Created in COMSOL Multiphysics 6.0

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 Nonisothermal 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.

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.

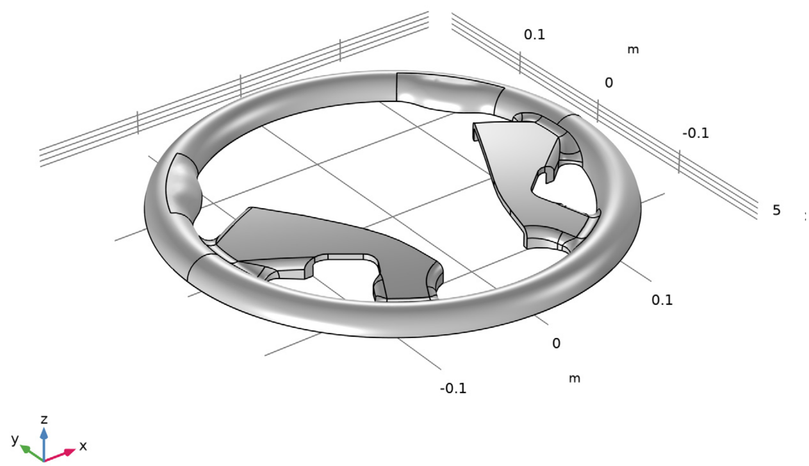


Figure 2: Polyurethane parts for a steering wheel. The top half of the grip is modeled in this example.

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.

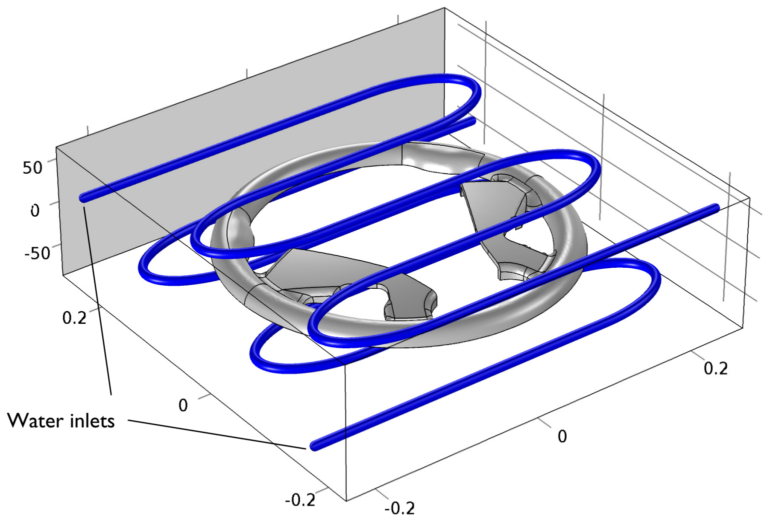


Figure 3: Mold block and cooling channels.

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:

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

$$\frac{\partial A\rho}{\partial t} + \nabla\cdot(A\rho\mathbf{u}) = 0 \qquad (1)$$

Above, $\mathbf{u}$ is the cross section averaged fluid velocity (m/s) along the tangent of the centerline 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 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 1 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 Nonisothermal Pipe Flow interface and is given by:

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

where

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

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

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.

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

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

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:

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

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:

$$\rho C_p \frac{\partial T_2}{\partial t} = \nabla \cdot k \nabla T_2 \quad (3)$$

Above, T_2 is the temperature in the solids. The source term Q_{wall} comes into play for the heat balance in Equation 3 through a line heat source where the pipe is situated. This coupling is automatically done by the Wall Heat Transfer feature in the Nonisothermal Pipe Flow interface.

Heat Exchange

The heat exchange term Q_{wall} (W/m) couple the two energy balances given by Equation 2 and Equation 3:

The heat transfer through the pipe wall is given by

$$Q_{\text{wall}} = hZ(T_{\text{ext}} - T) \quad (4)$$

In Equation 4 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 *Theory for the Heat Transfer in Pipes 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.

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

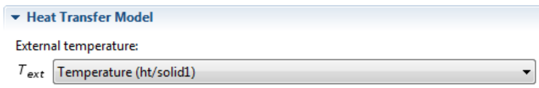


Figure 4: In the Wall Heat Transfer feature, set the External temperature to the temperature field computed by the Heat Transfer in Solids interface.

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:

$$h = \mathrm{Nu}\frac{k}{d_{\mathrm{h}}}$$

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):

$$\mathrm{Nu}_{\mathrm{int}} = \frac{(f_{\mathrm{D}}/8)(\mathrm{Re}-1000)\mathrm{Pr}}{1+12.7(f_{\mathrm{D}}/8)^{1/2}(\mathrm{Pr}^{2/3}-1)} \qquad (5)$$

where Pr is the Prandtl number:

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

Note that Equation 5 is a function of the friction factor, f_{D}, and therefore that the radial heat transfer increases 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 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 5).

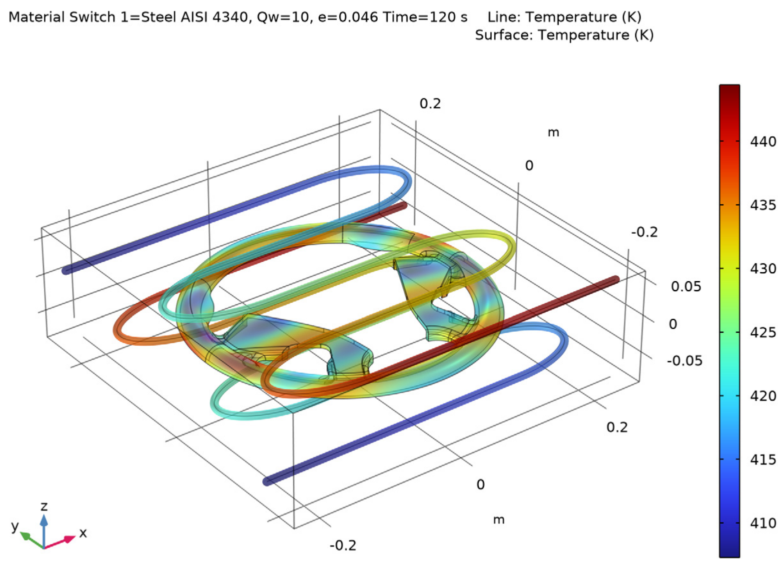


Figure 5: Temperature distribution in the polyurethane part and the cooling channels after 2 minutes of cooling.

[Figure 6](#) shows the temperature distribution in the steel mold after 2 minutes. The temperature footprint of the cooling channels is clearly visible.

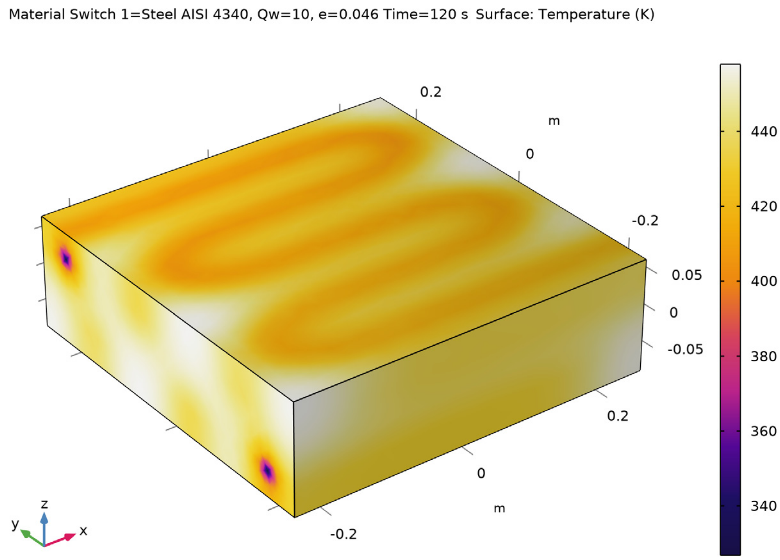


Figure 6: Temperature distribution in the steel mold block after 2 minutes of cooling.

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 7). Still, the faces with cooling channel inlets and outlets are more than 20 K hotter.

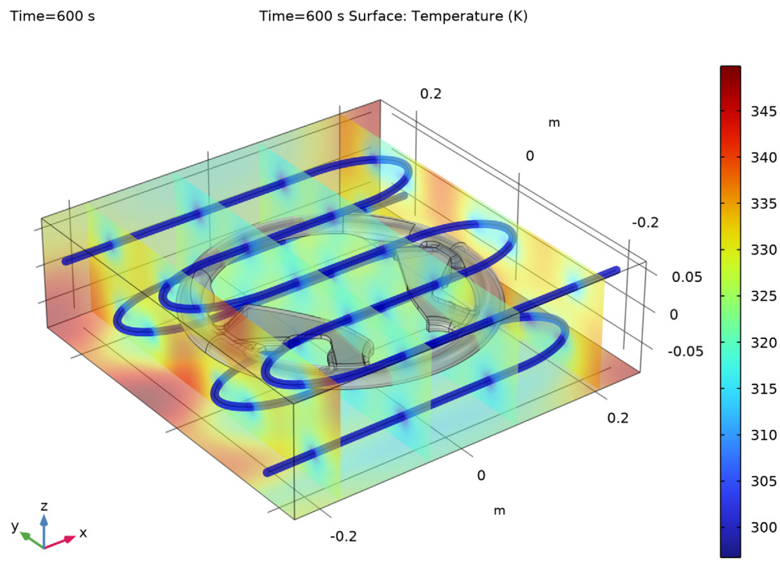


Figure 7: Temperature distribution in the steel mold block after 10 minutes of cooling.

To evaluate the influence of factors affecting the cooling time, use Material and Parametric sweeps. Figure 8 shows the average temperature of the polyurethane part as function of the cooling time for the several flow rates of the cooling water, the surface roughness of the cooling channels, and the mold materials.

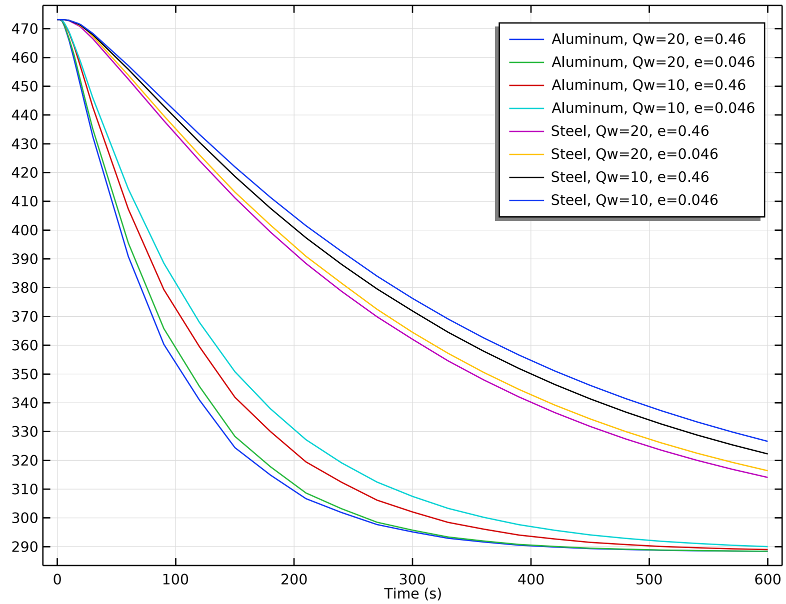


Figure 8: Average temperature of the polyurethane part as function of time and cooling conditions.

References

1. S.W. Churchill, "Friction factor equations span all fluid-flow regimes," *Chem. Eng.*, vol. 84, no. 24, p.91, 1997.

2. F.P. Incropera, D.P. DeWitt. *Fundamentals of Heat and Mass Transfer,* 5th ed., John Wiley & Sons, pp. 486–487, 2002.

3. V. Gnielinski, "New Equation for Heat and Mass Transfer in Turbulent Pipe and Channel Flow," *Int. Chem. Eng.* vol. 16, pp. 359–368, 1976.

Application Library path: `Pipe_Flow_Module/Heat_Transfer/mold_cooling`

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 **3D**.
2 In the **Select Physics** tree, select **Fluid Flow>Nonisothermal Flow>Nonisothermal Pipe Flow (nipfl)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Time Dependent**.
8 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 In the table, enter the following settings:

Name	Expression	Value	Description
T_init_mold	473.15[K]	473.15 K	Initial temperature, mold
T_coolant	288.15[K]	288.15 K	Steady-state inlet temperature, coolant
Qw	10[l/min]	1.6667E-4 m³/s	Coolant flow rate
e	0.046[mm]	4.6E-5 m	Surface roughness

Step 1 (step1)

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

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Step**.

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

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

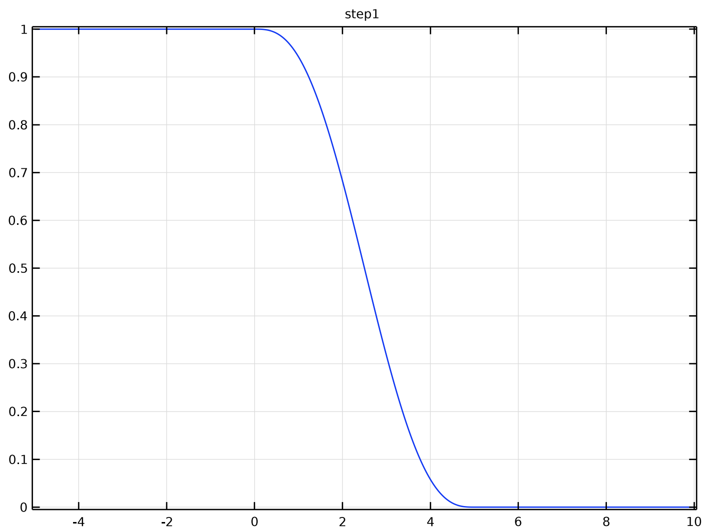
4 In the **From** text field, type `1`.

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

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

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

7 Click **Plot**.



Variables 1

1 In the **Home** toolbar, click a= **Variables** and choose **Global Variables**.

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

3 In the table, enter the following settings:

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

GEOMETRY 1

Import 1 (imp1)

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

1 In the **Home** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 Click **Browse**.
4 Browse to the model's Application Libraries folder and double-click the file `mold_cooling_top.mphbin`.
5 Click **Import**.
6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.
7 In the **New Cumulative Selection** dialog box, type `Wheel` in the **Name** text field.
8 Click **OK**.

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.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.
10 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
11 Browse to the model's Application Libraries folder and double-click the file `mold_cooling_geom_sequence.mph`.
12 In the **Geometry** toolbar, click **Build All**.
13 Click the **Zoom Extents** button in the **Graphics** toolbar.
14 Click the **Transparency** button in the **Graphics** toolbar.

Work Plane 1 (wp1)

Create the selections to simplify the model specification.

1 In the **Model Builder** window, click **Work Plane 1 (wp1)**.
2 In the **Settings** window for **Work Plane**, locate the **Selections of Resulting Entities** section.
3 Find the **Cumulative selection** subsection. Click **New**.
4 In the **New Cumulative Selection** dialog box, type `Cooling channels` in the **Name** text field.
5 Click **OK**.

MATERIALS

The next step is to specify material properties for the model. First, select water from the built-in materials database.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Water, liquid (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

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

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

Material Switch 1 (sw1)

Define the mold materials to switch between during a solver sweep.

1 In the **Materials** toolbar, click **More Materials** and choose **Local>Material Switch**.

2 Select Domain 1 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in>Aluminum**.

3 Right-click and choose **Add to Material Switch 1 (sw1)**.

4 In the tree, select **Built-in>Steel AISI 4340**.

5 Right-click and choose **Add to Material Switch 1 (sw1)**.

6 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Polyurethane

Next, create a material with the properties of polyurethane.

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 `Polyurethane` in the **Label** text field.

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

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	0.32	W/(m·K)	Basic
Density	rho	1250	kg/m³	Basic
Heat capacity at constant pressure	Cp	1540	J/(kg·K)	Basic

NONISOTHERMAL PIPE FLOW (NIPFL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Nonisothermal Pipe Flow (nipfl)**.

2 In the **Settings** window for **Nonisothermal Pipe Flow**, locate the **Edge Selection** section.

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

Pipe Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Nonisothermal Pipe Flow (nipfl)** click **Pipe Properties 1**.

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

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

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

5 Locate the **Flow Resistance** section. From the **Surface roughness** list, choose **User defined**.

6 In the **Surface roughness** text field, type `e`.

Temperature 1

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

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

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

Inlet 1

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

2 Select Points 3 and 4 only.

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

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

5 In the $q_{v,0}$ text field, type `Qw`.

Heat Outflow 1

1. In the **Physics** toolbar, click **Points** and choose **Heat Outflow**.
2. Select Points 269 and 270 only.

Wall Heat Transfer 1

1. In the **Physics** toolbar, click **Edges** and choose **Wall Heat Transfer**.
2. In the **Settings** window for **Wall Heat Transfer**, 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)**.

Internal Film Resistance 1

In the **Physics** toolbar, click **Attributes** and choose **Internal Film Resistance**.

Initial Values 1

1. In the **Model Builder** window, under **Component 1 (comp1)> Nonisothermal Pipe Flow (nipfl)** click **Initial Values 1**.
2. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3. In the u text field, type `0.1`.
4. In the T text field, type `T_init_mold`.

HEAT TRANSFER IN SOLIDS (HT)

Initial Values 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Initial Values 1**.
2. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3. In the $T2$ text field, type `T_init_mold`.

Heat Flux 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.
4. Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.
5. In the h text field, type `2`.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

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

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

Size 1

1 Right-click **Edge 1** and choose **Size**.

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

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

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

Even with a maximum element size of 0.003 m, the mesh contains some collapsed elements, resulting in solver errors. Trial and error gives that lowering the curvature factor somewhat will create a mesh with good quality.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Minimum element size** text field, type `0.003`.

5 In the **Curvature factor** text field, type `0.55`.

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

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Output times** text field, type `range(0,2,28) range(30,30,600)`.

DEFINITIONS

To evaluate the average temperature of the polyurethane part for different conditions and mold materials (Figure 8), perform parametric and material sweeps. To avoid accumulating a lot of data while solving, keep only last solution and save the average wheel temperature in a table. For this purpose, add a global probe to the model.

Domain Probe 1 (dom1)

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.
2 In the **Settings** window for **Domain Probe**, locate the **Source Selection** section.
3 From the **Selection** list, choose **Wheel**.
4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Temperature>T2 - Temperature - K**.
5 Locate the **Expression** section. Select the **Description** check box.
6 In the associated text field, type `Average wheel temperature`.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 From the **Sweep type** list, choose **All combinations**.
4 Click **Add**.
5 Click **Add**.
6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Qw (Coolant flow rate)	20 10	l/min
e (Surface roughness)	0.46 0.046	mm

7 Locate the **Output While Solving** section. Select the **Accumulated probe table** check box.
8 Locate the **Study Settings** section. Find the **Memory settings for jobs** subsection. From the **Keep solutions** list, choose **Only last**.

Material Sweep

1 In the **Study** toolbar, click **Material Sweep**.
2 In the **Settings** window for **Material Sweep**, locate the **Study Settings** section.
3 Click **Add**.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

Since the problem involves only heat conduction, you can solved it more efficiently by relaxing nonlinear setting.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

5 In the **Damping factor** text field, type 1.

6 From the **Jacobian update** list, choose **Minimal**.

7 In the **Maximum number of iterations** text field, type 4.

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

RESULTS

Study 1/Solution 1 (4) (sol1)

In the **Results** toolbar, click **More Datasets** and choose **Solution**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Wheel**.

Study 1/Solution 1 (5) (sol1)

In the **Results** toolbar, click **More Datasets** and choose **Solution**.

Study 1/Solution 1 (4) (sol1)

In the **Model Builder** window, click **Study 1/Solution 1 (4) (sol1)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 3 and 5 only.

Temperature (nipfl)

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

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

3 From the **Time (s)** list, choose **120**.

Line 1

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

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

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

4 In the associated text field, type 1.

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

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

Surface 1

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

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

3 From the **Dataset** list, choose **Probe Solution 3 (sol1)**.

4 From the **Time (s)** list, choose **120**.

5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Temperature>T2 - Temperature - K**.

6 In the **Temperature (nipfl)** toolbar, click **Plot**.

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

Temperature (ht)

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

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

3 From the **Time (s)** list, choose **120**.

4 In the **Temperature (ht)** toolbar, click **Plot**.

Surface

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

2 Click **Plot**.

3 Click the **Transparency** button in the **Graphics** toolbar.

3D Plot Group 7

In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Surface 1

1 Right-click **3D Plot Group 7** and choose **Surface**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (4) (sol1)**.

4 From the **Solution parameters** list, choose **From parent**.

5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Temperature>T2 - Temperature - K**.

Slice 1

1 In the **Model Builder** window, right-click **3D Plot Group 7** and choose **Slice**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (4) (sol1)**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Plane Data** section. In the **Planes** text field, type `4`.

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

7 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Temperature>T2 - Temperature - K**.

Mold Temperature

1 In the **Model Builder** window, under **Results** click **3D Plot Group 7**.

2 In the **Settings** window for **3D Plot Group**, type `Mold Temperature` in the **Label** text field.

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

4 In the **Title** text area, type `Time=600 s Surface: Temperature (K) .`

Surface 2

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

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

3 From the **Dataset** list, choose **Probe Solution 3 (sol1)**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Expression** section. In the **Expression** text field, type `0`.

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

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

Line 1

1 Right-click **Mold Temperature** and choose **Line**.

2 In the **Settings** window for **Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Nonisothermal Pipe Flow (Heat Transfer in Pipes)>T - Temperature - K**.

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

4 In the **Tube radius expression** text field, type `0.5*nipfl.dh`.

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

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

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

8 In the **Mold Temperature** toolbar, click **Plot**.

9 Click the **Transparency** button in the **Graphics** toolbar.

Evaluate the average temperature of the polyurethane part for different conditions and mold materials(Figure 8).

Accumulated Probe Table 1

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Accumulated Probe Table 1**.

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

3 From the **Presentation format** list, choose **Filled**.

4 Click **Update**.

Accumulated Probe Table 1.1

1 Right-click **Accumulated Probe Table 1** and choose **Duplicate**.

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

3 Find the **Filled table structure** subsection. From the **Parameter value** list, choose **2: matsw.comp1.sw1=1, Qw=10**.

Accumulated Probe Table 1.2

1 Right-click **Accumulated Probe Table 1** and choose **Duplicate**.

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

3 Find the **Filled table structure** subsection. From the **Parameter value** list, choose **3: matsw.comp1.sw1=2, Qw=20**.

Accumulated Probe Table 1.3

1 Right-click **Accumulated Probe Table 1** and choose **Duplicate**.

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

3 Find the **Filled table structure** subsection. From the **Parameter value** list, choose **4: matsw.comp1.sw1=2, Qw=10**.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Results>Probe Plot Group 6** node, then click **Probe Table Graph 1**.

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

3 From the **Table** list, choose **Accumulated Probe Table 1**.

4 From the **Plot columns** list, choose **All excluding x-axis**.

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

6 In the table, enter the following settings:

Legends
`Aluminum, Qw=20, e=0.46`
`Aluminum, Qw=20, e=0.046`

7 In the **Probe Plot Group 6** toolbar, click **Plot**.

Probe Table Graph 1.1

1 Right-click **Probe Table Graph 1** and choose **Duplicate**.

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

3 From the **Table** list, choose **Accumulated Probe Table 1.1**.

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

Legends
`Aluminum, Qw=10, e=0.46`
`Aluminum, Qw=10, e=0.046`

Probe Table Graph 1.2

1 Right-click **Probe Table Graph 1** and choose **Duplicate**.

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

3 From the **Table** list, choose **Accumulated Probe Table 1.2**.

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

Legends
`Steel, Qw=20, e=0.46`
`Steel, Qw=20, e=0.046`

Probe Table Graph 1.2.1

1 Right-click **Probe Table Graph 1.2** and choose **Duplicate**.

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

3 From the **Table** list, choose **Accumulated Probe Table 1.3**.

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

Legends
`Steel, Qw=10, e=0.46`
`Steel, Qw=10, e=0.046`

Average Temperature

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 6**.

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

3 In the **Average Temperature** toolbar, click **Plot**.