and  $RNF$  is the Faraday constant (equals 96485.309).

---

The activity of the solvent water ( $ACRH_2O$ ,  $A_w$ ), the osmotic coefficient of aqueous solution ( $OSMC$ ,  $O_s$ ), electronic affinity ( $A_h$ ), electronic activity  $\log_{10}ACRe$  ( $p_e$ ), ionic strength ( $IS$ ), total aqueous concentration, in molality ( $T_M$ ) and total alkaline concentrations under two definitions ( $A_{t1}/A_{t2}$ ) are calculated and listed for each equilibrium state.

POLY3 calculations for mass balances in Thermo-Calc are always based on site-fractions. Consequently, when functions for describing various properties of aqueous solutes are defined, such as molality ( $ML_i$ ), activity coefficient ( $RC_i$ ) and activity ( $AI_i$ ), they should be converted to molality-based quantities:

$$ML_i = Y(AQUEOUS, i) * AH_2O / YH_2O$$

$$RC_i = ACR(i, AQUEOUS) * YH_2O / Y(AQUEOUS, i)$$

$$AI_i = ACR(i, AQUEOUS) * AH_2O$$

Here,  $YH_2O$  is the site-fraction of solvent H<sub>2</sub>O and  $AH_2O$  equals 55.508435.

Many more variables, functions and tables can be entered for various purposes. For instance, an equilibrium constant for a homogeneous reaction or a solubility product for a solid dissolution reaction can be entered.

---

# Thermodynamic Data Requirements

---

**In this section:**

---

Required Thermodynamic Data .....	14
Including or Excluding the Gas Phase .....	15

---

## Required Thermodynamic Data

---

To calculate a Pourbaix diagram, thermodynamic data for at least the following four types of phases must be available:

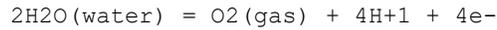
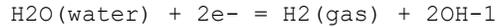
- The aqueous solution phase which applies a certain aqueous solution model. In Thermo-Calc, information about this phase could be retrieved from, for instance, the TCAQ or AQS database (these also include data for the `REF_ELECTRODE` phase that is used for setting the reference state of the standard hydrogen electrode).
- The alloy solution and/or compound phases for the primary matrix phases (and possibly also for the precipitated phases in the alloy. Examples of such phases include FCC\_A1, BCC\_A2, HCP\_A3, SIGMA and CEMENTITE. In Thermo-Calc, information about such phases could be retrieved from, for example, the TCFE database for steels/Fe-alloys, the TCNI database for Ni-based superalloys, the TCAL database for Al-based alloys, the TCMG database for Mg-based alloys, the TCSLD for Sn-/Au-/Bi-/Zn-based solder alloys or the SSOL for general alloys.
- The secondary solid phases that would form as a result of the heterogeneous chemical or electrochemical reactions. These phases could be, for example, oxides, hydroxides, silicates, hydrous silicates, sulphides, sulphates, carbonates, nitrates, phosphates, borates or halides. In Thermo-Calc, information about such phases could be retrieved from, for example, the SSUB database for pure solid compounds or the TCOX database for complex oxide solids.
- The gaseous mixture phase. Information about this phase could be retrieved from, for example, the SSUB database. However, note that it is also possible to perform Pourbaix diagram calculations that ignore the gaseous mixture phase.

All this thermodynamic data about the various phases must be critically assessed and internally consistent. Furthermore, when the data is retrieved from several databases, the data taken from each database must be consistent with the data taken from the other databases. When Pourbaix diagrams and other diagrams of steel corrosion processes are calculated, it is typically recommended that the TCAQ, TCFE, SSUB and TCOX databases or the AQS, TCFE, SSUB and TCOX are used in combination.

The public PAQ database contains thermodynamic data for all four phase types and can be used for calculating Pourbaix diagrams. They have been designed specifically for the purpose of demonstrating the POURBAIX module functionality in Thermo-Calc. However, these databases only allow you to perform test calculations for simple cases with major phases in which only a few elements can be considered.

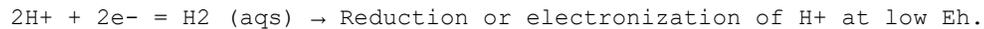
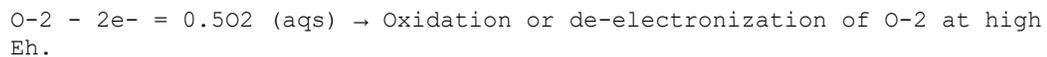
## Including or Excluding the Gas Phase

Under defined pressure and temperature conditions, the solvent water's thermodynamic stability limits are determined by the following two electrochemical reactions:

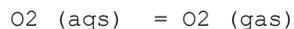


The first reaction describes the formation of H<sub>2</sub>-dominated gaseous mixture, under reducing conditions. The second reaction describes the formation of O<sub>2</sub>-dominated gaseous mixture, under oxidising conditions.

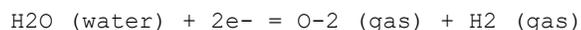
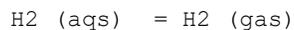
If the system reaches global equilibrium, then the water component is electrolyzed to H<sup>+</sup> and O<sup>2-</sup> at all pH conditions. The degree of electrolysis depends on the pH value in the aqueous solution phase. If Eh gets high enough, then the O<sup>2-</sup> anion is oxidized to O<sub>2</sub> (aq). On the other hand, if Eh gets low enough, then the H<sup>+</sup> cation is reduced to H<sub>2</sub> (aq). The major electrolysis and redox reactions are the following:



At a critically high Eh value under a given pH condition, an aqueous solution phase with a high enough O<sub>2</sub> activity becomes less stable than an O<sub>2</sub>-dominated gaseous mixture phase. At this point, the gas phase replaces the aqueous solution phase in the system. The replacement process can be characterised by the following phase-transformation on the aqueous-gas phase boundary and oxidation of remaining water:



Similarly, at a critically low Eh value under a given pH condition, an aqueous solution phase with a high enough H<sub>2</sub> activity becomes less stable than a H<sub>2</sub>-dominated gaseous mixture phase. At this point, the gas phase replaces the aqueous solution phase in the system, through the following phase-transformation on the aqueous-gas phase boundary and reduction of remaining water:



---

The phase transformation from an aqueous solution phase to an O<sub>2</sub>- or H<sub>2</sub>-dominated gaseous mixture phase also depends on the total molar Gibbs energies of the phases (which are complex functions of the phase constituents, the temperature and the pressure). The Gibbs energy minimization technique used in Thermo-Calc ensures that the phase transformation is accurately simulated.

The following Pourbaix diagrams show the result of a calculation where the gaseous mixture phase was included. In the upper region (high Eh) and lower region (low Eh), the aqueous solution phase has been oxidized and reduced, respectively, to an O<sub>2</sub>- or H<sub>2</sub>-dominated gaseous mixture phase.

The two example diagrams show that the upper and lower boundaries between the aqueous phase (water) and the gas phase (dominated by either O<sub>2</sub> or H<sub>2</sub>) can shift when solutes dissolve or when the temperature and pressure change.

Accordingly, if one does not take a gaseous mixture phase into account when performing a calculation of Pourbaix diagram (as above), then the aqueous solution phase may end up with an extremely high O<sub>2</sub> (aq) concentration at high Eh condition, or an extremely high H<sub>2</sub> (aq) concentration at low Eh condition. Under either of these two extreme circumstances, the concept of “aqueous solution phase” is no longer valid and consequently no proper aqueous solution model can actually be applied. Therefore, from a restrictive thermodynamic equilibrium point of view, one shall normally include a gaseous mixture phase in an aqueous-bearing heterogeneous interaction system. This is true for all types of equilibrium calculations for an aqueous-bearing heterogeneous interaction system, not only of Pourbaix diagram calculations.

Hence, to perform a completely accurate Pourbaix diagram calculation, the gaseous mixture phase must be taken into account. However, since these electrochemical reactions normally have much higher kinetic barriers and are slower than other electrochemical and chemical reactions in the interaction system, one can sometimes ignore the gaseous mixture phase in the calculation. Note that if the gaseous mixture phase is ignored in this way, then the Pourbaix diagram does not show the H<sub>2</sub>O-stability limiting lines.

**Examples**

Pure water at 25 °C and 1 bar

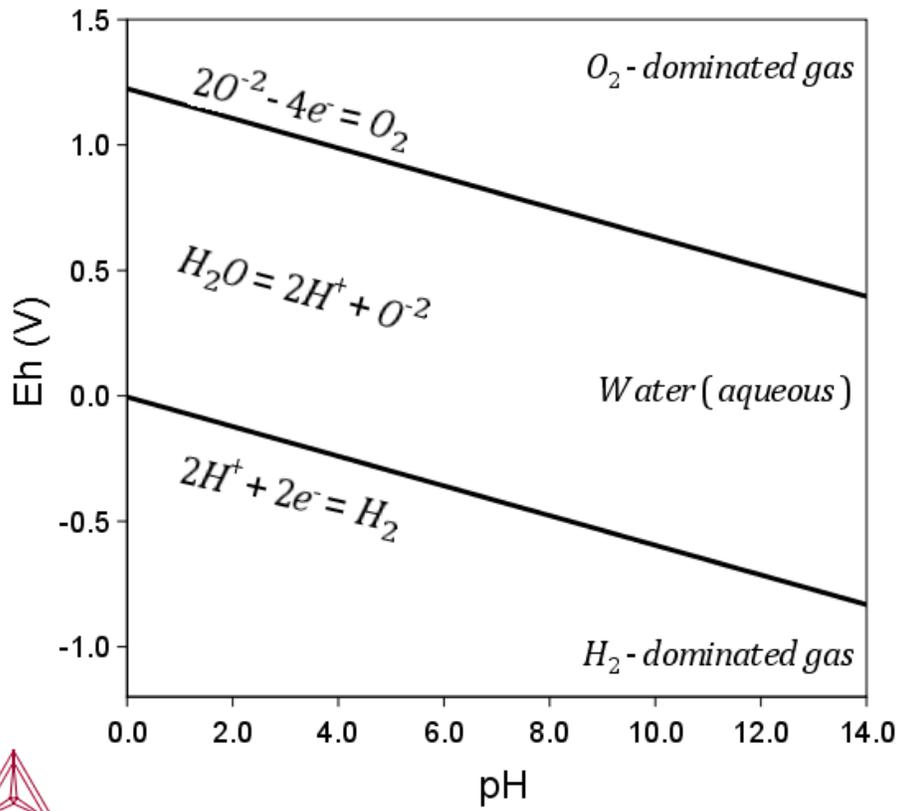


Figure 1: A pH-Eh diagram for pure water at 25° C and 1 bar.

3mNaCl-0.001mCO2-0.001mSO2 aqueous, 150 °C and 100 bar

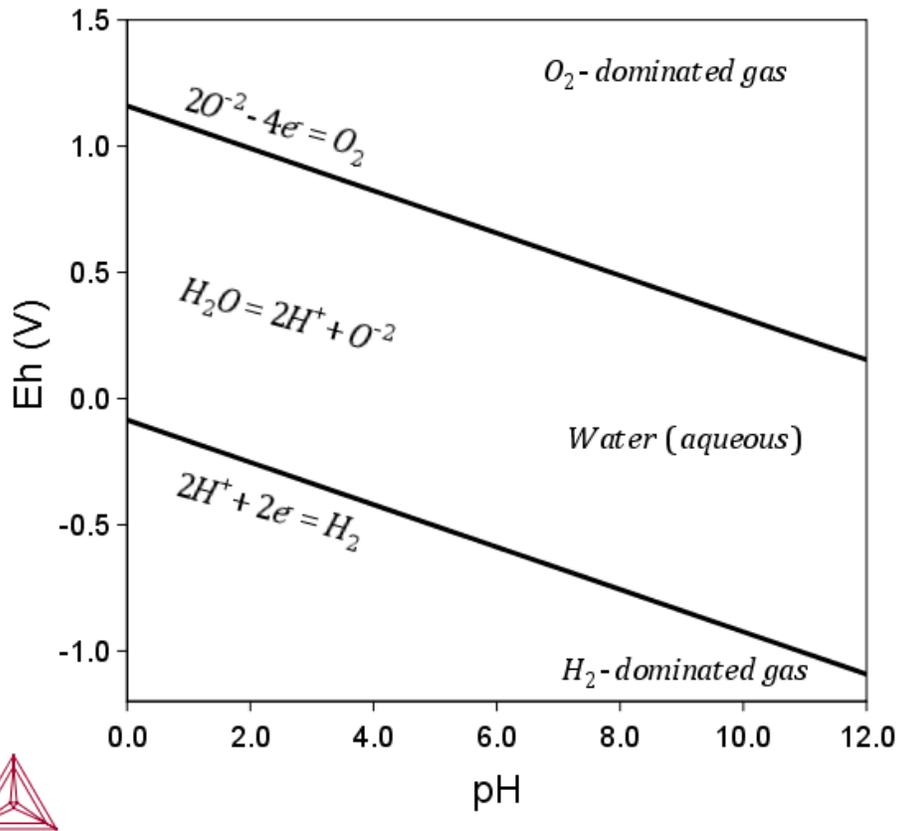


Figure 2: A pH-Eh diagram for 3mNaCl-0.001mCO2-0.001mSO2 aqueous solution at 150° C and 100 bar.

---

## Examples of Pourbaix Diagrams for Fe

---

The following two topics are examples of Pourbaix diagrams for Fe. For the first example, the calculations on which the diagrams are based have not taken the gas phase into account. The diagrams in the second example are based on calculations that have taken the gas phase into account.

**In this section:**

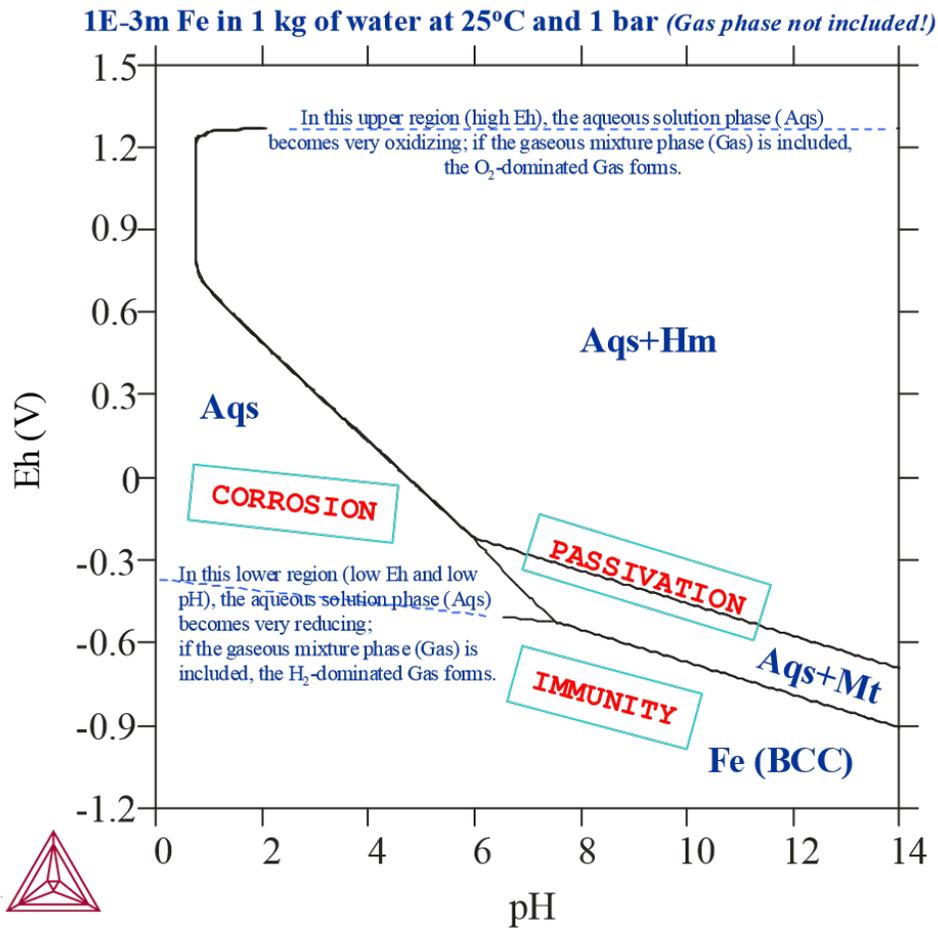
---

Pourbaix Diagrams with Gas Phase Excluded .....	20
Pourbaix Diagrams with Gas Phase Included .....	21

## Pourbaix Diagrams with Gas Phase Excluded

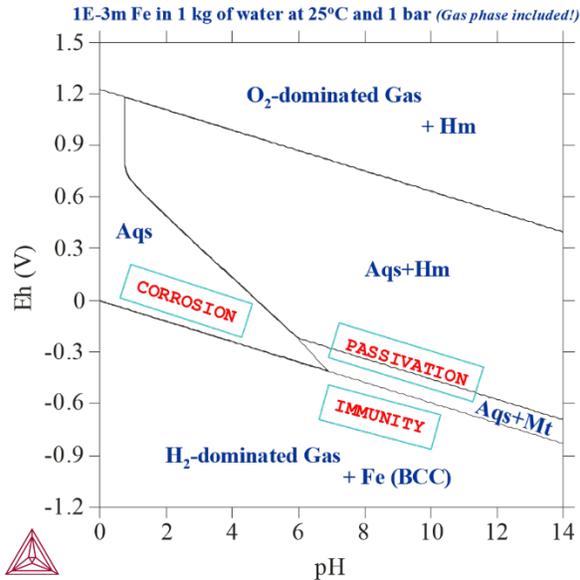
This is an example of a Pourbaix diagram of Fe, in which the gas phase has not been considered in the calculation. The system has 0.001 mole of Fe in 1 kg of pure water at a temperature of 25°C and a pressure of 1 bar. Magnetite (Mt, Fe<sub>3</sub>O<sub>4</sub>) and hematite (Hm, Fe<sub>2</sub>O<sub>3</sub>) co-exist with the Fe-containing dilute aqueous solution. The formation of magnetite and hematite represent the passivation of iron in the upper right pH-Eh field.

If pH is low and Eh is relatively high, then Fe eventually completely dissolves into the aqueous solution phase. Under conditions of low Eh, Fe remains in its stable solid state, the BCC phase, and neither dissolves into water nor into alternates to Fe (the immunity of Fe).



## Pourbaix Diagrams with Gas Phase Included

With all the possible redox reactions involving the aqueous solution phase and gaseous mixture phase being considered in the equilibrium system, that is, the gaseous mixture phase is included in the calculation, the Pourbaix diagram of Fe in a system with 0.001 mole of Fe in 1 kg of pure water at 25°C and 1 bar.



Comparing this Pourbaix diagram with that from [Basic Settings and Definitions](#), you can see that both diagrams are for the same interaction system under the same temperature-pressure condition, but the present diagram shows the results of calculations where the gaseous mixture phase has been included. It presents the complete Pourbaix diagram for the defined interaction system, in a full thermodynamic equilibrium. In the upper region (high Eh) and lower region (low Eh), the aqueous solution phase is oxidised and reduced to the O<sub>2</sub>- or H<sub>2</sub>-dominated gas phase. Hematite coexists with the O<sub>2</sub>-dominated gas, while Fe-BCC is stable with the H<sub>2</sub>-dominated gas. The stability field of magnetite may extend slightly into the H<sub>2</sub>-dominated gas region, where it coexists with the gaseous mixture instead of the aqueous solution. Its boundary with Fe (BCC) cannot be drawn due to the absence of aqueous solution phase.

It is not possible to calculate the pH value in both the O<sub>2</sub>- and H<sub>2</sub>-dominated gaseous mixture phase stability regions where the aqueous solution phase is absent. Hence, in a normal Pourbaix diagram, no line is drawn that indicates a phase boundary between the gas phase and the metallic phases or secondary phases (that is, metal-oxides).

## Other Examples of Pourbaix Diagrams

---

### In this section:

---

Variations of Pourbaix Diagrams .....	23
Pourbaix Diagrams for Complex Alloys .....	26

## Variations of Pourbaix Diagrams

The shape of a Pourbaix diagram of an alloy or condensed material and the stability relations of various secondary phases depend upon the following system factors:

- Initial amount of the alloy or other condensed materials
- Initial composition of the alloy or other condensed materials
- Initial amount of the interacting aqueous solution phase
- Initial composition of the interacting aqueous solution phase
- Temperature and pressure conditions

In the interaction system that the diagrams in [Pourbaix Diagrams with Gas Phase Excluded](#) and [Pourbaix Diagrams with Gas Phase Included](#) are based on, the initial amount of pure Fe that was taken to have effectively reacted with 1 kg of pure water at 25° C and 1 bar was 0.001 m. The following diagrams are Pourbaix diagrams of Fe where other initial amounts of pure Fe have been used, or where the interacting aqueous solution compositions have been alternated, or the temperature and/or pressure have been changed. Gaseous mixture phases have been included in all the calculations that these diagrams are based on.

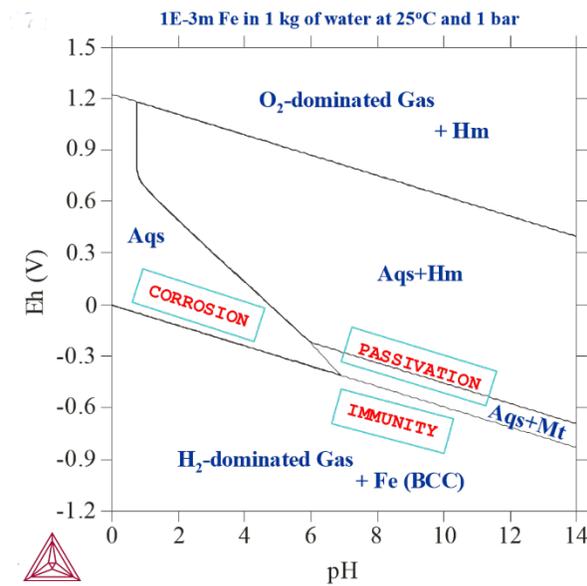


Figure 1:  $1E-3$  m Fe actively reacted with 1 kg of pure water at 25° C and 1 bar (as in the calculation in [Pourbaix Diagrams with Gas Phase Included](#)).

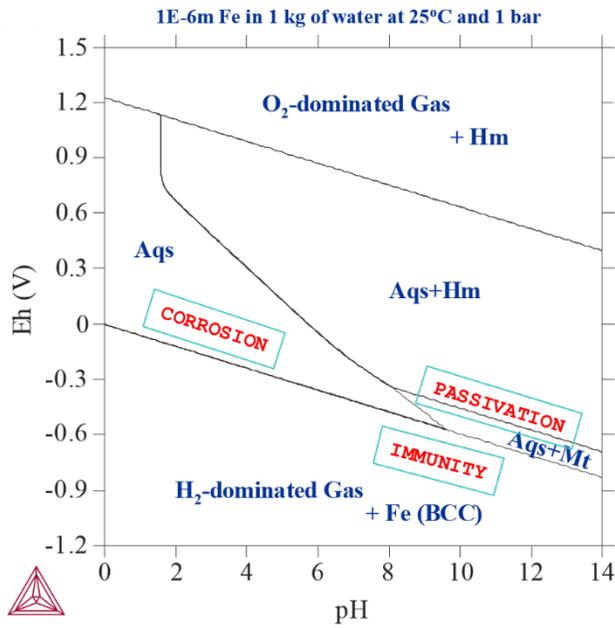


Figure 2: 1E-6 m Fe actively reacted with 1 kg of pure water at 25° C and 1 bar. Note that the active metal corrosion region gets enlarged as the initial Fe amount decreases from 1E-3m to 1E-6m.

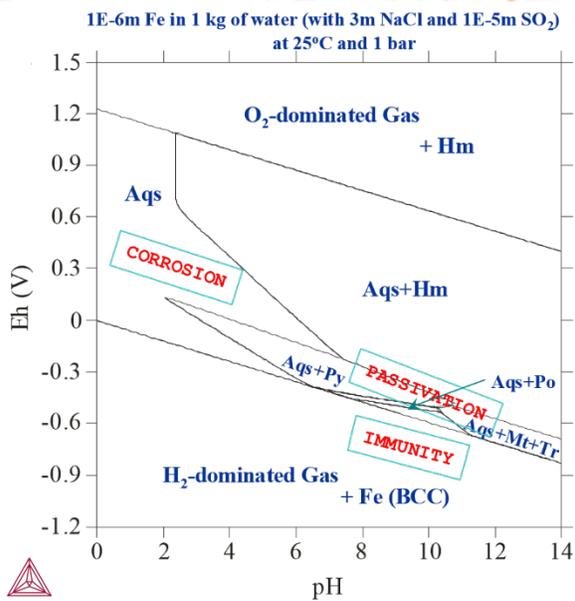


Figure 3: 1E-6 m Fe actively reacted with 1 kg of water, 3m NaCl and 1E-5 m SO<sub>2</sub> at 25° C and 1 bar. Introducing SO<sub>2</sub> into the system leads to the formation of various metal-sulphides (Py-pyrite, Popyrrhotite, Tr-troilite). In addition, the passivation region becomes larger.

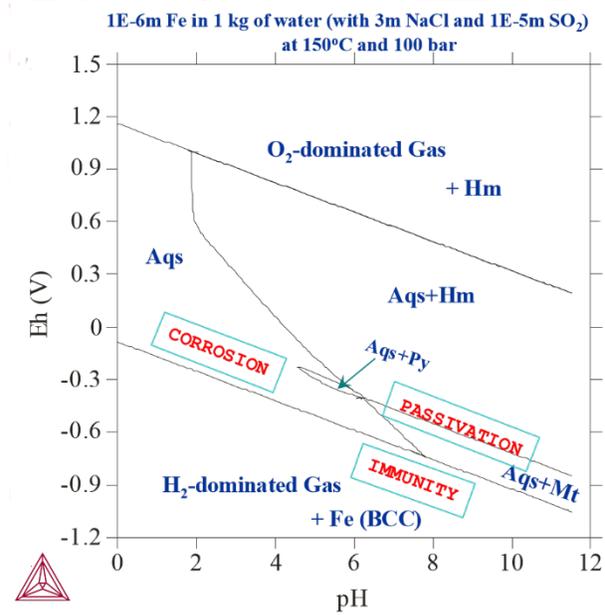


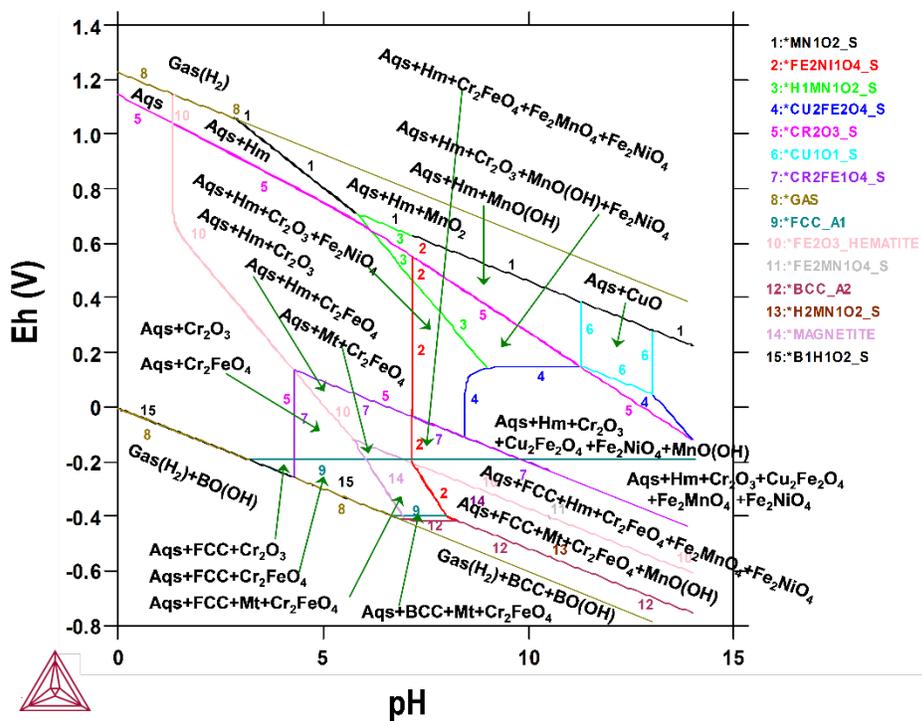
Figure 4: 1E-6 m Fe actively reacted with 1 kg of water, 3m NaCl and 1E-5 m SO<sub>2</sub> at 150° C and 100 bar. Here, changing the temperature and pressure affects the stability fields of various Fe-oxides/sulphides.

As you can see from the figures, the aqueous-gas phase boundaries shift as the initial bulk compositions, pressure and temperature conditions in the interaction system change.

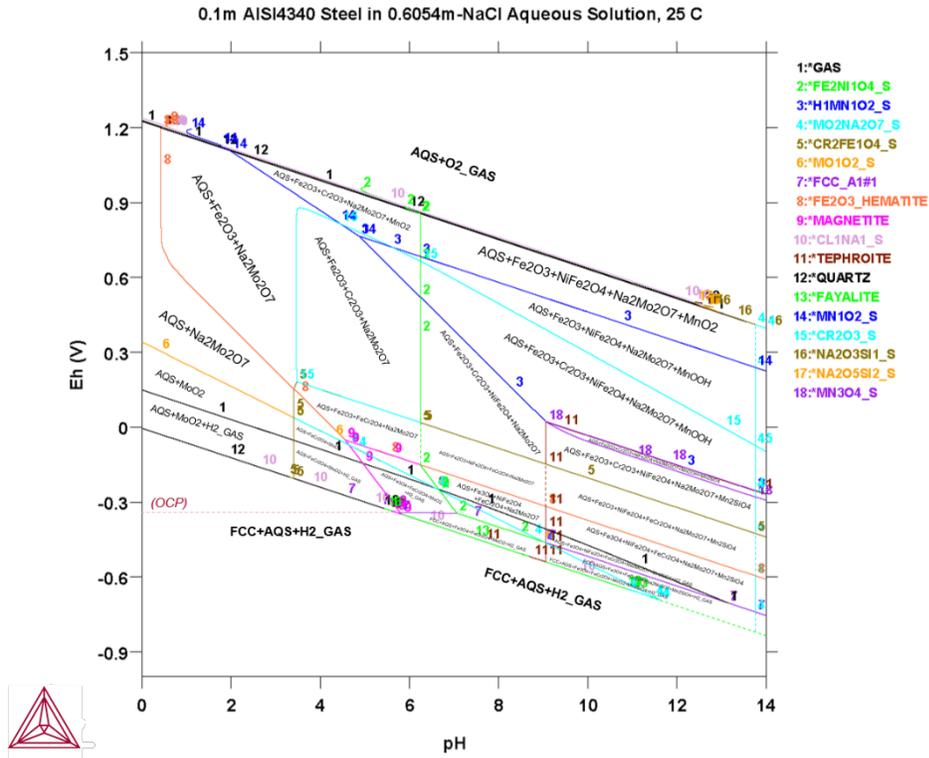
## Pourbaix Diagrams for Complex Alloys

Thermo-Calc can not only be used to simulate how pure metals interact with pure water or simple aqueous solutions under normal pressure and temperature conditions. It can also be used to calculate how complex alloys and concentrated aqueous solutions interact over a very wide pressure, temperature and composition ranges. This is illustrated with the two examples.

The first example is a Pourbaix diagram calculated for the heterogeneous interaction between 0.001 mole of steel (Fe-7.676Cr-5.0Ni-2.1887Mn-1.0Cu [at%]) and 1 kg of water (and with 1.2 m H<sub>3</sub>BO<sub>3</sub>, 0.022 m Li and 0.001 m NH<sub>3</sub>), at 25° C and 1 bar. This application is particularly useful for safety assessments of nuclear reactors and nuclear waste repositories.



The next example is a Pourbaix diagram calculated for the heterogeneous interaction systems between 0.1 mole of AISI4340 stainless steel (Fe-0.80Cr-1.85Ni-0.70Mn-0.25Mo-0.25Si-0.40C [wt%]) and 1 kg of seawater (with the equivalent of 0.6054 m NaCl), at 25° C and 1 bar. This application is particularly useful for failure analysis of petroleum exploitation industry.



---

## Pourbaix Diagram References

---

- [1973Pou] Pourbaix, M. (1973). *Lectures on Electrochemical Corrosion*. Plenum Press, New York.
- [1974Pou] Pourbaix, M. (1974). *Atlas of Electrochemical Equilibria in Aqueous Solutions, 2nd Ed*, Houston, Tex., National Association of Corrosion Engineers.
- [1981Hel] Helgeson, H.C., Kirkham, D.H., and Flowers, G.C. (1981). Theoretical prediction of the thermodynamic behavior of aqueous electrolytes at high pressures and temperatures: IV. Calculation of activity coefficients, osmotic coefficients, and apparent molal and standard and relative partial molal properties to 600° C and 5 kb. *Am. J. Sci.*, 281, 1249- 1516.
- [1990Cia] Ciavatta, L. (1990). The specific interaction theory in equilibrium analysis. Some empirical rules for estimating interaction coefficients of metal ion complexes. *Ann Chim. (Rome)*, 80, 255-263.
- [1991Joh] Johnson, J.M. and Norton, D. (1991). Critical phenomena in hydrothermal system: State, thermodynamic, electrostatic, and transport properties of H<sub>2</sub>O in the critical region. *Am. J. Sci.*, 291, 541-648.
- [2002And] Andersson J-O., Helander T., Höglund L., Shi P., and Sundman B. (2002). Thermo-Calc and DICTRA, Computational tools for materials science. *Calphad*, 26: 273-312.
- [2002Cra] Cramer, S.D. and Covino, B.S. Jr. (2003). *Corrosion: Fundamentals, Testing, and Protection*. ASM Handbook, Volume 13A.
- [2002Kau] Kaufman, L. (2002). Calculation of Pourbaix diagrams for C22 in various well water chemistries. In: *Proceedings for the 5th Nickel Development Institute Workshop on Fabrication of Welding of Nickel Alloys* (R. Moeller, ed.), Nickel Development Institute, Las Vegas, Nevada, 16-17 Oct., 2002.
- [2009Kau] Kaufman, L., Perepezko, J.H., Hildal, K., Farmer, J., Day, D., Yang, N., and Branagan, D. (2009). Transformation, stability and Pourbaix diagrams of high performance corrosion resistant (HPCRM) alloys. *Calphad*, 33: 89-99.
- [2011aShi] Shi Pingfang, Engström A, and Sundman B (2011a). Thermodynamic investigations on materials corrosion in some industrial and environmental processes. *Journal of Environmental Sciences*, 23 (Supplement): S1-S7.
- [2011bShi] Shi, P., Engström, A., Sundman, B., and Ågren, J. (2011b). Thermodynamic calculations and kinetic simulations of some advanced materials. *Materials Science Forum*, 675-677: 961-974.

- [2011Tho] Thompson, W.T., Kaye, M.H., Bale, C.W., and Pelton, A.D. (2011). Pourbaix diagrams for multielement systems. In: *Uhlig's Corrosion Handbook, 3rd Edition (Revised Edition, Ed.)*, The Electrochemical Society and John Wiley & Sons, New Jersey.
- [2011Ver] Verink, E.D. (2011). Simplified procedure for constructing Pourbaix diagrams. In: *Uhlig's Corrosion Handbook, 3rd Edition (Revised Edition, Ed.)*, The Electrochemical Society and John Wiley & Sons, New Jersey.
- [2013aShi] Shi, P. (2013a). *Thermo-Calc 3.0 (Console Mode) Manual (including TC3-CM User's Guide, and Thermo-Calc Software System)*, Thermo-Calc Software, Stockholm, Sweden.
- [2013bShi] Shi, P. (2013b). *Materials Corrosion Applications with Thermo-Calc Software*. Thermo-Calc Software, Stockholm, Sweden.

# Thermo-Calc: General Reference

## Miscellaneous Information

*Generally applicable to multiple applications within Thermo-Calc, i.e. in both Console Mode and Graphical Mode, or needed for a specific database calculation in Thermo-Calc (i.e. thermophysical properties), etc.*



# Parameters, Functions, and Variables

---



This section is applicable to both Console Mode and Graphical Mode with respect to understanding the format these are to be written in for Thermo-Calc to use the parameter, function, or variable. However, many of the sections are specific to Console Mode where the explicit use of these terms is more often used.



To learn about using variables and functions, in Graphical Mode, see [T\\_07: User-Defined Functions](#) and in Console Mode, see [example 44](#).

## In this section:

---

Specifying Parameters .....	3
About Operators and Functions .....	5
Thermodynamic Variables .....	6
Intensive Variables .....	8
Extensive Variables .....	9
Special Quantities .....	13
The u-Fraction Variable .....	15
Suffixes .....	16
Derived Variables and Partial Derivatives .....	23
Units of State Variables and Derived Variables .....	30
User-Specified Units .....	31
Examples of User-Defined Units .....	32
State Variables for Scheil .....	41
State Variables in Console Mode DICTRA .....	43

## Specifying Parameters

---



This section or topic is specific to Console Mode.

Running a command typically requires that several parameter values are specified, which can be done directly in the command line after the command name. For example, to set an axis variable for a stepping or mapping operation, enter:

```
SET_AXIS_VARIABLE 1 X(FCC,FE) 0 0.89 0.025
```

You can also use SET\_AXIS\_VARIABLE and press <Enter>. At the command line prompt, you then specify the parameters one at a time.

On all command lines, you can enter a question mark (?) for help. For some parameters, enter two question marks (??) for more detailed information.

### **Default Parameters Values**

When you are requested to specify a parameter, Thermo-Calc usually suggests an appropriate default value, which is displayed at the prompt between two slash signs (/). For example, this prompt means the default answer is `TERMINAL`:

```
OUTPUT FILE /TERMINAL/:
```

Press <Enter> to accept the default.

If you specify some parameters directly after the command name, then you can still choose to accept default values for some parameters.

To accept the default value for a parameter, type a comma (,) instead of a value for that parameter. For the comma to be interpreted correctly, type a blank space on each side of the comma. However, if you accept the default values for several parameters that follow each other, then the commas can be typed in without blank spaces separating them: (,,,) is interpreted in the same way as ( , , ).

For some commands and parameters, Thermo-Calc treats the values previously assigned as default values if the command is used again. For example, if the first stepping or mapping axis variable is specified as:

```
SET_AXIS_VARIABLE 1 X(FCC,FE) 0 0.89 0.025
```

Then if you want to change a parameter value for this axis variable, type:

```
SET_AXIS_VARIABLE 1 , 0.1 , , ,
```

This changes the minimum value from 0 to 0.1 at which a stepping or mapping operation halts. The other parameter values remain the same. Consequently, what is typed is equivalent to:

```
SET_AXIS_VARIABLE 1 X(FCC,FE) 0.1 0.89 0.025
```

## About Operators and Functions

Sometimes you need to specify functions when setting up a system. For example, to make global conditions vary as a function of time.

The following operators can be used in writing functions. Use a semi-colon (;) to mark the end of a function.

### Available Operators for Writing Functions

<i>Operator</i>	<i>Description</i>
+, -, *, /	Addition, subtraction, multiplication, division
**	Exponentiation, raising one quantity to the power of another, e.g. $x^2$ .
ABS()	Absolute value
ERF(X)	Error function
EXP(X)	Exponential
LOG(X)	Natural logarithm
LOG10(X)	Base 10 logarithm
SIGN(X)	Sign function SIGN(X) is -1 when $X < 0$ , is 0 when $X = 0$ and is 1 when $X > 0$ .
SIN(X), COS(X), TAN(X), ASIN(X), ACOS(X), ATAN(X)	<ul style="list-style-type: none"> <li>Trigonometric functions: sine, cosine, tangent</li> <li>Inverse trigonometric functions: arcsine, arccosine, arctangent</li> </ul>
SINH(X), COSH(X), TANH(X), ASINH(X), ACOSH(X), ATANH(X)	<ul style="list-style-type: none"> <li>Hyperbolic trigonometric functions</li> <li>Inverse hyperbolic functions</li> </ul>
SQRT(X)	Square root

---

## Thermodynamic Variables

---

There are different thermodynamic variables in Thermo-Calc. Some are *state variables*, for example temperature, pressure and mole fraction. These characterize the equilibrium state of a system. You set them when you define your system in the POLY module before performing a calculation. They are also used in other modules such as DATA, GIBBS and PARROT. State variables that involve components can be used for the defined components, but not for any species. (To define new components in a defined system, use [DEFINE COMPONENTS](#)).

Other useful thermodynamic variables are derive from state variables with a mathematical function. Some derived variables are pre-defined by Thermo-Calc, such as normalized energetic and compositional extensive state variables for example. You use these by appending various normalizing suffixes to abbreviations that are associated with the state variables.

Thermodynamic variables can also be divided into extensive variables and intensive variables. An extensive variable is a variable whose value depends on the size of the system, whereas an intensive variable is a variable whose value is independent of system size.

More information about thermodynamic variables is available in the online help in Thermo-Calc Console Mode. Use the command [INFORMATION](#), with a subject keyword such as one of the following as argument: STATE VARIABLES, INTENSIVE VARIABLES, EXTENSIVE VARIABLES, DERIVED VARIABLES, CONDITIONS (for condition settings), AXIS-VARIABLES (for stepping/mapping variable settings).

### Common Thermodynamic Variables

The tables below list common thermodynamic variables. Most of these can be used to define equilibrium conditions in the POLY module unless otherwise indicated.

The variables are divided into the following tables:

- Intensive variables
- Energy-related extensive variables for whole system or for a phase
- Compositional extensive variables (overall amount of components in the whole system, or amount of a component in system or in a phase)
- Constitutional composition-related extensive variables (amount of a constituent/species on a sublattice site in a phase)
- Special quantities

---

## How to Read the Tables

- The *Abbrev.* column shows the abbreviation that you use in the POLY module for referring to the variable. If the variable must be given any arguments, then these are given in parenthesis directly after the abbreviation. For most variables, this is also the abbreviation that you use to refer to the variable in other modules. However, some variables are referred to with a different abbreviation in the POST module. If this is the case, then this is noted at the bottom of the table.
- Variables that are specific to a species, in which case the species is specified as an argument (*sp* in the table below). If there are two or more sublattices in a given phase, then this argument is given as `constituent#sublattice`. This indicates a constituent on a specific sublattice or site in the phase (referred to by the # sign and a digit). For example,  $y_{(BCC, C\#2)}$  stands for the site fraction of the C species on the second sublattice site of the BCC\_A2 solution phase.
- The *Unit* column show which unit that you can use to express the variable in the POLY module. This unit is always the SI unit. In other modules, state variables may be expressed in other units.
- The *Suffix* column shows the suffixes you can append to the variables. The <sub>R</sub> suffix can be used with all compositional extensive state variables, but the suffix does not always change the value of the variable. This is indicated by putting the <sub>R</sub> in parenthesis.

These types of variables are listed in the following topics:

- [Intensive Variables](#)
- [Extensive Variables](#)
- [Special Quantities](#)
- [The u-Fraction Variable](#)
- [Suffixes](#)

## Intensive Variables

V	Abbrev.	Unit	Descript.	Domain	Suffix
T	T <sup>1</sup>	K	Temperature	System	
P	P	Pa	Pressure	System	
μ	MU(comp)	J/mol	Chemical potential	Component	R
	MU(sp,ph) <sup>2</sup>			Species relative to a solution phase	R
a	AC(comp)	N/A	Activity	Component	R
	AC(sp,ph) <sup>2</sup>			Species relative to a solution phase	R
	LNAC (comp) <sup>3</sup>		ln(Activity)	Component	R
	LNAC (sp,ph) <sup>2</sup>			Species relative to a solution phase	R

<sup>1</sup> When plotting in POST, besides T (Kelvin), you can also use T\_C or T\_F to plot temperature in °C or °F.

<sup>2</sup> Only for single-substitutional-lattice solution phases such as AQUEOUS solution and GASEOUS mixture phases and for interacting species on the substitutional sublattice of two-sublattice solution phases (such as the phases BCC\_A2 and M6C phases).

<sup>3</sup> In natural logarithm (lnAC=MU/RT)

## Extensive Variables

### Energy-Related Extensive Variables for Whole System or for a Phase

V	Abbrev.	Unit	Descript.	Domain	Suffix
V	V	m <sup>3</sup>	Volume	System	M, W, V, R
	V(ph) <sup>1</sup>			Phase	M, W, F, R
	VP(ph) <sup>2</sup>			Phase	M, W, V, R
G	G	J	Gibbs energy	System	M, W, V, R
	G(ph) <sup>1</sup>			Phase	M, W, V, F, R
A	A	J	Helmholtz energy	System	M, W, V, R
	A(ph) <sup>1</sup>			Phase	M, W, V, F, R
U	U	J	Internal energy	System	M, W, V, R
	U(ph) <sup>1</sup>			Phase	M, W, V, F, R
H	H	J	Enthalpy <sup>6</sup>	System	M, W, V, R
	H(ph)			Phase	M, W, V, F
S	S	J/K	Entropy <sup>6</sup>	System	M, W, V, R
	S(ph) <sup>1</sup>			Phase	M, W, V, F, R
Cp	HM.T <sup>3</sup>	J/mol/K	Heat capacity at constant pressure	System	R
	HM(ph).T <sup>3</sup>			Phase	R
Cv	UM.T <sup>4</sup>	J/mol/K	Heat capacity at constant volume	System	R

V	Abbrev.	Unit	Descript.	Domain	Suffix
	UM(ph).T <sup>4</sup>			Phase	R
D	DG(ph) <sup>5</sup>	N/A	Driving force (thermodynamic factor)	Phase	Always use with M, W, V or F. R can also be used.

<sup>1</sup> For this variable, if the phase is unstable, then the variable gets a value of zero.

<sup>2</sup> Do not use VP(ph) as a condition in POLY. Instead, use CHANGE\_STATUS PHASE <PHASE>=FIX <AMOUNT> where the fixed <AMOUNT> is roughly equal to VPF(ph), which cannot be directly evaluated or listed. If the phase is unstable, then VP(ph) and its normalized quantities get zero value.

<sup>3</sup> Use only if pressure has been set as a condition.

<sup>4</sup> Use only if volume has been set as a condition.

<sup>5</sup> Already divided by RT.

<sup>6</sup> When working in the POST module you can also enter the abbreviation for this variable into the Console, for example, `enth` or `enthalpy`, which corresponds to `HM`, and `entr` and `entropy`, which corresponds to `SM`.

## Compositional Extensive Variables

These are for the overall amount of components in whole system, or amount of a component in system or in a phase.

Do not use NP(ph) or BP(ph) as conditions in the POLY module. Instead use CHANGE\_STATUS PHASE <PHASE>=FIX <AMOUNT> where the fixed <AMOUNT> is roughly equal to NPF(ph) or BPF(ph), which cannot be directly evaluated or listed. If the phase is unstable, then NP(ph) or BP(ph) (and its normalized quantities) get zero value.

When plotting the following quantities in POST, refer to each as indicated:

- X(comp) -> Mole-Fraction <comp> (or M\_F)
- W(comp) -> Weight-Fraction <comp> (or W\_F)
- X%(comp) -> Mole-Percent <comp> (or M\_P)
- W%(comp) -> Weight-Percent <comp> (or W\_P).
- u-f(ph,comp) -> U-F

V	Abbrev.	Unit	Descript.	Domain	Suffix	Comment
n	N	mole	Moles	All components	M, W, V, (R)	
	N(comp)			Component	M, W, V, (R)	
	N(ph,comp)			Component in a phase	M, W, V, (R)	
	NP(ph) <sup>1</sup>			Phase	M, W, V, (R)	See above
b	B	gram	Mass	All components	M, W, V, (R)	
	B(comp)			Component	M, W, V, (R)	
	B(ph,comp)			Component in a phase	M, W, V, (R)	
	BP(ph) <sup>1</sup>			Phase	M, W, V, (R)	See above
x	X(comp)	N/A	Mole fraction	Component	(R)	See above
	X(ph,comp)			Component in a phase	(R)	
w	W(comp)	N/A	Mass (weight) fraction	Component	(R)	See above
	W(ph,comp)			Component in a phase	(R)	
x%	X%(comp) <sup>2</sup>	N/A	Mole percent	Component	(R)	See above
w%	W%(comp) <sup>2</sup>	N/A	Mass (weight) percent	Component	(R)	See above
u	u-f (ph,comp) <sup>1</sup>	N/A	u-fraction	Component in a stable phase	(R)	See above
in	IN(sp) <sup>2</sup>	mole	Input mole	Phase species	(R)	See above

V	Abbrev.	Unit	Descript.	Domain	Suffix	Comment
			number	in the system		
im	IM(sp) <sup>2</sup>	gram	Input mass unit	Phase species in the system	(R)	See above
<p><sup>1</sup> Do not use as a condition in the POLY module.</p> <p><sup>2</sup> Only available in TQ-Interface and TC-API.</p>						

### ***Constitutional Composition-Related Extensive Variable***

This is the amount of a constituent/species on a sublattice site in a phase.

V	Abbrev.	Unit	Descript.	Domain
y	Y(ph,cons#sub)	N/A	Site fraction	Constituent on a sublattice site (denoted by # and a digit) in a phase

## Special Quantities

Variable	Abbrev.	Unit	Description or Example	Domain	Suffix
Q	QF(ph) <sup>1</sup>	n/a	Phase stability function	Phase	R Under a condition and system definition, QFR(ph) = QF(ph)
Tc	TC(ph) <sup>2</sup>	K	Curie temperature	Phase	R Under a condition and system definition, TCR(ph) = TC(ph)
Mb	BMAG(ph) <sup>3</sup>	n/a	Bohr magneton number	Phase	R Under a condition and system definition, BMAGR(ph) = BMAG(ph)
D1G	D1G(phase, site-fraction)	n/a	For example, D1G(FCC_A1#1, Cr#1) for first derivative of G for first composition set of FCC_A1 phase with respect to Cr site fraction on first sublattice.   Search for TQGMDY in the <i>TQ-Interface Guide</i>	Phase	n/a
D2G	D2G(phase, site-fraction,	n/a	For example, D2G(FCC_A1#2, Cr#1, Cr#1) for	Phase	n/a

Variable	Abbrev.	Unit	Description or Example	Domain	Suffix
	site-fraction)		<p>second derivative of G for second composition set of FCC_A1 phase with respect to Cr site fraction on first sublattice.</p> <p>Search for TQDGY in the <i>TQ-Interface Guide</i></p> 		
<p><sup>1</sup> Negative when phase composition is inside a spinodal, otherwise positive. Can be used to find out if an equilibrium is within the miscibility gap for a solution phase. Cannot be used as a condition.</p> <p><sup>2</sup> Calculated for phases with magnetic contributions in an equilibrium state. Cannot be used as a condition, but can be plotted in POST or evaluated in POLY using SHOW_VALUE.</p> <p><sup>3</sup> Calculated for phases with magnetic contributions in an equilibrium state.</p> <p>n/a = not applicable</p>					

## The u-Fraction Variable

---

The u-fraction is an important quantity in some single-point or stepping paraequilibrium calculations in the POLY module and in the DICTRA module. You can apply the variable in a paraequilibrium calculation to the substitutional matrix component, the substitutional alloying components or the interstitial component. The u-fraction is denoted as  $u_i$  for the  $i$ -th system component  $n$  of a whole system or of a specific phase in various equilibrium states (such as full-equilibria, partial-equilibria, para-equilibria, local-equilibria).

Generally, the u-fraction is defined as:

$$\frac{x_i}{\sum_{j \in S} x_j}$$

The sum of  $x_j$  in the denominator is calculated for all substitutional alloying elements in the whole system or in a specific phase. The variable  $x_i$  in the numerator stands for the mole-fraction of the  $i$ -th component in the whole system or in the specific phase.

## Suffixes

Suffixes may be appended to all extensive variables and to some intensive variables. Some suffixes can be used to enter the value of variables in normalized form. These are referred to as normalizing suffixes. If the variable that you create with a normalizing suffix is based on a state variable that can be set as a condition in POLY, then the normalized variable can also be set as a condition in POLY.

The normalizing suffixes are M (per mole), W (per mass in gram), V (per volume in  $\text{m}^3$ ) and F (per mole formula unit). There is also a reference state suffix  $\text{R}$ , which you can use if you want the value of a thermodynamic variable to be calculated with respect to a reference state that have previously set.

### Normalizing Suffixes

When variables that express system and phase quantities are normalized, the following general rules are used:

- System quantities are normalized by the total system size (in terms of N, B or V).
- Phase quantities are normalized by the phase amount [in terms of NP(ph), BP(ph) or VP(ph)].

The normalized quantities of G(ph), A(ph), U(ph), H(ph), S(ph) and V(ph) are calculated according to the thermodynamic model used for the phase (e.g. GM(ph), AM(ph), UM(ph), HM(ph), SM(ph) and VM(ph)). These quantities are calculated using the first derivatives of the Gibbs energy expressed for the phase with respect to the current composition of the system.

The tables describe the normalizing suffixes for the different state variables.

### *G, A, U, H, S and V of a Whole System*

Suffix	Description	Example
M (per mole)	Total system size in terms of N.	GM is the Gibbs energy per mole of the system (J/mol). $GM = G/N$
W (per mass in gram)	The total system size in terms of B.	GW is the Gibbs energy per mass of the system (J/g).

Suffix	Description	Example
		$GW = G/B$
V (per volume in $m^3$ )	The total system size in terms of V. VV does not have to be evaluated.	GV is the Gibbs energy per volume of the system ( $J/m^3$ ). $GV = G/V$

### ***G(ph), A(ph), U(ph), H(ph), S(ph) and V(ph) of a Phase***

Suffix	Example
M (per mole)	For example, GM(ph) is Gibbs energy of the phase per mole of the phase (J/mol). $GM(ph) = G(ph)/NP(ph)$
W (per mass in gram)	For example, GW(ph) is Gibbs energy of the phase per mass of the phase (J/mol). $GW(ph) = G(ph)/BP(ph)$
V (per volume in $m^3$ )	For example, GV(ph) is Gibbs energy of the phase per volume of the phase (J/mol). $GV(ph) = G(ph)/V(ph)$
F (per mole formula unit)	For example, GF(ph) is the Gibbs energy of the phase per formula unit of the phase (J/mol). $GF(ph) = G(ph)/NP(ph)*NA$

### ***DG(ph) of a Phase***

Suffix	Description	Example(s)
M (per mole)	Theoretically, the first derivative of the variable with regard to the phase amount in terms of NP(ph). Since DG(ph) is not directly calculated, the second derivative of the Gibbs energy expressed for the phase in question with respect to the current compositions in the equilibrium state of the system is calculated instead.	DGM(ph) is driving force for precipitation of the phase per mole of components. $DGM(ph) = \partial DG(ph)/\partial NP(ph)$

Suffix	Description	Example(s)
	DGM(ph) cannot not be set as a condition since it is only calculated under a certain type of equilibrium state.	
W (per mass in gram)	<p>Theoretically, the first derivative of the variable with regard to the phase amount in terms of BP(ph). Since DG(ph) is not directly calculated, the second derivative of the Gibbs energy expressed for the phase in question with respect to the current compositions in the equilibrium state of the system is calculated instead.</p> <p>DGW(ph) cannot not be set as a condition since it is only calculated under a certain type of equilibrium state.</p>	<p>DGW(ph) is driving force for precipitation of the phase per mass of components.</p> $DGW(ph) = \partial DG(ph) / \partial BP(ph)$
V (per volume in m <sup>3</sup> )	<p>Theoretically, the first derivative of the variable with regard to the phase amount in terms of VP(ph). Since DG(ph) is not directly calculated, the second derivative of the Gibbs energy expressed for the phase in question with respect to the current compositions in the equilibrium state of the system is calculated instead.</p> <p>DGV(ph) cannot not be set as a condition since it is only calculated under a certain type of equilibrium state.</p>	<p>DGV(ph) is driving force for precipitation of the phase per volume of components.</p> $DGV(ph) = \partial DG(ph) / \partial VP(ph)$
F (per mole formula unit)	<p>Theoretically, the first derivative of the variable with regard to the phase amount in terms of NP(ph) and NA (NA is the total atomic number in the phase formula). Since DG(ph) is not directly calculated, the second derivative of the Gibbs energy expressed for the phase in question with respect to the current compositions in the equilibrium state of the system is calculated instead.</p> <p>DGF(ph) cannot not be set as a condition since it is only calculated under a certain type of equilibrium state.</p>	<p>DGF(ph) is driving force for precipitation of the phase per formula unit of components.</p> $DGF(ph) = \partial DG(ph) / \partial NP(ph) * \partial NA$

## ***N and B of a System***

Suffix	Description	Example(s)
M (per mole)	<p>The total system size in terms of N.</p> <p>NM does not have to be evaluated.</p> <p>BM cannot be set as a condition.</p>	<p>BM is mass (gram) of components per mole of the system (g/mol).</p> $BM = B/N$
W (per mass in gram)	<p>The total system size in terms of B.</p> <p>BW does not have to be evaluated.</p> <p>NW cannot be set as a condition.</p>	<p>NW is mole number of components per mass of the system (mol/g).</p> $NW = N/B$
V (per volume in m <sup>3</sup> )	<p>The total system size in terms of V.</p> <p>BV is the density of the entire system.</p>	<p>NV is mole number of components per volume of the system (mol/m<sup>3</sup>).</p> $NV = N/V$ <p>BV is the density of the entire system (g/m<sup>3</sup>).</p> $BV = B/V$

## ***N (comp) and B(comp) of a Component in the System***

Suffix	Description	Example(s)
M (per mole)	<p>The total system size in terms of N.</p> <p>BM(comp) cannot be set as a condition.</p>	<p>NM(comp) is the mole of a component per mole of the system (i.e. mole fraction, which also is expressed as X(comp)).</p> $NM(\text{comp}) = N(\text{comp})/N$
W (per mass in gram)	<p>The total system size in terms of B.</p> <p>NW(comp) cannot be set as a condition.</p>	<p>BW(comp) is mass (gram) of a component per mass of the system (i.e. mass fraction, which also is expressed as W(comp)).</p> $BW(\text{comp}) = B(\text{comp})/B$
V (per	The total system size	NV(comp) is mole number of a component per volume of the system

Suffix	Description	Example(s)
volume in m <sup>3</sup> )	in terms of V.	(mol/m <sup>3</sup> ). $NV(\text{comp}) = N(\text{comp})/V$

### ***NP(ph), BP(ph) and VP(ph) of a Phase in the System***

Suffix	Description	Example(s)
M (per mole)	BPM(ph) and VPM(ph): The phase amount in terms of NP(ph). NPM(ph): The total system size in terms of N.	BPM(ph) is mass (gram) of a phase per moles of the system (g/mol). $BPM(\text{ph}) = BP(\text{ph})/N$ NPM(ph) is number of moles of a phase per moles of the system (mole fraction). $NPM(\text{ph}) = NP(\text{ph})/N$
W (per mass in gram)	NPW(ph) and VPW(ph): The phase amount in terms of BP(ph). BPW(ph): The total system size in terms of B.	VPW(ph) is volume (m <sup>3</sup> ) of a phase per mass of the system (m <sup>3</sup> /g) $VPW(\text{ph}) = VP(\text{ph})/B$ BPW(ph) is mass (gram) of a phase per mass of the system (mass fraction) $BPW(\text{ph}) = BP(\text{ph})/B$
V (per volume in m <sup>3</sup> )	NPV(ph) and BPV(ph): The phase amount in terms of VP(ph). VPV(ph): The total system size in terms of V.	NPV(ph) is number of moles of a phase per volume of the system (mol/m <sup>3</sup> ). $NPV(\text{ph}) = NP(\text{ph})/V$ VPV(ph) is volume (m <sup>3</sup> ) of a phase per volume of the system (volume fraction). $VPV(\text{ph}) = VP(\text{ph})/V$

### ***N(ph,comp) and B(ph,comp) of a Component in a Phase***

Suffix	Description	Example(s)
M (per mole)	The phase amount in terms of NP(ph) of the phase.	NM(ph,comp) is number of moles of a component per mole of a phase (i.e. mole fraction of a component in a phase, which is equivalent to X(ph, comp)).

Suffix	Description	Example(s)
		$NM(\text{ph,comp}) = N(\text{ph,comp})/NP(\text{ph})$
W (per mass in gram)	The phase amount in terms of BP(ph) of the phase.	BW(ph,comp) is mass (gram) of a component per mass of a phase (i.e. mass fraction of a component in a phase, which is equivalent to W(ph,comp)). $BW(\text{ph,comp}) = B(\text{ph,comp})/BP(\text{ph})$
V (per volume in m <sup>3</sup> )	The phase amount in terms of VP(ph) of the phase.	NV(ph,comp) is mole number of a component per volume of a phase (mol/m <sup>3</sup> ). $NV(\text{ph,comp}) = N(\text{ph,comp})/VP(\text{ph})$

## The Reference State Suffix R



### SET\_REFERENCE\_STATE

You can use the reference state suffix <sub>R</sub> for some thermodynamic variables to calculate their value with respect to a reference state that you have previously set for a system component with the SET\_REFERENCE\_STATE command in POLY (or in response-driven modules such as the POURBAIX module). The value of energy-related variables that are used with the <sub>R</sub> suffix depends on the reference states of all the components in the defined system.



It is possible to use an <sub>R</sub> suffix on all compositional extensive state variables as well, but the value of the state variable is always the same, with or without the suffix.

If the reference state for a system component is the default reference state (the stable reference state (SER) which is defined in a Thermo-Calc database), then  $MUR(\text{comp}) = MU(\text{comp})$ ,  $ACR(\text{comp}) = AC(\text{comp})$  and  $LNACR(\text{comp}) = LNAC(\text{comp})$ .

In the case of some thermodynamic variables, you can also use the <sub>R</sub> suffix to express chemical potentials and activities of species relative to some single-substitutional-lattice solution phases (such as aqueous solution, gaseous mixture, metallic liquid solution, slag mixture or MO solid solution). These state variables are  $MU(\text{sp,ph})$ ,  $MUR(\text{sp,ph})$ ,  $AC(\text{sp,ph})$ ,  $ACR(\text{sp,ph})$ ,  $LNAC(\text{sp,ph})$  and  $LNACR(\text{sp,ph})$ .

The reference states and standard states of various solution species are pre-defined for some solution phases in some databases. For all solution species in any solution model in any database, it is always the case that  $MUR(sp,ph) = MU(sp,ph)$ ,  $ACR(sp,ph) = AC(sp,ph)$  and  $LNACR(sp,ph) = LNAC(sp,ph)$ .

## Derived Variables and Partial Derivatives

Many derived variables can be obtained easily by using partial derivatives of state variables, such as heat capacity, thermal expansivity, isothermal compressibility, among others.

Derivatives of state variables can be defined or evaluated by putting a dot (.) between two state variables. The variable after the dot must be a condition that you have set when you defined your system. For example, the heat capacity (at constant pressure or volume) of a system is the partial derivative of the state variable enthalpy with respect to temperature. To evaluate this derivative, use the designation HM.T, where temperature is a state variable that you have set already.

Any partial derivative of a state variable with respect to another can be defined or evaluated as a derived variable by using the dot (.) notation.

### Derived Variables Dot Notation Examples

The following table shows some examples:

Derived Variable	Description
$HM.T = \partial HM / \partial T$	Heat capacity for the system at either constant P if pressure is a condition or constant V if volume is a condition.
$HM(ph).T = \partial HM(ph) / \partial T$	Heat capacity for a phase at either constant P if pressure is a condition or constant V if volume is a condition.
$H.T = \partial H / \partial T$	Heat capacity for the system multiplied by total mole number of components, i.e., $\partial H / \partial T = C_p * N$ if pressure is a condition in a closed system or $\partial H / \partial T = C_v * N$ if volume is a condition in a closed system.
$H(ph).T = \partial H(ph) / \partial T$	Heat capacity for a phase multiplied by the sum of $NP(ph)$ and $HM(ph) * \partial NP(ph) / \partial T$ , i.e., $C_p(ph) * NP(ph) + HM(ph) * \partial NP(ph) / \partial T$ if pressure is a condition or $C_v(ph) * NP(ph) + HM(ph) * \partial NP(ph) / \partial T$ if volume is a condition.
$VM.T = \partial VM / \partial T$	Thermal expansivity of the system (already multiplied by the total molar volume), i.e., $\partial VM / \partial T = \alpha * VM$ .
$VM(ph).T = \partial VM(ph) / \partial T$	Thermal expansivity of a phase (already multiplied by its molar volume), i.e., $\partial VM(ph) / \partial T = \alpha(ph) * VM(ph)$ .

Derived Variable	Description
$V.T = \partial V / \partial T$	Thermal expansivity of the system (already multiplied by the total volume), i.e., $\partial V / \partial T = \alpha * V$ .
$V(ph).T = \partial V(ph) / \partial T$	Thermal expansivity of a phase (already multiplied by the phase volume) plus the $VM(ph) * \partial NP(ph) / \partial T$ term, i.e., $\partial V(ph) / \partial T = \alpha(ph) * V(ph) = \alpha(ph) * VM(ph) * NP(ph) + VM(ph) * \partial NP(ph) / \partial T$ .
$VM.P = -\partial VM / \partial P$	Isothermal compressibility of the system (already multiplied by the total molar volume), i.e., $\partial VM / \partial P = -\kappa * VM$ .
$VM(ph).P = -\partial VM(ph) / \partial P$	Isothermal compressibility of a phase (already multiplied by its molar volume), i.e., $\partial VM(ph) / \partial P = -\kappa(ph) * VM(ph)$ .
$V.P = -\partial V / \partial P$	Isothermal compressibility of the system (already multiplied by the total volume), i.e., $\partial V / \partial P = -\kappa * V$
$V(ph).P = -\partial V(ph) / \partial P$	Isothermal compressibility of a phase (already multiplied by the phase volume) plus the $VM(ph) * \partial NP(ph) / \partial P$ term, i.e., $\partial V(ph) / \partial P = -\kappa(ph) * V(ph) = -\kappa(ph) * VM(ph) * NP(ph) + VM(ph) * \partial NP(ph) / \partial P$ .
$T.X(comp) = \partial T / \partial X(comp)$	Slope of a phase boundary on a T-X(comp) phase diagram with respect to mole fraction of the component in the system.
$T.W(comp) = \partial T / \partial W(comp)$	Slope of a phase boundary on a T-W(comp) phase diagram with respect to mass of the component in the system.
$T.X(ph,comp) = \partial T / \partial X(ph,comp)$	Slope of a phase boundary on a T-X(ph,comp) phase diagram with respect to mole fraction of the component in the phase.
$T.W(ph,comp) = \partial T / \partial W(ph,comp)$	Slope of a phase boundary on a T-W(ph,comp) phase diagram with respect to mole fraction of the component in the phase.
$P.T = \partial P / \partial T$	Slope of a phase boundary on a P-T phase diagram (Note that the equilibrium with phase assemblage must be calculated first).

## Defining New Derived Variables and Functions

You can define additional derived variables or functions based on a state variable by using ENTER\_SYMBOL. Any derived variable or function must have a unique name that starts with a letter and has a maximum number of eight characters. These characters can include both uppercase and lowercase letters, numerical digits and the underscore (\_). No other characters are allowed.

The difference between new derived variables introduced with the ENTER\_SYMBOL command and new functions introduced with the same command is as follows. Whenever Thermo-Calc calculates the value of a function (as part of an equilibrium calculation for example), all functions related to the defined system is evaluated. An entered derived variable, on the other hand, is only evaluated when it is defined and when it is re-evaluated with EVALUATE\_FUNCTIONS.

## Derived Variables for Aqueous Solutions

Standard state variables used for other phases can be directly applied to the aqueous solution phase. In addition, there are pre-defined derived variables for the aqueous solution phase in particular.

Examples of predefined derived variables for the aqueous solution phase are listed and briefly described in the following tables. Some derived variables for a gaseous mixture phase are also shown. This is because EOS (Equation of State) expressions, standard thermodynamic properties and transport properties of the pure solvent H<sub>2</sub>O are also expressions and properties of the pure gaseous species H<sub>2</sub>O.

### For Aqueous Solution Phase

Name	Abbrev.	Units	Description	Comments
pH	PH	N/A	Acidity	of the aqueous solution phase $\text{pH} = -\log_{10}(\text{AC}(\text{H}^+)) = -\log_{10}(\text{ACR}(\text{H}^+, \text{AQ}))$
Eh	EH	V, mV	Hypothetical electric potential	of the aqueous solution phase $\text{Eh} = u(\text{ZE})/96485.309$
Pe	PE	N/A	Logarithm of the hypothetical electron activity	of the aqueous solution phase $\text{pe} = u(\text{ZE})/(2.3025851 * \text{RT})$
Ah	AH	kJ, kcal	Thermodynamic affinity per electron	of a redox couple wrt the standard hydrogen electrode in the aqueous solution phase $\text{Ah} = u(\text{ZE})$
yw	YH <sub>2</sub> O	N/A	Mole fraction	of the solvent H <sub>2</sub> O in the aqueous solution phase

Name	Abbrev.	Units	Description	Comments
				$Y_{H2O} = Y(AQ, H2O)$
Nw	AH2O	mole	Mole number	of 1.0 kg of solvent H2O $AH2O = 55.508435$
Nsp	NSH2O	mole	Mole number, $NS(AQ, H2O)$	of the solvent (H2O) in the aqueous solution phase $NS(AQ, H2O) = Y_{H2O} * NP(AQ)$
	NS#		Mole number, $NS(AQ, sp)$	of a solute species in the aqueous solution phase $NS(AQ, sp) = Y(AQ, sp) * NP(AQ)$
M	ML#	mol/kg_H2O	Molality, $ML(AQ, sp)$	of a solute species in the aqueous phase $ML(AQ, sp) = Y(AQ, sp) * AH2O / Y_{H2O}$
m*	TIM	equivalent molality	Total molality	of all solute species in the aqueous solution phase $TIM = \sum[ML(AQ, sp)]_{ions} + \sum[ML(AQ, sp)]_{complexes}$
mt	TIC#	equivalent molality	Total ionic concentration, TIC $(AQ, sp)$	of a cation I in the aqueous phase $TIC(AQ, spI) = \sum[ML(AQ, spI) * V(spI-in-spI)]$
I	ISTR	N/A	Ionic strength	of the aqueous solution phase $ISTR = 1/2 * \sum[ML(AQ, sp) * Z(AQ, sp) ** 2]$
Y	RCH2O	N/A	Activity coefficient, $RC(H2O, AQ)$	of the solvent (H2O) $RC(H2O, AQ) = ACR(H2O, AQ) / Y_{H2O}$
	RC#		Activity coefficient, $RC(sp, AQ)$	of a solute species $RC(sp, AQ) = ACR(sp, AQ) / Y(AQ, sp) * Y_{H2O}$

Name	Abbrev.	Units	Description	Comments
$\alpha_i$	AIH2O	N/A	Activity, AI(H2O,AQ)	of the solvent (H2O), AI(H2O,AQ) = ACR(H2O,AQ)
	AI#		Activity, AI(sp,AQ)	of a solute species related to the aqueous solution phase, AI(sp,AQ) = ACR (sp,AQ)*AH2O
	LogAI#		Log10 Activity, LogAI(H2O,AQ), LogAI(sp,AQ)	of the solvent or a solute species related to the aqueous solution phase in common logarithm, LogAI(H2O,AQ) = log10[AI (H2O,AQ)] LogAI(sp,AQ) = log10[AI (sp,AQ)]
$\alpha_w$	AW	N/A	Activity	of H2O in the aqueous solution phase AW = ACR(H2O,AQ)
$\phi$	OS	N/A	Osmotic coefficient	of aqueous solution phase OS = -55.508435*lnAW/TIM
At1	AT1	equil_mol/kg_ H2O	Titration alkalinity (definition 1)	of the aqueous solution phase Generally defined as the equivalent molality of carbonate and bicarbonate at the methyl orange endpoint (pH=4.5).
At2	AT2	equil_mol/kg_ H2O	Titration alkalinity (definition 2)	of the aqueous solution phase Generally defined as the equivalent molality of carbonate and bicarbonate, plus sulfide, at the methyl orange endpoint (pH=4.5).

## For Gaseous Mixture Phase

Name	Abbrev.	Units	Description	Comments
$\gamma$	RA#	N/A	Activity coefficient, RA(sp,GAS)	of a gaseous species in the gaseous mixture RA(sp,GAS) = function(Y,T,P)
$\gamma^*$	RF#	N/A	Fugacity coefficient, RF(sp,GAS)	of a pure gaseous species under TP RF(sp,GAS) = function(T,P/V)
F	FUG#	pa, bar, psi	Fugacity, FUG(sp,GAS)	of a gaseous species in the gaseous mixture FUG(sp,GAS) = RA(sp,GAS)*RF(sp,GAS)*Y(GAS,sp)*P
ft	TFUG	pa, bar, psi	Total gas fugacity	of the gaseous mixture phase TFUG = sum[FUG(sp,GAS)]

## Defining New Derived Variables for Aqueous Solutions

You can define many different additional derived variables and functions. For instance, the partition coefficient of a component between two phases is defined as follows:

$$Pc<name> = X(\text{phase1, component}) / X(\text{phase2, component})$$

Another example is the activity coefficient of a component in the system. It is defined as follows:

$$Rc<name> = ACR(\text{component}) / X(\text{component})$$

The activity coefficient of a species in a solution phase depends on which model definitions on the reference states that are used and it depends on the standard states for the species. If the standard state of a species is defined to be the same as the pure species in a substitutional phase, then the activity coefficient of the species can be calculated as:

$$RC<name> = ACR(\text{species, phase}) / Y(\text{phase, species})$$

In some cases, there are more than one sublattice in the phase and the reference states for the phase species cannot be presented by the pure species, as in the case of C in FCC [(Fe)1(C,N,VA)1], where the model may end at fifty percent C and fifty percent Fe. In those cases, the chemical potential and activity of all phase constituents is not well defined and you cannot properly define the activity coefficients of the species in the phase.

For an aqueous solution phase, no matter what model is used, the reference state and standard state for both the solvent and solutes are defined in a special way in the Thermo-Calc software. The reference state for the solvent is set as pure water, the same as its standard state (according to Raoult's Law). The standard state for a solute species is set the hypothetical solute species, whilst its reference state is defined as the hypothetical state at unit molality (one mole dissolved into 1.0 kg of solvent) but in which the environment of each molecular is the same as at infinite dilution (according to Henry's Law). Under these definitions, the activity coefficients for solvent species can be calculated as follows:

$$RC<H2O> = ACR(H2O, aqueous) / Y(aqueous, H2O)$$

The coefficients for the solute species can be calculated as follows:

$$RC<spec> = AI(species, aqueous) / ML(aqueous, species) \\ = ACR(species, aqueous) / Y(aqueous, species) * Y(aqueous, H2O)$$

AI is here the activity of solute species calculated from the model, and ML is the molality of the species.

## Thermodynamic Variables and Wildcard Characters

The ACR (species,phase) and the MUR (species,phase) quantities can only be appropriately and meaningfully applied to substitutional solution phases (such as GAS, AQUEOUS, LIQUID) and to solution phases without an interstitial sublattice set. This means that the wild card characters \* and \$ normally don't work properly for the activity and/or chemical potential properties.

For example, do not use ACR(\*,\*) or MUR(\*,\*) for all the species relative to all phases, nor ACR(\*,phase) or MUR(\*,phase) for all the species relative to a specific phase, nor ACR(species,\*) or MUR(species,\*) for a specific species relative to all phases.

Instead, use specific names for species and (applicable) phase, as in the following examples:

```
SHOW_VALUE acr(fe+2,aqueous)Lists activity of the Fe+2 species referred to
AQUEOUS solution phase

SHOW_VALUE mur(c2,gas)Lists chemical potential of the C2 species referred to GAS
mixture phase

SET_AXIS_VAR y ac(o2,gas)Sets activity of the O2 species referred to GAS mixture
phase as Y-axis
```

---

## Units of State Variables and Derived Variables

---

The units for the state variables in the Units column in the table in [User-Specified Units](#) are the SI units that are used in the POLY module. However, in the POST module, you can both use pre-defined symbols and define your own symbols which are based on state variables and derived variables. By doing this, you can express the value of the thermodynamic variables in non-default units. The following topic briefly describes how you can create your own symbols and suggests expressions that you can use to convert default units into other standard units which are not default units in Thermo-Calc.

The topic [Examples of User-Defined Units](#) describes some variables of specific species in a phase and the units in which they are expressed. These variables are not used in POLY or POST, but in the GIBBS, DATA, PARROT and ED\_EXP modules.

## User-Specified Units

---

You do not always need to use the default units for variables. Using [ENTER\\_SYMBOL](#), you can convert the value of variable from the default unit to your preferred unit. (Some response-driven modules also allow you to set conditions in non-default units.) User-defined symbols in the form of functions can be used for plotting in the POST module, but they cannot be used as conditions in POLY. User defined symbols in the form of variables can be used as conditions.

For example, if you want pressure to be plotted in bar rather than Pa, then you define the following function `PB`:

```
Enter_symbol function PB = P/1E5 ;
```

If you wanted heat capacity of a system to be shown as in terms of J/mol/K, then you can define the following function `Cp`:

```
Enter_symbol function Cp = Hm.T ;
```

If the variable you want to convert is a function of one or more derivatives, then you must enter the symbol as a variable rather as a function. For instance, a symbol expressing heat capacity of a system in a unit of cal/mol/K can be defined as the following variable `Cp2`:

```
Enter_symbol variable Cp2 = Hm.T/4.1858 ;
```



[Examples of User-Defined Units](#) for information about entering conventional functions and the associated units.

## Examples of User-Defined Units

The following tables give examples of expressions you can use to enter symbols. This enables you to show the values of thermodynamic variables in a variety of units.



[Units of State Variables and Derived Variables](#) and [User-Specified Units](#)



You can also use thermodynamic variables with normalizing suffixes in the expression when you enter a symbol. However, the tables do not have examples of these expressions.

In the expressions these are defined as follows:

- $i$  is for an auxiliary index or letter that corresponds to the component  $c$
- $j$  is for an auxiliary index or letter that corresponds to the phase  $ph$ , and
- $ij$  stands for an auxiliary index or letter that corresponds to the component  $c$  in the phase  $ph$ .

### ***Intensive and Extensive Variables of a Defined System***

Quantity	Suggested Name and Expression	Unit
<i>For Intensive Variables of a Defined System</i>		
Temperature	TempC = T-273.15	Celsius (C)
	TempF = 1.8*T-459.67	Fahrenheit (F)
	TempF = 1.8*TempC+32	

Quantity	Suggested Name and Expression	Unit
Pressure	$P_B = P/1E5$	bars (bar)
	$PKb = P/1E8$	kilobars (kbar)
	$P_{Atm} = P/101325$	atmosphere (atm)
	$PSI = P/6894.76$	pounds/sq. inch (psi)
	$PIHg = P/3342.11$	inches of Hg
	$PTor = P/133.322$	Tors (millimeters of Hg)
<i>For Extensive Variables of a Defined System</i>		
Amount	$BKg = 1E-3*B$	kilograms (kg)
	$BTon = 1E-6*B$	tons
Volume	$VDM = 1E-3*V$	cubic decimeters (dm <sup>3</sup> , l)
	$VCM = 1E-6*V$	cubic centimeters (cm <sup>3</sup> )
	$VMM = 1E-9*V$	cubic millimeters (mm <sup>3</sup> )
	$VCI = 1.6387064E-5*V$	cubic inches (in <sup>3</sup> )
	$VLi = 1E-3*V$	liters (l)
Energy	$GCal = G/4.1858$	cal
	$ACal = A/4.1858$	cal
	$UCal = U/4.1858$	cal
	$HCal = H/4.1858$	cal
	$SCal = S/4.1858$	Cal/K

Quantity	Suggested Name and Expression	Unit
Heat Capacity	$Cp2 = HM.T/4.1858$	cal/mol/K
	$Cp3 = HW.T/4.1858$	cal/g/K
	$Cp4 = HV.T/4.1858$	cal/m <sup>3</sup> /K
Thermal Expansion	$A2 = VM.T*1E-6/VM$	1/K
	$A3 = VW.T*1E-6/VW$	1/K
Isothermal Compressibility	$B2 = -VM.P*1E-9/VM$	1/Pa
	$B3 = -VW.P*1E-9/VW$	1/Pa

### ***Intensive and Extensive Variables of a System Component***

Quantity	Suggested Name and Expression	Unit
<i>For Intensive Variables of a System Component</i>		
Chemical Potential	$MU_i = MU(c)/4.1858$	cal/mol
	$MUR_i = MUR(c)/4.1858$	cal/mol
Special quantity for aqueous solution	$pH = -\text{LOG}_{10}(\text{ACR}(H+1))$	dimensionless
	$E_h = MUR(ZE)/RNF$ where $RNF=96485.309$	volts (V)
<i>For Extensive Variables of a System Component</i>		

Quantity	Suggested Name and Expression	Unit
Amount	$MF_i = N(c)/N = X(c)$	dimensionless
	$MP_i = N(c)/N*100$	dimensionless
	$WF_i = B(c)/B = W(c)$	dimensionless
	$WP_i = B(c)/B*100$	dimensionless
	$BK_{gi} = 1E-3*B(c)$	kilograms (kg)
	$BT_{oni} = 1E-6*B(c)$	tons

### ***Extensive Variables of a Phase***

Quantity	Suggested Name and Expression	Unit
Amount	$MF_j = NP(ph)/N$	dimensionless
	$MP_j = NP(ph)/N*100$	dimensionless
	$WF_j = BP(ph)/B$	dimensionless
	$WP_j = BP(ph)/B*100$	dimensionless
	$BK_{gj} = 1E-3*BP(ph)$	kilograms (kg)
	$BT_{onj} = 1E-6*BP(ph)$	tons
Volume	$VCM_j = 1E-3*V(ph)$	cubic decimeters (dm <sup>3</sup> , l)
	$VCM_j = 1E-6*V(ph)$	cubic centimeters (cm <sup>3</sup> )
	$VMM_j = 1E-9*V(ph)$	cubic millimeters (mm <sup>3</sup> )
Energy	$GCal_j = G(ph)/4.1858$	cal
	$ACal_j = A(ph)/4.1858$	cal

Quantity	Suggested Name and Expression	Unit
Isothermal Compressibility	$B_{2j} = -VM(ph).P/VM(ph)$	1/Pa

### ***Extensive Variables of a Component in a Phase***

Quantity	Suggested Name and Expression	Unit
Amount	$MF_{ij} = N(ph,c)/NP(ph) = X(ph,c)$	dimensionless
	$MP_{ij} = N(ph,c)/NP(ph)*100$	dimensionless
	$WFi = B(ph,c)/BP(ph) = W(ph,c)$	dimensionless
	$WP_{ij} = B(ph,c)/BP(ph)*100$	dimensionless
	$BKg_{ij} = 1E-3*B(ph,c)$	kilograms (kg)
	$BTon_{ij} = 1E-6*B(ph,c)$	tons

### ***Intensive and Extensive Variables of a Species in a Phase***

Quantity	Suggested Name and Expression	Unit
<i>For Intensive Variables of a Species in a Phase</i>		
Chemical Potential	$MUCal_{ij} = MU(sp,ph)/4.1858$	cal/mol
	$MUErg_{ij} = MU(sp,ph)*1E7$	cal/mol
Fugacity	$FA_{ij} = AC(sp,ph)*PATM$	atmosphere (atm)
	$FB_{ij} = AC(sp,ph)*PB$	bars (bar)
	$FC_{ij} = AC(sp,ph)*PKB$	kilobars (kbar)
	$FT_{ij} = AC(sp,ph)*PTOR$	Tors (millimeters of Hg)

Quantity	Suggested Name and Expression	Unit
ln(Fugacity)	$\text{LnFAij} = \text{LNAC}(\text{sp,ph}) + \text{LN}(\text{PATM})$	atmosphere (atm)
	$\text{LnFBij} = \text{LNAC}(\text{sp,ph}) + \text{LN}(\text{PB})$	bars (bar)
	$\text{LnFCij} = \text{LNAC}(\text{sp,ph}) + \text{LN}(\text{PKB})$	kilobars (kbar)
	$\text{LnFTij} = \text{LNAC}(\text{sp,ph}) + \text{LN}(\text{PTOR})$	Tors (millimeters of Hg)
Special quantity for aqueous solution	$\text{Alij} = \text{ACR}(\text{FE+3,AQ}) * \text{AH2O}$	dimensionless
	$\text{RCij} = \text{ACR}(\text{FE+3,AQ}) * \text{YH2O} / \text{Y}(\text{AQ,FE+3})$ where $\text{AH2O} = 55.508435$ and $\text{YH2O} = \text{Y}(\text{AQ,H2O})$	dimensionless
	$\text{WRCalij} = \text{WR}(\text{AQ,sp}) / 4.1858$ ( $\text{WR}(\text{AQ,sp})$ is only valid for aqueous species in a system in which the AQUEOUS solution phase is considered)	cal/mol
<i>For Extensive Variables of a Species in a Phase</i>		
Special quantity for aqueous solution	$\text{MLij} = \text{Y}(\text{AQ,FE+3}) * \text{AH2O} / \text{YH2O}$	molality (m)
	$\text{ISTR} = 1 * \text{IS1} + 1 * \text{IS2} + 1 * \text{IS3}$ where $\text{AH2O} = 55.508435$ , $\text{YH2O} = \text{Y}(\text{AQ,H2O})$ and $\text{ISn} = .5 * \text{MLi} * \text{Zi}^{**2} + .5 * \text{MLj} * \text{Zj}^{**2} + .5 * \text{MLk} * \text{Zk}^{**2}$	molality (m)

### ***Units for Variables of Specific Species in a Phase***

In the GIBBS, DATA, PARROT and ED\_EXP modules, there are quantities that describe various thermodynamic, physical, chemical and transport properties for a compound phase, and for a certain species or a given species combination in a specific solution phase. The following table describes the units for those variables.



These variables cannot be directly used in the POLY or POST modules.

Variables	Description and Unit
G(ph,sp) G(ph,sp;0) G(ph,sp1;sp2;...;0)	Gibbs energy (J/mol) of a pure substance or end-member.
G(ph,sp1,sp2,...) L(ph,sp1,sp2,...) G(ph,sp1,sp2,...;0) L(ph,sp1,sp2,...;0) G(ph,sp1,sp2,...;...;0) L(ph,sp1,sp2,...;...;0)	Zero-order interaction parameter (J/mol) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
G(ph,sp1,sp2,...;i) L(ph,sp1,sp2,...;i) G(ph,sp1,sp2,...;...;i) L(ph,sp1,sp2,...;...;i)	The <i>i</i> th -order interaction parameter (J/mol) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
TC(ph,sp) TC(ph,sp;0) TC(ph,sp1;sp2;...;0)	Curie temperature (K) of a pure substance or end-member.
TC(ph,sp1,sp2,...) TC(ph,sp1,sp2,...;0) TC(ph,sp1,sp2,...;...;0)	Zero-order Curie-temperature contribution (K) to a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
TC(ph,sp1,sp2,...;i) TC(ph,sp1,sp2,...;...;i)	The <i>i</i> th -order Curie-temperature contribution (K) to a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
BMAGN(ph,sp) BMAGN(ph,sp;0) BMAGN (ph,sp1;sp2;...;0)	Bohr magneton number (dimensionless) of a pure substance or end-member.
BMAGN(ph,sp1,sp2,...) BMAGN (ph,sp1,sp2,...;0)	Zero-order Bohr magneton number (dimensionless) to a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.

Variables	Description and Unit
BMAGN (ph,...;sp1,sp2,...;0)	
BMAGN (ph,sp1,sp2,...;i) BMAGN (ph,sp1,sp2,...;...;i)	The <i>i</i> th -order Bohr magneton number (dimensionless) to a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
V0(ph,sp) V0(ph,sp;0) V0(ph,sp1;sp2;...;0)	Molar volume (m <sup>3</sup> ) at 1 bar and reference T0 of a pure substance or end-member.
V0(ph,sp1,sp2,...) V0(ph,sp1,sp2,...;0) V0(ph,sp1,sp2,...;...;0)	Zero-order composition-dependent molar volume (m <sup>3</sup> ) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
V0(ph,sp1,sp2,...;i) V0(ph,sp1,sp2,...;...;i)	The <i>i</i> th-order composition-dependent molar volume (m <sup>3</sup> ) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
VA(ph,sp) VA(ph,sp;0) VA(ph,sp1;sp2;...;0)	Integrated thermal expansivity (m <sup>3</sup> /mol/K) of a pure substance or end-member.
VA(ph,sp1,sp2,...) VA(ph,sp1,sp2,...;0) VA(ph,sp1,sp2,...;...;0)	Zero-order composition-dependent thermal expansivity (m <sup>3</sup> /mol/K) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
VA(ph,sp1,sp2,...;i) VA(ph,sp1,sp2,...;...;i)	The <i>i</i> th -order composition-dependent thermal expansivity (m <sup>3</sup> /mol/K) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
VC(ph,sp)	High pressure fitting parameter (m <sup>3</sup> /mol) of a pure substance or endmember.
VC(ph,sp1,sp2,...)	Zero-order composition-dependent high pressure fitting parameter (m <sup>3</sup> /mol) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
VC(ph,sp1,sp2,...;i)	The <i>i</i> th -order composition-dependent high pressure fitting parameter (m <sup>3</sup> /mol) of a specified binary, ternary or higher-order interactions on a certain sublattice site

Variables	Description and Unit
	in a solution phase.
VK(ph,sp)	Isothermal compressibility ( $\text{m}^3/\text{mol}/\text{Pa}$ ) of a pure substance or endmember.
VK(ph,sp1,sp2,...)	Zero-order composition-dependent isothermal compressibility ( $\text{m}^3/\text{mol}/\text{Pa}$ ) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
VK(ph,sp1,sp2,...i)	The $i$ th -order composition-dependent isothermal compressibility ( $\text{m}^3/\text{mol}/\text{Pa}$ ) of a specified binary, ternary or higher-order interactions on a certain sublattice site in a solution phase.
WR(ph,sp) WR(ph,sp;0)	Standard Born function ( $\text{J}/\text{mol}$ ) of a specific aqueous species in the AQUEOUS solution phase (always with a single sublattice) under the reference-state temperature and pressure.

## State Variables for Scheil

Post processing is done the same way as POLY-3: specific variables for plotting in the Scheil module are listed.



[POLY Module Commands](#)



For a comparable list in Graphical Mode, see [Scheil Calculator Available Plot Variables](#).

The variables can be plotted along the X- and Y-axes. In the table, *ph* is an abbreviation of *phase*, and *el* is an abbreviation of *element*. An asterisk (\*) can be used as a wild character for *el* and *ph*. Use the abbreviations of the variables when you specify what the X- and Y-axes represent in a diagram.

Abbreviation	State Variable
T	Temperature in Celsius
NL/BL/VL	Mole/mass fraction of liquid
NS/BS/VS	Mole/mass fraction of all solid phases
NS(ph)/BS(ph)	Mole/mass fraction of a solid phase
VS(ph)	Volume fraction of a solid phase
W(ph,el)	Weight fraction of an element in a phase
X(ph,el)	Mole fraction of an element in a phase
Y(ph,el)	Site fraction of an element in a phase
NN(ph,el)	Distribution of an element in a phase
NH/BH	Total heat per mole/gram (top curve) and Latent heat per mole/gram (bottom curve)
CP/BCP	Apparent heat capacity per mole/gram
NV/NV(ph)	Molar volume of the system or a phase
DS/DS(ph)	Average density of the system or a phase

---

<i>Abbreviation</i>	<i>State Variable</i>
BT	Apparent volumetric CTE of the system
DVIS(ph)	Dynamic viscosity of a phase
KVIS(ph)	Kinematic viscosity of a phase
SURF(ph)	Surface tension of a liquid phase
ELRS/ELRS(ph)	Electrical resistivity of the system or a phase
ELCD/ELCD(ph)	Electrical conductivity of the system or a phase
THCD/THCD(ph)	Thermal conductivity of the system or a phase
THRS/THRS(ph)	Thermal resistivity of the system or a phase
THDF/THDF(ph)	Thermal diffusivity of the system or a phase
DGV	Driving force for evaporation
DHV	Evaporation enthalpy
MMG	Molar mass of gas
XAVG(el)	Mole fraction of an element in solid phases
WAVG(el)	Mass fraction of an element in solid phases

## State Variables in Console Mode DICTRA

State variables in the DICTRA POST module are similar to those defined in Thermo-Calc POLY-3 module. Examples of state variables are temperature, mole fraction, enthalpy, etc. In POLY-3 a general notation method based on character mnemonics is designed for a predefined set of state variables.

The numerical values of some state variables, e.g. activity, chemical potential, and the energetic extensive properties, are given relative to some reference state which has been defined in the thermodynamic database file. Sometimes you may have specified a reference state using SET\_REFERENCE\_STATE. To get numerical values relative to the user-defined reference state, append an R to the mnemonic names as defined.

### Intensive Properties

Mnemonic	Description
T	temperature
P	pressure
AC(component)	activity
MU(component)	chemical potential

### Extensive Properties

NORMALIZATION of extensive properties: For all extensive properties a suffix can be added to the mnemonic name to indicate a normalized extensive property. The following tables describe each mnemonic .

#### ***Extensive Property `Z at Each Grid Point***

Mnemonic	Description
Z	Extensive property `Z at each grid point.
ZM	Extensive property `Z per moles of atoms.

Mnemonic	Description
ZW	Extensive property $Z$ per mass (gram).
ZV	Extensive property $Z$ per volume (m <sup>3</sup> ).

### ***Extensive Property $Z$ for a Phase at Each Grid Point***

Mnemonic	Description
Z	Extensive property $Z$ for the current amount of the phase.
ZM	Extensive property $Z$ per mole of atoms of the phase.
ZW	Extensive property $Z$ per mass (gram) of the phase.
ZV	Extensive property $Z$ per volume (m <sup>3</sup> ) of the phase.
ZF	Extensive property $Z$ per mole formula unit of the phase.

### ***Energetic Extensive Properties***



Suffixes M, W, V and F can be applied.

Mnemonic	Description
S	entropy
S(phase)	entropy of a phase
V	volume
V(phase)	volume of a phase
G	Gibbs energy

Mnemonic	Description
G(phase)	Gibbs energy of a phase
H	enthalpy
H(phase)	enthalpy of a phase
A	Helmholtz energy
A(phase)	Helmholtz energy of a phase

### ***Amount of Components***



Suffixes M, W and V can be applied.

Mnemonic	Description
N(component)	Number of moles of a component.
N(phase,component)	Number of moles of a component in a phase.
B(component)	Mass of a component.
B(phase,component)	Mass of a component in a phase.

### ***Total Amount***



Suffixes M, W and V can be applied.

Mnemonic	Description
N	Number of moles

Mnemonic	Description
B	<p>Mass</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;">  The legal combination BW is not very interesting as it will always be unity. BV is the density         </div>

### ***Amount of a Phase***



Suffixes M, W and V can be applied.



The normalizing properties are calculated at each grid point.

Mnemonic	Description
NP(phase)	Number of moles of a phase.
BP(phase)	Mass of a phase.
VP(phase)	Volume of a phase.
Y(phase,species#sublattice)	<p>site fraction</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;">  This quantity is dependent upon the model chosen for the phase.         </div>

### ***Constitution***



This quantity is dependent upon the model chosen for the phase.

Mnemonic	Description
Y(phase,species#sublattice)	site fraction

---

# Thermophysical Properties Data and Variables

---

**In this section:**

---

About Thermophysical Properties Data Parameters .....	49
Thermophysical Properties Variables .....	53
Electrical Resistivity Model .....	55
Electrical Conductivity for Ionic Liquid Model .....	59
Thermal Conductivity Model .....	61
Surface Tension of Metallic Liquid Alloys Model .....	65
Surface Tension Model for Oxide Slag .....	69
Viscosity of the Metallic Liquids Model .....	70
Viscosity of the Ionic Liquids Model .....	72
Molar Volume Model .....	73

## About Thermophysical Properties Data Parameters

 [Parameters, Functions, and Variables](#)

 [Thermophysical Properties Variables](#)



The so-called classic syntax is used in the examples below. It can be used in Console Mode or when using the APIs, e.g. TC-Python. Graphical Mode in Thermo-Calc has readable names for all of these.



You can find information on our website about the [properties that can be calculated](#) with Thermo-Calc and the Add-on Modules. Additional resources are added on a regular basis so keep checking back or [subscribe to our newsletter](#).

### All Available Properties Data Parameters and Variables

Below is a summary of the available parameters and variables for the databases when working in Thermo-Calc. There are differences when you are working in Console Mode versus Graphical Mode as well as if you use the TC-Python SDK. More details are described in the online help.

<i>Property</i>	<i>Model Parameters</i>	<i>Variables to Show or Plot in Console Mode and TC-Python</i>
Electrical conductivity	ELQ**	ELCD for a system $ELCD(PHI)$ for phase PHI
Electrical resistivity	ELRS, ESPD	ELRS for a system $ELRS(PHI)$ for a phase PHI
Thermal conductivity	THCD	THCD for a system $THCD(PHI)$ for phase PHI
Thermal resistivity		THRS for a system $THRS(PHI)$ for phase PHI

<i>Property</i>	<i>Model Parameters</i>	<i>Variables to Show or Plot in Console Mode and TC-Python</i>
Thermal diffusivity		THDF for a system $THDF(PHI)$ for phase PHI
Surface tension	SIGM, XI*	SURF(LIQUID) SURF(ION)**
Dynamic viscosity	VISC	DVIS(LIQUID) DVIS(ION)**
Kinematic viscosity		KVIS(LIQUID) KVIS(ION)**
Molar volume	VO, VA	VM for a system $VM(PHI)$ for phase PHI
<p>* XI is not used in the TCOX database (all versions). As of 2023b it is also not used starting with the following versions of these databases: TCFE13, TCNI12.1, TCTI5.1, TCNOBL3, TCPMAG2, and TCCU6. As of 2024a, TCMG7, TCAL9, and TCHEA7.</p> <p>** ION is used in the TCS Metal Oxide Solutions Database (TCOX)</p>		

## Viscosity Model



[Viscosity of the Metallic Liquids Model](#)



[Viscosity of the Ionic Liquids Model](#)

To use this model, the database parameter VISC is needed. It can be used for liquids.

- Dynamic viscosity (DVIS)
- Kinematic viscosity (KVIS)

Examples in Console Mode:

- In POLY module: SHOW DVIS(LIQUID)
- Set as plot variable: SET-DIAGRAM-AXIS Y KVIS(LIQUID)

---

## **Thermal Conductivity Model**

### [Thermal Conductivity Model](#)

To use the thermal conduction parameters, the database parameter THCD is needed. The other two related properties are derived from THCD. These properties can be obtained for a phase or for the whole system.

- Thermal conductivity (THCD)
- Thermal diffusivity (THDF)
- Thermal resistivity (THRS)

Examples in Console Mode:

- In POLY module: SHOW THCD
- Set as plot variable: SET-DIAGRAM-AXIS Y THDF(FCC\_A1)

## **Electrical Conductivity and Electrical Resistivity Models**

### [Electrical Resistivity Model](#)

To use the electric properties parameters, the database parameter ELRS is needed. ELCD is derived from that. These properties can be obtained for a phase or for the whole system.

- Electrical conductivity (ELCD)
- Electrical resistivity (ELRS)

Examples in Console Mode:

- In POLY module: SHOW ELCD
- Set as plot variable: SET-DIAGRAM-AXIS Y ELRS(FCC\_A1)

## **Electrical Conductivity of Molten Slag Model**

### [Electrical Conductivity for Ionic Liquid Model](#)

To use the electric properties parameters, the database parameter ELQ is needed. The model parameters are the activation Gibbs energy of end-members and excess activation Gibbs energy.

- Electrical conductivity of molten slags  $ELCD(IONIC)$
- Electrical resistivity of molten slags  $ELRS(IONIC)$

## Surface Tension Model

-  [Surface Tension of Metallic Liquid Alloys Model](#)
-  [Surface Tension Model for Oxide Slag](#)

Surface tension for liquids can be calculated if the database contains the two database parameters SIGM and XI. From these, the surface energy is calculated:

- Surface energy (SURF)



XI is not used in the TCS Metal Oxide Solutions Database (TCOX)

Examples in Console Mode:

- In POLY module: SHOW SURF(LIQUID)
- Set as plot variable: SET-DIAGRAM-AXIS Y SURF(LIQUID)

## Thermophysical Properties Variables

The Thermo-Calc databases have a variety of properties data included such as molar volume, thermal conductivity, dynamic and kinematic viscosity, thermal resistivity, electrical resistivity, and surface tension. These properties data are being integrated into databases as each new version of a database is released.



Availability of properties data is dependent on both the database and its version. Some information is specific to use in Console Mode. There is more information also available for Graphical Mode and TC-Python for each of the databases where the properties data is included. More advanced information is also contained in the *Database Manager User Guide* [PARAMETER](#) section included with this documentation set.



To determine if the database you are using has this properties data, search the help (press F1) to review the technical information. The availability of properties data is also included with the Database Overview document available [on the Thermo-Calc website](#).

Variables	Abbrev.	Unit	Descript.	Domain
DVIS	DVIS(ph)	Pa·s	Dynamic viscosity	Liquid phase
KVIS	KVIS(ph)	m <sup>2</sup> /s	Kinematic viscosity	Liquid phase
ELCD	ELCD	S/m	Electric conductivity	System
ELCD	ELCD(ph)	S/m	Electric conductivity	Phase
ELRS	ELRS	Ωm	Electric resistivity	System
ELRS	ELRS(ph)	Ωm	Electric resistivity	Phase
THCD	THCD	W/(mK)	Thermal conductivity	System

---

Variables	Abbrev.	Unit	Descript.	Domain
THCD	THCD(ph)	W/(mK)	Thermal conductivity	Phase
THRS	THRS	mK/W	Thermal resistivity	System
THRS	THRS(ph)	mK/W	Thermal resistivity	Phase
THDF	THDF	m <sup>2</sup> /s	Thermal diffusivity	System
THDF	THDF(ph)	m <sup>2</sup> /s	Thermal diffusivity	Phase
SURF	SURF(ph)	J/m <sup>2</sup>	Surface tension	Liquid phase

## Electrical Resistivity Model

---

The modeling starts with unaries in their stable and metastable structures. The temperature dependence of electrical resistivity is described with polynomials for each unary and each end-member of compounds. In order to guarantee the reliability of the extrapolation, experimental data were first fitted to a semi-empirical model, which was established based on relevant theories. The experimental data, together with the extrapolated values, are then used for deriving the polynomials that are saved in the database. The magnetic contribution to electrical resistivity can be separately treated with a model implemented in Thermo-Calc.

The composition dependence of electrical resistivity is described with the Redlich-Kister expression. All binary systems are modeled by using experimental data or theoretical estimations. Ternary and higher-order parameters can be used if necessary.

Electrical resistivity is modeled and its descriptions are stored in the databases. Electrical conductivity is derived from electrical resistivity and can be retrieved from calculated results as well.

### Model Description

The semi-empirical model for electrical resistivity is built based on the Matthiessen's rule [1997DeB]. In general, the electrical resistivity of a metallic alloy consists of residual resistivity ( $\rho_0$ ) and electron-phonon resistivity ( $\rho_{e-p}$ ), as well as magnetic resistivity ( $\rho_m$ ). The residual resistivity is due to the scatterings of impurities and defects that exist in well-annealed and high purity alloys, and the value of  $\rho_0$  is generally considered independent of temperature.

$$[\text{Eq. 1}] \quad \rho = \rho_0 + \rho_{e-p} + \rho_m$$

$\rho_{e-p}$  is described with the modified Bloch-Grüneisen equation [1928Blo; 1933Gru] and corrections can be made in cases where other types of scatterings play considerable roles.

$$[\text{Eq. 2}] \quad \rho_{e-p} = A \cdot \left( \frac{T}{\theta_R} \right)^5 \cdot J_5 \left( \frac{\theta_R}{T} \right)$$

$$[\text{Eq. 3}] \quad A = \frac{\frac{3}{2} h q_D^6 (G')^2}{4e^2 (m^*)^2 n_c k_B \theta_D k_F^2 v_F^2}$$

$$[Eq. 4] \quad J_n \left( \frac{\theta_D}{T} \right) = \int_0^{\frac{\theta_D}{T}} \frac{x^n e^x}{(e^x - 1)^2} dx$$

The characteristic temperature  $\theta_R$  plays a similar role and is also approximated as the Debye temperature ( $\theta_D$ ), which can be readily obtained from low-temperature heat capacity and elastic constants data as well as theoretical computations. The approximation of  $\theta_R$  as  $\theta_D$  indicates that transverse phonons play a role in electron scattering.  $k_F$  is the electron wave number at the Fermi surface.  $q_D$  is the phonon Debye wave number.  $\hbar$  is the Planck constant.  $G'$  is a constant representing the strength of the e-p interaction and cannot be easily estimated.  $m^*$  is the electron effective mass.  $n_c$  is the number of unit cells per unit volume.  $v_F$  is the electron velocity at the Fermi surface.

To describe the drastic change of the magnetic resistivity ( $\rho_m$ ) around the Curie (or Neel) temperature,  $T_C$ , of a magnetic alloy, an empirical model is proposed using the dimensionless temperature  $T/T_C$  and a parameter  $\rho_{spd}$  that stands for the resistivity due to spin disordering. According to this model, [Eq. 5](#),  $\rho_m$  approaches  $\rho_{spd}$ , when the temperature is well above  $T_C$ , and is 0 when the temperature is far below  $T_C$ .  $\rho_{spd}$  depends only on composition and is described with the Redlich-Kister (R-K) expression. Apparently,  $\rho_{spd}$  has a positive value for magnetic elements, and compounds as well as endmembers, and is 0 for non-magnetic ones.

In a typical binary system made of a magnetic element A and a non-magnetic element B,  $\rho_{spd}$  decreases usually from  $\rho_{spd}^A$  to 0 at a certain composition ( $x_*$ ). Such a behavior is also seen in magnetic moment and Curie temperature.  $\rho_{spd}$  is expected to be zero in the composition range from  $x_*$  to element B, and will be forced to zero when the R-K expansion leads inevitably to negative values over this composition range.  $T_C$  is available from the thermodynamic description in the database.

$$[Eq. 5] \quad \rho_m = \rho_{spd} \cdot \left( 1 - e^{\left( -\frac{3}{2} \left( \frac{T}{T_c} \right)^3 \right)} \right)$$

The semi-empirical model (as given in [Eq. 1](#) to [Eq. 5](#)) is first fitted to experimental data. The description is then extrapolated to a wider temperature range if necessary, either to extreme temperatures where the structure is no longer stable or to where experimental data are missing. Polynomials are then derived with the experimental data, together with the extrapolated values, and stored in the database. The model as given in [Eq. 5](#) is implemented in Thermo-Calc, so the magnetic contribution can be excluded from the polynomials and separately modeled, and it will be automatically included during the calculations.

---

## Model Parameters

The extended TDB format accepts parameters describing electrical resistivity and thermal conductivity, together with spin disordering resistivity. Examples for unary and binary systems are given below:

```
ELRS (FCC_A1, AL:VA; 0)
ELRS (FCC_A1, AL, CO:VA; 0)
ESPD (FCC_A1, CO:VA; 0)
ESPD (FCC_A1, AL, CO:VA; 0)
```

## Calculation and Plot Variables

- ELRS – Electrical resistivity
- ELCD – Electrical conductivity

All the above quantities can be calculated, retrieved, shown and plot in Thermo-Calc via the variables in the Console Mode or via the quantity names in the Graphic Mode (see sections below).

## Console Mode

The quantities corresponding to the variables of ELRS and ELCD can be calculated in Console Mode for a phase, e.g. ELRS (FCC\_A1), or for a system, i.e. ELRS. The results can be shown in the POLY module with the command SHOW\_VALUE or illustrated as a plot in the POST module with the command PLOT\_DIAGRAM.

## Graphical Mode

On the **Plot Renderer** in Graphical Mode, electrical resistivity, electrical conductivity, thermal conductivity, thermal resistivity, and thermal diffusivity can be selected from the axes lists, tabulated and plotted using the quantity names, with options for a specific phase, all phases or for a system.

## TC-Python and TC-Toolbox for MATLAB®

With these Software Development Kits (SDKs), the quantities of ELRS and ELCD can be retrieved, for a phase, e.g. ELRS (FCC\_A1), or for a system, i.e. ELRS, via `get_value_of()` or `get_values_of()`, from most phase equilibrium calculations in TC-Python, for instance,

- `with_single_equilibrium_calculation()`,
- `with_property_digram_calculation()` **or**
- `with_batch_equilibrium_calculation()`.

## **References**

[1928Blo] F. Bloch, Über die Quantenmechanik der Elektronen in Kristallgittern (About the quantum mechanics of electrons in crystal lattices). Zeitschrift für Phys. 52, 555–600 (1928).

[1933Gru] E. Grüneisen, The dependence of the electrical resistance of pure metals on the temperature. Ann. Phys. 16, 530–540 (1933).

[1997DeB] R. De-Bruyn-Ouboter, Heike Kamerlingh Onnes's discovery of superconductivity. Sci. Am., 98–103 (1997).

## Electrical Conductivity for Ionic Liquid Model

The model for the electrical conductivity of molten slags is mainly based on the theory of the electrical conductivity of ionic solutions. The total conductivity is the sum of the contributions from all cations including those network-forming cations modeled as neutrals in the two sublattice ionic liquid model, such as liquid silicon oxide. For slags containing transition metal ions of different oxidation states, the electronic contribution due to the electron or hole exchange between these ions is also considered.

In short, the electrical conductivity of molten slags consists of the following:

1. Ionic conduction contributed by cations in the ionic liquid.
2. Electronic conduction from the cations having more than one oxidation states (Fe<sup>2+</sup>-Fe<sup>3+</sup>, Mn<sup>2+</sup>-Mn<sup>3+</sup> and so on).

The composition and temperature dependence of the total electrical conductivity of molten slags,  $\sigma$ , is represented by:

$$\begin{aligned}\sigma &= \sum x_{C_i} \sigma_{C_i} - x_{MeO} y_{MeO_3} \sigma_{Me^{2+}} + x_{B_k^0} \sigma_{B_k^0} \\ &= x_{Ca^{2+}} \sigma_{Ca^{2+}} + x_{Mg^{2+}} \sigma_{Mg^{2+}} + \dots + x_{Fe^{2+}} \sigma_{Fe^{2+}} - x_{FeO} y_{FeO_3} \sigma_{Fe^{2+}}^{2+} + \dots + x_{SiO_2} \sigma_{SiO_2}\end{aligned}$$

where  $x_{C_i}$  is the mole fraction of cation  $C_i$ , and its electrical conductivity,  $\sigma_{C_i}$ , is written in the form of the Nernst-Einstein equation as:

$$\sigma_{C_i} = \frac{1}{RT} \exp\left[-\frac{Q_{C_i}}{RT}\right] = \frac{1}{RT} \exp\left[-\frac{\sum_j y_{C_i} y_{A_j} Q_{C_i:A_j} + \sum_j y_{C_i} y_{B_k^0} Q_{C_i:B_k^0} + Q_{C_i}^E}{RT}\right]$$

where  $Q_{C_i}$  is the activation Gibbs energy, of which the composition dependence has been expanded in a CALPHAD fashion.

### Model Parameters

The model parameters are the activation Gibbs energy of end-members and excess activation Gibbs energy in the above equation. They can be entered in the GES module or TDB file by using the keyword `ELQ`, for example,

```
PARAM ELQ (IONIC_LIQ&Ca+2, CA+2:O-2; 0)
PARAM ELQ (IONIC_LIQ&Ca+2, CA+2:SiO4-4; 0)
PARAM ELQ (IONIC_LIQ&Ca+2, CA+2:SiO2; 0)
```

---

## Calculation and Plot Variables

All the quantities can be calculated, retrieved, shown and plot in Thermo-Calc via the variables in the Console Mode or via the quantity names in the Graphic Mode.

### Console Mode

The electrical conductivity and resistivity of molten slags modeled as the ionic liquid phase can be calculated usually through a single point or step equilibrium calculation in Console Mode. The results can be shown or plotted by using the variables `ELCD(IONIC)` and `ELRS(IONIC)`, respectively.

### Graphical Mode

On the **Plot Renderer** in Graphical Mode, the electrical conductivity and resistivity of molten slags modeled as the ionic liquid phase can be calculated by using the **Single point** or **One Axis** calculation type in Graphical Mode. The corresponding tabulation or plot variables are **Electrical Conductivity** and **Electrical Resistivity**, respectively.

### TC-Python and TC-Toolbox for MATLAB®

With these Software Development Kits (SDKs), this property can be plotted using `property-diagram_calculation()` or `with_batch_equilibrium_calculation()`. The plot variables are `ELCD(IONIC)` and `ELRS(IONIC)`. You can also get the individual values e.g. from an equilibrium calculation and use in whatever way you want, e.g. feed into other models: `result.get_value_of("ELCD(IONIC_LIQ)")`, where 'result' is the result of one equilibrium calculation. It matches `SHOW_VALUE` in Console Mode.

## Thermal Conductivity Model

The modeling starts with unaries in their stable and metastable structures. The temperature dependence of thermal conductivity is described with polynomials for each unary and each end-member of compounds. Thermal resistivity and thermal diffusivity can be derived from thermal conductivity and retrieved from calculated results. The composition dependence is described with the Redlich-Kister expression in the CALPHAD spirit, but the relation is applied to thermal resistivity (or the inverse of thermal conductivity).

### Model Description

Experimental data of thermal conductivity are used to derive descriptions in polynomials, which are stored in the database, despite that it consists of two components, electronic thermal conductivity ( $\kappa_e$ ) and lattice thermal conductivity  $\kappa_g$ . In other words, they are not separately treated.

In cases where experimental data are absent or sparse, however,  $\kappa_e$  and  $\kappa_g$  can be separately predicted and summed up to give the total thermal conductivity.  $\kappa_e$  can be predicted from  $\rho$  or vice versa, by employing the Wiedemann-Franz Law [1853Fra; 1985Jon], respectively, to the residual resistivity, magnetic resistivity and intrinsic resistivity.

$$[\text{Eq. 1}] \quad \frac{1}{\kappa_e} = W_e = \frac{\rho_0 + \rho_m + \rho_i}{L_0 T}$$

$L_0$  is known as the Sommerfeld value of the Lorenz number:  $2.4453 \times 10^{-8} \text{W}\Omega/\text{K}^2$ .

$$[\text{Eq. 2}] \quad \rho_i = \frac{A}{L_0 T} \cdot \left(\frac{T}{\theta_R}\right)^5 \cdot J_5\left(\frac{\theta_R}{T}\right) \cdot f$$

$$[\text{Eq. 3}] \quad f = 1 + \frac{3}{\pi^2} \cdot \left(\frac{k_F}{q_D}\right)^2 \cdot \left(\frac{\theta_R}{T}\right)^2 - \frac{1}{2\pi^2} \cdot \frac{J_7\left(\frac{\theta_R}{T}\right)}{J_5\left(\frac{\theta_R}{T}\right)}$$

The intrinsic resistivity ( $\rho_i$ ) is based on [Eq. 4](#)

$$[\text{Eq. 4}] \quad \rho_{e-p} = A \cdot \left(\frac{T}{\theta_R}\right)^5 \cdot J_5\left(\frac{\theta_R}{T}\right)$$

for the e-p processes, but corrections are provided, as given in [Eq. 3](#): the first term corresponds to large-angle scattering that satisfies the original Wiedemann-Franz Law; the second one is due to inelastic small-angle (vertical processes) and has no counterpart in the electrical resistivity; the third term is a correction that accounts for situations where large-angle scattering can reverse the electron direction without actually assisting to restore the distribution back to equilibrium.

$\kappa_g$  can be predicted with the Slack model in a temperature range where only anharmonic Umklapp phonon-phonon interactions are important. In other words, normal processes are neglected as a first approximation [1979Sla; 2002Mor]. The most reasonable version among several ones is given below [1965Jul].

$$[\text{Eq. 5}] \quad \kappa_p^U = A' \cdot \frac{M_a \cdot \theta_D^3 \cdot \delta}{\gamma^2 \cdot n_c^{\frac{2}{3}}} \cdot \frac{1}{T}$$

$$[\text{Eq. 6}] \quad A' = \frac{2.43 \cdot 10^{-5}}{1 - \frac{0.514}{\gamma} + \frac{0.228}{\gamma^2}}$$

The Debye temperature,  $\theta_D$  can be obtained from low-temperature heat capacity or elastic constant data.  $M_a$  is the average atomic mass,  $\delta^3$  the volume per atom,  $n_c$  the number of atoms in a unit cell, and  $\gamma$  the Grüneisen coefficient. A typical value for  $A'$  is evaluated to be  $3 \times 10^{-5}$  using  $\gamma = 2$  as the Grüneisen coefficient, and can be used in the absence of experimental information. It can also be obtained from e.g. DFT calculations of elastic properties.

## Model Parameters

The extended TDB format accepts parameters describing electrical resistivity and thermal conductivity, together with spin disordering resistivity. Examples for unary and binary systems are given below:

```
THCD (FCC_A1, AL:VA; 0)
```

```
THCD (FCC_A1, AL, CO:VA; 0)
```

## Calculation and Plot Variables

- THCD – Thermal conductivity
- THRS – Thermal resistivity
- THDF – Thermal diffusivity

All the above quantities can be calculated, retrieved, shown and plot in Thermo-Calc via the variables in the Console Mode or via the quantity names in the Graphic Mode (see sections below).

## Console Mode

The quantities corresponding to the variables of `THCD`, `THRS`, and `THDF` can be calculated in Console Mode for a phase, e.g. `THCD(FCC_A1)`, or for a system, i.e. `THCD`. The results can be shown in the POLY module with the command `SHOW_VALUE` or illustrated as a plot in the POST module with the command `PLOT_DIAGRAM`.

## Graphical Mode

On the **Plot Renderer** in Graphical Mode, electrical resistivity, electrical conductivity, thermal conductivity, thermal resistivity, and thermal diffusivity can be selected from the axes lists, tabulated and plotted using the quantity names, with options for a specific phase, all phases or for a system.

## TC-Python and TC-Toolbox for MATLAB®

The quantities of `THCD`, `THRS`, and `THDF` can be retrieved, for a phase, e.g. `THCD(FCC_A1)`, or for a system, i.e. `THCD`, via `get_value_of()` or `get_values_of()`, from most phase equilibrium calculations in these Software Development Kits (SDKs), for instance,

- `with_single_equilibrium_calculation()`,
- `with_property_digram_calculation()` or
- `with_batch_equilibrium_calculation()`.

## References

- [1852Fra] R. Franz, G. Wiedemann, Ueber die Wärme-Leitungsfähigkeit der Metalle. Ann. der Phys. und Chemie. 165, 497–531 (1853).
- [1965Jul] C. L. Julian, Theory of Heat Conduction in Rare-Gas Crystals. Phys. Rev. 137, A128–A137 (1965).
- 1979[Slack] G. A. Slack, The Thermal Conductivity of Nonmetallic Crystals, in Solid State Physics Volume 34, 1–71 (1979).
- [1985Jon] W. Jones, N. H. March, Theoretical Solid State Physics (Courier Dover Publications, 1985).

[2002Mor] D. T. Morelli, J. P. Heremans, G. A. Slack, Estimation of the isotope effect on the lattice thermal conductivity of group IV and group III-V semiconductors. Phys. Rev. B. 66, 195304 (2002).

## Surface Tension of Metallic Liquid Alloys Model

In general, for liquid metal alloys, the surface tension is assessed for all the elements and binary systems where experimental data are available. When experimental data are not available, the Butler model [1932But] is used to estimate the composition dependence of the surface tension. The experimental or estimated data are modeled with either the modified Guggenheim model [2019Ver] or the Redlich-Kister-Muggianu sub-regular solution model [1948Red; 1975Mug].



Search the help (press F1) to review the technical information for the database you are using.

### Modified Guggenheim Model Description

The surface tension of metallic liquid alloys by the modified Guggenheim model [2019Ver] and is expressed as:

$$\sigma(x_i^B, T) = -\frac{RT}{S(x_i^B, T)} \ln \left[ x_1^B \exp \left( -(1 + \Xi_1 \cdot x_2^S(x_i^B, T)) \frac{\sigma_1(T) S_1(T)}{RT} \right) + x_2^B \exp \left( -(1 + \Xi_2 \cdot x_1^S(x_i^B, T)) \frac{\sigma_2(T) S_2(T)}{RT} \right) \right]$$

where

- $x_i^B$  is the bulk composition
- $\sigma_i$  is the surface tension
- $S$  and  $S_i$  are the total and the partial molar surface of species  $i$  defined as

$$S_{(i)} = 1.901 \cdot N_A^{1/3} \cdot V_{m,(i)}^{2/3}$$

- $N_A$  being the Avogadro number and  $V_{m,(i)}$  the molar volume of the system or the species  $i$ .
- $x_i^S$  is the composition of  $i$  in surface, which is a function of bulk composition and  $T$  and defined as follows:

$$x_i^{S,id}(x_i^B, T) = \frac{x_i^B}{x_i^B + x_j^B \exp \left( \frac{S(x_i^B, T) [\sigma_i(T) - \sigma_j(T)]}{RT} \right)}$$

- $\Xi_1$  and  $\Xi_2$  are called damping factors and these parameters are fitted for species 1 and 2 of binary systems.

The extrapolation to higher-order systems is straightforward. The ideal surface composition for a multicomponent system is:

$$x_i^{S,id}(T, x_i^B, x_j^B, \dots) = \frac{x_i^B}{x_i^B + \sum_{j \neq i} x_j^B \exp\left(\frac{S_{ij}(x_i^B, T) \cdot |\sigma_i(T) - \sigma_j(T)|}{RT}\right)}$$

and the surface tension of a multicomponent alloy is obtained from:

$$\sigma(T, x_i^B, x_j^B, \dots) = -\frac{RT}{S(T, x_i^B, x_j^B, \dots)} \cdot \ln \left[ \sum_{i=1}^n \sum_{j \neq i} x_i^B \exp\left(-\left(1 + x_j^S(T, x_i^{B,ij}) \cdot \Xi_i^{ij}\right) \frac{\sigma_i(T) S_i(T)}{RT}\right)\right]$$

### Corresponding Variable - Modified Guggenheim

The corresponding variable for surface energy of pure elements is `SIGM`. For instance, the surface tension of pure A is written as `SIGM(LIQUID, A; 0)`. The corresponding  $X_1$  and  $X_2$  parameters for A-B binary system are respectively written as following:

```
PARAMETER XI (LIQUID, A, B; 0)
```

```
PARAMETER XI (LIQUID, A, B; 1)
```

## Redlich-Kister-Muggianu Model Description

In the case of using the Redlich-Kister-Muggianu (R-K-M) [1948Red; 1975Mug] sub-regular solution model for modeling the surface tension, the surface tension is given by:

$$\sigma = \sum y_i \sigma_i + \sigma^E$$

where  $y_i$  and  $\sigma_i$  are the site fraction and surface tension for pure component  $i$ , and the excess term is expressed using (R-K) for binary and (R-K-M) for ternary interactions.

### Corresponding Variable Redlich-Kister-Muggianu

The corresponding variable for surface energy of pure elements is `SIGM`. For instance, the surface tension of pure A is written as `SIGM(LIQUID, A; 0)`. The corresponding  $L$  parameters for the A-B binary system are written as follows:

```
PARAMETER SIGM (IONIC_LIQ, A, B; 0)
```

```
PARAMETER SIGM (IONIC_LIQ, A, B; 1)
```

---

PARAMETER SIGM (IONIC\_LIQ, A, B; 2)

## Plot Variables (Both Models)

### Console Mode

The surface energy can be calculated through step calculations in Console Mode. The plot variable is `SURF(LIQUID)`.

### Graphical Mode

The surface energy is calculated using the **One Axis Calculation Type** in Graphical Mode. The corresponding plot variable is **Surface tension**.

### TC-Python and TC-Toolbox for MATLAB®

With these Software Development Kits (SDKs), this property can be plotted using `property-diagram_calculation()` or `with_batch_equilibrium_calculation()`. The plot variable is `SURF(LIQUID)`.

## Graphical Mode and Console Mode Examples

There are two basic examples included with your software installation and that use demonstration (DEMO) databases.

Search the help (press F1 when in Thermo-Calc) for brief descriptions of these examples, or browse to the examples located in your installation. From the main menu, **Help**→**Examples Files**:

- Graphical Mode `T_11_Surface_tension_in_Cu-Zr.tcu`
- Console Mode `tcex56`

## References

[1932But] J. A. V. Butler, The thermodynamics of the surfaces of solutions. Proc. R. Soc. London. Ser. A, Contain. Pap. a Math. Phys. Character. 135, 348–375 (1932).

[1948Red] O. Redlich, A. T. Kister, Algebraic Representation of Thermodynamic Properties and the Classification of Solutions. Ind. Eng. Chem. 40, 345–348 (1948).

[1975Mug] Y. M. Muggianu, M. Gambino, J. P. Bros, Enthalpy of formation of liquid Bi-Sn-Ga alloys at 723 K, choice of an analytical expression of integral and partial excess quantities of mixing. J. Chim. Phys. 72, 83–88 (1975).

[2019Ver] M. Vermot des Roches, A. E. Gheribi, P. Chartrand, A versatile multicomponent database for the surface tension of liquid metals, *Calphad*. 65, 326–339 (2019).

## Surface Tension Model for Oxide Slag

---

The model for TCS Metal Oxide Solutions Database (TCOX) surface tension of oxide slag:

$$\sigma = \sum y_i \sigma_i + \sigma^E$$

$$\sigma^E = \sum_i y_A \cdot y_B (y_A - y_B)^i \cdot SIGM^i$$

where  $y_i$  and  $\sigma_i$  indicate the site fraction and surface tension for pure component  $i$ . For pure component, the surface tension is described with  $a + b * T$ . The excess term is expressed using Redlich-Kister polynomial, as one can find the example of a binary A-B.

SIGM is the parameter used for optimization, for example,

```
PARAMETER SIGM (IONIC_LIQ, A;0)
PARAMETER SIGM (IONIC_LIQ, B;0)
PARAMETER SIGM (IONIC_LIQ, A:B,0)
```

The user can calculate the surface tension of liquid oxide by calling the variable `SURF (IONIC)`.

## Viscosity of the Metallic Liquids Model

---

This section provides details about calculating the kinematic viscosity variable.

### Model Description

The viscosity of metallic liquid alloys is modeled as follows:

$$RT \ln \eta = RT \ln \eta_0 + E$$

where

- $\eta$  is the dynamic (or shear) viscosity of the liquid,
- $\eta_0$  is the viscosity at finite temperatures,
- $E$  is the activation energy in J/mole,
- $R$  is the gas constant, and
- $T$  is the absolute temperature.

The SI unit of viscosity is pascal.second (Pa.s). The viscosity parameters are expanded via Redlich-Kister polynomials.

For example, viscosity of a liquid A-B alloy is:

$$\begin{aligned} RT \ln \eta_{\text{alloy}} = & x_A (E_A + RT \ln \eta_0^A) \\ & + x_B (E_B + RT \ln \eta_0^B) \\ & + x_A x_B (L_0 + (x_A - x_B) \cdot L_1) \end{aligned}$$

The two first terms are the contributions for the end-members, A and B elements and the last term accounts for the excess viscosity. Basically, the activation energies are optimized.  $L_0$  and  $L_1$  etc. are the parameters that are optimized to fit the viscosity data.

The kinematic viscosity,  $\nu$ , is the ratio of the dynamic viscosity,  $\eta$ , to the density,  $\rho$ , of the alloy:

$$\nu = \eta / \rho$$

The SI unit of kinematic viscosity is square meter per second (m<sup>2</sup>/s).

---

## Corresponding Variables

In Console Mode, the viscosity can be plotted via a step calculation vs. temperature or composition. The calculated property in Thermo-Calc is VISC of the liquid phase. The calculated VISC property is not the viscosity ( $\eta$ ) but is related to  $\eta$  via  $RT \ln(\eta)$ , thus in order to plot the viscosity or  $\eta$  you define a function for  $\eta$  as follows:

$$\eta = \exp(\text{VISC}(\text{LIQUID})/RT)$$

The default unit is Pa.s.

However, it is also possible to directly get the values for dynamic and kinematic viscosity in Console Mode and using TC-Python and TC-Toolbox for MATLAB®. The relevant properties are DVIS (dynamic viscosity) and KVIS (kinematic viscosity). For example, once the equilibrium is calculated in Console Mode, one can get the viscosity value by:

```
show DVIS(liq) OR show KVIS(liquid).
```

In Graphical Mode, the viscosity ( $\eta$ ) is directly plotted when using the **One Axis Calculation Type** available with various calculators.

## Graphical Mode and Console Mode Examples

There are two basic examples included with your software installation and that use demonstration (DEMO) databases.

Search the help (press F1 when in Thermo-Calc) for brief descriptions of these examples, or browse to the examples located in your installation. From the main menu, **Help** → **Examples Files**:

- Graphical Mode **T\_12\_Viscosity\_in\_Cr-Ni.tcu**
- Console Mode `tcex55`

## Viscosity of the Ionic Liquids Model

The model for TCS Metal Oxide Solutions Database (TCOX) viscosity of slag:

$$RT \ln \eta = RT \ln \eta_0 + E$$

where  $\eta$  is viscosity,  $\eta_0 = \frac{hN_A}{V_m}$  and  $E$  is activation energy.

The excess parameters are expanded via R-K. An example of viscosity of A-B liquid oxide:

$$RT \ln \eta_{A-B} = y_A (RT \ln \eta_0^A + E_A) + y_B (RT \ln \eta_0^B + E_B) + \sum_i y_A \cdot y_B (y_A - y_B)^i \cdot VISC^i$$



The parameter of *VISC* stands for  $R \cdot T \cdot \ln(\text{viscosity})$ . In Thermo-Calc software, *VISC* is implemented as *VISC(ionic)*. In order to quickly acquire the actual viscosity, the parameter of *DVIS(ionic)* should be used. By coupling to the database, it makes use of the site fractions of each species, which reflects the structural change in the oxide melt.

Units for the viscosity of oxide slag:

- Pa·s (pascal-second) is the SI unit, mostly used for oxide slag
- CGS unit is poise (P), 1 poise=0.1 Pa·s
- Centipoise is also common (cP) because it is equivalent to mPa·s.

## Molar Volume Model

Molar volume can be used to establish a connection with some significant physical properties, for example, viscosity, electrical conductivity and surface tension. It is the reciprocal of density multiplied by molar mass.

### Model Description

The model used to describe the molar volume at ambient pressures is:

$$V_m(T) = V_0 \exp\left(\int_{T_0}^T 3\alpha dT\right)$$

A simple polynomial is used to model non-magnetic volumetric expansivity above 298 K:

$$3\alpha = a + bT + cT^2 + dT^3 + eT^{-2}$$

The model described above is implemented in Thermo-Calc software with two parameters,  $V_0$  and  $V_A$ , and  $\alpha$  is the linear thermal expansivity at 1 bar and  $3\alpha$  is the volumetric thermal expansivity.

### Molar Volume Descriptions

Parameter	Unit	Description
$V_0$ (phase, constituent array)	m <sup>3</sup> /mol	Volume at 1 bar and reference temperature $T_0$
$V_A$ (phase, constituent array)	None	$\int_{T_0}^T 3\alpha dT$

---

## Database Files

---

**In this section:**

---

About the Database Files .....	75
Database Installation Folder .....	77
About Cached Database Files .....	78
Editing the Database Initialization File or Unencrypted Database File .....	79
Database Checker .....	80

---

## About the Database Files

---

 [Database Manager User Guide](#)

### ***Encrypted vs Unencrypted Database Files***

Any licensed databases purchased from Thermo-Calc Software are encrypted in files with the extension `.TDC`; these files cannot be edited. These are referred to as *encrypted databases*.

*Unencrypted databases* have the file extension `.TDB` and can be edited. For example, PURE5 or PG35 (the free databases), or a user-defined database file.

 [Editing the Database Initialization File or Unencrypted Database File](#)

 If you develop your own databases on a regular basis, also see [About the TDB Editor](#).

### ***Database Initialization File***

The *database initialization file* is called `tc_initd.tdb`. Individual licenses are still required to use each database.

The file is used differently by the program depending on whether you are working with databases in Console Mode or Graphical Mode. For this reason, editing the file can be problematic if you do not understand the differences.

 [About Cached Database Files](#) and [Editing the Database Initialization File or Unencrypted Database File](#)

### ***User Database Files***

A user-defined database is considered to be an unencrypted database because you can make changes to it.

In Graphical Mode you can add user databases via the **Options** window. Once you have added the user database it is available from the database list in the System Definer Configuration window.

 [Global Settings: System Definer](#) to add a user database.

In Console Mode, users can decide to add their own user-defined database to the database initiation file, which then makes it available in the Console window.

 [Editing the Database Initialization File or Unencrypted Database File](#)

### *Custom Databases: GES5 and GES6 Calculation Engine Version*

As of Thermo-Calc version 2019b, the default calculation engine is set to GES6. For custom database users, you may sometimes see messages in the log indicating that additional preparation is being done by Thermo-Calc prior to a calculation. The preparation is carried out automatically; it is usually fast and only needed the first time a custom database is used after a modification. While databases with warnings are tolerated, databases that contain errors can only be loaded by GES6 after the errors are corrected.

The Database Checker tool uses the same parser as GES6 and can be used to verify the consistency and correctness of databases during development, before the databases are actually used.

 [Database Checker](#)



To globally set the default from GES6 back to GES5, go to the **Options** window [Global General Settings](#) tab and click **Version 5** for the *Preferred Gibbs Energy System* setting. The [SET\\_GES\\_VERSION](#) Console Mode command is used to change GES versions for a single Thermo-Calc session.



Contact Thermo-Calc Software AB at [info@thermocalc.com](mailto:info@thermocalc.com) for more information about the changes or if you encounter any issues you think are related to the new version of the calculation engine.

## Database Installation Folder

---

The standard Thermo-Calc databases and the database initialization file `tc_initd.tdb` are located in a subfolder to the Thermo-Calc installation. For example, in a Windows Standalone installation, you can find these in a folder called **data** where the file path is `C:\Program Files\Thermo-Calc\<version>\data`.

> This PC > OS (C:) > Program Files > Thermo-Calc > <Version> data

<input type="checkbox"/> Name	Type
<input checked="" type="checkbox"/> tc_initd.tdb	Thermo-Calc files
<input type="checkbox"/> STANDARD_DB_CLASSES.BIN	BIN File
<input type="checkbox"/> databases.ser	SER File
<input type="checkbox"/> TCZR1	File folder
<input type="checkbox"/> TCTI4	File folder

## About Cached Database Files

---



This is applicable to Graphical Mode.



Also see Global Settings [Global General Settings](#) where you can set the default database directory path. There is also an option to clear the database cache, which may be helpful if you are having issues.

The first time you open a database in Graphical Mode it can take a little while for it to load, especially if it is one of the large databases such as TCFE or TCNI. However, the next time the database opens quickly because the databases are now cached, or stored, in Graphical Mode.



Think of a cache as an intermediate storage area. Once a database cache is established the file is made easily accessible, even if you close the software program. It is the same concept as when you load a web page for the first time. The next time the page opens faster. This is also how it works for the databases in Thermo-Calc Graphical Mode.

Thermo-Calc caches the database files, both encrypted and unencrypted types, at two points—when you first open a database, sometimes called an *in-memory cache*, and then as a *disk-based cache*. The disk-based cache is what speeds up database loading and it is at this point that Thermo-Calc refers to the database initialization file for information. The disk-based cache uses a lookup key to find the abbreviated database name (e.g. TCFE or TCNI) contained in this initialization file.



It is important to understand the way databases are cached in Graphical Mode to ensure best practices for those who also use Console Mode, especially if the database initialization file is edited. If you plan to edit this file, see [Editing the Database Initialization File or Unencrypted Database File](#).

---

## Editing the Database Initialization File or Unencrypted Database File

---

Some advanced users may decide to edit the database initialization file, for example to reorder the list of encrypted databases displaying in the Console window or to add a user-defined database for use in Console Mode. It is important to understand that changes to this file impact the database caching process used in Graphical Mode.

 [About the Database Files](#)

 [About Cached Database Files](#)

The following recommendations are also applicable if you edit an unencrypted database file, which includes user databases and free databases such as PURE5 or PG35.

### **Best Practice Recommendations**

 [Database Installation Folder](#)

If you edit the database initialization file (`tc.initd.tdb` in Windows or `initd.tdb` in Mac and Linux), then:

- If a user database is added to the file, it is subject to the same caching as the encrypted databases. This means that further changes (to the user database) are not detected when you are working with databases in the System Definer in Graphical Mode.

If you have a user database, which is subject to regular updates, then:

- If you make changes to a user database, you then need to clear the cache.
- It is recommended that you create a separate directory to store unencrypted database files.

If you edit an unencrypted database file included with the installation (such as the PURE5 and PG35 files):

- Do not edit the file in the installation folder. Instead make a local copy and load it into the application as a user database.

## Database Checker

---

The Thermo-Calc software package includes a program to check that the syntax of unencrypted database files is correct. The program applies the syntax rules set out in the *Database Manager User Guide* and reports errors and issues warnings. This program is intended for advanced users who develop and manage databases.



Thermo-Calc accepts some deviations from these syntax rules. Although warnings can be ignored, errors must be corrected. Reported errors must be corrected in the TDB file before the database can be loaded by the application. Sometimes the warnings can also reveal the need for additional corrections as shown in the example below.

For example, a warning is generated if a phase name is detected that is also an abbreviation of another phase name in the database. Although allowed, it is not recommended as this can (in some cases) lead to unexpected results because abbreviations are also used when entering commands in the Console and included in macro files. The following shows why in this case an abbreviation is not recommended.

Imagine that a macro file uses the abbreviation `ABC` to refer to a phase in a database that is named `ABCDE`. As long as there is no other phase in the database with the same exact name or initial abbreviation, it works. But if a new phase is added to the database and is actually named `ABC` then the macro can in error refer to the wrong phase. The macro still runs but now refers to the phase `ABC` instead of `ABCDE` because `ABC` is an exact match, *which takes precedence over abbreviations*. The Database Checker displays a warning because this situation can only occur when one phase name is an abbreviation of another.

With respect to using this tool, the executable `DatabaseChecker` file is found in the Thermo-Calc home directory.

The program can also be launched by selecting **Tools** →  **Database Checker** from the main menu in Thermo-Calc.



[Database Manager User Guide](#)

# Global Settings

---

In the **Options** window you can globally set general settings for Graphical Mode as well as default settings for any new activities for both Graphical Mode and Console Mode.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

## In this section:

---

Global General Settings .....	82
Graphical Mode: Default Units .....	87
Graphical Mode: Activities .....	89
Global Settings: System Definer .....	90
Global Settings: Equilibrium Calculator .....	92
Global Settings: Diffusion Calculator .....	93
Global Settings: Precipitation Calculator .....	94
Global Settings: Process Metallurgy .....	95
Global Settings: Scheil Calculator .....	96
Global Settings: Tables .....	97
Global Settings: Graphical and Console Mode Plotting .....	98
Global Plot Settings: Themes .....	101
Predefined Theme Settings .....	108
Global Plot Settings: Plot Area Size .....	111
Global Plot Settings: Design the Look and Feel .....	113
Global Settings: Console Mode Default Appearance .....	119

---

## Global General Settings

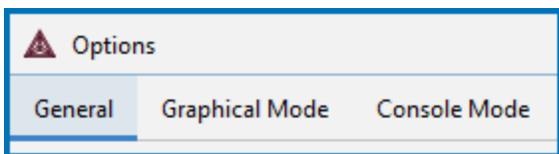
---

Make global changes to the various defaults used for the settings for both Graphical Mode and Console Mode.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

then click the **General** tab.



### Tooltips Enabled

Select whether to turn on tooltips information. By default the **Tooltips enabled** checkbox is selected. Hover the cursor above some buttons or other settings to display text with more information about the settings.

### Localization

To change the GUI language from the **Localization** list choose **English** (the default), **Swedish**, **Spanish**, **German**, **Russian**, **Chinese**, **Japanese**, or **French**.

### Look and Feel

To change the **Look and feel** of the GUI layout, choose **Light** (the default) or **Dark**.

For macOS, there are also the options **macOS Light** and **macOS Dark**.



The primary difference for the macOS themes is that the buttons and panels are a closer match to the Mac operating system. In addition, macOS Dark is darker than the standard Dark (more black than gray), although the macOS Light version is only slightly darker. It is recommended that users experiment with the options to determine which is best for your needs.



Also see [Global Plot Settings: Design the Look and Feel](#) for details about the **Color switching options** available and related to the **Light** and **Dark** modes. The use of these modes is also related to being able to right-click and access the **Plot Properties** options for local plots. See [Changing the Plot Properties](#).

## Log Level

Select the level and type of information to display in the **Event Log** window using the **Log level** slide bar. Choose from a **Debug**, **Info** (the default), **Warning**, or **Error** level of detail.

## Database Directory

In the **Database directory** field, the default path is to the **data** directory where all the database files and data are located. Click **Modify** to open the default folder to change this default but first review the important information below.

### Modifying the Database Directory Path (Graphical Mode Only)



Important Note: Database Directory Path (Graphical Mode only)

When you open a database for the first time, it is stored in a cache (an intermediate storage area), that is maintained even if you close the program. This improves the speed of loading the next time you open that database. The cache is used for all databases that are listed in the database initialization file, typically all the standard databases that are included with Thermo-Calc.

If you modify your own user-defined database, or if you choose to edit an unencrypted database, it is recommended that you create a separate directory to store these database files. This way the most recent version of the user-defined or edited databases are loaded into the program when the System Definer is performed.

---

## [About the Database Files](#)

### Model Directory

In the **Model directory** field, this is the default file location where the predefined and custom Property Models (used with the Property Model Calculator or TC-Python) are saved and stored. Click **Modify** to open the default folder or to create a custom folder for your own models.



All Property Models (whether these are predefined or custom models) must be located in a subdirectory of the Property Model directory. For that reason it is recommended that you do not change the directory location.



See [Property Model Calculator](#) for information about using the predefined models. Also see the [TC-Python help](#) on the website for details about creating custom models.

### Temporary Directory

In the **Temporary directory** field this is the default file path to the temporary folder, where files are stored during calculation, such as Console Mode state files (e.g. `State_4064824901731140183.POLY3`), log files, and so forth.

Click **Modify** to open the default folder, which is set during installation and based on your operating system.

### Property Model Python Interpreter

This is the path to the executable Python file that is needed to work with custom property models.



See the **Property Model Framework → Best Practices** section in the [TC-Python help](#) on the website.

### Check Update Interval

From the **Check update interval** list, choose **Do not check** (the default) or **On startup**.



You can also manually check for updates – choose **Help** → **Check for Updates**.

## Preferred Gibbs Energy System

Use this setting to globally set the default GES version to be either GES6 (**Version 6** the default) or GES5 (**Version 5**).



[About the Gibbs Energy System \(GES\) Module](#)



This setting is useful if you are using the POURBAIX module (aqueous solutions) or if you have a custom database that is not compatible with GES6. The [SET GES\\_VERSION](#) Console Mode command can also be used to change GES versions for a single Thermo-Calc session.



Contact Thermo-Calc Software AB at [info@thermocalc.com](mailto:info@thermocalc.com) for more information about the changes or if you encounter any issues you think are related to the new version of the calculation engine.

## Reset all Settings and the Database Cache to Defaults

Click **Factory reset** to reset all the settings to the original defaults, including clearing the database cache. The changes are applied when Thermo-Calc is restarted.



This may be useful when there are several versions of Thermo-Calc installed and there are conflicts between settings or loading databases becomes problematic. See [About Cached Database Files](#).

## Reset to Original Settings



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

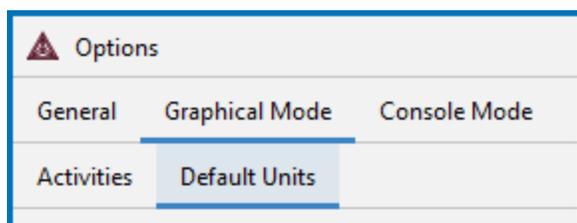
## Graphical Mode: Default Units

Make global changes to the default units used for the settings.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

then click the **Graphical Mode** → **Default Units** tabs.



### Reset to Original Settings



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

Setting	Options
Temperature	<b>Kelvin</b> (the default), <b>Celsius</b> , or <b>Fahrenheit</b>
Pressure	<b>Pascal</b> (the default), <b>Atmospheres</b> , or <b>Bar</b>
Amount	<b>Mole</b> (the default), <b>Gram</b> , <b>Kilogram</b> , <b>Tonne</b> , or <b>Pound</b>
Mass	<b>Tonne</b> (the default), <b>Kilogram</b> , <b>Gram</b> , or <b>Pound</b>
Composition	<b>Mass percent</b> (the default), <b>Mole percent</b> , <b>Mass fraction</b> , or <b>Mole fraction</b>
Mass composition	<b>Mass percent</b> (the default) or <b>Mass fraction</b>

<b>Setting</b>	<b>Options</b>
Energy	<b>Joule</b> (the default), <b>Calorie</b> , <b>Electron volt</b> , <b>Gigajoule</b> , <b>Watt hour</b> , or <b>Kilowatt hour</b>
Power	<b>Watt</b> (the default), <b>Kilowatt</b> , or <b>Megawatt</b>
Volume	<b>Cubic meter</b> (the default), <b>Cubic decimeter</b> , <b>Cubic centimeter</b> , or <b>Normal cubic meter</b>
Density	<b>Kilogram per cubic meter</b> , <b>Gram per cubic meter</b> , <b>Gram per cubic centimeter</b> (the default), or <b>Pound per cubic inch</b> .
Heat capacity	<b>Joule per mole Kelvin</b> (the default), <b>Calorie per mole Kelvin</b> , <b>Joule per kilogram Kelvin</b> , or <b>BTU per pound Fahrenheit</b>
Entropy	<b>Joule per Kelvin</b> (the default), <b>Calorie per Kelvin</b> , or <b>Electron volt per Kelvin</b>
Dynamic viscosity	<b>Pascal-second</b> (the default), <b>milliPascal-second</b> , <b>Poise</b> , or <b>centiPoise</b>
Kinematic viscosity	<b>Square meter per second</b> (the default), <b>Stokes</b> , or <b>Centistokes</b>
Length	<b>Meter</b> (the default), <b>Millimeter</b> , <b>Micrometer</b> , <b>Nanometer</b> , or <b>Ångström</b>
Time	<b>Seconds</b> (the default), <b>Minutes</b> , <b>Hours</b> , or <b>Days</b>
Velocity	<b>m/s</b> (the default), <b>mm/s</b> , <b>μm/s</b> , or <b>nm/s</b>
Surface tension	<b>Joule per m<sup>2</sup></b> (the default) or <b>Millijoule per m<sup>2</sup></b>
Surface energy	<b>Joule per m<sup>2</sup></b> (the default) or <b>Millijoule per m<sup>2</sup></b>

## Graphical Mode: Activities

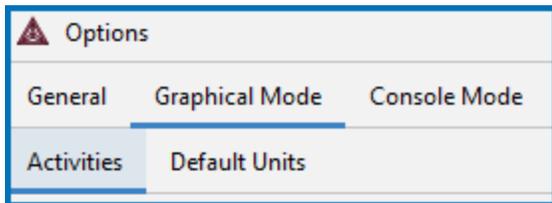
---

Make global changes to the various defaults used for the settings.

To open the **Options** window:

- Windows: Select **Tools** → ☰ **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **Graphical Mode** → **Activities** tabs.



Under **Default Configurations**, there are **System definition**, **Calculation**, **Precipitation**, **Diffusion**, **Scheil**, **Process Metallurgy**, **Plotting**, and **Tabulation** setting nodes. These are described individually.

- [Global Settings: System Definer](#)
- [Global Settings: Equilibrium Calculator](#)
- [Global Settings: Precipitation Calculator](#)
- [Global Settings: Diffusion Calculator](#)
- [Global Settings: Scheil Calculator](#)
- [Global Settings: Process Metallurgy](#)
- [Global Settings: Graphical and Console Mode Plotting](#)
- [Global Settings: Tables](#)

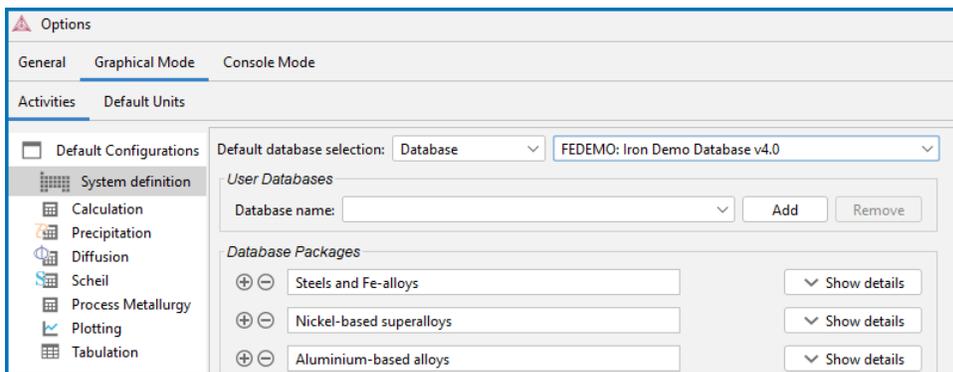
## Global Settings: System Definer

When you create a new System Definer, the default databases are taken from these defaults. The database selections for each System Definer can then be configured individually. Once created, the specific (local) settings for a System Definer are not affected if you later make changes to the global defaults.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

then click the **Activities** tab and the **System definition** node in the tree.



[System Definer](#)



[Creating a Database Package and Clearing a Database Package Message](#)



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

### Default Database Selection

The **Default database selection** can be either for a **Database** or a **Database Package**.

- If you choose **Database**, then select the default installed database to use globally for all System Definer nodes. For example, choose the latest version of the TCS Steel and Fe-alloys Database (TCFE) from the list if you have a license.
- If you choose **Database Package**, select an option from the list. These Database Packages are defined in the *Database Packages* section described below.

## User Databases

Under *User Databases*, define the **Database name** after adding a custom database.

Click **Add** to navigate to a custom database on your computer. After you have added the user database it is available from the list to choose as the default.

Once a user database is added to the **Database name** list, you can also click the **Remove** button to delete it from your project.

## Database Packages

Based on the installed databases, there are also default *Database Packages* available. Use this section to organize your own sets of databases that you use together often.

Create and remove *Database Packages* by clicking the **Add a database package**  button or **Remove this database package**  button.



The addition or removal of a database package does not take effect until a new System Definer node is defined.

Click **Show Details** to display the list of databases included with the package. Click **Hide Details** to only display the name of the package.



[Creating a Database Package](#)

## Global Settings: Equilibrium Calculator

---

When you create a new Equilibrium Calculator, its initial settings are taken from these defaults. The settings of each Equilibrium Calculator can then be configured individually. Once created, the specific (local) settings for an Equilibrium Calculator are not affected if you later make changes to the global defaults.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **Activities** tab and the **Calculation** node in the tree.



The available default settings are described in the topic [Options Tab Settings](#).



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

## Global Settings: Diffusion Calculator

---

When you create a new Diffusion Calculator, its initial settings are taken from these defaults. The settings for each Diffusion Calculator can then be configured individually. Once created, the specific (local) settings are not affected if you later make changes to the global defaults.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

then click the **Activities** tab and the **Diffusion** node in the tree.



The available default settings are described in the topic [Options Tab Settings](#).



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

## Global Settings: Precipitation Calculator

---

When you create a new Precipitation Calculator, some of its initial settings are taken from these defaults. The settings for each Precipitation Calculator can then be configured individually. Once created, the specific (local) settings for a Precipitation Calculator are not affected if you later make changes to the global defaults.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **Activities** tab and the **Precipitation** node in the tree.



The available default settings are described in the topics [Matrix Phase Settings](#) and [Precipitate Phase Settings](#)



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

## Global Settings: Process Metallurgy

---

When you create a new Process Metallurgy Calculator, its initial settings are taken from these defaults. The settings for each Process Metallurgy Calculator can then be configured individually. Once created, the specific (local) settings are not affected if you later make changes to the global defaults.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **Graphical Mode** and **Activities** tabs, followed by the **Process Metallurgy** node in the tree.



The available default settings are described in the topic [Options Tab](#).



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

## Global Settings: Scheil Calculator

---

When you create a new Scheil Calculator, its initial settings are taken from these defaults. The settings of each Scheil Calculator can then be configured individually. Once created, the specific (local) settings for a Scheil Calculator are not affected if you later make changes to the global defaults.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **Activities** tab and the **Scheil** node in the tree.



The available default settings are described in the topic [Advanced Options](#).



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

## Global Settings: Tables

---

When you create a new Table Renderer, its initial settings are taken from these default settings. The settings of each Table Renderer can then be configured individually. Once created, the settings of a Table Renderer are not affected by changes in the default settings.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

then click the **Activities** tab and the **Tabulation** node in the tree.



### [Configuration Settings](#)



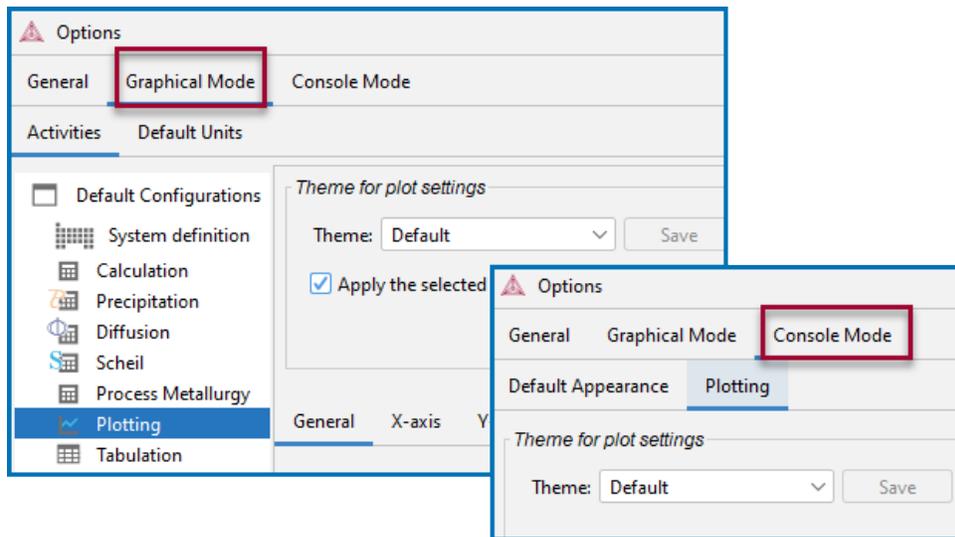
**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

## Global Settings: Graphical and Console Mode Plotting

Globally configure several default plot settings or work with a **Theme**.

To open the **Options** window:

- Windows: Select **Tools** → **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.



On the **Options** window you can access the same settings either from the **Graphical Mode** or **Console Mode** tabs.

- Click the **Graphical Mode** → **Activities** tabs and then click the **Plotting** node in the tree, or
- Click the **Console Mode** → **Plotting** tabs.



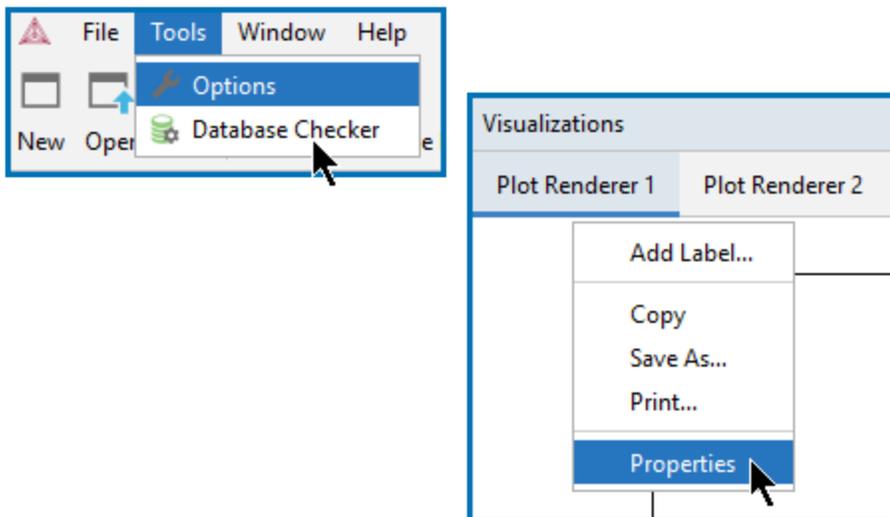
The Console Mode and Graphical Mode plot settings are independent of each other once you start making changes to any predefined or custom Themes, or when applying changes to individual plots in either Mode.

There are many options available and these are generally divided into these topics:

- [Global Plot Settings: Design the Look and Feel](#) to set a variety of options for titles, legends, labels, fonts, colors, lines, grids, data points, anchors, data points, numbers, and to adjust image quality.
- [Global Plot Settings: Themes](#) to create, edit, import, export, rename, or delete plot themes.
- [Global Plot Settings: Plot Area Size](#) to set the Plot Area Size.

## Global vs Local Plot Settings

For both Graphical Mode and Console Mode, you can work with plots and themes from both the global or local settings windows. Global settings are accessed on the **Options** window and local plot settings from the **Visualizations** window when you right-click an individual plot and choose **Properties**.



## Local Plot Property Settings

### *Plot Renderer (Graphical Mode)*



When you create a new Plot Renderer in Graphical Mode, its initial settings are taken from the default settings and theme. The settings can then be configured individually. Once created, the settings of a Plot Renderer are not affected by changes to the default global settings unless you are working with themes and then save or create a custom theme. See [Global Plot Settings: Themes](#).

## **Visualizations Window Individual Plots (Graphical or Console Mode)**

To make local changes to the appearance of a specific plot for either Graphical Mode or Console Mode, in the **Visualizations** window right-click a plot and select **Properties**. In the **Plot Properties** window a variety of settings can be made. These are mostly the same settings as in the **Options** window.

You can also edit some properties for individual plot lines (the color, the line width and type, and whether data points are included). In the **Visualizations** window, hover the mouse over a plot line. The crosshair cursor turns into a cursor resembling a pointing hand when it is over a line that can be clicked. Alternatively, hold down the <Ctrl> key while you move the cursor around the plot to only display it as a crosshair and prevent unintended edits.

When changes are made that differ from the plot theme applied (i.e. a predefined or custom theme) then blue text next to the theme section alerts you to the change. See [Global Plot Settings: Themes](#) for details.

### **Reset to Original Settings**



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

## Global Plot Settings: Themes

---

Make global changes to the various defaults used for the settings.

To open the **Options** window:

- Windows: Select **Tools** → ☰ **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

On the **Options** window you can access the same settings either from the **Graphical Mode** or **Console Mode** tabs.

- Click the **Graphical Mode** → **Activities** tabs and then click the **Plotting** node in the tree, or
- Click the **Console Mode** → **Plotting** tabs.



The Console Mode and Graphical Mode plot settings are independent of each other once you start making changes to any predefined or custom Themes, or when applying changes to individual plots in either Mode.

## About Plot Themes

### *Location of Local and Global Theme Settings*

Plot **Theme** settings are accessed in the *Theme for plot settings* section:

- Locally on the **Plot Properties** window (right-click any plot in the **Visualizations** window).
- Globally on the **Options** window for both Graphical Mode and Console Mode.

### *Predefined Plot Themes*

There are these predefined plot themes available: **Default**, **Large**, **Small**, or **Publication**. For these themes, you can use them as a template to create your own custom theme or export the json file for use elsewhere.

The predefined plot themes have many of the same settings where key differences include font sizes, plot area size and type, scaling, data point marker radius, tick mark length, and line width. For all predefined themes, the default font is SansSerif with the style Plain (normal).

 See [Predefined Theme Settings](#) for details and differences.

## Custom Themes

You can create, edit, delete, rename, import, and export custom themes.

### ***Plotting Input is Changed: Ignore, Create, Update, or Discard the Changes***



When any change is made to a predefined or custom theme, either locally or globally, a blue text message under the *Theme for plot settings* section reminds you that *at least one plotting input has changed*.

When you see this blue text you can:

- Ignore it and apply the local changes to a specific plot. In this case when you click **Perform** or **Perform Tree** to update the plot in the **Visualizations** window the change only applies to this one plot. The blue text continues to be visible.
- Create a new custom theme (click **Save as**). If this is done locally on the **Plot Properties** window, the new theme is added to the global list but only applied locally to the plot. The blue text is gone.
- Update an existing custom theme (click **Save**). If this is done on the **Plot Properties** window, the changes are applied globally and applied locally to the plot. The blue text is gone.
- Discard the changes and click **Reset to original settings** (for global **Options** windows you may need to scroll, it is in the bottom right of the window) or **Reset** (for local settings on the **Plot Properties** window). The blue text is gone.

## **Working with Plot Themes**

Whether working with a predefined or custom theme, there are common tasks you can do such as using a theme as a template to create a new custom theme, or you can import and export .json files.

Once created, there is more flexibility with the custom themes.

### ***Applying Predefined or Custom Themes to a Plot Globally or Locally***

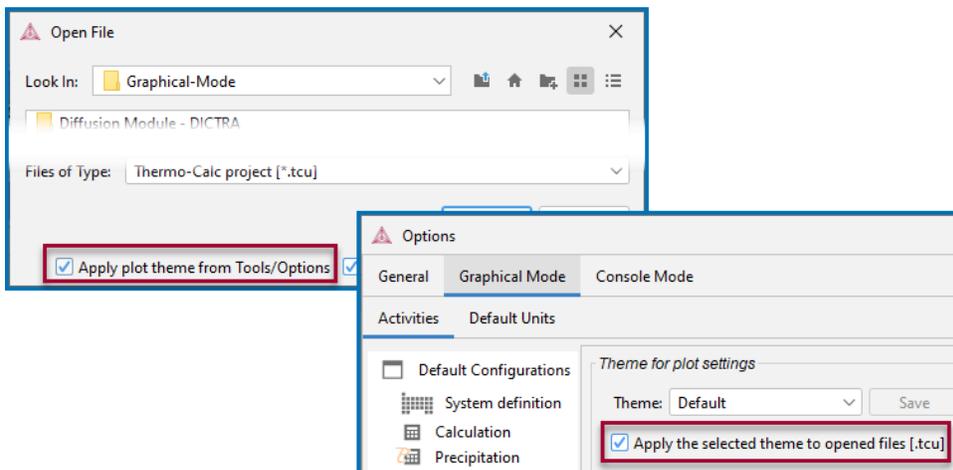
1. In the *Theme for plot settings* section, choose an option from the **Theme** list.
  2. Click **OK** to apply the theme. Click close (X) or **Cancel** to exit without saving, or **Reset** to revert any changes to the original saved settings.
- On the **Options** window this is applied globally.
  - On the **Plot Properties** window this is applied locally to the specific plot in the **Visualizations** window.

### ***Applying Plot Themes to Project Files (Graphical Mode Only)***

When opening project files, a predefined or custom plot theme can be automatically applied to all plots. There are two checkboxes that control this, one is a global setting and one is a local setting applied to a specific file as it is opened.



If you double-click or drag and drop to open a project file it automatically uses the global settings in Thermo-Calc (which may or may not be to apply a theme). This means you may inadvertently apply a theme to your plots without realizing it.



There is some interaction between these checkboxes as detailed in the table below.

Checkbox	Location	Actions	Comments
Apply the selected theme to opened file [.tcu]	<b>Options</b> window.	Select the checkbox to apply a theme globally to all project files being opened.  Click to clear the checkbox to allow each file to keep existing plot settings.	When this checkbox is selected or deselected, it also changes the <b>Apply plot theme from Tools/Options</b> checkbox to match, i.e. these are connected in one direction.
Apply plot theme from Tools/Options	<b>Open File</b> window.  See <a href="#">Opening Project Files</a>	Select the checkbox to apply a theme locally to the specific project file being opened.  Click to clear the checkbox to allow the specific file to keep existing plot settings.	If this checkbox is selected or deselected it only applies to the specific project file being opened, i.e. the <b>Apply the selected theme to opened file [.tcu]</b> checkbox does not change.



If you have a project file with calculated results and local settings applied to plots i.e. applied from the **Visualizations** window to a specific plot via the **Plot Properties** settings window, then click to clear the local setting as you open the file using **Apply plot theme from Tools/Options** checkbox. Otherwise the default global settings are applied to all plots and you would need to individually change these settings back to your original design. Alternatively, experiment with applying a predefined theme, e.g. **Publication**, as this could be sufficient for your final output.

### ***Creating a Custom Plot Theme in Thermo-Calc***

To create a custom theme, you start by making changes to the selected theme type and then save it with a new name using the **Save as** option.



You can work with themes from both the global or local settings windows. Global settings are accessed on the **Options** window (Windows: **Tools** → **Options**; Mac: **Thermo-Calc <version>** → **Preferences**) and local plot settings accessed from the **Visualizations** window when you right-click an individual plot and choose **Properties**.

For example, the **Background color** is changed to RGB **#FF4EE2** for the **Default** theme. Next to the *Theme for plot settings* section, a message in blue alerts you that *at least one plotting input has changed*.

To save the new settings including the change to the background color (you cannot replace the predefined themes), you also must make sure to click **OK** at the end.

## HOW TO CREATE A CUSTOM PLOT THEME

1. In the *Theme for plot settings* section, click **Save as**. The **New theme plot setting** window opens.
2. In the **Theme name** field enter the new custom theme name. The new theme is now available from the list.
3. On the **Plot Properties** or **Options** window, click **OK** to finish. Click close (X) or **Cancel** to exit without saving, or **Reset** to revert any changes to the original saved settings.

## Importing a Custom Theme

Use the **Import** option to add your own theme with the file type of \*.json. Once the file is imported, you can make further changes and save the new theme settings as needed. If you import a theme that uses the same name one already in the list (e.g. **Publication**), the imported theme name is changed automatically by adding a number at the end of its name (e.g. **Publication\_1**). You can then **Rename** the theme to give it a unique name.

## HOW TO IMPORT A CUSTOM PLOT THEME

1. In the *Theme for plot settings* section, click **Import**.
2. Navigate to where you saved the .json file, select it, then click **Open**.
3. During the import, and if there are incorrect input values, you are prompted with an error message describing the issue with the file.

This could be for various reasons, such as entering a value that is not available (i.e. it is out of bounds), or the file contains information about an unavailable setting. In particular these settings must be correctly entered in the import file: font sizes, tie line, number of decimals, border width, data points marker radius, x and y axis tick marks length, and jpeg compression level.

4. If it is a successful import, click **Rename** to give the theme a unique name.
5. On the **Plot Properties** or **Options** window, click **OK** to finish. Click close (X) or **Cancel** to exit without saving, or **Reset** to revert any changes to the original saved settings.

## ***Exporting a Theme***

Use the **Export** option to export an existing theme or after creating a custom theme.



You can also export an existing theme, edit it in a text editor (making sure to save the file with file extension .json), and then **Import** this new .json file. However, it is better to use the Graphical User Interface to prepare your custom theme in order to minimize errors during import.

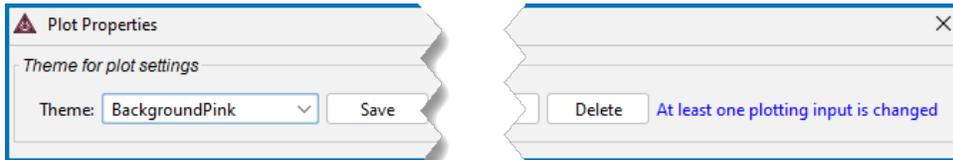
## **HOW TO EXPORT A PLOT THEME**

1. In the *Theme for plot settings* section, select the **Theme** to export.
2. Click **Export**.
3. Navigate to where you want to save the .json file. Enter a **File name** as required then click **Save**.

## ***Deleting a Custom Theme***

After you have created a custom theme, it can be deleted from the list. Predefined themes cannot be deleted.

If the selected theme is deleted from the global Options section, and you open a project or plot that uses the now deleted theme, the default theme is selected but it includes the message in blue next to the section to remind you that *at least one plotting input has changed*. In the example the custom theme, **BackgroundPink**, was deleted at the global level but at the local level for the specific plot open in the **Visualizations** window the settings are retained and the message displays.



## HOW TO DELETE A CUSTOM PLOT THEME

1. In the *Theme for plot settings* section, select the **Theme** to delete.
2. Click **Delete**.
3. Click **OK** on the window that opens.
4. On the **Plot Properties** or **Options** window, click **OK** to finish. Click close (X) or **Cancel** to exit without saving, or **Reset** to revert any changes to the original saved settings.

## Renaming a Custom Theme

After you have created or imported a custom theme, it can be renamed. Predefined themes cannot be renamed.

## HOW TO RENAME A CUSTOM PLOT THEME

1. In the *Theme for plot settings* section, select the custom **Theme** to rename.
2. Click **Rename**.
3. Enter the new name in the field. Click **OK**.
4. On the **Plot Properties** or **Options** window, click **OK** to finish. Click close (X) or **Cancel** to exit without saving, or **Reset** to revert any changes to the original saved settings.

## Predefined Theme Settings

The following lists the applied attributes for each of the predefined themes: **Default**, **Large**, **Small**, or **Publication**.

 [Global Plot Settings: Themes](#)

### Image Quality

Plot Setting	Default	Large	Small	Publication
Default file format and DPI	.png and Use default resolution (300 DPI)	.png and Use default resolution (400 DPI)	.png and Use default resolution (300 DPI)	.png and 600 DPI

### Fonts, Style, Size, and Format

Plot Setting	Default	Large	Small	Publication
Title font (SansSerif) and Style (Plain/normal). The size is different for each.	18	26	16	24
Legend font (SansSerif) and Style (Plain/normal). The size is different for each	11	22	9	20
Label font (SansSerif) and Style (Plain/normal). The size is different for each.	16	24	14	22
Header font (SansSerif) and Style (Plain/normal). The size is different for each.	12	18	10	16
X and Y Axis Label font (SansSerif) and Style (Plain/normal). The size is different for each.	18	28	16	26
X and Y Axis Tick Label font (SansSerif) and Style (Plain/normal). The size is different for each.	14	24	12	22
Title format, Label format, Axis label format (X- and Y-axis)	Plain text			

## Colors

<i>Plot Setting</i>	<i>Default</i>	<i>Large</i>	<i>Small</i>	<i>Publication</i>
Title color, Legend color, Label color, Header color, Border color, Axis label color (X- and Y-axis), Axis tick label color (X- and Y-axis)	rgb ff000000			
Legend background color, Background color, and Canvas color	rgb ffffffff			
Color option	Vivid (in json file it is PlotColorOption 6)			

## Checkbox Selections

Selected = Yes

<i>Plot Setting</i>	<i>Default</i>	<i>Large</i>	<i>Small</i>	<i>Publication</i>
Show grid	No			
Show legend	Yes			
Show anchor	Yes			
Retain labels	Yes			
Show header	No			
Show tick labels (X- and Y-axis)	Yes			
Tick marks outside (X- and Y-axis)	Yes			

## Plot Area Size

<i>Plot Setting</i>	<i>Default</i>	<i>Large</i>	<i>Small</i>	<i>Publication</i>
Fit the plot area size to the enclosing window (selected), with the plot area height and width preset as indicated	400 x 400	700 x 700	400 x 400	Not selected. By default it is 600 x 600

<i>Plot Setting</i>	<i>Default</i>	<i>Large</i>	<i>Small</i>	<i>Publication</i>
Fit the plot area size to a square with a width/height of the enclosing window	Not selected (in json file it is plotAreaSizeType: Window_Fit)			Selected (in json file it is Square_Fit)
Define the plot area size	Not selected			
Scale plot attributes using the system scaling factor	yes			

## ***Other Settings***

<i>Plot Setting</i>	<i>Default</i>	<i>Large</i>	<i>Small</i>	<i>Publication</i>
Show every Nth tieline (only relevant for phase diagrams)	1			
Border width	1			
Data points marker	None (in json file this is called Symbol and set to 0)			
Legend phase caption style	None (in json file it is set to 0)			
Data points marker size (if you choose a data points marker symbol, then this is the default size applied)	3	5	3	5
Tick mark length (X- and Y-axis)	6	8	4	6
Number of decimals	2			
Numeric format	Auto (in json file it is 2)			
Line width	0	2	0	1

## Global Plot Settings: Plot Area Size

---

Make global changes to the various defaults used for the settings.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press <⌘> on the keyboard.

On the **Options** window you can access the same settings either from the **Graphical Mode** or **Console Mode** tabs.

- Click the **Graphical Mode** → **Activities** tabs and then click the **Plotting** node in the tree, or
- Click the **Console Mode** → **Plotting** tabs.



The Console Mode and Graphical Mode plot settings are independent of each other once you start making changes to any predefined or custom Themes, or when applying changes to individual plots in either Mode.

For local or global application, the default settings applied in the *Plot Area Size* section are initially based on the selected predefined theme. Once a custom theme is available from the list, these default settings can also be related to how the custom theme is set up.



See [Predefined Theme Settings](#) for details and differences between the predefined themes initially available in the list.

In the *Plot Area Size* section, choose an option.

Click **Fit the plot area size to the enclosing window** and as the **Visualizations** window is resized, the plot moves and adjusts at the same time. This setting is the default for the **Default**, **Large**, and **Small** themes.

Click **Fit the plot area size to a square with a width/height of the enclosing window**. With this option, the minimum value of the width and height of the **Visualizations** window is used as the dimension of the fixed plot area and in a square.

Click **Define the plot area size** to enter a **Plot area height** and **Plot area width (not applicable to triangular plots)**. The units are pixels and the size is kept as the size on the screen (96 dpi). This is the default for the **Publication** theme.

For the predefined themes this is already set for each as indicated below. As needed, you can change these defaults for one plot or create a new theme.

- **Default** and **Small** are set to 400 x 400
- **Large** is set to 700 x 700
- **Publication** is set to 600 x 600

Click to deselect the **Scale plot attributes using the system scaling factor** checkbox to scale all plot attributes (i.e. labels, font size, axes, tick marks, etc.) independently of the screen size and resolution. This maintains consistent attribute sizing between different plots.

The minimum value of the width and height of the **Visualizations** window, or the fixed plot is used for resizing, in the following parts of the plot:

- X- and Y-axis label
- Legend font
- Tick label font
- Tick mark length
- Title
- Header
- Line width
- Data point width
- Experiment data size
- Logo

## Global Plot Settings: Design the Look and Feel

Make global changes to the various defaults used for the settings.

To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc <version>** → **Preferences** or **Settings** or press  $\langle \text{⌘} \rangle$  on the keyboard.

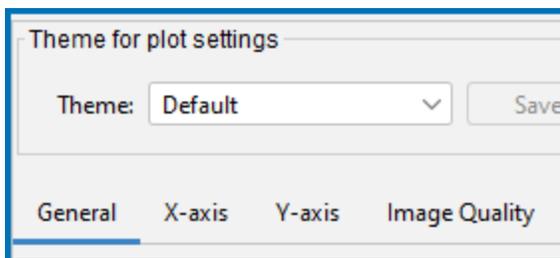
On the **Options** window you can access the same settings either from the **Graphical Mode** or **Console Mode** tabs.

- Click the **Graphical Mode** → **Activities** tabs and then click the **Plotting** node in the tree, or
- Click the **Console Mode** → **Plotting** tabs.



The Console Mode and Graphical Mode plot settings are independent of each other once you start making changes to any predefined or custom Themes, or when applying changes to individual plots in either Mode.

The following global design settings are available for both Graphical Mode and Console Mode from the **General** tab or as indicated for the **X-axis**, **Y-axis**, and **Image Quality** tabs.



The plot settings listed below are grouped into these general categories

- [Titles, Legends, Headers, and Labels](#)
- [Fonts and Colors](#)
- [Lines and Grids](#)
- [Data Points, Anchors, and Numbers](#)
- [Image Quality](#)

---

## ***Titles, Legends, Headers, and Labels***

### **Title Format, Label Format, and Axis Label Format**

On the **General** tab (**Title format** and **Label format**) and the **X-axis** and **Y-axis** tabs (**Axis label format**) select **Plain text** (the default) or **LaTeX formatted text**.



For **LaTeX formatted text** you can use the LaTeX math text to enter text. See [ADD\\_LABEL\\_TEXT](#) and [Plot Labels](#) for more details.

### **Show Legend**

Select the **Show legend** checkbox to include a legend on the plot. By default no legend is displayed.

### **Tick Mark Length, Location, and Show Tick Labels**

On the **X-axis** and **Y-axis** tabs, choose or enter a **Tick mark length** (6 is the default).

The defaults are to have the **Tick marks outside the chart** and to **Show tick labels**. Click to clear the checkboxes as required.

### **Legend Phase Caption Style**

For the **Legend phase caption style**, choose **None**, **All**, **Constitution description**, or **Ordering description**.



[About Legend Styles](#)

### **Retain Labels**

By default the **Retain labels** checkbox is selected. Plot labels are kept (retained) when plots are updated. It can be applied globally or locally to individual plots.

## Show Header

Click to select the **Show header** checkbox to display the basic details about the plot along the top. This includes the date and time the plot is generated, the database used, and the properties.

## Fonts and Colors

### Color Switching

This is related to the **Light** and **Dark** look and feel modes set under **Tools** → **Options** then on the **General** tab. See [Global General Settings](#).



This section is about Global changes to the default output for all plots for any future project files. Global settings take effect for *new* project files and/or *new* plots *added* to existing projects. That is, the default setting only affects how the plots start out initially. Once a plot is created the settings are locked to the mode and can only be changed locally.



You can locally change an existing plot in a project. See [Changing the Plot Properties](#) for details.

Select a **Color switching** option: **Switch automatically between Light mode and Dark mode** (the default), **Always use Light mode colors**, or **Always use Dark mode colors**.



Tie lines and invariant lines are the same color for both Light and Dark modes, i.e. these colors do not change.



Use the **Color switching** option to override the look and feel settings that may not be optimal. For example, when preparing a plot for publication, the **Dark** mode plot or line colors may not work well or do not follow a journal's guidelines. In other words, you can view and export the plot with **Light** mode color settings even though the interface is using Dark mode, and vice versa.

## Fonts: Title, Legend, Label, Header, Axis Label, and Tick Label

On the **General** tab (**Title**, **Legend**, **Label**, **Header**, and on the **X-axis** and **Y-axis** tabs (**Axis Label**, **Tick Label**), click **Modify** to edit the **Font Name**, **Style**, **Size**, and **Color**.

## Legend Background Color, Background Color, Canvas Color, Border Color

To change various background color attributes, click **Modify** to choose a color or enter a specific color.

## Lines and Grids

### Show Grid

Click to select the **Show grid** checkbox to include a grid on the plot. By default no grid is displayed.

### Show Every Nth Tieline



Tielines are only applicable to phase diagrams.

Choose an integer value ranging from 0 to 99.

### Color Option (Plot Lines)

For the for plot line colors, choose a **Color option** from the list: **Legacy**, **Printer friendly**, **JFree chart**, **Pastel**, **Medium dark**, **Bright dark**, **Vivid**, or **Earth**.

This option is also available for specific plot lines. To change the color of an individual plot line double-click it in the **Visualizations** window and use the color palette to define it or enter a specific color.

### Border Width

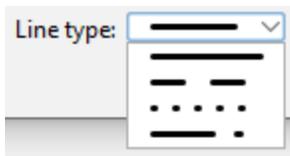
Choose a plot **Border width** between **1** (thin, the default) and **10** (thick).

### Line Width

For the plot line width, choose a **Line width** from the list. These options are also available for specific plot lines. To change an individual plot line width double-click it in the **Visualizations** window.

### Line Type

This option is only available for individual plot lines. To change an individual plot line type double-click it in the **Visualizations** window and choose a **Line type** from the list.



## ***Data Points, Anchors, and Numbers***

### Show Anchor

Click to select the **Show anchor** checkbox. By default, an anchor between the label and the plot point is displayed.

### Data Points Marker

From the **Data points marker** list, scroll to select a symbol to use to display the data points on plot lines. This option is also available for specific plot lines. To show data points on an individual plot line double-click it in the **Visualizations** window and select a symbol from the **Marker** list.

### Data Points Marker Size

Choose the **Data points marker size** between **1** (small) and **10** (large). The default is 3.

## Number of Decimals

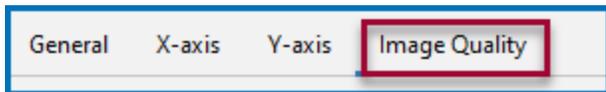
Choose the **Number of decimals** to display in the plot. The default is 2.

## Numeric Format

Choose the **Numeric format** to display in the plot **Auto**, **Decimal**, or **Scientific**.

## Image Quality

Click the **Image Quality** tab to access the settings below.



Choose a **Default file format**: **Portable Network Graphics .png** (the default), **Graphical Interchange Format .gif**, **Enhanced Windows Metafile .emf**, **Portable Document Format .pdf**, **PostScript .ps**, **Scalable Vector Graphics .svg**, or **Joint Photographic Experts Group .jpg**.



The following settings are only available for .png and .jpg file formats.

To change the resolution of a \*.png or \*.jpg image, click to clear the **Use default resolution** checkbox. Then enter a numerical value in the **Dots per inch (DPI)** field. For \*.jpg files, you can also use the slider to choose a number between 0 and 1 for the **JPG quality**. The default is 0.9. The lower the number, the smaller the file.

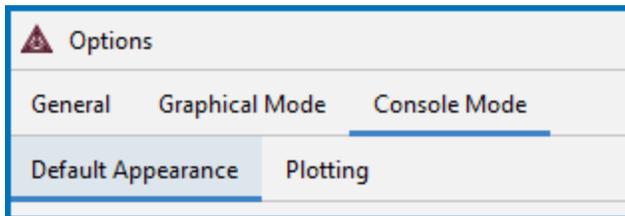
## Global Settings: Console Mode Default Appearance

Make global changes to the various defaults used for the settings.

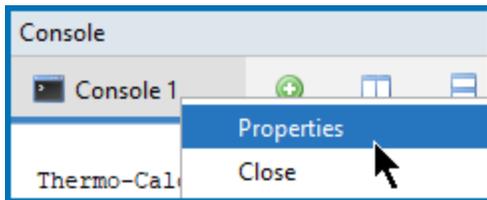
To open the **Options** window:

- Windows: Select **Tools** →  **Options**.
- Mac: Select **Thermo-Calc** <version> → **Preferences** or **Settings** or press <⌘> on the keyboard.

then click the **Console Mode** → **Default Appearance** tabs.



To change these defaults for an individual Console tab, in Console Mode, right-click the tab and select **Properties**.



**Reset to original settings:** In the lower corner of the window, click the **Reset to original settings** button to restore the original settings. You may have to scroll or expand the window to see the button. For local plot settings, there is a **Reset** button available instead.

### Console Name

The default **Console name** prefix is `Console` or enter another name to display as the tab label.

## Buffer Size

The default **Buffer size** is 40,000.

In Console Mode, sometimes the output overflows the window text buffer. This is how to increase the buffer size.

1. In the Console window right-click the tab to edit. For example, if this is the first tab, it is labeled **Console 1**.
2. Select **Properties** and either enter a number in the field or use the arrows to increase or decrease the **Buffer size**.

## Fonts: System Output, Command Prompt, or User Input

Click **Modify** to open the **Select Font** window and makes changes to the **Font Name, Style, Size**, or choose from a variety of color options for the font as required. The **Show only fonts that support Greek characters** checkbox is selected by default.

## Background Color

Click **Modify** to open the **Select Color** window to choose a background color for the Console Mode window where you enter commands.

## Default Directory

Click **Modify** to change the default directory file path to where the various file types are saved.