



# 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.

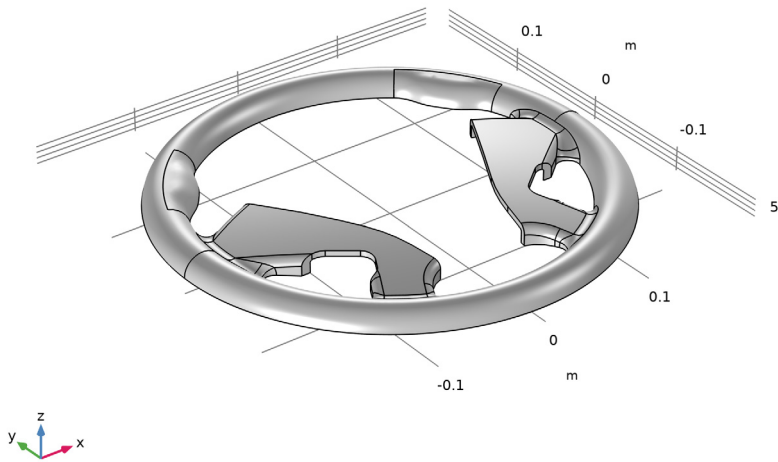
This example shows how you can use the **Nonisothermal Pipe Flow** interface together with the **Heat Transfer in Solids** interface to model a mold cooling process. The equations describing the cooling channels are fully coupled to the heat transfer equations of the mold and the polyurethane part using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



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

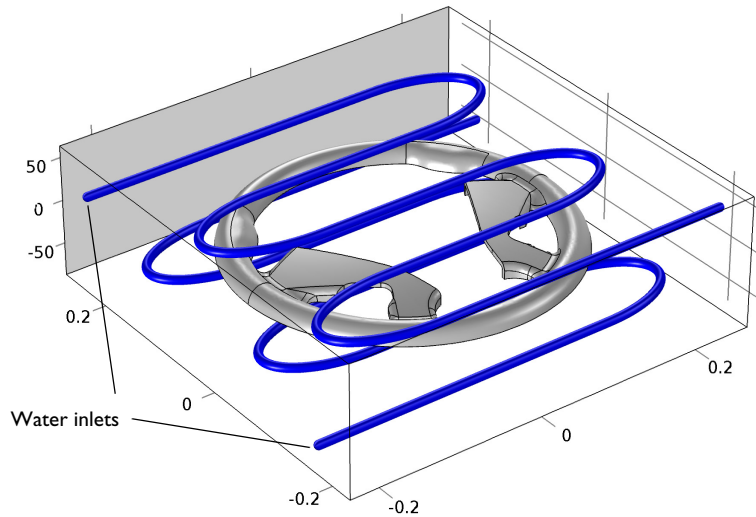
**MODEL GEOMETRY AND PROCESS CONDITIONS**

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.*

After injection of the polyurethane, the average temperature of the mold and 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.

### *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  $\mu\text{m}$ . After two minutes

of cooling, the hottest and coldest parts of the polyurethane part differ by approximately 40 K (Figure 4).

Material Switch 1=Steel AISI 4340, Qw=10 l/min, e=0.046 mm Time=120 s Surface: Temperature (K)  
Line: Temperature (K)

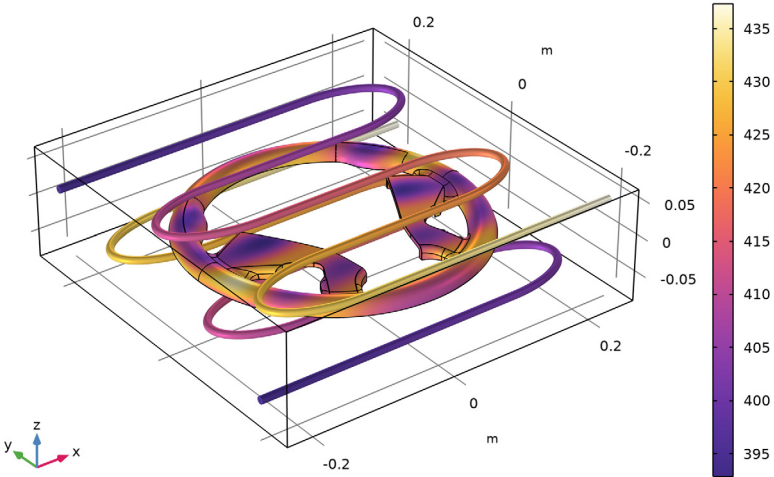


Figure 4: Temperature distribution in the polyurethane part and the cooling channels after 2 minutes of cooling.

Figure 5 shows the temperature distribution in the steel mold after 2 minutes. The temperature footprint of the cooling channels is clearly visible.

Material Switch 1=Steel AISI 4340, Qw=10 l/min, e=0.046 mm Time=120 s Temperature (K)

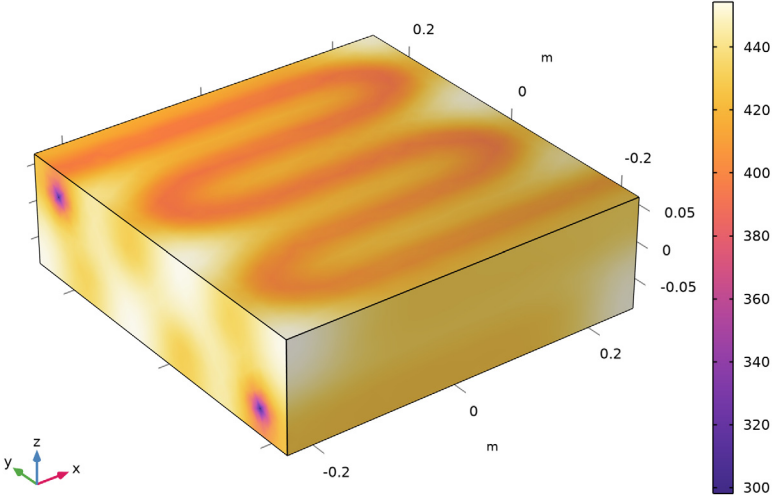


Figure 5: Temperature distribution in the steel mold block after 2 minutes of cooling.

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

Material Switch 1=Steel AISI 4340, Qw=10 l/min, e=0.046 mm Time=600 s Temperature (K)

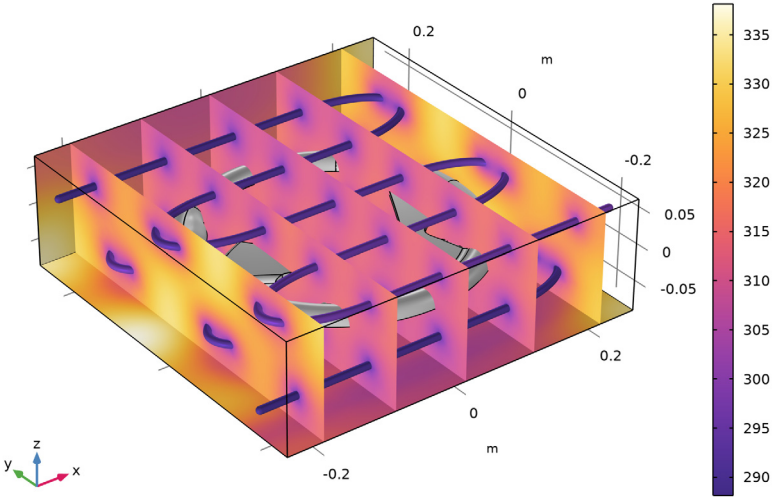


Figure 6: Temperature distribution in the steel mold block after 10 minutes of cooling.

To evaluate the influence of factors affecting the cooling time, use Material and Parametric sweeps. Figure 7 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.

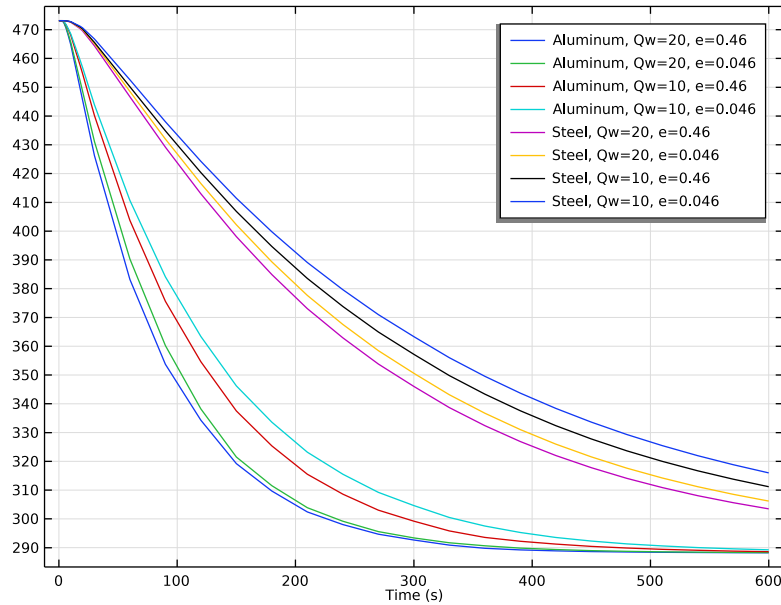


Figure 7: Average temperature of the polyurethane part as function of time and cooling conditions.

---

**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 (nipf)**.



- 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:

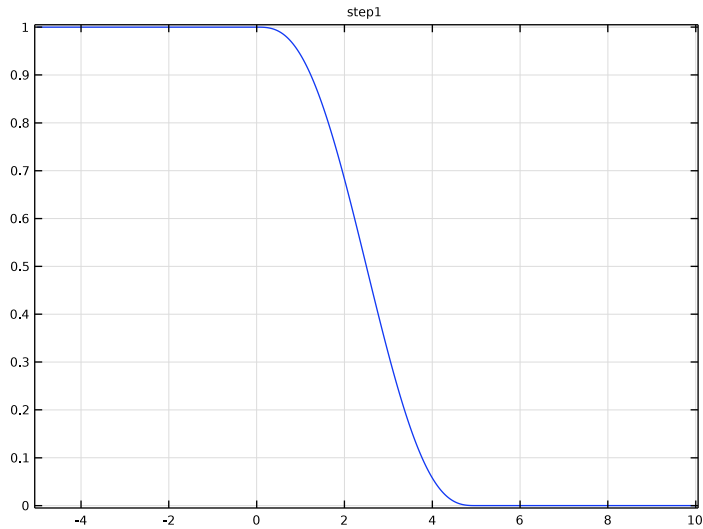
Name	Expression	Value	Description
T_init_mold	473.15[K]	473.15 K	Initial temperature, mold
T_coolant	288.15[K]	288.15 K	Steady-state inlet temperature, coolant
Qw	10[l/min]	1.6667E-4 m <sup>3</sup> /s	Coolant flow rate
e	0.046[mm]	4.6E-5 m	Surface roughness

### 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  **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  **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/m <sup>3</sup>	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.


### *Initial Values 1*

- 1 In the **Model Builder** window, 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.


### *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 I

- 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**.

### Internal Film Resistance I


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

## HEAT TRANSFER IN SOLIDS (HT)

### Initial Values I


- 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  $T^2$  text field, type  $T_{init\_mold}$ .

### Heat Flux I

- 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.


## MULTIPHYSICS

### Pipe Wall Heat Transfer I (pwhtc1)

- 1 In the **Physics** toolbar, click  **Multiphysics Couplings** and choose **Edge>Pipe Wall Heat Transfer**.

## MESH I

### Edge I


- 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 I

- 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 7), 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.

- 6 Select the **Description** check box. In the associated text field, type Average wheel temperature.

## 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.


### 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 solve 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 **Study** toolbar, click  **Compute**.

## RESULTS

In the **Home** toolbar, click  **Add Predefined Plot**.

### ADD PREDEFINED PLOT




- 1 Go to the **Add Predefined Plot** window.
- 2 In the tree, select **Study 1/Solution 1 (sol1)>Heat Transfer in Solids>Temperature (ht)**.
- 3 Click **Add Plot** in the window toolbar.

## RESULTS

*Wheel and Cooling Channels Temperature*

- 1 In the **Settings** window for **3D Plot Group**, type **Wheel and Cooling Channels Temperature** in the **Label** text field.
- 2 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.
- 3 From the **Time (s)** list, choose **120**.

*Selection 1*



- 1 In the **Model Builder** window, expand the **Wheel and Cooling Channels Temperature** node.
- 2 Right-click **Surface** and choose **Selection**.
- 3 In the **Settings** window for **Selection**, locate the **Selection** section.
- 4 From the **Selection** list, choose **Wheel**.
- 5 In the **Wheel and Cooling Channels Temperature** toolbar, click  **Plot**.
- 6 Click the  **Transparency** button in the **Graphics** toolbar.
- 7 In the **Wheel and Cooling Channels Temperature** toolbar, click  **Plot**.

*Wheel and Cooling Channels Temperature*



In the **Model Builder** window, under **Results** click **Wheel and Cooling Channels Temperature**.

*Line 1*

- 1 In the **Wheel and Cooling Channels Temperature** toolbar, click  **Line**.

- 2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.
- 3 From the **Line type** list, choose **Tube**.
- 4 In the **Tube radius expression** text field, type  $0.5 * n_{ipf1} . dh$ .
- 5 Clear the **Rounded end caps** check box.
- 6 Click to expand the **Inherit Style** section. Locate the **Coloring and Style** section. From the **Scale** list, choose **Logarithmic**.
- 7 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface**.
- 8 Clear the **Color and data range** check box.
- 9 In the **Wheel and Cooling Channels Temperature** toolbar, click  **Plot**.
- 10 Locate the **Coloring and Style** section.
- 11 Select the **Radius scale factor** check box. In the associated text field, type 1.
- 12 In the **Wheel and Cooling Channels Temperature** toolbar, click  **Plot**.

#### *Mold Temperature*

- 1 In the **Model Builder** window, right-click **Results** and choose **Add Predefined Plot>Study 1/ Parametric Solutions 1 (sol2)>Heat Transfer in Solids>Temperature (ht)**.
- 2 In the **Settings** window for **3D Plot Group**, type **Mold Temperature** in the **Label** text field.
- 3 Locate the **Data** section. From the **Time (s)** list, choose **120**.
- 4 In the **Mold Temperature** toolbar, click  **Plot**.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 6 In the **Title** text area, type **Temperature (K)**.
- 7 In the **Mold Temperature** toolbar, click  **Plot**.

#### *Temperature (pwhtc1)*


- 1 Right-click **Results** and choose **Add Predefined Plot>Study 1/ Parametric Solutions 1 (sol2)>Pipe Wall Heat Transfer 1>Temperature (pwhtc1)**.
- 2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.
- 3 From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type **Temperature (K)**.

#### *Surface 1*

- 1 Right-click **Temperature (pwhtc1)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type 1.

- 4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 5 From the **Color** list, choose **Gray**.


*Selection 1*

- 1 Right-click **Surface 1** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.
- 3 From the **Selection** list, choose **Wheel**.
- 4 In the **Temperature (pwhtc1)** toolbar, click  **Plot**.

*Surface 2*


- 1 In the **Model Builder** window, right-click **Temperature (pwhtc1)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type T2.
- 4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

*Selection 1*

- 1 Right-click **Surface 2** and choose **Selection**.
- 2 Select Boundaries 3 and 5 only.
- 3 In the **Temperature (pwhtc1)** toolbar, click  **Plot**.

Evaluate the average temperature of the polyurethane part for different conditions and mold materials(Figure 7).

*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.swl=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.swl=2, Qw=10.**

*Probe Table Graph 1*

- 1 In the **Model Builder** window, expand the **Results>Probe Plot Group 1** 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:

<b>Legends</b>
Aluminum, Qw=20, e=0.46
Aluminum, Qw=20, e=0.046

- 7 In the **Probe Plot Group 1** 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:

<b>Legends</b>
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 I.2**.

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

---

<b>Legends</b>
Steel, Qw=20, e=0.46
Steel, Qw=20, e=0.046

---

*Probe Table Graph I.2.1*

1 Right-click **Probe Table Graph I.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 I.3**.

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

---

<b>Legends</b>
Steel, Qw=10, e=0.46
Steel, Qw=10, e=0.046

---

*Average wheel temperature*

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

2 In the **Settings** window for **ID Plot Group**, type Average wheel temperature in the **Label** text field.

