Created in COMSOL Multiphysics 6.0

Continuous Casting — Arbitrary Lagrangian-Eulerian Method

Introduction

This example simulates the process of continuous casting of a metal rod from a molten state (Figure 1) using the **Phase Change Interface** boundary condition. Continuous Casting — Apparent Heat Capacity Method is a variant of this model using the **Phase Change Material** domain condition.

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.

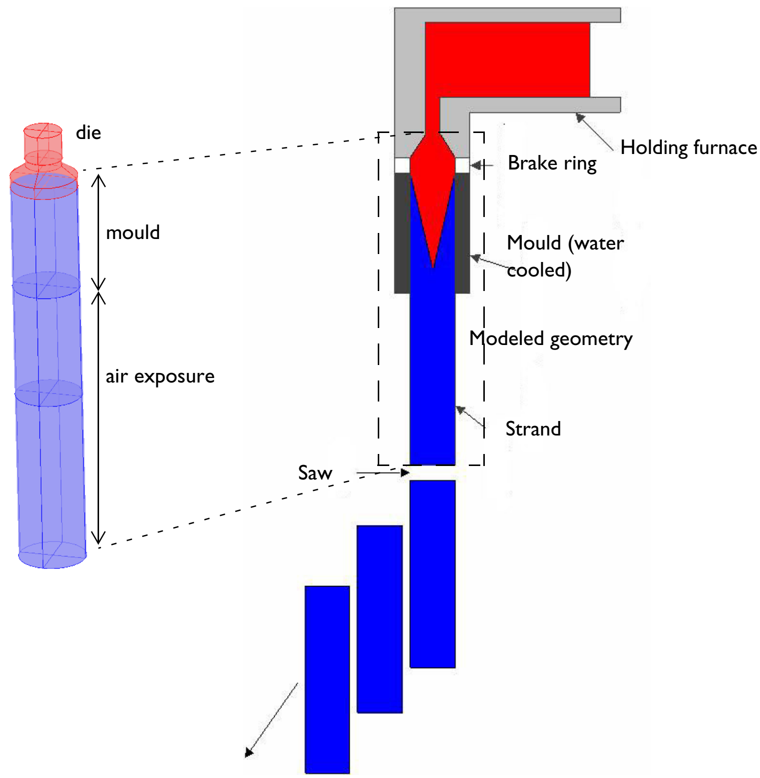


Figure 1: Continuous metal-casting process with a view of the modeled section.

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.

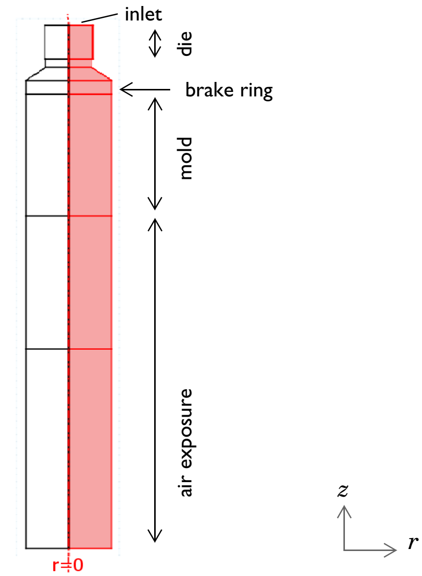


Figure 2: 2D axisymmetric model of the casting process.

As the melt cools down in the mold it solidifies. The phase transition releases latent heat, which the model includes. For metal alloys, the transition is often spread out over a temperature range. However, using the ALE approach to model the phase transition, a sharp interface is assumed between the two phases, and the latent heat of phase change is released at the corresponding boundary.

This example models the casting process with a transient study until it reaches a stationary state. The Heat Transfer in Fluids interface combined with the Laminar Flow interface are used.

Model Definition

The transient heat transport is described by the equation:

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

where k, ρ, C_p, and Q denote thermal conductivity, density, 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.

In this example, the **Phase Change Interface** boundary condition is used to model the phase change interface. This feature uses the Stefan Condition, which derives the normal interface velocity from the incoming heat fluxes, the melting latent heat and the solid density. To allow this interface to move in the geometry according to the calculated normal velocity, this feature is used along with a **Deformed Geometry** interface.

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

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

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

where ρ is the density (in this case constant), μ is the viscosity, and κ is the dilatational viscosity (here assumed to be zero).

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 (N·s/m^2)	0.0434	-

Furthermore, the melting temperature, T_{m}, and enthalpy of phase change, ΔH, are set to 1356 K and 205 kJ/kg, respectively.

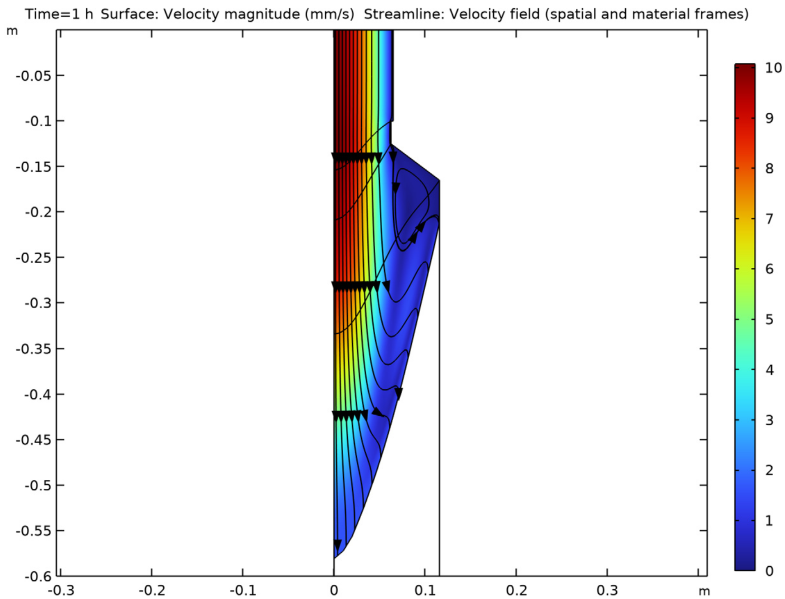


Figure 3: Velocity field with streamlines near the inlet part of the process.

In Figure 3, velocity streamlines are plot along with the phase change interface that delimits the fluid outlet. This interface stretches out toward the center of the rod because of poorer cooling in that area. 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.

To help determine how to optimize process cooling, Figure 4 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.

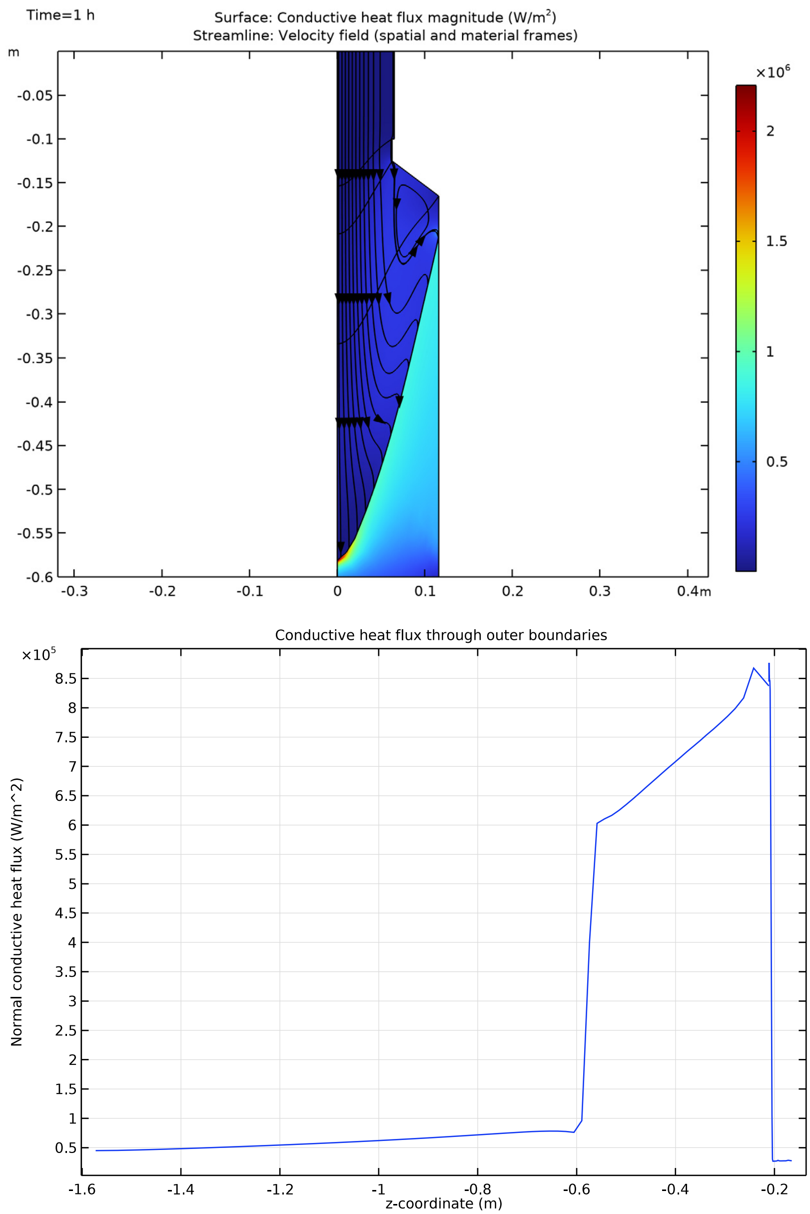


Figure 4: The cooling viewed as conductive heat flux in the domains (top), and through the outer boundary (the cooling zones) after the die (bottom).

Furthermore, by plotting the conductive heat flux at the outer boundary for the process as in the lower plot in Figure 4, 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).

This method allows a coarser mesh compared to the Continuous Casting — Apparent Heat Capacity Method model and by consequence a faster calculation. It provides also transient results hence the ability to compute the response of the system with time varying input (typically the casting velocity).

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_ale`

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 **Fluid Flow>Nonisothermal Flow>Laminar Flow**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

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 `continuous_casting_ale_parameters.txt`.

DEFINITIONS

Piecewise 1 (pw1)

1. In the **Home** toolbar, click f(x) **Functions** and choose **Local>Piecewise**.
2. In the **Settings** window for **Piecewise**, locate the **Definition** section.
3. From the **Smoothing** list, choose **Continuous function**.
4. In the **Size of transition zone** text field, type `0.01`.
5. Find the **Intervals** subsection. Click **Load from File**.
6. Browse to the model's Application Libraries folder and double-click the file `continuous_casting_ale_pw1.txt`.
7. Locate the **Units** section. In the **Arguments** text field, type `m`.
8. In the **Function** text field, type `W/m^2/K`.

Piecewise 2 (pw2)

1. In the **Home** toolbar, click f(x) **Functions** and choose **Local>Piecewise**.
2. In the **Settings** window for **Piecewise**, locate the **Definition** section.
3. From the **Smoothing** list, choose **Continuous function**.
4. In the **Size of transition zone** text field, type `0.01`.
5. Find the **Intervals** subsection. Click **Load from File**.
6. Browse to the model's Application Libraries folder and double-click the file `continuous_casting_ale_pw2.txt`.
7. Locate the **Units** section. In the **Arguments** text field, type `m`.
8. In the **Function** text field, type `1`.

Since the boundary edges will be translated due to the deformed geometry sliding conditions, add variables to impose spacially fixed boundary condition coefficients.

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
h_rod	pw1(Z)	W/(m²·K)	Heat transfer coefficient along the rod
eps_rod	pw2(Z)		Surface emissivity along the rod

GEOMETRY 1

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 `0.065`.

4 In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **z** text field, type `-0.1`.

6 In the **Geometry** toolbar, click **Build All**.

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 `0.0625`.

4 In the **Height** text field, type `0.025`.

5 Locate the **Position** section. In the **z** text field, type `-0.125`.

6 In the **Geometry** toolbar, click **Build All**.

Rectangle 3 (r3)

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 `0.11575`.

4 In the **Height** text field, type `1.4075`.

5 Locate the **Position** section. In the **z** text field, type `-1.5725`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.04

7 Clear the **Layers on bottom** check box.

8 Select the **Layers on top** check box.

9 In the **Geometry** toolbar, click **Build All**.

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

Polygon 1 (pol1)

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

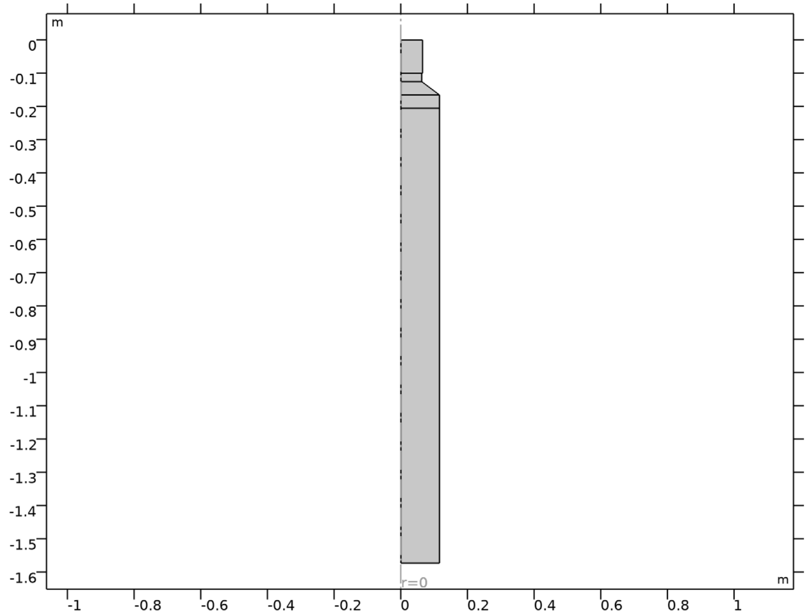
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (m)	z (m)
0	-0.125
0	-0.165
0.11575	-0.165
0.0625	-0.125
0	-0.125

4 In the **Geometry** toolbar, click **Build All**.

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



This completes the geometry modeling stage.

MATERIALS

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

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
Dynamic viscosity	mu	0.0434	Pa·s	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	200	W/(m·K)	Basic

Property	Variable	Value	Unit	Property group
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 Select Domains 2–5 only.

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

Property	Variable	Value	Unit	Property group
Dynamic viscosity	mu	0.0434	Pa·s	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	200	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	Cp_l	J/(kg·K)	Basic

COMPONENT 1 (COMP1)

In the **Definitions** toolbar, click **Deformed Geometry** and choose **Domains>Deforming Domain**.

DEFORMED GEOMETRY

Deforming Domain 1

1 In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.

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

Symmetry/Roller 1

1 In the **Definitions** toolbar, click **Deformed Geometry** and choose **Boundaries>Symmetry/Roller**.

2 Select Boundaries 1, 3, 5, 7, and 9 only.

Prescribed Normal Mesh Displacement 1

1 In the **Definitions** toolbar, click **Deformed Geometry** and choose **Boundaries>Prescribed Normal Mesh Displacement**.

2 Select Boundaries 16 and 17 only.

Fixed Boundary 1

1 In the **Definitions** toolbar, click **Deformed Geometry** and choose **Boundaries>Fixed Boundary**.

2 Select Boundaries 2 and 11–15 only.

LAMINAR FLOW (SPF)

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

2 Select Domains 2–5 only.

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

Inlet 1

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

2 Select Boundary 11 only.

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

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

Outlet 1

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

2 Select Boundary 4 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

DEFINITIONS

Ambient Properties 1 (ampr1)

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

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

3 In the T_{amb} text field, type `300[K]`.

This defines the ambient temperature for heat transfer between the outer surfaces and the surroundings.

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`.

Solid 1

1 In the **Physics** toolbar, click **Domains** and choose **Solid**.

2 Select Domain 1 only.

Translational Motion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Translational Motion**.

2 In the **Settings** window for **Translational Motion**, locate the **Translational Motion** section.

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

0	r
v_cast	z

Inflow 1

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

2 Select Boundary 11 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 Boundaries 16 and 17 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_rod`.

6 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Surface-to-Ambient Radiation 1

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

2 Select Boundaries 16 and 17 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_rod`.

5 From the T_{amb} list, choose **Ambient temperature (ampr1)**.

Phase Change Interface 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Phase Change Interface**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Phase Change Interface**, locate the **Phase Change Interface** section.

4 In the T_{pc} text field, type `T_m`.

5 In the $L_{\mathrm{s} \to \mathrm{f}}$ text field, type `dH`.

6 From the **Solid side** list, choose **Downside**.

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 **Finer**.

4 Click **Build All**.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check boxes for **Laminar Flow (spf)** and **Deformed geometry (Component 1)**.

4 In the table, clear the **Solve for** check box for **Nonisothermal Flow 1 (nitf1)**.

Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent> Time Dependent**.
2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3 From the **Time unit** list, choose **h**.
4 Click to expand the **Values of Dependent Variables** section.

A fully coupled solver is more robust and faster for this model. Tweak the solver sequence accordingly with the instructions below.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node.
4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.
5 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
6 In the **Damping factor** text field, type `0.9`.
7 From the **Jacobian update** list, choose **Once per time step**.
8 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.
9 In the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

To reproduce the plot in Figure 3, plot the velocity field as a combined surface and streamline plot.

Surface

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 From the **Unit** list, choose **mm/s**.

Velocity (spf)

In the **Model Builder** window, click **Velocity (spf)**.

Streamline 1

1 In the **Velocity (spf)** toolbar, click **Streamline**.
2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3 From the **Positioning** list, choose **Magnitude controlled**.
4 In the **Density** text field, type `12`.
5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.
6 In the **Velocity (spf)** toolbar, click **Plot**.

Pressure (spf)

The second default plot shows the pressure profile in the 2D slice.

Velocity, 3D (spf)

The third default plot shows the velocity magnitude in 3D obtained by revolution of the 2D axisymmetric dataset.

Temperature, 3D (ht)

This default plot shows the temperature in 3D obtained by revolution of the 2D axisymmetric dataset.

Temperature, Velocity Fields

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Temperature, Velocity Fields` in the **Label** text field.

Surface 1

1 In the **Temperature, Velocity Fields** 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>Temperature>T - Temperature - K**.
3 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.
4 In the **Temperature, Velocity Fields** toolbar, click **Plot**.

Temperature, Velocity Fields

In the **Model Builder** window, click **Temperature, Velocity Fields**.

Streamline 1

1 In the **Temperature, Velocity Fields** toolbar, click **Streamline**.
2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

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

4 In the **Density** text field, type `12`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

6 In the **Temperature, Velocity Fields** toolbar, click **Plot**.

Proceed to reproduce the heat flux plots shown in Figure 4.

Conductive Heat Flux

1 In 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.

Surface 1

1 In the **Conductive Heat Flux** 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>Domain fluxes>ht.dfluxMag - Conductive heat flux magnitude - W/m²**.

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

Conductive Heat Flux

In the **Model Builder** window, click **Conductive Heat Flux**.

Streamline 1

1 In the **Conductive Heat Flux** toolbar, click **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

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

4 In the **Density** text field, type `12`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

6 In the **Conductive Heat Flux** toolbar, click **Plot**.

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

Conductive Heat Flux through Outer Boundaries

1 In 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 Boundaries` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **Last**.

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

5 In the **Title** text area, type `Conductive heat flux through outer boundaries`.

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

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

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

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

Line Graph 1

1 In the **Conductive Heat Flux through Outer Boundaries** toolbar, click **Line Graph**.

2 Select Boundaries 16 and 17 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>Boundary fluxes>ht.ndflux - Normal conductive heat flux - W/m²**.

4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Geometry> Coordinate (spatial and material frames)>z - z-coordinate**.

5 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

6 Click to collapse the **Quality** section. In the **Conductive Heat Flux through Outer Boundaries** toolbar, click **Plot**.

Compare the result with the lower plot of Figure 4.