Created in COMSOL Multiphysics 6.0

Cooling and Solidification of Metal

Introduction

This example is a model of a continuous casting process. Liquid metal is poured into a mold of uniform cross section. The outside of the mold is cooled and the metal solidifies as it flows through the mold. When the metal leaves the mold, it is completely solidified on the outside but still liquid inside. The metal then continues to cool and eventually solidify completely, at which point it can be cut into sections. This tutorial simplifies the problem somewhat by not computing the flow field of the liquid metal and assuming there is no volume change during solidification. It is also assumed that the velocity of the metal is constant and uniform throughout the modeling domain. The phase transition from molten to solid state is modeled via the apparent heat capacity formulation. Issues of convergence and mesh refinement are addressed for this highly nonlinear model.

The Continuous Casting — Arbitrary Lagrangian-Eulerian Method model is similar to this one, except that the velocity is computed from the Laminar Flow interface instead of being considered constant and uniform. For a detailed description of the application, see Continuous Casting — Arbitrary Lagrangian-Eulerian Method.

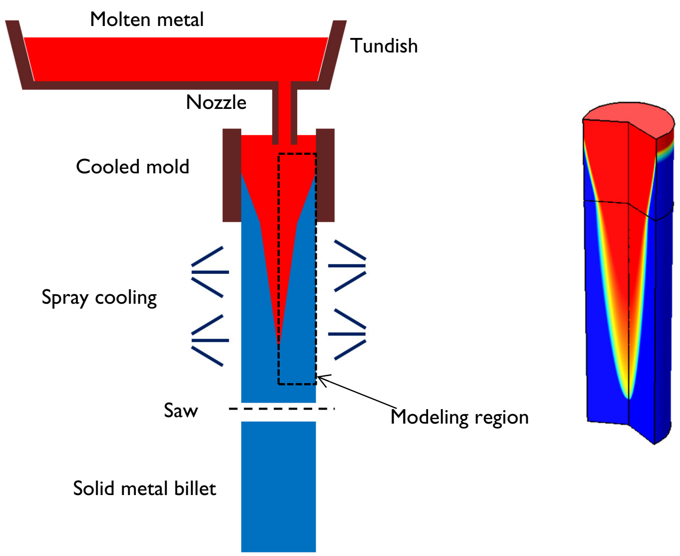


Figure 1: A continuous casting process. The section where the metal is solidifying is being modeled.

Model Overview

The model simplifies the 3D geometry of the continuous casting to a 2D axisymmetric model composed of two rectangular regions: one representing the strand within the mold, and one the spray cooled region outside of the mold, prior to the saw cutoff. In the second section, there is also significant cooling via radiation to the ambient. In this region it is assumed that the molten metal is in a hydrostatic state, that the only motion in the fluid is due to the bulk downward motion of the strand. This simplification allows the assumption of bulk motion throughout the domain.

Since this is a continuous process, the system can be modeled at steady state. The heat transport is described by the equation:

$$\rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot (-k \nabla T) = 0$$

where k and C_p denote thermal conductivity and specific heat, respectively. The velocity, $\mathbf{u}$, is the fixed casting speed of the metal in both liquid and solid states.

As the metal cools down in the mold, it solidifies. During the phase transition, a significant amount of latent heat is released. The total amount of heat released per unit mass of alloy during the transition is given by the change in enthalpy, ΔH. In addition, the specific heat capacity, C_p, also changes considerably during the transition.

As opposed to pure metals, an alloy generally undergoes a broad temperature transition zone, over several kelvins, in which a mixture of both solid and molten material coexist in a “mushy” zone. To account for the latent heat related to the phase transition, the Apparent Heat capacity method is used through the Heat Transfer with Phase Change domain condition. The objective of the analysis is to make ΔT, the half-width of the transition interval small, such that the solidification front location is well defined.

Table 1 reviews the material properties in this tutorial.

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	SYMBOL	MELT	SOLID
Density	ρ (kg/m^3)	8500	8500
Heat capacity at constant pressure	C_p (J/(kg·K))	530	380
Thermal conductivity	k (W/(m·K))	150	300

The melting temperature, T_{m}, and enthalpy, ΔH, are 1356 K and 205 kJ/kg, respectively.

This example is a highly nonlinear problem and benefits from taking an iterative approach to finding the solution. The location of the transition between the molten and solid state is a strong function of the casting velocity, the cooling rate in the mold, and the cooling rate in the spray cooled region. A fine mesh is needed across the solidification front to resolve the change in material properties. However, it is not known where this front will be.

By starting with a gradual transition between liquid and solid, it is possible to find a solution even on a relatively coarse mesh. This solution can be used as the starting point for the next step in the solution procedure, which uses a sharper transition from liquid to solid. This is done using the continuation method. Given a monotonic list of values to solve for, the continuation method uses the solution to the last case as the starting condition for the next. Once a solution is found for the smallest desired ΔT, the adaptive mesh refinement algorithm is used to refine the mesh to put more elements around the transition region. This finer mesh is then used to find a solution with an even sharper transition. This can be repeated as needed to get better and better resolution of the location of the solidification front.

In this example, the parameter ΔT is first ramped down from 300 K to 75 K, then the adaptive mesh refinement is used such that a finer mesh is used around the solidification front. The resultant solution and mesh are then used as starting points for a second study, where the parameter ΔT is further ramped down from 50 K to 25 K.

Results and Discussion

The solidification front computed with the coarsest mesh, and for $\Delta T = 75$ K, is shown in Figure 2. A wide transition between the molten and solid state is observed. The adaptive mesh refinement algorithm then refines the mesh along the solidification front because this is the region where the results are strongly dependent upon mesh size. This solution, and refined mesh, are used as the starting point for the next solution, which ramps the ΔT parameter down to 25 K. These results are shown in Figure 3.

The point of complete solidification moves slightly as the transition zone is made smaller. As the transition zone becomes smaller, a finer mesh is needed, otherwise the model might not converge. If it is desired to get an even better resolution of the solidification front, the solution procedure used here should be repeated to get an even finer mesh, and further ramp down the ΔT parameter.

The solid phase fraction is plotted along the r-direction at the line at the bottom of the mold in Figure 4, and Figure 5 shows the solid fraction along the centerline of the strand.

For smaller values of ΔT, the transition becomes sharper, and the model gives confidence that the metal is completely solidified before the strand is cut.

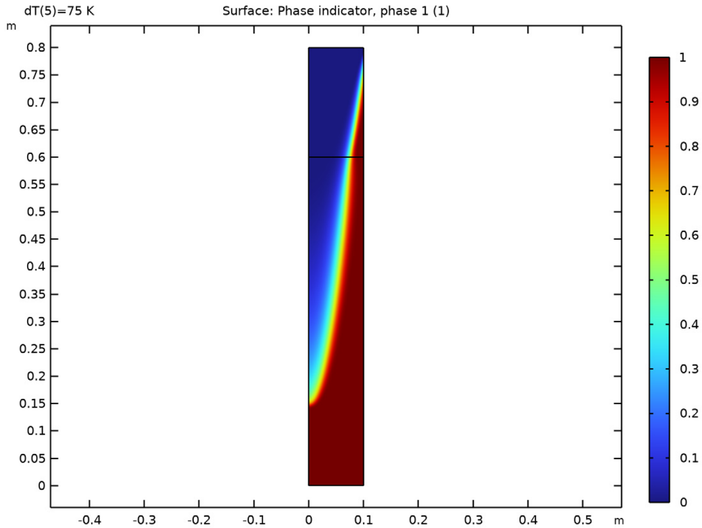


Figure 2: The fraction of solid phase for ΔT = 75 K shows a gradual transition between the liquid and solid phase.

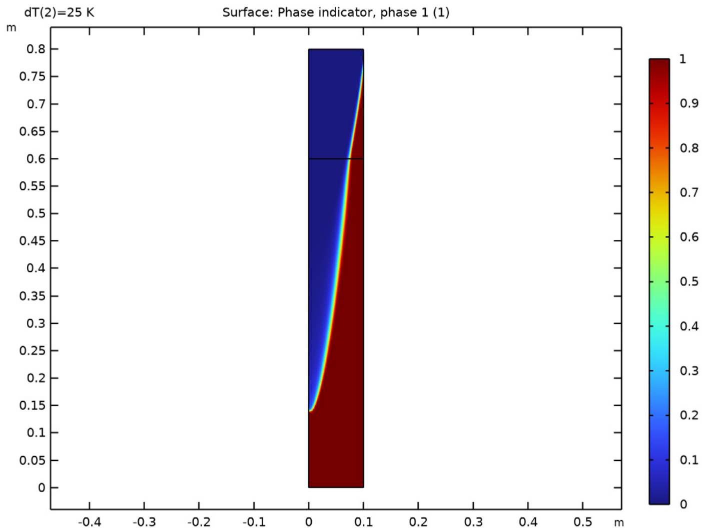


Figure 3: The fraction of solid phase for ΔT = 25 K shows a sharp transition between the liquid and solid phase.

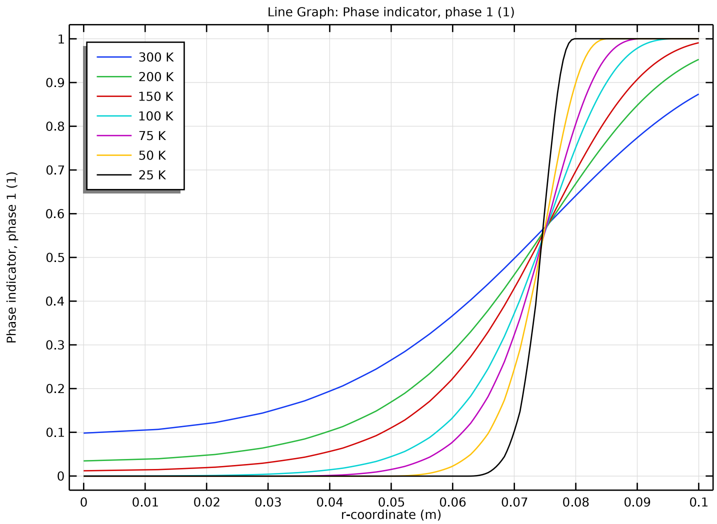


Figure 4: The fraction of solid phase through the radius for all values of ΔT*. For smaller values of* ΔT*, the transition is sharper.*

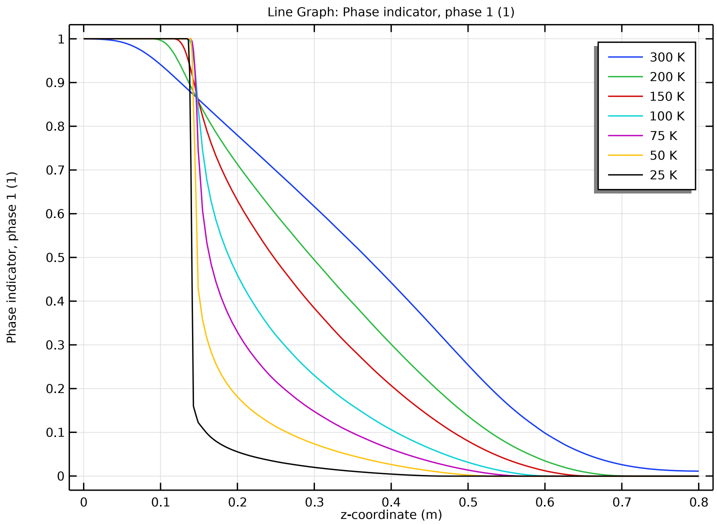


Figure 5: The fraction of solid phase along the centerline for all values of ΔT. For smaller values of ΔT, the transition is sharper.

Application Library path: `Heat_Transfer_Module/Thermal_Processing/cooling_solidification_metal`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

First, set up the parameters and variables needed for this simulation of a continuous casting process.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cooling_solidification_metal_parameters.txt`.

GEOMETRY 1

Create two rectangles representing the strand within the mold, and the spray cooled region outside of the mold.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.6`.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

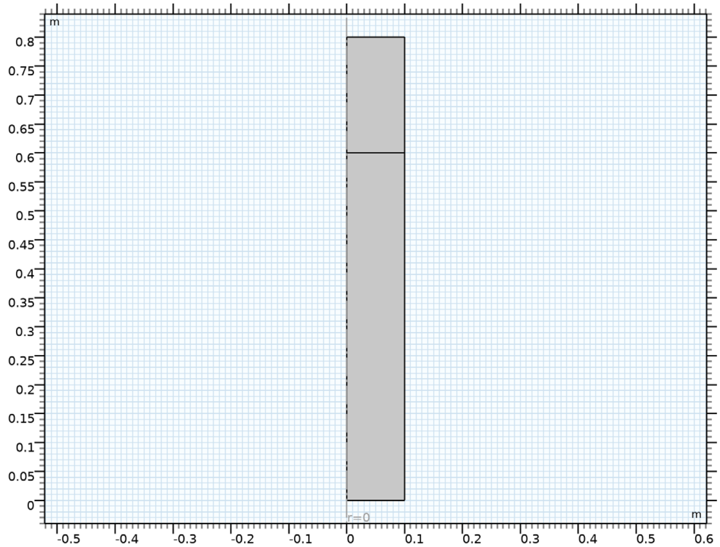
3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **z** text field, type `0.6`.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.



MATERIALS

Solid Metal Alloy

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Solid Metal Alloy` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	300	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	Cp_S	J/(kg·K)	Basic

Liquid Metal Alloy

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Liquid Metal Alloy` in the **Label** text field.

3 Click in the **Graphics** window and then press Ctrl+A to select both domains.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	150	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	Cp_L	J/(kg·K)	Basic

Set up the physics.

HEAT TRANSFER IN FLUIDS (HT)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T_in`.

Fluid 1

1 In the **Model Builder** window, click **Fluid 1**.

2 In the **Settings** window for **Fluid**, locate the **Heat Convection** section.

3 Specify the $\mathbf{u}$ vector as

0	r
-v_cast	z

Phase Change Material 1

1 In the **Physics** toolbar, click **Attributes** and choose **Phase Change Material**.

2 In the **Settings** window for **Phase Change Material**, locate the **Phase Change** section.

3 In the $T_{\mathrm{pc},1 \rightarrow 2}$ text field, type `T_m`.

4 In the $\Delta T_{1 \rightarrow 2}$ text field, type `dT`.

5 In the $L_{1 \rightarrow 2}$ text field, type `dH`.

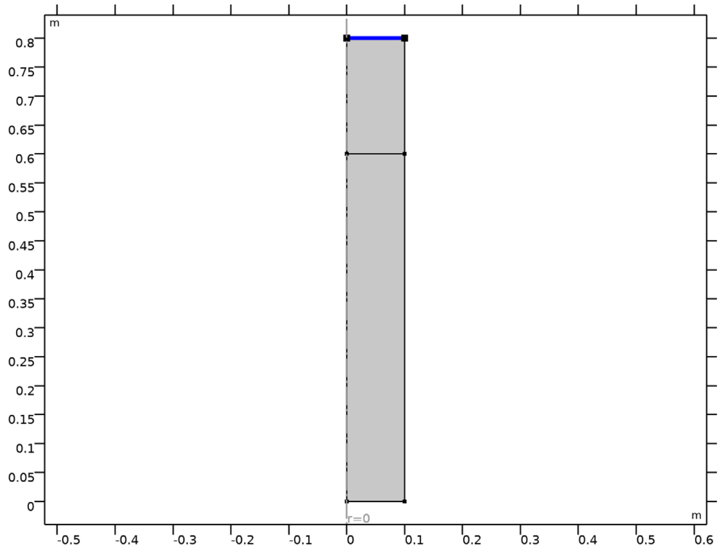
6 Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Solid Metal Alloy (mat1)**.

7 Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Liquid Metal Alloy (mat2)**.

Inflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 Select Boundary 5 only.



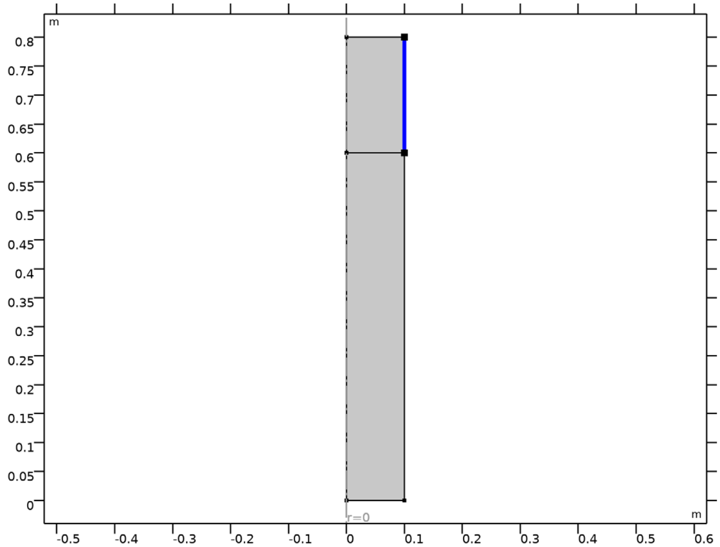
3 In the **Settings** window for **Inflow**, locate the **Upstream Properties** section.

4 In the T_{ustr} text field, type `T_in`.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 7 only.



3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

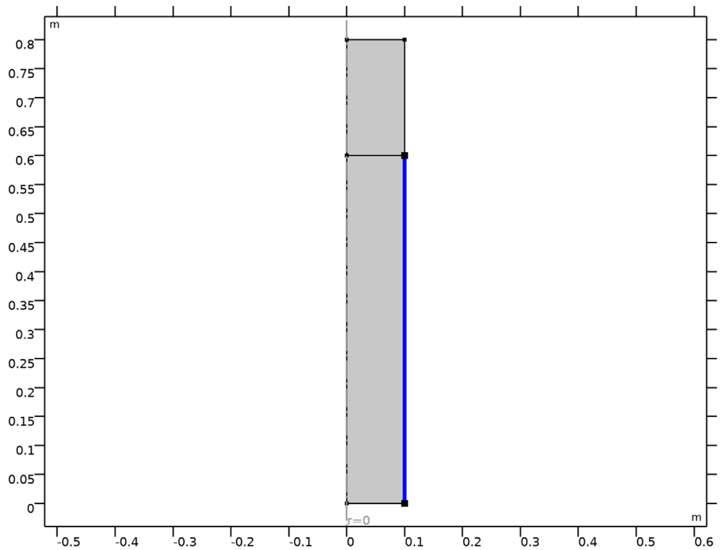
5 In the h text field, type `h_mold`.

6 In the T_{ext} text field, type `T0`.

Heat Flux 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 6 only.



3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `h_spray`.

6 In the T_{ext} text field, type `T0`.

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 Select Boundary 6 only.

3 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.

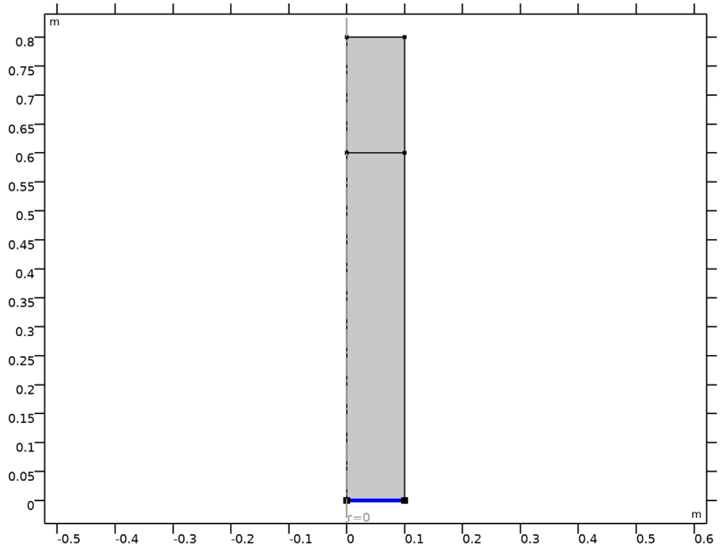
4 From the ε list, choose **User defined**. In the associated text field, type `eps_s`.

5 In the T_{amb} text field, type `T0`.

Outflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 2 only.



MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extremely fine**.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

Step 1: Stationary

Set up an auxiliary continuation sweep for the dT parameter.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click + **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT (Temperature transition zone half width)	300 200 150 100 75	K

6 Click to expand the **Adaptation and Error Estimates** section. From the **Adaptation and error estimates** list, choose **Adaptation and error estimates**.

7 Find the **Mesh adaptation** subsection. From the **Adaptation method** list, choose **Rebuild mesh**.

8 In the **Home** toolbar, click **Compute**.

RESULTS

Solid and Liquid Phases (Adaptive Mesh)

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Solid and Liquid Phases (Adaptive Mesh)` in the **Label** text field.

Surface 1

1 In the **Solid and Liquid Phases (Adaptive Mesh)** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Fluids>Phase change>ht.theta1 - Phase indicator, phase 1**.

3 In the **Solid and Liquid Phases (Adaptive Mesh)** toolbar, click **Plot**.

The reproduced figure describes the fraction of solid phase for ΔT = 75 K.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.

3 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 1, Stationary**.

6 From the **Solution** list, choose **Adaptive Mesh Refinement Solutions 1 (sol2)**.

7 From the **Use** list, choose **Level 2 Refined Solution 5 (sol5)**.

8 From the **Parameter value (dT (K))** list, choose **75 K**.

9 Click to expand the **Mesh Selection** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

10 Click **Add**.

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT (Temperature transition zone half width)	50 25	K

Solution 6 (sol6)

1 In the **Study** toolbar, click **Show Default Solver**.

Use a tighter relative tolerance to capture the phase change effect also for the small phase transition temperature interval.

2 In the **Model Builder** window, expand the **Solution 6 (sol6)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-5`.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Solid and Liquid Phases

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Solid and Liquid Phases` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 6 (sol6)**.

Surface 1

1 In the **Solid and Liquid Phases** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Heat Transfer in Fluids>Phase change>ht.theta1 - Phase indicator, phase 1**.

3 In the **Solid and Liquid Phases** toolbar, click **Plot**.

This shows the fraction of solid phase for $\Delta T = 25$ K.

Phase Indicator at Symmetry Axis

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Phase Indicator at Symmetry Axis` in the **Label** text field.

Line Graph 1

1 In the **Phase Indicator at Symmetry Axis** toolbar, click **Line Graph**.

2 Select Boundaries 1 and 3 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Heat Transfer in Fluids>Phase change>ht.theta1 - Phase indicator, phase 1**.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `z`.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 6 (sol6)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 In the **Phase Indicator at Symmetry Axis** toolbar, click **Plot**.

Compare the resulting plot with Figure 5 showing the fraction of solid phase through the centerline for all values of Δ T.

Phase Indicator through Radius

1 In the **Model Builder** window, right-click **Phase Indicator at Symmetry Axis** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Phase Indicator through Radius` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Phase Indicator through Radius** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 4 only.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `r`.

6 In the **Phase Indicator through Radius** toolbar, click **Plot**.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 4 only.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `r`.

Phase Indicator through Radius

1 In the **Model Builder** window, click **Phase Indicator through Radius**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 In the **Phase Indicator through Radius** toolbar, click **Plot**.

Compare the resulting plot with Figure 4 showing the fraction of solid phase through the radius for all values of Δ T.