Continuous Casting

Introduction

This example simulates the process of continuous casting of a metal rod from a molten state (Figure 1). To optimize the casting process in terms of casting rate and cooling, it is helpful to model the thermal and fluid dynamic aspects of the process. To get accurate results, you must model the melt flow field in combination with the heat transfer and phase change. The model includes the phase transition from melt to solid, both in terms of latent heat and the varying physical properties.

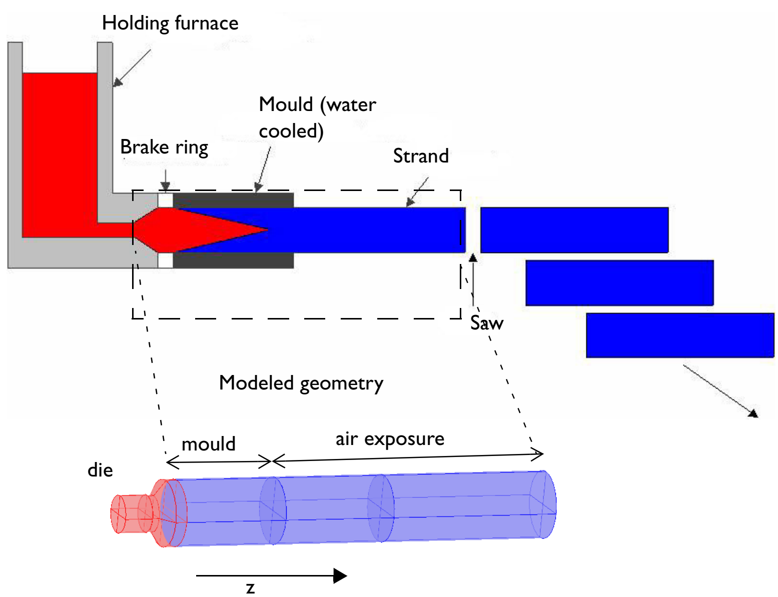


Figure 1: Continuous metal-casting process with a view of the modeled section.

This example simplifies the rod's 3D geometry in Figure 1 to an axisymmetric 2D model in the *rz*-plane. Figure 2 shows the dimensions of the 2D geometry.

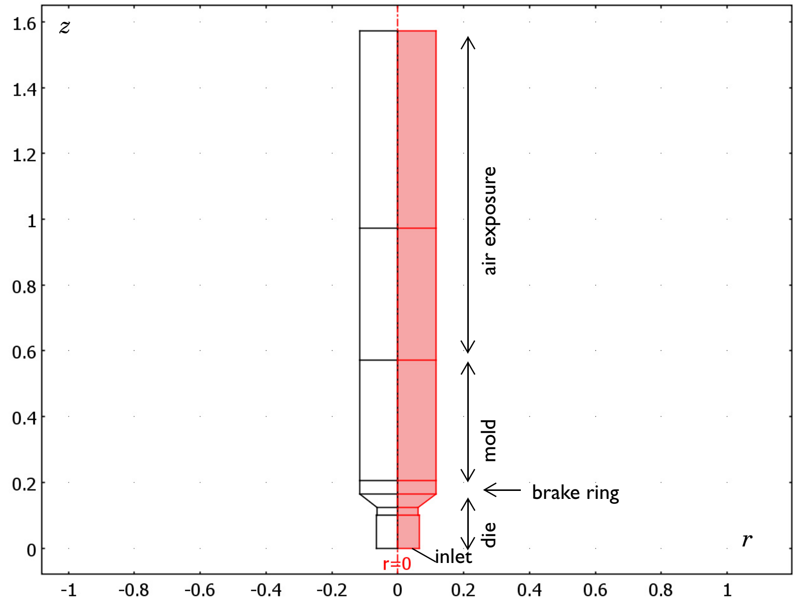


Figure 2: 2D axisymmetric model of the casting process.

As the melt cools down in the mold it solidifies. The phase transition releases latent heat, which the model includes. Furthermore, for metal alloys, the transition is often spread out over a temperature range. As the material solidifies, the material properties change considerably. Finally, the model also includes the "mushy" zone—a mixture of solid and melted material that occurs due to the rather broad transition temperature of the alloy and the solidification kinetics.

This example models the casting process as being stationary using the Heat Transfer in Fluids interface combined with the Laminar Flow interface.

Model Definition

The process operates at steady state, because it is a continuous process. The heat transport is described by the equation:

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

where k, C_p, and Q denote thermal conductivity, specific heat, and heating power per unit volume (heat source term), respectively.

As the melt 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. The difference in specific heat before and after transition can be approximated by

$$\Delta C_p = \frac{\Delta H}{T}$$

As opposed to pure metals, an alloy generally undergoes a broad temperature transition zone, over several kelvin, in which a mixture of both solid and molten material co-exist 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 half-width of the transition interval, ΔT, is set to 10 K in this case, and represents half the transition temperature span.

This example models the laminar flow by describing the fluid velocity, $\mathbf{u}$, and the pressure, p, according to the equations

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho\mathbf{u}\cdot\nabla\mathbf{u} = \nabla\cdot\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \left(\frac{2\mu}{3} - \kappa\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right] + \mathbf{F}$$

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0$$

where ρ is the density (in this case constant), μ is the viscosity, and κ is the dilatational viscosity (here assumed to be zero). Here, the role of the source term, $\mathbf{F}$, is to dampen the velocity at the phase-change interface so that it becomes that of the solidified phase after the transition. The source term follows from the equation (see Ref. 1)

$$\mathbf{F} = \frac{(1-\alpha)^2}{\alpha^3 + \varepsilon}A_{\mathrm{mush}}(\mathbf{u} - \mathbf{u}_{\mathrm{cast}})$$

where α can be seen as the volume fraction of the liquid phase; A_{mush} and ε represent arbitrary constants (A_{mush} should be large and ε small to produce a proper damping); and $\mathbf{u}_{\mathrm{cast}}$ is the velocity of the cast rod.

Table 1 reviews the material properties in this model.

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))	200	200
Dynamic viscosity	μ (Ns/m^2)	0.0434	-

Furthermore, the melting temperature, T_m, and enthalpy, ΔH, are set to 1356 K and 205 kJ/kg, respectively.

The model uses the parametric solver in combination with adaptive meshing to solve the problem efficiently. In particular, using an adaptive mesh makes it possible to resolve the steep gradients in the mushy zone at a comparatively low computational cost.

Results and Discussion

The plots in Figure 3 display the temperature and phase distributions, showing that the melt cools down and solidifies in the mold region. Interestingly, the transition zone stretches out towards the center of the rod because of poorer cooling in that area.

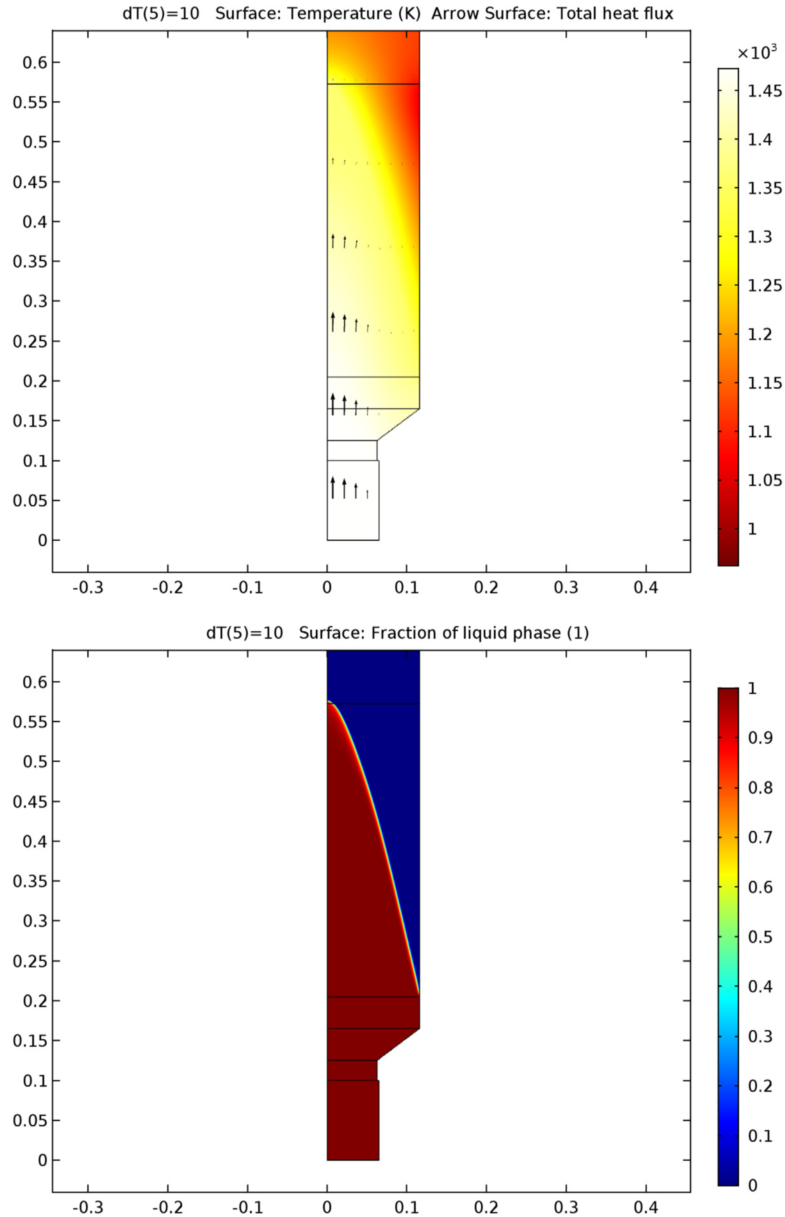


Figure 3: Temperature distribution (top) and fraction of liquid phase (bottom) in the lower part of the cast at a casting rate of 1.6 mm/s.

With the modeled casting rate, the rod is fully solidified before leaving the mold (the first section after the die). This means that the process engineers can increase the casting rate without running into problems, thus increasing the production rate.

The phase transition occurs in a very narrow zone although the model uses a transition half width, ΔT, of 10 K. In reality it would be even more distinct if a pure metal were being cast but somewhat broader if the cast material were an alloy with a wider ΔT.

It is interesting to study in detail the flow field in the melt as it exits the die.

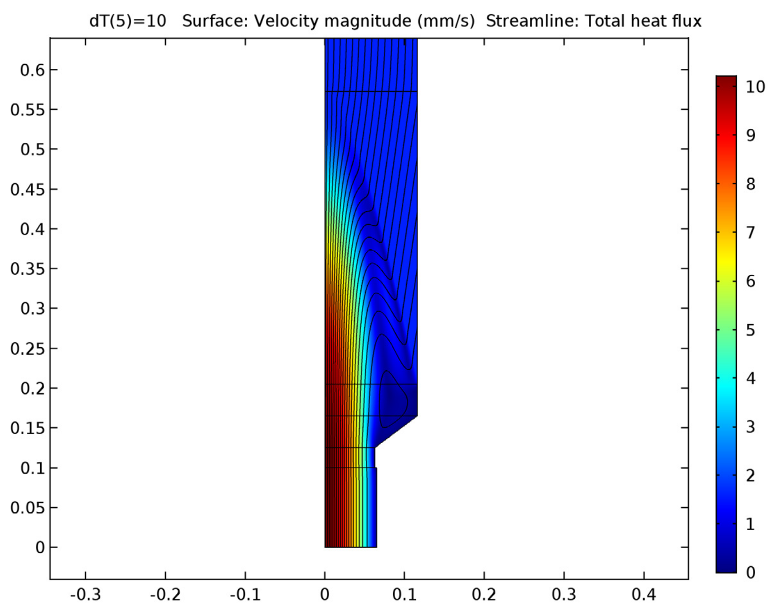
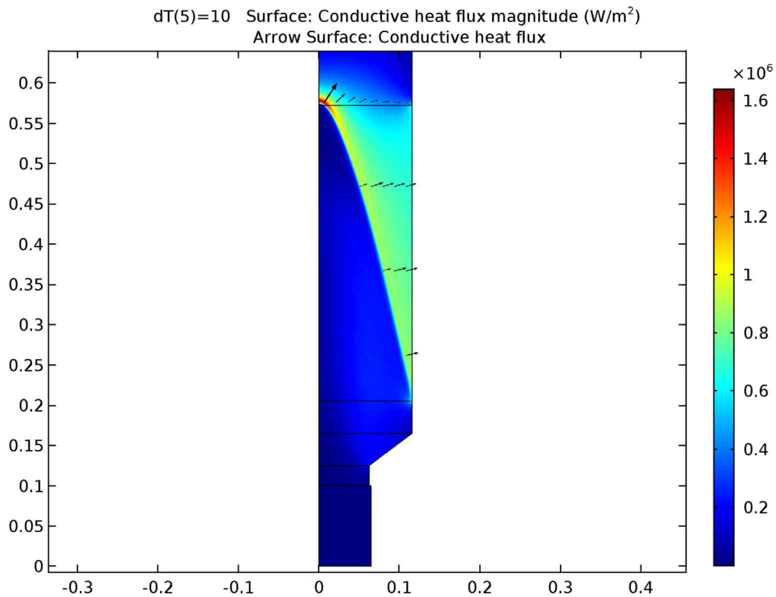


Figure 4: Velocity field with streamlines in the lower part of the process.

In Figure 4, notice the disturbance in the streamlines close to the die wall resulting in a vortex. This eddy flow could create problems with nonuniform surface quality in a real process. Process engineers can thus use the model to avoid these problems and find an optimal die shape.

To help determine how to optimize process cooling, Figure 5 plots the conductive heat flux. It shows that the conductive heat flux is very large in the mold zone. This is a consequence of the heat released during the phase transition, which is cooled by the water-cooling jacket of the mold. An interesting phenomenon of the process is the peak of conductive heat flux appearing in the center of the flow at the transition zone.



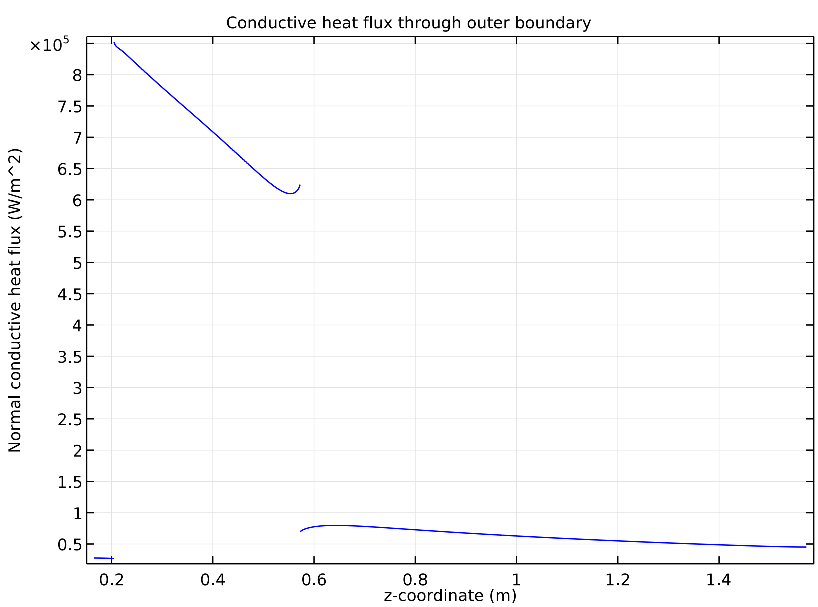


Figure 5: The cooling viewed as conductive heat flux in the domains (top), and through the outer boundary (the cooling zones) after the die (bottom).

Furthermore, by plotting the conductive heat flux at the outer boundary for the process as in the lower plot in Figure 5, you can see that a majority of the process cooling occurs in the mold. More interestingly, the heat flux varies along the mold wall length. This information can help in optimizing the cooling of the mold (that is, the cooling rate and choice of cooling method).

You solve the model using a built-in adaptive meshing technique. This is necessary because the transition zone—that is, the region where the phase change occurs—requires a fine discretization. Figure 6 depicts the final mesh of the model. Notice that the majority of the elements are concentrated to the transition zone.

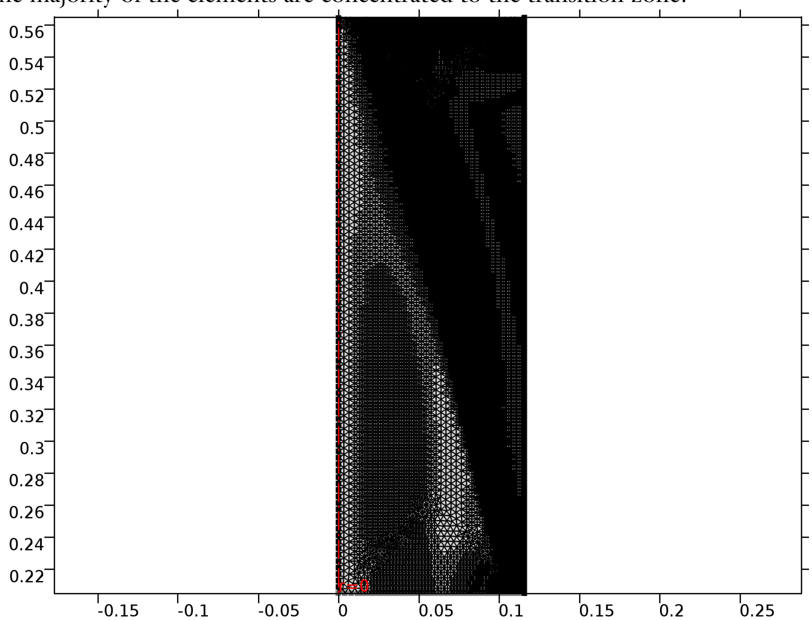


Figure 6: Close-up of the final computational mesh, resulting from the built-in adaptive technique.

The adaptive meshing technique allows for fast and accurate calculations even if the transition width is brought down to a low value, such as for pure metals.

Reference

1. V.R. Voller and C. Prakash, "A fixed grid numerical modeling methodology for convection—diffusion mushy region phase-change problems," *Int.J.Heat Mass Transfer*, vol. 30, pp. 1709–1719, 1987.

Application Library path: `Heat_Transfer_Module/Thermal_Processing/continuous_casting`

Modeling Instructions

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

NEW

1 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 In the **Select physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the application's Application Library folder and double-click the file `continuous_casting_parameters.txt`.

Note, in particular, the value of the parameter `dT`, which represents the parameter ΔT in the Model Definition section. It will apply when you solve with adaptive mesh refinement because that solution stage does not use the parametric. It is then crucial that the value of `dT` matches that of the final parameter step for the parametric solution that is used as the initial solution.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.
2 In the **Settings** window for Variables, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **All domains**.

Define the variables by loading the corresponding text file provided.

5 Locate the **Variables** section. Click **Load from File**.
6 Browse to the application's Application Library folder and double-click the file `continuous_casting_variables.txt`.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.
3 In the **Width** text field, type `0.065`.
4 In the **Height** text field, type `0.1`.

Rectangle 2 (r2)

1 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.
2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
3 In the **Settings** window for Rectangle, locate the **Size and Shape** section.
4 In the **Width** text field, type `0.0625`.
5 In the **Height** text field, type `0.025`.
6 Locate the **Position** section. In the **z** text field, type `0.1`.

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.
2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
3 In the **Settings** window for Rectangle, locate the **Size and Shape** section.
4 In the **Width** text field, type `0.11575`.
5 In the **Height** text field, type `0.04`.
6 Locate the **Position** section. In the **z** text field, type `0.165`.

Rectangle 4 (r4)

1 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

4 In the **Width** text field, type `0.11575`.

5 In the **Height** text field, type `0.3675`.

6 Locate the **Position** section. In the **z** text field, type `0.205`.

Rectangle 5 (r5)

1 Right-click **Rectangle 4 (r4)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

4 In the **Width** text field, type `0.11575`.

5 In the **Height** text field, type `0.4`.

6 Locate the **Position** section. In the **z** text field, type `0.5725`.

Rectangle 6 (r6)

1 Right-click **Rectangle 5 (r5)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

4 In the **Width** text field, type `0.11575`.

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

6 Locate the **Position** section. In the **z** text field, type `0.9725`.

7 Right-click **Rectangle 6 (r6)** and choose **Build Selected**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

Bézier Polygon 1 (b1)

1 Use the **Zoom Box** button in the Graphics toolbar to zoom in on the gap between two rectangles in the geometry.

2 Click **Geometry 1**. Then, using the **Draw Line** toolbar button, close the gap with a solid polygon. Note that you need to right-click at the end to close the polygon.

This completes the geometry modeling stage.

MATERIALS

Now, add the following two materials to the model. The Solid Metal Alloy material is used in the **Heat Transfer with Phase Change** feature for the solid phase while the Liquid Metal Alloy material is used for the liquid phase. The Liquid Metal Alloy also defines fluid properties used in the **Laminar Flow** interface.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **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	Name	Value	Unit	Property group
Thermal conductivity	k	200	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	Cp_s	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic
Dynamic viscosity	mu	0.0434	Pa·s	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **All domains**.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	200	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	Cp_l	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic
Dynamic viscosity	mu	0.0434	Pa·s	Basic

HEAT TRANSFER IN FLUIDS (HT)

Heat Transfer with Phase Change 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Transfer with Phase Change**.
2 In the **Settings** window for Heat Transfer with Phase Change, locate the **Domain Selection** section.
3 From the **Selection** list, choose **All domains**.
4 Locate the **Model Inputs** section. From the p_A list, choose **Absolute pressure (spf)**.
5 From the $\mathbf{u}$ list, choose **Velocity field (spf)**.
6 Locate the **Phase Change** section. In the $T_{\text{pc},1 \to 2}$ text field, type `T_m`.
7 In the $\Delta T_{1 \to 2}$ text field, type `2*dT`.

   The parameter `dT` is multiplied by 2 because it is only the half width of the phase change interval.
8 In the $L_{1 \to 2}$ text field, type `dH`.
9 Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Solid Metal Alloy (mat1)**.
10 Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Liquid Metal Alloy (mat2)**.

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`.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundary 2 only.
3 In the **Settings** window for Temperature, locate the **Temperature** section.
4 In the T_0 text field, type `T_in`.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundary 20 only.
3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.
4 Click the **Convective heat flux** button.

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

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

Heat Flux 2

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

2 Select Boundary 21 only.

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

4 Click the **Convective heat flux** button.

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

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

Heat Flux 3

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

2 Select Boundaries 22 and 23 only.

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

4 Click the **Convective heat flux** button.

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

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

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 Select Boundaries 22 and 23 only.

3 In the **Settings** window for Diffuse Surface, locate the **Surface Emissivity** section.

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

5 Locate the **Ambient** section. In the T_{amb} text field, type `T0`.

Outflow 1

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

2 Select Boundary 15 only.

LAMINAR FLOW (SPF)

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Fluid Properties 1**.

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

3 From the ρ list, choose **User defined**. In the associated text field, type `ht.rho`.

The Heat Transfer interface provides `ht.rho` which is the mixture density between solid and liquid phases. Using this variable as the fluid density in the flow interface ensures that the continuity equation is verified. This should be done in all models coupling the continuity equation and **Heat Transfer with Phase Change**. Note that in this model both phases have the same density so this change would not affect the results.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Initial Values 1**.

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

3 Specify the **u** vector as

0	r
v_cast	z

4 In the **Model Builder** window, click **Laminar Flow (spf)**.

Volume Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Volume Force**.

2 In the **Settings** window for Volume Force, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Volume Force** section. Specify the **F** vector as

Fr	r
Fz	z

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 2 only.

3 In the **Settings** window for Inlet, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundaries 15 and 21–23 only.

3 In the **Settings** window for Outlet, locate the **Boundary Condition** section.

4 From the list, choose **Velocity**.

5 Locate the **Velocity** section. Click the **Velocity field** button.

6 Specify the $\mathbf{u}_0$ vector as

0	r
v_cast	z

7 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

8 In the **Model Builder** window, click **Laminar Flow (spf)**.

9 In the **Settings** window for Laminar Flow, click to expand the **Advanced settings** section.

10 Locate the **Advanced Settings** section. Find the **Pseudo time stepping** subsection. Select the **Use pseudo time stepping for stationary equation form** check box.

To improve convergence, the pseudo time stepping option is enabled.

MESH 1

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

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

3 From the **Element size** list, choose **Finer**.

Boundary Layers 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Boundary Layers 1** and choose **Delete**. Click **Yes** to confirm.

This is necessary to be able to use the adaptive mesh functionality.

Size 1

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

2 Select Boundaries 16–21 only.

3 In the **Settings** window for Size, locate the **Element Size** section.

4 From the **Predefined** list, choose **Fine**.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

Calculate the solution using a three-step process. First, solve the problem using `dT` as a continuation parameter with the parametric solver on the default mesh, gradually decreasing the value of dT. Then, use the adaptive solver to adapt the mesh. Finally, use the parametric solver again to decrease dT further down to a value of 10 K.

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.

Disable default plots from this study because they will be added from the last study.

Step 1: Stationary

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 Locate the **Study Extensions** section. 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	300 100 50 30	

Stationary 2

On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

Step 2: Stationary 2

1 In the **Settings** window for Stationary, click to expand the **Values of dependent variables** section.

2 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

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

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

5 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Adaptive mesh refinement** check box.

6 On the **Study** toolbar, click **Compute**.

Before proceeding with the final solution stage, inspect the adapted mesh. You find it under the automatically created **Meshes** branch in the model tree.

MESH 2

1 In the **Model Builder** window, expand the **Meshes** node, then click **Mesh 2**.

2 Click the **Zoom Box** button on the Graphics toolbar and then use the mouse to zoom in on the transition zone where the mesh is the densest.

The mesh should look like that in Figure 6.

Add a second study for the second parametric study step.

ADD STUDY

1 On the **Study** 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 **Preset Studies>Stationary**.

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

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

STUDY 2

In order to get faster convergence, you use the previous solution as the initial value for this study.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes** node, then click **Study 2>Step 1: Stationary**.

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

3 Locate the **Values of Dependent Variables** section. 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 2**.

6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT	25 20 16 13 10	

Notice that **Mesh 2**, the adapted mesh, is the default selection in the Mesh list. Keep this setting.

In the parametric sweep of dT from 25 K to 10 K, the model becomes highly nonlinear. You have to make sure that each solution converges enough since it is used as initial condition for the next parametric sweep step. To do so, specify a lower error tolerance than the default setting.

Solution 4 (sol4)

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.
3 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Stationary Solver 1**.
4 In the **Settings** window for Stationary Solver, locate the **General** section.
5 In the **Relative tolerance** text field, type `1e-6`.
6 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 4 (sol4)>Stationary Solver 1** node, then click **Fully Coupled 1**.
7 In the **Settings** window for Fully Coupled, click to expand the **Method and termination** section.
8 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Automatic (Newton)**.
9 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature, 3D (ht)

The first default plot shows the temperature in 3D obtained by a revolution of the 2D axisymmetric data set.

Isothermal Contours (ht)

The second default plot shows the isothermal contours in the 2D slice.

Velocity (spf)

To reproduce the upper plot in Figure 4, plot the velocity field as a combined surface and streamline plot.

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Surface 1**.
2 In the **Settings** window for Surface, locate the **Expression** section.
3 From the **Unit** list, choose **mm/s**.
4 On the **Velocity (spf)** toolbar, click **Plot**.
5 Again, use the mouse to zoom in on the transition zone.
6 Right-click **Results>Velocity (spf)** and choose **Streamline**.
7 In the **Settings** window for Streamline, locate the **Streamline Positioning** section.

8 From the **Positioning** list, choose **Magnitude controlled**.

9 On the **Velocity (spf)** toolbar, click **Plot**.

Compare the plot in the Graphics window with that in Figure 4.

Pressure (spf)

This default plot shows the pressure profile in the 2D slice.

Velocity (spf) 1

This default plot shows the velocity magnitude in 3D obtained by a revolution of the 2D axisymmetric data set.

Proceed to reproduce the lower plot in Figure 3, showing the fraction of liquid phase.

2D Plot Group 6

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

2 In the **Settings** window for 2D Plot Group, type `Fraction of Liquid Phase` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 4 (sol4)**.

Fraction of Liquid Phase

1 Right-click **Fraction of Liquid Phase** and choose **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>Definitions>Variables>alpha - Fraction of liquid phase**.

3 On the **Fraction of Liquid Phase** toolbar, click **Plot**.

Notice, in particular, the narrow transition zone between the two phases.

To reproduce the upper plot in Figure 3, which visualizes the temperature and velocity fields, proceed as follows.

2D Plot Group 7

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

2 In the **Settings** window for 2D Plot Group, type `Temperature` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 4 (sol4)**.

Temperature

1 Right-click **Temperature** and choose **Surface**.

2 In the **Settings** window for Surface, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **ThermalLight**.

4 On the **Temperature** toolbar, click **Plot**.

5 Right-click **Temperature** and choose **Arrow Surface**.

6 In the **Settings** window for Arrow Surface, locate the **Arrow Positioning** section.

7 Find the **r grid points** subsection. In the **Points** text field, type `8`.

8 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

9 On the **Temperature** toolbar, click **Plot**.

Proceed to reproduce the heat flux plots shown in Figure 3.

2D Plot Group 8

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

2 In the **Settings** window for 2D Plot Group, type `Conductive Heat Flux` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 4 (sol4)**.

Conductive Heat Flux

1 Right-click **Conductive Heat Flux** and choose **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>Heat Transfer in Fluids>Domain fluxes>ht.dfluxMag - Conductive heat flux magnitude**.

3 On the **Conductive Heat Flux** toolbar, click **Plot**.

4 Right-click **Conductive Heat Flux** and choose **Arrow Surface**.

5 In the **Settings** window for Arrow Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer in Fluids>Domain fluxes>ht.dfluxr,ht.dfluxz - Conductive heat flux**.

6 Locate the **Arrow Positioning** section. Find the **r grid points** subsection. In the **Points** text field, type `8`.

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

8 On the **Conductive Heat Flux** toolbar, click **Plot**.

Compare the result with the upper plot in Figure 5.

The following steps reproduce the lower plot in the same figure, showing the conductive heat flux through the outer boundary after the die.

1D Plot Group 9

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

2 In the **Settings** window for 1D Plot Group, type `Conductive Heat Flux through Outer Boundary` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 4 (sol4)**.

4 From the **Parameter selection (dT)** list, choose **Last**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Conductive heat flux through outer boundary`.

7 Locate the **Plot Settings** section. Select the **x-axis label** check box.

8 In the associated text field, type `z-coordinate (m)`.

9 Select the **y-axis label** check box.

10 In the associated text field, type `Normal conductive heat flux (W/m^2)`.

Line Graph 1

On the **Conductive Heat Flux through Outer Boundary** toolbar, click **Line Graph**.

Conductive Heat Flux through Outer Boundary

1 Select Boundaries 20–23 only.

2 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>Heat Transfer in Fluids>Boundary fluxes>ht.ndflux - Normal conductive heat flux**.

3 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Geometry>Coordinate>z - z-coordinate**.

4 Click to expand the **Legends** section. Click to collapse the **Legends** section. Click to expand the **Quality** section. From the **Recover** list, choose **Within domains**.

5 Click to collapse the **Quality** section. On the **Conductive Heat Flux through Outer Boundary** toolbar, click **Plot**.

Compare the result with the lower plot of Figure 5.

Data Sets

Finally, verify that the final mesh is sufficiently fine to resolve the latent heat's temperature-dependence.

Cut Line 2D 1

On the **Results** toolbar, click **Cut Line 2D**.

Data Sets

1 In the **Settings** window for Cut Line 2D, locate the **Line Data** section.

2 In row **Point 1**, set **r** to `0.045`.

3 In row **Point 1**, set **z** to `0.42`.

4 In row **Point 2**, set **r** to `0.085`.

5 In row **Point 2**, set **z** to `0.43`.

These values are chosen such that the two points are on opposite sides of and approximately perpendicular to the transition zone.

Alternatively, you can select the two end points and create the Cut Line 2D data set by first clicking the **Fraction of Liquid Phase** node and then clicking in the Graphics window after first selecting, in turn, **First Point for Cut Line** and **Second Point for Cut Line** on the main toolbar.

6 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 4 (sol4)**.

1D Plot Group 10

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Temperature Dependence, Latent Heat` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

Line Graph 1

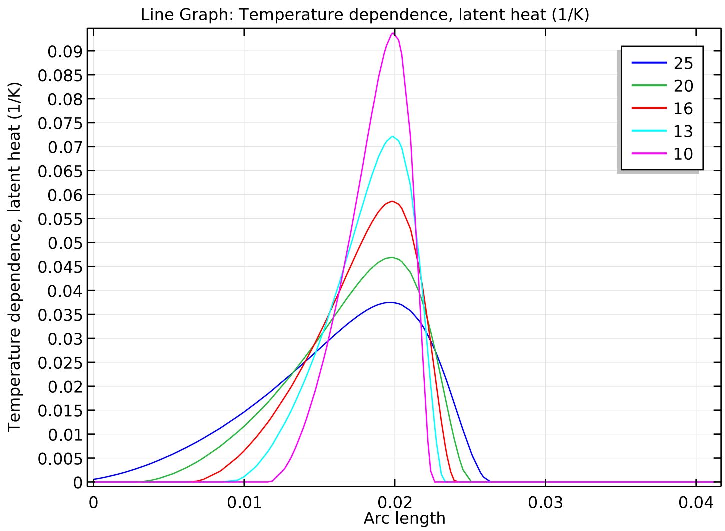
On the **Temperature Dependence, Latent Heat** toolbar, click **Line Graph**.

Temperature Dependence, Latent Heat

1 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>Definitions>Variables>D - Temperature dependence, latent heat**.

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

3 On the **Temperature Dependence, Latent Heat** toolbar, click **Plot**.



As you can see, the curves for the lower ΔT values, in particular $\Delta T = 10$ K, are not entirely smooth. Thus, if you were to reduce ΔT further to model the casting of some pure metal, you would need to increase the mesh resolution.