Cooling of an Injection Mold

Introduction

Cooling is an important process in the production of injection molded plastics. First of all, the cooling time may well represent more than half of the production cycle time. Second, a homogeneous cooling process is desired to avoid defects in the manufactured parts. If plastic materials in the injection molding die are cooled down uniformly and slowly, residual stresses can be avoided, and thereby the risk of warps and cracks in the end product can be minimized.

As a consequence, the positioning and properties of the cooling channels become important aspects when designing the mold.

The simulation of heat transfer in molds of relatively complex geometries requires a 3D representation. Simulation of 3D flow and heat transfer inside the cooling channels are computationally expensive. An efficient short-cut alternative is to model the flow and heat transfer in the cooling channels with 1D pipe flow equations, and still model the surrounding mold and product in 3D.

This example shows how you can use the Non-Isothermal Pipe Flow interface together with the Heat Transfer in Solids interface to model a mold cooling process. The equations describing the cooling channels are fully coupled to the heat transfer equations of the mold and the polyurethane part.

Figure 1: The steering wheel of a car, made from polyurethane.

©2013 COMSOL

Model Definition

MODEL GEOMETRY AND PROCESS CONDITIONS

The polyurethane material used for a steering wheel is produced by several different molds. The part considered in this model is the top half of the wheel grip, shown in gray in Figure 2.

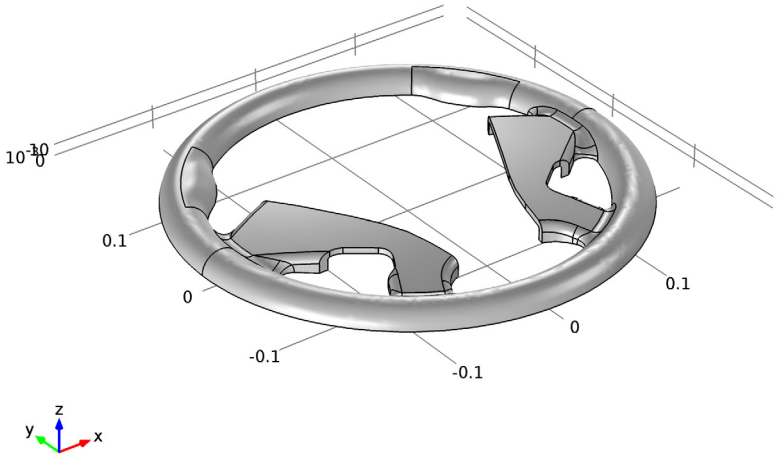


Figure 2: Polyurethane parts for a steering wheel. The top half of the grip is modeled in this example.

©2013 COMSOL

The mold consists of a 50-by-50-by-15 cm steel block. Two cooling channels, 1 cm in diameter, are machined into the block as illustrated in Figure 3.

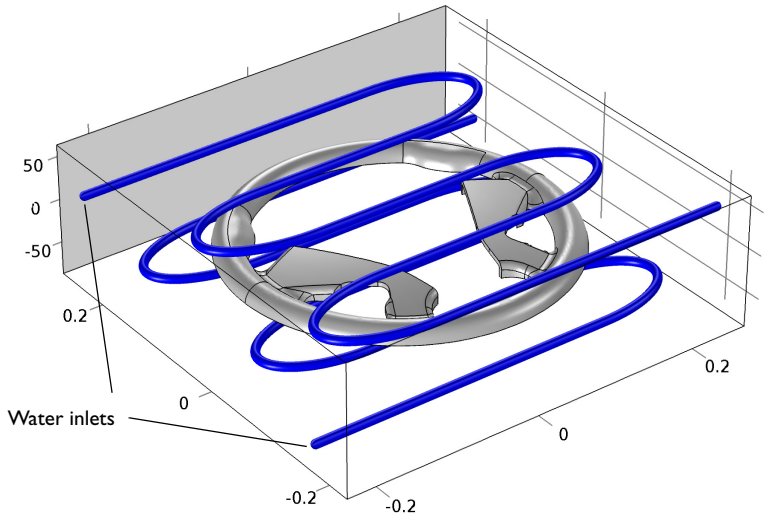


Figure 3: Mold block and cooling channels.

The after injection of the polyurethane, the average temperature of the mold a the plastic material is 473 K. Water at room temperature is used as cooling fluid and flows through the channels at a rate of 10 liters/min. The model simulates a 10 min cooling process.

For numerical stability reasons, the model is set up with an initial water temperature of 473 K, which is ramped down to 288 K during the first few seconds.

PIPE FLOW EQUATIONS

The momentum and mass conservation equations below describe the flow in the cooling channels:

$$\rho\frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_{\mathrm{D}}\frac{\rho}{2d_{\mathrm{h}}}\mathbf{u}|\mathbf{u}| \tag{1}$$

$$\frac{\partial A\rho}{\partial t} + \nabla\cdot(A\rho\mathbf{u}) = 0 \tag{2}$$

Above, **u** is the cross section averaged fluid velocity (m/s) along the tangent of the center line of a pipe. A (m^2) is the cross section area of the pipe, ρ (kg/m^3) is the density, and p (N/m^2) is the pressure. For more information, refer to the section Theory for the Pipe Flow User Interface in the *Pipe Flow Module User's Guide*.

Expressions for the Darcy Friction Factor

The second term on the right hand side of Equation 2 accounts for pressure drop due to viscous shear. The Pipe Flow physics uses the Churchill friction model (Ref. 1) to calculate f_D. It is valid for laminar flow, turbulent flow, and the transitional region in between. The Churchill friction model is predefined in the Non-Isothermal Pipe Flow interface and is given by:

$$f_D = 8\left[\left(\frac{8}{Re}\right)^{12} + (A+B)^{-1.5}\right]^{1/12} \tag{3}$$

where

$$A = \left[-2.457\ln\left(\left(\frac{7}{Re}\right)^{0.9} + 0.27(e/d)\right)\right]^{16} \tag{4}$$

$$B = \left(\frac{37530}{Re}\right)^{16} \tag{5}$$

©2013 COMSOL

As seen from the equations above, the friction factor depends on the surface roughness divided by diameter of the pipe, e/d. Surface roughness values can be selected from a list in the Pipe Properties feature or be entered as user-defined values.

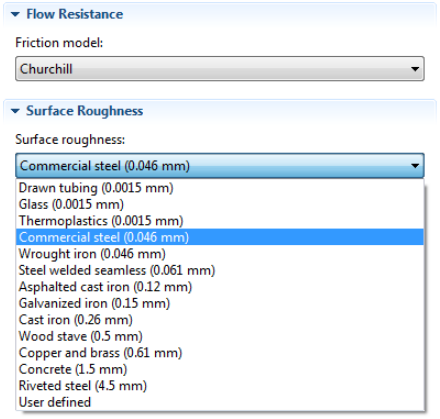


Figure 4: The Friction model and Surface roughness settings are found in the Pipe Properties feature.

In the Churchill equation, f_D is also a function of the fluid properties, flow velocity and geometry, through the Reynolds number:

$$\mathrm{Re} = \frac{\rho u d}{\mu} \tag{6}$$

The physical properties of water as function of temperature are directly available from the software's built-in material library.

HEAT TRANSFER EQUATIONS

Cooling Channels

The energy equation for the cooling water inside the pipe is:

$$\rho A C_p \frac{\partial T}{\partial t} + \rho A C_p \mathbf{u} \cdot \nabla T = \nabla \cdot A k \nabla T + f_D \frac{\rho A}{2 d_h} |\mathbf{u}|^3 + Q_{\mathrm{wall}} \tag{7}$$

where C_p (J/(kg·K)) is the heat capacity at constant pressure, T is the cooling water temperature (K), and k (W/(m·K)) is the thermal conductivity. The second term on the right hand side corresponds to heat dissipated due to internal friction in the fluid. It is negligible for the short channels considered here. Q_{wall} (W/m) is a source term that accounts for the heat exchange with the surrounding mold block.

Mold Block and Polyurethane Part

Heat transfer in the solid steel mold block as well as the molded polyurethane part is governed by conduction:

$$\rho C_p \frac{\partial T_2}{\partial t} = \nabla \cdot k \nabla T_2 \tag{8}$$

Above, T_2 is the temperature in the solids. The source term Q_{wall} comes into play for the heat balance in Equation 8 through a line heat source where the pipe is situated. This coupling is automatically done by the Wall Heat Transfer feature in the Non-Isothermal Pipe Flow interface.

Heat Exchange

The heat exchange term Q_{wall} (W/m) couple the two energy balances given by Equation 7 and Equation 8:

The heat transfer through the pipe wall is given by

$$Q_{\text{wall}} = hZ(T_{\text{ext}} - T) \tag{9}$$

In Equation 9 Z (m) is the perimeter of the pipe, h (W/(m^2·K)) a heat transfer coefficient and T_{ext} (K) the external temperature outside of the pipe. Q_{wall} appears as a source term in the pipe heat transfer equation.

The Wall heat transfer feature requires the external temperature and at least an internal film resistance.

T_{ext} can be a constant, parameter, expression, or given by a temperature field computed by another physics interface, typically a 3D Heat Transfer interface. h is automatically calculated through film resistances and wall layers that are added as subnodes. For details, refer to the section Theory for the Heat Transfer in Pipes User Interface in the *Pipe Flow Module User's Guide*.

In this model example, T_{ext} is given as the temperature field computed by a 3D heat transfer interface, and automatic heat transfer coupling is done to the 3D physics side as a line source. The temperature coupling between the pipe and the surrounding domain is implemented as a line heat source in the 3D domain. The source strength is proportional to the temperature difference between the pipe fluid and the surrounding domain.

©2013 COMSOL

The Wall Heat Transfer feature is added to the Non-Isothermal Pipe Flow interface, and the **External temperature** is set to the temperature of the Heat Transfer in solids interface.

Figure 5: In the Wall Heat Transfer feature, set the External temperature to the temperature field calculated by the Heat Transfer in Solids interface.

The heat transfer coefficient, h, depends on the physical properties of water and the nature of the flow and is calculated from the Nusselt number:

$$h = \mathrm{Nu}\frac{k}{d_\mathrm{h}} \tag{9-6}$$

where k is the thermal conductivity of the material, and Nu is the Nusselt number. d_h is the hydraulic diameter of the pipe.

COMSOL detects if the flow is laminar or turbulent. For the laminar flow regime, an analytic solution is available that gives $\mathrm{Nu} = 3.66$ for circular tubes (Ref. 2). For turbulent flow inside channels of circular cross sections the following Nusselt correlation is used (Ref. 3):

$$\mathrm{Nu}_\mathrm{int} = \frac{(f_\mathrm{D}/8)(\mathrm{Re}-1000)\mathrm{Pr}}{1+12.7(f_\mathrm{D}/8)^{1/2}(\mathrm{Pr}^{2/3}-1)} \tag{10}$$

where Pr is the Prandtl number:

$$\mathrm{Pr} = \frac{C_p\mu}{k} \tag{11}$$

Note that Equation 10 is a function of the friction factor, f_D, and therefore that the radial heat transfer will increase with the surface roughness of the channels.

Note: All the correlations discussed above are automatically used by the Wall Heat Transfer feature in the Pipe Flow Module, and it detects if the flow is laminar or turbulent for automatic selection of the correct correlation.

Results and Discussion

The steel mold and polyurethane part, initially at 473 K, are cooled for 10 minutes by water at room temperature. Figures below show sample results when flow rate of the cooling water is 10 liters/minute and the surface roughness of the channels is 46 μm. After two minutes of cooling, the hottest and coldest parts of the polyurethane part differ by approximately 40 K (Figure 7).

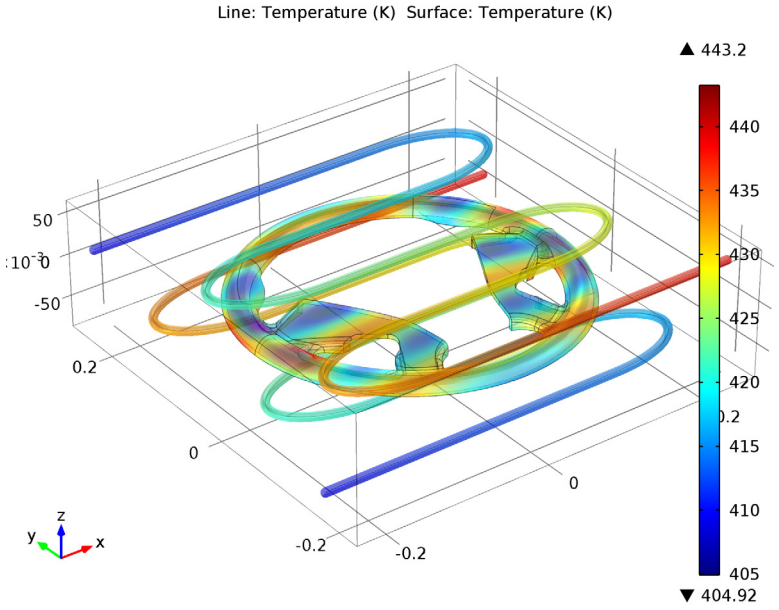


Figure 7: Temperature distribution in the polyurethane part and the cooling channels after 2 minutes of cooling.

©2013 COMSOL

Figure 8 shows the temperature distribution in the steel mold after 2 minutes. The temperature footprint of the cooling channels is clearly visible.

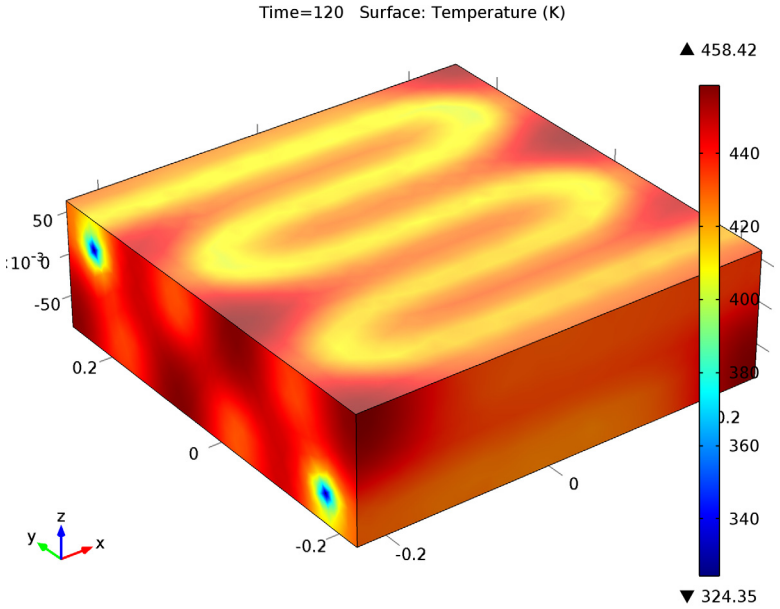


Figure 8: Temperature distribution in the steel mold block after 2 minutes of cooling.

After 10 minutes of cooling, the temperature in the mold block is more uniform, with a temperature at the center of approximately 333 K (Figure 9). Still, the faces with cooling channel inlets and outlets are more than 20 K hotter.

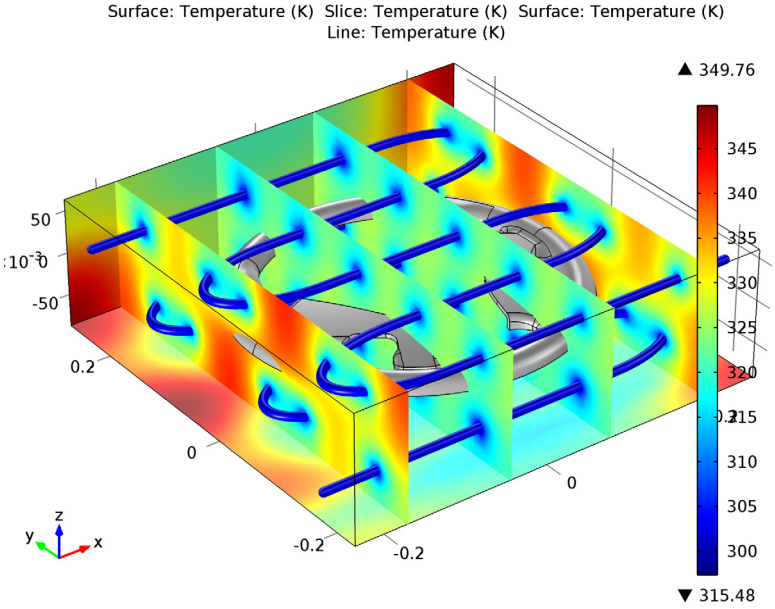


Figure 9: Temperature distribution in the steel mold block after 10 minutes of cooling.

The blue line in Figure 10 shows the average temperature of the polyurethane part as function of the cooling time. The temperature is 333 K after 10 minutes of cooling. To evaluate the influence of factors affecting the cooling time, additional simulations were run varying the flow rate of the cooling water, the surface roughness of the cooling channels, and the mold material. The conditions are summarized in the table below.

TABLE 1: COOLING CONDITIONS

MOLD MATERIAL	WATER FLOW RATE (L/MIN)	SURFACE ROUGHNESS (MM)	AVERAGE T AFTER 10 MIN (K)	LINE COLOR
Steel	10	0.046	333	Blue
Steel	20	0.046	325	Green
Steel	10	0.46	328	Red
Aluminium	10	0.046	301	Magenta

©2013 COMSOL

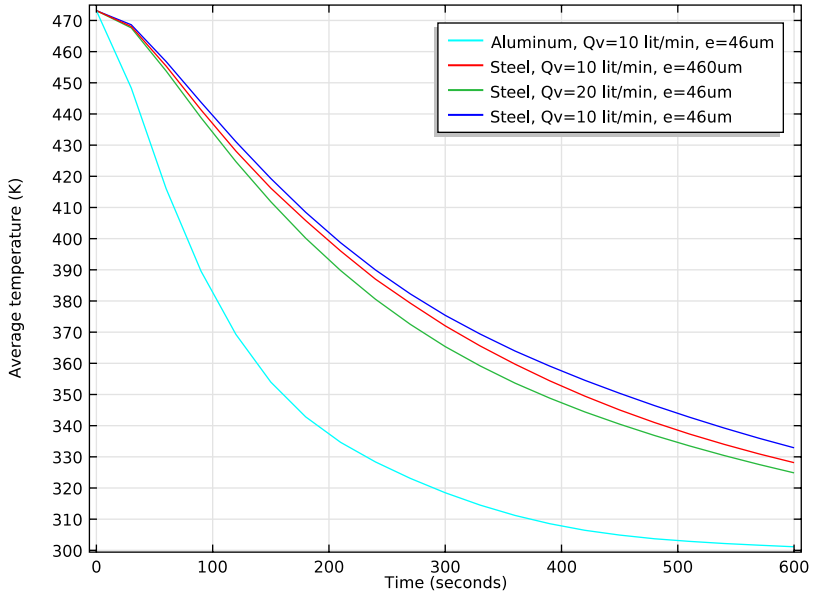


Figure 10: Average temperature of the polyurethane part as function of time and cooling conditions.

Clearly, the thermal conductivity of the mold material is the most important factor in this comparison, followed by flow rate and surface roughness of the cooling channels. Assuming that 340 K is an acceptable temperature at the end of the production cycle, it can be found that changing the mold material reduces the cooling time by 67%, increasing the flow rate reduces the cooling time by 17%, and increasing surface roughness reduces the cooling time by 11%.

References

1. S.W. Churchill, *Friction factor equations span all fluid-flow regimes*, Chem. Eng., 84 (24), 1997, 91.

2. Incropera, Frank P.; DeWitt, David P. (2002). *Fundamentals of Heat and Mass Transfer (5th ed.)*. Hoboken: Wiley. pp. 486, 487. ISBN 0-471-38650-2.

3. V. Gnielinski, *Int. Chem. Eng.* 16, 1976, 359.

Model Library path: `Pipe_Flow_Module/Heat_Transfer/mold_cooling`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Fluid Flow>Non-Isothermal Flow>Non-Isothermal Pipe Flow (nipfl)**.

4 Click **Add Selected**.

5 In the **Add physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

6 Click **Add Selected**.

7 Click **Next**.

8 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Time Dependent**.

9 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
T_init_mold	473.15[K]	Initial temperature, mold
T_coolant	288.15[K]	Steady-state inlet temperature, coolant

Step 1

Create a smooth step function to decrease the coolant temperature at the beginning of the process.

1 Right-click **Global Definitions** and choose **Functions>Step**.

2 In the **Step** settings window, locate the **Parameters** section.

 ©2013 COMSOL

3 In the **Location** edit field, type `2.5`.

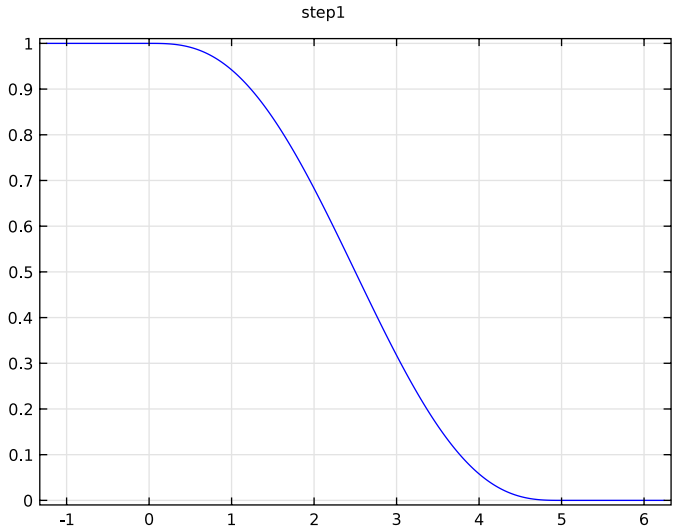
4 In the **From** edit field, type `1`.

5 In the **To** edit field, type `0`.

6 Click to expand the **Smoothing** section. In the **Size of transition zone** edit field, type `5`.

Optionally, you can inspect the shape of the step function:

7 Click the **Plot** button.



Variables 1

1 Right-click **Global Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Description
`T_inlet`	`T_coolant+(T_init_mold-T_coolant)*step1(t[1/s])`	`Ramped inlet temperature, coolant`

GEOMETRY 1

First, import the steering wheel part from a CAD design file.

Import 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Import**.

2 In the **Import** settings window, locate the **Import** section.

3 Click the **Browse** button.

4 Browse to the model's Model Library folder and double-click the file `mold_cooling_top.mphbin`.

5 Click the **Import** button.

Second, draw the mold and cooling channels. To simplify this step, insert a prepared geometry sequence from file. After insertion you can study each geometry step in the sequence.

6 In the **Model Builder** window, right-click **Geometry 1** and choose **Insert Sequence from File**.

7 Browse to the model's Model Library folder and double-click the file `mold_cooling_geom_sequence.mph`.

8 Click the **Build All** button.

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

10 Click the **Transparency** button on the Graphics toolbar.

DEFINITIONS

Create the selections to simplify the model specification.

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

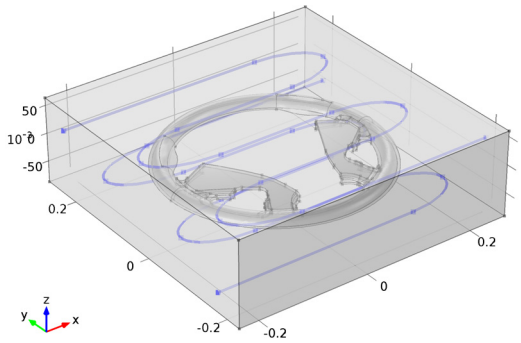
3 From the **Geometric entity level** list, choose **Edge**.

Select one segment of the upper channel and one segment of the lower.

4 Select Edges 6 and 7 only.

 ©2013 COMSOL

5 Select the **Group by continuous tangent** check box to select entire channels.



6 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.

7 Go to the **Rename Explicit** dialog box and type `Cooling channels` in the **New name** edit field.

8 Click **OK**.

MATERIALS

The next step is to specify material properties for the model. Select water and steel from the built-in materials database.

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Water, liquid**.

3 Click **Add Material to Model**.

Water, liquid

1 In the **Model Builder** window, under **Model 1>Materials** click **Water, liquid**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 From the **Selection** list, choose **Cooling channels**.

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Steel AISI 4340**.

3 Click **Add Material to Model**.

Steel AISI 4340

1 In the **Model Builder** window, under **Model 1>Materials** click **Steel AISI 4340**.

2 Select Domain 1 only.

Next, create a material with the properties of polyurethane.

Material 3

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Select Domain 2 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value
Thermal conductivity	k	0.32
Density	rho	1250
Heat capacity at constant pressure	Cp	1540

5 Right-click **Model 1>Materials>Material 3** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type `Polyurethane` in the **New name** edit field.

7 Click **OK**.

NON-ISOTHERMAL PIPE FLOW

1 In the **Non-Isothermal Pipe Flow** settings window, locate the **Edge Selection** section.

2 From the **Selection** list, choose **Cooling channels**.

Pipe Properties 1

1 In the **Model Builder** window, under **Model 1>Non-Isothermal Pipe Flow** click **Pipe Properties 1**.

2 In the **Pipe Properties** settings window, locate the **Pipe Shape** section.

3 From the list, choose **Round**.

4 In the d_i edit field, type `1[cm]`.

5 Locate the **Flow Resistance** section. From the **Surface roughness** list, choose **Commercial steel (0.046 mm)**.

Temperature 1

1 In the **Model Builder** window, under **Model 1>Non-Isothermal Pipe Flow** click **Temperature 1**.

 ©2013 COMSOL

2 In the **Temperature** settings window, locate the **Temperature** section.

3 In the T_{in} edit field, type `T_inlet`.

Inlet 1

1 In the **Model Builder** window, right-click **Non-Isothermal Pipe Flow** and choose the point condition **Pipe Flow>Inlet**.

2 Select Points 3 and 4 only.

3 In the **Inlet** settings window, locate the **Inlet Specification** section.

4 From the **Specification** list, choose **Volumetric flow rate**.

5 In the $q_{v,0}$ edit field, type `10[dm^3/min]`.

Heat Outflow 1

1 Right-click **Non-Isothermal Pipe Flow** and choose the point condition **Heat Transfer in Pipes>Heat Outflow**.

2 Select Points 269 and 270 only.

Wall Heat Transfer 1

1 Right-click **Non-Isothermal Pipe Flow** and choose the edge condition **Heat Transfer in Pipes>Wall Heat Transfer**.

2 In the **Wall Heat Transfer** settings window, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Cooling channels**.

4 Locate the **Heat Transfer Model** section. From the T_{ext} list, choose **Temperature (ht)**.

5 Right-click **Model 1>Non-Isothermal Pipe Flow>Wall Heat Transfer 1** and choose **Internal Film Resistance**.

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Non-Isothermal Pipe Flow** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the u edit field, type `0.1`.

4 In the T edit field, type `T_init_mold`.

HEAT TRANSFER IN SOLIDS

1 In the **Model Builder** window, expand the **Model 1>Heat Transfer in Solids** node, then click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the $T2$ edit field, type `T_init_mold`.

Heat Flux 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose **Heat Flux**.

2 In the **Heat Flux** settings window, locate the **Boundary Selection** section.

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

4 Locate the **Heat Flux** section. Click the **Inward heat flux** button.

5 In the h edit field, type 2.

MESH 1

Edge 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Edge**.

2 In the **Edge** settings window, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Cooling channels**.

Size 1

1 Right-click **Model 1>Mesh 1>Edge 1** and choose **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

Free Tetrahedral 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

Size 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

6 Click the **Build Selected** button.

STUDY 1

Step 1: Time Dependent

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

2 In the **Time Dependent** settings window, locate the **Study Settings** section.

 ©2013 COMSOL

3 In the **Times** edit field, type `range(0,30,600)`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Data Sets

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Solution**.

2 Right-click **Results>Data Sets>Solution 2** and choose **Add Selection**.

3 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 2 only.

6 In the **Model Builder** window, right-click **Data Sets** and choose **Solution**.

7 Right-click **Results>Data Sets>Solution 3** and choose **Add Selection**.

8 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

9 From the **Geometric entity level** list, choose **Boundary**.

10 Select Boundaries 3 and 5 only.

Temperature (nipfl)

1 In the **Model Builder** window, expand the **Results>Temperature (nipfl)** node, then click **Line 1.1**.

2 In the **Line** settings window, locate the **Coloring and Style** section.

3 Select the **Radius scale factor** check box.

4 In the associated edit field, type `1`.

5 Clear the **Color legend** check box.

6 Locate the **Data** section. From the **Data set** list, choose **Solution 1**.

7 From the **Time** list, choose **120**.

8 In the **Model Builder** window, right-click **Temperature (nipfl)** and choose **Surface**.

9 In the **Surface** settings window, locate the **Data** section.

10 From the **Data set** list, choose **Solution 2**.

11 From the **Time** list, choose **120**.

12 Click the **Plot** button.

Temperature (ht)

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

2 In the **3D Plot Group** settings window, locate the **Data** section.

3 From the **Time** list, choose **120**.

4 Click the **Plot** button.

5 Click the **Transparency** button on the Graphics toolbar.

6 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

7 In the **Surface** settings window, locate the **Coloring and Style** section.

8 From the **Color table** list, choose **Rainbow**.

9 Click the **Plot** button.

3D Plot Group 6

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 6** and choose **Surface**.

3 In the **Surface** settings window, locate the **Data** section.

4 From the **Data set** list, choose **Solution 3**.

5 In the **Model Builder** window, right-click **3D Plot Group 6** and choose **Slice**.

6 In the **Slice** settings window, locate the **Plane Data** section.

7 In the **Planes** edit field, type `4`.

8 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

9 Right-click **3D Plot Group 6** and choose **Surface**.

10 In the **Surface** settings window, locate the **Data** section.

11 From the **Data set** list, choose **Solution 2**.

12 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

13 From the **Color** list, choose **Gray**.

14 Right-click **3D Plot Group 6** and choose **Line**.

15 In the **Line** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Non-Isothermal Pipe Flow (Heat Transfer in Pipes)>Temperature (T)**.

16 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

17 In the **Tube radius expression** edit field, type `0.5*nipfl.dh`.

18 Select the **Radius scale factor** check box.

19 From the **Coloring** list, choose **Uniform**.

20 From the **Color** list, choose **Blue**.

21 Click the **Plot** button.

 ©2013 COMSOL

Derived Values

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Average>Volume Average**.

2 In the **Volume Average** settings window, locate the **Selection** section.

3 Click **Paste Selection**.

4 Go to the **Paste Selection** dialog box.

5 In the **Selection** edit field, type 2.

6 Click the **OK** button.

7 Click the **Evaluate** button.

8 In the **Table** window, click **Table Graph**.

©2013 COMSOL