

# Additive Manufacturing (AM) Module User Guide

Thermo-Calc Version 2026a



Copyright 2026 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

---

## **Introduction to the Additive Manufacturing (AM) Module** ..... 5

*About the Additive Manufacturing (AM) Module* ..... 6

*Available Options* ..... 10

*Additive Manufacturing (AM) Module Help Resources* ..... 15

*Network License Restrictions* ..... 17

## **General Theory and Feature Overview** ..... 18

*Additive Manufacturing Module Theory* 19

*About the Heat Source Models* ..... 29

*About Heat Source Calibration* ..... 39

*About the Keyhole Model* ..... 43

*About the Absorptivity Model* ..... 47

*Additive Manufacturing Simulation Types* ..... 59

*Steady-state Calculation Types* ..... 61

## **AM Plots and Tables** ..... 63

*About the AM Plot Types* ..... 64

*AM Plot Type: 3D Plot* ..... 70

*AM Plot Type: Plot Over Line (2D)* ..... 77

*AM Plot Type: Probe (2D)* ..... 79

*AM Plot Type: Heat Source Calibration Parameters* ..... 81

*AM Plot Type: Melt Pool Dimensions* ..... 83

---

*AM Plot Type: Printability Map* ..... 84

*AM Plot Type: Parity Plot* ..... 89

*AM Plot Type: Melt Pool vs Energy Density* ..... 93

*AM Plot Type: Thermal Gradient vs. Solidification Rate* ..... 95

*AM Plots: Table View Options* ..... 98

## **Additive Manufacturing Workflow** 99

*Setting Up the Additive Manufacturing Simulation* ..... 100

*Additive Manufacturing Templates* ..... 105

## **Working with the Activity Nodes for an AM Simulation** ..... 107

*AM Calculator* ..... 108

*AM Calculator Conditions Settings* ..... 111

*AM Calculator Heat Source Settings* ..... 123

*AM Calculator Experiment File Requirements* ..... 134

*AM Calculator Materials Properties Settings* ..... 139

*AM Calculator Options Settings* ..... 143

*AM Calculator: Working with Probe Data* ..... 145

*AM Calculator: Plot Renderer Settings* 147

*Working with AM Visualizations* ..... 164

*AM Visualizations Window Plot Toolbar* ..... 170

*Menu or Keyboard Options: Geometry and 3D Plots* ..... 173

---

<i>Rotate, Zoom, and Pan 3D Plots:</i>	
<i>AM Calculations</i> .....	176
<i>Overlaying Plots in the AM Module</i> .....	177
<i>Visualizing Heat Source Calibrations</i> .....	178
<i>Visualizing Batch Calculations in the AM Module</i> .....	188

**Additive Manufacturing (AM) Module Examples** .....194

<i>AM_01: Transient Simulation of a Single Track</i> .....	196
<i>AM_02: Transient and Steady-state Simulations of a Single Track</i> .....	202
<i>AM_03: Steady-state Simulations</i> .....	208
<i>AM_04: Scheil Transient Steady-state</i> .....	213
<i>AM_05: Using AM Calculator Probe Data with the Diffusion Module (DICTRA)</i> .....	220
<i>AM_06a: Calibrating a Heat Source for a 316L Steel</i> .....	229
<i>AM_06b: Using the Calibrated Heat Source for a 316L Steel</i> .....	236
<i>AM_07: Batch Calculations for an IN718 Alloy</i> .....	241
<i>AM_08a: Grid Calculation for a Ti64 Alloy</i> .....	247
<i>AM_08b: Batch Calculations for a Ti64 Alloy</i> .....	253
<i>AM_09a: Grid Calculation for an SS316L Alloy</i> .....	259
<i>AM_09b: Batch Calculations for an SS316L Alloy</i> .....	264

---

<i>AM_10: CET Transition in an IN718 Alloy</i> .....	270
<i>AM_11: Comparing Single Tracks Printed on Casted and LPBF Substrates</i> .....	275
<i>AM_12: Using AM Calculator Probe Data with the Precipitation Module (TC-PRISMA)</i> .....	285
<i>AM_13: Using the Core-ring Beam Shape</i> .....	293
<i>AM_14: Using the Top-hat Beam Shape</i> .....	301
<i>AM_15: Electron Beam Melting in an IN718 Alloy</i> .....	309

# Introduction to the Additive Manufacturing (AM) Module

---

In this section:

About the Additive Manufacturing (AM) Module .....	6
Available Options .....	10
Additive Manufacturing (AM) Module Help Resources .....	15
Network License Restrictions .....	17

# 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](#)" on page 15 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. You can also simulate the powder bed fusion-electron beam (PBF-EB) process, commonly called Electron Beam Melting (EBM), using the electron beam heat source. 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 printability (aka process) maps.

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
- Thermal gradient and solidification rate
- Keyhole mode (with Gaussian, Core-ring, Top-hat, or Electron Beam heat sources) with or without fluid flow.

### ***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
- Thermal gradient and solidification rate
- Connect the above temperature vs. time response with the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA)
- Keyhole mode (with a *Transient with heat source from Steady-state* simulation using a Gaussian, Core-ring, Top-hat, or Electron Beam heat source) with or without fluid flow.

## **Overview of the Content**

### ***In This Guide***

There are ["Additive Manufacturing \(AM\) Module Help Resources"](#) on page 15 and ["Network License Restrictions"](#) on page 17 to consider and access.

Theory is included in a variety of feature topics. Some basic equations pertaining to the logic behind the simulation and settings are discussed in ["Additive Manufacturing Module Theory"](#) on page 19, as well as particular details about other features are discussed in ["About Heat Source Calibration"](#) on page 39, ["About the Keyhole Model"](#) on page 43, and ["About the Absorptivity Model"](#) on page 47, and ["About the Heat Source Models"](#) on page 29 for example.

Feature overviews are covered in a summary of the ["Additive Manufacturing Simulation Types"](#) on page 59 as well as the ["Steady-state Calculation Types"](#) on page 61 and associated ["About the AM Plot Types"](#) on page 64, which links you to feature-based topics for each calculation or plot type.

The workflow for the AM Module is outlined in ["Setting Up the Additive Manufacturing Simulation" on page 100](#) and there are two default ["Additive Manufacturing Templates" on page 105](#) 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" on page 100](#).

This set of topics specifically includes details about the settings for the ["AM Calculator" on page 108](#) and the specialized ["AM Calculator: Plot Renderer Settings" on page 147](#). There are many additional specialized sections related to working with, for example, probe data as well as experimental data files that are used in heat source calibrations and batch calculations.

There are many details related to visualizing the plots discussed in several sections, starting with ["Working with AM Visualizations" on page 164](#) and then extended with specialized sections ["Visualizing Heat Source Calibrations" on page 178](#) and ["Visualizing Batch Calculations in the AM Module" on page 188](#). The power of the AM Module is that there are many ways visualize both the set up (i.e. you can preview a variety of settings) and then finally examine and work with your results using the ["AM Visualizations Window Plot Toolbar" on page 170](#) and many other settings.

There are also several examples included with the installation that are generally discussed and described in the examples section or as part of the *Graphical Mode Examples Guide*.

## **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.

## Available Databases

The Additive Manufacturing Module is currently available together with the following databases, starting with the version indicated:

- Aluminum-based Alloys Database (TCAL9 and newer)
- Copper-based Alloys Database (TCCU6)
- High Entropy Alloys Database (TCHEA6 and newer)
- Magnesium-based Alloys Database (TCMG6 and newer)
- Molybdenum-based Alloys Database (TCMO1)
- Nickel-based Superalloys Database (TCNI12 and newer)
- Niobium-based Alloys Database (TCNB1)
- Noble Metal Alloys Database (TCNOBL3 and newer)
- Permanent Magnetic Materials Database (TCPMAG3)
- Solder Alloy Solutions Database (TCSLD5 and newer)
- Steel and Fe-alloys Database (TCFE13 and newer)
- Titanium and TiAl-based Alloys Database (TCTI5 and newer)



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.



The Additive Manufacturing (AM) Module is not available with the free Educational version of Thermo-Calc.

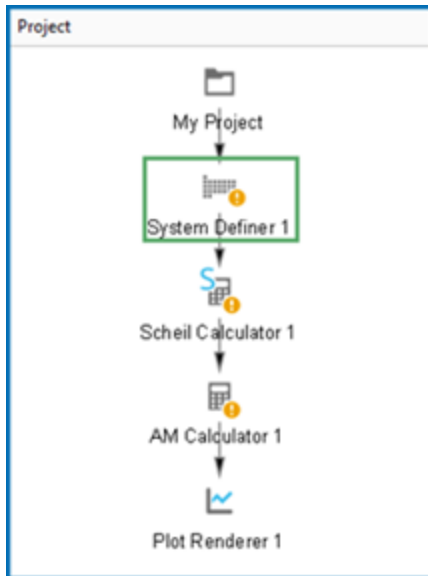
### Additive Manufacturing Templates

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 (demonstration) mode then this is indicated by the addition of **DEMO** text. See [Figure 1](#).

#### *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 an additional AM 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.

### What is Included?

- Some examples (i.e. AM\_01, AM\_02, AM\_03, 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: AM Calculation in Demo mode has limited functionality.` See [Figure 2](#).

- 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.
- All examples except those listed above require a full license to the Additive Manufacturing (AM) Module to run, plus often additional licenses (i.e. for databases or other Add-on Modules). 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.



You can also view the *Getting Started Guide* [on the website](#), which uses AM\_04 in the example.



Figure 1: The templates indicate that you are in DEMO mode and would need an AM license to fully use the templates or run certain examples as listed in the text.

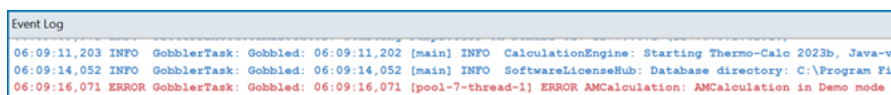


Figure 2: When in DEMO mode and you click on the AM Calculator node, the above note is included at the bottom of the Configuration window to remind you that a license is needed for full functionality. The Perform Tree button is not always available and sometimes even if it is available to click, the 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

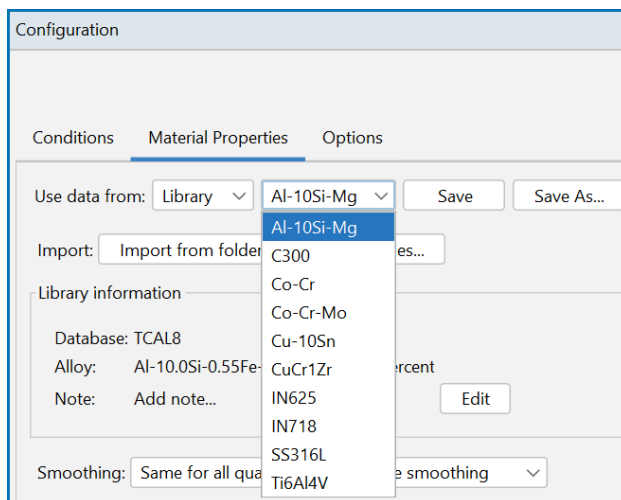


All predefined Material Properties libraries are available if you have a license for the Additive Manufacturing (AM) Module, even if you do not have the alloy database it is created with.

### 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).



## Other Options

### *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" on page 145.

### *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

The technical content is available in HTML format as an Online Help system that you can access from two locations. Both versions are viewed in your browser.

Go to these locations to access the same content:

- **Locally Installed Help:** When in Thermo-Calc, press F1 to open the current version of the help in a local browser. You can also click **Online Help** from the **My Project** page to open the file. Then search or navigate to the applicable folder to browse the contents. This does not require an Internet connection.
- **Web Help:** Go to the [Documentation](#) page to link to the most recent version of the web help. Then search or navigate to a relevant folder to browse the contents.



Between releases there may be minor updates to the web version compared to the installed version. All updates made to the web version are included in the next installed release version.



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.




## Opening the Web Version of the Help

A web version **specific to only the most recent release version** is available at <https://help.thermocalc.com/2026a/content/intro/help-home.htm>

## Opening the Installed 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 to go to the local browser version of the help:

- 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**.

## 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.



For more Additive Manufacturing (AM) Module learning resources, visit the video tutorials on our [website](#) or our [YouTube playlist](#).

## 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 Feature Overview

---

In this section:

Additive Manufacturing Module Theory .....	19
About the Heat Source Models .....	29
About Heat Source Calibration .....	39
About the Keyhole Model .....	43
About the Absorptivity Model .....	47
Additive Manufacturing Simulation Types .....	59
Steady-state Calculation Types .....	61

## 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" on page 108 using one of the "Additive Manufacturing Simulation Types" on page 59.



The settings related to the following are defined for the "AM Calculator" on page 108 starting with the "AM Calculator Conditions Settings" on page 111 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:

$$[Eq. 1] \quad \dot{H}_V = -div(J_Q) + \dot{Q}$$

$H_V$  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:

$$[Eq. 2] \quad H_V = \frac{H_m}{V_m}$$

$div$  denotes the divergence operator and  $J_Q$  is the heat flux given by Fourier's law:

$$[Eq. 3] \quad J_Q = -\kappa \text{ grad}(T)$$

where  $\kappa$  is the heat conductivity,  $grad$  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:

$$[Eq. 4] \quad H_m = \sum_{\beta} f^{\beta} H_m^{\beta} (T, x_k^{\beta})$$

$$[Eq. 5] \quad V_m = \sum_{\beta} f^{\beta} V_m^{\beta} (T, x_k^{\beta})$$

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

$$[Eq. 11] \quad \dot{V}_m = \dot{f}^L \Delta V_m + V_m \alpha \dot{T}$$

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:

$$[Eq. 12] \quad \dot{H}_V = \frac{1}{V_m} \left( \dot{f}^L \Delta H_m^L + c_P \dot{T} \right)$$

and Eq. 1 becomes:

$$[Eq. 13] \quad \frac{1}{V_m} \left( \dot{f}^L \Delta H_m^L + c_P \dot{T} \right) = \text{div}(\kappa \text{grad}(T)) + \dot{Q}$$

## 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:

$$[Eq. 14] \quad \dot{f}^L = \frac{df^L}{dT} \dot{T}$$

and Eq. 13 may be further simplified to:

$$[Eq. 15] \quad \frac{c_p^{eff}}{V_m} \dot{T} = \text{div}(\kappa \text{grad}(T)) + \dot{Q}$$

where the effective heat capacity  $c_p^{eff}$  is given by:

$$[Eq. 16] \quad c_p^{eff} = \left( \frac{df^L}{dT} \Delta H_m + c_P \right)$$

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.

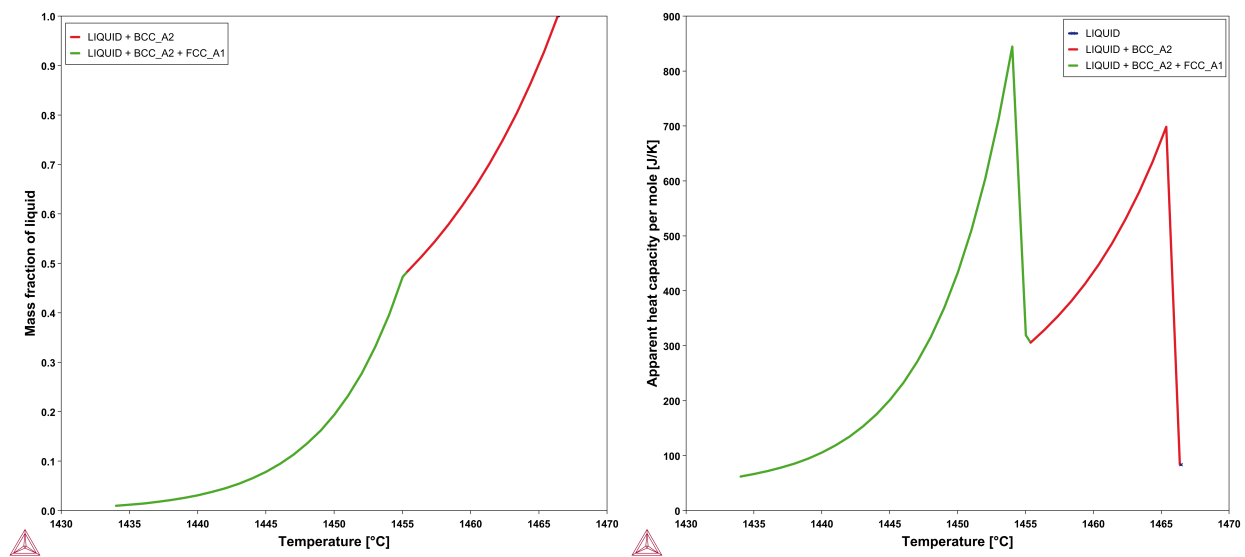


Figure 3: Results of a Scheil simulation of Fe-18Cr-10Ni alloy showing the mass fraction of the liquid (left) and the apparent heat capacity (right) both as a function of temperature.

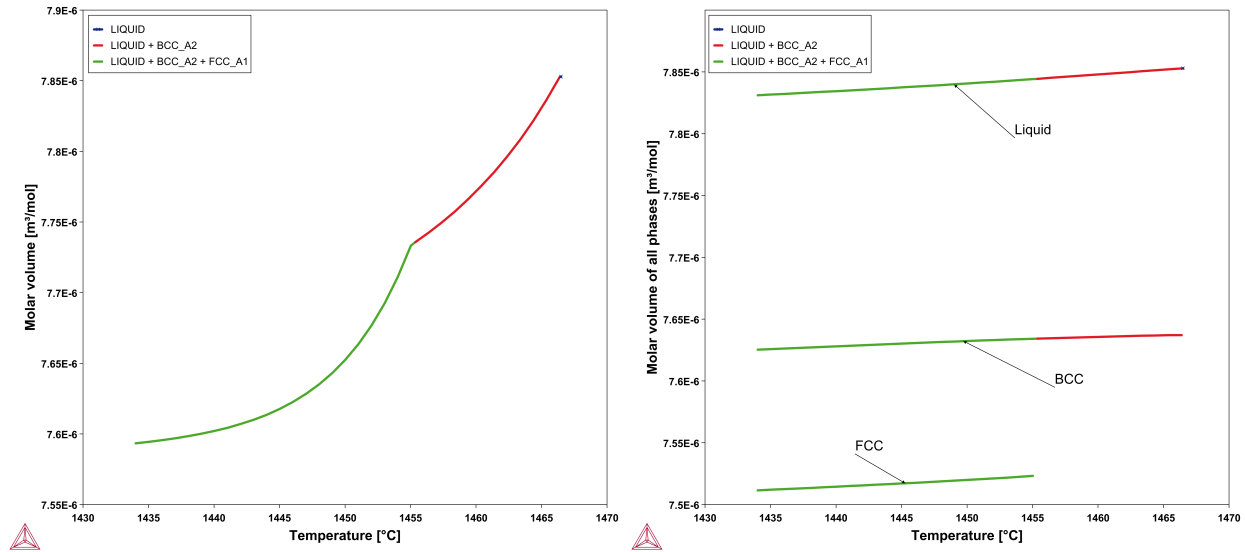


Figure 4: Molar volume from a Scheil simulation of Fe-18Cr-10Ni alloy. The right plot shows all phases as a function of temperature.

## The Enthalpy Method Using Thermo-Calc

In the enthalpy method we keep Eq. 1 and use Eq. 9 to write

$$[Eq. 17] \quad \frac{\dot{H}_m}{V_m} = -div(J_Q) + \dot{Q}$$

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.

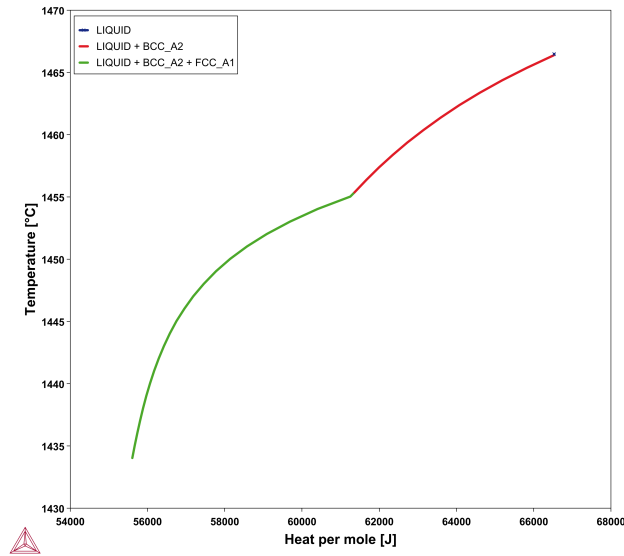


Figure 5: Enthalpy from a Scheil simulation of Fe-18Cr-10Ni alloy.

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:

$$[Eq. 18] \quad \text{div}[\kappa \text{grad}(T)] - \frac{v_b}{V_m} \text{grad}(H_m) + \dot{Q} = 0$$

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:

$$[Eq. 19] \quad \frac{\dot{H}_m}{V_m} = \text{div}[\kappa \text{grad}(T)] - \frac{v}{V_m} \text{grad}(H_m) + \dot{Q}$$

where  $v$  is the velocity vector field and may be imposed or obtained from the solution of the Navier-Stokes equation:

$$[Eq. 20] \quad \rho \dot{v} + \rho(v \text{grad})v - \mu(\text{divgrad})v = F_V - \text{grad}P$$

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.

$$[\text{Eq. 22}] \quad \text{div}(v) = 0$$

## Large Eddy Simulation Used for Fluid Flow

The use of a subgrid model [1963Sma] relies on the eddy-viscosity assumption and this is applied to account for the unresolved turbulent motions that cannot be directly simulated due to computational limitations. The Smagorinsky model is the default for fluid flow, and it provides a simple way to estimate *Subgrid-scale (SGS)* viscosity in large eddy simulation as,

$$\nu_{SGS} = (C_S \Delta)^2 |\bar{S}|$$

where:

- $C_S$  is the Smagorinsky constant (default value is 0.18).
- $\Delta$  represents the filter width (length scale).
- $|\bar{S}|$  is the magnitude of the resolved strain rate tensor.

## Heat Source Models



See additional theory related to heat sources in ["About the Heat Source Models"](#) on page 29 and ["About Heat Source Calibration"](#) on page 39. Settings information specific to heat sources is also found in ["AM Calculator Heat Source Settings"](#) on page 123.

## 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.

$$[Eq. 23] \quad Q_L = h(T - T_{ambient}) + \epsilon\sigma(T^4 - T_{ambient}^4)$$

where  $h$  is the convective heat transfer coefficient,  $\epsilon$  is the surface radiation emissivity and

$$\sigma = 5.669 \times 10^{-8} \text{ W/m}^2 \text{ K}^4$$

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:

$$[Eq. 24] \quad Q_N = 0$$

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:

$$[Eq. 25] \quad T = T_{baseplate}$$

$T_{baseplate}$  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:

$$[Eq. 26] \quad Q_E = 0.82J_E\Delta H_v$$

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:

$$[Eq. 27] \quad J_E = \frac{1}{\sqrt{2\pi MRT}} \left( P_o \exp\left(-\frac{\Delta G_v}{RT}\right) - P \right)$$

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 used for the bottom surface 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.

For the melted liquid at the top, the symmetry (front - in a steady-state simulation), and the keyhole surfaces, the free-slip boundary is used.

The most important boundary condition in Navier-Stokes setup is the Marangoni boundary condition that reflects the tangential force on the surface and the keyhole 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:

$$[Eq. 28] \quad \tau = \frac{\partial \gamma}{\partial T} \nabla_s T$$

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:

---

$$[Eq. 29] \quad k_{\text{powder}} = (6.3 + 22\sqrt{0.09k_{\text{solid}} - 0.016}) \frac{k_{\text{solid}} (1-\phi)}{(k_{\text{solid}} / k_{\text{air}})(10^{0.523-0.594\phi})^{-1}}$$

where  $k_{\text{air}} = 0.05784$ .

## References

- [1963Sma] J. Smagorinsky, General circulation experiments with the primitive equations: I. The basic experiment. *Mon. Weather Rev.* 91, 99–164 (1963).
- [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.

## About the Heat Source Models

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 (AM) Module, these heat source models are available: **Gaussian**, **Core-ring**, **Top-hat**, **Electron Beam**, **Double-ellipsoidal**, and **Conical**.

The **Configuration** window settings for each of these heat sources are further described in both the "AM Calculator Conditions Settings" on page 111 and "AM Calculator Heat Source Settings" on page 123 topics. There is a variety of additional background theory topics available based on the selected heat source.

Topic	Heat Source Models	Link to Theory
General Theory	For all models	"Additive Manufacturing Module Theory" on page 19
Keyhole model	Gaussian, Core-ring, Top-hat, and Electron Beam	"About the Keyhole Model" on page 43
Absorptivity	Gaussian, Core-ring, Top-hat, and Electron Beam	"About the Absorptivity Model" on page 47
Heat source calibrations	Gaussian, Double-ellipsoidal, and Conical	"About Heat Source Calibration" on page 39

### Gaussian Heat Source

The **Gaussian** 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 ( $W/m^2$ ) 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 [Figure 6](#).

$$[Eq. 1] \quad Q = \frac{2\epsilon P}{\pi r^2} \exp\left[-2\frac{(x-X_p)^2+(y-Y_p)^2}{r^2}\right]$$

Where  $Q$  is the power density deposited on the top surface ( $\text{W}/\text{m}^2$ ),  $\epsilon$  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.

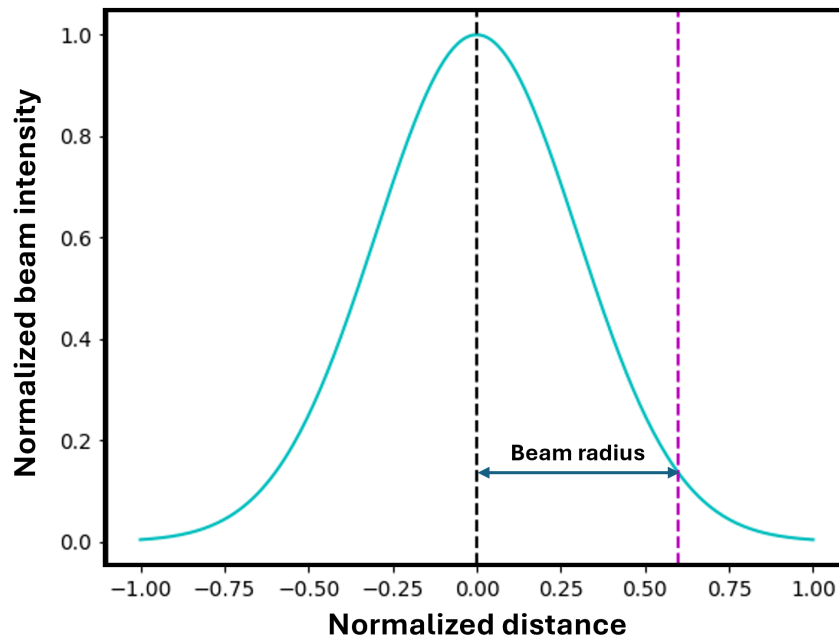


Figure 6: Schematic representation of energy distribution in Gaussian heat source model.



There are various examples using this heat source, i.e. "AM\_04: Scheil Transient Steady-state" on page 213 and "AM\_08a: Grid Calculation for a Ti64 Alloy" on page 247.



Also see additional theory related to using a Gaussian heat source with the keyhole option in "About the Keyhole Model" on page 43.

## Core-ring Heat Source

Gaussian beam distribution in additive manufacturing often leads to high energy intensity peaks and strong thermal gradients causing manufacturing defects such as keyhole porosity and suboptimal material processing performance. To overcome these challenges, spatial beam shaping is available as an alternate method to customize energy distribution and control the resulting thermal field [2023Bi; 2025Bir]. One of the alternative beam shapes is the **Core-ring** profile where energy is distributed between a central core and an outer ring.



## "AM\_13: Using the Core-ring Beam Shape" on page 293

From a production point of view, the primary benefit of **Core-ring** and **Top-hat** beam shapes is that there is a much wider and more stable melt pool. This means that you can increase the productivity of the process so that it prints faster, consequently saving on printing costs [2021Grü].



The alternative beam shape, **Top-hat**, is described in the next section.

A **Core-ring** heat source, like a Gaussian heat source, is a surface heat source model, which means that the power input from the heat source is only defined on the top surface of the workpiece, as a Neumann boundary condition. The intensity of a core-ring heat source can be modeled as a combination of a central Gaussian core beam and an outer ring beam [2024Gal].

The intensity of the core beam, as shown in Eq. 1, is given as follows:

$$[Eq. 2] \quad Q_{core} = \frac{2\epsilon P}{\pi r_{core}^2} \exp\left[-2 \frac{(x-X_p)^2 + (y-Y_p)^2}{r_{core}^2}\right]$$

Where

- $Q_{core}$  is the power density of the Gaussian core ( $W/m^2$ ),
- $\epsilon$  denotes the absorptivity of the laser beam,
- $P$  is the power of the heat source (W),
- $r_{core}$  is the  $1/e^2$  radius of the core beam,
- $x$  and  $y$  denote coordinates of the computational domain, and
- $X_p$  and  $Y_p$  represent the location of the heat source.

The intensity of the ring beam is derived by assuming the conservation of the total laser power,  $P$ , and is given as follows:

$$[Eq. 3] \quad Q_{ring} = \frac{\epsilon P}{2\pi w} \exp\left[-2 \frac{(r-w_{ring})^2}{r_{ring}^2}\right]$$

Where

- $w$  is the factor that ensures conservation of the total laser power,
- $w_{ring}$  is the radial distance of the outer ring beam from the center of the heat source, and

- $r_{ring}$  is the  $1/e^2$  radius of the ring beam.

Assuming that the total power remains constant,  $\varpi$  takes the following form:

$$\varpi = -\frac{\sqrt{2}w_{ring}r_{ring}}{8} \left( -2\sqrt{\pi} \left( 2 - \operatorname{erfc} \left( \frac{\sqrt{2}w_{ring}}{r_{ring}} \right) \right) - \frac{\sqrt{2}r_{ring}e^{-\frac{w_{ring}^2}{r_{ring}^2}}}{w_{ring}} \right)$$

Finally,  $r$  is given as follow:

$$r = \sqrt{(x - X_p)^2 + (y - Y_p)^2}$$

The total intensity of the core-ring beam is then given as:

$$Q = \alpha Q_{ring} + (1 - \alpha) Q_{core}$$

Where  $\alpha$  denotes the fraction of power in the ring beam and  $1 - \alpha$  consequently expresses the fraction of power in the core beam.

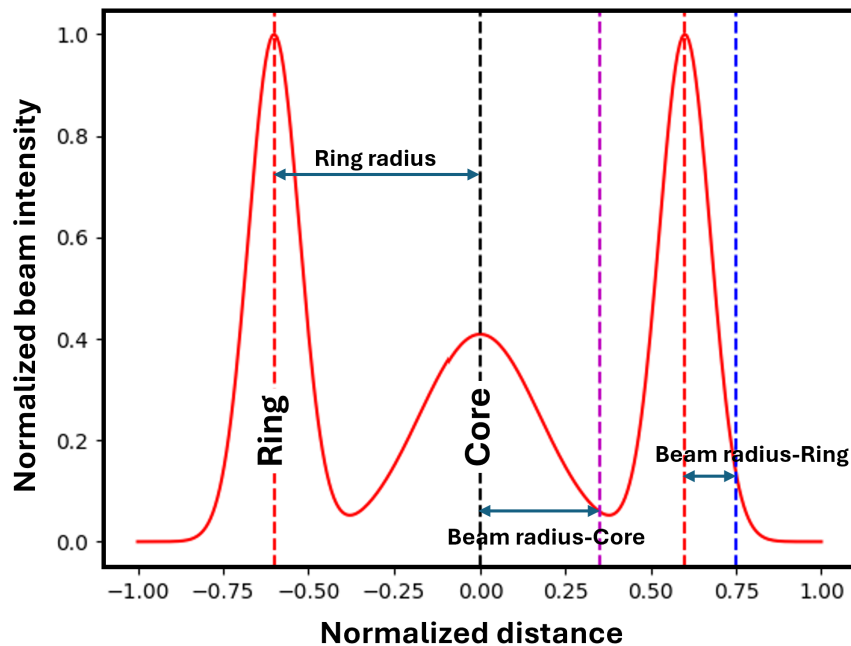


Figure 7: A schematic of the intensity profile of a core-ring heat source.

## Index Mode (Core-ring)

The intensity of the core-ring beam depends on different radii and the distribution of power between the core beam and the ring beam. For most users, these radii are not known since they are selected by the laser optics system of the additive manufacturing (AM) machine being used. The **Index Mode** is available with the **Core-ring** heat source to guide the selection of settings for the **Beam radius-Ring**, **Ring radius**, and **Amount of power in ring**, based on the **Beam radius-Core** and **Index** number. The AM Module incorporates configurations commonly provided by AM machines and laser optics systems as described below.

The ratio between the **Beam radius-Core** and the outer radius i.e. **Beam radius-Ring + Ring radius** is assumed to be constant and is equal to 0.38. Furthermore, **Beam radius-Ring** is assumed to be a function of the **Beam radius-Core**. For instance, for a **Beam radius-Core** of 50  $\mu\text{m}$  (i.e. diameter of 100  $\mu\text{m}$ ), the **Beam radius-Ring** would be 33  $\mu\text{m}$  and the **Ring radius** would then be 98.58  $\mu\text{m}$ . Similarly, for **Beam radius-Core** of 100  $\mu\text{m}$ , the other Beam radii would also scale accordingly, such that the **Beam radius-Ring** would become 66  $\mu\text{m}$  and the **Ring radius** would be 197.16  $\mu\text{m}$ .

For the **Index Mode**, you provide the **Beam radius-Core** and select the **Index** number and the rest of the heat source parameters are set as described above.

The **Amount of power in ring** depends on the selected **Index** number and is set as follows:

Index	0	1	0	3	4	5	6
Amount of power in ring	7%	25%	35%	50%	70%	80%	90%

## Custom Mode (Core-ring)

The **Custom Mode** for the **Core-ring** heat source allows you to manually enter all the parameters of the heat source. This requires an awareness of all the radii, either from the optics system of the AM machine in use, or from the calibration measurements of the laser beam.



If you plan to use the beam radii from the **Index Mode** but perhaps require a different power distribution in the core/ring, you can first select the **Index Mode**, enter the **Beam radius-Core**, then change to **Custom Mode** and enter the user-defined **Amount of power in ring**.

## Top-hat Heat Source

In addition to the Core-ring beam shape as an alternative to the Gaussian heat source, the **Top-hat** heat source is available with a flat and uniform energy distribution within a circular region and drops to zero outside the region.



"AM\_14: Using the Top-hat Beam Shape" on page 301

A **Top-hat** heat source is also called a *flat-top* and is considered a surface heat source model. In the AM Module, the top-hat heat source is approximated with a super-Gaussian beam, which means that the intensity of the beam at the edges decays softly but more steeply than a Gaussian beam, resulting in a flatter peak and sharper edges.

The intensity of such a beam is given as follows [1996Lu]:

$$[Eq. 4] \quad Q = Q_0 \exp\left(-2\left(\frac{r}{r_{tophat}}\right)^{2n}\right)$$

Where

- $r_{tophat}$  is the  $1/e^2$  radius of the top-hat beam and

$$r = \sqrt{(x - Xp)^2 + (y - Yp)^2}$$

- $n$  is the order of the super Gaussian beam, which determines the steepness of the beam edges. To get a better approximation of the top-hat beam, Thermo-Calc uses  $n = 5$ .
- $Q_0$  is the peak intensity of the top-hat beam, and given the conservation of the total power, it takes the following form:

$$Q_0 = \left(\frac{\epsilon n P}{\pi r_{tophat}^2}\right) \cdot \frac{2^{1/n}}{\Gamma\left(\frac{1}{n}\right)}$$

where  $\Gamma\left(\frac{1}{n}\right)$  represents the Gamma function evaluated at  $\frac{1}{n}$ ,  $\epsilon$  denotes the absorptivity of the laser beam, and  $P$  is the power of the heat source (W).

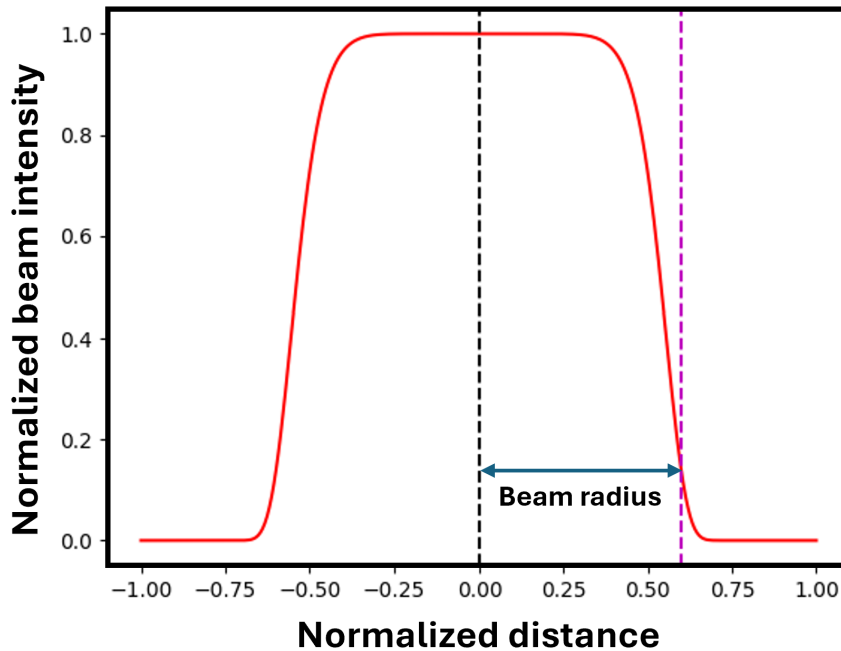


Figure 8: A schematic of the intensity profile of a top-hat heat source.

## Electron Beam Heat Source

Electron Beam Melting (EBM) is an additive manufacturing process in which a high-energy electron beam selectively melts metal powder layer-by-layer inside a vacuum chamber. The powder bed is preheated to a high temperature throughout the build, resulting in low residual stresses.

A major advantage of EBM over Laser Powder Bed Fusion (LPBF) is its ability to process materials that are prone to cracking when exposed to the steep thermal gradients of laser-based melting. The high build temperature, vacuum conditions, and efficient energy absorption of the electron beam make EBM suitable for alloys that are difficult to print with lasers, such as gamma titanium aluminides and certain nickel-based superalloys.

In the AM Module, the heat source for the EBM process is approximated by a Gaussian profile, which means that the heat intensity applied on the top surface of the domain is described as follows:

$$Q = \frac{2 \cdot A \cdot P}{\pi r^2} \exp \left[ -2 \frac{(x - X_p)^2 + (y - Y_p)^2}{r^2} \right]$$

Where  $Q$  is the power density ( $W/m^2$ ),  $A$  denotes the absorptivity of the electron beam,  $P$  is the power of the electron beam given as the product of the beam current and acceleration voltage,  $r$  is the  $1/e^2$  radius of the electron beam, and  $x$  and  $y$  denote coordinates of the computational domain while  $X_P$  and  $Y_P$  represent the location of the heat source.



Examples using this heat source: "AM\_12: Using AM Calculator Probe Data with the Precipitation Module (TC-PRISMA)" on page 285 and "AM\_15: Electron Beam Melting in an IN718 Alloy" on page 309.

## Double Ellipsoidal (Goldak) Heat Source

The **Double ellipsoidal** heat source model is a volumetric heat source model which was first proposed by Goldak et al. [1984Gol], and is now commonly used in LBP 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.

$$[Eq. 5] \quad Q_f = f_f \frac{6\sqrt{3}\varepsilon P}{\pi^{\frac{3}{2}} a_f b c} \exp \left[ -2 \frac{(x - X_p)^2}{a_f^2} + \frac{(y - Y_p)^2}{b^2} + \frac{(z - Z_p)^2}{c^2} \right]$$

$$Q_r = f_r \frac{6\sqrt{3}\varepsilon P}{\pi^{\frac{3}{2}} a_r b c} \exp \left[ -2 \frac{(x - X_p)^2}{a_r^2} + \frac{(y - Y_p)^2}{b^2} + \frac{(z - Z_p)^2}{c^2} \right]$$

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:

$$[Eq. 6] \quad f_f = \frac{2a_f}{a_f + a_r}$$

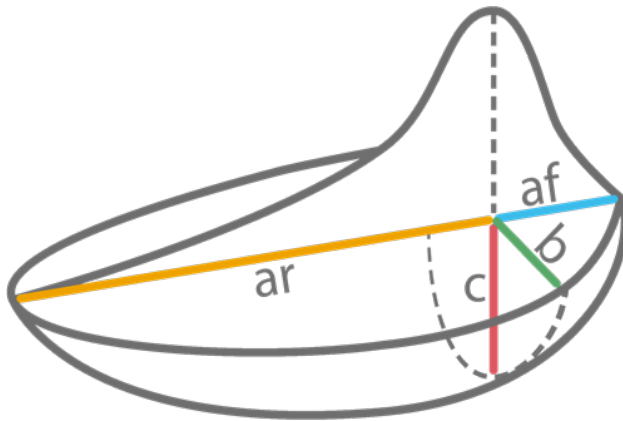


Figure 9: Schematic representation of energy distribution in double ellipsoidal heat source model.



For examples using this heat source, see "AM\_01: Transient Simulation of a Single Track" on page 196 and "AM\_02: Transient and Steady-state Simulations of a Single Track" on page 202.

## Conical Heat Source

The **Conical** heat source model 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.

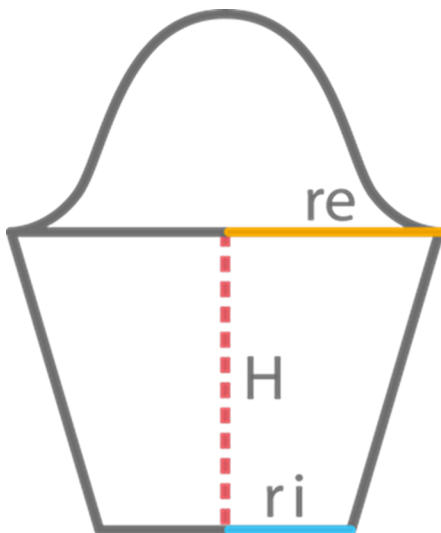


Figure 10: Schematic representation of energy distribution in conical heat source model.

The power density for the conical heat source is given as follows [2019Zha]:

$$[Eq. 7] \quad Q = \frac{6\epsilon P}{\pi H(r_e^2 + r_e r_i + r_i^2)} \exp\left[-2 \frac{(x-X_p)^2 + (y-Y_p)^2}{r_o^2}\right]$$

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.

## 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).
- [1996Lu] B. Lü, B. Zhang, X. Wang, Three-dimensional intensity distribution of focused super-Gaussian beams. *Opt. Commun.* 126, 1–6 (1996).
- [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).
- [2021Grü] J. Grünwald, F. Gehringer, M. Schmöller, K. Wudy, Influence of Ring-Shaped Beam Profiles on Process Stability and Productivity in Laser-Based Powder Bed Fusion of AISI 316L. *Metals (Basel)*. 11, 1989 (2021).
- [2023Bi] J. Bi, L. Wu, S. Li, Z. Yang, X. Jia, M. D. Starostenkov, G. Dong, Beam shaping technology and its application in metal laser additive manufacturing: A review. *J. Mater. Res. Technol.* 26, 4606–4628 (2023).
- [2024Gal] F. Galbusera, L. Caprio, B. Previtali, A. G. Demir, Analytical modeling and characterization of ring beam profiles for high-power lasers used in industrial manufacturing. *J. Manuf. Process.* 117, 387–404 (2024).
- [2025Akb] F. Akbari, Prediction of electron-solid interaction parameters using machine learning. *Med. Phys.* 52, 652–661 (2025).
- [2025Bir] M. Birg, T. Novotny, F. Bayer, “Beam Shaping in Additive Manufacturing A Toolbox of New Possibilities (White paper)” (2025); <https://www.eos.info/content/blog/entering-the-next-era-of-am-with-beam-shaping>.

## About Heat Source Calibration

---

There are models available to define the heat source in the AM process and that are also available with the **Heat Source Calibration** *Calculation Type*: **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 Type*) 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. The absorptivity of the liquid can be calculated but often requires correction due to uncertainty in the model, surface oxides, and so forth. 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, Core-ring, and Top-hat heat sources. See ["About the Keyhole Model"](#) on page 43 for some additional information.



For information about using calculated or constant absorptivity with a Gaussian Heat Source Calibration, there is additional theory in ["About the Absorptivity Model"](#) on page 47. The associated settings for this type of calibration is in the subsection ["Heat Source Calibration"](#) on page 56.

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](#)" on page 19 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).



When a **Users heat source** option is available to choose in the heat source list, then only the **Power** of the heat source needs 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**, **Core-ring**, **Top-hat**, **Double ellipsoidal**, or **Conical**.



For more details about including the **Heat Source Calibration** *Calculation Type* during the simulation set up, go to "[AM Calculator Conditions Settings](#)" on page 111. Then go to "[Visualizing Heat Source Calibrations](#)" on page 178 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 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.



"AM Calculator Experiment File Requirements" on page 134

Once the experiment data is entered, you 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** column. When this is done, click **Perform Tree** to perform the calibration.



If you select **Depth** from the **Calibrate for** list, then the experimental values for only the melt pool depth is needed.

## 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" on page 229: This example demonstrates the calibration of the heat source using the **Gaussian** and **Double ellipsoidal** heat sources.



"AM\_06b: Using the Calibrated Heat Source for a 316L Steel" on page 236: 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 based on [1994Kap] to pre-compute the keyhole shape and the corresponding mesh.



Example "[AM\\_06b: Using the Calibrated Heat Source for a 316L Steel](#)" on page 236 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.



Also see examples [AM\\_07](#), [AM\\_08a](#), [AM\\_08b](#), [AM\\_09a](#), [AM\\_09b](#), [AM\\_13](#), and [AM\\_15](#) where the keyhole model is used with fluid flow, or [AM\\_14](#) without fluid flow



By default, the **Use keyhole model** checkbox is selected. To make changes to this setting, go to "[AM Calculator Heat Source Settings](#)" on page 123 for additional information.



See "[About the Heat Source Models](#)" on page 29 for background details about the heat sources.

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.

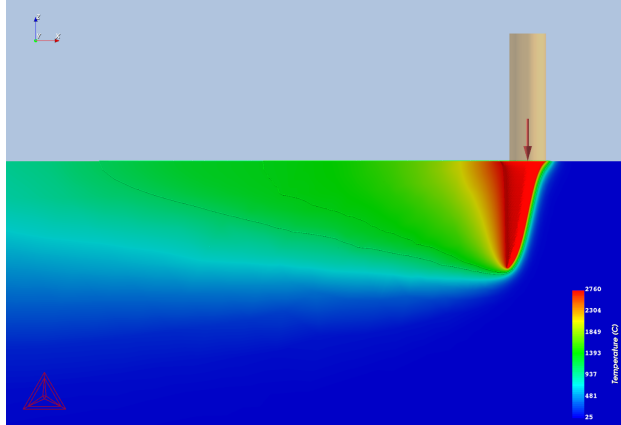


Figure 11: Side-view of a keyhole and melt pool for the material SS316L with Gaussian beam radius of  $40\ \mu\text{m}$  and the absorptivity set to 40%.

When a keyhole is formed, the energy is transmitted into the material through numerous reflections within the keyhole (see Figure 12).

For metals with an absorption factor of around 30% the reflections can increase the effective absorption to values around 90% 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 Coviello publication [2022Cov]. This strategy allows for computation of a keyhole shape, including multi reflections using ray tracing.

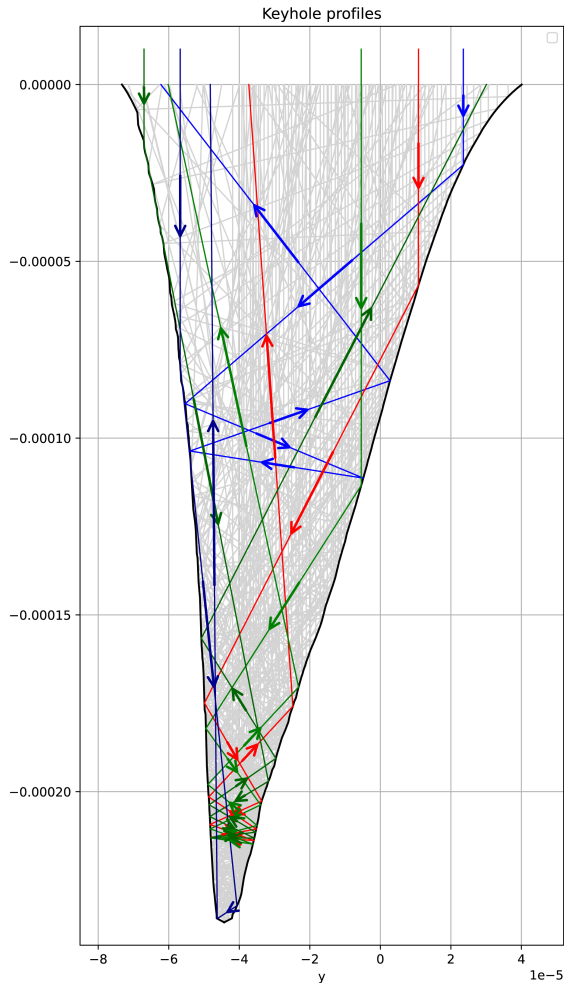


Figure 12: Ray tracking of the laser beam reflected at the wall of the keyhole. Blue, green and red lines highlight the path for three rays. Gray lines represent a fraction of all the reflected rays.

The keyhole model is limited to the surface heat sources (Gaussian, Core-ring, Top-hat, and Electron Beam). An additional input is the beam quality factor  $M^2$  of the laser beam where the default value is 1.0. The beam quality factor is a measure of laser beam quality, it relates the beam divergence of a laser beam to the minimum focused spot size that can be achieved.

## Multiple Reflections in the Keyhole

When using an **Electron Beam** heat source, the electron beam interacts with the material differently than a laser. When a laser hits the surface, part of the light reflects and continues traveling until it reaches another surface. Accounting for these multiple laser reflections in the keyhole is therefore essential for a laser heat source.

In EBM, high-energy electrons have low reflectivity and mainly undergo inelastic collisions with the material. Some of the non-absorbed energy is emitted as X-rays, while the rest appears as back-scattered electrons that spread in many directions with varying energies. Because this scattered energy is diffuse and relatively small, secondary reflections are ignored with the EBM heat source and the keyhole model.



For additional theory, see [Electron Beam Heat Source](#).

## 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).

[2022Cov] D. Coviello, A. D'Angola, D. Sorgente, Numerical Study on the Influence of the Plasma Properties on the Keyhole Geometry in Laser Beam Welding. *Front. Phys.* 9, 1–9 (2022).

## About the Absorptivity Model

The absorptivity of a flat metal surface is a function of the alloy composition, temperature, the wavelength of the heat source, and the angle of incidence.

The absorptivity of a metallic powder is more complex. Most metallic powders are oxidized at the surface where a thin layer of oxide often increases absorptivity. Multiple reflections of the laser rays also increases the effective absorptivity, e.g. at surface roughness or between the particles within the powder layer. The additional complications of the metallic powder are neglected in this model, which focuses on an ideal flat surface of the liquid phase after the powder has melted. This simplification can be justified at steady-state conditions when a melt pool is established and most laser rays hit the liquid surface in the melt pool.



Additional related theory topics include: ["Additive Manufacturing Module Theory"](#) on page 19, ["About the Heat Source Models"](#) on page 29, and ["About the Keyhole Model"](#) on page 43.

### Laser Beam Reflection

A fraction of the energy of a laser ray ( $E$ ) is absorbed when it strikes a surface, the rest of the energy is reflected ( $R$ ), i.e. the absorptivity is given by  $A = (1 - R) * E$ .

The Fresnel equation describes how the absorptivity depends on the angle of incidence  $\alpha$ .

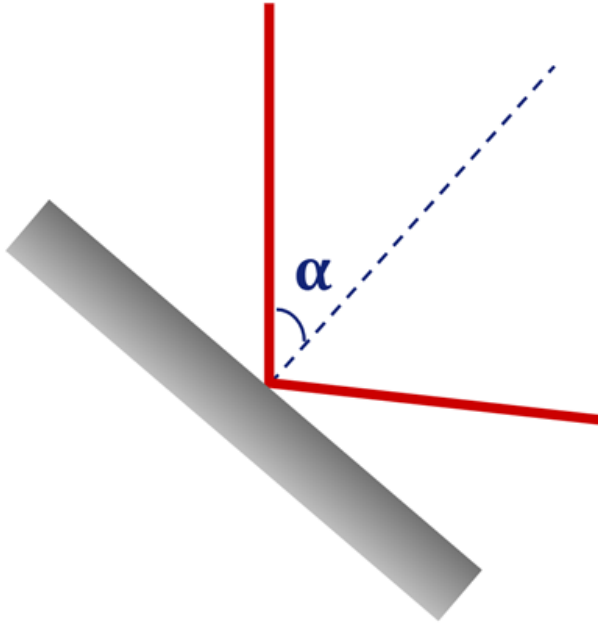


Figure 13: A laser ray strikes a surface at the incident angle  $\alpha$ .

For an unpolarized laser beam, the reflection coefficient  $R$  is the average of the parallel and perpendicular component, denoted by  $R_s$  and  $R_p$ :

$$R = (R_s + R_p)/2$$

Both components of  $R$  are given by the Fresnel equations:

$$R_s = \left| \frac{\cos \alpha - \sqrt{\hat{n}^2 - \sin^2 \alpha}}{\cos \alpha + \sqrt{\hat{n}^2 - \sin^2 \alpha}} \right|^2$$

$$R_p = \left| \frac{\hat{n}^2 \cos \alpha - \sqrt{\hat{n}^2 - \sin^2 \alpha}}{\hat{n}^2 \cos \alpha + \sqrt{\hat{n}^2 - \sin^2 \alpha}} \right|^2$$

The refractive index  $\hat{n}$  is a complex number  $\hat{n} = n - ik$  where  $k$  corresponds to the extinction coefficient.

The refractive index is directly related to the electric permittivity,  $\hat{\epsilon} = \epsilon_r - i\epsilon_i$ :

$$n = \frac{1}{\sqrt{2}} \left[ (\epsilon_r^2 + \epsilon_i^2)^{1/2} + \epsilon_r \right]^{1/2}$$

$$k = \frac{1}{\sqrt{2}} \left[ (\epsilon_r^2 + \epsilon_i^2)^{1/2} - \epsilon_r \right]^{1/2}$$

## About the Lorentz-Drude (LD) Model

The Lorentz-Drude model (LD) relates the electric permittivity with the electric resistivity [2021Yan].



The electric resistivity thermophysical property is included in many Thermo-Calc databases. For a list of databases that include it, see the [Properties that Can Be Calculated](#) brochure on the website or refer to the specific database documentation.

The LD model only considers the intraband absorption and is neglecting the interband part. The interband part of absorption is, however, often less dominant for wavelengths larger than 1  $\mu\text{m}$ ; for smaller wavelengths the model shows larger deviations when compared to experimental data.



For additive manufacturing lasers, the most common wavelength used is 1.06  $\mu\text{m}$ .

The LD model is used as a first approximation of the dielectric constants of an alloy. The model is then further calibrated against the available experimental information of dielectric constants, refraction index, and absorptivity of liquid metal, i.e. the correction functions  $f^r$  and  $f^i$  are modeled:

$$\epsilon_r = \epsilon_r^{LD} \cdot \left[ \sum_{i \in n} x_i \cdot f_i^r(\lambda, T) + \sum_{i \in n, j \in n} \sum_i x_i \cdot x_j \cdot f_{i-j}^r(\lambda, T) \right]$$

$$\epsilon_i = \epsilon_i^{LD} \cdot \left[ \sum_{i \in n} x_i \cdot f_i^i(\lambda, T) + \sum_{i \in n, j \in n} \sum_j x_i \cdot x_j \cdot f_{i-j}^i(\lambda, T) \right]$$

where

- $x_i$  is the mole-fraction of element  $i$
- $\lambda$  the laser wavelength, and
- $T$  the temperature.

The correction improves the model accuracy for the entire wavelength spectrum.

Following the work by Yang [2021Yan], the LD model obtains the real and imaginary parts of the electric permittivity:

$$\epsilon_r^{LD} = 1 - \frac{\omega_p^2}{f_L^2 + \gamma^2}$$

$$\epsilon_i^{LD} = \frac{\gamma}{f_L} \cdot \frac{\omega_p^2}{f_L^2 + \gamma^2}$$

where

- $\omega_p$  is the plasma frequency,
- $f_L$  is the laser frequency, and
- $\gamma$  is the damping frequency.

The plasma frequency is given by:

$$\omega_p = \sqrt{\frac{N_e q_e^2}{m_e \epsilon_0}}$$

where

- $N_e$  is the number density of free electrons,
- $q_e$  the absolute value of elementary charge,  $q_e = 1.6 \times 10^{-19} C$ ,
- $m_e$  is the electron mass,  $m_e = 9.11 \times 10^{-31} kg$ , and
- $\epsilon_0$  is the vacuum permittivity,  $\epsilon_0 = 8.85 \times 10^{-12} Fm^{-1}$ .

The damping frequency can be expressed as:

$$\gamma = \omega_p^2 \cdot \epsilon_0 \cdot \rho(T, x)$$

where

- $\rho(T, x)$  is the temperature- and composition-dependent electrical resistivity.

The laser frequency can be expressed as:

$$f_l = 2\pi \cdot \frac{c}{\lambda}$$

where

- $c$  is the speed of light in vacuum, and
- $\lambda$  is the wavelength of the laser.

The free electron density  $N_e$  is assumed to be given directly by the atomic mass and mass density:

$$N_e = N_A \cdot \rho_m \sum_{i \in N} \frac{x_i Z_i}{m_i}$$

where

- $N_A$  is the Avagadros constant,
- $\rho_m$  is the mass density of the alloy,
- $m_i$  is the atomic weight of element  $i$ , and
- $Z$  is the number of valence electrons per atom of element  $i$ .

## Heat Sources and Absorptivity

The LD model for liquid absorptivity describes a material property and is applicable to the **Gaussian**, **Core-ring**, and **Top-hat** heat sources where the energy is added to the surface of the material.



The LD model for absorptivity is not applicable to the volume heat sources **Double ellipsoidal** and **Conical**. Volume heat sources assume an effective absorptivity that indirectly can include the effect of keyholing by increasing the absorptivity as a function of energy density.

### Surface Heat Sources

The angle of incidence for a laser ray is zero when no keyhole forms. The angle of incidence varies greatly within the keyhole and the full angle dependent absorptivity model is used.

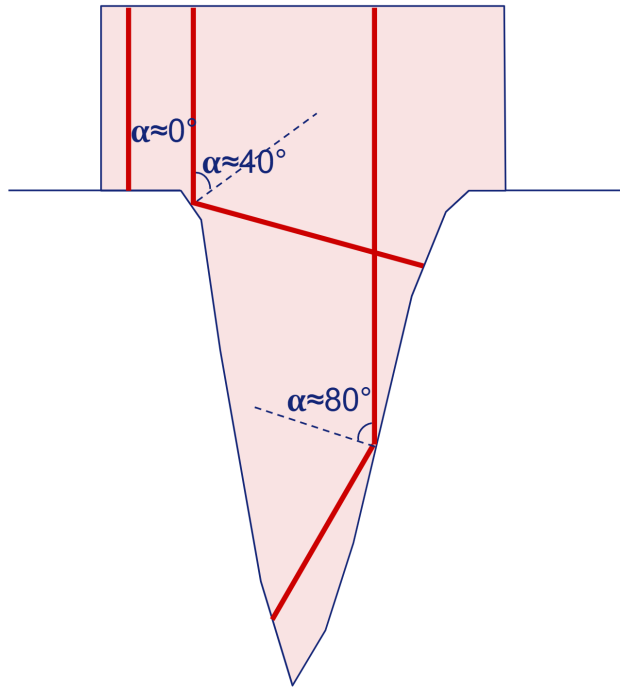


Figure 14: A schematic keyhole where three individual laser rays hit the surface at different incident angle  $\alpha$ . The keyhole model in the AM Module accounts for multiple reflections, but only the first reflection for each ray is shown for clarity.

The absorptivity model takes the wavelength of the heat source as input together with a prefactor that can scale the calculated absorptivity up or down. The scaling can be required to calibrate the simulations against experiments.



The following absorptivity settings are described in the ["AM Calculator Heat Source Settings"](#) on page 123. The examples use Gaussian, but Core-ring and Top-hat area also available. It is also available with the Electron Beam heat source, although the settings are slightly different and described in the settings section.

Heat Source

Gaussian Power: 100.0 W Absorptivity: Calculated with prefactor 1.0

Figure 15: Gaussian Heat Source using the Calculated Absorptivity. The calculated value is scaled with the value of "prefactor" where the default is 1.0, i.e. to use the calculated value as-is.

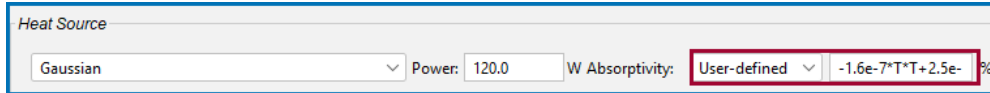
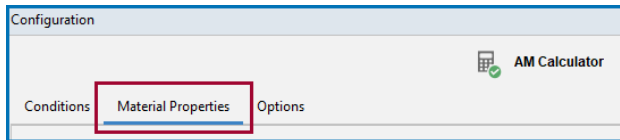


Figure 16: Gaussian Heat Source with User-defined function of the absorptivity.

The resulting absorptivity (given the wavelength and prefactor) can be visualized from the **Material Properties** tab on the **AM Calculator Configuration** window.



The following absorptivity options are described in "AM Calculator Heat Source Settings" on page 123.

From the **Plot** list, select:

- **Absorptivity** to visualize the temperature dependent absorptivity for the liquid phase of the selected material (Figure 17).
- **Absorptivity vs angle** to visualize the angle dependent absorptivity at liquidus and the evaporation temperature (Figure 18).
- **Electric resistivity** to visualize it as a function of temperature (Figure 19).



The electric resistivity is not directly used in the AM simulations, only indirectly through the absorptivity model that uses the electric resistivity when modeling the composition, temperature, and angle dependent absorptivity.

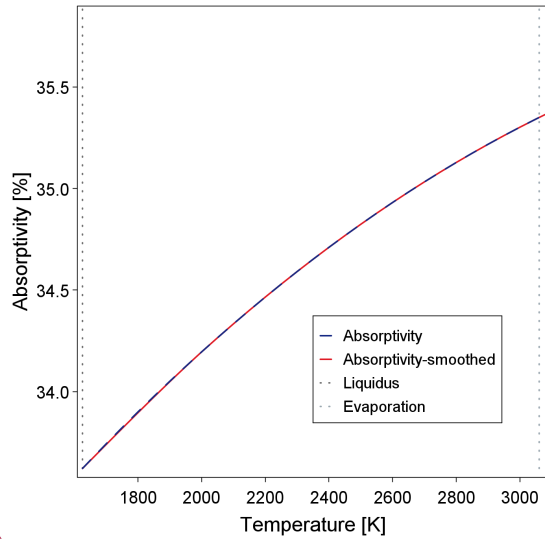


Figure 17: The calculated absorptivity as a function of temperature for the alloy IN738LC.

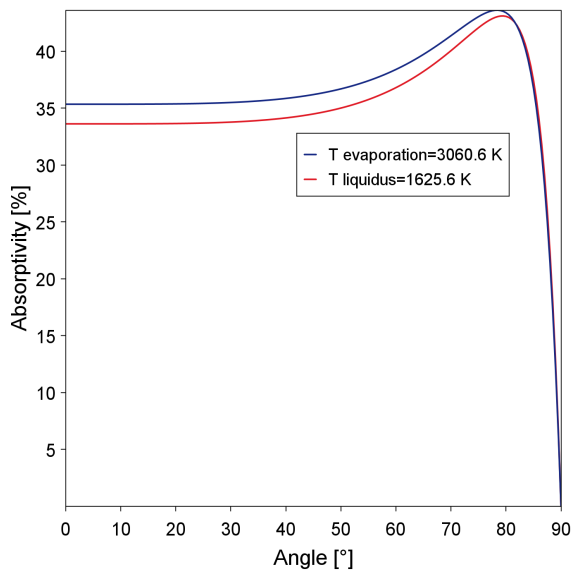


Figure 18: The calculated absorptivity as a function of angle of incidence for the alloy IN738LC. The red curve shows the absorptivity at the liquidus temperature and the blue curve the corresponding absorptivity at the evaporation temperature.

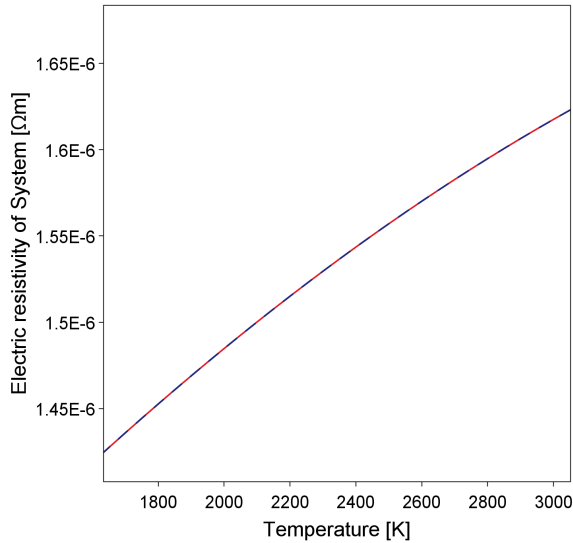


Figure 19: The electric resistivity of the liquid phase for the alloy IN738LC. The electric resistivity is used in the model for absorptivity for the liquid phase.

### Absorptivity of the Electron Beam Heat Source

The back-scattering coefficient  $\eta$ , of the electron beam on pure elements, can be described using various empirical equations [1989Aug], expressed as a function of the element's atomic number, the electron beam energy (keV), and the incident angle. Absorptivity is then given by

$$A = 1 - \eta$$

A model of back-scattering as a function of beam energy and incident angle was assessed using experimental backscatter constants collected from the literature for 48 elements.

An example of the back-scattering constant for Ti is shown below. This dataset reveals trends that apply to all elements: electron back-scattering increases with the incident angle, meaning absorptivity decreases. In contrast, for a laser heat source, absorptivity often increases with the incident angle. That means that a keyhole will be shallower for EBM when including this angle of incidence effect.

Another trend is that the back-scattering constant decreases at very low and very high electron acceleration voltages, which the empirical model [1989Aug] does not capture. However, typical EBM beam energies (60–160 keV) fall within the range where the model [1979Hun] is generally reliable. The empirical model is used for elements without experimental data.

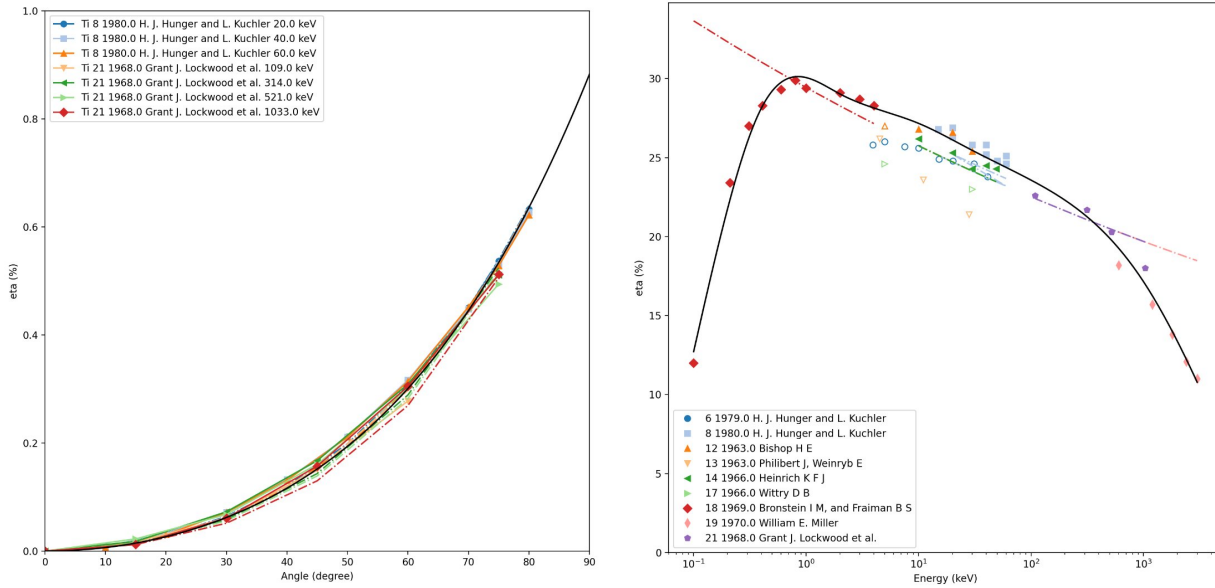


Figure 20: Back scatter constant for Ti, as function of incident angle (left) and as a function of electron beam energy (right). The dash dotted line shows August Wernisch model [1989Aug].

## Heat Source Calibration



Additional settings are described in the "AM Calculator Conditions Settings" on page 111 topic.



"AM\_06a: Calibrating a Heat Source for a 316L Steel" on page 229

For **Gaussian Heat Source Calibration** calculations, you can use the temperature and angle dependent calculated absorptivity and calibrate the scaling prefactor to match the given experimental data. The settings are on the **AM Calculator Configuration** window.

For **Absorptivity** select **Calculated (prefactor calibrated)** then enter the **Wavelength** of the heat source or use the default value.

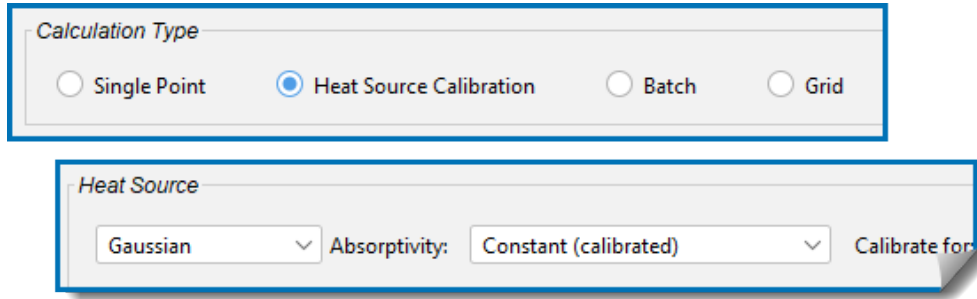
Calculation Type

Single Point
  Heat Source Calibration
  Batch
  Grid

Heat Source

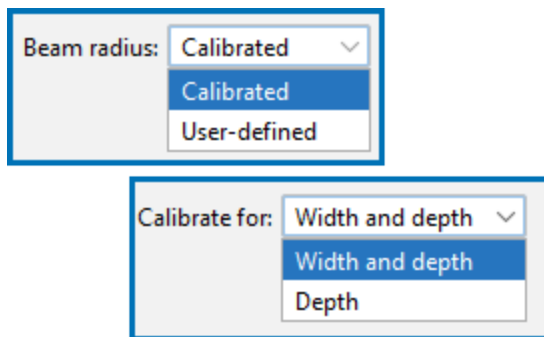
Gaussian ▼
 Absorptivity: Calculated (prefactor calibrated) ▼
 Wave length: 1064.0 nm
Calibrate

In order to use a constant **Absorptivity**, you can instead select **Constant (calibrated)**, which uses a constant value of absorptivity i.e. it is not dependent on temperature or incidence angle. Then you can calibrate that value to match the given experimental data.



For both **Absorptivity** options (**Calculated (prefactor calibrated)** or **Constant (calibrated)**) you can also either calibrate the **Beam radius** along with the absorptivity or the prefactor (choose **Calibrated**), or use a fixed value of **Beam radius** and calibrate only the absorptivity (choose **User-defined**).

Another setting available with the Gaussian Heat Source Calibration, is to calibrate for both the **Width and depth** of the melt pool from the experimental data or use only the **Depth** of the melt pool for the calibration. If you choose **Width and depth**, the *Experiment Data* values in the table require both the width and depth of the melt pool whereas if you only calibrate the depth then the data file needs to only include the information for the depth of the melt pool.



## References

- [1979Hun] H.-J. Hunger, L. Küchler, Measurements of the electron backscattering coefficient for quantitative EPMA in the energy range of 4 to 40 keV. Phys. Status Solidi 56, K45–K48 (1979).
- [1989Aug] H.-J. August, J. Wernisch, Analytical Expressions for the Electron Backscattering Coefficient. Phys. Status Solidi 114, 629–633 (1989).

[2021Yan] Z. Yang, A. Bauereiß, M. Markl, C. Körner, Modeling Laser Beam Absorption of Metal Alloys at High Temperatures for Selective Laser Melting. *Adv. Eng. Mater.* 23, 2100137 (2021).

# Additive Manufacturing Simulation Types

Three simulation types are available on the "AM Calculator" on page 108 – *Steady-state*, *Transient*, and *Transient with heat source from Steady-state*.



"Additive Manufacturing Module Theory" on page 19 and "About the Heat Source Models" on page 29.



The **keyhole model** is available when **Gaussian, Core-ring, Top-hat, or Electron Beam** heat sources are used for either the *Steady-state* or *Transient with heat source from Steady-state* types of simulation (see "About the Keyhole Model" on page 43). It is available with or without fluid flow.

## Steady-state



For **Steady-state** simulations, you can also choose to run different "Steady-state Calculation Types" on page 61.

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, 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 involves the assumption that the temperature distribution and fluid flow inside the melt pool instantly reach steady state. First you 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 you have the steady state solution, use this solution inside the melt pool and map it as a “heat source” in the transient simulations. Using this approach, you do not have to solve for the complicated Navier-Stokes equations at each time step in the transient simulation, yet it still captures the effect of fluid flow on the shape of the melt pool. Furthermore, it also reduces 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 considerably decreases the numerical complexity of the full-scale transient simulations, which consequently reduces the simulation time without making a significant compromise on the accuracy of the solution.

---

## Steady-state Calculation Types

---

When setting up a **Steady-state** simulation on the **AM Calculator**, you can also select different *Calculation Types*.



"Additive Manufacturing Simulation Types" on page 59 and for overall general theory, see "Additive Manufacturing Module Theory" on page 19.



For background theory related to Heat Sources, see "About the Heat Source Models" on page 29 and "About the Absorptivity Model" on page 47. For *Gaussian*, *Double ellipsoidal*, and *Conical* heat sources, also see "About Heat Source Calibration" on page 39 for additional details.

### ***Single Point***

The standard steady-state calculation for a single set of processing parameters. This is available with all heat sources.

See [Calculation Type \(Steady-state\)](#) for settings information on the AM Calculator.

### ***Heat Source Calibration***

To calibrate a heat source based on the imported or entered experimental data instead of performing a single point calculation. This is available with **Gaussian**, **Double ellipsoidal**, or **Conical** heat sources.

See [Calculation Type \(Steady-state\)](#) for settings information on the AM Calculator. For information about previewing configuration settings (on the AM Calculator) as well as plot results (on the Plot Renderer), see "[Visualizing Heat Source Calibrations](#)" on page 178.

## **Batch**

To include power and scan speed data, which is read from a file such as a spreadsheet. You can optionally include experimental melt pool width and depth (if there is data), and compare this to the calculated results using a Parity plot on the Plot Renderer. This is available with all heat sources.

For settings information on the AM Calculator, see [Calculation Type \(Steady-state\)](#). For information about previewing configuration settings (on the AM Calculator) as well as plot results (on the Plot Renderer), see "[Visualizing Batch Calculations in the AM Module](#)" on [page 188](#).

## **Grid**

To evaluate two axis variables of power and scanning speeds in the specified range and number of steps. This is available with all heat sources.

For settings information on the AM Calculator, see [Calculation Type \(Steady-state\)](#). For Plot Renderer Configuration settings, see "[AM Calculator: Plot Renderer Settings](#)" on [page 147](#).

---

# AM Plots and Tables

---


In this section:

About the AM Plot Types .....	64
AM Plot Type: 3D Plot .....	70
AM Plot Type: Plot Over Line (2D) .....	77
AM Plot Type: Probe (2D) .....	79
AM Plot Type: Heat Source Calibration Parameters .....	81
AM Plot Type: Melt Pool Dimensions .....	83
AM Plot Type: Printability Map .....	84
AM Plot Type: Parity Plot .....	89
AM Plot Type: Melt Pool vs Energy Density .....	93
AM Plot Type: Thermal Gradient vs. Solidification Rate .....	95
AM Plots: Table View Options .....	98

## About the AM Plot Types

When working with an **AM Calculator** with the **Plot Renderer** as a successor, the visualization is different from the other calculators available in Thermo-Calc. When setting up the **AM Calculator**, there are many options available that impact what plot types are going to be available. Further, when defining the **Plot Renderer**, there are additional ways and axis variables or other setting requirements that enable you to display certain plot types, as well as preview a variety of settings on the **Visualizations** window.

In addition, when a **Plot Renderer** is added as a successor to the **AM Calculator**, the default layout of the **Configuration** window, with respect to the different tabs, relates to how the **AM Calculator** is set up. This is then further related to the tabs located on the **Visualizations** window.

 See ["Working with AM Visualizations"](#) on page 164 for information about how these tabs are connected between the windows. Also see ["Overlaying Plots in the AM Module"](#) on page 177.

### AM Plot Types

<i>Simulation Type</i>	<i>Calculation Type*</i>	<i>Plot Type</i>	<i>Export Result Files</i>	<i>Display Type</i>	<i>Notes</i>	<i>Tab Name Configuration Window (Setting)</i>	<i>Tab Name Visualizations Window (Plot)</i>	<i>Link to More information</i>
Steady-state, Transient, or Transient with heat source from Steady-state	Steady-state → Single Point, Heat Source Calibration, Batch, and Grid	3D Plot	Yes	Surface colormap	From the <b>Display</b> list select <b>Surface colormap</b> . Available when Temperature, Surface tension, Thermal conductivity, or Dynamic	3D Plot	3D Plot  When overlaying certain plots, there is also a suffix in brackets that matches the name of the AM Calculator it is connected to.	<a href="#">"AM Plot Type: 3D Plot" on page 70</a>

<i>Simulation Type</i>	<i>Calculation Type*</i>	<i>Plot Type</i>	<i>Export Result Files</i>	<i>Display Type</i>	<i>Notes</i>	<i>Tab Name Configuration Window (Setting)</i>	<i>Tab Name Visualizations Window (Plot)</i>	<i>Link to More information</i>
					viscosity is selected as an Axis variable.			
				Slice	From the <b>Display</b> list select <b>Surface colormap</b> or <b>Contour</b> and then click to select the <b>Slice</b> checkbox.			
				Contour	From the <b>Display</b> list select <b>Contour</b> . Available when Temperature, Surface tension, Thermal conductivity, or Dynamic viscosity is selected as an Axis variable.		e.g. <b>3D Plot (AM Calculator 1)</b>	
				Iso-surface	From the <b>Display</b> list select <b>Iso-surface</b> . Available when			

<i>Simulation Type</i>	<i>Calculation Type*</i>	<i>Plot Type</i>	<i>Export Result Files</i>	<i>Display Type</i>	<i>Notes</i>	<i>Tab Name Configuration Window (Setting)</i>	<i>Tab Name Visualizations Window (Plot)</i>	<i>Link to More information</i>
					Temperature, Surface tension, Thermal conductivity, or Dynamic viscosity is selected as an Axis variable.			
				Arrow, 2D glyph, Cone, or Line	Available when <b>Flow velocity field</b> is selected as an Axis variable. Requires that the <b>Fluid flow including Marangoni effect</b> checkbox is selected on the AM Calculator.			
Steady-state, Transient, or Transient with heat source from Steady-state	Steady-state → Single Point, Heat Source Calibration, Batch, and Grid	Plot Over Line (2D)	No	not applicable	Once the plot is performed, switch between a plot and a table by clicking the <b>Table View</b> button on the <b>Visualizations</b>	Plot Over Line	Plot Over Line	"AM Plot Type: Plot Over Line (2D)" on page 77 "AM Plots: Table View Options" on

<i>Simulation Type</i>	<i>Calculation Type*</i>	<i>Plot Type</i>	<i>Export Result Files</i>	<i>Display Type</i>	<i>Notes</i>	<i>Tab Name Configuration Window (Setting)</i>	<i>Tab Name Visualizations Window (Plot)</i>	<i>Link to More information</i>
					window.			page 98
Steady-state	Heat Source Calibration	Heat Source Calibration (2D)	Yes	not applicable	This plot is available once the calibration is completed.	Heat Source Calibration	Heat Source Calibration Parameters	"AM Plot Type: Heat Source Calibration Parameters" on page 81
Steady-state	Heat Source Calibration	Melt Pool Dimensions (2D)	Yes	not applicable	This plot is available once the calibration is completed.	Heat Source Calibration	Melt Pool Dimensions	"AM Plot Type: Melt Pool Dimensions" on page 83
Steady-state	Batch and Grid	Printability Map (2D)	Yes	not applicable	Also toggle between <b>Show Table</b> and <b>Hide Table</b> on the <b>Configuration</b> window.	Batch or Grid	Printability Map	"AM Plot Type: Printability Map" on page 84 "AM Plots: Table View Options" on page 98
Steady-state	Batch and Grid	Meltpool vs Energy Density (2D)	Yes	not applicable	Also toggle between <b>Show Table</b> and <b>Hide Table</b> on the <b>Configuration</b> window.	Batch or Grid	Meltpool vs Energy Density	"AM Plot Type: Melt Pool vs Energy Density" on page 93 "AM Plots: Table View

<i>Simulation Type</i>	<i>Calculation Type*</i>	<i>Plot Type</i>	<i>Export Result Files</i>	<i>Display Type</i>	<i>Notes</i>	<i>Tab Name Configuration Window (Setting)</i>	<i>Tab Name Visualizations Window (Plot)</i>	<i>Link to More information</i>
								<a href="#">Options"</a> on page 98
Steady-state	Batch	Parity Plot (2D)	Yes	not applicable	Also toggle between <b>Show Table</b> and <b>Hide Table</b> on the <b>Configuration</b> window.	Batch	Parity Plot	"AM Plot Type: Parity Plot" on page 89 "AM Plots: Table View Options" on page 98
Steady-state, Transient, or Transient with heat source from Steady-state	Single Point, Heat Source Calibration, Batch, or Grid	Thermal Gradient vs Solidification Rate (2D)	No	not applicable	Once the plot is performed, switch between a plot and a table by clicking the <b>Table View</b> button on the <b>Visualizations</b> window.  It is also possible to visualize this via the <b>3D Plot</b> tab when you define the Axis variables <b>Solidification rate</b> and <b>Thermal gradient</b> .	Thermal Gradient vs Solidification Rate	Thermal Gradient vs Solidification Rate	"AM Plot Type: Thermal Gradient vs. Solidification Rate" on page 95

<i>Simulation Type</i>	<i>Calculation Type*</i>	<i>Plot Type</i>	<i>Export Result Files</i>	<i>Display Type</i>	<i>Notes</i>	<i>Tab Name Configuration Window (Setting)</i>	<i>Tab Name Visualizations Window (Plot)</i>	<i>Link to More information</i>
Transient or Transient with heat source from Steady-state	not applicable	Probe (2D)	No	not applicable	<p>Probe positions and time steps available.</p> <p>Defined and available from a specific a tab on the Plot Configuration window.</p> <p>Once the plot is performed, switch between a plot and a table by clicking the <b>Table View</b> button on the <b>Visualizations</b> window.</p>	Probe	Probe	<p><a href="#">"AM Plot Type: Probe (2D)" on page 79</a></p> <p><a href="#">"AM Plots: Table View Options" on page 98</a></p>

\* For **Transient** and **Transient with heat source from Steady-state** simulations there are no additional *Calculation Types* selected.

## AM Plot Type: 3D Plot

The **3D Plot** is available with all simulation types (*Steady-state*, *Transient*, and *Transient with heat source from Steady-state*). It is also available with *Steady-state* calculation types **Heat Source Calibration**, **Batch**, and **Grid**.

The **3D Plot** type has its own tab on both the **Plot Renderer Configuration** window and the **Visualizations** window. This plot type is further visualized using additional **Display** options.

Further, when connecting two AM Calculators to plot combinations of axis variables, separate tabs are added to the **Visualizations** window that have a suffix of the name of the AM Calculator added in brackets, e.g. **3D Plot (AM Calculator 1)**. It is recommended that the AM Calculator is renamed to help you identify which tab belongs to which calculation.



See "Overlying Plots in the AM Module" on page 177 for more details and "AM Plot Type: Thermal Gradient vs. Solidification Rate" on page 95 for an example.

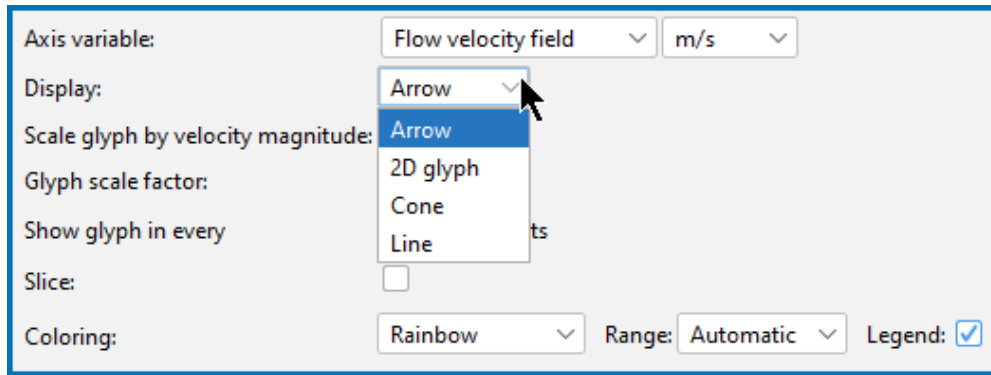
### 3D Plot Display Options

The **Display** options available are also based on the **Axis variable** selected. The **Slice** plot, which is a unique type of plot, is also available when either **Surface colormap** or **Contour** is selected.

Surface colormap, Contour, and Iso-surface are available when **Temperature**, **Surface tension**, **Thermal conductivity**, or **Dynamic viscosity** is selected as an *Axis variable*.

Axis variable:	Temperature	Kelvin
Display:	Surface colormap	Opacity: 100 %
Slice:	Surface colormap	
Coloring:	Iso-surface	: Automatic Legend: <input checked="" type="checkbox"/>

Arrow, 2D Glyph, Cone, or Line are available when **Flow velocity field** is selected as an *Axis variable*. It also requires that the **Fluid flow including Marangoni effect** checkbox is also selected on the **AM Calculator**.



## Examples

### Surface Colormap

This is an example from "AM\_01: Transient Simulation of a Single Track" on page 196.

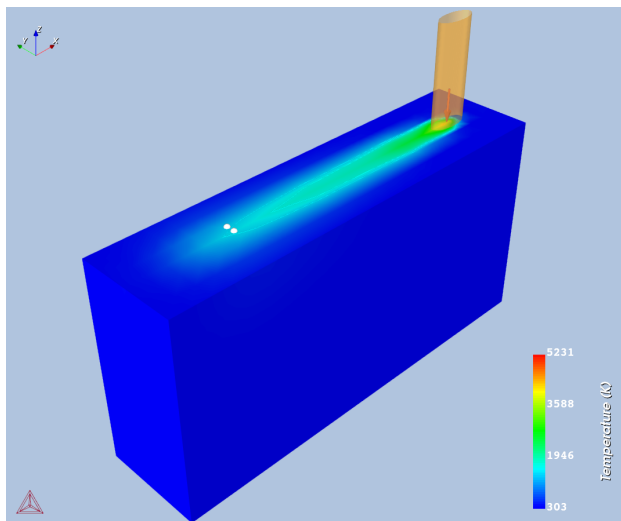


Figure 21: Surface plot of temperature distribution after the single track scan is completed. Probes P1 and P2 can also be seen on the top surface.

### Slice Plot

This is an example from "AM\_01: Transient Simulation of a Single Track" on page 196.

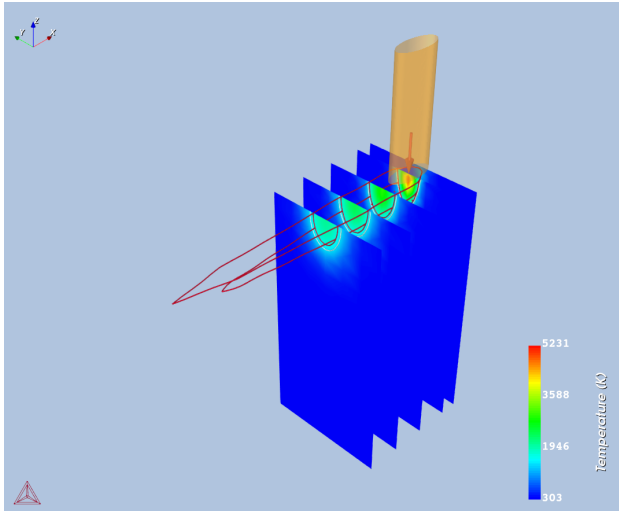
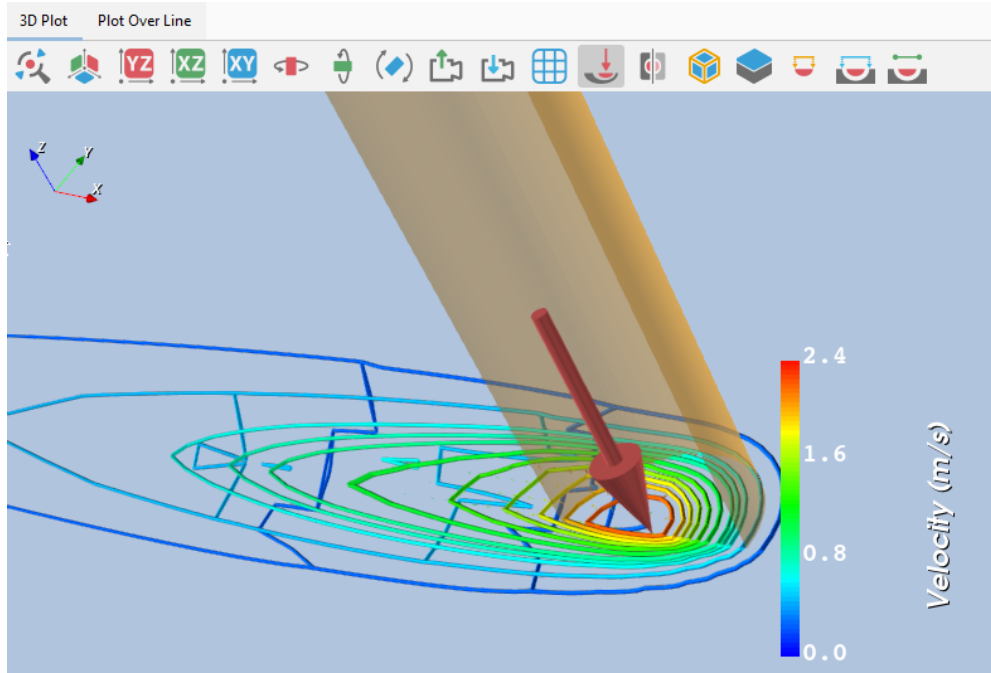


Figure 22: Slices of the computational domain, orthogonal to the scanning direction, showing temperature distribution at different cross-sections. Red tubes show the contour of the melt pool (at liquidus temperature).

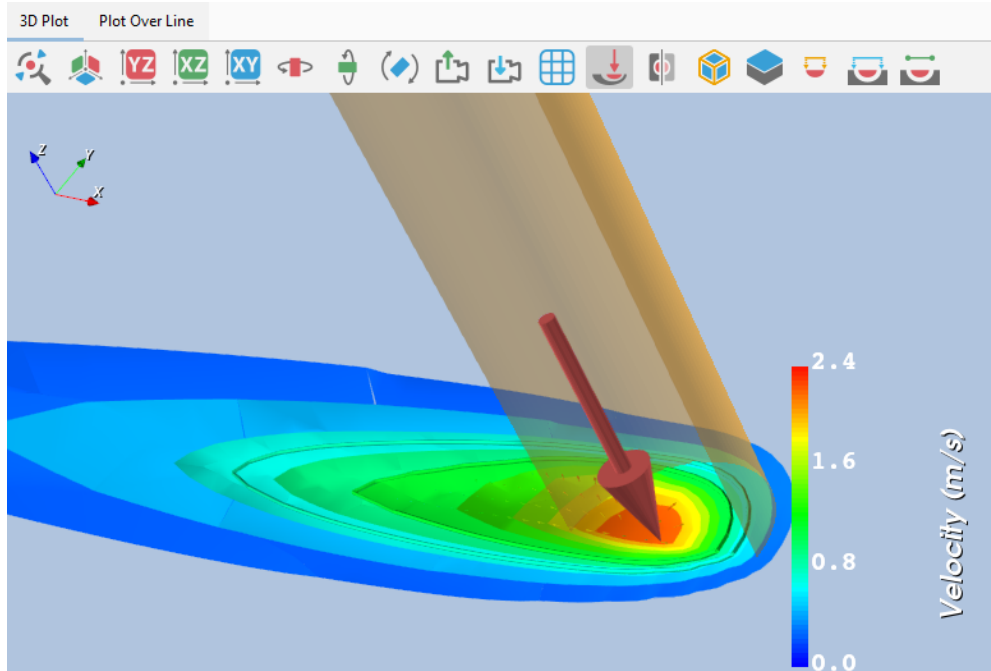
## Contour

This is an example using "AM\_03: Steady-state Simulations" on page 208 to zoom into the **AM Calculator**, which in that example is renamed to **Gaussian HS, with Fluid**. On the **Plot Renderer**, the **Display** setting is changed from the default to **Contour**.



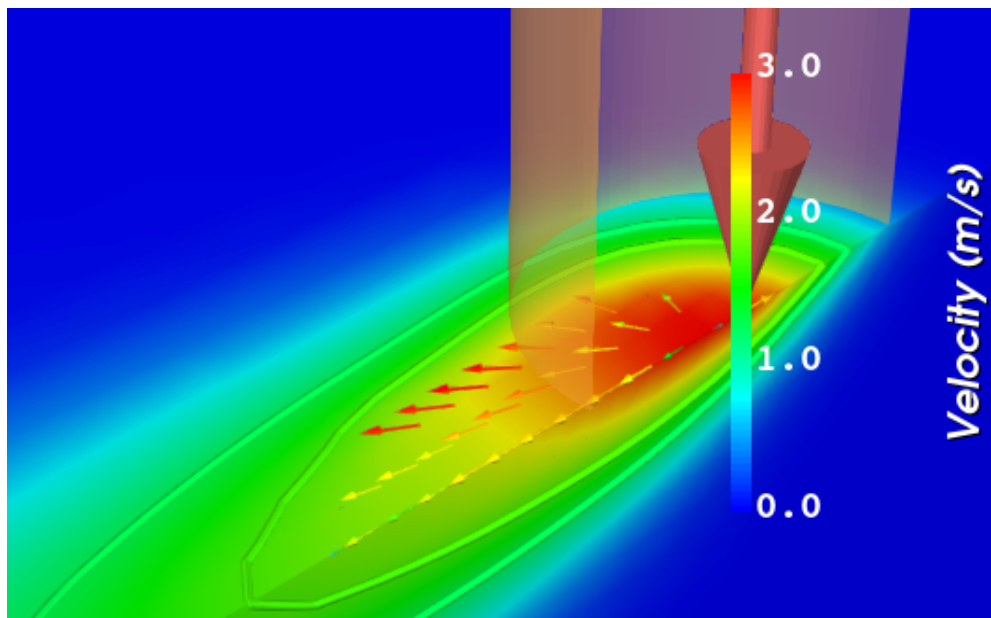
### *Isosurface*

This is an example using "AM\_03: Steady-state Simulations" on page 208 to zoom into the **AM Calculator**, which in that example is renamed to **Gaussian HS, with Fluid**. On the **Plot Renderer**, the **Display** setting is changed from the default to **Iso-surface**.



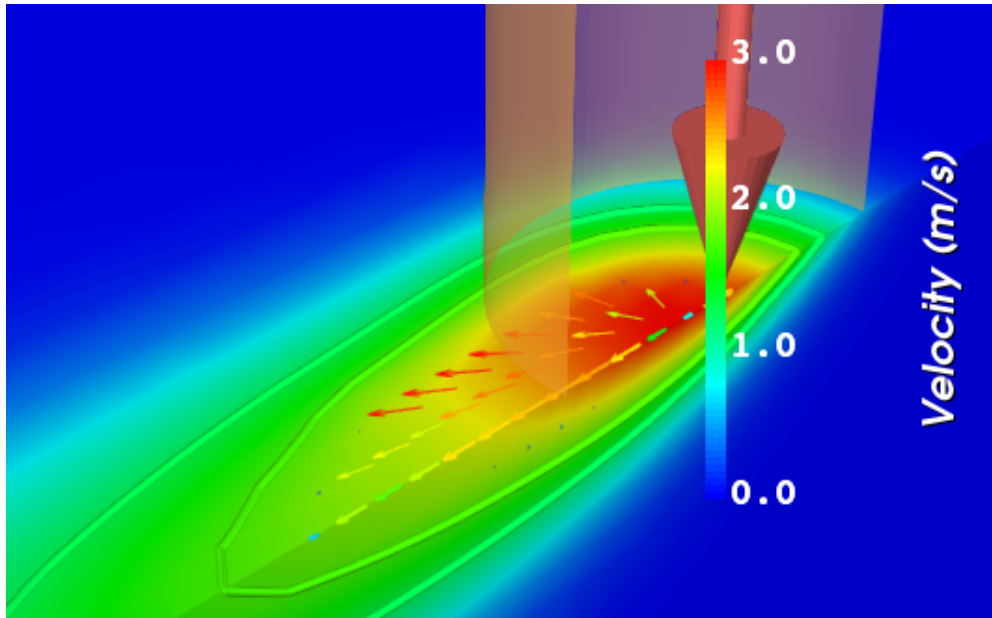
## Arrow

This is an example using "AM\_04: Scheil Transient Steady-state" on page 213 to zoom into one of the AM Calculators, which in this case is renamed to **Steady-state**. On the **Plot Renderer**, and when the *Axis variable* is set to **Flow velocity field**, the **Display** default setting is **Arrow**.



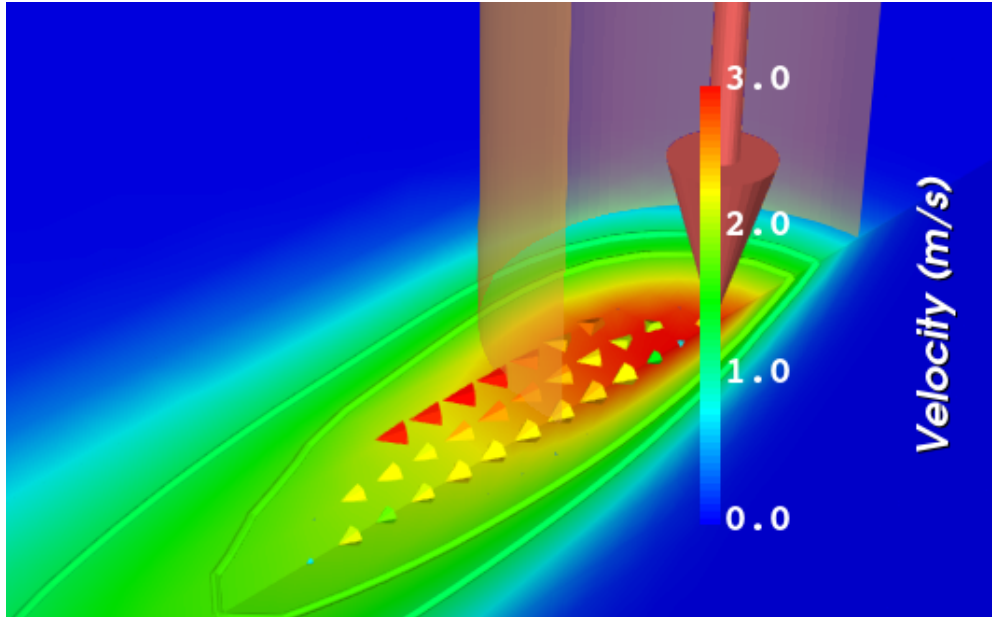
## 2D Glyph

This is an example using "AM\_04: Scheil Transient Steady-state" on page 213 to zoom into one of the AM Calculators, which in this case is renamed to **Steady-state**. On the **Plot Renderer**, and when the *Axis variable* is set to **Flow velocity field**, the **Display** setting is changed from the default to **2D Glyph**.



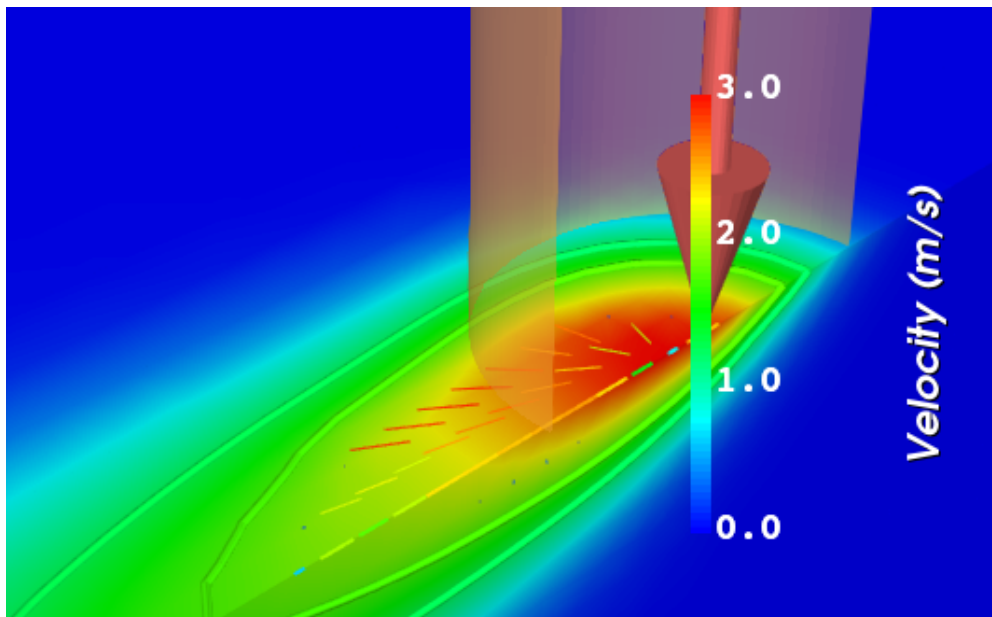
## Cone

This is an example using "AM\_04: Scheil Transient Steady-state" on page 213 to zoom into one of the AM Calculators, which in this case is renamed to **Steady-state**. On the **Plot Renderer**, and when the *Axis variable* is set to **Flow velocity field**, the **Display** setting is changed from the default to **Cone**.



### Line

This is an example using "AM\_04: Scheil Transient Steady-state" on page 213 to zoom into one of the AM Calculators, which in this case is renamed to **Steady-state**. On the **Plot Renderer**, and when the *Axis variable* is set to **Flow velocity field**, the **Display** setting is changed from the default to **Line**.



---

## AM Plot Type: Plot Over Line (2D)

---

The **Plot Over Line** plot type is available with all simulation types (**Steady-state**, **Transient**, and **Transient with heat source from Steady-state**). For Steady-state simulations, it is also available with **Heat Source Calibration**, **Batch**, and **Grid** calculation types.

The **Plot Over Line** 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.

Once the plot is performed, you can also switch between a plot and a **Table View** using the button available on the **Visualizations** window.



The coordinates for the **Plot Over Line Transient with heat source from Steady-state** and its **Steady-state** model are synchronized. This means that if you switch between these computational models, the **Plot Over Line** coordinates show the same result.

### **Example**

This is an example from "[AM\\_01: Transient Simulation of a Single Track](#)" on page 196.

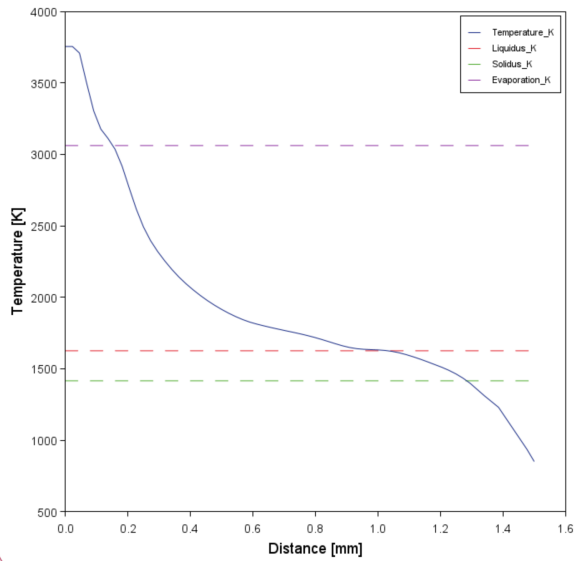


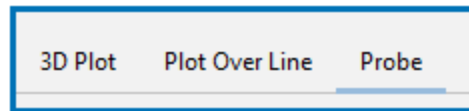
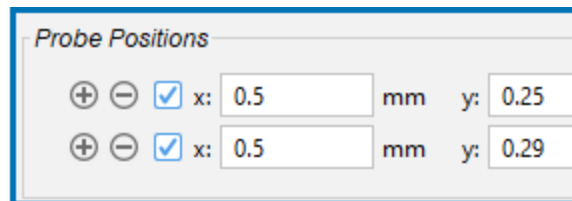
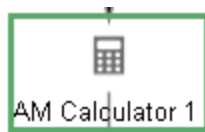
Figure 23: 2D plot over line showing temperature in the tail of the melt pool as a function of distance plotted along the scanning track.

## AM Plot Type: Probe (2D)

The **Probe** plot type is available with the **Transient** and **Transient with heat source from Steady-state** simulation types.

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.

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.



Once the plot is performed, you can also switch between a plot and a **Table View** using the button available on the **Visualizations** window.



Probe data can also be used with the Diffusion Calculator and Precipitation Calculator when these Calculators are added as successors to the AM Calculator.

### Example

The example below is taken from "AM\_01: Transient Simulation of a Single Track" on page 196.

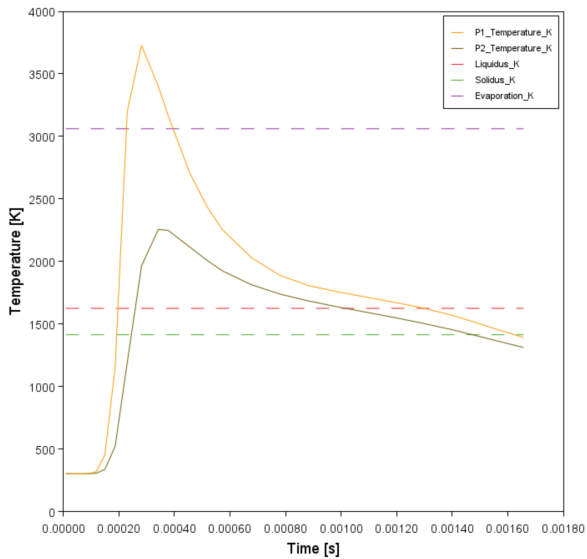


Figure 24: 2D probe plot showing temperature evolution as a function of time at probes P1 and P2.



For use of probe data with the Diffusion Module (DICTRA) see "AM\_05: Using AM Calculator Probe Data with the Diffusion Module (DICTRA)" on page 220.



For use of probe data with the Precipitation Module (TC-PRISMA) see "AM\_12: Using AM Calculator Probe Data with the Precipitation Module (TC-PRISMA)" on page 285.

---

## AM Plot Type: Heat Source Calibration Parameters

---

The **Heat Source Calibration Parameters** plot type is available with a **Steady-state** simulation and when **Heat Source Calibration** is selected as the *Calculation Type*.

During the set up, the **Heat Source Calibration Parameters** type also has its own tab called **Heat Source Calibration** on the **Plot Renderer Configuration** window. For more information see "Visualizing Heat Source Calibrations" on page 178.



"About Heat Source Calibration" on page 39

### Example

The following is from "AM\_06a: Calibrating a Heat Source for a 316L Steel" on page 229.



Also see the companion example, "AM\_06b: Using the Calibrated Heat Source for a 316L Steel" on page 236.

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.

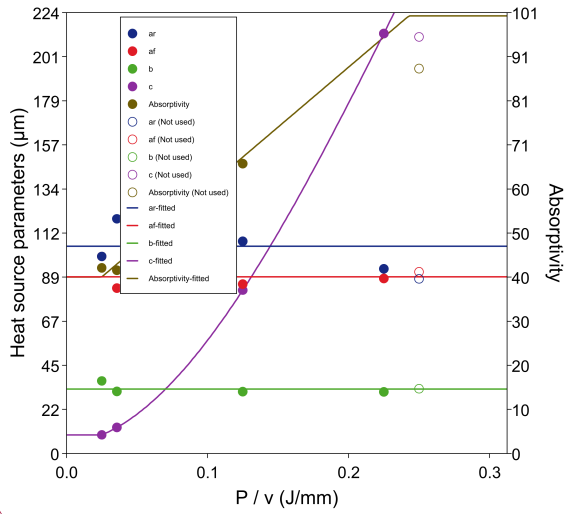


Figure 25: From example AM\_06a, the Heat Source Calibration Parameters plot for the double ellipsoidal calibration.

For instance, you can deselect the points that seem to be outliers or where the error bar in experimental data is large.

## AM Plot Type: Melt Pool Dimensions

The **Melt Pool Dimensions** plot type is available with a **Steady-state** simulation and when **Heat Source Calibration** is selected as the *Calculation Type*. For more information see "[Visualizing Heat Source Calibrations](#)" on page 178.

 "[About Heat Source Calibration](#)" on page 39

### Example

The following is from "[AM\\_06a: Calibrating a Heat Source for a 316L Steel](#)" on page 229.



Also see the companion example, "[AM\\_06b: Using the Calibrated Heat Source for a 316L Steel](#)" on page 236.

A **Melt Pool Dimensions** plot shows 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.

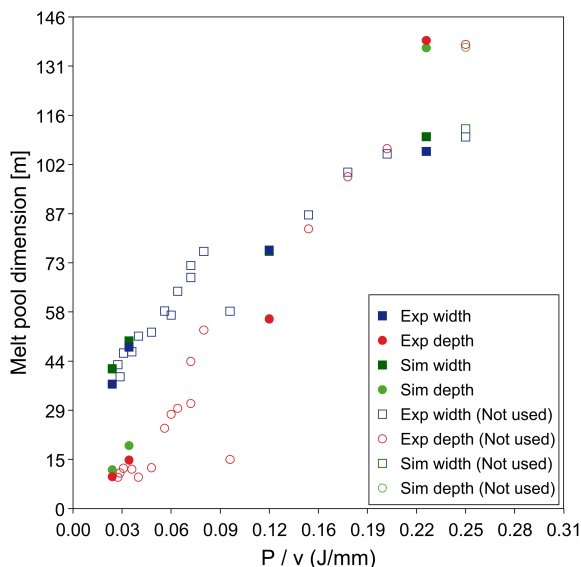


Figure 26: From example *AM\_06a*, the Melt Pool Dimensions plot for the double ellipsoidal calibration.

## AM Plot Type: Printability Map

This topic describes the plot type **Printability Map**, which is available with **Batch** or **Grid** calculations for a *Steady-state* simulation. A printability map is also known as a process map.



This plot is available with **Batch** and **Grid** calculations. However, using it with the **Grid** calculation is the preferred method as it provides full coverage of the plot area. Otherwise, if you are doing a **Batch** calculation and only base it on experimental conditions, it often leaves out the most extreme cases so you get areas in the Printability Map that are blank.

Printability maps allow you to plot the likelihood of three possible defects that occur during additive manufacturing: *Keyholing*, *lack of fusion*, and *balling*.



There is a video tutorial about the **Printability Map** on our [website](#) and on our [YouTube channel](#). It is also included in the Additive Manufacturing Module [YouTube playlist](#).



These examples highlight the use of the printability map: "[AM\\_08a: Grid Calculation for a Ti64 Alloy](#)" on page 247 and "[AM\\_09a: Grid Calculation for an SS316L Alloy](#)" on page 259.

These defects occur based on the speed and power used during the AM process. Printability maps allow you to reduce the risks of these defects by showing the speed and power settings that are optimal, which in turn allows you to calibrate your system to avoid these issues.

- *Keyholing* occurs with low scanning speed and high power.
- *Lack of fusion* occurs with high scanning speed and low power.
- *Balling* occurs if you increase the scanning speed.

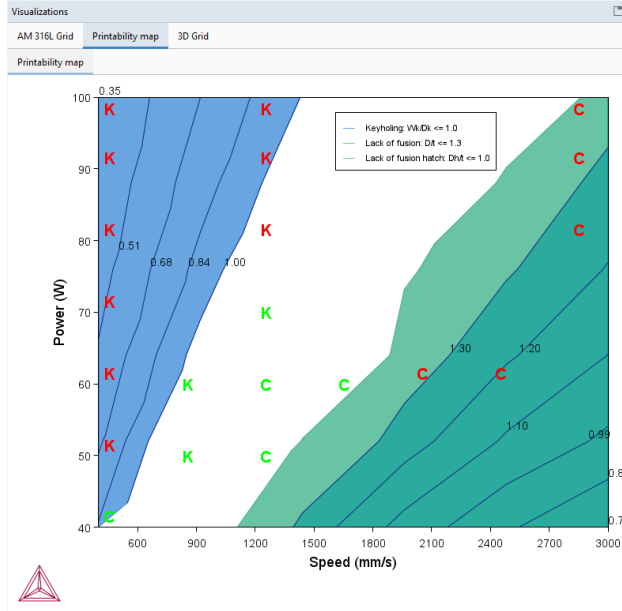


Figure 27: Printability map for 316L showing regions of keyholing and lack-of-fusion. Experimental information from Hu et. al. [2019Hu] overlaid as coloured labels showing keyhole porosity (**K**), conduction mode with lack-of-fusion porosity (**C**). Green labels showing experiments without defects for conduction mode (**C**) and keyhole mode (**K**).

The defect regions in the printability maps are defined by functions of the keyhole dimension ( $W_k$ ,  $D_k$ ), melt pool dimensions ( $W$ ,  $D$ ,  $L$ ), hatch distance ( $h$ ), and powder thickness ( $t$ ) [2019Joh].

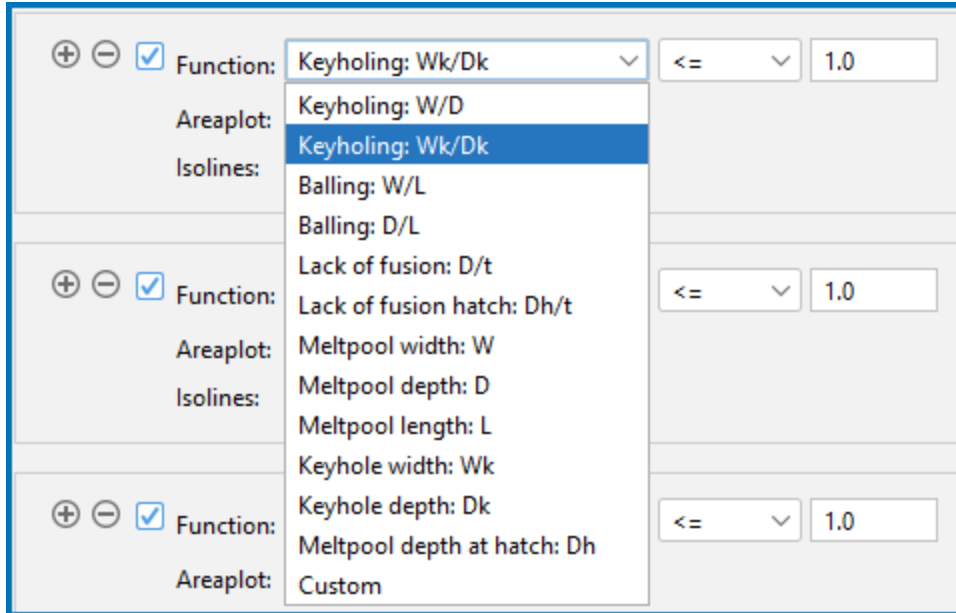


Figure 28: When defining the Printability map on the Plot Renderer for an AM Calculator, you can select from a variety of Functions.

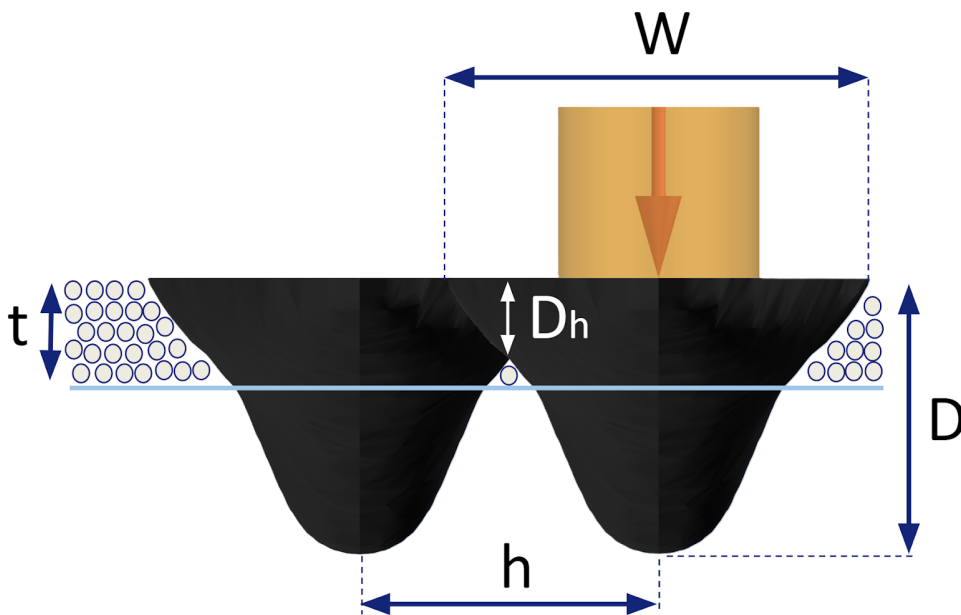


Figure 29: Two single tracks with the melt pool width ( $W$ ) and depth ( $D$ ). The hatch distance ( $h$ ), powder thickness ( $t$ ), and the melt pool depth at half of the hatch distance ( $D_h$ ).

## Keyholing Porosity

Keyholing is a critical feature in additive manufacturing that occurs at high energy densities ( $P/v$ ). Keyholing is not only negative, the increased depth of the melt pool can increase productivity. However, deep and narrow keyholes are unstable and can collapse, and lead to keyholing porosity. Keyhole porosity appears close to spherical in shape.

Keyholing porosity can be defined to occur if the width/depth ratio of the keyhole is less than or equal to 1 (i.e.  $W_k/D_k \leq 1$ ). Similarly if a volume heat source is used (that lacks information of the keyhole geometry) the ratio of the melt pool ( $W/D$ ) can be used instead, where  $W/D \leq 1$ .

## Lack of Fusion

Lack of fusion porosity occurs if the melt pool, due to low energy density ( $P/v$ ), fails to penetrate the powder thickness. Lack of fusion gives porosity uneven in shape, compared to keyhole porosity that is close to spherical in shape.

Lack of fusion for a single track can be defined to occur if the melt pool depth is smaller than the powder thickness, i.e.  $D/t < 1.0$ . Additionally the lack of fusion between two tracks, at half of the hatch distance, can be defined to occur if  $D_h/t < 1.0$ .

## Balling

The scan tracks can break up and produce agglomerated particles in spherical shape instead of a single bead. Balling has been defined to occur if ratios  $W/L$  [2019Joh] or  $D/L$  [2023Kat] are small .

Additional user-defined defect criteria can be defined with the Custom function using any combination of the keyhole dimension ( $W_k, D_k$ ), melt pool dimensions ( $W, D, L$ ), hatch distance ( $h$ ) and powder thickness ( $t$ ).

## References

[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, 1–9 (2019).

[2019Joh] L. Johnson, M. Mahmoudi, B. Zhang, R. Seede, X. Huang, J. T. Maier, H. J. Maier, I. Karaman, A. Elwany, R. Arróyave, Assessing printability maps in additive manufacturing of metal alloys. *Acta Mater.* 176, 199–210 (2019).

[2023Kat] J. Katagiri, M. Kusano, S. Minamoto, H. Kitano, K. Daimaru, M. Tsujii, M. Watanabe, Melt Pool Shape Evaluation by Single-Track Experiments and Finite-Element Thermal Analysis: Balling and Lack of Fusion Criteria for Generating Process Window of Inconel738LC. Materials (Basel). 16, 1729 (2023).

## AM Plot Type: Parity Plot

This topic describes the plot type **Parity Plot** which is available with Batch calculations for a Steady-state simulation. This plot type is only to select on the Plot Renderer when the simulation uses the **Batch Calculation Type** with an **AM Calculator**.



These examples highlight the use of this plot: "AM\_07: Batch Calculations for an IN718 Alloy" on page 241, "AM\_08b: Batch Calculations for a Ti64 Alloy" on page 253 and "AM\_09b: Batch Calculations for an SS316L Alloy" on page 264.

Parity plots are used when validating the Steady-state Batch calculation against experimentally measured melt pool width and depth. The plot shows the experimental width/depth on the X-axis and the calculated width/depth on the Y-axis. The data should follow the diagonal line for a perfect match between the calculated and experimental values.

The following are from example AM\_07.

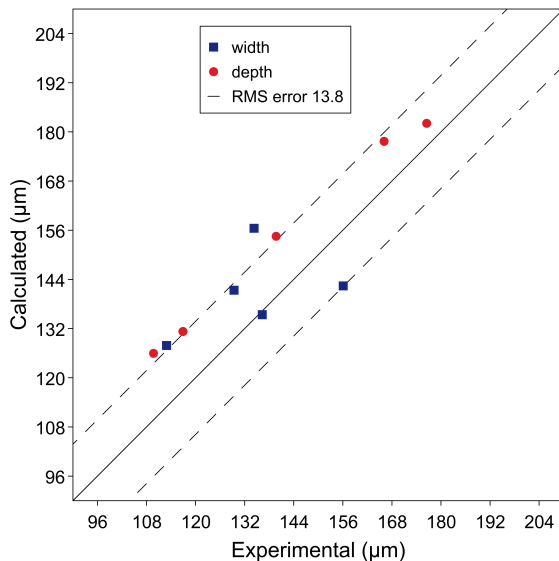


Figure 30: Parity plot comparing experimental versus calculated melt pool width and depth for all the experiments. The experiments are single tracks on bare plate IN718 with varied power and scan speed. The Root Mean Square (RMS) error can also be shown as a dashed line.

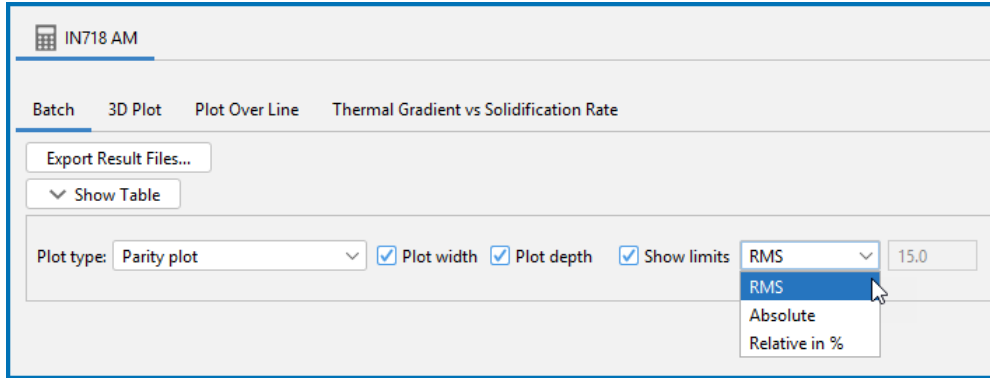


Figure 31: Alternatively, lines for user-defined Absolute or Relative in % error can be shown instead by selecting these options on the Configuration window.

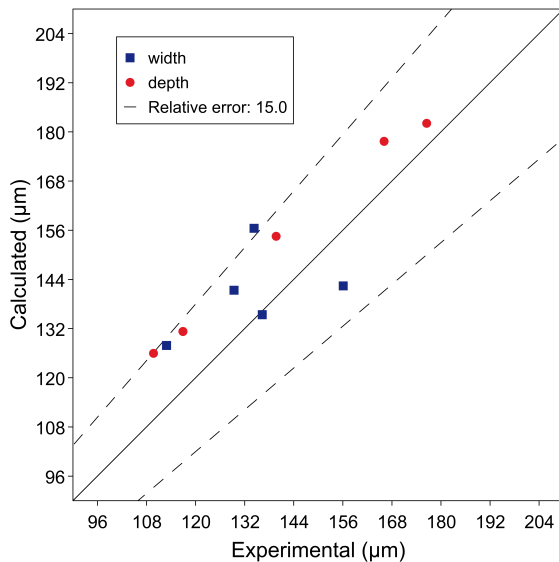


Figure 32: Parity plot comparing experimental versus calculated melt pool width and depth for all the experiments. The experiments are single tracks on bare plate IN718 with varied power and scan speed. In this plot, it is configured to show a Relative in % error of 5 instead of the RMS error.

## Keyhole Model with Fluid Flow

This is an example application of using the parity plot when the keyhole model includes fluid flow, which can provide a better prediction of the melt pool size.



"About the Keyhole Model" on page 43

Keyholing can occur with low scanning speed and high power. Including fluid flow in the simulation makes the predictions of the entire melt pool size more accurate, which is especially important at higher energy densities when Marangoni forces are strong.

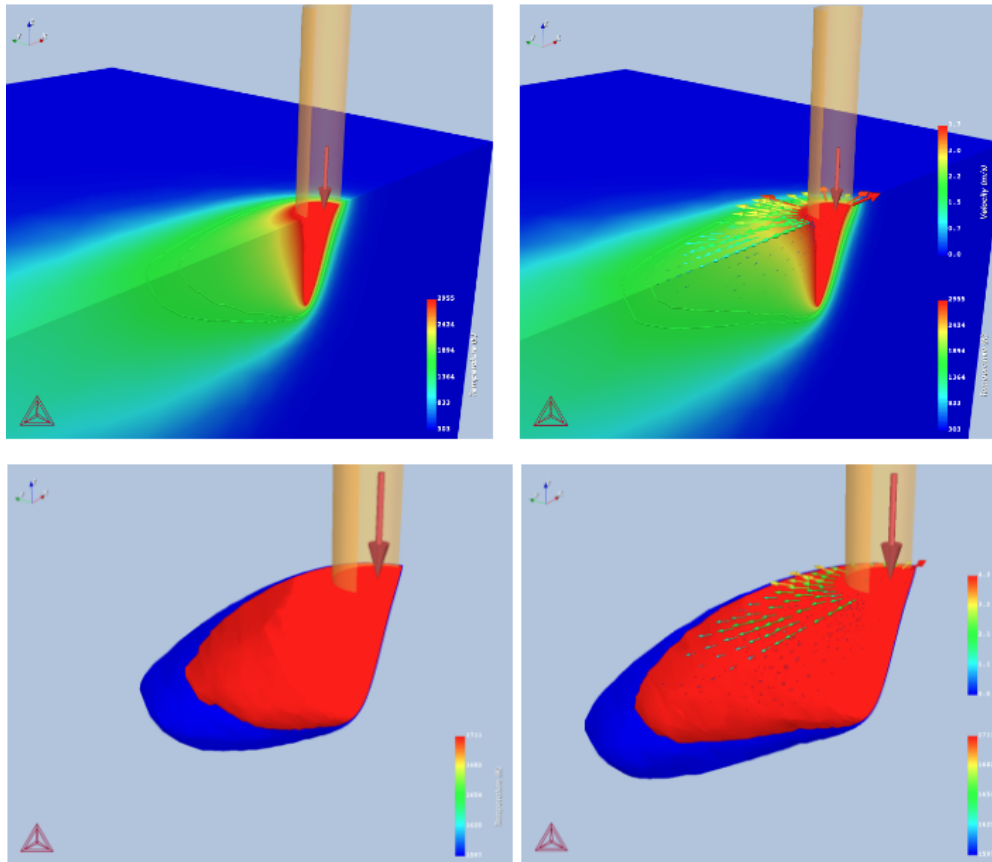


Figure 33: Fluid flow can make the melt pool wider and longer. This example shows the effect when enabling fluid flow for 316L for power 50 W and scan speed 400 mm/s. Left images show without fluid flow and right images with fluid flow enabled. Top row shows the temperature color map. Bottom row shows the isosurfaces for liquidus and solidus, i.e. the size of the melt pool and mushy zone.

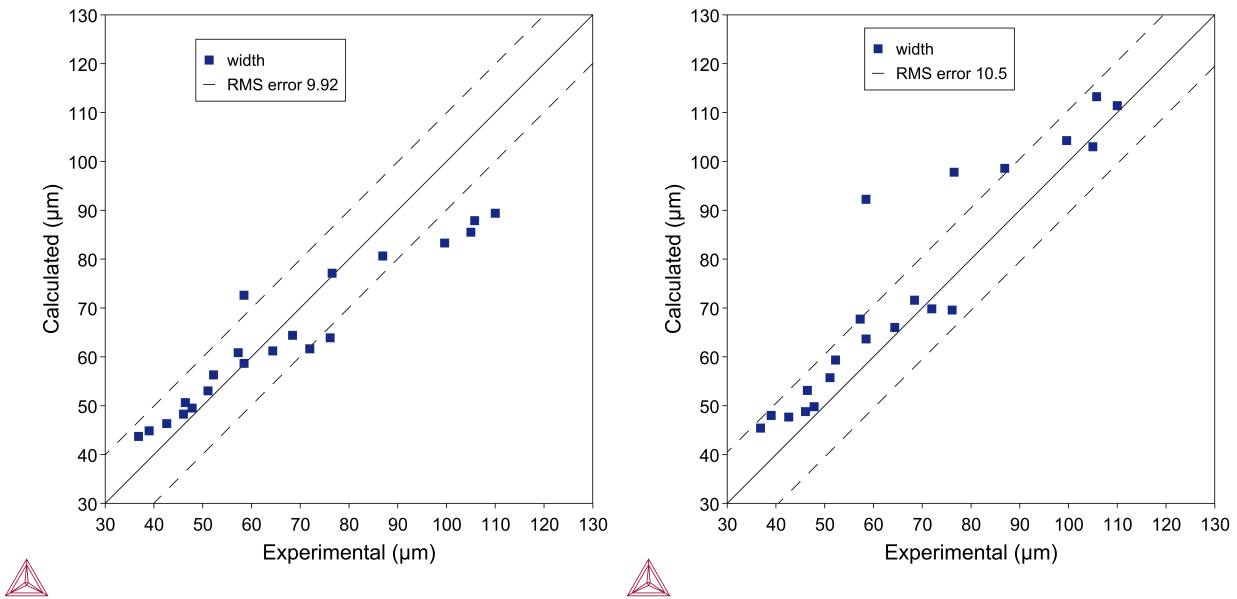


Figure 34: Parity plot for 316L showing how the RMS is reduced for the melt pool width when excluding fluid flow (left) compared to when it is included (right). Experiments from Hu et. al [2019Hu].

## 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, 1–9 (2019).

## AM Plot Type: Melt Pool vs Energy Density

This topic describes the plot type **Melt Pool vs Energy Density** which is available with **Batch** or **Grid** calculations for a *Steady-state* simulation.



These examples highlight the use of this plot: "AM\_08b: Batch Calculations for a Ti64 Alloy" on page 253 and "AM\_09b: Batch Calculations for an SS316L Alloy" on page 264.

This plot type is used when validating the Steady state Batch calculation against experimentally measured melt pool width and depth. The plot shows the experimental and simulated melt pool dimensions as a function of the energy density ( $P/v$ ). The plot type can also be used when experimental melt pool dimension are lacking, e.g. after the Grid calculation, but then naturally only shows the simulated melt pool dimensions as a function of the energy density ( $P/v$ ).

This uses example AM\_09b.

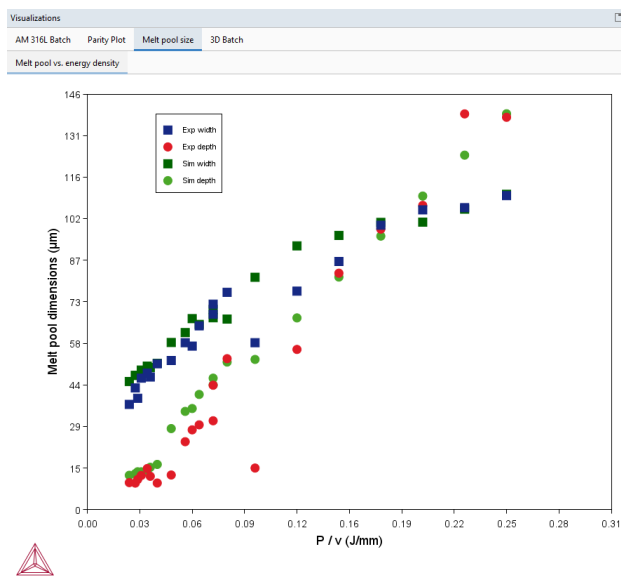


Figure 35: Plot comparing experimental versus calculated melt pool dimensions for all the Batch calculations. The experiments are single tracks with SS316L with varied power and scan speed. The melt pool width and depth are shown on the Y-axis and the energy density ( $P/v$ ) on the X-axis.

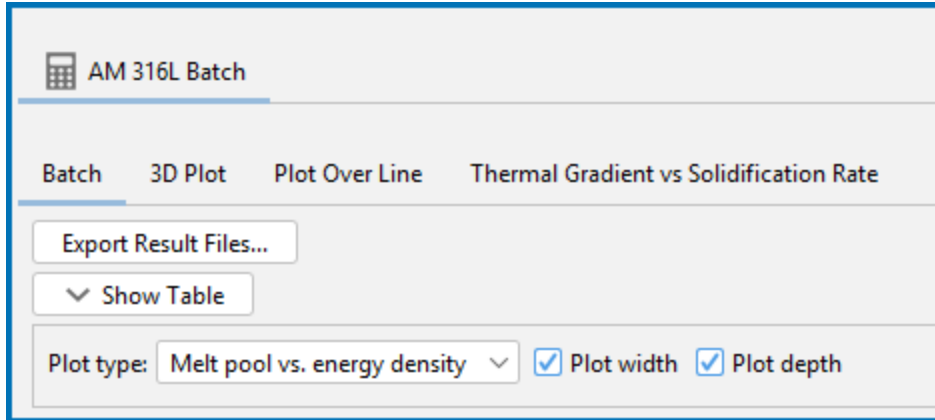


Figure 36: For the plot, you can choose what to include on the plot, to show width and/or depth by selecting the relevant checkboxes on the Configuration window.

## AM Plot Type: Thermal Gradient vs. Solidification Rate

This topic describes 2D and 3D versions of plots comparing *thermal gradient* to *solidification rate* along the liquid isotherm. Both the 2D and 3D versions are available for steady-state and transient simulations.

- The 2D version is accessed on the **Plot Renderer Configuration** window as a tab named **Thermal Gradient vs Solidification Rate**. It is visualized on a tab of the same name on the **Visualizations** window.
- The 3D version is available on the **3D Plot** tab on the **Configuration** (settings) window. It is further visualized on two **3D Plot** tabs on the **Visualizations** window.



"AM\_10: CET Transition in an IN718 Alloy" on page 270 includes an example of this plot in both 2D and 3D.

### 2D Plot

The 2D version of this plot is available on the **Thermal Gradient vs Solidification Rate** tabs on both the **Configuration** (settings) and **Visualizations** window.




You do not need to configure anything on the **AM Calculator** to access the 2D plot.

On the **Thermal Gradient vs Solidification Rate** tab on **Plot Renderer Configuration** window you can adjust the settings for the **Solidification rate** (X-axis) and **Thermal gradient** (Y-axis).

The screenshot shows the **Plot Renderer Configuration** window with the **Thermal Gradient vs Solidification Rate** tab selected. The settings are as follows:

- 3D Plot** (selected)
- Plot Over Line** (selected)
- Thermal Gradient vs Solidification Rate** (selected)
- Probe** (selected)
- Filter by material types:** Show all (dropdown)
- X: Solidification rate [m/s]** (highlighted)
- Axis type:** Linear (dropdown)
- Limits:** 0.0 to 1.0 step 0.1 (checkbox checked)  **Automatic scaling**
- Y: Thermal gradient [K/m]** (highlighted)
- Axis type:** Linear (dropdown)
- Limits:** 0.0 to 1.0 step 0.1 (checkbox checked)  **Automatic scaling**

As you adjust settings, watch it dynamically change on the **Visualizations** window. The plot is on the tab of the same name (**Thermal Gradient vs Solidification Rate**). You can switch between the plot and a table format using the **Table View**  button.

 "AM Plots: Table View Options" on page 98

This uses example AM\_10.

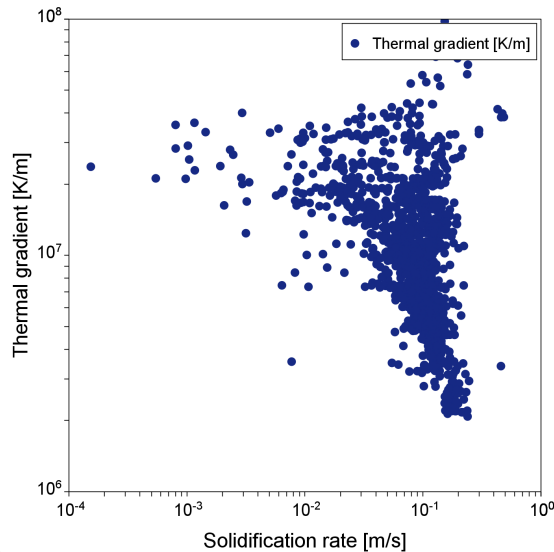


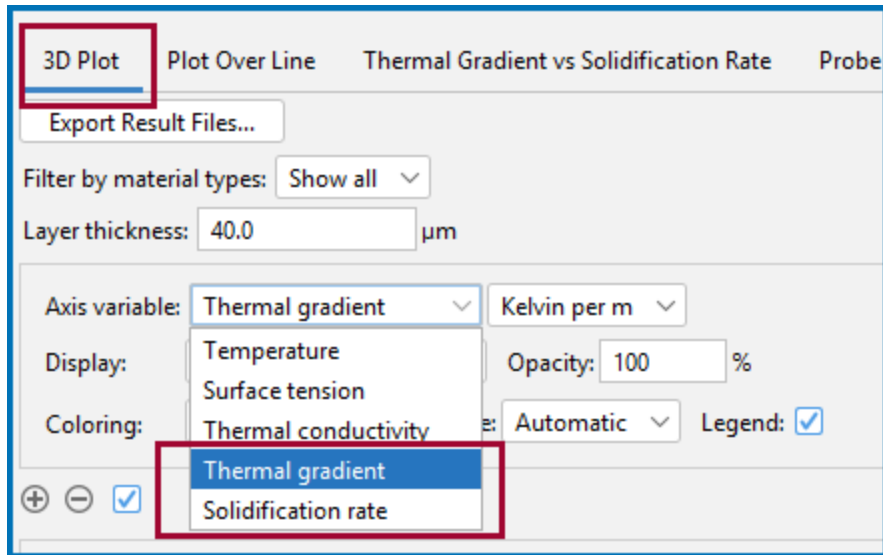


Figure 37: The solidification rate vs thermal gradient is shown on the AM Thermal Gradient & Solidification Rate Plot Renderer.

## 3D Plot

The 3D version of this plot is available on the **3D Plot** tab on the **Configuration** (settings) window. You choose **Solidification rate** and **Thermal gradient** as the **Axis variable**. Use the  and  buttons as usual to add and remove quantities.



On the **Visualizations** window the comparisons for the different calculation types and variable combinations are plotted on two **3D Plot** tabs.

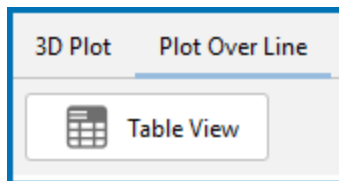
## AM Plots: Table View Options

There are ways to view AM plot results as a table on either the **Configuration** or **Visualization** windows for various plots and calculation types.

### 2D Plots

For 2D plots (**Plot Over Line**, **Probe**, and **Thermal Gradient vs Solidification Rate**), 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.

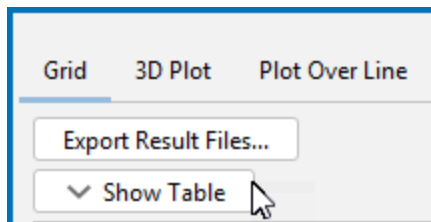
These plots are available with all calculation types (*Steady-state*, *Transient*, and *Transient with heat source from Steady-state*) simulations.



### Batch and Grid Plots

For *Steady-state* **Batch** and **Grid** calculations, the available plots **Printability map**, **Meltpool vs energy density**, and **Parity plot**, can also be viewed as a table.

Once the plot is performed, you can toggle to between **Show Table** and **Hide Table** on the **Configuration** window.



#	Power (W)	Speed (mm/s)
1	40.0	3000.0
2	40.0	2480.0

# Additive Manufacturing Workflow

---

In this section:

Setting Up the Additive Manufacturing Simulation .....	100
Additive Manufacturing Templates .....	105

---

# 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 property 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 different *Calculation Type* options such as a *Heat Source Calibration* to calibrate a heat source based on the provided experimental data instead of using just a *Single point* calculation. *Batch* and *Grid* types are also available.
  - c. For *Steady-state* or *Transient with heat source from Steady-state* simulations plus a Gaussian, Core-ring, Top-hat, or Electron Beam heat source, choose to include a *keyhole model* with or without *fluid flow*.
  - d. Set the simulation conditions.
4. Run the simulation and visualize the results (Workflow 1 or 2) based on the settings, for example:
  - a. *3D Plot*
  - b. *Plot Over Line* or *Thermal Gradient vs Solidification Rate (2D)*. Once the plot is performed you can also switch to view it in a table.

- 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). Once the plot is performed you can also switch to view it in a table.
- d. *Heat Source Calibration Parameters* and *Melt Pool Dimensions*: These plots are available for Heat Source Calibration calculations.
- e. *Printability Map*, *Parity Plot*, and *Melt Pool vs Energy Density*: These plots are available for Batch and Grid calculations. This can also be viewed as a table.

## 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 (Workflow 1)***

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.

## ***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 need to change these settings yourself. If you are working with different materials, different settings might be more suitable to your material.

## ***Materials Properties Library (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" on page 10

## AM Calculator (Workflow 1 or 2)

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.



Transient simulations currently support only cuboid geometries. 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" on page 108, "AM Calculator Conditions Settings" on page 111, and "AM Calculator Heat Source Settings" on page 123

### 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" on page 164

## ***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.



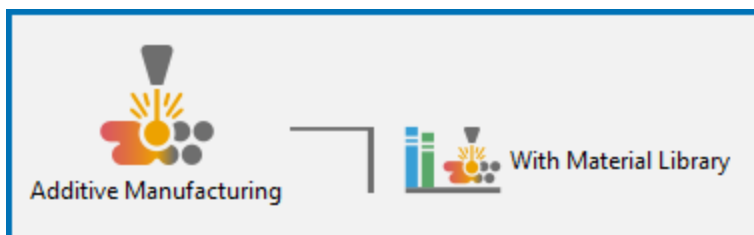
You can also add nodes individually to build your own tree.



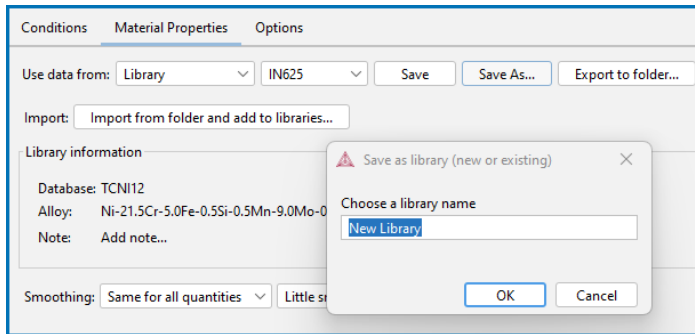
When you also have licenses either for the Diffusion Module (DICTRA) and/or the Precipitation Module (TC-PRISMA), for transient simulations you can additionally include probe data from the AM Calculator in the set up of the thermal profile for diffusion and precipitation calculations, respectively.

### 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.

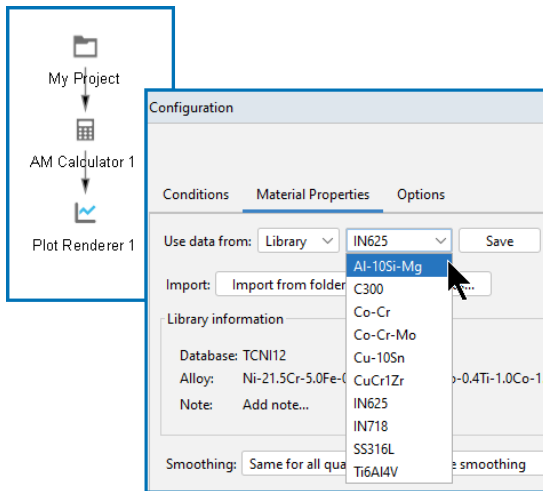


There is also an option on the AM Calculator to save the material properties in a material library, and use these later for a new AM Calculator where you want to make calculations related to the same alloy.



## 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 .....	108
AM Calculator Conditions Settings .....	111
AM Calculator Heat Source Settings .....	123
AM Calculator Experiment File Requirements .....	134
AM Calculator Materials Properties Settings .....	139
AM Calculator Options Settings .....	143
AM Calculator: Working with Probe Data .....	145
AM Calculator: Plot Renderer Settings .....	147
Working with AM Visualizations .....	164
AM Visualizations Window Plot Toolbar .....	170
Menu or Keyboard Options: Geometry and 3D Plots .....	173
Rotate, Zoom, and Pan 3D Plots: AM Calculations .....	176
Overlaying Plots in the AM Module .....	177
Visualizing Heat Source Calibrations .....	178
Visualizing Batch Calculations in the AM Module .....	188

## 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](#)" on page 105 and "[Setting Up the Additive Manufacturing Simulation](#)" on page 100.

### Configuration Window

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](#)" on page 59. Also review the "[Additive Manufacturing Module Theory](#)" on page 19 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 following *Calculation Types*:

- **Single Point**, which is the standard steady-state calculation for a single set of processing parameters.
  - **Heat Source Calibration** to calibrate a heat source based on the imported or entered experimental data instead of performing a single point calculation. This is available with **Gaussian**, **Double ellipsoidal**, or **Conical** heat sources.
  - **Batch** to include power and scan speed data, which is read from a file such as a spreadsheet. You can optionally include experimental melt pool width and depth (if there is data), and compare this to the calculated results.
  - **Grid** to evaluate two axis variables of power and scanning speeds in the specified range and number of steps.
- 
- **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 Gaussian, Core-ring, Top-hat, or Electron Beam heat sources (*Steady-state* or *Transient with heat source from Steady-state*) you can also include a **Keyhole model**. This is available with or without fluid flow, or if you use separate material properties for powder.

There are several settings and these are described separately.

- ["AM Calculator Conditions Settings"](#) on page 111
- ["AM Calculator Heat Source Settings"](#) on page 123
- ["AM Calculator Materials Properties Settings"](#) on page 139
- ["AM Calculator Options Settings"](#) on page 143

### ***Plot Renderer as a Successor***

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 ["About the AM Plot Types"](#) on page 64 and ["AM Calculator: Plot Renderer Settings"](#) on page 147 as starting points.



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. There are several related topics to visualize results as well as when you are setting up the simulations. See ["Working with AM Visualizations"](#) on page 164 as a starting point to learn more.

# AM Calculator Conditions Settings

Once you have added an "AM Calculator" on page 108, 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" on page 139, and "AM Calculator Options Settings" on page 143



"Additive Manufacturing Module Theory" on page 19

## 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




Also see the theory section, "[Fluid Flow](#)" on page 24 and if also using the keyhole model, see "[About the Keyhole Model](#)" on page 43.

The use of the **Fluid flow including Marangoni effect** checkbox in combination with separate material properties for powder and the keyhole model is dependent on the simulation type.

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.

### **FEATURE CHECKBOX AVAILABILITY: FLUID FLOW, SEPARATE MATERIAL PROPERTIES WITH POWDER, AND KEYHOLE MODEL**

Depending on what you want to simulate, there are some setting combinations that are available based on the different simulation types. The details are relevant to using the **Fluid flow including Marangoni effect**, **Use separate material properties for powder**, and **Use keyhole model** checkboxes in a variety of combinations. The software automatically guides your choices. However, it can be useful to know in advance what is available to better design your simulation.

<i>Simulation Type</i>	<i>Feature (checkbox) Availability</i>		
	<i>Use Separate Material Properties for Powder</i>	<i>Fluid Flow including Marangoni Effect</i>	<i>Use Keyhole Model (Heat Source setting)</i>
Steady-state	available	available	available
Transient (with fluid flow)	not available	available	not available
Transient (without fluid flow)	available	not available	not available
Transient with heat source from Steady-state (Single-track scanning strategy)	available	available	available
Transient with heat source from Steady-state (Bidirectional or Unidirectional scanning strategies).	not available   <div style="border: 2px solid blue; padding: 10px; margin: 10px 0;"> <p>In this case, you can select the checkbox but the <b>Perform</b> button is unavailable.</p> </div>	available	available

## Use Separate Material Properties for Powder


The use of the **Use separate material properties for powder** checkbox in combination with fluid flow and the keyhole model is dependent on the simulation type.

The **Use separate material properties for powder** checkbox is selected by default to use different density and thermal conductivity for the powder layer; otherwise by default the same properties as for the solid are used. Enter a **Powder density** as a percentage (0-100%) of the solid material. The default is 80 % of the solid material.

### FEATURE CHECKBOX AVAILABILITY: FLUID FLOW, SEPARATE MATERIAL PROPERTIES WITH POWDER, AND KEYHOLE MODEL

Depending on what you want to simulate, there are some setting combinations that are available based on the different simulation types. The details are relevant to using the **Fluid flow including Marangoni effect**, **Use separate material properties for powder**, and **Use keyhole model** checkboxes in a variety of combinations. The software automatically guides your choices. However, it can be useful to know in advance what is available to better design your simulation.

Simulation Type	Feature (checkbox) Availability		
	Use Separate Material Properties for Powder	Fluid Flow including Marangoni Effect	Use Keyhole Model (Heat Source setting)
Steady-state	available	available	available
Transient (with fluid flow)	not available	available	not available
Transient (without fluid flow)	available	not available	not available
Transient with heat source from Steady-state (Single-track scanning strategy)	available	available	available
Transient with heat source from Steady-state (Bidirectional or Unidirectional scanning strategies).	not available	available	available

Simulation Type	Feature (checkbox) Availability		
	Use Separate Material Properties for Powder	Fluid Flow including Marangoni Effect	Use Keyhole Model (Heat Source setting)
	<div style="border: 2px solid blue; padding: 10px; display: inline-block;">  <p>In this case, you can select the checkbox but the <b>Perform</b> button is unavailable.</p> </div>		

## Geometry

For a **Steady-state** simulation you only need to enter the height. 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 you 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 settings are described in a separate topic: "[AM Calculator Heat Source Settings](#)" on page 123. The theory is also described in "[About the Heat Source Models](#)" on page 29.

## 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*

Enter a **Scanning speed**, which is the velocity of the moving heat source. The default is 1000 mm/s.

### *Layer Thickness*

Enter a **Layer thickness**, which is the thickness of the powder layer. The default is 40 mm. When you change the layer thickness, the meshing size changes too.

### *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. This field is available when more than 1 is entered in the **Number of layers** field.
- **Cooling time:** Enter a cooling time (unit = s) This is the time for which the material is left to cool down after the scanning is completed. During cooling time, the heat source is completely turned off.

### 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.
- **Cooling time:** Enter a cooling time (unit = s) This is the time for which the material is left to cool down after the scanning is completed. During cooling time, the heat source is completely turned off.

## Top Boundary Conditions



"Numerical Boundary Conditions" on page 25 in the theory section.

---

## **Radiation Emissivity**

Enter a **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**

Enter a **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. The default is 20 W/m<sup>2</sup>.

## **Evaporation**

Select the **Evaporation** 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.

## **Calculation Type (Steady-state)**

### **Single Point**

By default for **Steady-state** calculations, **Single point** is selected, which by default uses the **Gaussian Heat Source**. In addition to settings described in this topic, you can also find more details in "[AM Calculator Heat Source Settings](#)" on page 123. This is available with all heat sources.

## Heat Source Calibration



For some background, see ["About Heat Source Calibration"](#) on page 39. Also see ["Visualizing Heat Source Calibrations"](#) on page 178 for various options related to visualizing the calibration settings.


Select **Heat Source Calibration** to calibrate a heat source based on the imported or entered experimental data instead of performing a single point calculation. All types of heat sources can be used. See below for additional information to enter the *Experiment Data*. This is available with **Gaussian**, **Double ellipsoidal**, or **Conical** heat sources.


### Experiment Data (Heat Source Calibration)




It is important to prepare the experiment data file correctly so the simulation performs smoothly. See ["AM Calculator Experiment File Requirements"](#) on page 134.

This section is available when **Heat Source Calibration** is selected as the *Calculation Type*.

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.

## Batch

Select **Batch** to include power and scan speed data, which is read from a file such as a spreadsheet and entered or imported to the *Batch Experiment Data* table. You can optionally include experimental melt pool width and depth (if there is data), and compare this to the calculated results using a **Parity plot** on the Plot Renderer.

This is available with all heat sources.




In addition to settings described in this topic, you can also find more details in "[AM Calculator Heat Source Settings](#)" on page 123 and "[Visualizing Batch Calculations in the AM Module](#)" on page 188.


## Batch Experiment Data




It is important to prepare the experiment data file correctly so the simulation performs smoothly. See "[AM Calculator Experiment File Requirements](#)" on page 134.

This section is available when **Batch** is selected as the *Calculation Type*.

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.

## Grid

Select **Grid** to evaluate two axis variables of power and scanning speeds in the specified range and number of steps. Then continue with the *Grid Definitions*. In addition to settings described in this topic, you can also find more details in "AM Calculator Heat Source Settings" on [page 123](#).

This is available with all heat sources.

## Grid Definitions

This section is available when **Grid** is selected as the *Calculation Type*.

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.

Enter a **Min**, **Max**, and **Number of steps** for:



- **Power (W)** for the selected *Heat Source* model.
- **Scanning speed (mm/s)** for the velocity of the moving heat source.

## 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.

Some advance options can be selected from a popup menu;

- Double-click on the geometry to pick a coordinate
- Hold Ctrl key to pick a probe in the middle of a layer
- Hold Shift key to map the picking position with a scanning path, i.e. you can pick a probe that snaps to a nearest scanning path
- Hold Shift+Alt keys to map the picking position to the middle of two scanning paths, i.e. pick a probe in the middle of the two nearest scanning paths.



["AM Calculator: Working with Probe Data" on page 145](#)

### ***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.

## AM Calculator Heat Source Settings

These settings are found on the AM Calculator Conditions tab. The other settings are described in "AM Calculator Conditions Settings" on page 111.

Choose the heat source model: **Gaussian**, **Core-ring**, **Top-hat**, **Electron Beam**, **Double ellipsoidal**, or **Conical**. When available you can also choose predefined *Users heat sources* from the list.



For background theory related to Heat Sources, see "About the Heat Source Models" on page 29 and "About the Absorptivity Model" on page 47. For *Gaussian*, *Double ellipsoidal*, and *Conical* heat sources, also see "About Heat Source Calibration" on page 39 for additional details.



When a **Users heat source** option is available to choose in the heat source list, then only the **Power** of the heat source needs 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.



The **Core-ring**, **Top-hat**, and **Electron Beam** heat sources are not available with **Heat Source Calibration** calculations.

### Gaussian Heat Source

**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.

- Settings below: Power, Absorptivity (incl prefactor, Wave length), Beam radius, Use keyhole model, Beam quality factor  $M^2$ .



For specific theory see [Gaussian Heat Source](#).

## Core-ring Heat Source

A **Core-ring** heat source, like a Gaussian heat source, is a surface heat source model which means that the power input from the heat source is only defined on the top surface of the workpiece, as a Neumann boundary condition. The intensity of a core-ring heat source can be modeled as a combination of a central Gaussian core beam and an outer ring beam.


- Settings below: Power, Absorptivity (incl prefactor, Wave length), Beam radius-Core, Beam radius-ring, Ring radius, Amount of power in ring, Use keyhole model, Beam quality factor  $M^2$ , Mode (Index or Custom).

 For specific theory see [Core-ring Heat Source](#).

## Top-hat Heat Source

A **Top-hat** heat source is also called a *flat-top* and is considered a surface heat source model. In the AM Module, the top-hat heat source is approximated with a super-Gaussian beam, which means that the intensity of the beam at the edges decays softly but more steeply than a Gaussian beam, resulting in a flatter peak and sharper edges.

- Settings below: Power, Absorptivity (incl prefactor, Wave length), Beam radius, Use keyhole model, Beam quality factor  $M^2$ , Mode (Index or Custom).

 For specific theory see [Top-hat Heat Source](#).

## Electron Beam Heat Source

The heat source for the **Electron Beam** melting process is modeled as a Gaussian heat source. The heat is applied on the top surface of the domain, which is then diffused into the material based on the diffusivity of the material and absorptivity of the electron beam.

- Settings below: Power, Acceleration voltage, Absorptivity (incl prefactor), Beam radius, Use keyhole model.

 For specific theory see [Electron Beam Heat Source](#).

## Double Ellipsoidal Heat Source

The **Double ellipsoidal** heat source, also referred to as Goldak, is a volume 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.

- Settings below: Power, Absorptivity (% only), ar, af, b, c.

 For specific theory see [Double Ellipsoidal \(Goldak\) Heat Source](#).

## Conical Heat Source

The **Conical** option 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.

- Settings below: Power, Absorptivity (% only), re, ri, H.

 For specific theory see [Conical Heat Source](#).

## Settings

### POWER

For a **Single Point** calculation, enter the **Power**. The default is 200 W.

### ABSORPTIVITY (GAUSSIAN, CORE-RING, TOP-HAT, ELECTRON BEAM)

The **Absorptivity** settings are based on the *Heat Source* and *Calculation Type* selected.

For a **Single Point**, **Batch**, or **Grid Calculation Type**:

- **Gaussian, Core-Ring or Top-Hat:**
  - Select **Calculated** (the default) to use the alloy's liquid resistivity to calculate the absorptivity as a function of temperature. To adjust this calculation, enter a different prefactor than the default (1.0) in the **with prefactor** field.
  - Choose **User-defined** to enter absorptivity in % where you also can enter this as a function of temperature (T) i.e.  $-1.6e-7 * T * T + 2.5e-3 * T + 31$ .
- **Electron Beam:**
  - Select **Calculated** (the default) to evaluate a back-scattering coefficient ( $\eta$ ) for the given alloy chemistry as a function of acceleration energy and angle of incidence. Absorptivity is then given as  $(1 - \eta) * 100$ .

- To adjust this calculation, enter a different prefactor than the default (1.0) in the **with prefactor** field.
- Choose **User-defined** to enter absorptivity in %.

For a **Heat Source Calibration** *Calculation Type* the absorptivity is calibrated separately for each row in the **Experiment Data** table. Select an option to calibrate absorptivity:

- **Calculated (prefactor calibrated)** (the default) to use the calculated absorptivity and calibrate the prefactor.
- **Constant (calibrated)** to calibrate the absolute value of absorptivity.



"About the Absorptivity Model" on page 47 and the section "Heat Source Calibration" on page 56 for more.

### **ABSORPTIVITY (DOUBLE ELLIPSOIDAL, CONICAL)**

The **Absorptivity** settings are based on the *Heat Source* and *Calculation Type* selected.

For this heat source and with a **Single Point**, **Batch**, or **Grid** *Calculation Type* enter the absorptivity % for the selected heat source model. The default is 60 %. For **Heat Source Calibration** calculations there are no further settings.

### **ACCELERATION VOLTAGE (ELECTRON BEAM)**

Enter an **Acceleration voltage** of the electron beam in kV.

### **WAVE LENGTH (GAUSSIAN, CORE-RING, TOP-HAT)**

This field is available:

- For a **Single Point**, **Batch**, or **Grid** *Calculation Type* and when **Calculated** is selected as the **Absorptivity**.
- For a **Heat Source Calibration** *Calculation Type* and when **Calculated (prefactor calibrated)** is selected as the **Absorptivity**.

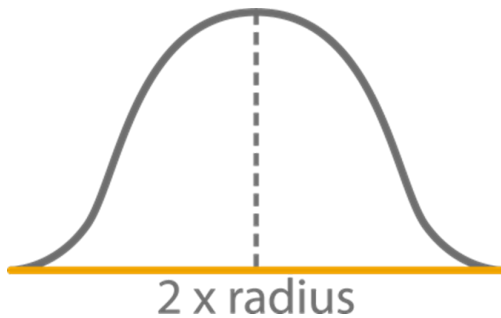


"About the Absorptivity Model" on page 47 and the section "Heat Source Calibration" on page 56 for more.

Enter the **Wave length** in nm.

### BEAM RADIUS (GAUSSIAN/ELECTRON BEAM)

The beam radius is where the power reduces to  $1/e^2$  as compared to its peak value. This is only applied on the top surface and then heat is diffused into the material based on the diffusivity of the material.



For a **Single Point**, **Batch**, or **Grid Calculation Type** enter the **Beam radius** size for the **Gaussian** or **Electron Beam** heat source model. The defaults are 40  $\mu\text{m}$  (Gaussian) or 200  $\mu\text{m}$  (Electron Beam). Hover over the field in the GUI to view the example of it as below.

For a **Heat Source Calibration Calculation Type** and when **Calculated (prefactor calibrated)** is selected as the **Absorptivity**.

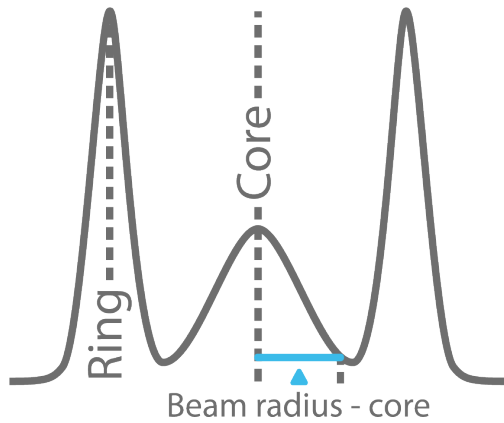
- Select **Calibrated** to calibrate the beam radius and the absorptivity or **User-defined** to enter a specific beam radius in the field (unit =  $\mu\text{m}$ ) and only calibrate the absorptivity.
- Then choose what to **Calibrate for**. Select **Width and depth** of the melt pool, or **Depth** to calibrate using only the depth of the melt pool.



"About the Absorptivity Model" on page 47 and the section "Heat Source Calibration" on page 56 for more.

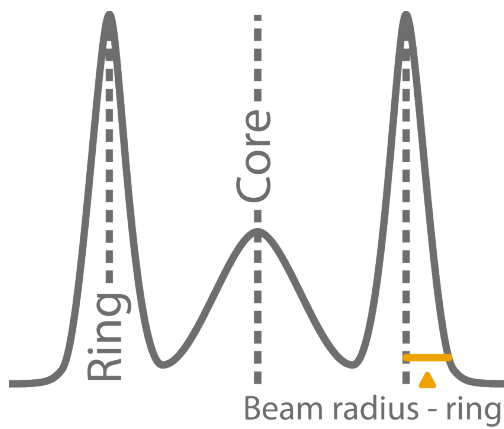
### BEAM RADIUS-CORE (CORE-RING)

The **Beam radius-Core** for a **Core-ring** heat source is the  $1/e^2$  radius of the core beam where the power reduces to  $1/e^2$  as compared to its peak value. Also see the equations in "Core-ring Heat Source" on page 30.



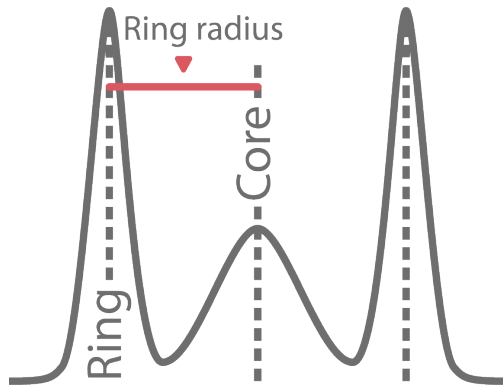
### ***BEAM RADIUS-RING (CORE-RING)***

The **Beam radius-Ring** for a **Core-ring** heat source is the  $1/e^2$  radius of the ring beam. Also see the equations in "Core-ring Heat Source" on page 30.



### ***RING RADIUS (CORE-RING)***

The **Ring radius** for a **Core-ring** heat source is the ring radius i.e. the distance between the center of the core beam to the center of the ring beam. Also see the equations in "Core-ring Heat Source" on page 30.



### AMOUNT OF POWER IN RING (CORE-RING)

Enter a value for the percent (%) of **Amount of power in ring**. This is the % of power for the ring beam. When using the **Index Mode**, the amount of power in the ring is set based on the selected index value (0-6).



"Index Mode (Core-ring)" on page 33

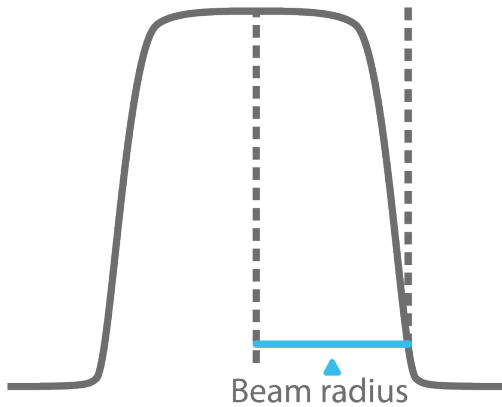
### MODE (CORE-RING)

Select a **Mode**:

- **Custom** to manually enter all parameters (the Beam radius-Core, Beam radius-Ring, Ring radius, and Amount of power in ring). See "[Custom Mode \(Core-ring\)](#)" on page 33 in the theory section for more details.
- **Index** to select the index value (0, 1, 2, 3, 4, 5, or 6). Enter a value for **Beam radius-Core** and then preset values for Beam radius-Ring, Ring radius, and Amount of power in ring are based on this selection. See "[Index Mode \(Core-ring\)](#)" on page 33 in the theory section for more details.

### BEAM RADIUS (TOP-HAT)

The **Beam radius** for a **Top-hat** heat source is the  $1/e^2$  radius of the top-hat beam i.e. where the power reduces to  $1/e^2$  as compared to its peak value. Also see the equations in detailed in "[Top-hat Heat Source](#)" on page 34.



## Keyhole Model




"About the Keyhole Model" on page 43

The use of the **Use keyhole model** checkbox in combination with fluid flow and separate material properties for powder is dependent on the simulation type.

The **Use keyhole model** checkbox is selected by default and available for a **Gaussian**, **Core-ring**, **Top-hat**, or **Electron Beam Heat Source**.

### **FEATURE CHECKBOX AVAILABILITY: FLUID FLOW, SEPARATE MATERIAL PROPERTIES WITH POWDER, AND KEYHOLE MODEL**

Depending on what you want to simulate, there are some setting combinations that are available based on the different simulation types. The details are relevant to using the **Fluid flow including Marangoni effect**, **Use separate material properties for powder**, and **Use keyhole model** checkboxes in a variety of combinations. The software automatically guides your choices. However, it can be useful to know in advance what is available to better design your simulation.

Simulation Type	Feature (checkbox) Availability		
	Use Separate Material Properties for Powder	Fluid Flow including Marangoni Effect	Use Keyhole Model (Heat Source setting)
Steady-state	available	available	available
Transient (with fluid flow)	not available	available	not available
Transient (without fluid flow)	available	not available	not available
Transient with heat source from Steady-state (Single-track scanning strategy)	available	available	available
Transient with heat source from Steady-state (Bidirectional or Unidirectional scanning strategies).	not available   <p>In this case, you can select the checkbox but the <b>Perform</b> button is unavailable.</p>	available	available

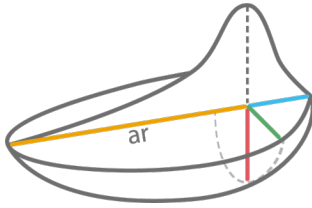
### **BEAM QUALITY FACTOR $M^2$**

Enter the **Beam quality factor  $M^2$**  of the laser beam.

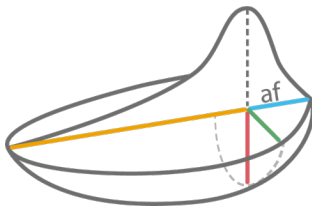
### **AR, AF, B, AND C (DOUBLE ELLIPSOIDAL)**

For **Double ellipsoidal**, also enter the following. For each field you can hover over it in the GUI to see these images that show you the location.

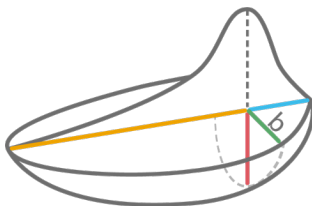
- **ar** (default is 70  $\mu\text{m}$ ).



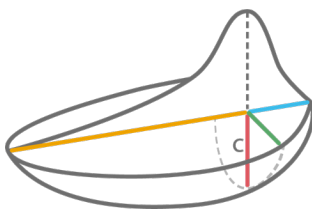
- **af** (default is 70  $\mu\text{m}$ ).



- **b** (default is 85  $\mu\text{m}$ ).



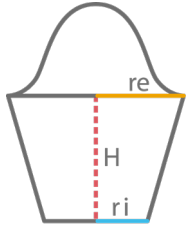
- **c** (default is 200  $\mu\text{m}$ ).



### **RE, RI, H (CONICAL)**

For **Conical**, 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  $\mu\text{m}$ ).
- **ri** (default is 60  $\mu\text{m}$ ).
- **H** (default is 100  $\mu\text{m}$ ).



## AM Calculator Experiment File Requirements

The following describes the requirements for importing experimental file data to be used with an **AM Calculator** either for a **Heat Source** or **Batch Calculation Type** and when setting it up on the **Configuration** window.

 "AM Calculator Conditions Settings" on page 111



It is important that the experiment data file contents are correctly entered in order for the calculation to perform smoothly.

### Heat Source Calibration Calculations

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.

These examples use experiment data, key holing, and heat source calibration:



"AM\_06a: Calibrating a Heat Source for a 316L Steel" on page 229 and "AM\_06b: Using the Calibrated Heat Source for a 316L Steel" on page 236.

Experiment Data						
Experiment file				delimiter Comma		
#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp. width (µm)	Exp. depth (µm)	Use
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 38: From example AM\_06a, this shows the data imported from an experiment file into the table. The "Use" column allows you to choose the points to include or exclude for the simulation. Here row (experiment) #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.

## Batch Calculations

When the **Batch Calculation Type** is selected in the **Steady-state** mode, there is an option to import a file with experimental data points.

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.



These examples use Batch calculations "AM\_07: Batch Calculations for an IN718 Alloy" on page 241, "AM\_08b: Batch Calculations for a Ti64 Alloy" on page 253, and "AM\_09b: Batch Calculations for an SS316L Alloy" on page 264.

Batch Experiment Data						
Experiment file <input type="text"/>						delimiter <input type="checkbox"/>
#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp.width (μm)	Exp.depth (μm)	Use
1	50.000000	500.000000	0.100000	67.379660	16.058920	<input checked="" type="checkbox"/>
2	50.000000	750.000000	0.066667	50.318180	11.830310	<input checked="" type="checkbox"/>

Figure 39: From example AM\_08b, this shows the data imported from an experiment file into the table. Here the "Use" checkbox for Experiment 1 is about to be deselected to exclude the data point after previewing it on the Visualizations window.

## 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.



If you select **Depth** from the **Calibrate for** list, then the experimental values for only the melt pool depth is needed.

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**. 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.

	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 (μm)	Depth (μm)	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 40: 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.



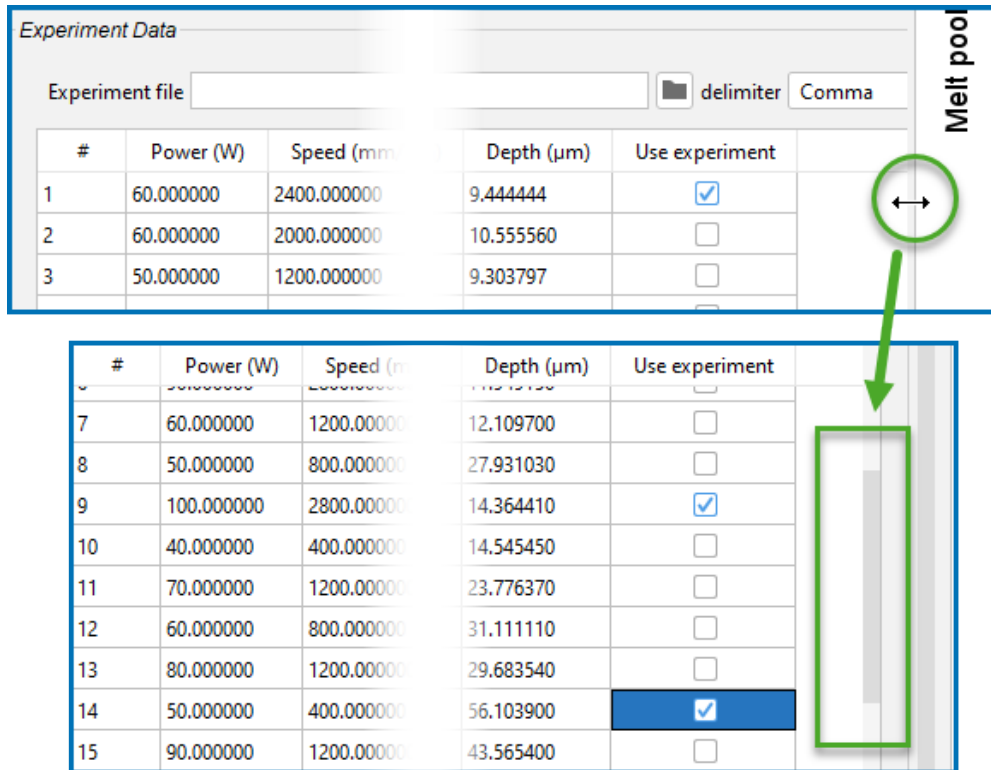
Once a set of data is imported to the table (or entered manually), expand the **Configuration** window to reveal the scroll bars. Then you can scroll up or down, left or right, within the table to select checkboxes for the relevant data points you want to include or exclude. This is applicable to the tables found on the AM Calculator as well as those on the Plot Renderer for the 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" on page 108, 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" on page 111 and "AM Calculator Options Settings" on page 143

### ***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.



**Export to folder** is intended for using the data in an external simulation software, where using smoothed data is expected to result in fewer numerical issues. **Save** and **Save as** are intended for when the data is to be used in the AM Module. The data will be smoothed according to the smoothing setting before use in the FEM simulation.

### ***Account for Interface Scattering***

When **Use data from Scheil Calculator** is selected, you can also select the **Account for interface scattering** checkbox.

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 **Phase interface scattering constant** is found to be  $4.0 \times 10^{-8} \Omega\text{m}$  for aluminum alloys produced with conventional methods. A higher value could be required for the AM process where the rapid solidification results in a very fine microstructure with a large amount of phase interfaces. The contribution to thermal conductivity is assumed to be related to that to electrical resistivity, following the Wiedemann-Franz law.

Either keep the default **Phase interface scattering constant**, which is a constant for evaluating thermal conductivity due to scattering at phase interfaces, or enter another value.

## ***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

- Electric resistivity
- Absorptivity
- Absorptivity vs angle

## **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
- Electric resistivity

### **Plot: Absorptivity Options**

The following absorptivity options depend on what combination of heat source and calculation type is selected on the **Conditions** tab. See "[AM Calculator Heat Source Settings](#)" on page 123.

**Absorptivity** and **Absorptivity vs angle** plot options are available when the conditions below are met:

- When a **Gaussian**, **Core-ring**, **Top-hat**, or **Electron Beam** heat source is selected, plus any one of the **Single Point**, **Batch**, or **Grid** calculation types AND when **Calculated** or **User-defined** is selected.
- When a **Gaussian** heat source is selected, plus a **Heat Source Calibration** calculation type AND when **User-defined** is selected.

Also, for a **Gaussian** heat source, **Heat Source Calibration** calculation type, AND when **Calculated** is selected, then the options are:

- **Absorptivity (wavelength is <value from Heat Source settings> with prefactor 1.0).**
- **Absorptivity vs angle (wavelength is <value from Heat Source settings> with prefactor 1.0).**



The wavelength <value from Heat Source settings > for the **Absorptivity** and **Absorptivity vs angle** is the same as what is entered on the **Conditions** tab. For example, if 1064 nm is entered, it displays on the **Material Properties** tab as **Absorptivity (wavelength is 1064 nm with prefactor 1.0).**

### **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" on page 108, the **Configuration** window has these settings tabs where there are many available conditions to set on the **Conditions**, **Materials Properties**, and **Options** tabs, as well as with the specialized **Plot Renderer**.

This section describes the settings for the **Options** tab.



"AM Calculator Conditions Settings" on page 111, "AM Calculator Heat Source Settings" on page 123, and "AM Calculator Materials Properties Settings" on page 139

### ***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.

### ***Smagorinsky Constant***

Enter a **Smagorinsky constant** in the field. This is the value of the constant to use in the fluid flow model. The default is 0.18.



For more information see "Fluid Flow" on page 24 in the theory section.

### ***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 installed examples to visualize how the probes are set up for a Diffusion Calculator (see "[AM\\_05: Using AM Calculator Probe Data with the Diffusion Module \(DICTRA\)](#)" on page 220) and for the Precipitation Calculator (see "[AM\\_12: Using AM Calculator Probe Data with the Precipitation Module \(TC-PRISMA\)](#)" on page 285).

1. Set up the probes on the **AM Calculator**. Probes are available for transient calculations only. See "[AM Calculator Conditions Settings](#)" on page 111. 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.
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 Calculator: Plot Renderer Settings

Once you have added an "AM Calculator" on page 108, 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 listed below are made available based on a variety of combinations, e.g. the type of *simulation* (**Steady-state**, **Transient**, or **Transient with heat source from Steady-state**), whether *fluid flow* or the *keyhole* checkboxes are selected, whether *probes* are used, and so on. There are also *Calculation Types* (**Single Point**, **Heat Source Calibration**, **Batch**, and **Grid**), that impact the settings shown, and these options are further included on separate tabs (i.e. **Batch**, **Grid**, **Heat Source Calibration**, **3D Plot**, **Plot Over Line**, **Thermal Gradient vs Solidification Rate**, etc.), which are further fine-tuned based on the *plot types* or *display types* selected on the **Plot Renderer**.



The software guides you by only providing the relevant settings to define. Due to the number of settings available, these are not in order as to what you see on the **Configuration** window. Where relevant or logical, settings are grouped together and some detail of when or how you would access the setting is included. Scroll or search for the setting or field name as there are many simulation, calculation, display options, and plot type combinations that are available on the relevant tab on the **Plot Renderer Configuration** window.



"About the AM Plot Types" on page 64 and "Working with AM Visualizations" on page 164

---

## General

### *Add a Quantity and Remove a Quantity or Function Buttons*

Use the **Add quantity** or **Add function** ⊕ and **Remove quantity** or **Remove function** ⊖ buttons to add additional setting groups that include all of the available settings. Click the checkbox next to these buttons to enable or disable any particular settings group.

### *Show Table or Hide Table*

These buttons are available on the **Batch** or **Grid** tabs after running either a **Batch** or **Grid** calculation, respectively.

Use the **Show Table** or **Hide Table** buttons to view the simulation data and using the table you can fine tune the plot. Also see "[Visualizing Batch Calculations in the AM Module](#)" on page 188 for more information about the columns in the table.

### *Export Result Files*

This option is available on the **Plot Renderer Configuration** window for the following tabs (and the associated plots that are available for each): **3D Plot**, **Grid**, **Batch**, or **Heat Source Calibration**. The button is available after the simulation is run.

Click **Export Result Files** to export the files and visualize the results in external software. The result file(s) in Exodus format (\*.e) and the json file info .json are copied to the folder selected. The json info file contains the settings and results from each data point. You can open the json info file in a text reader (i.e. Notepad) to review it before using the Exodus file in the external software.

When exporting from the **Grid**, **Batch**, and **Heat Source Calibration** tabs, the results are saved in separate subfolders, each containing the .e and .json files. The Exodus file format is used for finite element analysis.



The folders containing the sets of results files are exported in random order. To match the folder to the original datapoint, open the json file in a text editor. In this file there is a `number` field that matches the row (experiment) number in the table found on the **Plot Renderer Configuration** window for the applicable tab (click **Show Table**). You can use this to match the folder contents back to the datapoint as needed.

#	Power (W)	Speed (mm/s)	P/V (J)
1	60.0	2400.0	0.025
2	60.0	2000.0	0.03
3	50.0	1200.0	0.0417
4	60.0	1600.0	0.0375
5	80.0	2800.0	0.0286
6	90.0	2800.0	0.0321
7	60.0	1200.0	0.05
8	50.0	800.0	0.0625
9	100.0	2800.0	0.0357

```

37 "separateMaterial": false,
38 "percentagePowderDensity": 0.8,
39 "beamRadius": [
40   2.2E-5,
41   2.2E-5
42 ],
43 "power": 60.0,
44 "speed": 1200.0,
45 "absorptivity": 0.3,
46 "layerThickness": 10.0,
47 "liquidusTemperature": 1710.799,
48 "solidusTemperature": 1596.869,
49 "evaporateTemperature": 2954.5496863877147,
50 "mainFileName": "result.e",
51 "number": 7,
52 "expWidth": 52.21519,
53 "expDepth": 12.1097

```

In the **Export Result Files** window, in the **Folder Name** field, navigate to the location or enter a file path to where you want to save the files. Create a new folder as needed. Click **Save**. The individual exported files in each folder are named `info.json` and `result.e` and can be used in external software. When there are multiple subfolders (i.e. for Grid, Batch and Heat Source Calibration) the number of folders exported is the same as the number of rows of data selected to be used. If the **Use** checkbox is not selected, the result folder (and files) for that data point is not exported. Click the **Show Table** button to review the rows of data and choose to include or exclude data points from the **Use** column.

### Filter by Material Types


This option is available when **Use separate material properties for powder** is selected on the **AM Calculator**.

Select **Filter by material types** to visualize specific material types from the list.

## Layer Thickness

Enter a **Layer thickness** in  $\mu\text{m}$ . The printed layer thickness is equal to the powder thickness first after multiple layers have been printed. For a single track, or a few layers, the printed layer is often much less than the powder thickness.

## Table View

Click the  **Table View** button to toggle it ON and OFF between a plot and a table.

 "AM Plots: Table View Options" on page 98

## Transient-related Simulations

### 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



Button	Action
⏪	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 for **Transient** simulations.

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.

## Axis Variables and Display Options

### 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 and options vary.

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 when the **Fluid flow including Marangoni effect** checkbox is selected:
  - **Dynamic viscosity** (Pascal-second, milliPascal-second, Poise, or centiPoise)
  - **Flow velocity field** (m/s, mm/s, μm/s, or nm/s)
  - **Solidification rate** (m/s, mm/s, μm/s, or nm/s)
- **Thermal gradient** (Kelvin per m)

### Display

Based on the **Axis variable** selected, choose the way to **Display** it.

- For **Temperature**, **Surface tension**, **Thermal conductivity**, or **Dynamic viscosity**, choose **Surface colormap**, **Contour**, or **Iso-surface**. For Surface colormap and Contour, you can also click to select the **Slice** checkbox.
- For **Flow velocity field**, choose the object shape to display velocity vectors: **Arrow**, **2D glyph**, **Cone**, or **Line**.

The settings are further described below.

## Tubes

Available for display type **Contour**.

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***

This is available for display types **Surface colormap** or **Contour**.

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.

## 2D 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.

### *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.

## Grid Calculations (Grid, 3D Plot, and Plot Over Line Tabs)

These settings are specific to **Grid** calculations and are on the **Grid** tab. Also see plot related settings: "[Melt Pool vs Energy Density Plot Type \(Batch and Grid Tabs\)](#)" on page 157 and "[Printability Map Plot Type \(Batch and Grid Tabs\)](#)" on page 157.

### *Grid Data*

This is available on the **3D Plot** and **Plot Over Line** tabs after running a **Grid Calculation Type**.

The data in the **Grid data** list shows all the power and scan speed combinations and it is sorted in order of energy density.

### *Grid Calculation Table*

The following is on the **Grid** tab after a **Grid** calculation.

Use the **Show Table** or **Hide Table** buttons to view the simulation data and using the table you can fine tune the plot. The table includes these columns:

- The **Power (W)** and **Speed (mm/s)** columns are the from what was entered under *Grid Definitions*, on the **AM Calculator** i.e. the **Min**, **Max**, and **Number of steps**.
- The **P/v (J/mm)** column is the automatically calculated energy density, defined as the ratio of power and scanning speed.
- The calculated width and depth points, with the prefix **Sim.**, i.e. **Sim.width (μm)** and **Sim.depth (μm)**, to indicate it is the result of the simulation.
- The **Use** column has a checkbox available to include or exclude these points on the plots. When a **Use** checkbox is not selected, the results are also not included if you choose to export the result files. See "[Export Result Files](#)" on page 148.

## Batch Calculations

These settings are specific to **Batch** calculations. Also see plot related settings: "[Parity Plot \(Batch Tab\)](#)" on the next page, "[Melt Pool vs Energy Density Plot Type \(Batch and Grid Tabs\)](#)" on page 157, and "[Printability Map Plot Type \(Batch and Grid Tabs\)](#)" on page 157.



After the **Batch Experiment Data** set up is completed on the **AM Calculator** and the simulation run, then on the **Plot Renderer** you can start to work with the data you want to plot. Also see [Including or Excluding Data Points](#) to understand the interaction between these two tables.

## Batch Experiments

**Plot Renderer** → **3D Plot**, **Plot Over Line**, and **Thermal Gradient vs Solidification Rate** tabs after running a **Batch Calculation Type**.

The data in the **Batch Experiments** list shows all the power and scan speed combinations and it is sorted in order of energy density.

## Batch Calculation Table

**Plot Renderer** → **Batch** tab.

You may have already included or excluded data points earlier when previewing the data. Use the **Show Table** or **Hide Table** buttons to view the included or excluded data and continue to fine tune the plot.

On the **Batch** tab the table includes these columns:

- The original experiment data for **Power (W)** and **Speed (mm/s)**, plus the **P/v (J/mm)**.
- The original experiment data for **Width** and **Depth**, now with a prefix **Exp.** e.g. **Exp.width (μm)**, to indicate this is the experimental data.
- The calculated **Width** and **Depth** points, with the prefix **Sim.**, e.g. **Sim.width (μm)**, 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** column has a checkbox available to continue to include or exclude those points on the plots. When a checkbox is not selected, the results are also not included if you choose to export the result files.



Only points that were originally included in the simulation can be selected or deselected from the **Use** 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.

## Parity Plot (Batch Tab)

The following settings are available on the **Batch** tab when a **Parity plot** is selected as the plot type.

### **Plot Width and Plot Depth**

Click the **Plot width** and **Plot depth** checkboxes to toggle between displaying these points on the associated tab on the **Visualizations** window for the plot.

---

## Show Limits

Select the **Show limits** checkbox to include additional visual cues on the plot. Choose **RMS**, **Absolute**, or **Relative in %**. For **Absolute** or **Relative in %** enter a number in the field. The plot dynamically updates as you toggle between these options.

## Melt Pool vs Energy Density Plot Type (Batch and Grid Tabs)

The following settings are available on the **Batch** or **Grid** tab when a **Melt pool vs energy density** is selected as the plot type.

### Plot Width and Plot Depth

Click the **Plot width** and **Plot depth** checkboxes to toggle between displaying these points on the associated tab on the **Visualizations** window for the plot.

## Printability Map Plot Type (Batch and Grid Tabs)

The following settings are available on the **Batch** or **Grid** tab when a **Printability map** is selected as the plot type and after running a **Batch** or **Grid** calculation, respectively. The example used in the images is AM\_08a.

### Show Result Points

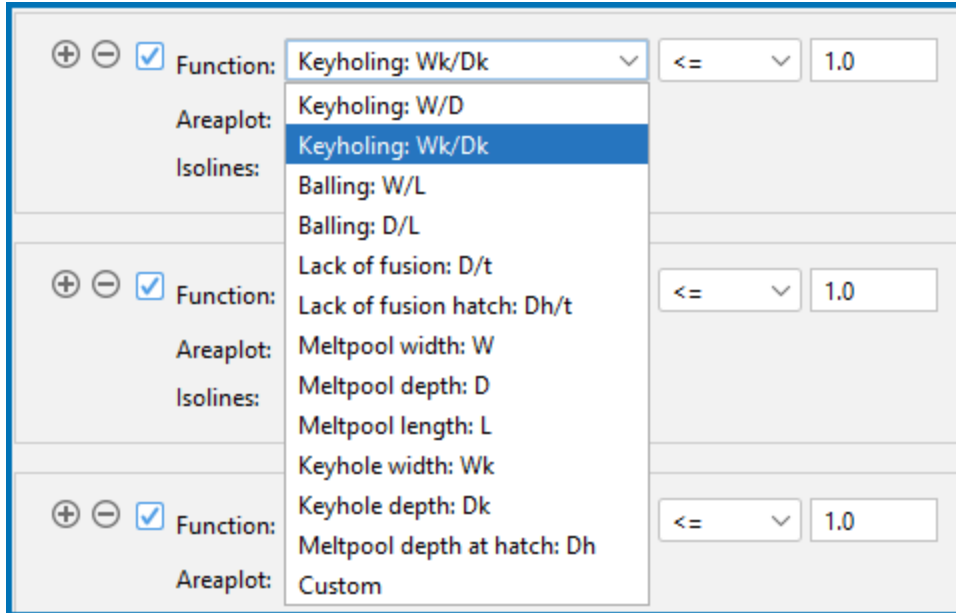
Select or deselect the **Show result points** checkbox to toggle these on and off on the **Visualizations** window. The plot updates dynamically.

### Hatch Distance

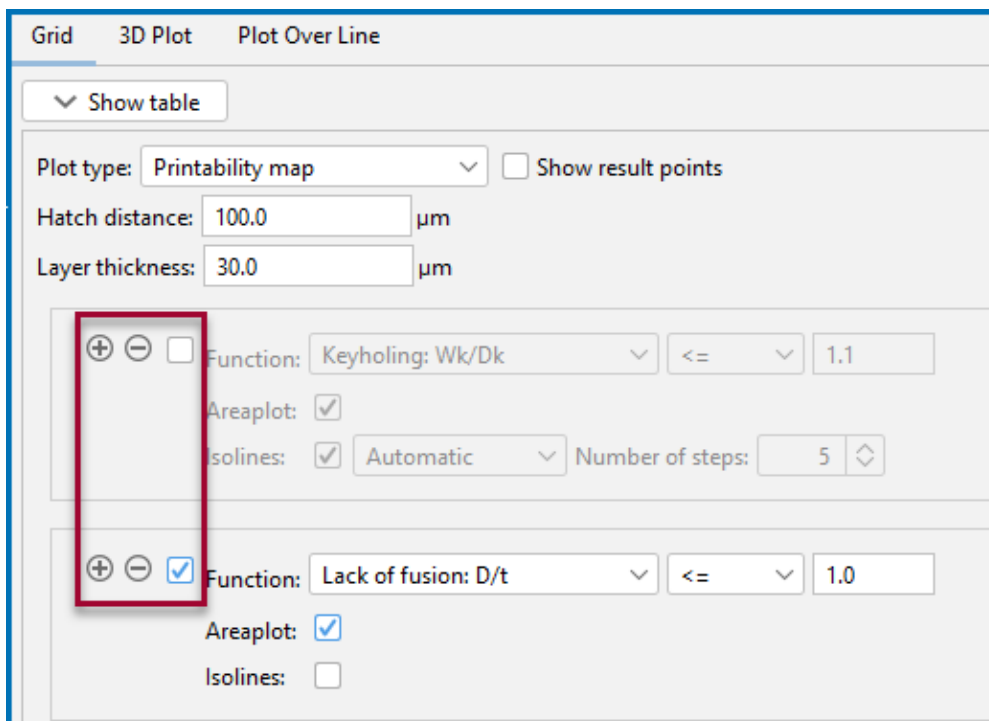
Enter a **Hatch distance** in  $\mu\text{m}$ .

### Function

**Printability maps** with varied power and scanning speeds can be simulated showing regions where the defects keyholing porosity, lack of fusion, and balling occur. The defect regions are defined by functions of the keyhole dimension (**Wk**, **Dk**), melt pool dimensions (**W**, **D**, **L**), hatch distance (**h**), and powder thickness (**t**). Use the  $\oplus$  plus and  $\ominus$  minus buttons to add and remove sections to define each **Function**.



Click to toggle the **Function** checkbox to show and hide the selected functions as needed on the **Visualizations** window.



From the list select one of the following for each section added then for each choose **All**, **<=** (less than or equal to), or **>=** (greater than or equal to) and enter a number in the field associated field. On the **Visualizations** window, there are numbered labels to indicate experiments with defects, in **red**, or without defects, in **green**.

- **Keyholing: W/D**
- **Keyholing: Wk/Dk**
- **Balling: W/L**
- **Balling: D/L**
- **Lack of fusion: D/t**
- **Lack of fusion hatch: Dh/t**
- **Meltpool width: W**
- **Meltpool depth: D**
- **Meltpool length: L**
- **Keyhole width: Wk**
- **Keyhole depth: Dk**
- **Meltpool depth at hatch: Dh**
- **Custom**. Then enter a **Name** in the field.

### ***Areaplot***

Click the **Areaplot** checkbox to toggle between including the colored sections on the plot.

### ***Isolines***

Click the **Isolines** checkbox to include or exclude isolines.

Choose **Automatic**, **Custom**, or **Custom values**.

- For **Automatic**, enter the **Number of steps**.
- For **Custom**, enter the distance between lines i.e. 1.0 1.25 1.5.
- For **Custom values**, enter a **Min**, **Max**, and **Number of steps**.

## Heat Source Calibration Calculations



See "AM Calculator Conditions Settings" on page 111 and "Visualizing Heat Source Calibrations" on page 178 for more detailed information.

### Heat Source Calibration Experiments

**Plot Renderer** → **3D Plot** and **Plot Over Line** tabs after running a **Heat Source Calibration Calculation Type**.

The data in the **Heat Source Calibration Experiments** list shows all the power and scan speed combinations and it is sorted in order of energy density.

### Heat Source Calibration Table

**Plot Renderer** → **Heat Source Calibration** tab.

Click the **Show Table** or **Hide Table** buttons to view or hide the experimental and simulation data. To fine-tune the choice of data points, click the checkbox for the **Use** column for each row (experiment #) to observe the dynamic changes on the **Heat Source Calibration Parameters** and/or **Melt Pool Dimensions** tabs on the **Visualizations** window.



Experiment data and other settings must be configured for this to be available. The data points available in the table 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** column.

On the **Heat Source Calibration** tab (Plot Renderer) the table includes these columns:

- The original experiment data for **Power (W)** and **Speed (mm/s)**, plus the energy density **P/v (J/mm)**.
- The original experiment data for **Width** and **Depth**, now with a prefix **Exp.** to indicate this is the experimental data, e.g. **Exp. width (μm)**.

- The calculated **Width** and **Depth** points, with the prefix **Sim.** to indicate it is the result of the simulation, , e.g. **Sim. width ( $\mu\text{m}$ )**.



Only the experiment data points selected for inclusion when setting up the AM Calculator have data in these two **Sim.** columns.

- The **Use** column has a checkbox available to include or exclude those points on the *Heat Source Calibration Parameters* and *Melt Pool Dimensions* tabs on the **Visualizations** window. Furthermore, if a simulation is not used in a 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** 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 Thermal Gradient vs Solidification Rate								
Export Result Files...								
^ Hide Table								
#	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
1	60.0	2400.0	0.025	36.8421	9.4444	37.2076	9.7557	<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 41: 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 column. Rows 2-4 were not included in this simulation and are greyed out. When a checkbox is not selected, the results are also not included if you choose to export the result files.

## Heat Source Parameters

Plot Renderer → Heat Source Calibration tab.

These are the parameters you choose on the **Plot Renderer Configuration** window. There is also a **Heat Source Calibration Parameters** tab on the **Visualizations** window that is used in combination with the table to choose data points.

The heat source parameters available to plot for each *Heat Source* type (i.e. *Gaussian*, *Double ellipsoidal*, or *Conical*) selected on the **AM Calculator**:

- *Gaussian*: **Beam radius**. Then based on the **Absorptivity** setting—**Calculated (prefactor calibrated)** or **Constant (calibrated)**—also **Absorptivity prefactor** plus additional options.
- *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 Heat Source Settings](#)" on page 123.

For all heat source parameters, you can also choose to select:

- A **Constant function** value i.e. the average value from all experiments or a function of energy density, which could also be a **Linear function**, **Cubic function**, or a **Sigmoid function**.
- A **2d cubic function P and v** of *Power (P)* and *Scanning speed (v)*.
- 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.

## Working with AM Visualizations



The AM Calculator visualization using the Plot Renderer is different from the other Calculators available in Thermo-Calc.



"About the AM Plot Types" on page 64

### 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 different 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.

#### Window Connectivity

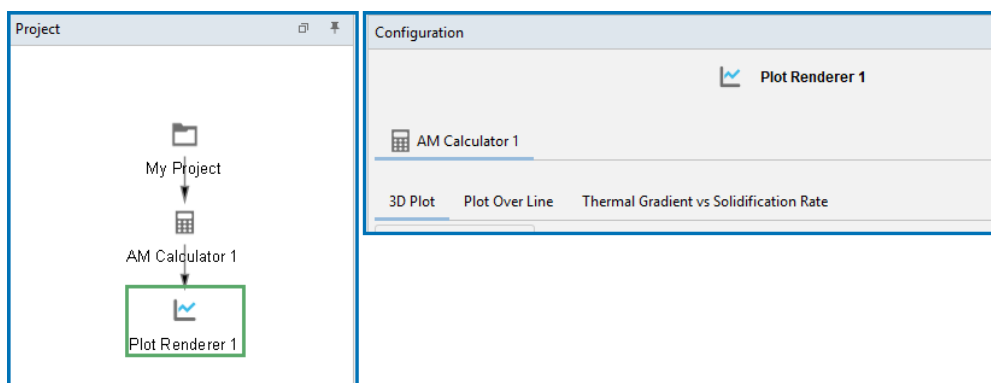
In general:

- 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, Probe Plot, etc.), 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.

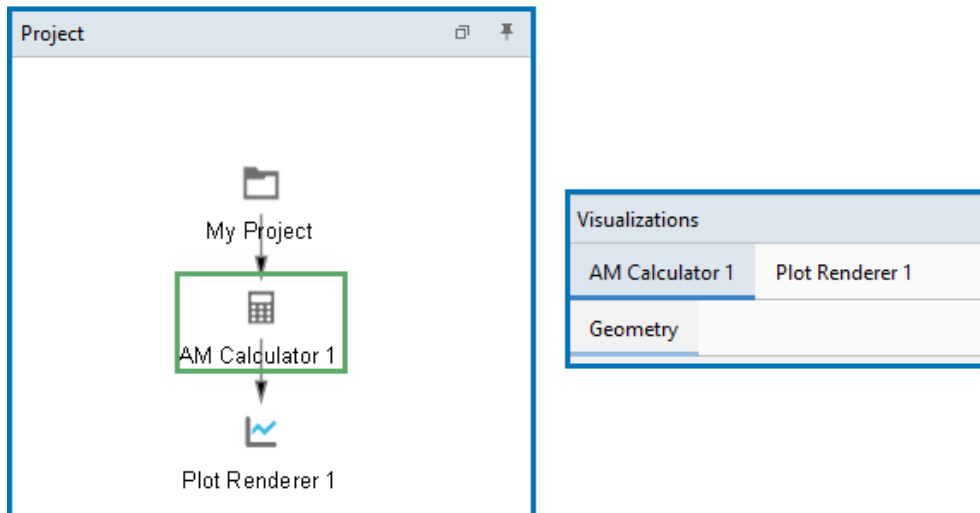
## Configuration Window

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**, **Probe**, and **Thermal Gradient vs Solidification Rate** 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.

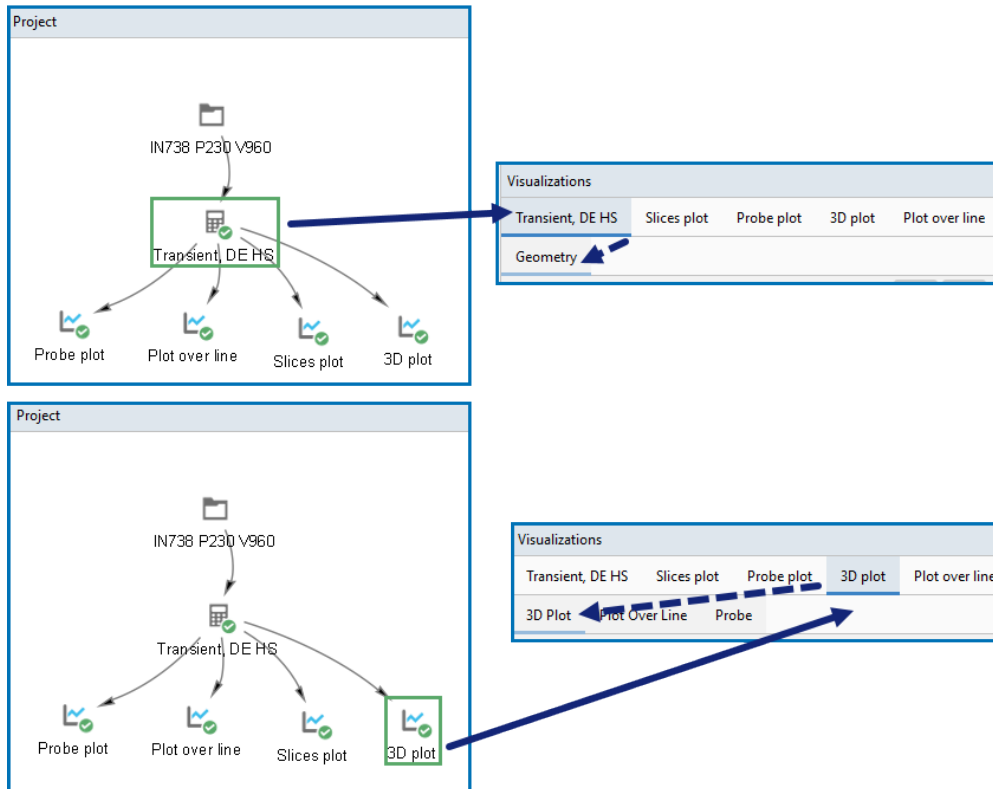
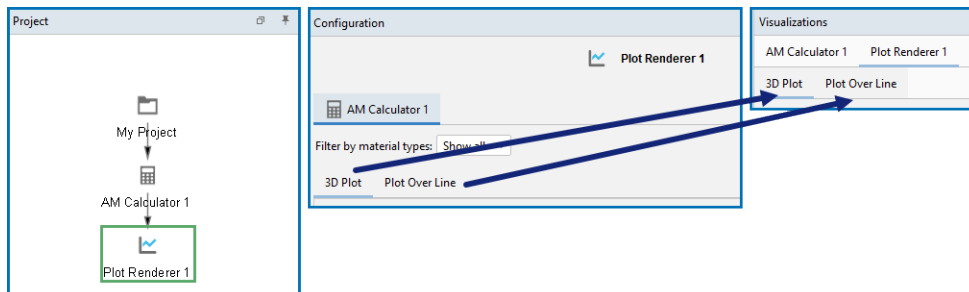


Figure 42: When you click a node in the Project window, the associated tab is highlighted and underlined in the Visualizations window. Then the second level of tabs are further underlined to indicate what is associated to the selected top level tab, and this is what actually displays in the Visualizations 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" on page 39](#)

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" on page 178](#).

## Visualizing Batch and Grid Calculations

The following is available for a **Steady-state** → **Batch** or **Grid** *Calculation Type*.

Results from **Steady-state** simulations using **Batch** or **Grid** calculations contain multiple steady-state results, one for each power and scan speed. The results can be visualized individually using a **3D Plot** and **Plot Over Line** (described above). The combined result from all steady-state simulations can additionally be visualized from the **Batch** tab (available on the **Plot Renderer Configuration** window after the **Batch** calculation), and similarly from the **Grid** tab after a **Grid** calculation. On these tabs you choose additional plot types **Printability map**, **Parity plot**, and **Melt pool vs energy density**.

A table of the experimental and calculated values can be viewed when you click to toggle between the **Show Table** or **Hide Table** buttons. The table is displayed in the **Plot Renderer Configuration** window for the relevant **Batch** or **Grid** tab.

The screenshot shows a software interface with two main panels. The left panel, titled 'Project', contains a tree view with a folder icon labeled '316L', a sub-item 'AM 316L Batch' with a green checkmark, and a '3D Batch' icon highlighted with a green box. The right panel, titled 'Configuration', shows a '3D Batch' status at the top right. Below it is a '3D Plot' tab selected, with other tabs for 'Batch', 'Plot Over Line', and 'Thermal Gradient vs Solidification Rate'. There is an 'Export Result Files...' button and a 'Hide Table' button. A table displays simulation data for four different runs.

#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp. width (μm)	Exp. depth (μm)
1	60.0	2400.0	0.025	36.8421	9.4444
2	60.0	2000.0	0.03	39.0093	10.5556
3	50.0	1200.0	0.0417	51.076	9.3038
4	60.0	1600.0	0.0375	46.4396	11.6667

Figure 43: The simulated and experimental melt pool dimension can also be tabulated for all data using the toggle button to Show Table or Hide Table. This is from example AM\_09b.



More details about this are described in separate topics. See "[Visualizing Batch Calculations in the AM Module](#)" on page 188 and for the various plot settings for a Grid, these are included with the general topic, "[AM Calculator: Plot Renderer Settings](#)" on page 147.












There is a video tutorial about the **Printability Map** on our [website](#) and on our [YouTube channel](#). It is also included in the Additive Manufacturing Module [YouTube playlist](#).










## 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" on page 176](#)

 ["Working with AM Visualizations" on page 164](#)

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 YZ, 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	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.

Button	Name	Action
	zoom level	<p>Then when you click the <b>Apply saved camera view and zoom level</b> , it mirrors 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>
	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. This button also toggles displaying the <b>Heat Source Intensity (W/mm<sup>2</sup>)</b> legend on and off in the plot area.
	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 line between powder and solid	Click to turn on and off a line and plane around the powder layer thickness.
	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> </ul>

---

<i>Button</i>	<i>Name</i>	<i>Action</i>
		<ul style="list-style-type: none"><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>

---

## Menu or Keyboard Options: Geometry and 3D Plots

---

When working with the AM Calculator (probes) and its associated Plot Renderer (for 3D Plots), there are menu options or tooltip guidelines available to access certain actions for probes, markers, or to save a snapshot image from the **Visualizations** window.

### Plot- and Geometry-related Menus to Save Images

When available, right-click in the **Visualizations** window and choose **Save As** from the menu to save the current view in png, gif, or jpg format. This saves an image snapshot of the 3D Plot or Geometry at the zoom level and angle you are currently viewing.

- **Geometry:** When setting up probes, you can right-click on the Geometry to save the current view.
- **3D Plot:** When working with a 3D Plot, you can right-click anywhere on the 3D Plot to save the current view. This option is only available with AM Calculator 3D plots.

### Probe-related Menus and Tooltips

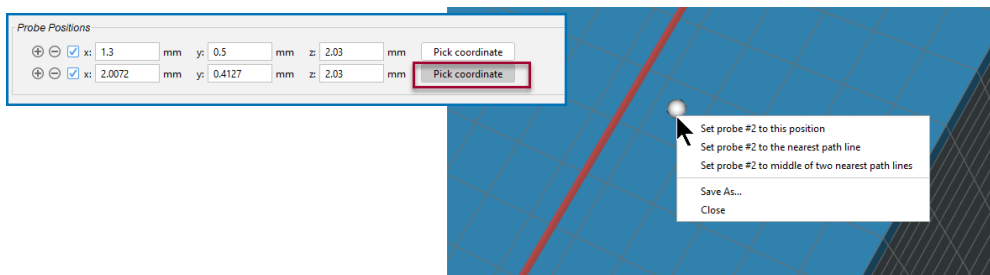


"AM Calculator: Working with Probe Data" on page 145

When you are picking the coordinates for the probe location, there is a menu available when you right-click a point on the Geometry in the Visualizations window. There are also tooltips that can help guide you.

1. On the **Configuration** window for an **AM Calculator**, under *Probe Positions*, click **Pick coordinate**. Hover over the button to view the tooltips with options to:
  - a. Double-click on the geometry to pick a coordinate.
  - b. Hold the <Ctrl> key and double-click to pick the position at the middle of a layer.
  - c. Hold the <Shift> key and double-click to map the picking position with a scanning path.
  - d. Hold the <Shift+Alt> keys and double-click to map the picking position to the middle of the scanning path.
2. In the **Visualizations** window on the **Geometry** tab, use one of the methods to add a probe point to the geometry, for example double-click to add a probe.

3. In the **Visualizations** window, right-click to choose an option from the menu associated to the new **Pick coordinate** probe point:
  - a. **Set probe to this position**: Sets to the location where the probe is located at the time of selecting.
  - b. **Set probe to the nearest path line**: Moves the probe to the path line, which can be where other probes are already set up.
  - c. **Set probe to the middle of the two nearest path lines**: This only works if there are two path lines, otherwise the probe does not move.
  - d. **Save As...**: Select this to save an image of the current view of the probes (a snapshot at the zoom level chosen). You can save in png, gif, or jpg formats.
  - e. **Close** : Closes this menu.

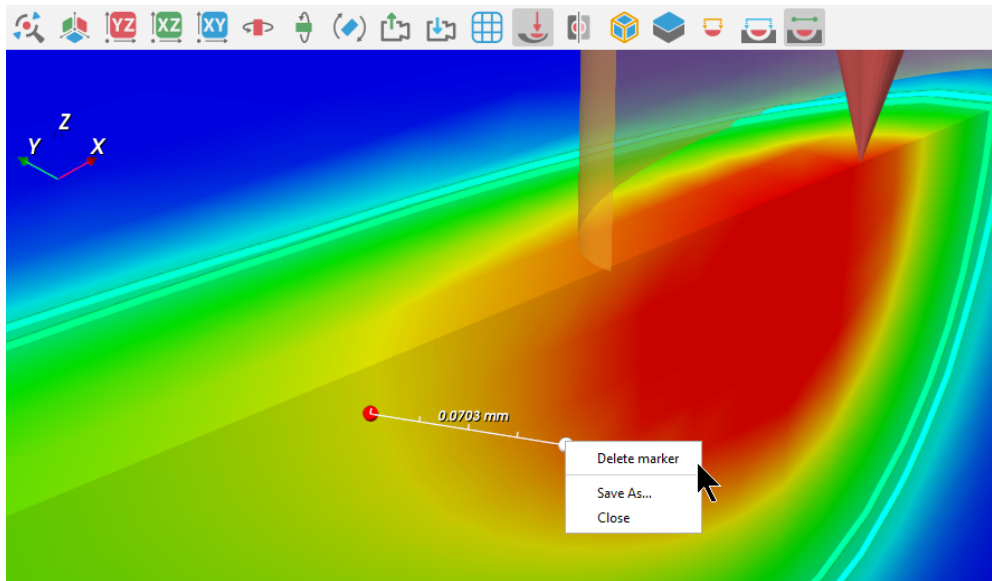


## Marker-related Menus and Tooltips for 3D Plots (AM Module Only)

Markers are added to 3D plots in the **Visualizations** window. For example, you are measuring the distance between two points on a heat source.

1. On the **Visualizations** window for an AM Calculator **Plot Renderer** (and after running a **3D Plot**), click the **Show Manual Ruler** button on the toolbar. Hover over the button to view the tooltips with options to:
  - a. Double-click each time to add the start and end points.
  - b. 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.
  - c. To move a ruler: Click and hold a point, then drag to reposition.
  - d. Right-click a point to delete the ruler.

2. Click in the plot area to add markers for the start and end points, for example. Once the marker is on the plot, you can delete it or save an image of the plot at the current view.
  - To delete a marker, zoom as close to it as possible and right-click the marker to bring up the menu. Select **Delete marker** from the list. You might have to move the mouse around the area of the marker or zoom further to access the menu options.
  - To save an image of the plot at the current view, click anywhere in the plot and select **Save as**. Select this to save an image in png, gif, or jpg format.



## Rotate, Zoom, and Pan 3D Plots: AM Calculations

When working with the Additive Manufacturing (AM) Module and the results from an "AM Calculator" on page 108 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" on page 170) 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.

## Overlaying Plots in the AM Module

In general, it is now possible to overlay plots from any Calculator with plots from any other Calculator.

2D plots are overlaid pair-wise which means if you add two or more AM Calculators as a predecessor in one Plot Renderer, similar 2D plots (such as Probe with Probe and Plot Over Line with Plot Over Line, etc.) are grouped and overlaid in one tab on the Visualizations window.

The most important use case is that you can overlay Probe plots from different Transient and Transient with heat source from Steady-state Calculators and Plot Over Line plots from all AM Calculators. Similarly, you can also overlay Meltpool vs energy density and Parity plot from different Batch, Grid, and Heat Source Calibration calculators.



You need to click **Perform** to make plot overlaying work.



It is not possible to overlay two 3D plots. However, when you want to plot two or more variables in a comparable way to overlaying plots in 2D, you use one **Plot Renderer** connected to at least two AM Calculators. Then on the **3D Plot** tab on the **Configuration** (settings) window you choose different quantities from the **Axis variable** list. Use the **+** and **-** buttons as usual to add and remove quantities. The plots are then viewed on separate **3D Plot** tabs on the **Visualizations** window. After defining the axis variables, click **Perform** to update the plots on the respective **Visualization 3D Plot** tabs. It is also important to rename your AM Calculators in order to better work with these plots.

## 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.



To import data for use in batch calculations, it is recommended you review "AM Calculator Experiment File Requirements" on page 134. Also review "Working with AM Visualizations" on page 164, which provides an overview of how most of the windows are connected.

It is recommended to open and run one of the examples to best understand the wide variety of visualization outcomes.



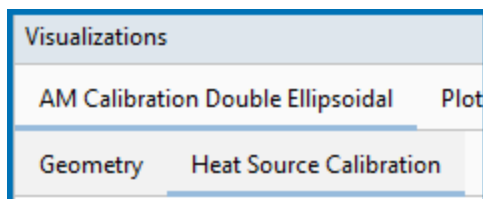
"AM\_06a: Calibrating a Heat Source for a 316L Steel" on page 229 and "AM\_06b: Using the Calibrated Heat Source for a 316L Steel" on page 236

### Working with AM Calculator Heat Source Calibration 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.

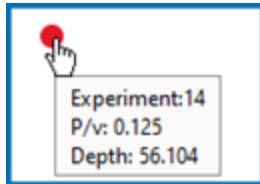
Experiment Data							
Experiment file					delimiter	Comma	
#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp. width (μm)	Exp. depth (μm)	Use	
1	60.000000	2400.000000	0.025000	36.842110	9.444444	<input checked="" type="checkbox"/>	
2	60.000000	2000.000000	0.030000	39.009290	10.555560	<input type="checkbox"/>	
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** 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 46](#)). 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.

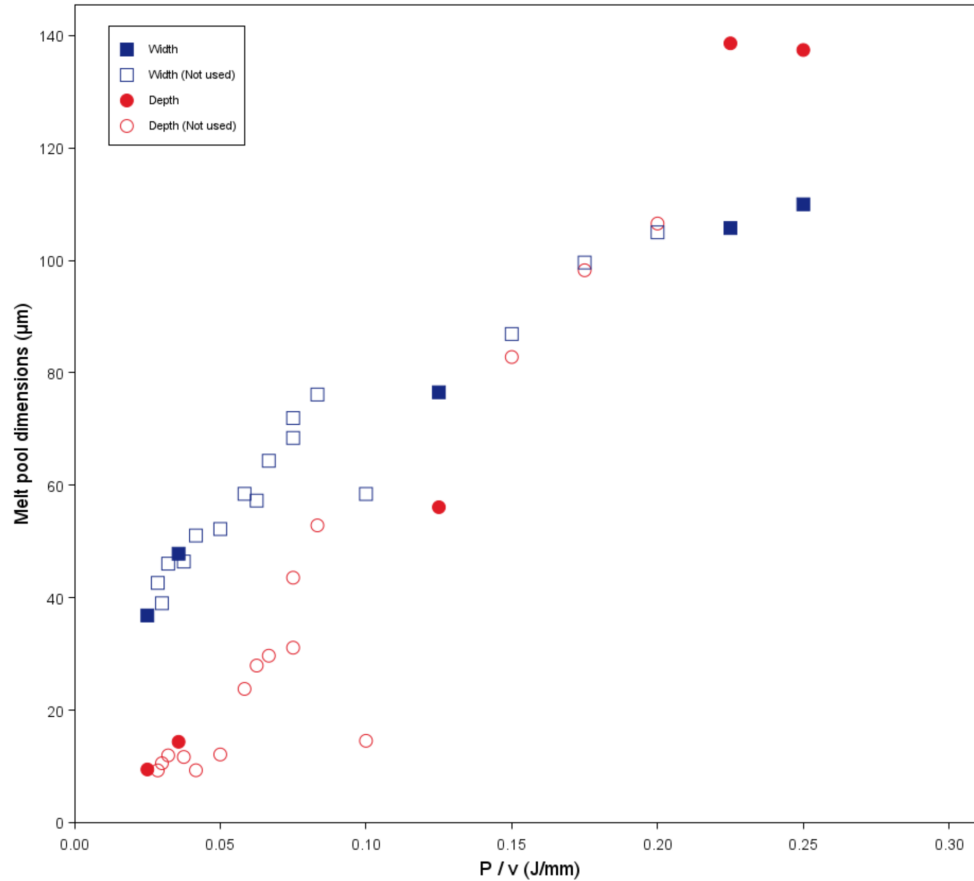


Figure 44: From example AM\_06a, this is the heat source calibration preview for the data imported to the AM Calculator node (renamed to AM Calibration Double Ellipsoidal). As per the legend, both "used" (solid color) and "not used" (hollow with outline color) points are included in this preview.



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 data.

Experiment Data

Experiment file  delimiter

#	Power (W)	Speed (mm)	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"/>

Melt pool

#	Power (W)	Speed (mm)	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"/>

## Working with the Plot Renderer for Heat Source Calibrations

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.

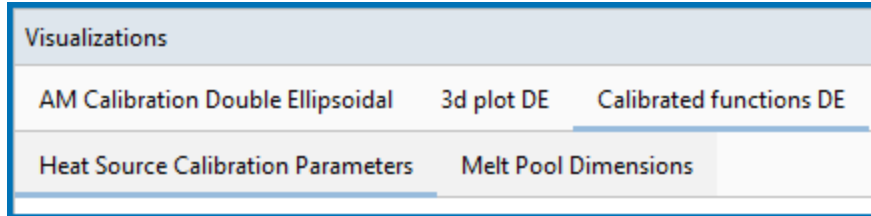


For additional **Plot Renderer** settings (and other visualization options for plotting) go to "AM Calculator: Plot Renderer Settings" on page 147.

### 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** list. The points in this list match the points selected in the **Use** 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 Parameters* and *Melt Pool Dimensions* tabs in the **Visualizations** window on the secondary level of tabs. These

tabs are where you fine-tune the choice of data points: Click the checkbox for the **Use** column for each row (experiment #) to observe the dynamic changes on the **Heat Source Calibration Parameters** and **Melt Pool Dimensions** tabs on the **Visualizations** window.



### ***Navigating the Heat Source Calibration Table***

On the **Heat Source Calibration** tab (Plot Renderer) the table includes these columns:

- The original experiment data for **Power (W)** and **Speed (mm/s)**, plus the energy density **P/v (J/mm)**.
- The original experiment data for **Width** and **Depth**, now with a prefix **Exp.** to indicate this is the experimental data, e.g. **Exp. width (μm)**.
- The calculated **Width** and **Depth** points, with the prefix **Sim.** to indicate it is the result of the simulation, , e.g. **Sim. width (μm)**.



Only the experiment data points selected for inclusion when setting up the AM Calculator have data in these two **Sim.** columns.

- The **Use** column has a checkbox available to include or exclude those points on the *Heat Source Calibration Parameters* and *Melt Pool Dimensions* tabs on the **Visualizations** window. Furthermore, if a simulation is not used in a 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** 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 Thermal Gradient vs Solidification Rate								
Export Result Files...								
^ Hide Table								
#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp. width (μm)	Exp. depth (μm)	Sim. width (μm)	Sim. depth (μm)	Use
1	60.0	2400.0	0.025	36.8421	9.4444	37.2076	9.7557	<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 45: 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 column. Rows 2-4 were not included in this simulation and are greyed out. When a checkbox is not selected, the results are also not included if you choose to export the result files.

## 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 46](#) for an example of the available parameters for a **Double ellipsoidal Heat Source**.

	R2	Below P/v	0.025	Above P/v	0.25	Plot
Absorptivity:	Linear function of P/v	0.96	Constant	Constant		<input checked="" type="checkbox"/>
ar:	Constant function of P/v	0.00	Constant	Constant		<input checked="" type="checkbox"/>
af:	Constant function of P/v	0.00	Constant	Constant		<input checked="" type="checkbox"/>
b:	Constant function of P/v	0.00	Constant	Constant		<input checked="" type="checkbox"/>
c:	Cubic function of P/v	1.00	Constant	Extrapolate linear		<input checked="" type="checkbox"/>

Figure 46: 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 (i.e. *Gaussian*, *Double ellipsoidal*, or *Conical*) selected on the **AM Calculator**:

- *Gaussian*: **Beam radius**. Then based on the **Absorptivity** setting—**Calculated (prefactor calibrated)** or **Constant (calibrated)**—also **Absorptivity prefactor** plus additional options.
- *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 Heat Source Settings](#)" on page 123.

For all heat source parameters, you can also choose to select:

- A **Constant function** value i.e. the average value from all experiments or a function of energy density, which could also be a **Linear function**, **Cubic function**, or a **Sigmoid function**.

- A **2d cubic function P and v** of *Power (P)* and *Scanning speed (v)*.
- 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.

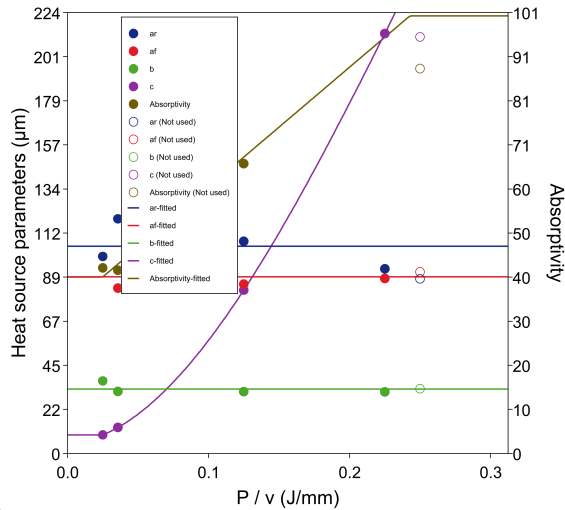


Figure 47: From example AM\_06a, the Heat Source Calibration Parameters plot for the double ellipsoidal calibration.

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** column.

### Melt Pool Dimensions Plot

A **Melt Pool Dimensions** plot shows 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.

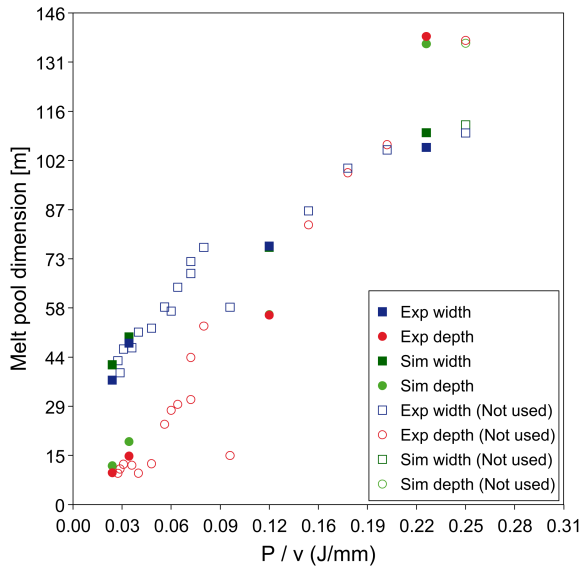


Figure 48: From example AM\_06a, the Melt Pool Dimensions plot for the double ellipsoidal calibration.

### 3D Plot and Plot Over Line

Use a **3D Plot** to visualize all the experiments for which the calibration is performed. To show the 3D plot for a certain experiment, select the corresponding experiment from the **Heat Source Calibration Experiments** menu. The configuration in the **3D Plot** works in the same way as for a single point steady-state calculation.

For **Plot Over Line**, plot the data along a line for the selected experiment. To show the plot over line for a specific experiment, select the corresponding experiment from the **Heat Source Calibration Experiments** menu. The configuration in the **Plot Over Line** works in the same way as for a single point steady-state calculation.



"About the AM Plot Types" on page 64

---

## Visualizing Batch Calculations in the AM Module

---

This topic describes the **AM Calculator** visualizations related to the **Batch** calculations for a **Steady-state** simulation. There are further settings and visualization possibilities that are available when you are working on the **Plot Renderer**.



To import data for use in batch calculations, it is recommended you review "AM Calculator Experiment File Requirements" on page 134. Also review "Working with AM Visualizations" on page 164, which provides an overview of how most of the windows are connected.

It is recommended to open and run one of the examples to best understand the wide variety of visualization outcomes.

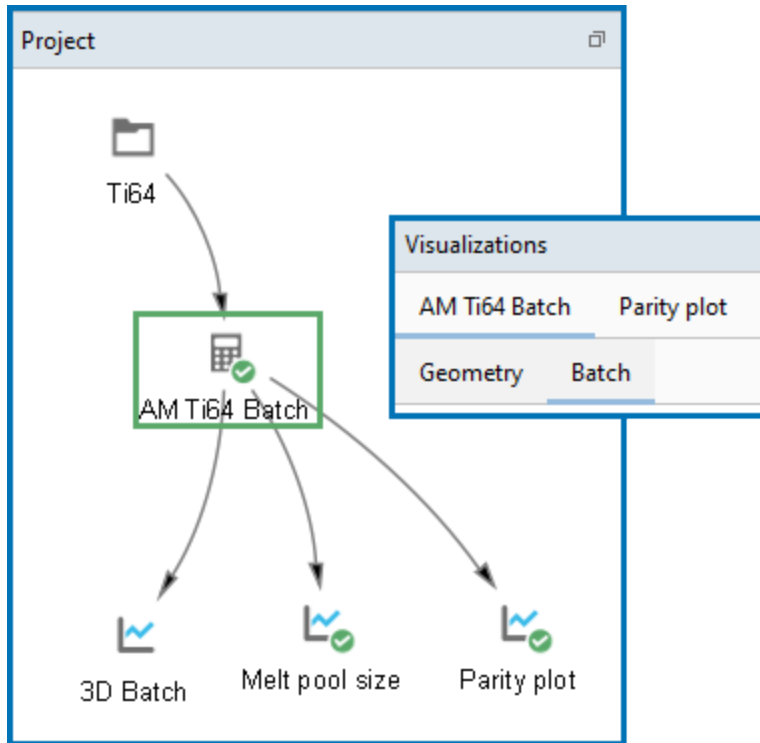


"AM\_07: Batch Calculations for an IN718 Alloy" on page 241, "AM\_08b: Batch Calculations for a Ti64 Alloy" on page 253, and "AM\_09b: Batch Calculations for an SS316L Alloy" on page 264.

### Working with AM Calculator Batch Experiment Data

From example AM\_08b, where the general behavior related to importing data to the table is the same as other places in Thermo-Calc.

In the **Project** window, click the relevant **AM Calculator** node (in this example renamed to **AM Ti64 Batch**) to work on the **Configuration** window and preview the table data via a **Batch** tab on the **Visualizations** window.



During the set up of a calculation, the *Batch Experiment Data* is imported into the **AM Calculator Configuration** window, where you can review the data and choose to include or exclude data points by selecting and deselecting the checkboxes as needed in the **Use** column (see [Figure 49](#)). At the same time, you can observe the change as this is updated in the **Visualizations** window (see [Figure 50](#)). Data can also be entered directly into the table.

Calculation Type							
<input type="radio"/> Single Point <input type="radio"/> Heat Source Calibration <input checked="" type="radio"/> Batch <input type="radio"/> Grid							
Batch Experiment Data							
Experiment file					delimiter	Comma	
#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp.width (μm)	Exp.depth (μm)	Use	
1	50.000000	500.000000	0.100000	67.379660	16.058920	<input checked="" type="checkbox"/>	
2	50.000000	750.000000	0.066667	50.318180	11.830310	<input checked="" type="checkbox"/>	
3	50.000000	1000.000000	0.050000	47.357410	9.654352	<input checked="" type="checkbox"/>	
4	50.000000	1200.000000	0.041667	45.508170	5.108316	<input checked="" type="checkbox"/>	
5	100.000000	500.000000	0.200000	118.025200	44.462060	<input checked="" type="checkbox"/>	
6	100.000000	750.000000	0.133333	98.663500	32.019900	<input checked="" type="checkbox"/>	
7	100.000000	1000.000000	0.100000	75.846620	24.027610	<input checked="" type="checkbox"/>	
8	100.000000	1200.000000	0.083333	72.846300	21.534230	<input checked="" type="checkbox"/>	
9	150.000000	500.000000	0.300000	145.364300	101.268300	<input checked="" type="checkbox"/>	
10	150.000000	750.000000	0.200000	135.499000	72.058650	<input checked="" type="checkbox"/>	

Figure 49: The experimental data used for the Batch calculation for the single track experiments of Ti64 is imported to the AM Calculator table where you can review the data points and include or exclude as needed using the checkboxes.

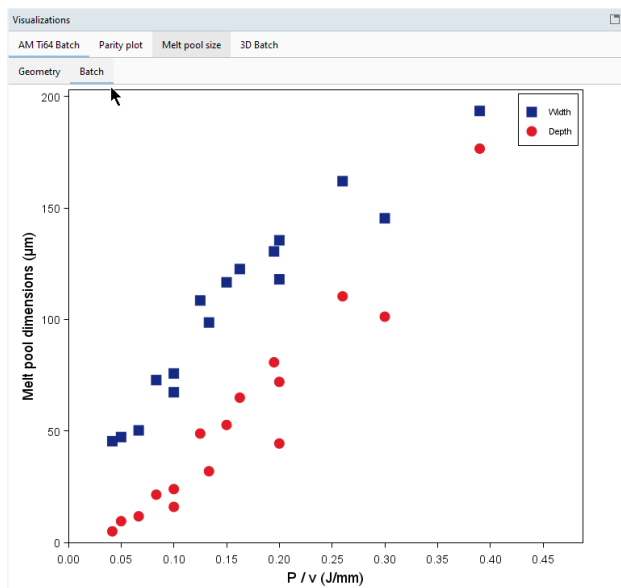
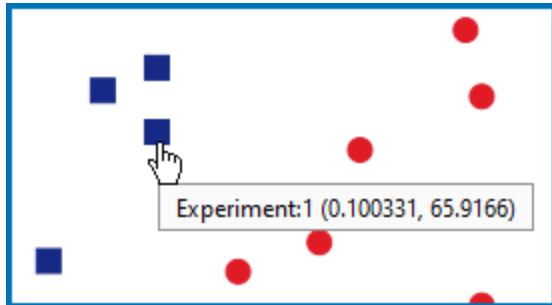


Figure 50: The visualization of the experimental melt pool dimensions are shown as a function of the energy density  $P/v$ . You can adjust the selected points in the Batch Experiment Data table and watch the updates dynamically in this window.

## Including or Excluding Data Points

The following discusses visualization options available on the **Configuration** windows for both the **AM Calculator** and **Plot Renderer**.

You can click or hover over any point on the **Batch** preview to display the label, which also includes the corresponding table row (i.e. *Experiment*) number for the point.



Use the table's **Use** 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*).

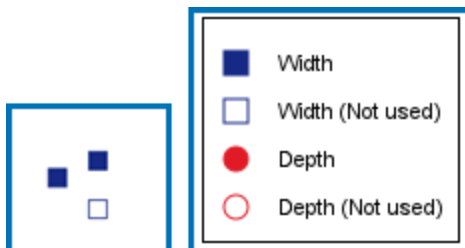
Batch Experiment Data						
Experiment file						delimiter
#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp.width (μm)	Exp.depth (μm)	Use
1	50.000000	500.000000	0.100000	67.379660	16.058920	<input checked="" type="checkbox"/>
2	50.000000	750.000000	0.066667	50.318180	11.830310	<input checked="" type="checkbox"/>

Figure 51: From example AM\_08b, on the AM Calculator, the checkbox for row (experiment) #1 can be deselected to exclude the data point in the calculations.

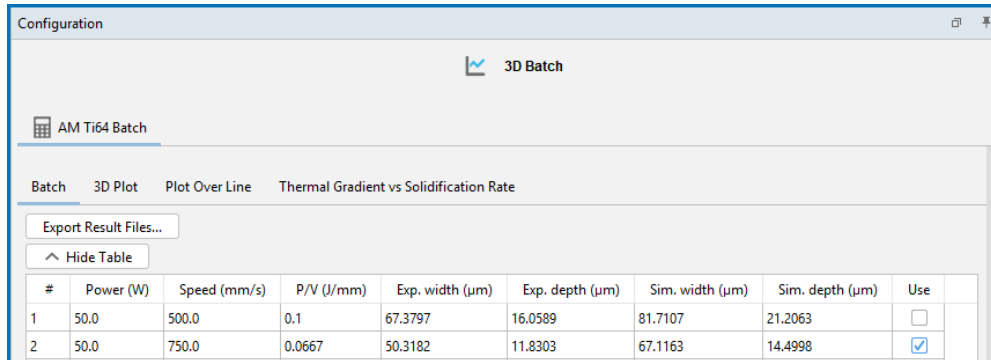
On the **Batch** tab on the **Visualizations** window, the preview shows the **P/v** on the X-axis and **Melt pool dimensions** on the Y-axis:

- **Width** uses blue squares (solid color means used, hollow with same color outline means not used)
- **Depth** uses red circles (solid color means used, hollow with same color outline means not used)

When you toggle the **Use** checkbox selection, the preview dynamically updates in the **Visualizations** window to indicate which data points are used (filled) or not used (hollow), both on the preview and in the legend.



The points that are selected in the *Batch Experiment Data* table are used in the simulation results and are later labeled **Exp.Width** and **Exp.Depth** in the **Plot Renderer Configuration** table (see [Figure 52](#)). Then the calculated results are added to new columns labeled **Sim.Width** and **Sim.Depth**.



#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp. width (µm)	Exp. depth (µm)	Sim. width (µm)	Sim. depth (µm)	Use
1	50.0	500.0	0.1	67.3797	16.0589	81.7107	21.2063	<input type="checkbox"/>
2	50.0	750.0	0.0667	50.3182	11.8303	67.1163	14.4998	<input checked="" type="checkbox"/>

*Figure 52: From example AM\_08b, this is the Batch tab on the Plot Renderer Configuration window where row (experiment) #1 data point is excluded from the results after selecting it on the AM Calculator. When a checkbox is not selected here, the results are also not included if you choose to export the result files. The details about Plot Renderer settings are in the topic "AM Calculator: Plot Renderer Settings" on page 147.*

## Working with the Batch Table on the Plot Renderer

After the **Batch Experiment Data** set up is completed on the **AM Calculator** and the simulation run, then on the **Plot Renderer Configuration** window you can start to work with the data you want to plot. Also see [Including or Excluding Data Points](#) to understand the interaction between these two tables.



For additional **Plot Renderer** settings (and other visualization options for plotting) go to "Batch Calculations" on page 155.

### Navigating the Batch Table

You may have already included or excluded data points earlier when previewing the data. Use the **Show Table** or **Hide Table** buttons to view the included or excluded data and continue to fine tune the plot.

On the **Batch** tab the table includes these columns:

- The original experiment data for **Power (W)** and **Speed (mm/s)**, plus the **P/v (J/mm)**.
- The original experiment data for **Width** and **Depth**, now with a prefix **Exp.** e.g. **Exp.width (μm)**, to indicate this is the experimental data.
- The calculated **Width** and **Depth** points, with the prefix **Sim.**, e.g. **Sim.width (μm)**, 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** column has a checkbox available to continue to include or exclude those points on the plots. When a checkbox is not selected, the results are also not included if you choose to export the result files.



Only points that were originally included in the simulation can be selected or deselected from the **Use** 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.

# Additive Manufacturing (AM) Module Examples



Without a license, the Additive Manufacturing (AM) Module examples are available to open and view the settings. However, no calculations can be done.



The first time you open an Additive Manufacturing (AM) Module example after a new Windows installation of Thermo-Calc, it copies various files that can take some time to download. Then the example itself can take further time to open where it seems that the program is frozen. After the first instance, the other examples should not take as long to load.

In this section:

AM_01: Transient Simulation of a Single Track .....	196
AM_02: Transient and Steady-state Simulations of a Single Track .....	202
AM_03: Steady-state Simulations .....	208
AM_04: Scheil Transient Steady-state .....	213
AM_05: Using AM Calculator Probe Data with the Diffusion Module (DICTRA) .....	220
AM_06a: Calibrating a Heat Source for a 316L Steel .....	229
AM_06b: Using the Calibrated Heat Source for a 316L Steel .....	236
AM_07: Batch Calculations for an IN718 Alloy .....	241
AM_08a: Grid Calculation for a Ti64 Alloy .....	247
AM_08b: Batch Calculations for a Ti64 Alloy .....	253
AM_09a: Grid Calculation for an SS316L Alloy .....	259
AM_09b: Batch Calculations for an SS316L Alloy .....	264
AM_10: CET Transition in an IN718 Alloy .....	270
AM_11: Comparing Single Tracks Printed on Casted and LPBF Substrates .....	275

AM_12: Using AM Calculator Probe Data with the Precipitation Module (TC-PRISMA) .....	285
AM_13: Using the Core-ring Beam Shape .....	293
AM_14: Using the Top-hat Beam Shape .....	301
AM_15: Electron Beam Melting in an IN718 Alloy .....	309

## 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.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: `AM_01_Transient_DE_No_Marangoni.tcu`



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) are available to all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO (demonstration) mode. However, the AM Module is not available with the Educational version of Thermo-Calc. Search the help for Available Options with the Additive Manufacturing (AM) Module. See "[Available Options](#)" on page 10 for details.

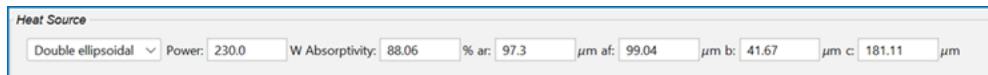
### Material Properties

These properties are the same for both AM\_01 and AM\_02 [2021Gra].

- 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.

## 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 53.



The screenshot shows the 'Heat Source' configuration window. It features a dropdown menu set to 'Double ellipsoidal'. Below it, several input fields are filled with numerical values: Power: 230.0 W, Absorptivity: 88.06 %, arc: 97.3 μm, af: 99.04 μm, b: 41.67 μm, and c: 181.11 μm.

Figure 53: 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 54. 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.

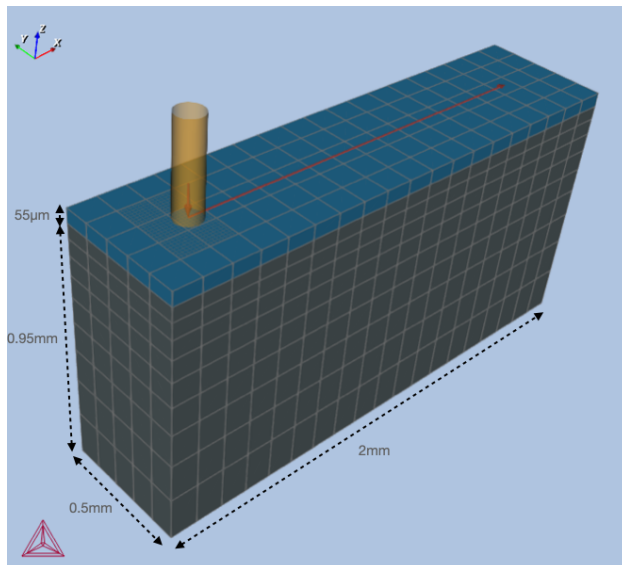


Figure 54: The geometry used for the Additive Manufacturing (AM) Module examples AM\_01 and AM\_02.

Two probes monitor the temporal evolution of temperature at positions shown in [Figure 55](#). The results can be presented by tabulated data or as 2D graph plots.

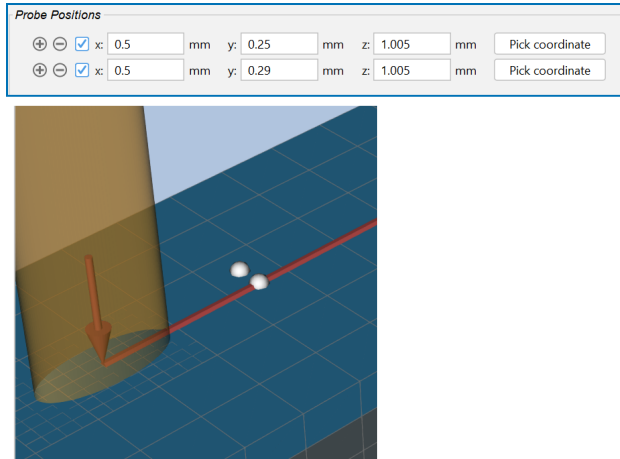


Figure 55: Probe positions for Probes P1 and P2, where P1 is placed exactly on the scanning track while P2 is placed at a distance of 0.04 mm from the track, as shown in a close up of the probes on the Visualizations window.

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it takes at least 30 minutes for the calculations to complete.

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 needed.
- 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.

In the surface plot it shows the 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.

<i>Melt Pool Dimension</i>	<i>Size (mm)</i>
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.

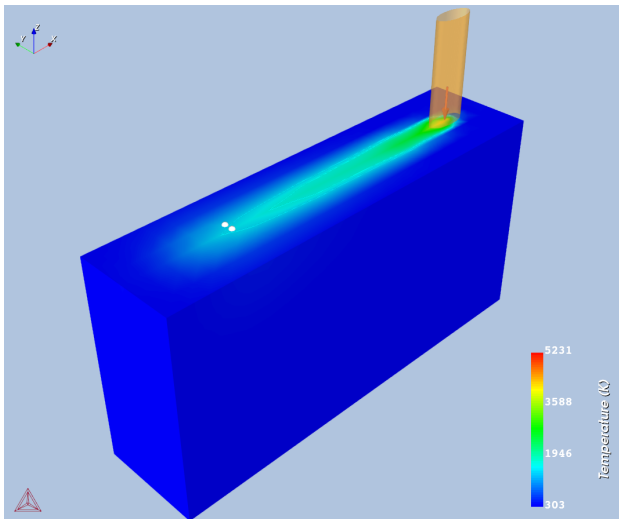


Figure 56: Surface plot of temperature distribution after the single track scan is completed. Probes P1 and P2 can also be seen on the top surface.

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. The next figure shows the temperature slices plotted at cross-sections orthogonal to the scanning direction.

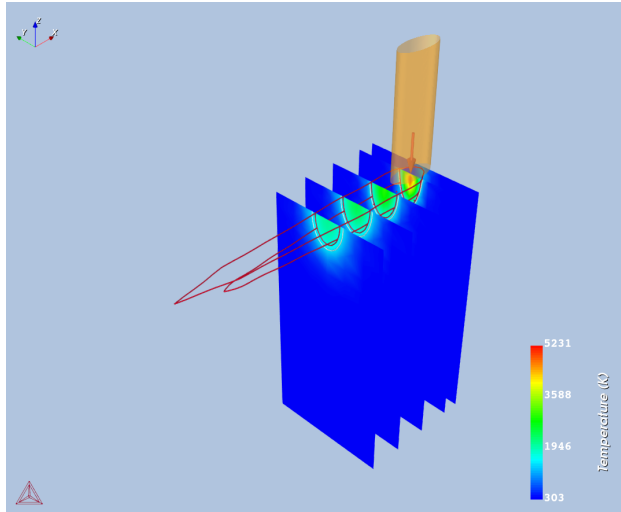


Figure 57: Slices of the computational domain, orthogonal to the scanning direction, showing temperature distribution at different cross-sections. Red tubes show the contour of the melt pool (at liquidus temperature).

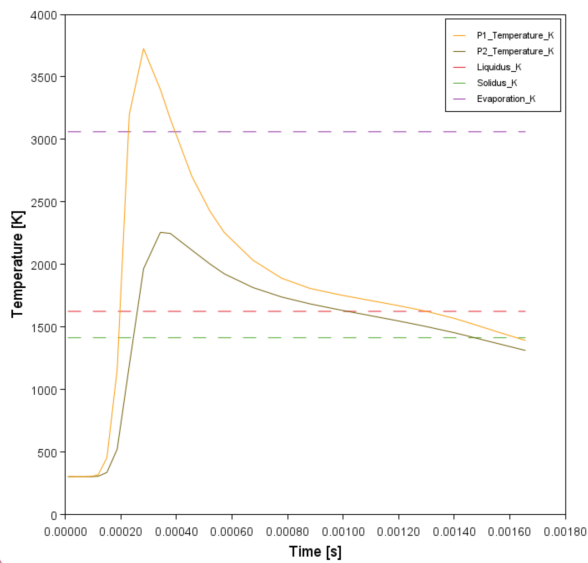


Figure 58: 2D probe plot showing temperature evolution as a function of time at probes P1 and P2.

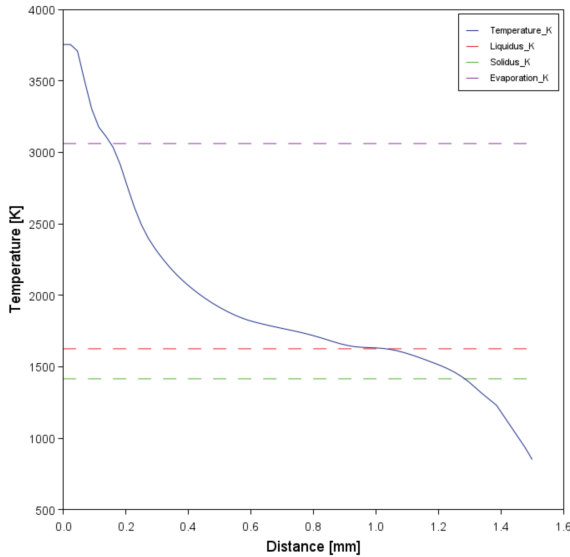


Figure 59: 2D plot over line showing temperature in the tail of the melt pool as a function of distance plotted along the scanning track.

## 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).

## 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.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: AM\_02\_TransientSS\_DE.tcu



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) are available to all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO (demonstration) mode. However, the AM Module is not available with the Educational version of Thermo-Calc. Search the help for Available Options with the Additive Manufacturing (AM) Module. See "[Available Options](#)" on page 10 for details.

## Material Properties

These properties are the same for both AM\_01 and AM\_02 [2021Gra].

- 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.

## 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 60](#).

The screenshot shows a 'Heat Source' configuration window with the following parameters:

Parameter	Value	Unit
Model Type	Double ellipsoidal	
Power	230.0	W
Absorptivity	88.06	%
arc	97.3	μm
a	99.04	μm
b	41.67	μm
c	181.11	μm

*Figure 60: 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 61](#). 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.

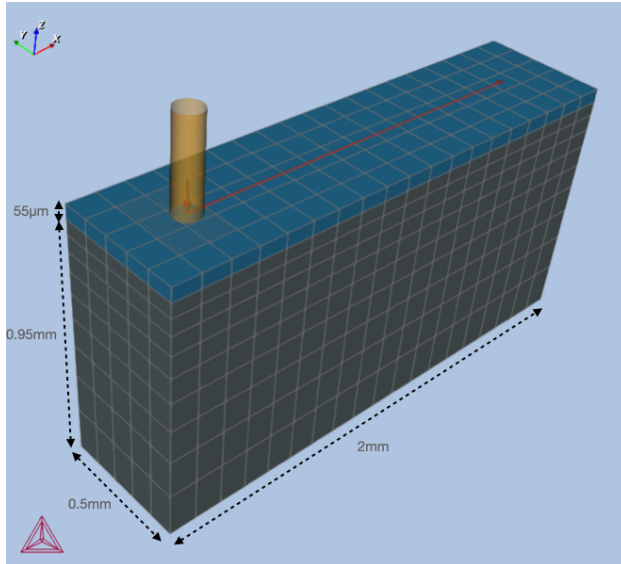


Figure 61: The geometry used for the Additive Manufacturing (AM) Module examples AM\_01 and AM\_02.

Two probes monitor the temporal evolution of temperature at positions shown in Figure 62. The results can be presented by tabulated data or as 2D graph plots.

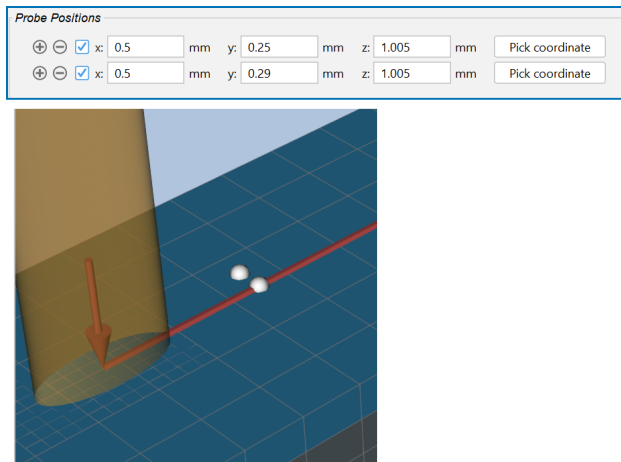


Figure 62: Probe positions for Probes P1 and P2, where P1 is placed exactly on the scanning track while P2 is placed at a distance of 0.04 mm from the track, as shown in a close up of the probes on the Visualizations window.

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects,

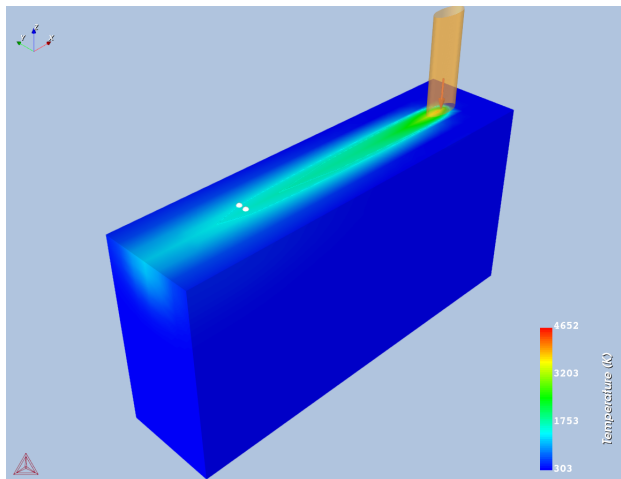
you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.

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.

The figure below shows the surface plot of temperature distribution after the single track scan 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 63: 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 64](#).

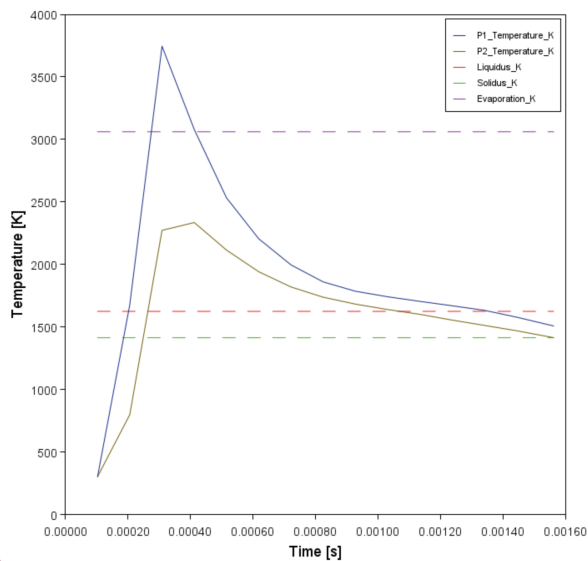


Figure 64: 2D probe plot showing temperature evolution as a function of time at probes P1 and P2.

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 65](#).

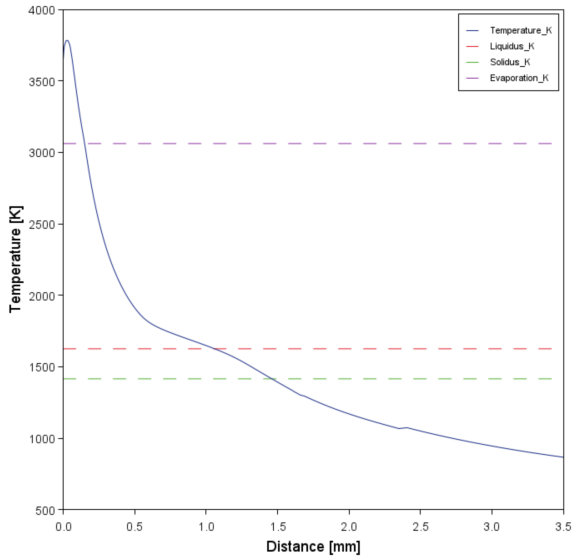


Figure 65: 2D plot over line showing temperature in the tail of the melt pool as a function of distance plotted along the scanning track.

## 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).

## 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**.

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 powder and fluid flow in the melt pool due to Marangoni effect.

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.



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 and License Information***

- Folder: **Additive Manufacturing**
- File name: `AM_03_Steady.tcu`



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) are available to all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO (demonstration) mode. However, the AM Module is not available with the Educational version of Thermo-Calc. Search the help for Available Options with the Additive Manufacturing (AM) Module. See "Available Options" on page 10 for details.

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.

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.

## Results Discussion

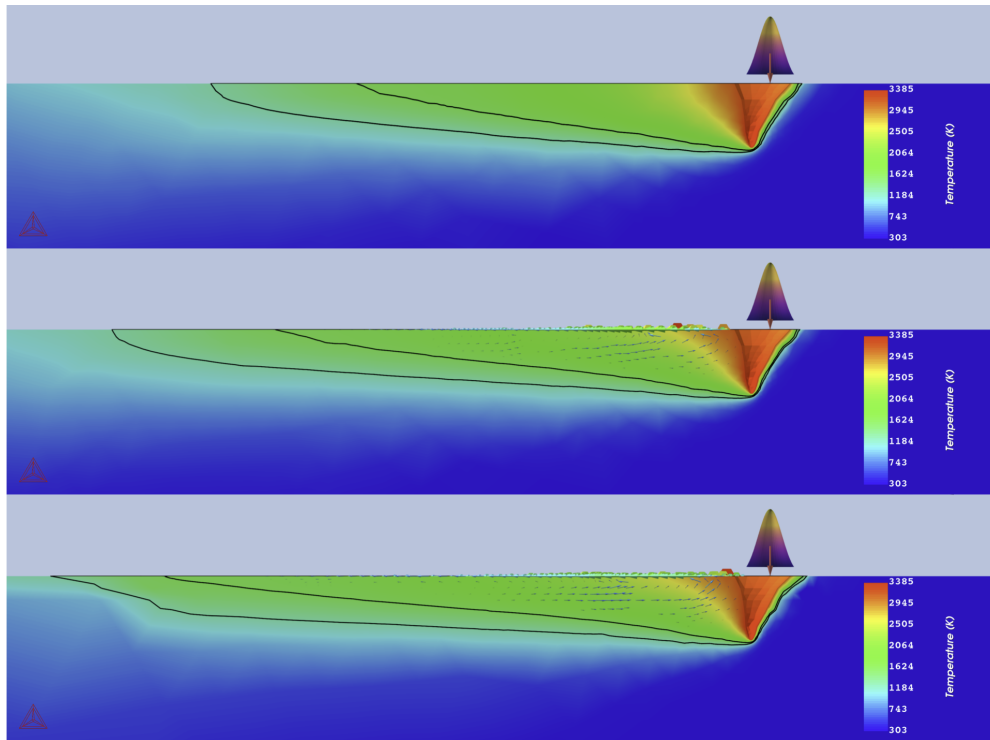


Figure 66: A comparison of the melt pool depth for different simulation conditions; (top) without fluid flow, (middle) with fluid flow in the melt pool and (bottom) with both fluid flow and different material properties for the powder layer. Isocontours show the boundaries of (inner) melt pool and (outer) 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	0.115	0.111	0.736
With fluid flow	0.147	0.111	0.869
Separate powder material properties	0.194	0.111	1.064

Figure 66 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 67.

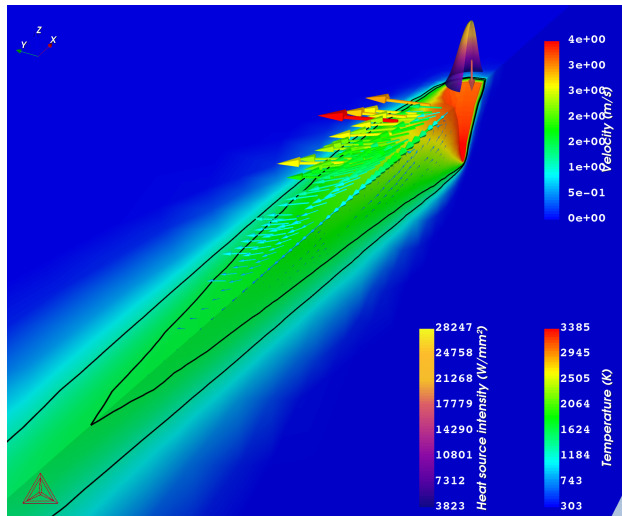


Figure 67: Vectors showing flow field in the melt pool due to gradient in surface tension. The colors of the vectors represent the magnitude of the velocity. Surface colormap shows the temperature distribution.

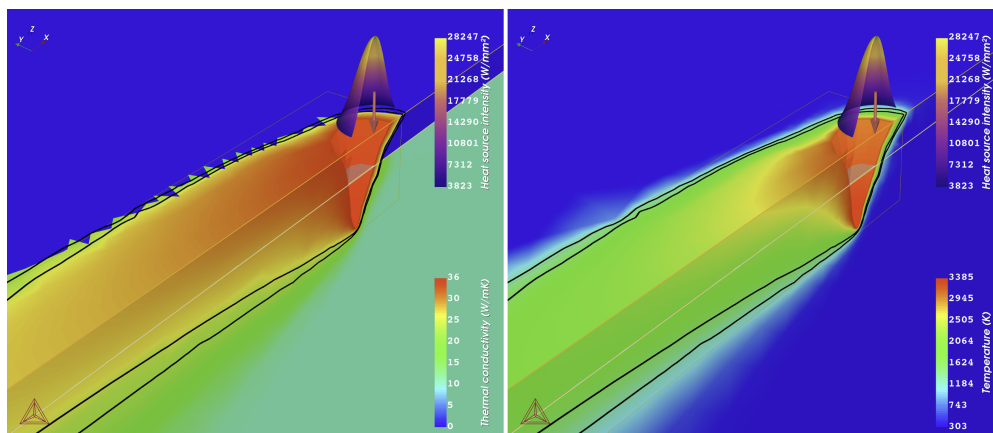


Figure 68: Surface colormap of temperature dependent thermal conductivity showing different values in powder and liquid/solid and (right) the corresponding temperature distribution.

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 68) 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.

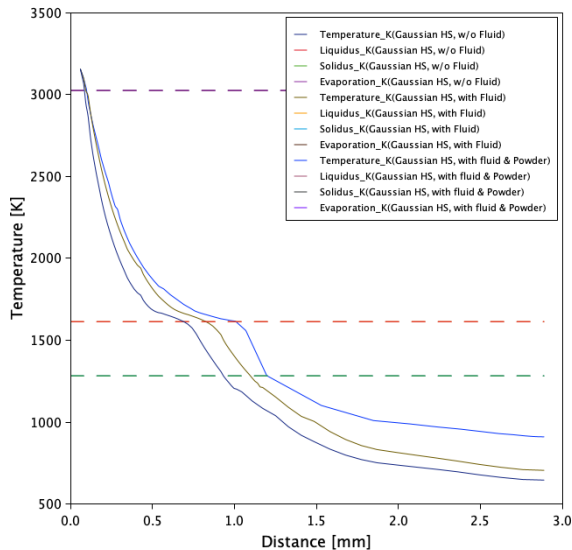


Figure 69: Temperature distribution along the top surface of the melt pool from the heat source position toward the tail. The plot compares three conditions: no fluid flow (solid blue line), fluid flow only (dashed green line), and combined powder and fluid flow (dash-dotted red line).

## 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.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: AM\_04\_Scheil\_TransientSS.tcu



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.

### Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it takes at least 30 minutes for the calculations to complete.

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 100 W and scanning speed of 600 mm/s. The beam radius in this example is taken as 100  $\mu\text{m}$ . For the geometry, the steady-state model only needs the height of the solid substrate as the input, which is taken as 2.0 mm. The solid substrate is covered with a powder layer of thickness 55  $\mu\text{m}$ . This example also takes into account the effect of fluid flow due to Marangoni forces inside the melt pool.

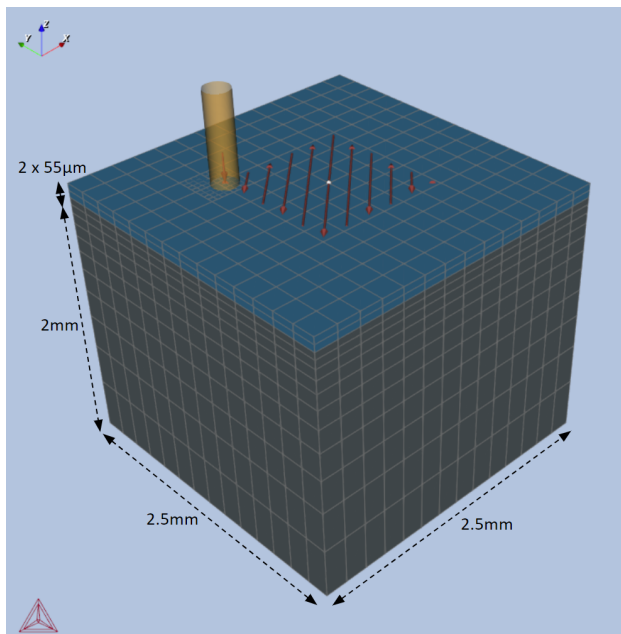


Figure 70: Geometry definition for Transient Multilayer. The scanning strategy shows the scanning pattern on the top most layer.

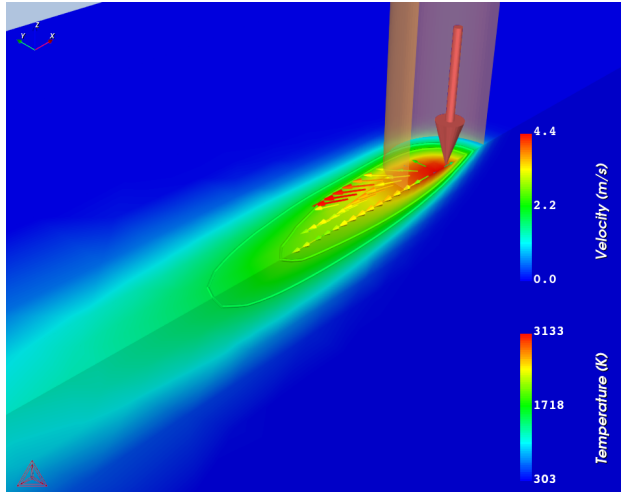


Figure 71: Temperature distribution around the melt pool and mushy zone for the Steady-state case. Velocity vectors representing the fluid flow inside the melt pool can also be seen.

## Transient Single Track

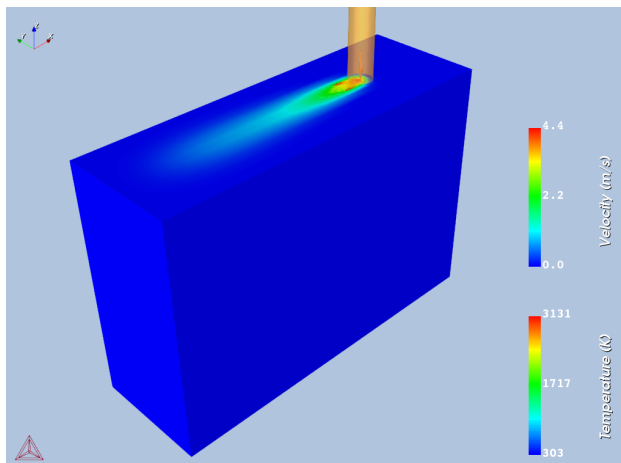


Figure 72: 3D Surface colormap showing temperature distribution at the end of single track transient simulation. The velocity vectors around the heat source show the direction of fluid flow in the melt pool.

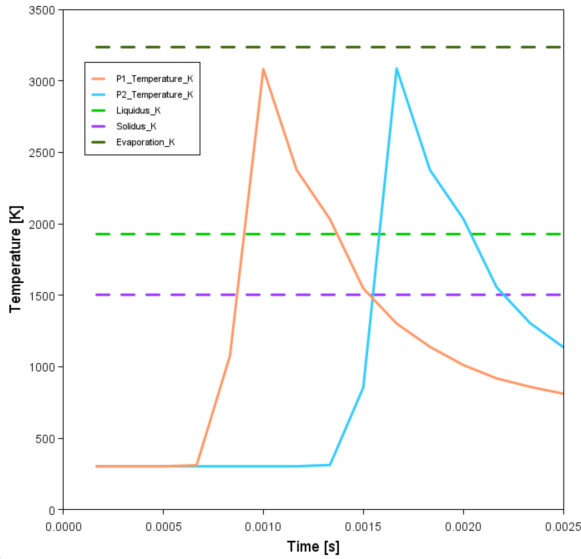


Figure 73: 2D Plot showing temperature as a function of time from Probe 1 (orange) and Probe 2 (blue) after the single track scan is completed.

## Transient Multilayer

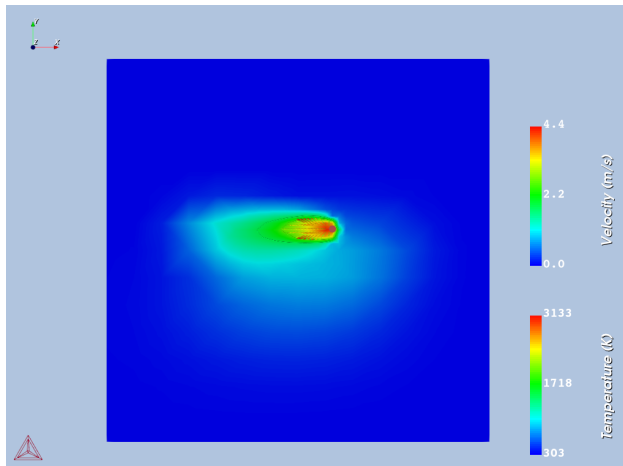


Figure 74: Temperature distribution on the top surface of the workpiece during scanning of the first layer.

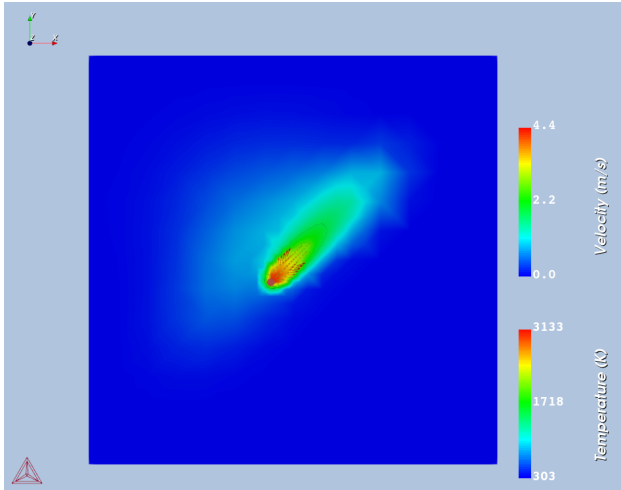


Figure 75: Temperature distribution on the top surface of the workpiece during scanning of the second layer.

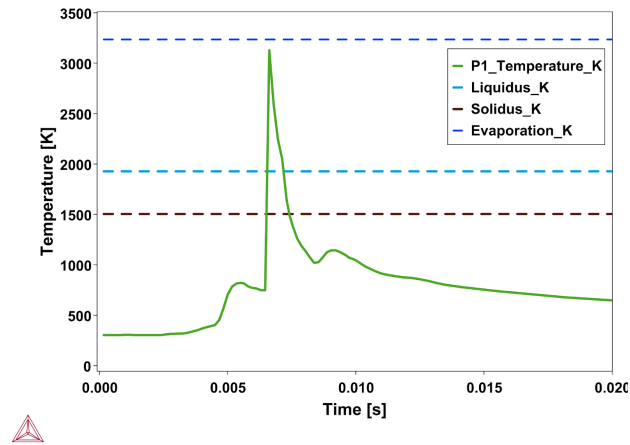


Figure 76: 2D Plot showing temperature as a function of time from Probe 1 after scanning the first layer.

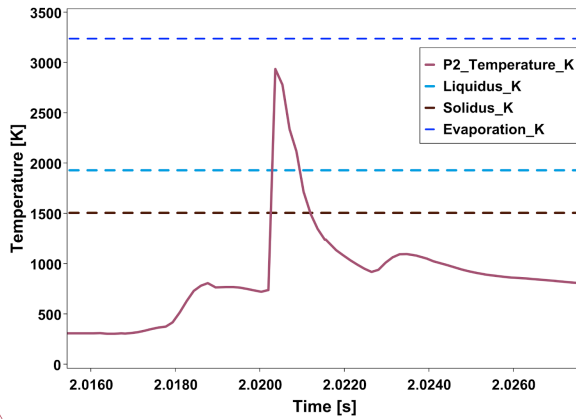


Figure 77: 2D Plot showing temperature as a function of time from Probe 2 after scanning the second layer.

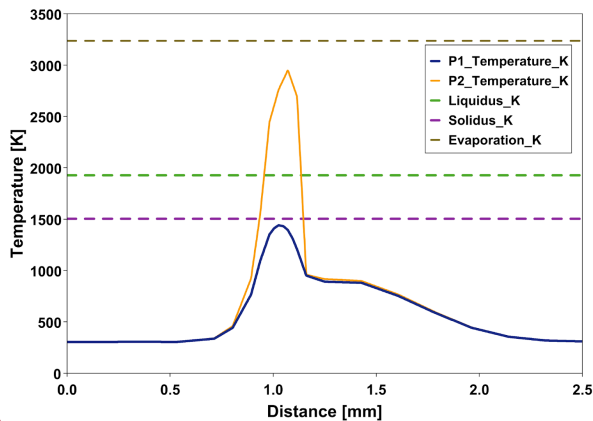


Figure 78: 2D Plot Over Line showing temperature profiles plotted along the lines, parallel to the width of the geometry placed on the top of the first layer (P1) and the second layer (P2).

## 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\_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.

### *Project File and License Information*

- Folder: **Additive Manufacturing**
- File name: `AM_05_AM_Probe_to_Diffusion.tcu`



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).

### **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 79](#).

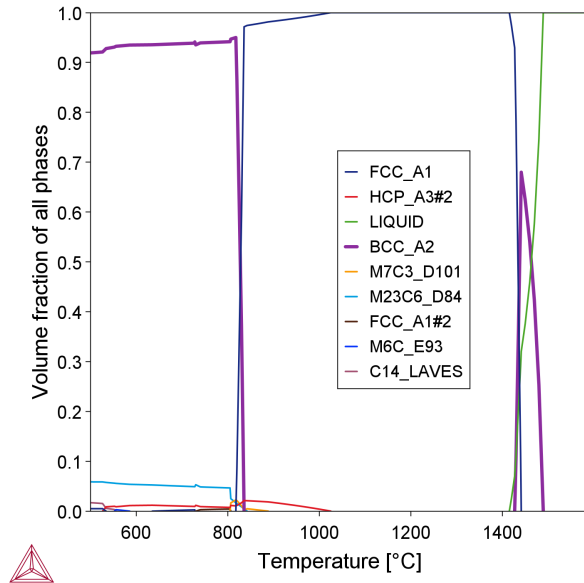


Figure 79: Equilibrium property diagram calculated in Thermo-Calc of the tool steel showing  $\delta$ -ferrite would be the first solid phase to form during solidification according to equilibrium.

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 80](#) (or refer to the example if you have it open) to see the layout.

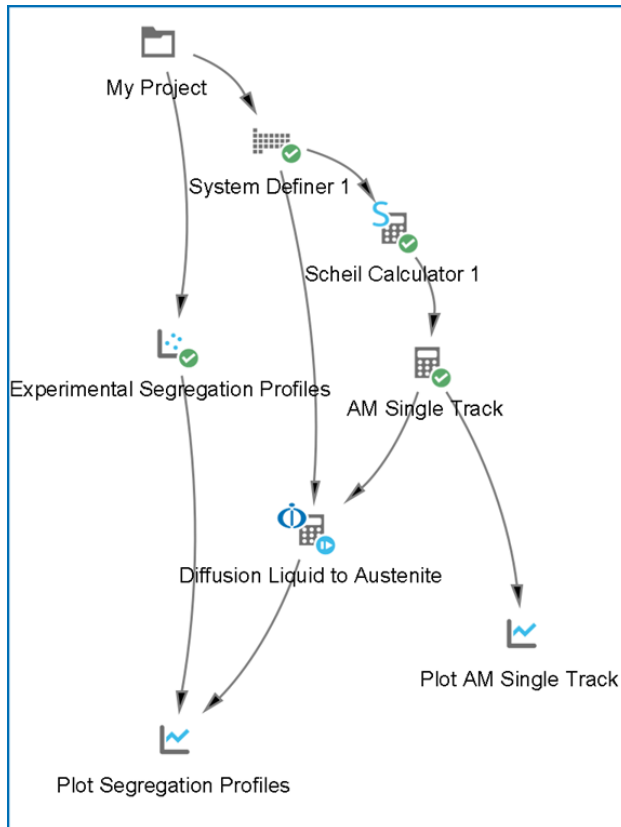


Figure 80: Project tree of the example containing activities from the default Additive Manufacturing template with additional Diffusion Calculator and Experimental File Reader nodes added, where the segregation profiles datasets are pulled from the Experimental File Reader.

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.

Configuration

AM Single Track

Conditions Material Properties Options

Steady-state
  Transient
  Transient with heat source from Steady-state

**Global Settings**

Gas pressure: 100000.0 Pascal

Temperature unit: Celsius

Base plate temperature: 200.0

Ambient temperature: 23.0

Fluid flow including Marangoni effect:

Use separate material properties for powder:  Powder density: 80.0 % of solid materi...

**Geometry**

Height: 2.0 mm Width: 1.0 mm Length: 4.0 mm

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.

**Heat Source**

Double ellipsoidal Power: 260.0 W Absorptivity: 42.0

**Scanning Strategy**

Scanning speed: 900.0 mm/s

Layer thickness: 30.0 μm

Pattern: Single track Margin: 1.0 mm Number of layers:

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).

**Probe Positions**

x: 1.3 mm y: 0.5 mm z: 2.03 mm

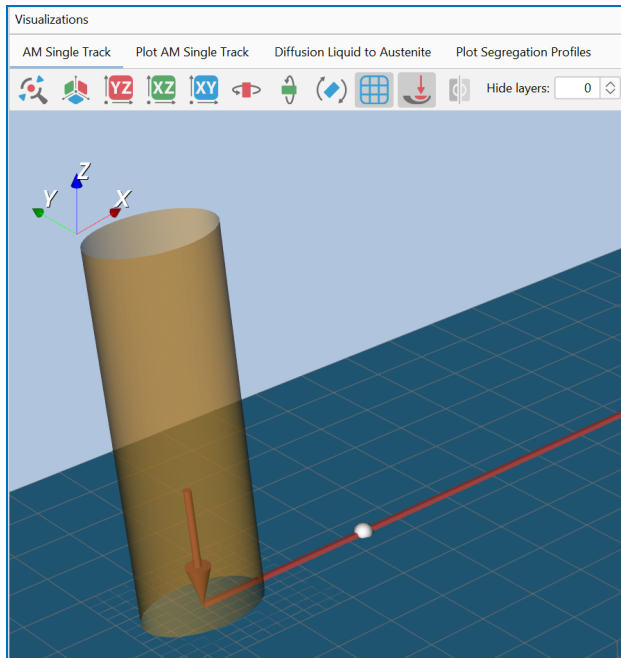


Figure 81: Close up of the probe point added to the geometry for the single track AM simulation.

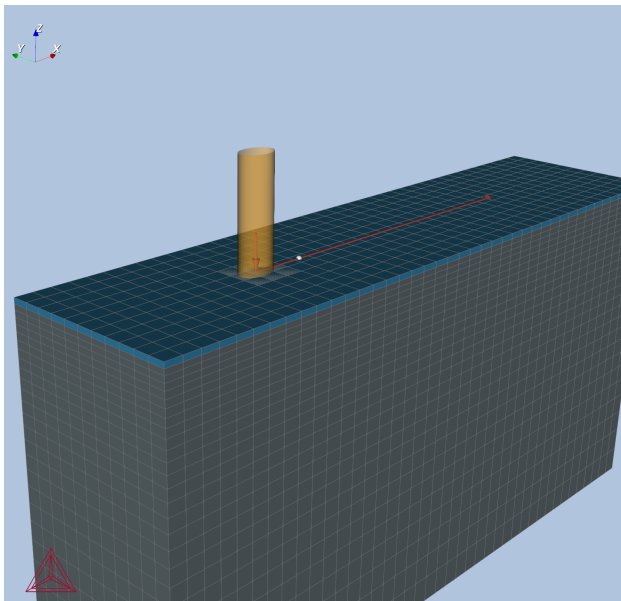
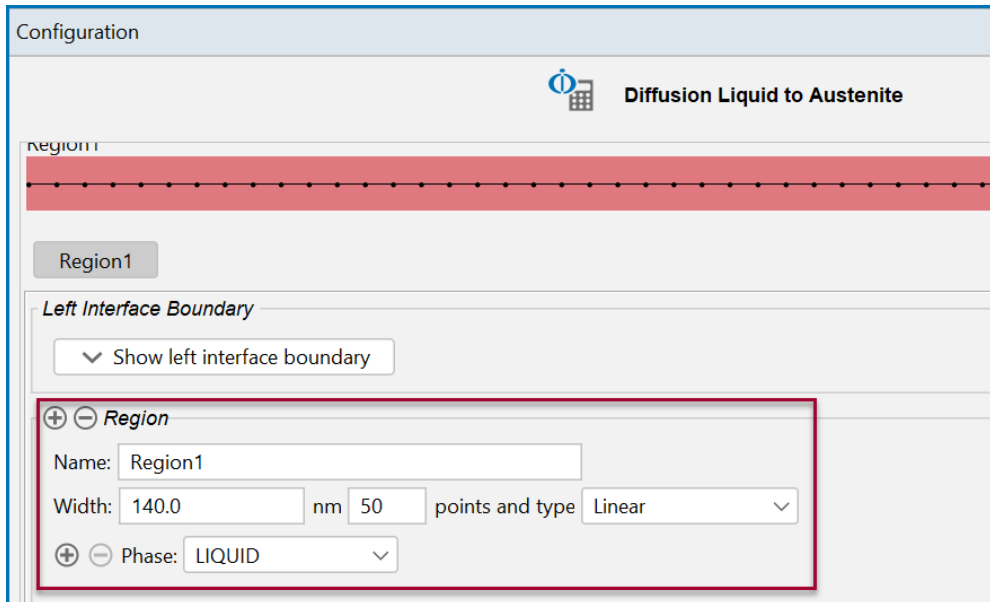


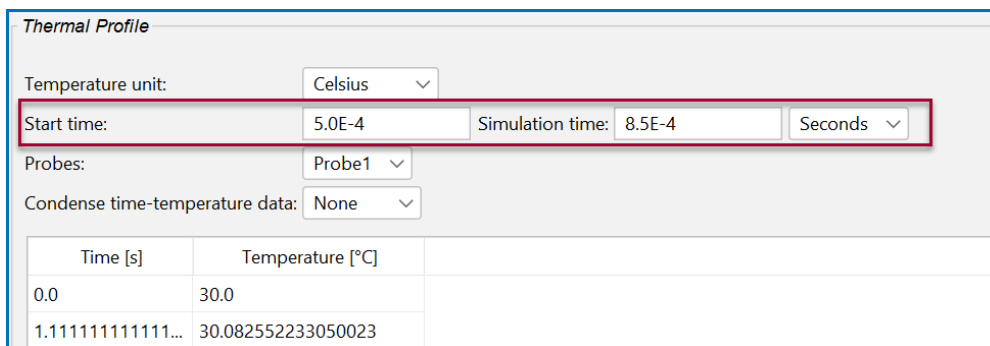
Figure 82: The full geometry set up for the single track AM simulation with a probe point added to the middle of the scan. Open the example to better see the set up and to try working within the window.

## 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.



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 83](#). 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.



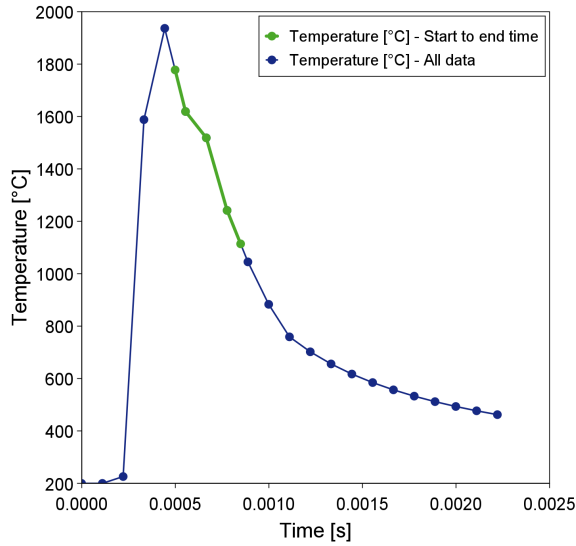


Figure 83: Time temperature profile of the probe in the AM simulation. The green curve show the part of the profile used during the diffusion simulation with the Diffusion Calculator.

The calculated segregation profiles are finally compared with the STEM-EDS line scans as shown in Figure 84.

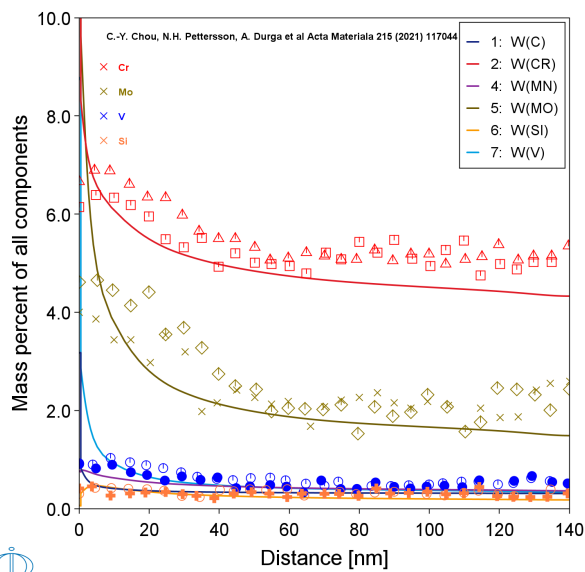


Figure 84: The simulated segregation profile with overlaid experimental STEM-EDS line scans from C.-Y. Chou et al. [2021Cho].

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it can take over three hours to complete the calculations.

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 81](#) and [Figure 82](#) 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 83](#).
- **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 84](#).

## 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" on page 236.

### Project File and License 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) are available to all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO (demonstration) mode. However, the AM Module is not available with the Educational version of Thermo-Calc. Search the help for Available Options with the Additive Manufacturing (AM) Module. See "Available Options" on page 10 for details.

### Material Properties

- SS316L: Fe-17.0Cr-12.0Ni-2.5Mo-0.03C Mass percent
- Database: TCFE13
- The material properties are precalculated, and stored as a built-in material library with the Additive Manufacturing (AM) Module.

---

## 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 this AM Calculator, other settings include:

- **Constant (calibrated)**, which means the absorptivity is not a function of temperature or angle, and a constant value is calibrated for each experiment.
- The **Beam radius** is set to **Calibrated**, which means that both beam radius and absorptivity are calibrated.
- **Calibrate for** is set to **Width and depth**, which means that the experimental values for both width and depth are used for calibration.

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

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects,

you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it can take about an hour to complete the calculations.

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](#)" on page 178 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 85](#).

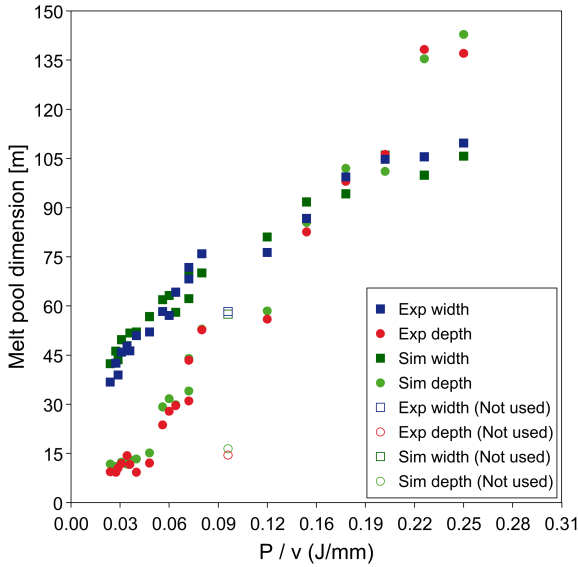


Figure 85: A comparison of melt pool dimension for Gaussian heat source calibration. The "Not used" points shown in the legend are because a checkbox is not selected in the 'Use' column for row 10 on the Plot Renderer Configuration window.

The resulting calibrated heat source parameters for **AM Calibration Gaussian**, are shown in Figure 86.

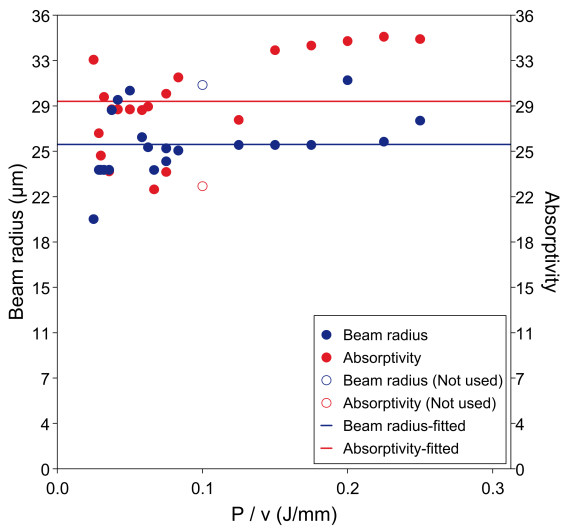


Figure 86: Calibrated heat source parameters for Gaussian heat source calibration.

In order to obtain the expressions for **Absorptivity** and **Beam radius**, one of the experiments (*Experiment # 10*) is not included. This is because *Experiment # 10* is an outlier and you can see that the values for *Exp width* and *Exp depth* (hollow square) then the *Sim width* and *Sim depth* (hollow circle) are excluded in [Figure 85](#).

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.

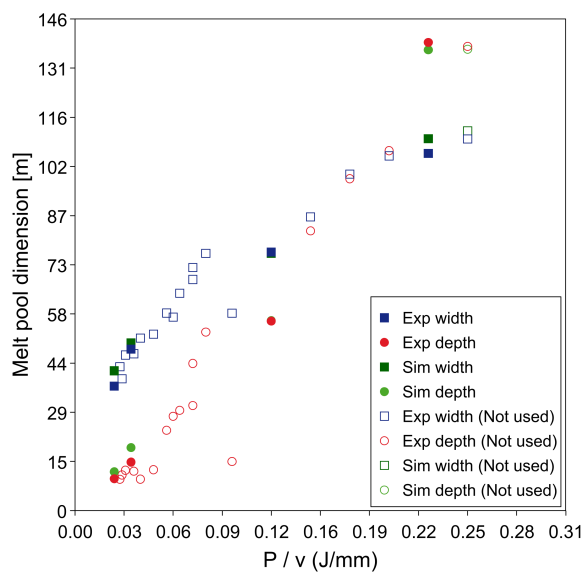


Figure 87: A comparison of melt pool dimension for double ellipsoidal heat source calibration.

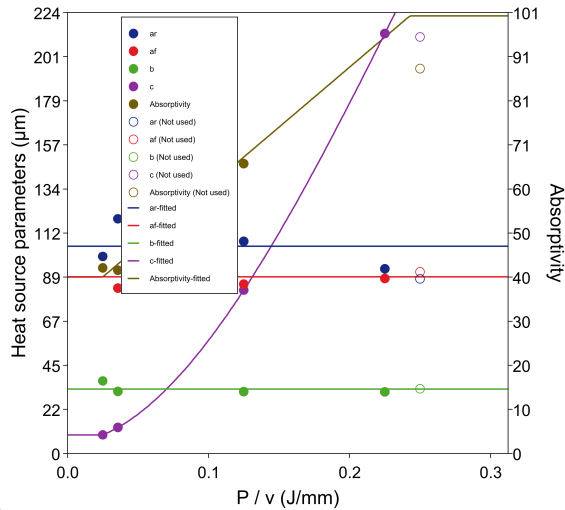


Figure 88: Calibrated heat source parameters for double ellipsoidal heat source calibration.

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).

## 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\_06b: Using the Calibrated 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].



"AM\_06a: Calibrating a Heat Source for a 316L Steel" on page 229

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: AM\_06b\_Use\_Calibrated\_Heat\_Sources\_316L.tcu



Other than an Additive Manufacturing (AM) Module license, this example does not require an additional license to run it, although its companion example, AM\_06a, does.

### Material Properties

- SS316L: Fe-17.0Cr-12.0Ni-2.5Mo-0.03C 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

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.

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](#)" on page 178 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 15 um** 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  $P = 80$  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 89](#).

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	53.30	47.82	12.93	14.36
AM use calibrated Gaussian P80 u400	80	400	93.25	105	102.71	106.57

*Table 1: A comparison of the melt pool dimensions from the simulations and experiments using the calibrated Gaussian heat source.*

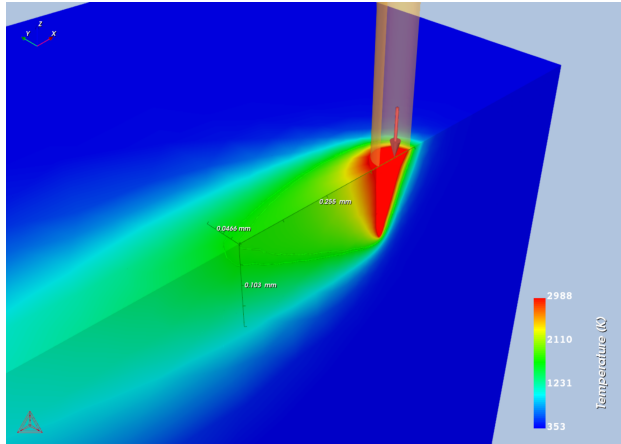


Figure 89: Steady-state temperature distribution around the melt pool for SS316L with  $P = 80$  W and scanning speed = 400 mm/s using the calibrated Gaussian heat source with the keyhole model. A keyhole can also be seen formed just below the location of the heat source.

## Double Ellipsoidal Heat Source

For the **Double ellipsoidal - 316L - beam d 15  $\mu$ m** 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 90*.

Calculator Name	Power [W]	Scanning Speed [mm/s]	Melt Pool Width		Melt Pool Depth	
			Simulation [ $\mu$ m]	Experiment [ $\mu$ m]	Simulation [ $\mu$ m]	Experiment [ $\mu$ m]
AM use calibrated DE P80 u1200	100	1200	65.07	64.34	31.63	29.68
AM use calibrated DE P80 u400	80	400	103.65	105	112.29	106.57

Table 2: A comparison of the melt pool dimensions from the simulations and experiments using the calibrated double ellipsoidal heat source.

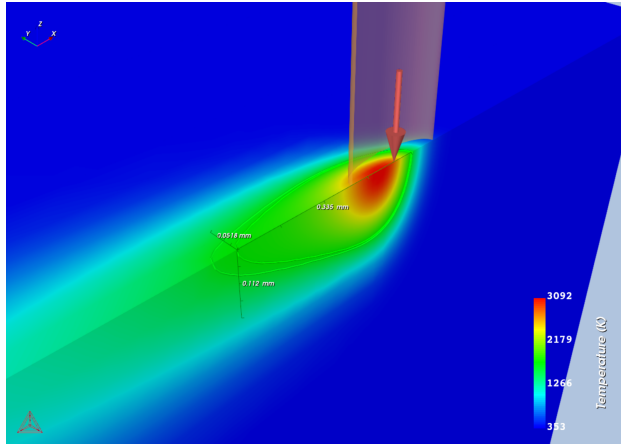


Figure 90: Steady-state temperature distribution around the melt pool for SS316L with  $P = 80$  W and scanning speed = 400 mm/s using the calibrated double ellipsoidal heat source.

## 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).

## 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\_07: Batch Calculations for an IN718 Alloy

This example shows the use of the **AM Calculator** with a **Steady-state** mode and **Batch Calculation Type** where it compares calculated and measured melt pool dimensions. The experimental data are from the 2022 NIST AM-Bench Test Series [2022NIST] where single track experiments were performed on a IN718 bare plate at different power and scan speeds.

The use of different **Plot types** in this example include a **Parity plot**, **3D plot** showing the keyhole, and **Printability map**.



This example is part of a set using a **Steady-state** simulation with a **Gaussian** heat source, plus the **Keyhole model** including **Fluid flow**. These examples collectively show the use of **Batch** and **Grid** calculation types plus various plot types such as **Printability maps**, **Parity plots**, and **Melt pool vs energy density**. The examples are numbered AM\_07 to AM\_09b.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: AM\_07\_Batch\_IN718.tcu



This example requires an Additive Manufacturing (AM) Module license.

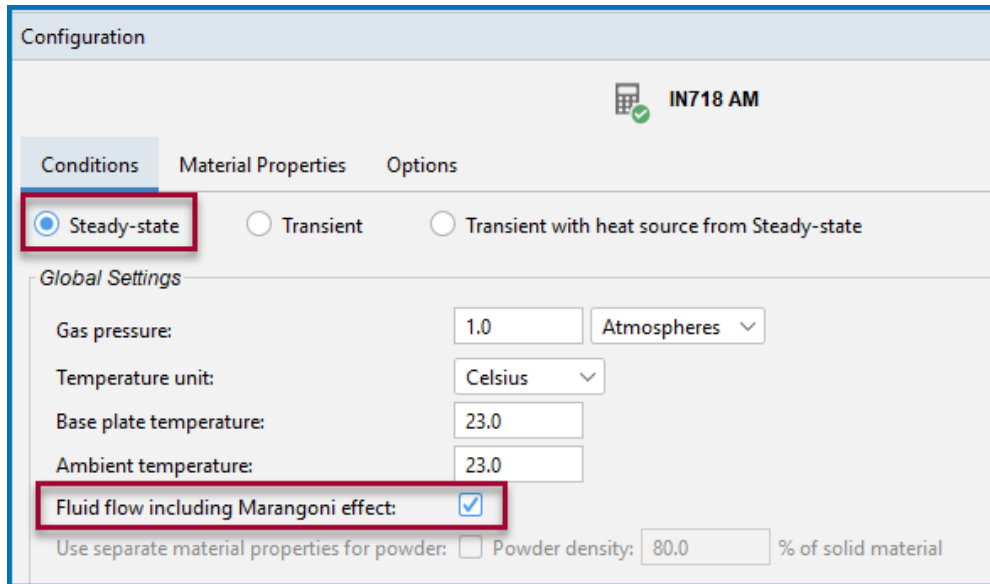


Some examples (AM\_01, AM\_02, AM\_03, and AM\_06b) are available to all users. These examples can be run without an additional Additive Manufacturing license when you are in DEMO (demonstration) mode. However, the AM Module is not available with the Educational version of Thermo-Calc. Search the help for Available Options with the Additive Manufacturing (AM) Module. See "[Available Options](#)" on page 10 for details.

### Configuration and Calculation Set Up

Below highlights the main settings for this example.

The **Steady-state** simulation is selected and the **Fluid flow including Marangoni effect** checkbox is selected.



Configuration

IN718 AM

Conditions Material Properties Options

Steady-state  Transient  Transient with heat source from Steady-state

Global Settings

Gas pressure: 1.0 Atmospheres

Temperature unit: Celsius

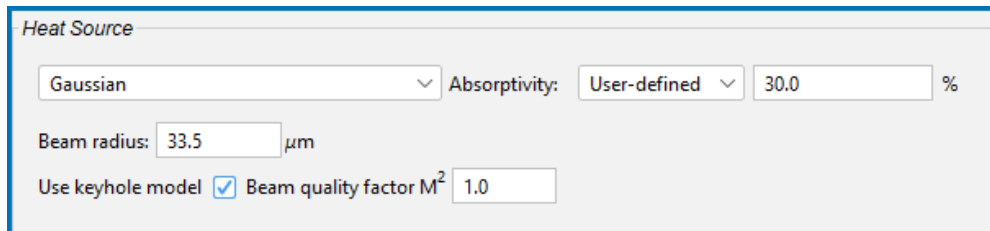
Base plate temperature: 23.0

Ambient temperature: 23.0

Fluid flow including Marangoni effect:

Use separate material properties for powder:  Powder density: 80.0 % of solid material

The *Heat Source* is set to **Gaussian** and uses the **Keyhole model**. The printers in the experiments had a beam diameter of  $67\ \mu\text{m}$  so the Gaussian **Beam radius** is set to  $33.5\ \mu\text{m}$ . The **Absorptivity** is set to  $30.0\ \%$ .



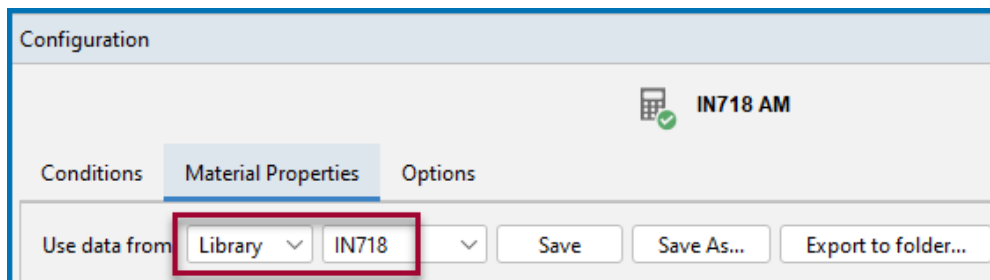
Heat Source

Gaussian Absorptivity: User-defined 30.0 %

Beam radius: 33.5  $\mu\text{m}$

Use keyhole model  Beam quality factor  $M^2$  1.0

The **IN718** material is selected from the **Material Properties** library. The material properties are precalculated and stored as a built-in material **Library**.



Configuration

IN718 AM

Conditions Material Properties Options

Use data from Library IN718 Save Save As... Export to folder...

The **Batch Calculation Type** is used to set up all the conditions from the multiple experiments in a single calculation. The experimental *Power* and scan *Speed* as well as the measured melt pool *Width* and *Depth* were collected in a CSV file and read into the software. This data is then saved in the project file.

In the *Batch Experiment Data* table you can see that the power ranges between 245 W to 285 W and the scan speed ranges between 800 mm/s to 1200 mm/s.

Calculation Type

Single Point
  Heat Source Calibration
  Batch
  Grid

Batch Experiment Data

Experiment file  delimiter Comma

#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp.width (μm)	Exp.depth (μm)	Use
1	285.000000	960.000000	0.296875	136.300000	139.700000	<input checked="" type="checkbox"/>
2	285.000000	1200.000000	0.237500	112.900000	109.700000	<input checked="" type="checkbox"/>
3	285.000000	800.000000	0.356250	156.100000	176.500000	<input checked="" type="checkbox"/>
4	325.000000	960.000000	0.338542	134.300000	166.100000	<input checked="" type="checkbox"/>
5	245.000000	960.000000	0.255208	129.400000	116.900000	<input checked="" type="checkbox"/>

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it takes at least 30 minutes for the calculations to complete.



There is a wide variety of information shown both in the **Visualizations** and **Plot Renderer Configuration** windows that can be viewed during configuration and after performing the calculation(s). Not all views, such as the **Geometry** or previews, nor all additional output (i.e. plots) are shown in this section and it is recommended that you open and run the example to review all available options and results.

## Parity Plot

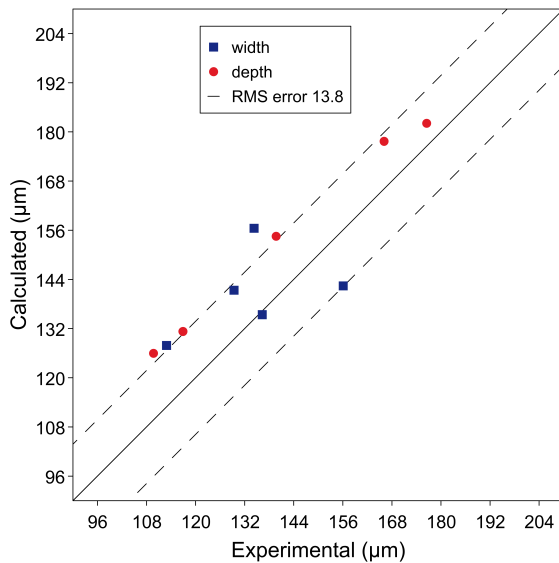


Figure 91: Parity plot comparing experimental versus calculated melt pool width and depth for all the experiments. The experiments are single tracks on bare plate IN718 with varied power and scan speed. The Root Mean Square (RMS) error can also be shown as a dashed line.

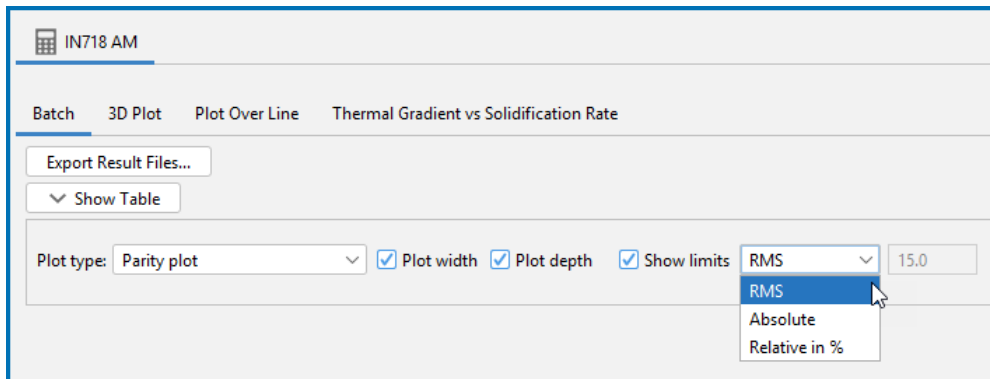


Figure 92: Alternatively, lines for user-defined Absolute or Relative in % error can be shown instead by selecting these options on the Configuration window.

## 3D Plot with Surface Colormap

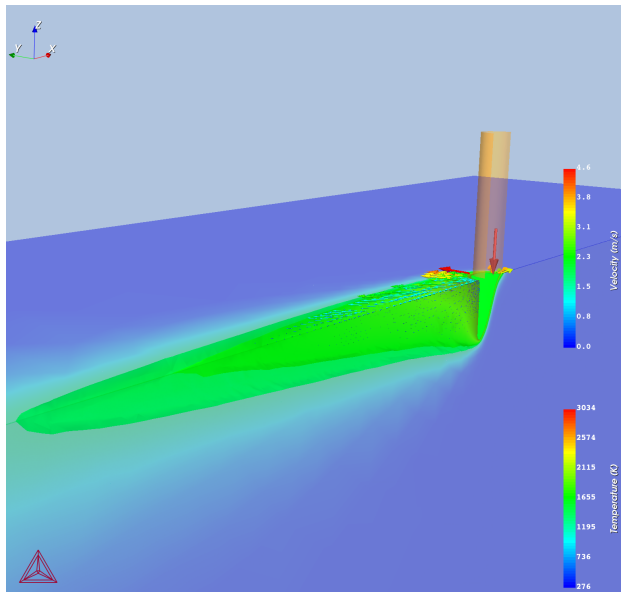


Figure 93: 3D plot showing a keyhole for the second simulation that uses power 285 W and scan speed 1200 mm/s.

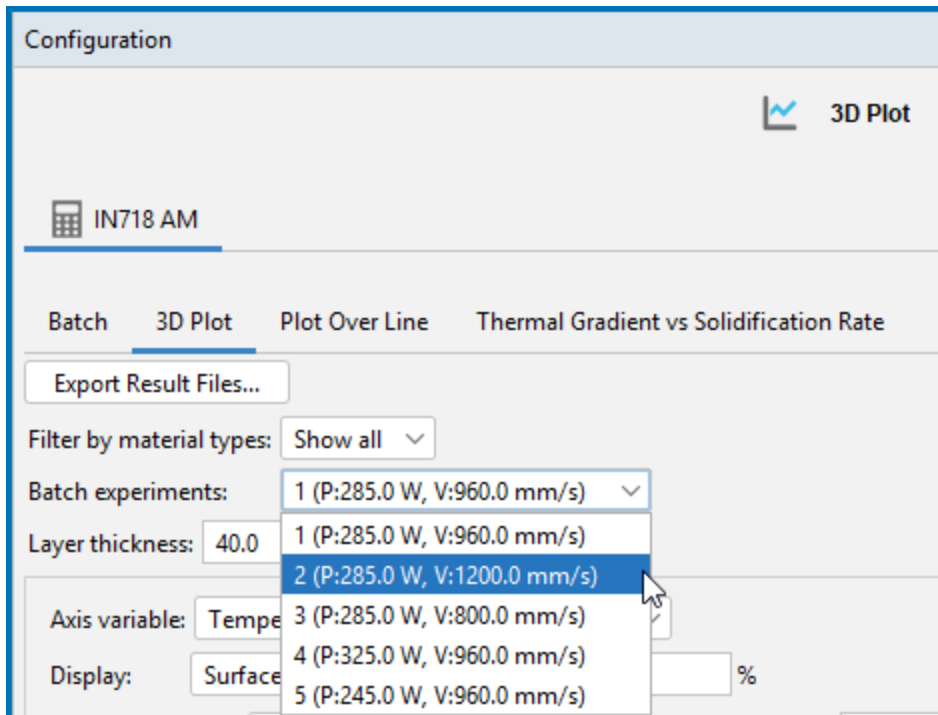


Figure 94: Selecting the Batch experiment number to display in the Visualizations window for the 3D plot shown in Figure 93.



"Visualizing Batch Calculations in the AM Module" on page 188

## Reference

[2022NIST] National Institute of Standards and Technology (NIST), Additive Manufacturing Benchmark Test Series (AM-Bench) (2022), (available at <https://www.nist.gov/ambench>).

## 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\_08a: Grid Calculation for a Ti64 Alloy

This example shows the use of the **AM Calculator** with a **Steady-state** mode and **Grid Calculation Type** where it compares the calculated and measured printability map. Printability maps are also known as *process maps*. The experiments are from Dilip *et. al* [2017DiI] where they performed single track experiments with the alloy Ti64 at different power and scan speeds. They also printed cubes and performed measurements of the porosity amounts for each experimental condition.

The use of different **Plot types** in this example include a **Printability map** and a **3D plot** with surface colormap.



This example is part of a set using a **Steady-state** simulation with a **Gaussian** heat source, plus the **Keyhole model** including **Fluid flow**. These examples collectively show the use of **Batch** and **Grid** calculation types plus various plot types such as **Printability maps**, **Parity plots**, and **Melt pool vs energy density**. The examples are numbered AM\_07 to AM\_09b.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: AM\_08a\_Printability\_Map\_Ti64.tcu



This example requires an Additive Manufacturing (AM) Module license.

### Configuration and Calculation Set Up

Below highlights some of the settings for this example to compare with experimental data of Ti64.



This example builds on the previous one (AM\_07) and it is recommended to review this and to open the example file to locate and follow along for the settings described here and found on the **Configuration** window.

The **Steady-state** calculation includes **Fluid flow**.

Configuration

AM Ti64 Grid

Conditions Material Properties Options

Steady-state  Transient  Transient with heat source from Steady-state

Global Settings

Gas pressure: 100000.0 Pascal

Temperature unit: Kelvin

Base plate temperature: 303.15

Ambient temperature: 296.15

Fluid flow including Marangoni effect:

Use separate material properties for powder:  Powder density: 80.0 % of solid material

The *Heat Source* is set to **Gaussian** and uses the **Keyhole model**. The printers in the experiments had a beam diameter of 100  $\mu\text{m}$  so the Gaussian **Beam radius** is set to 50  $\mu\text{m}$ . The **Absorptivity** is set to **Calculated**.

Heat Source

Gaussian Absorptivity: Calculated with prefactor 1.0 Wave length: 1064.0 nm

Beam radius: 50.0  $\mu\text{m}$

Use keyhole model  Beam quality factor  $M^2$  1.0

The **Ti6Al4V** from **project file** material is selected from the **Material Properties** library. The material properties are precalculated and stored as a built-in material **Library**.

Configuration

AM Ti64 Grid

Conditions Material Properties Options

Use data from: Library Ti6Al4V from project file Save Save As...

The **Grid Calculation Type** is used to cover all the conditions from the experiments in a single calculation. The **Power** ranges between 50–200 W and the **Scanning speed** ranges between 500–1200 mm/s.

Calculation Type			
<input type="radio"/> Single Point	<input type="radio"/> Heat Source Calibration	<input type="radio"/> Batch	<input checked="" type="radio"/> <b>Grid</b>
Grid Definitions			
Quantity	Min	Max	Number of steps
Power (W)	50.0	200.0	5
Scanning speed (mm/s)	500.0	1200.0	5

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



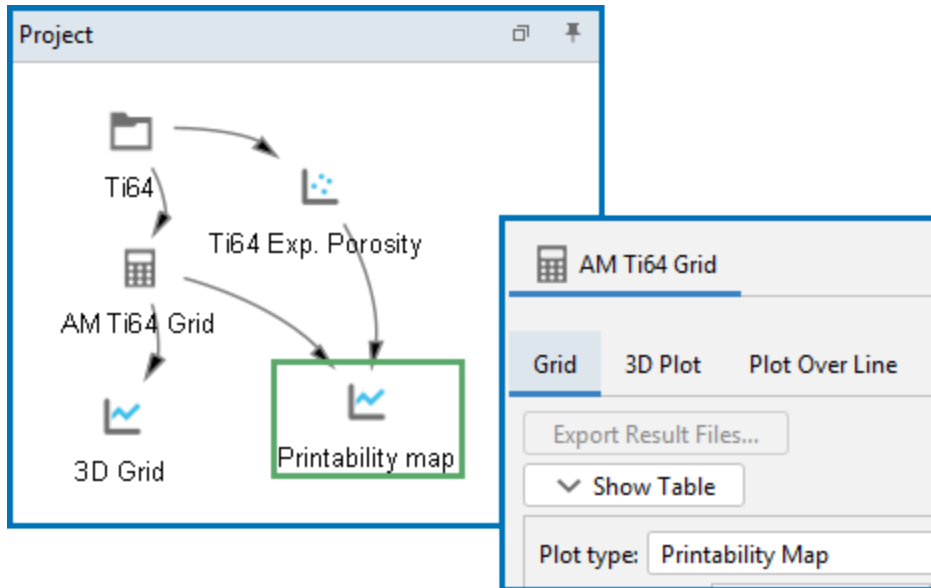
When you run (Perform) this example, it can take around two hours to complete the calculations.



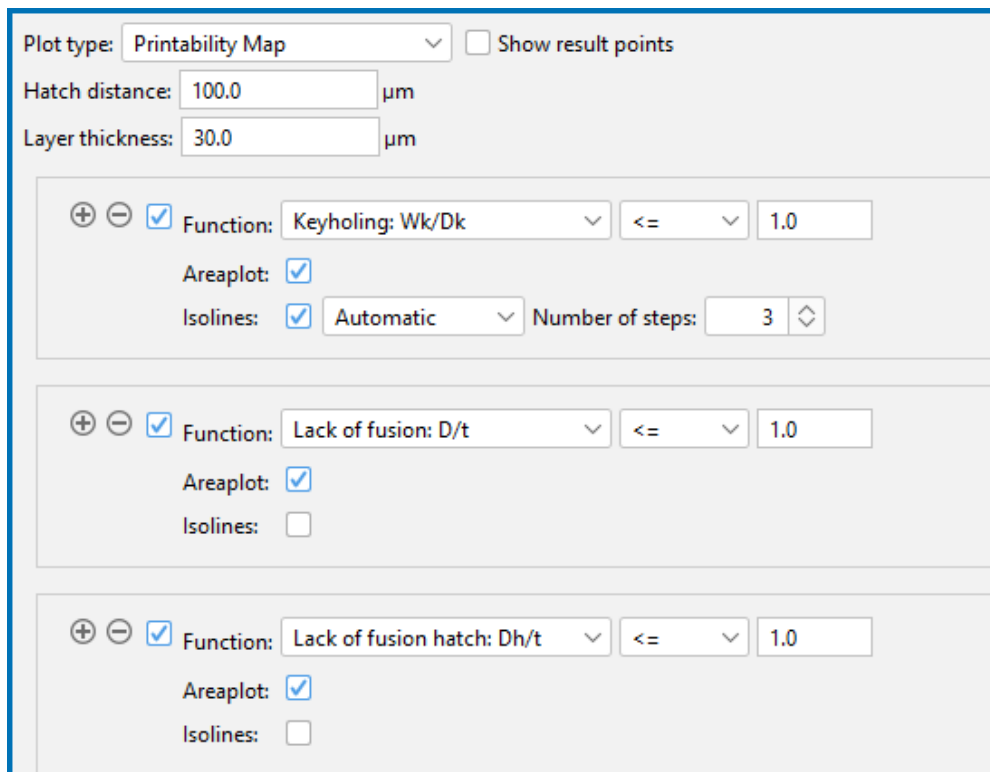
There is a wide variety of information shown both in the **Visualizations** and **Plot Renderer Configuration** windows that can be viewed during configuration and after performing the calculation(s). Not all views, such as the **Geometry** or previews, nor all additional output (i.e. plots) are shown in this section and it is recommended that you open and run the example to review all available options and results.

## Plot Renderer Configuration Window

The combined results from the **Grid** calculation can be viewed under the matching **Grid** tab on the Plot Renderer **Configuration** window where it is set to use the **Printability map** plot type. In this example, the **Plot Renderer** node is renamed to **Printability map** in the **Project** window.



The experimental value of  $30\ \mu\text{m}$  powder thickness (**Layer thickness**) and  $100\ \mu\text{m}$  for the **Hatch distance** seems to be too big to produce dense builds according to this lack of fusion criteria. Full density can then in principle only be achieved by melting the regions between tracks by shifting the layers printed on top so the full depth of the melt pool covers unmelted regions.



## Printability Map and 3D Plot

There is a video tutorial about the **Printability Map** on our [website](#) and on our [YouTube channel](#). It is also included in the Additive Manufacturing Module [YouTube playlist](#).

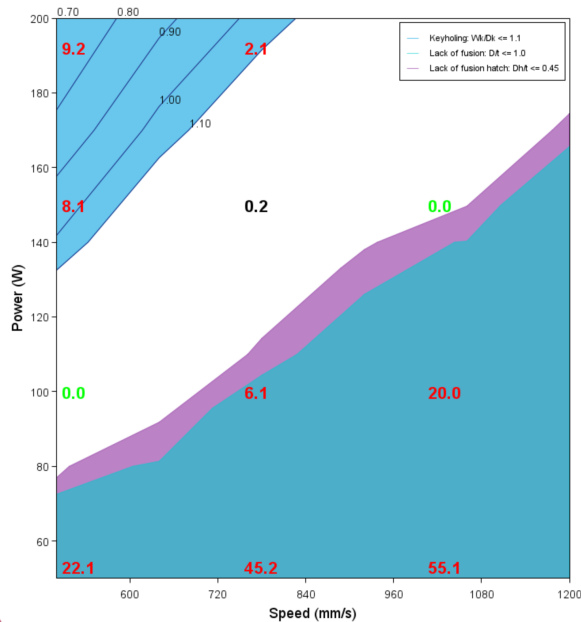


Figure 95: The calculated printability map for Ti64 showing the regions for keyholing porosity (upper left) and lack of fusion porosity to the lower right. The labels show the measured amount of porosity. Labels in red show regions with severe amounts of porosity and labels in black/green show regions with little or no defects.

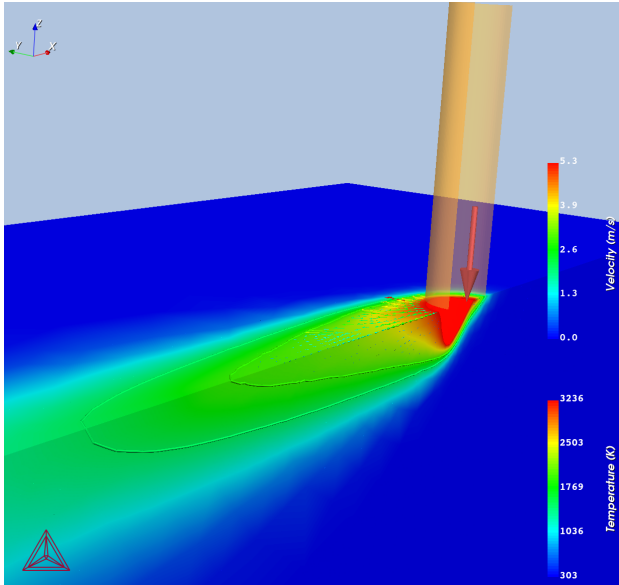


Figure 96: 3D plot showing a keyhole for the simulation that uses power 200 W and scan speed 1200 mm/s.

## Reference

[2017Dil] J. J. S. Dilip, S. Zhang, C. Teng, K. Zeng, C. Robinson, D. Pal, B. Stucker, Influence of processing parameters on the evolution of melt pool, porosity, and microstructures in Ti-6Al-4V alloy parts fabricated by selective laser melting. *Prog. Addit. Manuf.* 2, 157–167 (2017).

## 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.

## AM\_08b: Batch Calculations for a Ti64 Alloy

This example shows the use of the **AM Calculator** with a **Steady-state** mode and **Batch Calculation Type** where it compares the calculated and measured melt pool dimensions. The experiments are from Dilip *et. al* [2017Dil] where they performed single track experiments with the alloy Ti64 at different power and scan speeds.

The use of different **Plot types** in this example include a **Parity plot**, **Melt pool vs energy density**, **3D plot** showing the keyhole, and **Printability map**.



This example is part of a set using a **Steady-state** simulation with a **Gaussian** heat source, plus the **Keyhole model** including **Fluid flow**. These examples collectively show the use of **Batch** and **Grid** calculation types plus various plot types such as **Printability maps**, **Parity plots**, and **Melt pool vs energy density**. The examples are numbered AM\_07 to AM\_09b.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: `AM_08b_Batch_Ti64.tcu`



This example requires an Additive Manufacturing (AM) Module license.

### Configuration and Calculation Set Up



Below highlights some of the settings for this example. This example builds on the previous ones (AM\_07 and AM\_08a) and it is recommended to review these and to open the example file to locate and follow along for the settings described here and found on the **Configuration** window.

The **Steady-state** calculation is configured with the **Gaussian Heat Source** with the **Keyhole model** and includes **Fluid flow**. The printers had a beam diameter of 100  $\mu\text{m}$  so the **Gaussian Beam radius** is set to 50  $\mu\text{m}$ . The **Absorptivity** is set to 27 %.

The **Ti6Al4V** material is selected from the **Material Properties** library. The material properties are precalculated and stored as a built-in material **Library**.

The **Batch Calculation Type** is used to set up all the conditions from the experiments in a single calculation. The experimental *Power* and scan *Speed* as well as the measured melt pool *Width* and *Depth* were collected in a CSV file and read into the software. This data is then saved in the project file.

In the *Batch Experiment Data* table (see [Visualizing the Batch Calculation Experimental Data](#)) you can see that the power ranges between 50–195 W and the scan speed ranges between 500–1200 mm/s.

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it takes at least 30 minutes for the calculations to complete.



There is a wide variety of information shown both in the **Visualizations** and **Plot Renderer Configuration** windows that can be viewed during configuration and after performing the calculation(s). Not all views, such as the **Geometry** or previews, nor all additional output (i.e. plots) are shown in this section and it is recommended that you open and run the example to review all available options and results.

## Visualizing the Batch Calculation Experimental Data

During the set up of a calculation, the *Batch Experiment Data* is imported into the **AM Calculator Configuration** window, where you can review the data and choose to include or exclude data points by selecting and deselecting the checkboxes as needed in the **Use** column (see [Figure 97](#)). At the same time, you can observe the change as this is updated in the **Visualizations** window (see [Figure 98](#)). Data can also be entered directly into the table.

Calculation Type

Single Point
  Heat Source Calibration
  Batch
  Grid

Batch Experiment Data

Experiment file  delimiter

#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp.width (μm)	Exp.depth (μm)	Use
1	50.000000	500.000000	0.100000	67.379660	16.058920	<input checked="" type="checkbox"/>
2	50.000000	750.000000	0.066667	50.318180	11.830310	<input checked="" type="checkbox"/>
3	50.000000	1000.000000	0.050000	47.357410	9.654352	<input checked="" type="checkbox"/>
4	50.000000	1200.000000	0.041667	45.508170	5.108316	<input checked="" type="checkbox"/>
5	100.000000	500.000000	0.200000	118.025200	44.462060	<input checked="" type="checkbox"/>
6	100.000000	750.000000	0.133333	98.663500	32.019900	<input checked="" type="checkbox"/>
7	100.000000	1000.000000	0.100000	75.846620	24.027610	<input checked="" type="checkbox"/>
8	100.000000	1200.000000	0.083333	72.846300	21.534230	<input checked="" type="checkbox"/>
9	150.000000	500.000000	0.300000	145.364300	101.268300	<input checked="" type="checkbox"/>
10	150.000000	750.000000	0.200000	135.499000	72.058650	<input checked="" type="checkbox"/>

Figure 97: The experimental data used for the Batch calculation for the single track experiments of Ti64 is imported to the AM Calculator table where you can review the data points and include or exclude as needed using the checkboxes.

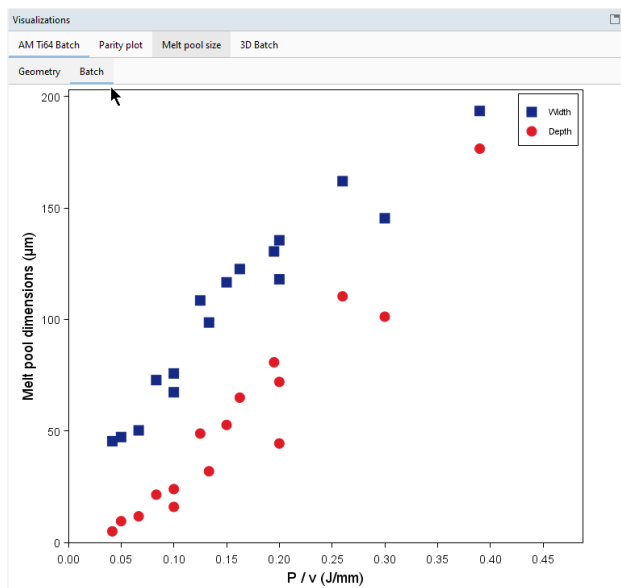


Figure 98: The visualization of the experimental melt pool dimensions are shown as a function of the energy density  $P/v$ . You can adjust the selected points in the Batch Experiment Data table and watch the updates dynamically in this window.

## Parity, Melt Pool, and 3D Plot

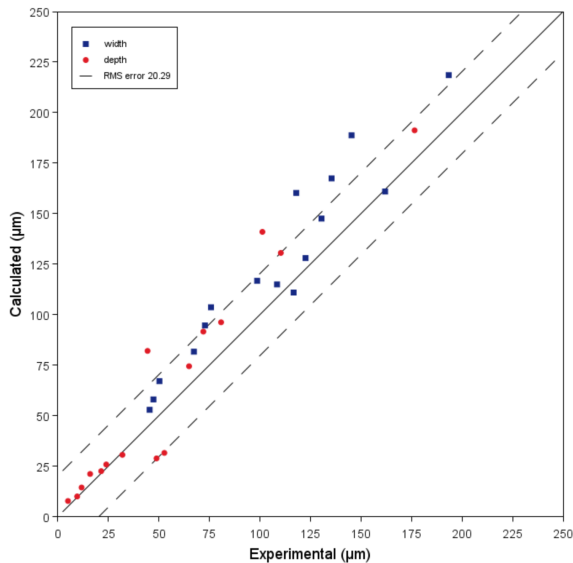


Figure 99: Parity plot comparing experimental versus calculated melt pool width and depth for all the Batch calculations. The experiments are single tracks with Ti64 with varied power and scan speed.

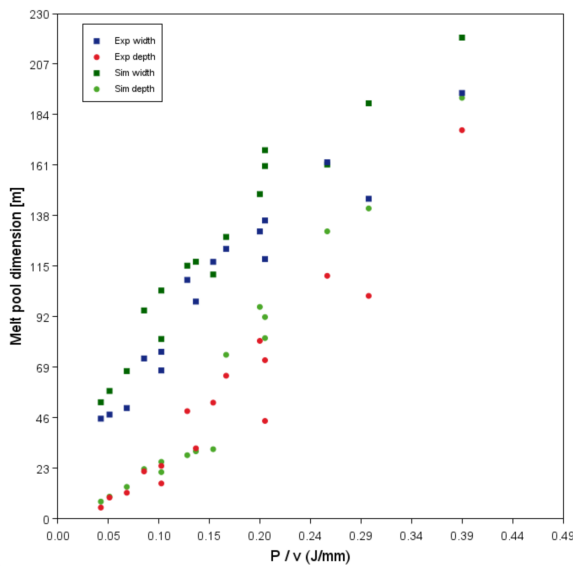


Figure 100: Plot comparing experimental versus calculated melt pool dimensions for all the Batch calculations. The experiments are single tracks with Ti64 with varied power and scan speed. The melt pool width and depth are shown on the Y-axis and the energy density ( $P/v$ ) on the X-axis.

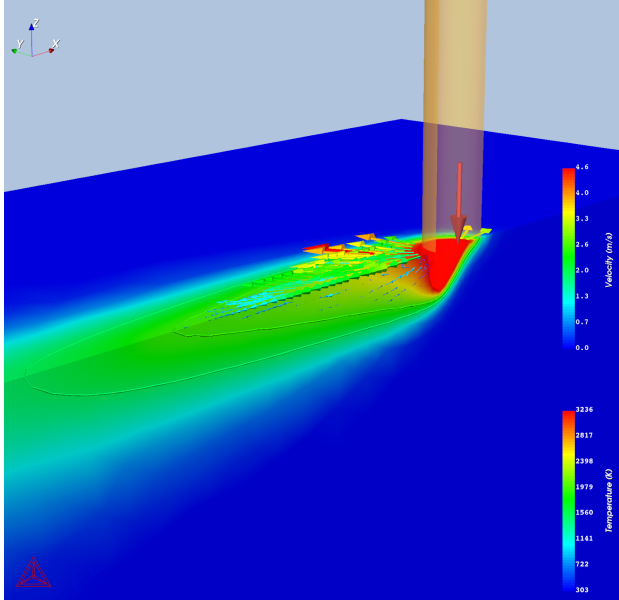


Figure 101: 3D plot showing a keyhole for the 15th simulation that uses power 195 W and scan speed 1000 mm/s.



"Visualizing Batch Calculations in the AM Module" on page 188

## Reference

[2017Dil] J. J. S. Dilip, S. Zhang, C. Teng, K. Zeng, C. Robinson, D. Pal, B. Stucker, Influence of processing parameters on the evolution of melt pool, porosity, and microstructures in Ti-6Al-4V alloy parts fabricated by selective laser melting. *Prog. Addit. Manuf.* 2, 157–167 (2017).

## 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\_09a: Grid Calculation for an SS316L Alloy

This example shows the use of the **AM Calculator** with a **Steady-state** mode and **Grid Calculation Type** where it compares the calculated and measured printability map. Printability maps are also known as *process maps*. The experiments are from Hu *et. al* [2019Hu] where they performed single track experiments with the alloy SS316L at different power and scan speeds.

The use of different **Plot types** in this example include a **Printability map** and a **3D plot** with surface colormap.



This example is part of a set using a **Steady-state** simulation with a **Gaussian** heat source, plus the **Keyhole model** including **Fluid flow**. These examples collectively show the use of **Batch** and **Grid** calculation types plus various plot types such as **Printability maps**, **Parity plots**, and **Melt pool vs energy density**. The examples are numbered AM\_07 to AM\_09b.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: AM\_09a\_Printability\_Map\_316L.tcu



This example requires an Additive Manufacturing (AM) Module license.

### Configuration and Calculation Set Up



Below highlights some of the settings for this example. This example builds on the previous ones (AM\_07, AM\_08a, and AM\_08b) and it is recommended to review these and to open the example file to locate and follow along for the settings described here and found on the **Configuration** window.

The **Steady-state** calculation is configured with the **Gaussian Heat Source** with the **Keyhole model** and includes **Fluid flow**.

The Gaussian **Beam radius** is set to 22  $\mu\text{m}$ . The **Absorptivity** is set to 30 % and in the **Scanning Strategy** section, the powder **Layer thickness** is set to 10  $\mu\text{m}$ .

The **SS316L** material is selected from the **Material Properties** library. The material properties are precalculated and stored as a built-in material **Library**.

The **Grid Calculation Type** is used to cover all the conditions from the experiments in a single calculation. The **Power** ranges between 40–100 W and the **Scanning speed** ranges between 400–3000 mm/s.

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



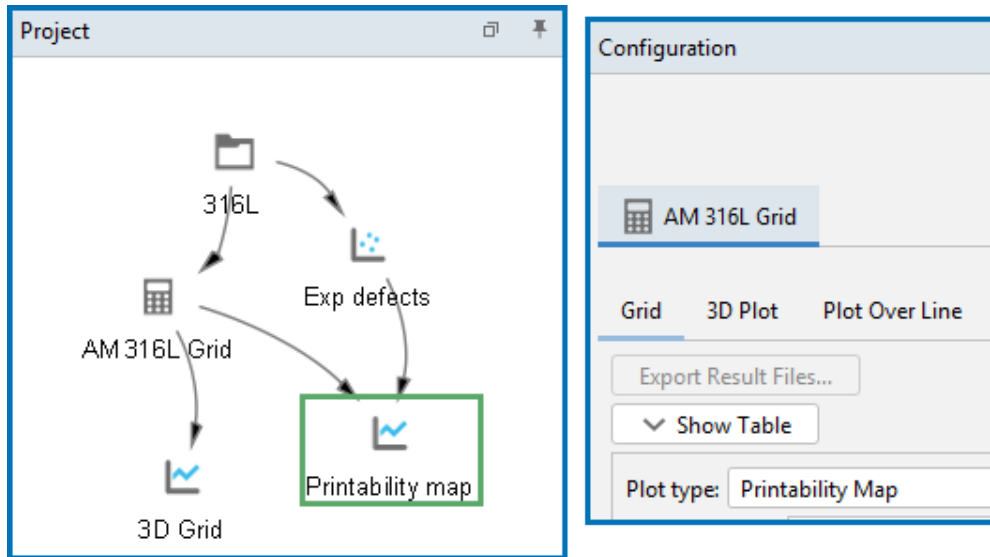
When you run (Perform) this example, it can take around two hours to complete the calculations.



There is a wide variety of information shown both in the **Visualizations** and **Plot Renderer Configuration** windows that can be viewed during configuration and after performing the calculation(s). Not all views, such as the **Geometry** or previews, nor all additional output (i.e. plots) are shown in this section and it is recommended that you open and run the example to review all available options and results.

## Plot Renderer Configuration Window

The combined results from the **Grid** calculation can be viewed under the matching **Grid** tab on the Plot Renderer **Configuration** window where it is configured to use the **Printability map** plot type. In this example, the **Plot Renderer** node is renamed to **Printability map** in the **Project** window.



Next each **Function** is defined and limits were adjusted to match the experimental regions of keyholing and lack of fusion porosity.

- The keyholing limit (**Keyholing: Wk/Dk**) is kept at default 1.0.
- The **Lack of fusion: D/t** is increased from default of 1.0 to 1.3.
- The experiments were only single-track and the lack of fusion at half of the hatch distance (**Lack of fusion hatch: Dh/t**) is kept at default 1.0 and only included together with the **Hatch distance** of 35  $\mu\text{m}$  for demonstration purposes.

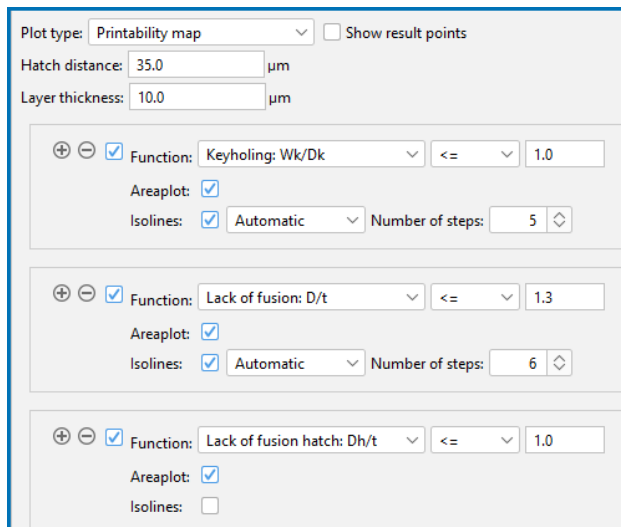


Figure 102: The settings for the printability map with the Function limits defined for keyholing, lack of fusion, and lack of fusion at half of the hatch distance (as described in the text).

## Printability Map and 3D Plot

There is a video tutorial about the **Printability Map** on our [website](#) and on our [YouTube channel](#). It is also included in the Additive Manufacturing Module [YouTube playlist](#).

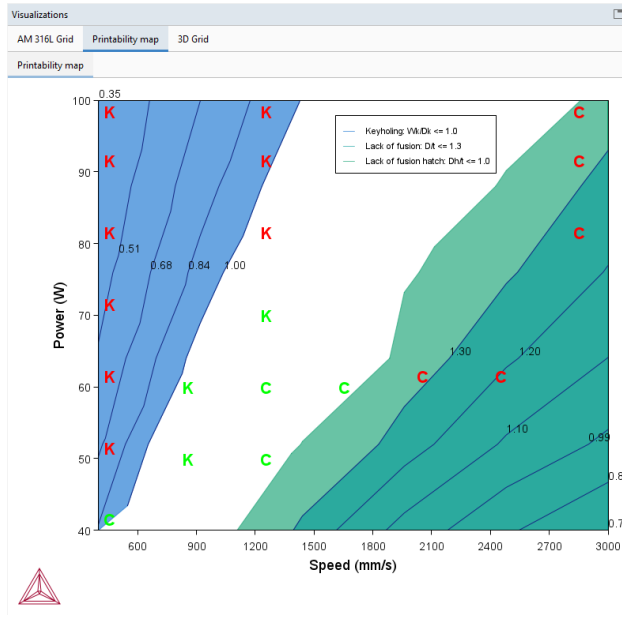


Figure 103: Printability map for 316L showing regions of keyholing and lack-of-fusion. Experimental information from Hu et. al. [2019Hu] overlaid as coloured labels showing keyhole porosity (**K**), conduction mode with lack-of-fusion porosity (**C**). Green labels showing experiments without defects for conduction mode (**C**) and keyhole mode (**K**).

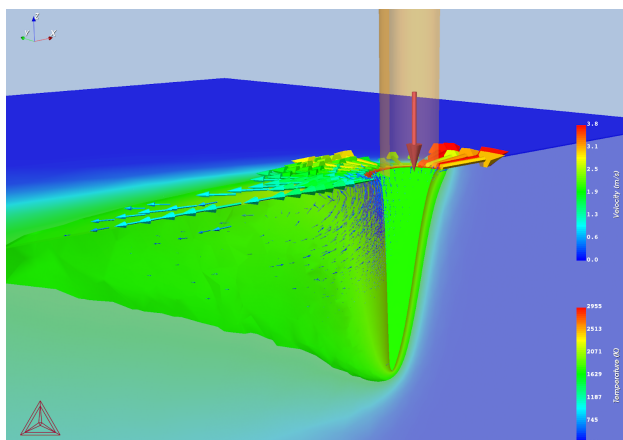


Figure 104: 3D plot showing a keyhole for the simulation that uses power 100 W and scan speed 400 mm/s.

## 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, 1–9 (2019).

## 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.

## AM\_09b: Batch Calculations for an SS316L Alloy

This example shows the use of the **AM Calculator** with a **Steady-state** mode and **Batch Calculation Type** where it compares the calculated and measured melt pool dimensions. The experiments are from Hu *et. al* 2019 where they performed single track experiments with the alloy SS316L at different power and scan speeds.

The use of different **Plot types** in this example include a **Parity plot**, **Melt pool vs energy density**, **3D plot** showing the keyhole, and **Printability map**.



This example is part of a set using a **Steady-state** simulation with a **Gaussian** heat source, plus the **Keyhole model** including **Fluid flow**. These examples collectively show the use of **Batch** and **Grid** calculation types plus various plot types such as **Printability maps**, **Parity plots**, and **Melt pool vs energy density**. The examples are numbered AM\_07 to AM\_09b.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: AM\_09b\_Batch\_316L.tcu



This example requires an Additive Manufacturing (AM) Module license.

### Configuration and Calculation Set Up



Below highlights some of the settings for this example. This example builds on the previous ones (AM\_07, AM\_08a, AM\_08b, and AM\_09a) and it is recommended to review these and to open the example file to locate and follow along for the settings described here and found on the **Configuration** window.

The **Steady-state** calculation is configured with the **Gaussian Heat Source** with the **Keyhole model** and includes **Fluid flow**.

The Gaussian **Beam radius** is set to 22  $\mu\text{m}$ . The **Absorptivity** is set to 30 % and in the **Scanning Strategy** section, the powder **Layer thickness** is set to 10  $\mu\text{m}$ .

The **SS316L** material is selected from the **Material Properties** library. The material properties are precalculated and stored as a built-in material **Library**.

The **Batch Calculation Type** is used to set up all the conditions from the experiments in a single calculation. The experimental *Power* and scan *Speed* as well as the measured melt pool *Width* and *Depth* were collected in a CSV file and read into the software. This data is then saved in the project file.

In the *Batch Experiment Data* table the power ranges between 50–100 W and the scan speed ranges between 400–2800 mm/s.

## Visualizations



There is a video tutorial about the **Printability Map** on our [website](#) and on our [YouTube channel](#). It is also included in the Additive Manufacturing Module [YouTube playlist](#).

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



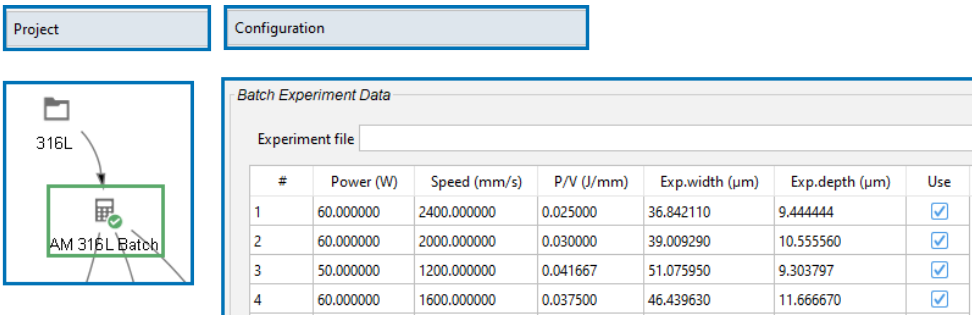
When you run (Perform) this example, it can take about an hour to complete the calculations.



There is a wide variety of information shown both in the **Visualizations** and **Plot Renderer Configuration** windows that can be viewed during configuration and after performing the calculation(s). Not all views, such as the **Geometry** or previews, nor all additional output (i.e. plots) are shown in this section and it is recommended that you open and run the example to review all available options and results.

## Visualizing the Batch Calculation Experimental Data

During the set up of this calculation, the *Batch Experiment Data* is imported into the AM Calculator **Configuration** window, where you can review the data and choose to include or exclude data points by selecting and deselecting the checkboxes as needed in the **Use** column. At the same time, you can observe the change as this is updated in the **Visualizations** window.



The screenshot shows the 'Configuration' window with a 'Batch Experiment Data' table. The table has columns for '#', 'Power (W)', 'Speed (mm/s)', 'P/V (J/mm)', 'Exp.width (μm)', 'Exp.depth (μm)', and 'Use'. All 'Use' checkboxes are checked.

#	Power (W)	Speed (mm/s)	P/V (J/mm)	Exp.width (μm)	Exp.depth (μm)	Use
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 checked="" type="checkbox"/>
3	50.000000	1200.000000	0.041667	51.075950	9.303797	<input checked="" type="checkbox"/>
4	60.000000	1600.000000	0.037500	46.439630	11.666670	<input checked="" type="checkbox"/>

Figure 105: The experimental data used for the Batch calculation for the single track experiments of SS316L is imported to the AM Calculator table where you can review the data points and include or exclude as needed using the checkboxes.

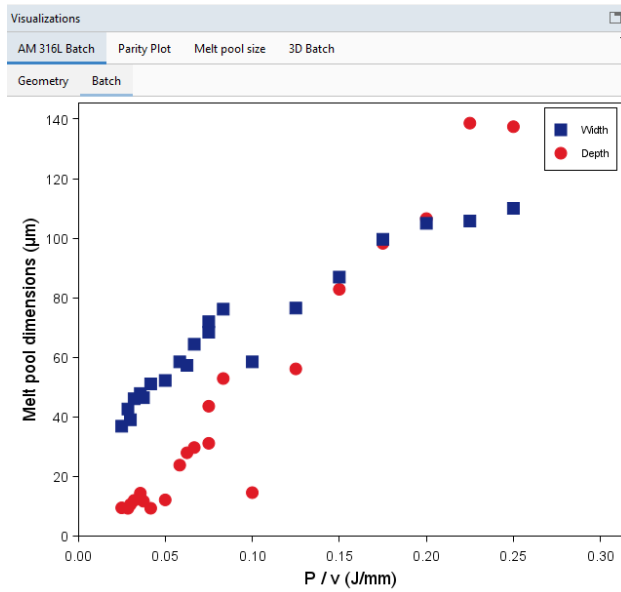


Figure 106: The visualization of the experimental melt pool dimensions are shown as a function of the energy density  $P/v$ . You can adjust the selected points in the Batch Experiment Data table and watch the updates dynamically in this window.

## Parity, Melt Pool, and 3D Plot

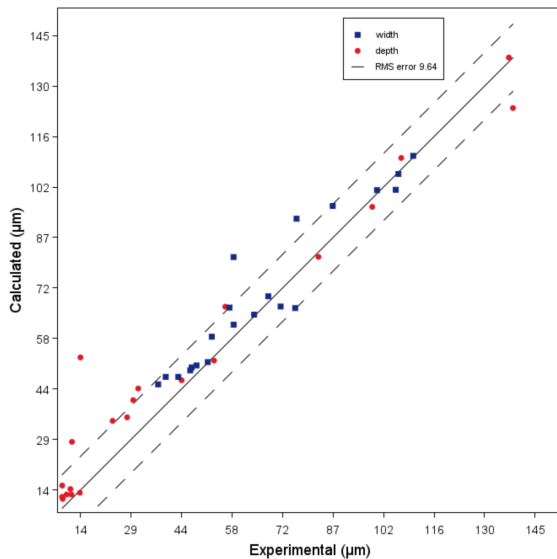


Figure 107: Parity plot comparing experimental versus calculated melt pool width and depth for all the Batch calculations. The experiments are single tracks with SS316L with varied power and scan speed.

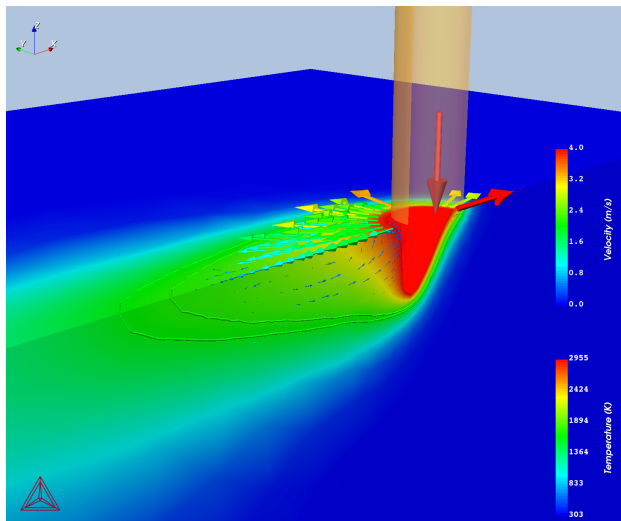


Figure 108: 3D plot showing a keyhole for the 12th simulation that uses power 60 W and scan speed 800 mm/s.

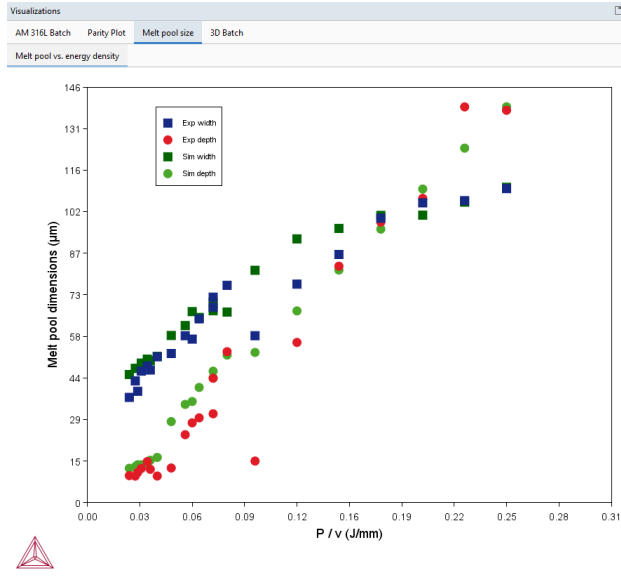


Figure 109: Plot comparing experimental versus calculated melt pool dimensions for all the Batch calculations. The experiments are single tracks with SS316L with varied power and scan speed. The melt pool width and depth are shown on the Y-axis and the energy density ( $P/v$ ) on the X-axis.

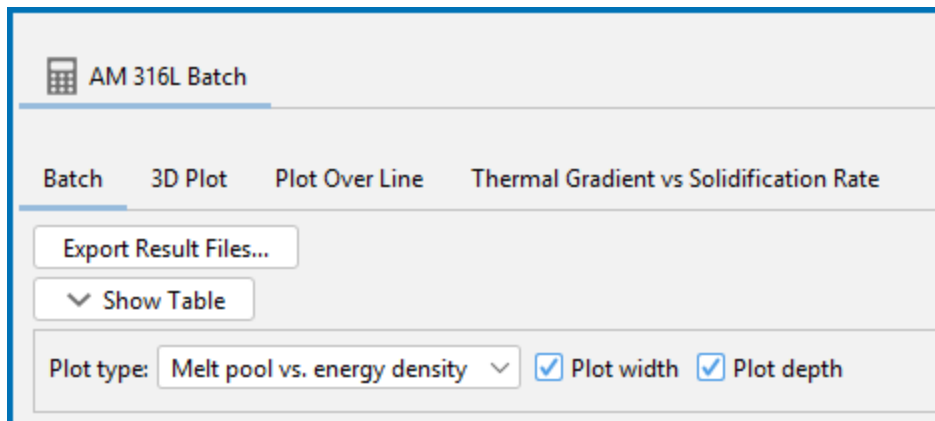


Figure 110: For the plot, you can choose what to include on the plot, to show width and/or depth by selecting the relevant checkboxes on the Configuration window.



"Visualizing Batch Calculations in the AM Module" on page 188

## 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, 1–9 (2019).

## 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.

## AM\_10: CET Transition in an IN718 Alloy

The example uses a **Scheil Calculator**, **AM Calculator**, and the **Property Model Calculator** with the **Columnar to Equiaxed Transition** Property Model to compare calculated CET curves for an IN718 alloy (from example PM\_G\_17) with the solidification conditions of the melt pool in the AM simulation.



For more background about this Property Model, search the Help or the main Thermo-Calc documentation set for *About the Columnar to Equiaxed Transition (CET) model*.



This example is similar to, and based on, example *PM\_G\_17: Columnar to Equiaxed Transition of an IN718 Alloy*. This is available from the **Property Models** → **General** folder.

A Scheil with solute trapping calculation is done first on the **Scheil Calculator** to generate thermophysical properties data to the **AM Calculator**. A steady-state AM simulation is done to predict the solidification conditions in the melt pool, the solidification rates, and thermal gradients.

Then using the **Property Model Calculator** with the **Columnar to Equiaxed Transition (CET)** model, the calculated CET curves for an IN718 alloy are fed into the **AM + CET** node (the renamed Plot Renderer), where together the data from the AM and Property Model calculations are combined and overlaid on one plot.

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.

## Project File and License Information

- Folder: **Additive Manufacturing**
- File name: AM\_10\_Columnar\_to\_Equiaxed\_Transition\_IN718.tcu



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. This example also requires additional database licenses for the TCS Ni-based Superalloys Database (TCNI) (TCNI12 and newer), and the TCS Ni-alloys Mobility Database (MOBNI) (MOBNI6 and newer).

## Configuration and Calculation Set Up

On the **Scheil Calculator** (renamed to *Scheil with solute trapping*), the **Scheil with solute trapping** is used with these specific settings:

- Trans-interface diffusivity: **Same for all elements**
- Prefactor:  $5.0E-9$
- Maximum velocity for infinite driving force:  $2000$  m/s
- Model: **Aziz**
- Interface driving force: **Driving energy**

For the **AM Calculator** (renamed to *AM Steady-state*), the **Steady-state** calculation is configured as a **Gaussian Heat Source** with the **Keyhole model** and includes **Fluid flow**. It also uses a **Single Point** calculation with the **Calculated Absorptivity**.

On the **Property Model Calculator** (renamed to *Columnar to Equiaxed Transition*) a **One Axis** calculation for the **FCC\_L12** primary phase is done using the CET parameters entered on the **Configuration** window.

The settings on the Property Model Calculator 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
- Solidification rate:  $\log_{10}(\dot{v})$

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



There is a wide variety of information shown both in the **Visualizations** and **Plot Renderer Configuration** windows that can be viewed during configuration and after performing the calculation(s). Not all views, such as the **Geometry** or previews, nor all additional output (i.e. plots) are shown in this section and it is recommended that you open and run the example to review all available options and results.

The thermal gradients and solidification rate can be visualized in 2D or 3D. [Figure 111](#) shows the thermal gradient along the melt pool boundary in a 3D plot. Only the data corresponding to solidification conditions are shown. The thermal gradients corresponding to melting conditions are filtered out and only the tail part of the melt pool are shown.

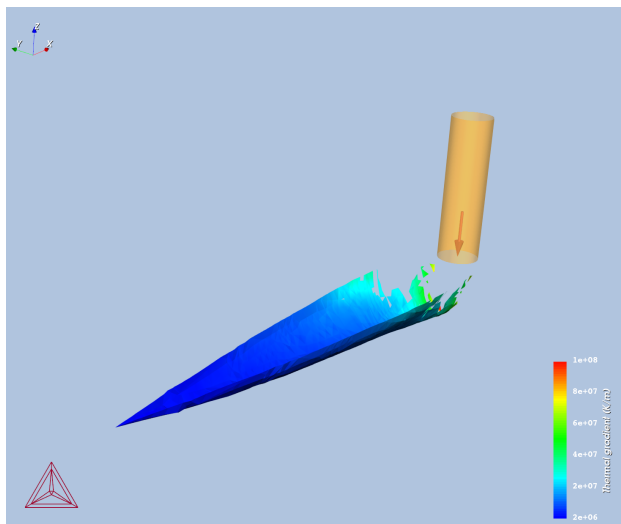


Figure 111: The 3D Plot for the 3D Thermal Gradient Plot Renderer.

Both the thermal gradients and solidification rates from the melt pool can be visualized at the same time in a 2D scatter plot when the Plot Renderer tab **Thermal Gradient vs Solidification Rate** is selected.

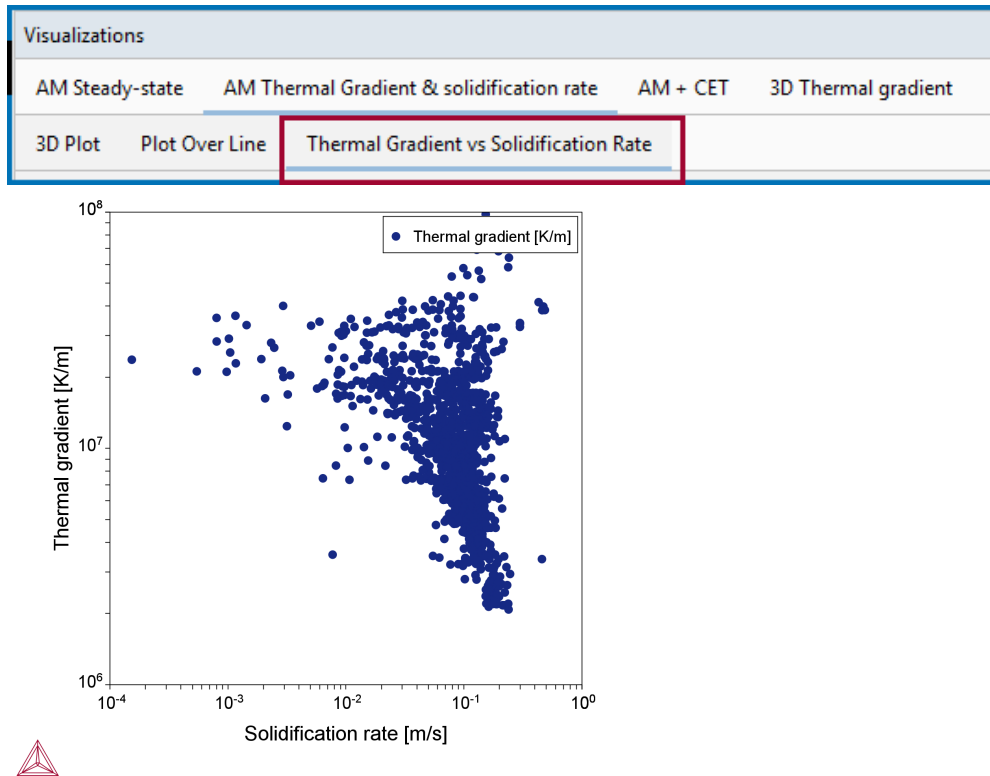


Figure 112: The solidification rate vs thermal gradient is shown on the AM Thermal Gradient & Solidification Rate Plot Renderer.

Figure 113 is an example of a plot where the results from both the AM simulation and the CET model for the IN718 thermal gradient vs solidification rate are overlaid on the same plot. The blue, red, and green lines show the increasing equiaxed fraction from the CET Property Model and the points show the solidification conditions at the melt pool calculated with the Additive Manufacturing (AM) Module. When the results are overlaid like this, you can see that nearly all the points (those below the purple line) exist in a fully columnar region.

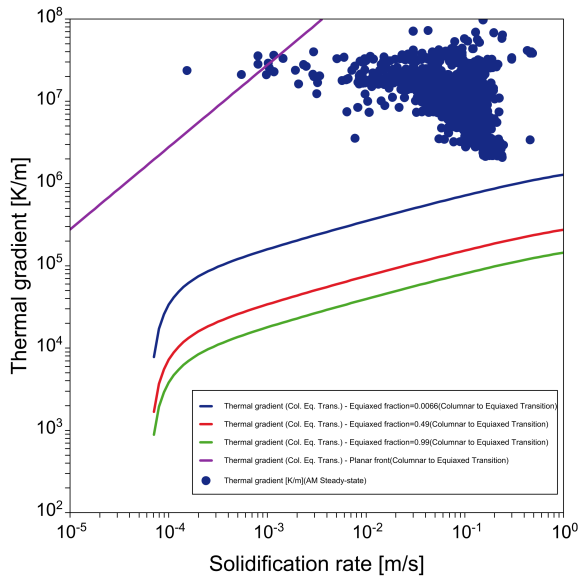


Figure 113: The solidification rate vs the thermal gradient overlaying both plots from the AM Calculator and Property Model Calculator (via the AM + CET Plot Renderer).

## 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.

## 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\_11: Comparing Single Tracks Printed on Casted and LPBF Substrates

Bogdonova et al. [2024Bog] performed single track experiments with the alloy Al10SiMg. The study compared the printing of single tracks on two types of substrates—one fabricated with traditional casting and the other with laser powder bed fusion (LPBF).

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: `AM_11_Batch_Al10SiMg.tcu`



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. This example also requires an additional database license for the TCS AI-based Alloy Database (TCAL) (TCAL9 and newer).

### Background

It is well known that the thermal conductivity of the alloy Al10SiMg is drastically reduced in the as-printed condition when fabricated by LPBF compared to conventionally manufactured counterparts [2022Gha].

Thermal conductivity in Thermo-Calc is modeled as a temperature- and composition-dependent property per phase, and the conductivity of the alloy is taken as an average over the stable phases. The composition of the primary phase can change greatly depending on the fabrication method where the LPBF method results in rapid solidification and solute trapping of solute alloy elements.

Thermal conductivity can generally be reduced by different scattering phenomena (pores, defects, phase interfaces) within the microstructure and can therefore be lower than the calculated conductivity averaged over the phases.

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 phase interface scattering constant is found to be  $4.0e-8 \Omega m$  for aluminum alloys. The contribution to thermal conductivity is assumed to be related to that of electrical resistivity, following the Wiedemann-Franz law.

The rapid solidification during LPBF generally results in a much finer microstructure compared to traditional casting. LPBF fabricated Al10SiMg solidifies with a cellular primary FCC structure and a eutectic structure between the cells [2021Lef]. An increased interface scattering constant more than  $4.0E-8$  can be argued given the much finer structure that has a large number of phase interfaces.

In this example the effect on material properties and melt pool dimensions is compared when printing single tracks on the two types of substrates. The material properties for the traditionally cast alloy is calculated using regular Scheil and the default interface scattering constant of  $4.0E-8$ . The material properties for the LPBF fabricated alloy is calculated using Scheil with solute trapping and the effect of increasing the interface scattering is shown.

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



There is a wide variety of information shown both in the **Visualizations** and **Plot Renderer Configuration** windows that can be viewed during configuration and after performing the calculation(s). Not all views, such as the **Geometry** or previews, nor all additional output (i.e. plots) are shown in this section and it is recommended that you open and run the example to review all available options and results.



When you run (Perform) this example, it can take around two hours to complete the calculations.

## Material Properties for the As-cast Substrate Material

The TCS Al-based Alloy Database (TCAL) is selected on the **System Definer** (renamed to *Scheil As-cast*). The as-cast Al10SiMg alloy has the composition of Al-10.8Si-0.3Mg-0.1Cu-0.1Fe.

The **Scheil Calculator** (renamed to *Scheil As-cast*) is configured with default settings for AM with an evaluation of thermophysical properties from 5000 K down to room temperature. The **Calculation type** is set to **Classic Scheil**.

## Material Properties for the LPBF Printed Substrate Material

The TCS Al-based Alloy Database (TCAL) is selected in the **System Definer** (renamed to *Scheil printed*). The LPBF Al10SiMg alloy has almost the same composition (Al-10.8Si-0.3Mg-0.1Cu-0.2Fe) with only a small difference in Fe content.

The **Scheil Calculator** (renamed to *Scheil Solute-Trapping*) is configured with the default settings for AM with an evaluation of thermophysical properties from 5000 K down to room temperature. The **Calculation type** is set to **Scheil with solute trapping**.

## Comparing Material Properties Between the Substrates

Two plots overlay results from the two Scheil Calculators in Figure 114. The Plot Renderer, renamed to *Si in primary FCC*, shows the Si content in the primary FCC phase. The classic Scheil calculation gives less than 1.5 at% Si. The solute trapping simulation results in twice the Si content with about 3 at% Si in the primary FCC phase. Atom probe experiments for a LPBF printed Al10SiMg alloy measured 3.08 at% Si in the FCC cell structure for the as-printed condition [2021Lef].

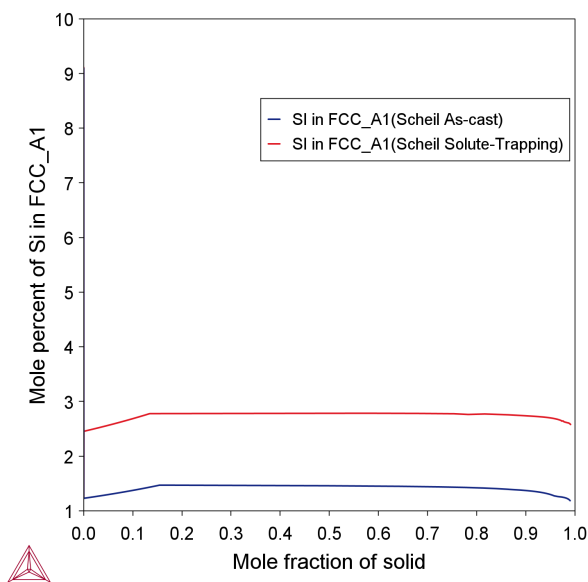


Figure 114: Silicon content in the primary FCC phase comparing regular (classic) Scheil with Scheil and solute trapping. The solute trapping almost doubles the Si content in the primary FCC phase.

Figure 115 is another overlaid plot that takes the results from two Scheil Calculator predecessors to show how the thermal conductivity varies with temperature. The conductivity for the as-cast substrate (blue line), that uses classic Scheil and the default setting for the phase interface scattering constant ( $C = 4.0E-8$ ), naturally gives the highest thermal conductivity. The red line shows the reduction in conductivity due to solute trapping and the increased Si

content in the primary phase. The green line shows further reduction in thermal conductivity due to the increased interface scattering when the scattering constant was increased ( $C = 1.0E-7$ ).

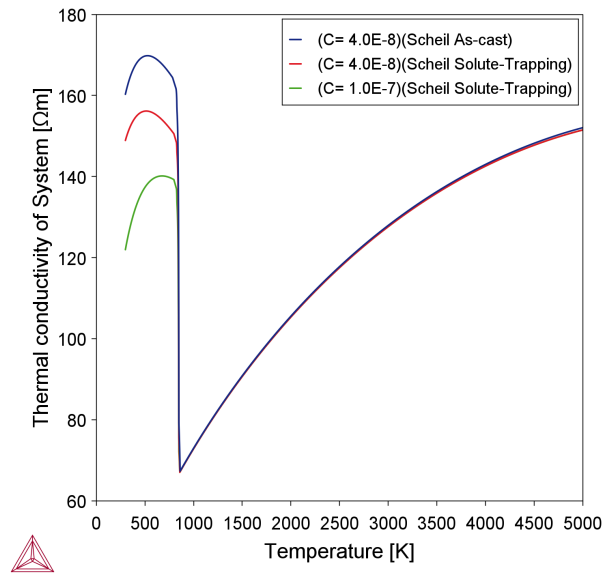


Figure 115: Thermal conductivity for as-cast substrate and printed substrate. Blue line shows conductivity for as-cast structure using the default setting for the interface scattering constant ( $C = 4.0E-8$ ). The red line shows the reduced conductivity due to the solute trapping. Finally, the green line shows further reduction in thermal conductivity due to increased interface scattering where  $C = 1.0E-7$ .

## Batch AM Steady-State Simulations

Two AM Calculators are next set up. These are renamed to *As-cast Batch* and *AM Batch*. These AM batch steady-state simulations are configured with similar settings, except for the interface scattering and the measured experimental melt pool dimensions that are different for the as-cast and LPBF fabricated substrates. The interface scattering settings are found on the **Material Properties** tab for both AM Calculators:

- The as-cast substrate uses the default interface scattering ( $C = 4.0E-8$ ).
- The LPBF as-printed substrate uses the increased value for interface scattering ( $C = 1.0E-7$ ).

The shared settings in the AM Calculator configuration (on the **Conditions** tab) are:

- Uses **Fluid flow including Marangoni effect** and the **keyhole model**.
- **Absorptivity** is selected as **Calculated** and the **with prefactor** is set to 1.8 times the calculated value in order to match the experimental melt pool dimensions.



The need to use a large prefactor is probably related to surface oxides and that the calculated absorptivity is for an oxide free liquid surface. The stable oxide  $\text{Al}_2\text{O}_3$  often forms at the surfaces of Al-alloys where it remains in solid form to above 2000 °C and evaporates well above 3000 °C.

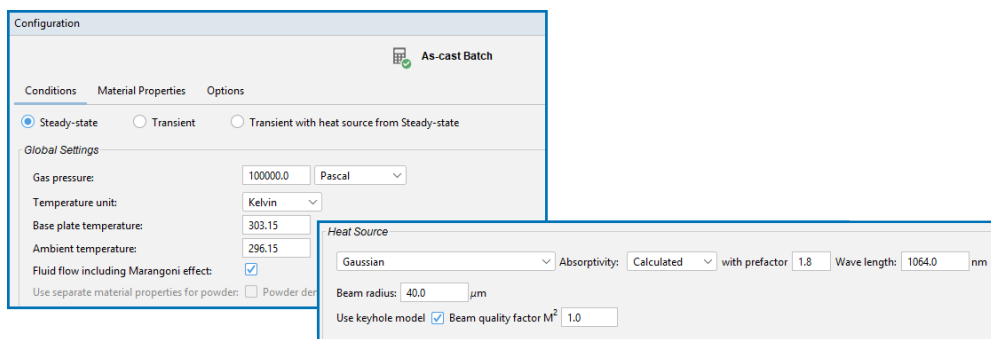


Figure 116: The settings window for an AM steady-state simulation for the As-cast Batch calculator. Fluid flow and Gaussian heat source with keyhole model are enabled, plus it uses a calculated absorptivity with a prefactor set to 1.8.



To reduce total calculation time, for each AM Calculator only 5 of the 12 experiments are selected on the Plot Renderer (click **Show Table** to see the details). However, the example still takes about two hours to perform.

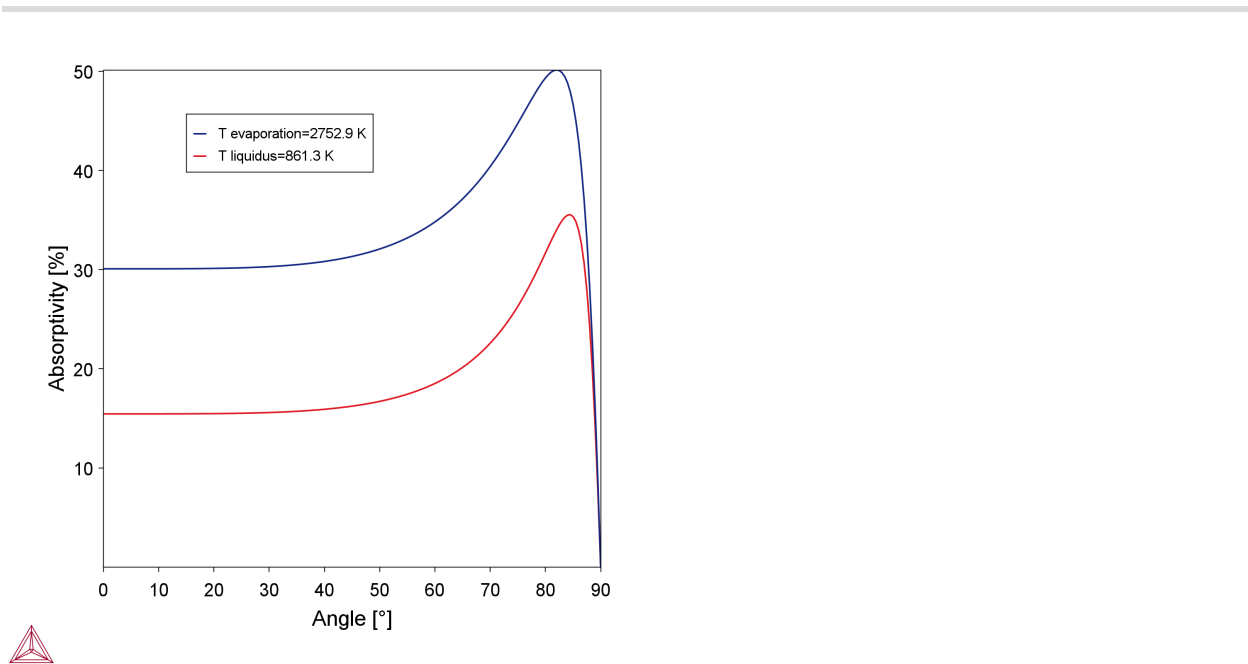


Figure 117: The temperature- and incident angle-dependent absorptivity. Red curve shows the absorptivity as a function of incident angle at the liquidus temperature, the blue curve shows the corresponding absorptivity at the evaporation temperature.

## Turbulent Flow - Large Eddy Simulations

Al-alloys have in general lower viscosity compared to other metallic alloys (e.g. Ti-, Fe- or Ni-based alloys). The low viscosity in combination with high fluid flow rates can result in turbulent flows. This happens in this example for the simulations at the higher energy densities. The turbulent flow model implemented in the AM simulation is based on *Large Eddy Simulations* (LES) where the large-scale motions are represented directly and smaller-scale motions are modeled. A larger Smagorinsky constant filters more small-scale motions to be modeled. The example increases the Smagorinsky constant to 0.5. The simulations with the highest energy densities will otherwise fail to converge with the default value of 0.18.

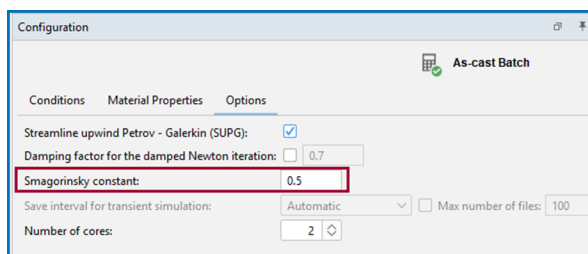


Figure 118: On the Options tab for the AM Steady-State simulation (As-cast Batch) with an increased Smagorinsky constant to handle the high turbulent flows.

The following parity plot examples compare the experimental versus calculated melt pool dimensions. In general a good trend can be seen for both the simulations with the as-cast and LPBF fabricated substrates. The experimental melt pools are in general deeper than the simulated with an increasing error with increasing depth of the melt pool.

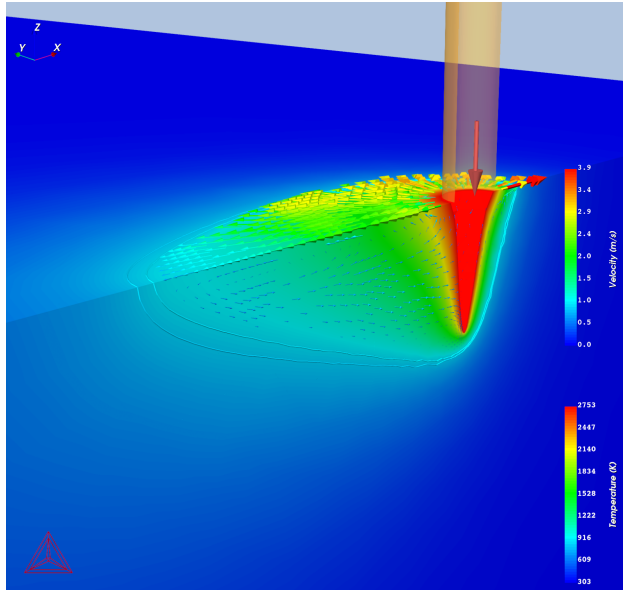


Figure 119: 3D plot of melt pool for as-cast substrate where  $P = 325 \text{ W}$  and scanning speed =  $600 \text{ mm/s}$ .

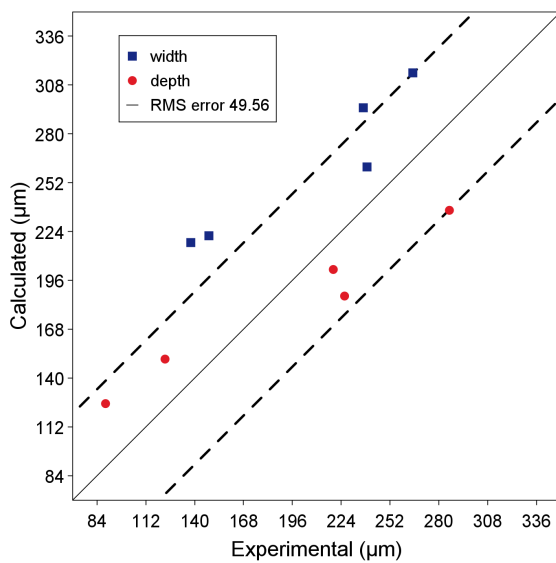


Figure 120: Parity plot for as-cast substrate when 5/12 experiments are calculated.

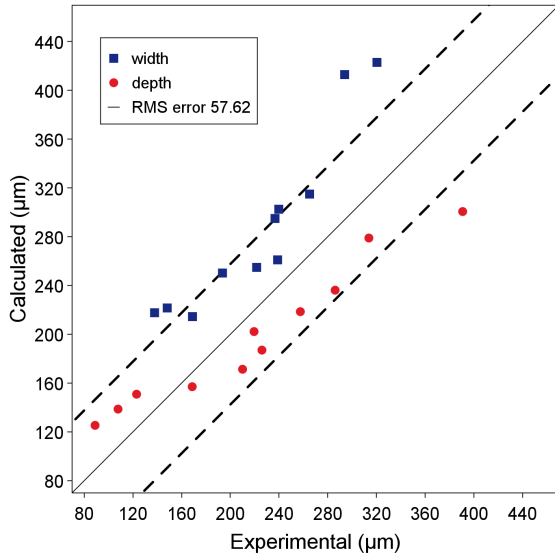
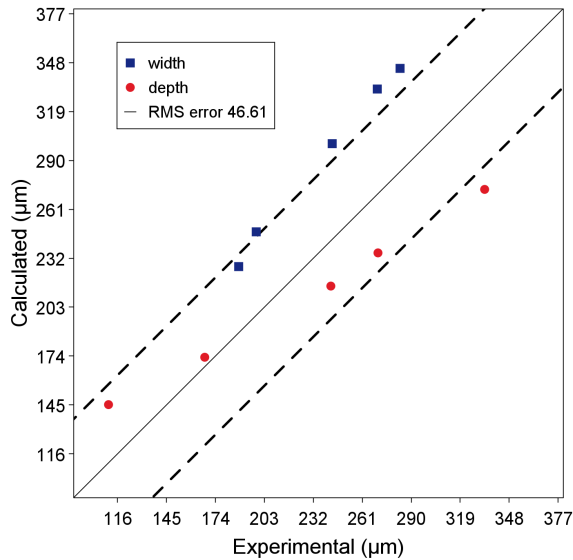


Figure 121: Parity plot for as-cast substrate when all 12 experiments are calculated.



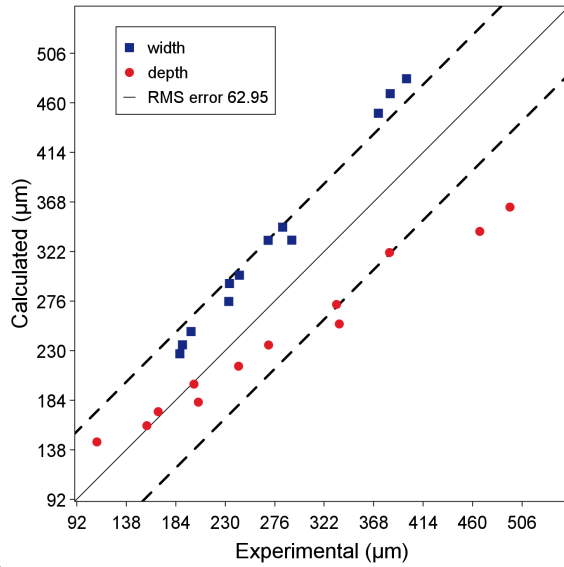


Figure 123: Parity plot for LPBF printed substrate when all 12 experiments are calculated.

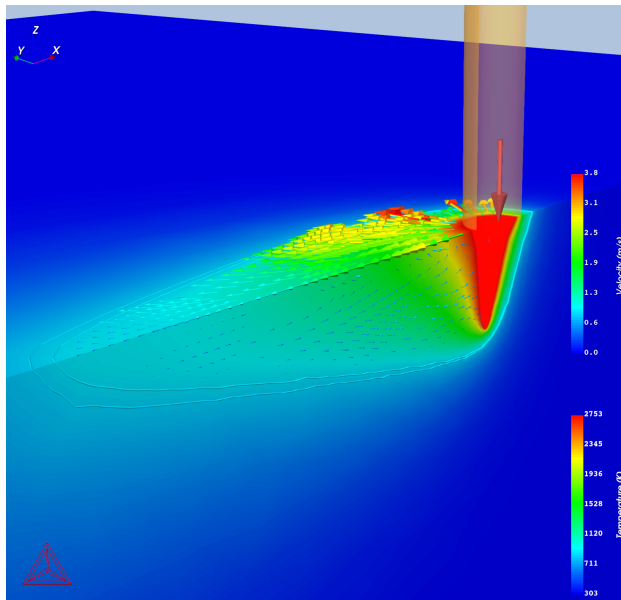


Figure 124: 3D plot of melt pool for LPBF printed substrate where  $P = 325 \text{ W}$  and scanning speed =  $1200 \text{ mm/s}$ .

## References

- [2021Lef] W. Lefebvre, G. Rose, P. Delroisse, E. Baustert, F. Cuvilly, A. Simar, Nanoscale periodic gradients generated by laser powder bed fusion of an AlSi10Mg alloy. Mater. Des. 197, 109264 (2021).

[2022Gha] A. Ghasemi, E. Fereiduni, M. Balbaa, M. Elbestawi, S. Habibi, Unraveling the low thermal conductivity of the LPBF fabricated pure Al, AlSi12, and AlSi10Mg alloys through substrate preheating. *Addit. Manuf.* 59, 103148 (2022).

[2024Bog] M. Bogdanova, S. Chernyshikhin, A. Zakirov, B. Zotov, L. Fedorenko, S. Belousov, A. Perepelkina, B. Korneev, M. Lyange, I. Pelevin, I. Iskandarova, E. Dzidziguri, B. Potapkin, A. Gromov, Mesoscale Simulation of Laser Powder Bed Fusion with an Increased Layer Thickness for AlSi10Mg Alloy. *J. Manuf. Mater. Process.* 8, 7 (2024).

## 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\_12: Using AM Calculator Probe Data with the Precipitation Module (TC-PRISMA)

This example demonstrates the coupling of the **AM Calculator** with the **Precipitation Calculator** via probe data from an AM simulation. Both a thermodynamic and mobility database is needed for the precipitation calculation, so the AM Material Library cannot be used. Instead, a **Scheil Calculator** is used to calculate the material properties used in the AM simulation.

The example simulates the incipient melting and re-precipitation of gamma prime in CMSX-4 during Selective Electron Beam Melting (SEBM) during a single pass of the beam. The SEBM process is simulated using an Electron Beam heat source. A transient with heat source from Steady-state simulation of a single track is simulated, with a cooling time of 150 s. The bed and ambient temperature is elevated to 950 °C. The process parameters are taken from Ramsperger et al. [2016aRam].

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: *AM\_12\_AM\_Probe\_to\_Precipitation.tcu*



A separate license is required to perform calculations with the Additive Manufacturing (AM) Module. This example also requires a license for the Precipitation Module (TC-PRISMA), plus additional database licenses for the TCS Ni-based Superalloys Database (TCNI) (TCNI13 and newer is recommended for best results), and the TCS Ni-alloys Mobility Database (MOBNI) (MOBNI6 and newer).

### Background

Ramsperger and Körner [2016bRam] demonstrated that it is possible to print single crystal CMSX-4 components using SLBM with an elevated bed temperature. The main challenge encountered is fracture during printing. The formation of gamma prime may incur a drop in ductility that contributes to fracture. Understanding the precipitation kinetics of gamma prime during 3D printing is important with regards to identifying optimum process parameters, and determining how best to heat treat the final component.

Wahlmann et al. [2019Wah] studied gamma prime kinetics in emulated SLBM thermal conditions, indirectly measuring the size of the gamma prime using X-ray diffraction during repeated nucleation and dissolution thermal cycles imitating the thermal history of an SLBM build. Figure 125 shows the calibration of the Precipitation Calculator to capture the measured kinetics.

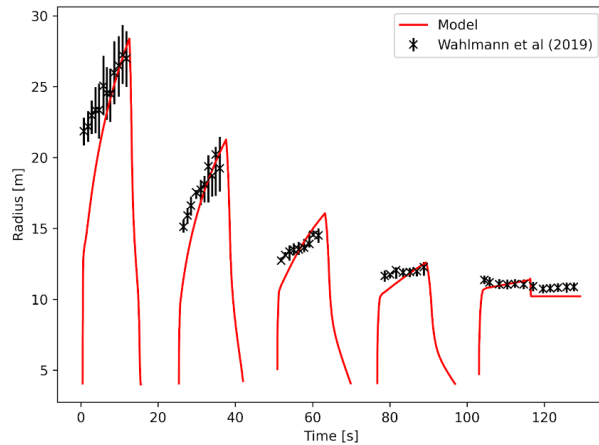


Figure 125: A comparison of the indirectly measured gamma prime size evolution during thermal heat treatment emulated an SLBM process with predictions from the Precipitation Calculator using the Thermo-Calc Ni-based superalloys databases (TCNI12 and MOBNI6) [2019Wah].

This AM Calculator to Precipitation Calculator example uses the calibrated parameters for gamma prime kinetics shown in Figure 125. The Precipitation Calculator includes the possibility for incipient melting of the gamma prime, which is predicted to occur during the first pass of the electron beam. Incipient melting describes the phenomena where solid-state precipitates are heated rapidly to a high temperature beyond their solvus temperature. The precipitates reach a temperature where they melt before the parent matrix phase. The Precipitation Calculator removes the gamma prime dispersion upon melting.

## Configuration and Calculation Set Up

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 **Precipitation 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 precipitation simulation.



Open the example to see the **Project** tree layout and review the settings for each activity. It is useful to click around in the interface to understand where the settings are located and configured. This section highlights the relevant settings for this example.

## System Definer

On the **System Definer**, the nickel-based superalloy CSMX-4 composition was approximated by Ni-9.8Co-6.4Ta-6.5Cr-6.4W-5.7Al-2.8Re-0.97Ti-0.62Mo-0.086Hf, (mass%). The thermodynamic (TCNI) and mobility (MOBNI) nickel-based superalloy databases were selected, with the phase **DIS\_FCC\_A1** included.



You can include or exclude phases from the **Phases and Phase Constitution** tab on the System Definer.

## AM Calculator

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 **Geometry** of 5 mm x 5 mm x 5 mm (height x width x length). The **Base plate temperature** and **Ambient temperature** is set to 950 °C with a gas pressure of 1 Pa.

Additional settings are then made on the **Configuration** window as follows.

- In the *Heat Source* section, **Electron Beam** is selected with a **Power** of 1110 W, a **Calculated Absorptivity**, an **Acceleration voltage** of 60 kV, and a **Beam radius** of 250 µm. The **keyhole model** checkbox is selected.
- In the *Scanning Strategy* section, the **Scanning speed** is 2000 mm/s and the **Layer thickness** is 50 µm. A **Cooling time** of 150 s is included.
- In the *Top Boundary Conditions* section, the **Evaporation** checkbox is selected to include this in the model.
- In the *Probe Positions* section, a single probe is added to the center of the top surface of the component, below the powder layer. See the **Geometry** tab as shown in [Figure 126](#).

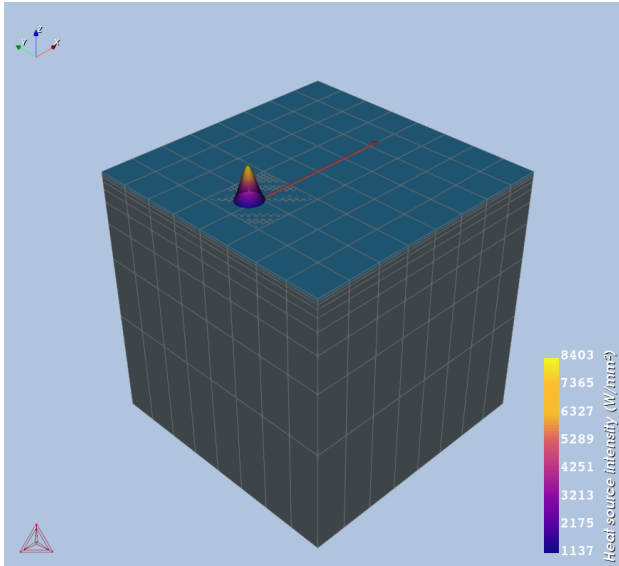


Figure 126: The full geometry set up for the single track AM simulation with a probe point added to the middle of the scan. Open the example to better see the set up and try working in the Visualizations window.

## Precipitation Calculator

The following highlights the relevant settings to make on the **Configuration** window for this example.

On the **Options** tab:

- Both the **Preprocess equilibrium data** and **Include incipient melting** checkboxes are selected.

On the **Conditions** tab, under *Matrix Phase*:

- **DIS\_FCC\_A1** is selected.



Also see "Selecting the Disordered Phase as a Matrix Phase" in the *Precipitation Module User Guide*, or search for this in the help.

- Click **Show Details**, then for **Mobility adjustment > Prefactor**, keep **Same for all elements** and enter 6.0 for the **Prefactor**.

On the **Conditions** tab, under *Precipitate Phase*:

- **FCC\_L12#2** is selected.
- **Nucleation sites = Bulk**
- **Interfacial energy** =  $0.035 \text{ J/m}^2$
- Click **Show Details**, then:
  - **Phase energy addition** =  $-55.0 \text{ J/mol}$
  - Select the **Preexisting size distribution** checkbox.
  - Click **Edit Particle Size Distribution**, and note the *Compositions* entered.

Compositions	
Dependent component:	Ni <input type="button" value="v"/> Mass percent
Al	7.585
Co	5.382
Cr	2.6
Hf	0.135
Mo	0.288
Re	0.394
Ta	9.568
Ti	1.455
W	4.525



In the **Preexisting Particle Size Distribution** window you can also adjust other settings. In this case, the size distribution is approximated by a Weibull distribution function with a mean radius of 200 nm, an Alpha of 2.0, and a fraction offset of 0.5 Search for "Particle Size Distribution (PSD)" in the *Precipitation Module User Guide*, or in the help.

On the **Conditions** tab, under *Calculation Type*:

- **Temperature unit = Kelvin**
- **Start time = 0.0**
- **Simulation time = 150.01 Seconds**
- **Probes: Probe1**
- **Condense time-temperature data = None**

## Visualizations



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](#).

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.

There is a variety of information shown in the **Visualizations** window that can be viewed during configuration and after performing the calculation(s).

Figure 127 and Figure 128 show the predicted evolution of the precipitate dispersion including the change in mean radius and volume fraction, respectively. The second Y axis in both figures shows the temperature. The initial gamma prime size distribution describes the gamma prime that has formed during the production of the AM powder. It is too large to dissolve during the rapid heating as the electron beam heats the component, and melts before the matrix. The gamma prime re-precipitates upon resolidification and cooling below the gamma prime solvus, and coarsens whilst held at elevated temperature. The incipient melting of the initial size distribution is clearly evident.

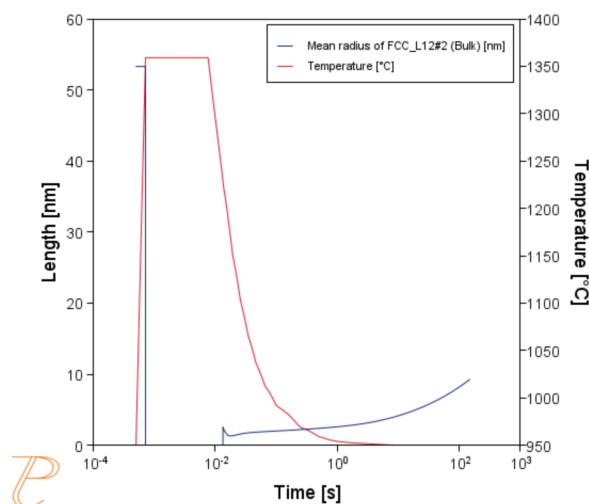


Figure 127: The mean radius and temperature as a function of time, with the mean radius on the left axis, and the temperature on the right axis.

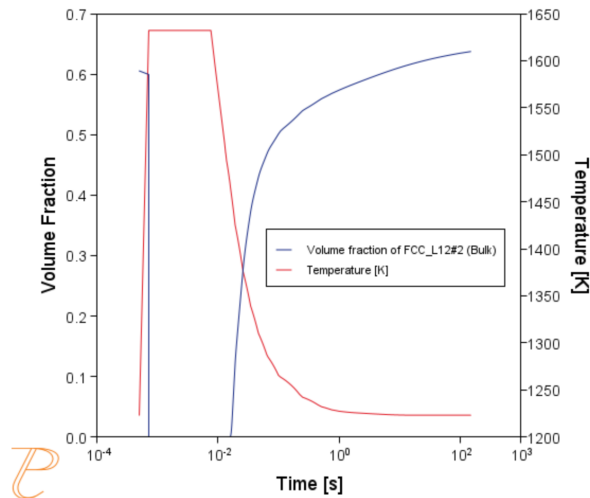


Figure 128: The volume fraction and temperature as a function of temperature, with the volume fraction on the left axis, and the temperature on the right axis.

The gamma prime distribution just prior to incipient melting is observed at 0.004 s. It has not dissolved significantly during the rapid heating as the electron beam passes. At 1 s the material has solidified and cooled to a temperature where gamma prime has nucleated and grown. The size distribution at the end of the simulation is shown at 150.1 s. Figure 129 shows the size distributions at these times during the single track AM simulation.

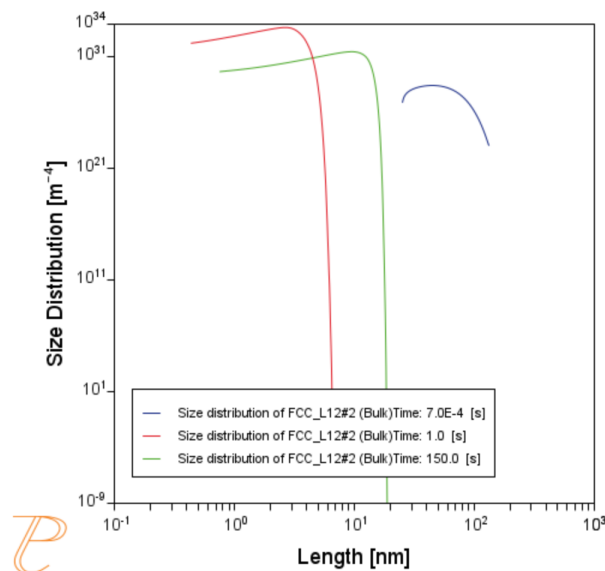


Figure 129: The size distributions of gamma prime at key points during the simulation.

---

## References

- [2016aRam] M. Ramsperger, R. F. Singer, C. Körner, Microstructure of the Nickel-Base Superalloy CMSX-4 Fabricated by Selective Electron Beam Melting. *Metall. Mater. Trans. A* 47, 1469–1480 (2016).
- [2016bRam] M. Ramsperger, C. Körner, “Selective Electron Beam Melting of the Single Crystalline Nickel-Base Superalloy CMSX-4®: From Columnar Grains to a Single Crystal” in *Superalloys 2016* (John Wiley & Sons, Inc., Hoboken, NJ, USA, 2016), pp. 341–349.
- [2019Wah] B. Wahlmann, F. Galgon, A. Stark, S. Gayer, N. Schell, P. Staron, C. Körner, Growth and coarsening kinetics of gamma prime precipitates in CMSX-4 under simulated additive manufacturing conditions. *Acta Mater.* 180, 84–96 (2019).

## 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.

## AM\_13: Using the Core-ring Beam Shape

This **Steady-state** example demonstrates the use of **Core-ring Heat Source** model available in the AM Module with **Batch** calculations.

The heat source parameters for the **Core-ring** heat source, as well as the processing parameters (power and speed), are taken from Holla et al. [2024Hol]. The example also compares the predicted melt pool dimensions with the experimental data from this paper.



For TC-Python users, there is an example using **Core-ring**: `pyx_AM_09_CoreRing_BeamShape.py`.



There is a companion example in Graphical Mode using **Top-hat**: "AM\_14: Using the Top-hat Beam Shape" on page 301.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: `AM_13_CoreRing_BeamShape.tcu`



This example requires an Additive Manufacturing (AM) Module license.

### Background

In a core-ring heat source, the energy is distributed between the central core and the surrounding ring, allowing for more precise control of heat input and a more uniform temperature distribution, reducing the risk of unstable keyhole formation [2023Bi]. Furthermore, a wider melt pool with the **Core-ring** beam shape as compared to a Gaussian profile allows enlargement of the hatch distances and together with the possibility of using higher scanning speeds, results in faster printing process and increased productivity [2021Grü].



"About the Heat Source Models" on page 29

The inclusion of a **Core-ring** heat source model in the AM Module, together with the keyhole model, allows you to predict the temperature evolution and melt pool shape along with the transition from conduction mode to keyhole mode for a **Core-ring** heat source. The experimental data taken from the literature [2024Hol] demonstrates the use of a **Core-ring** heat source for single track experiments on stainless steel 316L bare plates. A wide range of process parameters (power = 200 W – 900 W and scanning speed = 200 mm/s – 800 mm/s) also shows different cases for both conduction as well as the keyhole mode, which are correctly predicted in this example by the simulation results.

## Material Properties

- SS316L: Fe-17.0Cr-12.0Ni-2.5Mo-0.03C Mass percent
- Database: TCFE13 and with interface scattering =  $1e-07$ .
- The material properties are precalculated, and stored as a built-in material library with the Additive Manufacturing (AM) Module.

## AM Calculator Configuration Settings

This example contains one AM Calculator, which is renamed to **Core-ring\_20/80**. On the **Configuration** window, **Conditions** tab, this **Steady-state** calculation is configured with the **Core-ring Heat Source** with the **Keyhole model** and includes **Fluid flow**.

The **Custom Mode** is used in order to set all the heat source parameters as given in Table 2 of [2024Hol]. Individual settings are entered to match:

- **Beam radius-Core:** 47.6  $\mu\text{m}$
- **Beam radius-Ring:** 25.29  $\mu\text{m}$
- **Ring radius:** 68.4  $\mu\text{m}$

The **Amount of power in the ring** is set to **80%**, which is equivalent to 20/80 beam shape in [2024Hol].

The **Absorptivity** is set to **Calculated** with a **prefactor** of 1.2. It uses the **keyhole model** where the **Beam quality factor  $M^2$**  is kept as 1.0.

The **Batch Calculation Type** is used to set up all the conditions from the experiments in a single calculation. In the **Batch Experiment Data** section, the experimental **Power** and scan **Speed** as well as the measured melt pool Width (**Exp width**) and Depth (**Exp. depth**) were collected in a CSV file and imported into the software. This data is then saved in the project file.

In the **Batch Experiment Data** table, the **Power** ranges between 200 W to 900 W, and the scan **Speed** ranges between 200 mm/s to 800 mm/s.



"AM Calculator Conditions Settings" on page 111 and "AM Calculator Heat Source Settings" on page 123

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.



For more information also see "[Working with AM Visualizations](#)" on page 164.

## Parity Plot

This specific **Parity** plot is shown by clicking the **Parity Plot** node in the **Project** window. Then you can see on the **Visualizations** window it is in the tab of the same name. The settings are located on the **Configuration** window on the **Batch** tab.

The plot compares experimental and calculated melt pool width and depth for all experiments. The experiments are single tracks on bare plate SS316L with varied power and scan speed using a **Core-ring** heat source.

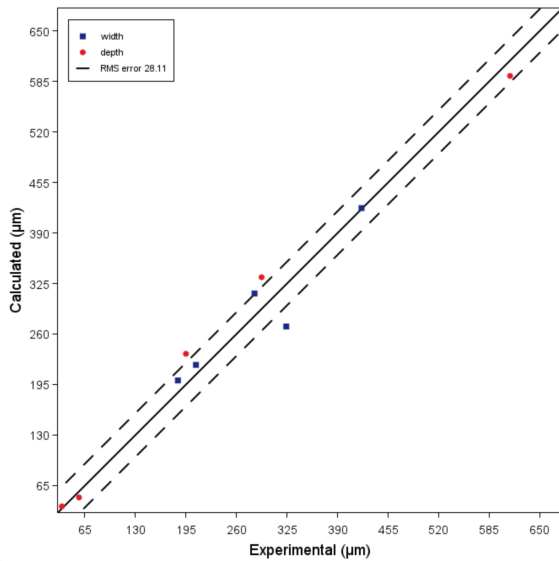


Figure 130: Parity plot comparing experimental and calculated melt pool width and depth for all the experiments when using a Core-ring heat source. The Root Mean Square (RMS) error can also be seen as a dashed line.

## 3D Plots

The specific **3D Plot** that is highlighted in this section is shown by clicking the **3D Plot** node in the **Project** window. Then on the **Visualizations** window on the tab of the same name, you can use various toolbar buttons to navigate and experiment with the views, for example:



### Zoom to Heat Source position

In [Figure 131](#), and using a combination of the **Plot Renderer Configuration** window settings on the **3D Plot** tab and the **Visualizations** window, different zoom levels and adjusting the **Glyph scale factor** setting helps to clearly show the melt pool and velocity vectors.

Further, the plot on the left is displayed by choosing the applicable **Batch experiments** results from the list (P=200 W, V=500 mm/s) and this set of experiments undergoes melting in conduction mode. Then the plot on the right is displayed by choosing the applicable **Batch experiments** results (P=600 W, V=200 mm/s) and that one forms a keyhole.

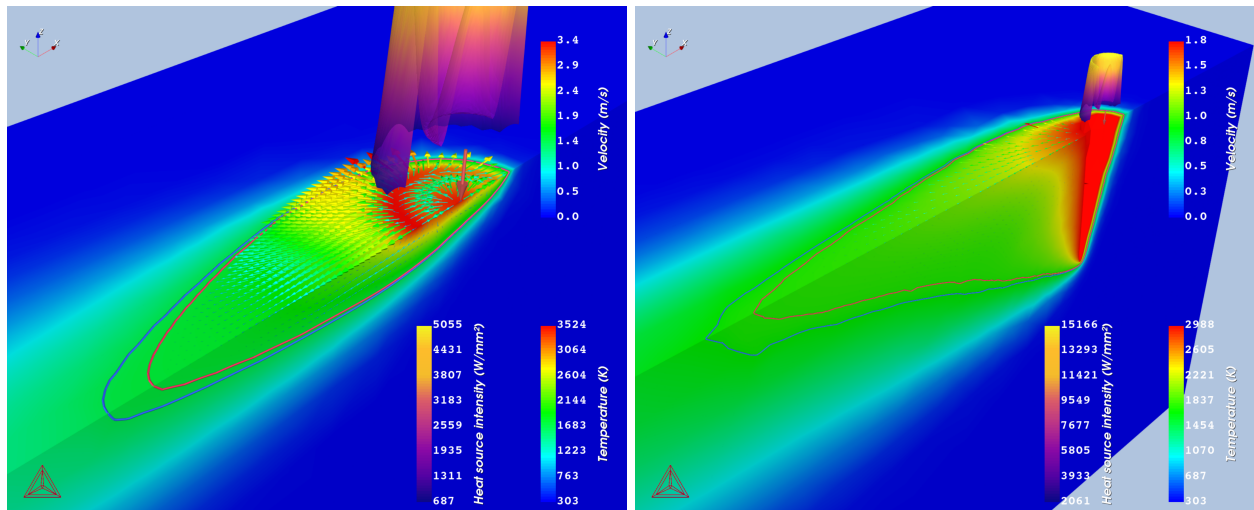
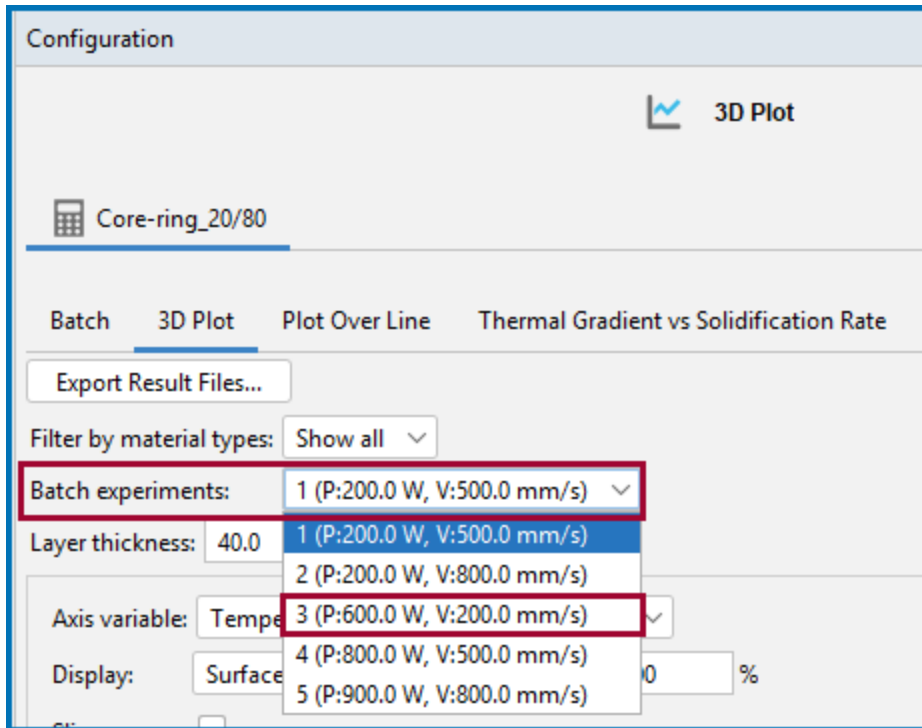




Figure 131: A comparison of the melt pools for two different processing conditions: Red and blue isocontours show liquidus and solidus temperatures, respectively. See the text for details.

In Figure 132 you can click the  **Set View** button plus  **Mirror Geometry** button to obtain this top view of a **Surface colormap**.

In general, you can play around with the **3D Plot Configuration** window settings. For example, for the right-hand side figure, you can view the **Flow velocity field** with different **Display** types such as an **Arrow** (shown).

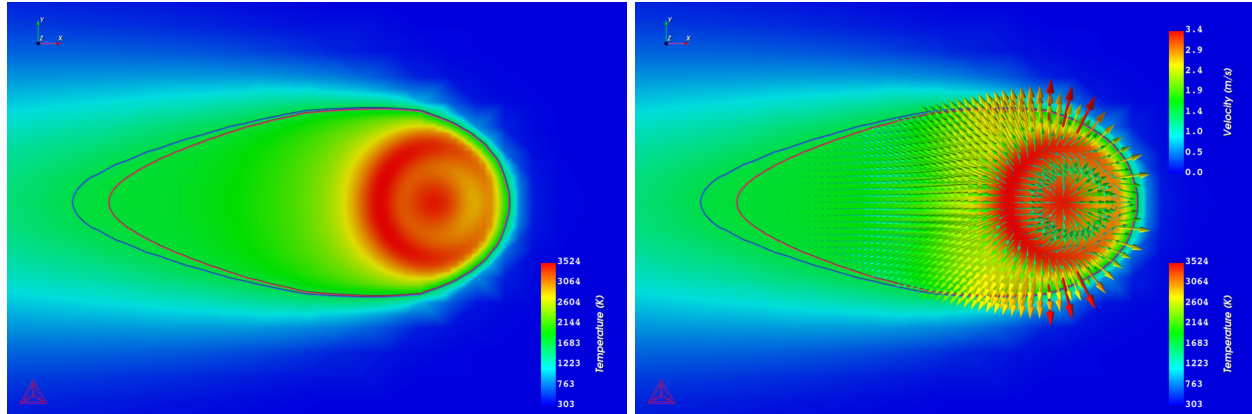


Figure 132: Top view (XY) of the surface colormap showing (left) temperature distribution due to the 20/80 Core-ring beam shape, and (right) velocity field caused by the distinctive temperature profile, resulting in widening of the melt pool.  $P=200$  W,  $V=500$  mm/s. Red and blue isocontours show liquidus and solidus temperatures, respectively.

In Figure 133, this cross-section compares the melt pool shape and size for **Batch experiment 1** ( $P=200$  W,  $V=500$  mm/s) to the experimental results as shown in Figure 9 in the paper by [2024Hol].

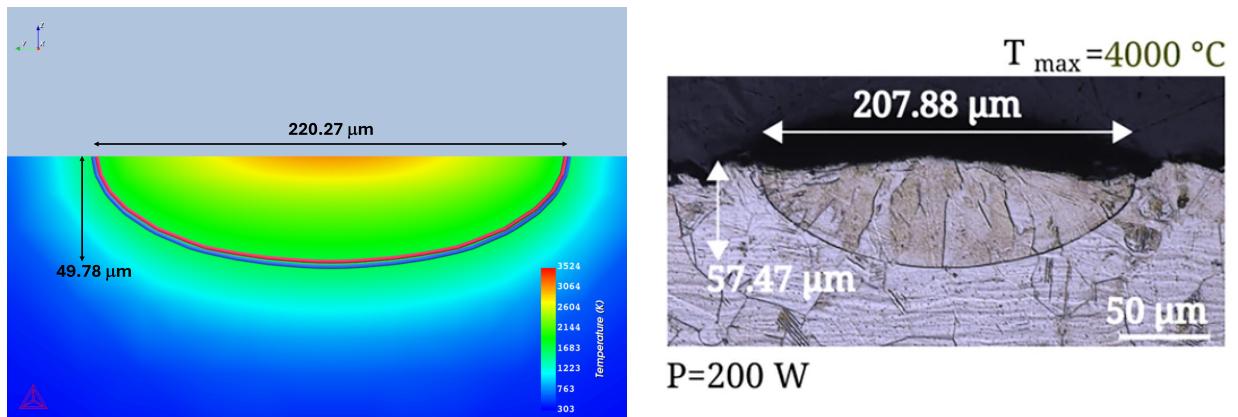


Figure 133: A cross-section comparing melt pool shape and size for  $P=200$  W,  $V=500$  mm/s from (left) simulation and (right) experiments [2024Hol-Fig9]. Red and blue isocontours in the simulation result show liquidus and solidus temperatures, respectively.

In Figure 134, the formation of the melt pool is shown on the left and compared to a cross-section that is taken from **Batch experiment 3** ( $P=600$  W,  $V=200$  mm/s) to show the formation of a keyhole and the maximum melt pool size, where you can also see the widening of the melt pool due to fluid flow. Compare this to Figure 135, which are the experimental results as shown in Figure 9 in the paper by [2024Hol].

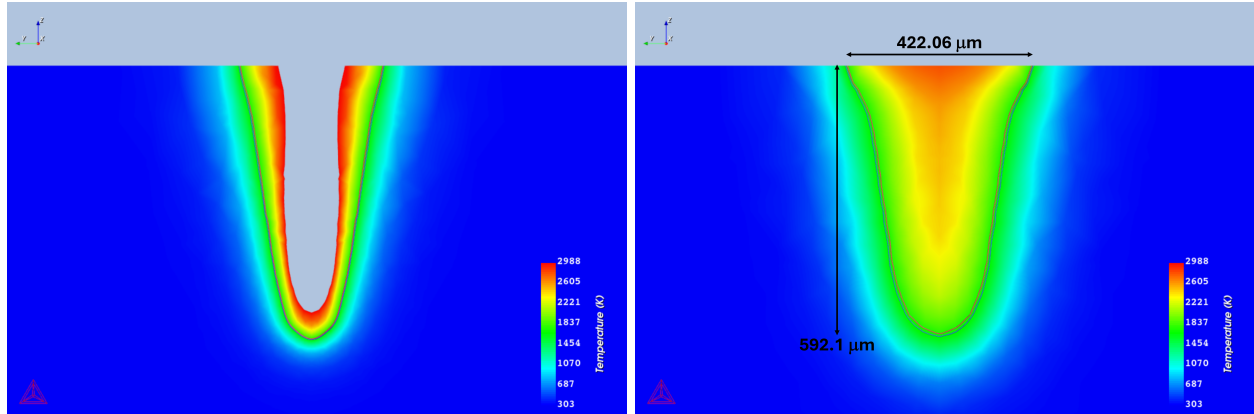


Figure 134: Cross-sections at different locations along the scanning direction showing (left) formation of keyhole and (right) maximum melt pool dimensions for  $P=600\text{ W}$ ,  $V=200\text{ mm/s}$ . See the text for more.



Figure 135: Cross-section showing the experimental melt track for the same processing conditions shown in Figure 134. From [2024Hol-Fig9].

## 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](#).

## References

- [2021Grü] J. Grünwald, F. Gehringer, M. Schmöller, K. Wudy, Influence of Ring-Shaped Beam Profiles on Process Stability and Productivity in Laser-Based Powder Bed Fusion of AISI 316L Metals (Basel). 11, 1989 (2021).
- [2023Bi] J. Bi, L. Wu, S. Li, Z. Yang, X. Jia, M. D. Starostenkov, G. Dong, Beam shaping technology and its application in metal laser additive manufacturing: A review. J. Mater. Res. Technol. 26, 4606–4628 (2023).
- [2024Hol] V. Holla, J. Grünwald, P. Kopp, P. M. Praegla, C. Meier, K. Wudy, S. Kollmannsberger, Validity of Thermal Simulation Models for Different Laser Beam Shapes in Bead-on-Plate Melting. Integr. Mater. Manuf. Innov. 13, 969–985 (2024).
- [2024Hol-Fig9]: V. Holla et. al, Validity of Thermal Simulation Models for Different Laser Beam Shapes in Bead-on-Plate Melting. Integr. Mater. Manuf. Innov. 13, 969–985 (2024). Creative Commons Attribution [4.0 International License](#), Figure 9, accessed May 2025, cropped individual images from source. <https://link.springer.com/article/10.1007/s40192-024-00382-2/figures/9>

## AM\_14: Using the Top-hat Beam Shape

This **Steady-state** example demonstrates the use of **Top-hat Heat Source** model in the AM Module with **Batch** calculations.

The processing parameters (power and speed) and other processing conditions such as the layer thickness are taken from Sow et al. [2020Sow]. The example also compares the predicted melt pool dimensions with the experimental data.



For TC-Python users, there is an example using **Top-hat**: `pyx_AM_10_TopHat_BeamShape.py`.



There is a companion example in Graphical Mode using **Core-ring**: "[AM\\_13: Using the Core-ring Beam Shape](#)" on page 293.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: `AM_14_TopHat_BeamShape.tcu`



This example requires an Additive Manufacturing (AM) Module license.

### Background

The primary benefit of the Top-hat heat source beam shape, in comparison to the Gaussian beam shape, is uniform energy distribution across the beam spot, leading to even heating of the material surface. With lower peak intensity as compared to the Gaussian beam, a Top-hat beam also allows the use of larger beam size that not only increases productivity but also reduces vapor-induced instabilities [2020Sow]. The inclusion of Top-hat heat source model in the AM Module, together with the keyhole model, allows you to predict the temperature evolution and melt pool shape along with the transition from conduction mode to keyhole mode for a Top-hat heat source. The example uses experimental data taken from the literature [2020Sow] and demonstrates the use of Top-hat heat source with a rather large beam size for single track experiments on an IN625 alloy.



"[About the Heat Source Models](#)" on page 29

## Material Properties

- IN625: Ni-21.5Cr-5.0Fe-0.5Si-0.5Mn-9.0Mo-0.4Ti-1.0Co-1.8Nb-1.8Ta-0.4Al Mass percent
- Database: TCNI12 and with solute trapping and interface scattering = 1e-07.
- The material properties are precalculated, and stored as a built-in material library with the Additive Manufacturing (AM) Module.

## AM Calculator Configuration Settings

This example contains one AM Calculator, which is renamed to **Top-hat**. On the **Configuration** window, **Conditions** tab, this **Steady-state** calculation is configured with the **Top-hat Heat Source** with the **Keyhole model**.



Fluid flow is not used in this example (the **Fluid flow including Marangoni effect** checkbox is not selected). Although fluid flow plays a crucial role in determining the shape and size of the melt pool, in this case it was found that it overestimates the melt pool width for most cases when fluid flow is included.

In the *Geometry* section, the **Height** of the base plate is set to 3.0 mm and in the *Scanning Strategy* section the **Layer thickness** is changed to 50.0  $\mu\text{m}$ .

In the *Heat Source* section, the **Beam radius** is set to 450  $\mu\text{m}$ .



In the [2020Sow] paper, it is mentioned that the beam diameter for the top-hat distribution is 500  $\mu\text{m}$ . However, in Figure 2 of this paper, where intensity is plotted against the distance for the top-hat beam, the actual diameter looks much larger than that. In order to match the intensity distribution, the beam radius is therefore set to 450  $\mu\text{m}$ .

Also in the *Heat Source* section, the **Absorptivity** is set to **Calculated** with a **prefactor** of 1.0 and in the keyhole model, **Beam quality factor  $M^2$**  is kept as 1.0.

The **Batch Calculation Type** is used to set up all the conditions from the experiments in a single calculation. In the *Batch Experiment Data* section, the experimental **Power** and scan **Speed** as well as the measured melt pool Width (**Exp width**) and Depth (**Exp. depth**) were collected in a CSV file and imported into the software. This data is then saved in the project file.

Data with the error bars is also read and stored in the project file, that is later used to plot error bars together with the experimental values of melt pool width and depth.

In the *Batch Experiment Data* table, the **Power** is always 1000 W and the scan **Speed** ranges between 25 mm/s to 500 mm/s.



"AM Calculator Conditions Settings" on page 111 and "AM Calculator Heat Source Settings" on page 123

## Visualizations

Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



When you run (Perform) this example, it takes a few minutes for the calculations to complete.



For more information also see "[Working with AM Visualizations](#)" on page 164.

## Parity Plot

This specific **Parity** plot is shown by clicking the **Parity Plot** node in the **Project** window. Then you can see on the **Visualizations** window it is in the tab of the same name. The settings are located on the **Configuration** window on the **Batch** tab.

The plot compares experimental and calculated melt pool width and depth. It can be seen that the calculated melt pool width shows a fairly good agreement with the measured values. A mixed trend is however, seen for the melt pool depth. For lower energy densities, the calculated and measured melt pool depth matches quite well, however it starts to deviate once the melting appears to happen in the keyhole mode in the experiments, while it is still in the conduction mode in the calculations. For the last three points, where the calculations also predict a keyhole mode, the melt pool depth matches quite well with the measured value, except for the very last point where it is somehow overpredicted. The difference in depth could be attributed to the uncertainty in the actual beam size of the top-hat beam.

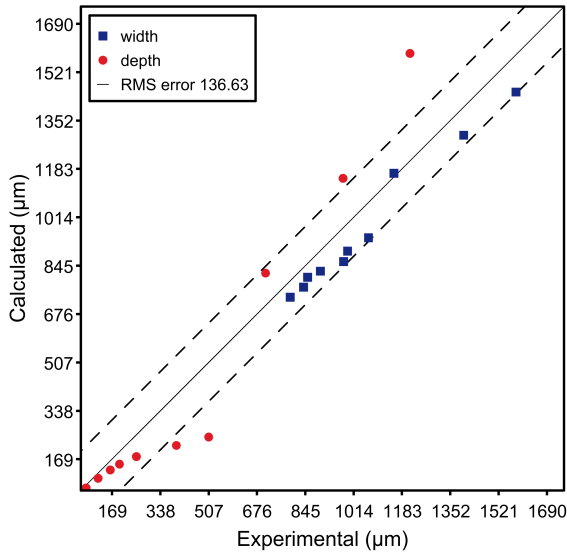


Figure 136: Parity plot comparing experimental and calculated melt pool width and depth for all the experiments. The experiments are single tracks on IN625 with constant power and varied scan speed using the Top-hat heat source. The Root Mean Square (RMS) error can also be seen as a dashed line.

## Melt Pool Dimensions

This **Melt Pool vs Energy Density** plot is shown by clicking the **Melt Pool Dimensions** node in the **Project** window. Then you can see on the **Visualizations** window it is in the tab of the same name. The settings are located on the **Configuration** window on the **Batch** tab.

The plot shows a comparison of the experimental and calculated melt pool width and depth, where the error bars are also plotted with the measured data. Rather large error bars, especially for the higher energy density cases, show that the overall trend for the calculated values of melt pool width and depth is in good agreement with the measured values. Some improvements are however needed for the low energy density cases where a huge deviation is found between the calculated and measured values.

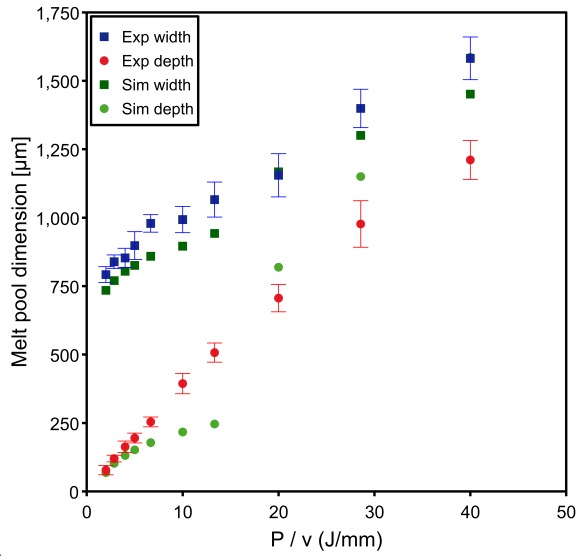


Figure 137: A comparison of the experimental and calculated melt pool width and depth for all the experiments. The error bars on the experimental data are also shown.

## 3D Plots

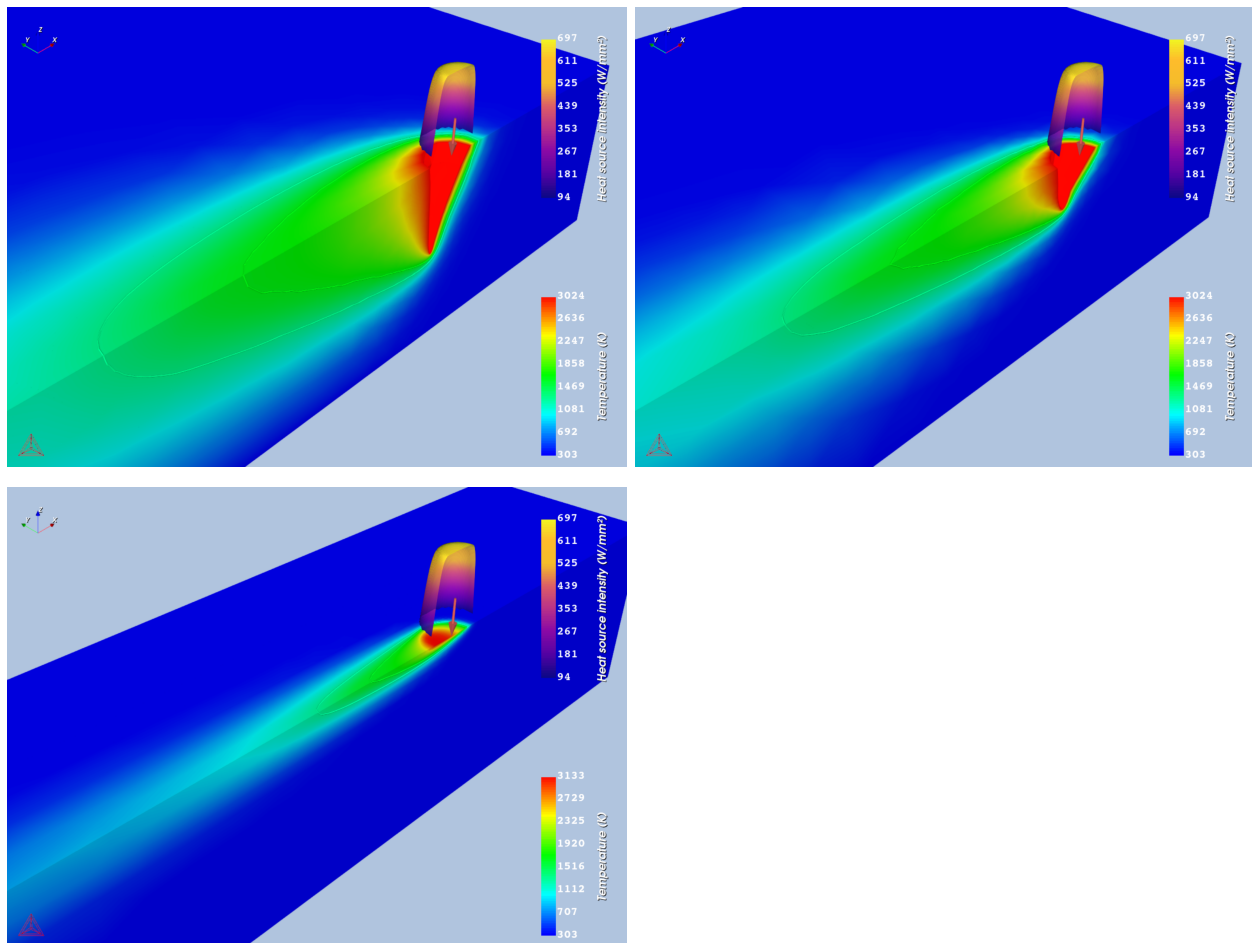


Figure 138: A comparison of the temperature distribution for three different processing conditions: (top-left)  $P=1000$  W,  $V=25$  mm/s, (top-right)  $P=1000$  W,  $V=50$  mm/s, both of which form keyhole during melting and (bottom-left)  $P=1000$  W,  $V=250$  mm/s that undergoes melting in conduction mode.

For the comparisons in Figure 139, all cross-sections are taken where the melt pools attain maximum width and depth. Red and blue isocontours show liquidus and solidus temperatures, respectively.

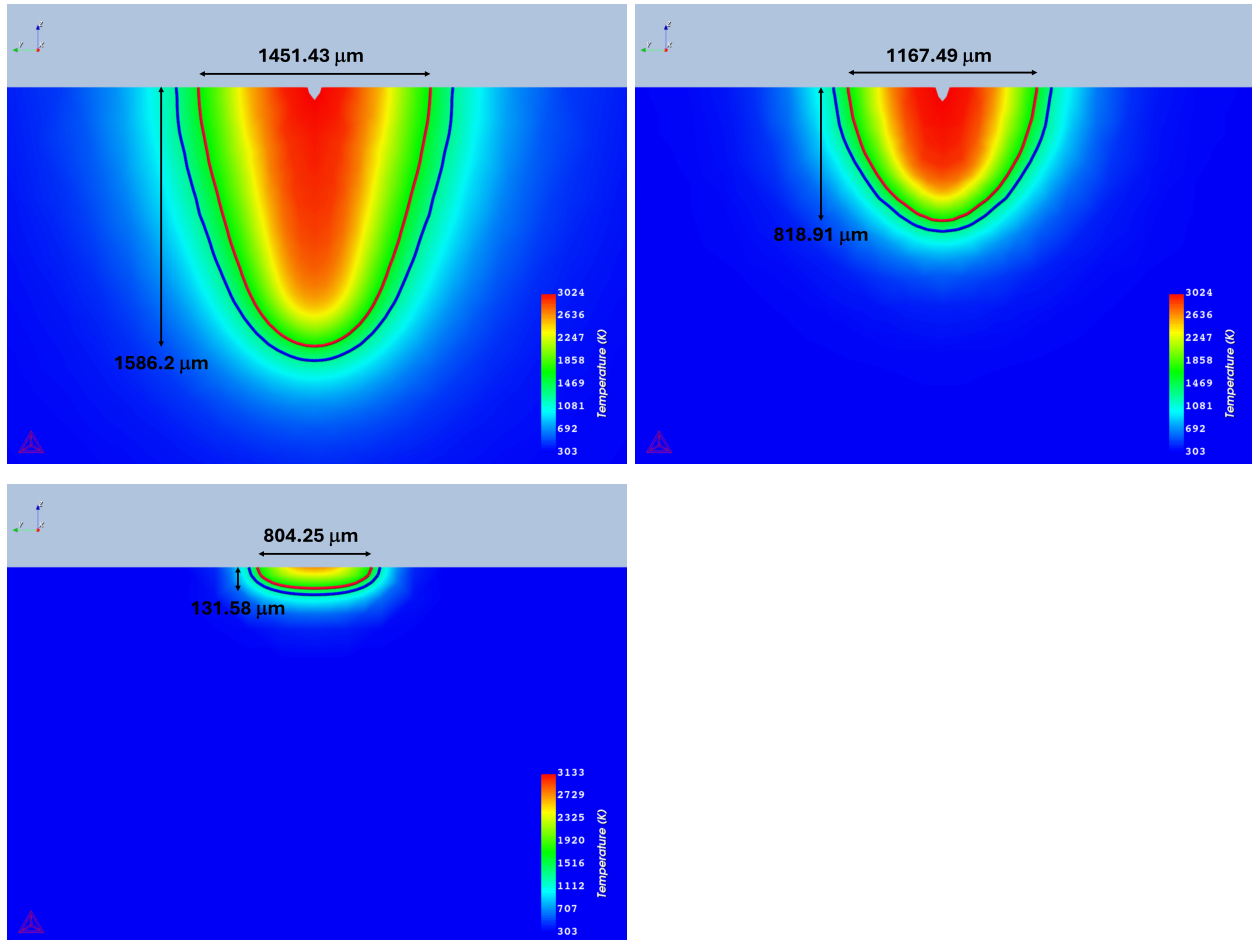


Figure 139: Cross-sections comparing melt pool shape and size for (top-left)  $P=1000$  W,  $V=25$  mm/s, (top-right)  $P=1000$  W,  $V=50$  mm/s and (bottom-left)  $P=1000$  W,  $V=250$  mm/s.

## 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](#).

## Reference

[2020Sow] M. C. Sow, T. De Terris, O. Castelnau, Z. Hamouche, F. Coste, R. Fabbro, P. Peyre, Influence of beam diameter on Laser Powder Bed Fusion (L-PBF) process. *Addit. Manuf.* 36, 101532 (2020).

## AM\_15: Electron Beam Melting in an IN718 Alloy

This steady-state example demonstrates the use of **Electron Beam Heat Source** for **Batch** calculations.

The experimental data as well as the process conditions are taken from [2019Din]. The steady-state calculations represent single bead experiments where electron beam was used to scan an IN718 base plate at 1025 °C. The example also compares the predicted melt pool dimensions with the experimental data from this paper.



For TC-Python users, there is a comparable example: `pyex_AM_12_Electron_Beam_Melting_IN718.py`.

### Project File and License Information

- Folder: **Additive Manufacturing**
- File name: `AM_15_Electron_Beam_Melting_IN718.tcu`



This example requires an Additive Manufacturing (AM) Module license.

### Background

In an Electron Beam melting (EBM) process, a high energy electron beam is used to selectively melt the powder layer inside a vacuum chamber. In the Additive Manufacturing (AM) Module, the energy distribution from the electron beam is approximated using a Gaussian profile. In the paper [2019Din], authors performed a series of single bead experiments on the IN718 base plate (Case B in the paper) using an A2X EBM system (Arcam AB). The process was performed under a controlled vacuum.



"About the Heat Source Models" on page 29

---

## Material Properties

- IN718: Ni-0.05C-0.14Mn-0.08Si-18.27Cr-0.57Al-2.9Mo-5.15Nb-0.01Ta-1.04Ti-0.79Co-19.18Fe-0.04Cu Mass percent
- Database: TCNI12 and with interface scattering =  $1e-07$ . The material properties are precalculated, and stored as a built-in material library with the Additive Manufacturing (AM) Module.

## AM Calculator Configuration Settings

This example contains one AM Calculator, which is renamed to **Electron Beam Melting**. On the **Configuration** window, **Conditions** tab, this **Steady-state** calculation is configured with the **Electron Beam** Heat Source with the **Keyhole model** and includes **Fluid flow**.

The **Gas pressure** is set to 0.2 Pascal. The **Base plate temperature** as well as the **Ambient temperature** are set to 1025 °C. The **Height** of the base plate is set to 3.0 mm.

The **Absorptivity** of the Electron Beam heat source is set to be **Calculated** with a **prefactor** of 1.0. The **Acceleration voltage** of the beam, as mentioned in the paper, is set 60.0 kV.

The **Beam radius** of the electron beam is not mentioned in the experimental setup of the paper. We have set the Beam radius to 125.0  $\mu\text{m}$ , which is also the minimum beam radius of the A2X EBM system. The **Convective heat transfer coefficient** is set to 0.0  $\text{W}/\text{m}^2$  to eliminate any heat loss due to convection in the vacuum chamber.

Experimental data for the **Batch** calculation is taken from Case B in the paper (single bead experiments on a bare metal plate). Data for low scanning speed i.e. < 300 mm/s is not included in the example.

## Visualizations

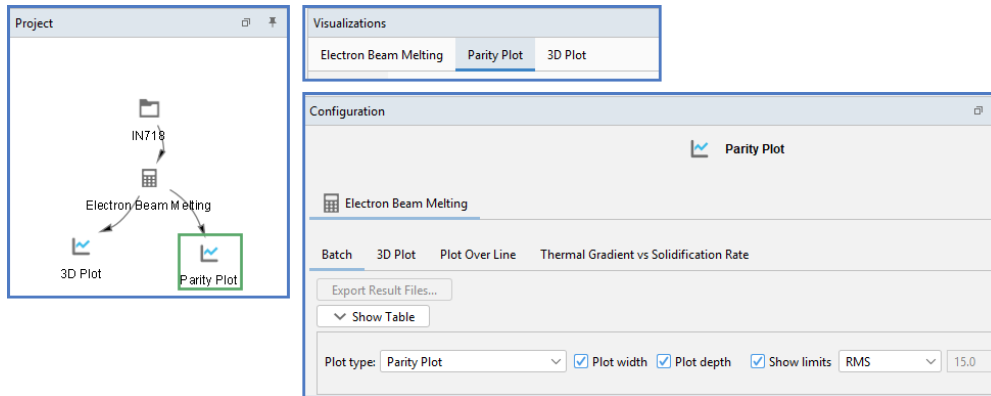
Open the example project file to review the node setup on the **Project** window and the associated settings on the **Configuration** window for each node. For some types of projects, you can also adjust settings on the **Plot Renderer Configuration** window to preview results before performing the simulation. Click **Perform Tree** to generate plots and tables to see the results on the **Visualizations** window.



For more information also see "[Working with AM Visualizations](#)" on page 164.

## Parity Plot

This specific Parity plot is shown by clicking the **Parity Plot** node in the **Project** window. Then you can see on the **Visualizations** window it is in the tab of the same name. The settings are located on the **Configuration** window on the **Batch** tab.



The plot compares experimental and calculated melt pool width and depth for all experiments. The experiments are single bead experiments on bare plate IN718 with varied power and scan speed using an **Electron Beam** heat source.

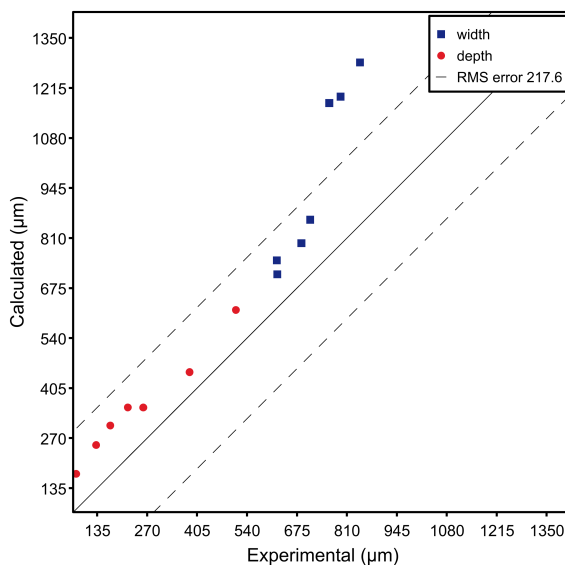


Figure 140: Parity plot comparing experimental and calculated melt pool width and depth for all the experiments when using an Electron Beam heat source. The Root Mean Square (RMS) error can also be seen as a dashed line.

## 3D Plots

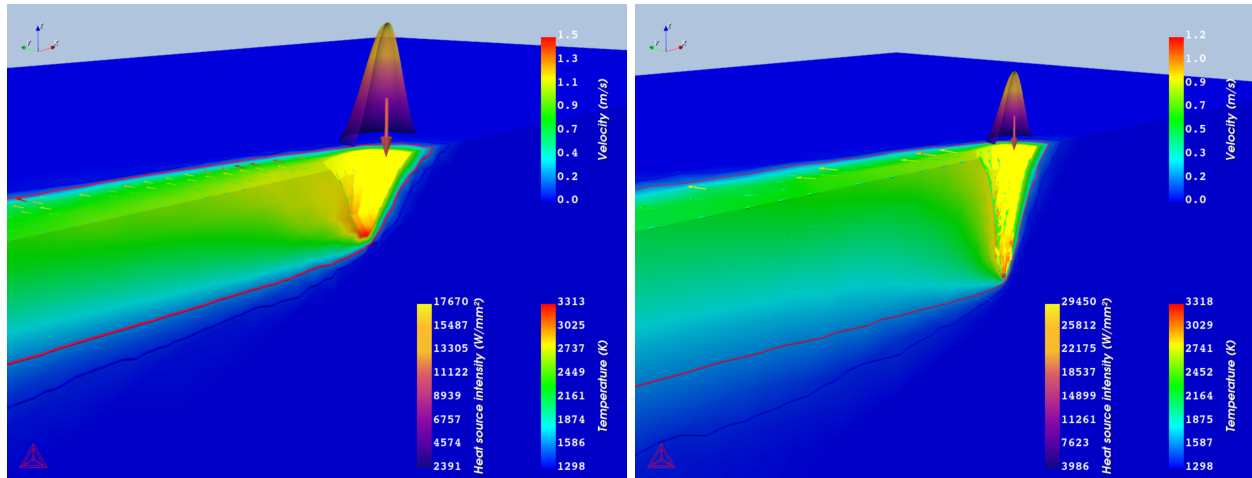


Figure 141: A comparison of the keyhole profile for two different processing conditions (left)  $P=600$  W,  $v=1000$  mm/s, and (right)  $P=1000$  W,  $v=300$  mm/s.

## 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](#).

## Reference

[2019Din] X. Ding, Y. Koizumi, D. Wei, and A. Chiba, “Effect of process parameters on melt pool geometry and microstructure development for electron beam melting of IN718: A systematic single bead analysis study,” *Addit. Manuf.*, vol. 26, pp. 215–226, Mar. 2019.