Heat Transfer Module

Application Library Manual

5.2a

COMSOL

Heat Transfer Module Application Library Manual

Version: COMSOL 5.2a

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM020802

Created in COMSOL Multiphysics 5.2a

Heat Generation in a Disc Brake

Introduction

Cars need a brakes for obvious reasons, and you do not want these to fail. Brake failure can be caused by many things, one of which is the overheating of the brake's disc. This example models the heat generation and dissipation in a disc brake of an ordinary car during panic braking and the following release period. When the driver is pressing down on the brakes, kinetic energy is transformed into thermal energy. If the brake discs overheat, the brake pads cease to function through brake fade where the material properties of the brake change due to the temperature overload. Braking power starts to fade already at temperatures above 600 K. This is why it is so important during the design-stages to simulate the transient heating and convective cooling to figure out what the minimum interval between a series of brake engagements is.

In this application, an 1,800 kg car is traveling at 25 m/s (90 km/h or about 56 mph), until the driver suddenly panic brakes for 2 seconds. At that point the eight brake pads slow the car down at a rate of 10 m/s^2 (assuming the wheels do not skid against the road). Upon braking for two seconds the driver releases the brake, leaving the car traveling at 5 m/s for eight seconds without engaging the brakes. The questions to analyze with the model are:

- How hot do the brake discs and pads get when the brake is engaged?
- How much do the discs and pads cool down during the rest that follows the braking?

Model Definition

Model the brake disc as a 3D solid with shape and dimensions as in Figure 1. The disc has a radius of 0.14 m and a thickness of 0.013 m.

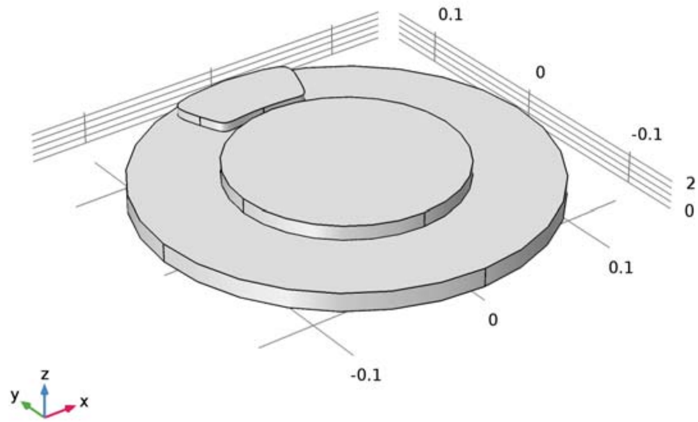


Figure 1: Model geometry, including disc and pad.

The model also includes heat conduction in the disc and the pad through the transient heat transfer equation. The heat dissipation from the disc and pad surfaces to the surrounding air is described by both convection and radiation. Table 1 summarizes the thermal properties of the materials used in this application (Ref. 1).

TABLE 1: MATERIAL PROPERTIES

PROPERTY	DESCRIPTION	DISC	PAD	AIR
ρ (kg/m^3)	Density	7870	2000	1.170
C_p (J/(kg·K))	Heat capacity at constant pressure	449	935	1100
k (W/(m·K))	Thermal conductivity	82	8.7	0.026
ε	Surface emissivity	0.28	0.8	-

After 2 s, contact is made at the interface between the disc and the pad. Neglecting drag and other losses outside the brakes, the brakes' retardation power is given by the negative of the time derivative of the car's kinetic energy:

$$P = -\frac{d}{dt}\left(\frac{mv^2}{2}\right) = -mv\frac{dv}{dt}$$

Here m is the car's mass (1800 kg) and v denotes its speed. Figure 2 shows the profile of v and Figure 3 shows the corresponding acceleration profile.

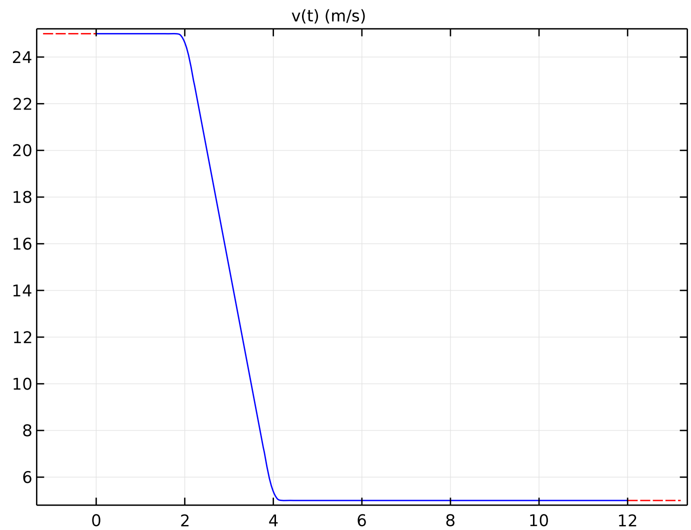


Figure 2: Velocity profile of the disc.

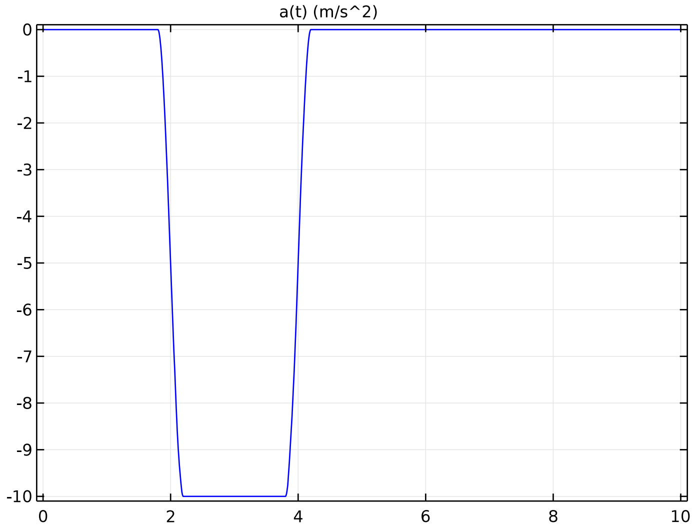


Figure 3: Acceleration profile of the disc.

At one of the eight brakes, the frictional heat source is:

$$P_\mathrm{b} = \frac{P}{8} = -\frac{1}{8}mv\frac{dv}{dt}$$

The contact pressure between the disc and the pad is related to the frictional heat source per unit area, P_b, according to:

$$p = \frac{P_\mathrm{b}}{\mu v}$$

where the friction coefficient μ is here equal to 0.3.

The disc and pad dissipate the heat produced at the boundary between the brake pad and the disc by convection and radiation. This example models the rotation as convection in the disc. The local disc velocity vector is

$$\mathbf{v}_\mathrm{d} = \frac{v}{R}(-y, x)$$

At the end of the computation, produced and dissipated heat can be recovered using the relations

$$
\begin{aligned}
W_{\text{prod}} &= \int_0^{t_0} Q_{\text{prod}} dt \\
W_{\text{diss}} &= \int_0^{t_0} Q_{\text{diss}} dt
\end{aligned}
\tag{1}
$$

Results and Discussion

The surface temperatures of the disc and the pad vary with both time and position. At the contact surface between the pad and the disc the temperature increases when the brake is engaged and then decreases again as the brake is released. You can best see these results in *COMSOL Multiphysics* by generating an animation. Figure 4 displays the surface temperatures just before the end of the braking. A "hot spot" is visible at the contact between the brake pad and disc, just at the pad's edge. This is the area that could overheat to the point of brake failure or fade. The figure also shows the temperature decreasing along the rotational trace after the pad. During the rest, the temperature becomes significantly lower and more uniform in the disc and the pad.

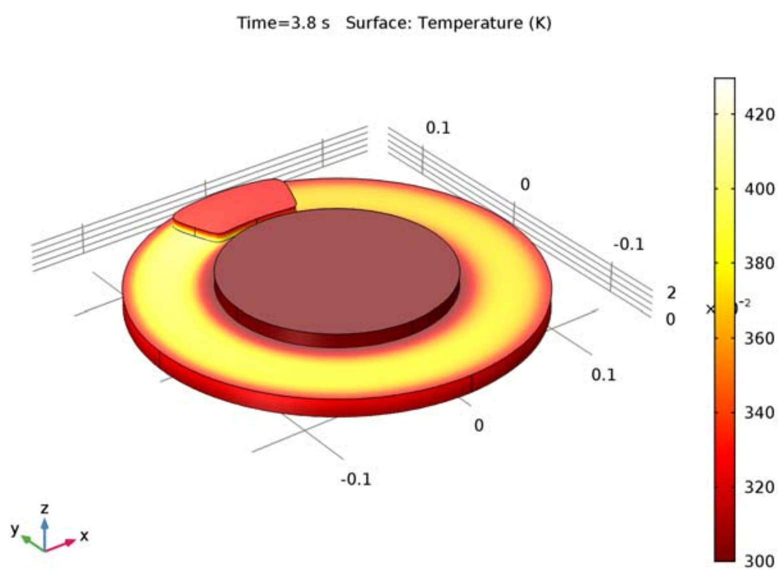


Figure 4: Surface temperature of the brake disc and pad just before releasing the brake (t = 3.8 s).

To investigate the position of the hot spot and the time of the temperature maximum, it is helpful to plot temperature versus time along the line from the center to the pad's edge shown in Figure 5. The result is displayed in Figure 6. You can see that the maximum temperature is approximately 430 K. The hot spot is positioned close to the radially outer edge of the pad. The highest temperature occurs approximately 1 s after engaging the brake.

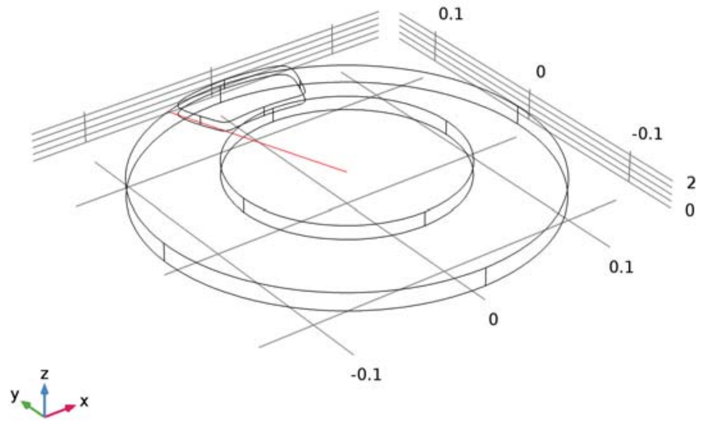


Figure 5: The radial line probed in the temperature vs. time plot in Figure 6.

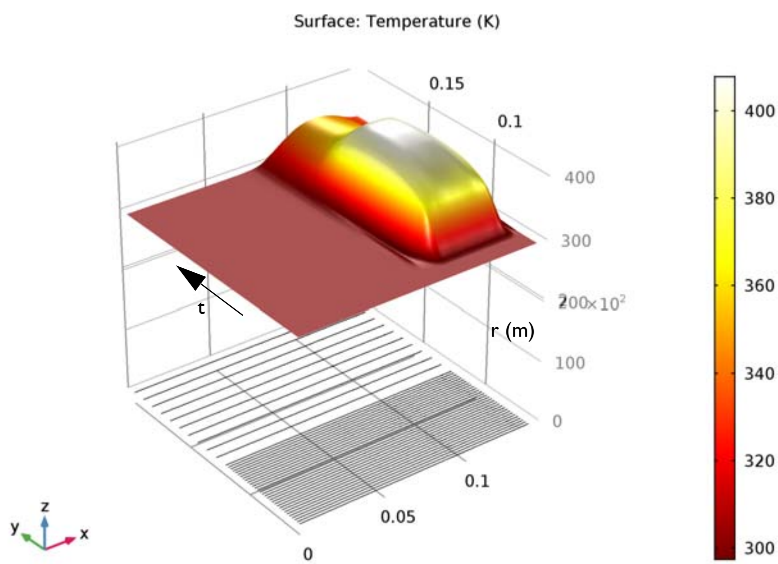


Figure 6: Temperature profile along the line indicated in Figure 5 *at the disc surface*

(z = 0.013 m) as a function of time.

To investigate how much of the generated heat is dissipated to the air, study the surface integrals of the produced heat and the dissipated heat. These integrals give the total heat rate (W) for heat production, Q_{prod}, and heat dissipation, Q_{diss}, as functions of time for the brake disc. The time integrals of these two quantities, W_{prod} and W_{diss}, give the total heat (J) produced and dissipated, respectively, in the brake disc. Figure 7 shows a plot of the total produced heat and dissipated heat versus time. Eight seconds after the driver has stopped braking, a mere fraction of the produced heat has dissipated. In other words, in order to cool down the system sufficiently the brake needs to remain disengaged for a lot longer period than these eight seconds (100 seconds, in fact).

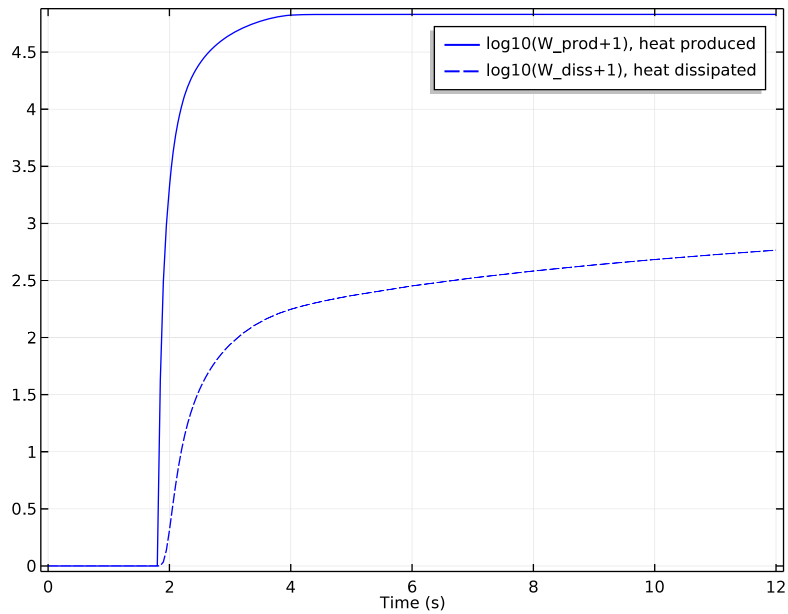


Figure 7: Comparison of total produced heat (solid line) and dissipated heat (dashed).

The results of this application can help engineers investigate how much abuse, in terms of specific braking sequences, a certain brake-disc design can tolerate before overheating. It is also possible to vary the parameters affecting the heat dissipation and investigate their influence.

Reference

1. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 1, eq. 9.88; material properties from appendix A2.

Application Library path: `Heat_Transfer_Module/Thermal_Contact_and_Friction/brake_disc`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

GLOBAL DEFINITIONS

Define the global parameters by loading the corresponding text file provided.

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the application's Application Libraries folder and double-click the file `brake_disc_parameters.txt`.

Geometry 1

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.14`.

4 In the **Height** text field, type `0.013`.

5 On the **Geometry** toolbar, click **Build All**.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.08`.

4 In the **Height** text field, type `0.01`.

5 Locate the **Position** section. In the **z** text field, type `0.013`.

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

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `0.013`.

4 Click **Show Work Plane**.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

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

3 Find the **Added segments** subsection. Click **Add Cubic**.

4 Find the **Control points** subsection. In row **1**, set **yw** to `0.135`.

5 In row **2**, set **xw** to `0.02`, and **yw** to `0.135`.

6 In row **3**, set **xw** to `0.05`, and **yw** to `0.13`.

7 In row **4**, set **xw** to `0.04`, and **yw** to `0.105`.

8 Find the **Added segments** subsection. Click **Add Cubic**.

9 Find the **Control points** subsection. In row **2**, set **xw** to `0.03`, and **yw** to `0.08`.

10 In row **3**, set **xw** to `0.035`, and **yw** to `0.09`.

11 In row **4**, set **xw** to `0`, and **yw** to `0.09`.

12 Find the **Added segments** subsection. Click **Add Cubic**.

13 Find the **Control points** subsection. In row **2**, set **xw** to `-0.035`.

14 In row **3**, set **xw** to `-0.03`, and **yw** to `0.08`.

15 In row **4**, set **xw** to `-0.04`, and **yw** to `0.105`.

16 Find the **Added segments** subsection. Click **Add Cubic**.

17 Find the **Control points** subsection. In row **2**, set **xw** to `-0.05`, and **yw** to `0.13`.

18 In row **3**, set **xw** to `-0.02`, and **yw** to `0.135`.

19 Click **Close Curve**.

To complete the pad cross section, you must make the top-left and top-right corners sharper. Do so by changing the weights of the Bézier curves.

20 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 1 (cubic)**.

21 Find the **Weights** subsection. In the **3** text field, type `2.5`.

22 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 4 (cubic)**.

23 Find the **Weights** subsection. In the **2** text field, type `2.5`.

24 On the **Work Plane** toolbar, click **Build All**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

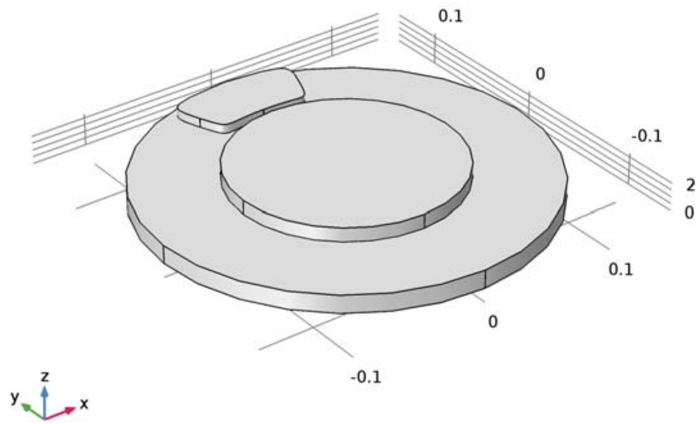
2 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (m)
0.0065

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

The model geometry is now complete.



Next, define some selections of certain boundaries. You will use them when defining the settings for component couplings, boundary conditions, and so on.

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Disc Faces` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1, 2, 4–6, 8, 13–15, and 18 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Pad Faces` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 9, 10, 12, 16, and 17 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Contact Faces` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

To select the contact surface boundary, it is convenient to temporarily switch to wireframe rendering.

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

5 Select Boundary 11 only.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar again to return to the original state.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `External Surfaces` in the **Label** text field.

3 Locate the **Input Entities** section. Select the **All domains** check box.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

These instructions make you select the external boundaries of the wheel and the pad.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

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

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

4 From the **Selection** list, choose **Contact Faces**.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

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

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

4 From the **Selection** list, choose **External Surfaces**.

Now, define the velocity and acceleration of the car through these two piecewise and analytic functions.

Piecewise 1 (pw1)

1 On the **Definitions** toolbar, click **Piecewise**.

2 In the **Settings** window for Piecewise, type `v` in the **Function name** text field.

3 Locate the **Definition** section. In the **Argument** text field, type `t`.

4 From the **Smoothing** list, choose **Continuous second derivative**.

5 From the **Transition zone** list, choose **Absolute size**.

6 In the **Size of transition zone** text field, type `0.2`.

The function definition expects nondimensional quantities for the interval starts and ends, and the function values. The function definition below uses unit conversions to do so.

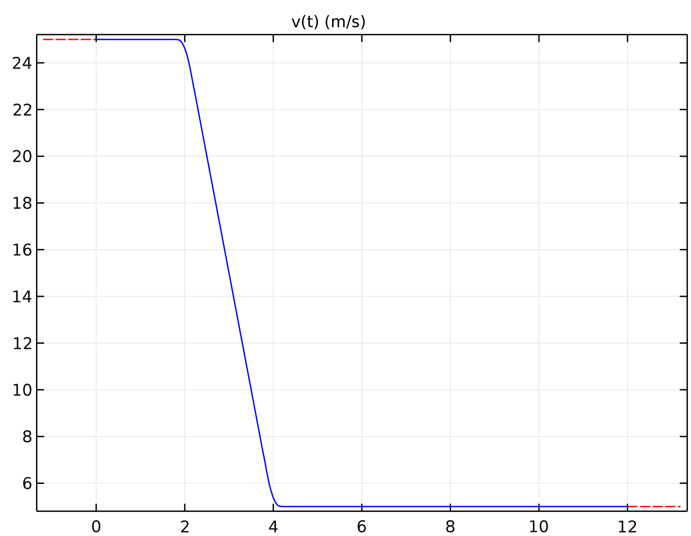
7 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
`0`	`t_brake_start[1/s]`	`v0[s/m]`
`t_brake_start[1/s]`	`t_brake_end[1/s]`	`v0[s/m]+a0*(t[s]-t_brake_start)[s/m]`
`t_brake_end[1/s]`	`12`	`v0[s/m]+a0*(t_brake_end-t_brake_start)[s/m]`

8 Locate the **Units** section. In the **Arguments** text field, type `s`.

9 In the **Function** text field, type `m/s`.

10 Click **Plot**.



Analytic 1 (an1)

1 On the **Definitions** toolbar, click **Analytic**.

2 In the **Settings** window for Analytic, type `a` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `d(v(t),t)`.

4 In the **Arguments** text field, type `t`.

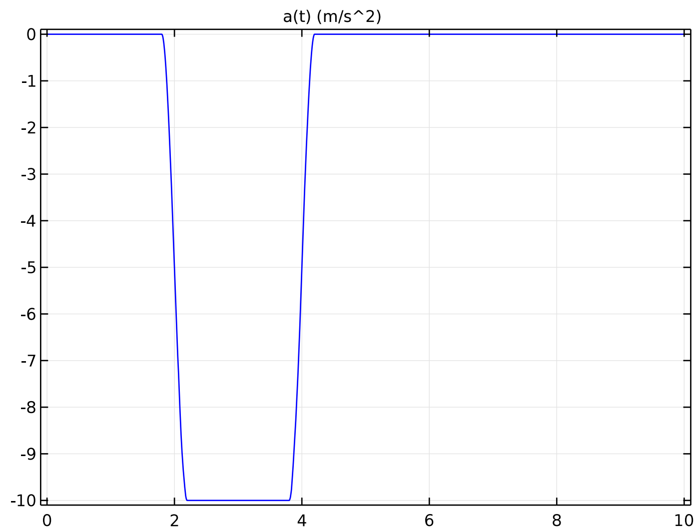
5 Locate the **Units** section. In the **Arguments** text field, type `s`.

6 In the **Function** text field, type `m/s^2`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
t	0	10

8 Click **Plot**.



MATERIALS

Material 1 (mat1)

1 On the **Materials** toolbar, click **Blank Material**.

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

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	82	W/(m·K)	Basic
Density	rho	7870	kg/m³	Basic
Heat capacity at constant pressure	Cp	449	J/(kg·K)	Basic

Material 2 (mat2)

1 On the **Materials** toolbar, click **Blank Material**.
2 In the **Settings** window for Material, type `Pad` in the **Label** text field.
3 Select Domain 3 only.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	8.7	W/(m·K)	Basic
Density	rho	2000	kg/m³	Basic
Heat capacity at constant pressure	Cp	935	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Solid 1

In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.

Translational Motion 1

1 On the **Physics** toolbar, click **Attributes** and choose **Translational Motion**.
2 Select Domains 1 and 2 only.
3 In the **Settings** window for Translational Motion, locate the **Translational Motion** section.
4 Specify the $\mathbf{u}_{\mathrm{trans}}$ vector as

`-y*v(t)/r_wheel`	x
`x*v(t)/r_wheel`	y
`0`	z

Heat Flux 1

1 On 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. Click the **Convective heat flux** button.
5 From the **Heat transfer coefficient** list, choose **External forced convection**.
6 In the L text field, type `0.14`.
7 In the U text field, type `v(t)`.

8 In the T_{ext} text field, type T_air.

Thermal Contact 1

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

2 Select Boundary 11 only.

3 In the **Settings** window for Thermal Contact, locate the **Contact Surface Properties** section.

4 In the p text field, type ht.tc1.Qb/(mu*v(t)).

5 In the H_c text field, type 800[MPa].

6 Locate the **Thermal Friction** section. Click the **Heat rate** button.

7 In the P_b text field, type -m_car*v(t)*a(t)/8.

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, type T_air in the T text field.

Diffuse Surface 1

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

2 In the **Settings** window for Diffuse Surface, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Disc Faces**.

4 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type 0.28.

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

Diffuse Surface 2

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

2 In the **Settings** window for Diffuse Surface, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pad Faces**.

4 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type 0.8.

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

Symmetry 1

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

2 Select Boundary 3 only.

To compute the produced dissipated heats, integrate the corresponding heat rate variables, `Q_prod` and `Q_diss`, over time. For this purpose, define two ODEs using a **Global Equations** node.

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

Global Equations 1

1 On the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for Global Equations, locate the **Units** section.

3 Find the **Dependent variable quantity** subsection. From the list, choose **Energy (J)**.

4 Find the **Source term quantity** subsection. From the list, choose **Power (W)**.

5 Locate the **Global Equations** section. In the table, enter the following settings:

Name	f(u,ut,utt,t) (W)	Initial value (u_0) (J)	Initial value (u_t0) (W)	Description
`W_prod`	`W_prodt-intop1(ht.tc1.Qb)`	`0`	`0`	`Produced heat`
`W_diss`	`W_disst+(intop2(ht.q0+ht.rflux))`	`0`	`0`	`Dissipated heat`

Here, `W_prodt` (resp. `W_disst`) is COMSOL Multiphysics syntax for the time derivative of `W_prod` (resp. `W_diss`). The quantities `intop1(ht.tc1.Qb)` and `intop2(ht.q0+ht.rflux)` correspond to `Q_prod` and `Q_diss`. The table thus defines the first-order ODEs corresponding to Equation 1, so that `W_prod` and `W_diss` host the produced and dissipated heats. The initial values follow from setting $t = 0$.

MESH 1

On the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Free Triangular 1

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

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

3 Select Boundaries 4, 7, and 11 only.

4 Click the **Transparency** button on the **Graphics** toolbar again to return to the original state.

Size

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

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

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

4 On the **Mesh** toolbar, click **Swept**.

Swept 1

Click **Distribution**.

Distribution 1

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

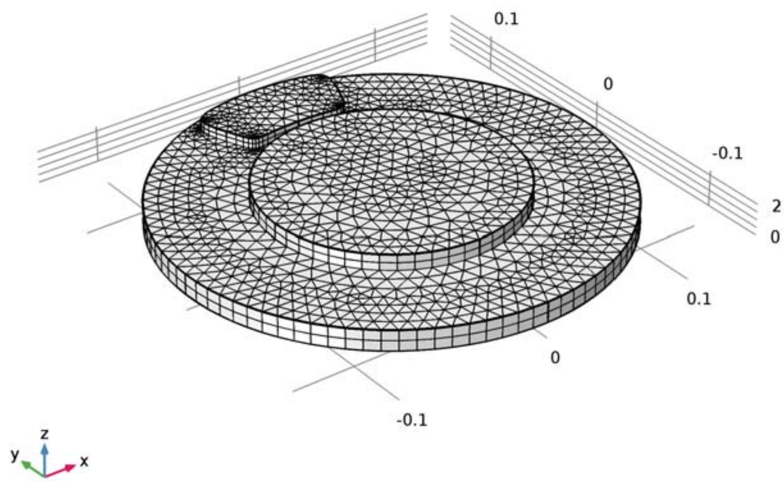
2 In the **Settings** window for Distribution, locate the **Distribution** section.

3 In the **Number of elements** text field, type 2.

4 In the **Model Builder** window, click **Mesh 1**.

5 In the **Settings** window for Mesh, click **Build All**.

The complete mesh consists of roughly 5,700 elements.



STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Times** text field, type `range(0,0.5,1.5) range(1.55,0.05,3) range(3.2, 0.2,5) range(6,1,12)`.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

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

4 In the **Settings** window for Time-Dependent Solver, click to expand the **Absolute tolerance** section.

5 Locate the **Absolute Tolerance** section. In the **Tolerance** text field, type `1e-4`.

6 Click to expand the **Time stepping** section. Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

This setting forces the solver to take at least one step in each specified interval.

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

RESULTS

Temperature (ht)

The first of the two default plots displays the surface temperature of the brake disc and pad at the end of the simulation interval. Modify this plot to show the time just before releasing the brake.

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

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

3 From the **Time (s)** list, choose **3.8**.

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

Compare the result to the plot shown in Figure 4.

To compare the total produced heat and the dissipated heat, as done in Figure 7, follow the steps given below.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for 1D Plot Group, type `Dissipated and Produced Heats` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
4 Click to collapse the **Title** section. Locate the **Plot Settings** section. Select the **x-axis label** check box.
5 In the associated text field, type `Time (s)`.

Point Graph 1

1 On the **Dissipated and Produced Heats** toolbar, click **Point Graph**.
2 Select Point 1 only.
3 In the **Settings** window for Point Graph, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `log10(W_prod+1)`.
5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
6 Click to expand the **Legends** section. Select the **Show legends** check box.
7 From the **Legends** list, choose **Manual**.
8 In the table, enter the following settings:

Legends
`log10(W_prod+1), produced heat`

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.
2 In the **Settings** window for Point Graph, locate the **y-Axis Data** section.
3 In the **Expression** text field, type `log10(W_diss+1)`.
4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
5 Locate the **Legends** section. In the table, enter the following settings:

Legends
`log10(W_diss+1), dissipated heat`

Dissipated and Produced Heats

Finally, follow the steps below to reproduce the plot in Figure 6.

Cut Line 3D 1

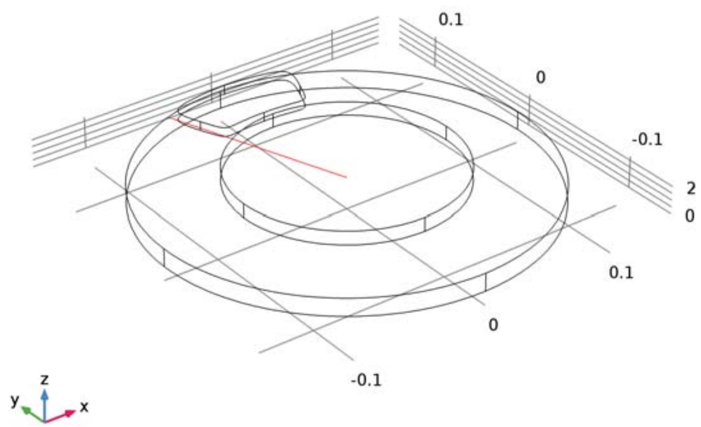
1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for Cut Line 3D, locate the **Line Data** section.

3 In row **Point 1**, set **z** to `0.013`.

4 In row **Point 2**, set **x** to `-0.047`, **y** to `0.1316`, and **z** to `0.013`.

5 Click **Plot**.



Parametric Extrusion 1D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Parametric Extrusion 1D**.

2 In the **Settings** window for Parametric Extrusion 1D, locate the **Data** section.

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

4 Click and shift-click in the list to select all time steps from 1.5 s through 5 s.

2D Plot Group 4

1 On the **Results** toolbar, click **2D Plot Group**.

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

Surface 1

1 Right-click **Temperature Profile vs Time** and choose **Surface**.

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

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

4 Right-click **Results>Temperature Profile vs Time>Surface 1** and choose **Height Expression**.

Height Expression 1

On the **Temperature Profile vs Time** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Buoyancy Flow in Water

Introduction

This example studies the stationary state of free convection in a cavity filled with water and bounded by two vertical plates. To generate the buoyancy flow, the plates are heated at different temperatures, bringing the regime close to the transition between laminar and turbulent.

An important step in setting up a convection model is to assess whether the flow stays laminar or becomes turbulent. It is also important to approximate how fine should be the mesh needed to resolve velocity and temperature gradients. Both of these approximations rely on the velocity scale of the model. This makes the set-up of natural convection problems challenging since the resulting velocity is part of the nonlinear solution. There are, however, tools to approximate scales for natural convection problems. These tools are demonstrated in this application using simple 2D and 3D geometries.

A first 2D model representing a square cavity (see Figure 1) focuses on the convective flow.

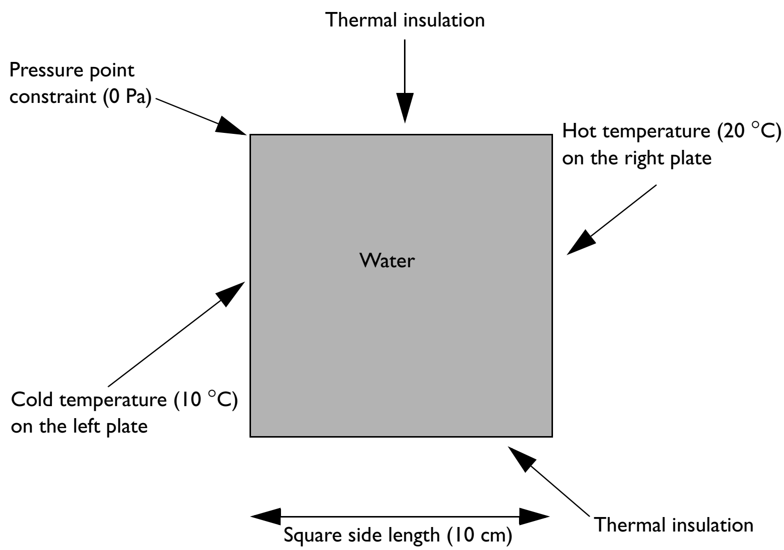


Figure 1: Domain geometry and boundary conditions for the 2D model (square cavity).

The 3D model (see Figure 2) extends the geometry to a cube. Compared to the 2D model, the front and back sides are additional boundaries that may influence the fluid behavior.

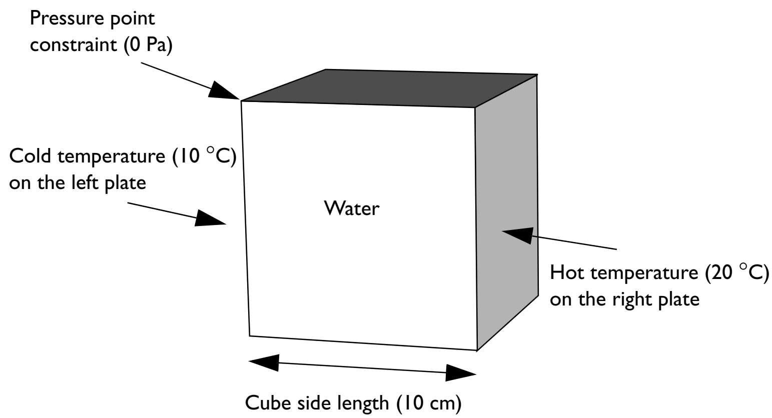


Figure 2: Domain geometry and boundary conditions for the 3D model (cubic cavity).

Both models calculate and compare the velocity field and the temperature field. The predefined Non-Isothermal Flow interface available in the Heat Transfer Module provides appropriate tools to fully couple the heat transfer and the fluid dynamics.

Model Definition

2D MODEL

Figure 1 illustrates the 2D model geometry. The fluid fills a square cavity with impermeable walls, so the fluid flows freely within the cavity but cannot leak out. The right and left edges of the cavity are maintained at high and low temperatures, respectively. The upper and lower boundaries are insulated. The temperature differential produces the density variation that drives the buoyant flow.

The compressible Navier-Stokes equation contains a buoyancy term on the right-hand side to account for the lifting force due to thermal expansion that causes the density variations:

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = -\nabla p + \nabla\cdot\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \frac{2}{3}\nabla(\mu(\nabla\cdot\mathbf{u})) + \rho\mathbf{g}$$

In this expression, the dependent variables for flow are the fluid velocity vector, $\mathbf{u}$, and the pressure, p. The constant $\mathbf{g}$ denotes the gravitational acceleration, ρ gives the temperature-dependent density, and μ is the temperature-dependent dynamic viscosity.

Because the model only contains information about the pressure gradient, it estimates the pressure field up to a constant. To define this constant, you arbitrarily fix the pressure at a point. No slip boundary conditions apply on all boundaries. The no slip condition results in zero velocity at the wall but does not set any constraint on p.

At steady-state, the heat balance for a fluid reduces to the following equation:

$$\rho C_p \mathbf{u} \cdot \nabla T - \nabla \cdot (k \nabla T) = 0$$

Here T represents the temperature, k denotes the thermal conductivity, and C_p is the specific heat capacity of the fluid.

The boundary conditions for the heat transfer interface are the fixed high and low temperatures on the vertical walls with insulation conditions elsewhere, as shown in Figure 1.

3D MODEL

Figure 2 shows the geometry and boundary conditions of the 3D model. The cavity is now a cube with high and low temperatures applied respectively at the right and left surfaces. The remaining boundaries (top, bottom, front, and back) are thermally insulated.

MODEL ANALYSIS

Before starting the simulations, it is recommended to estimate the flow regime. To this end, four indicators are presented: the Reynolds, Grashof, Rayleigh, and Prandtl numbers. They are calculated using the thermophysical properties of water listed in Table 1. The thermophysical properties are given at 290 K which is in the range of the temperatures observed in the model.

TABLE 1: THERMOPHYSICAL PROPERTIES FOR WATER AT 290 K

PARAMETER	DESCRIPTION	VALUE
ρ	Density	999 kg/m^3
μ	Dynamic viscosity	$1.08 \cdot 10^{-3}$ $N \cdot s/m^2$
k	Thermal conductivity	0.60 W/(m·K)
C_p	Heat capacity at constant pressure	4.18 kJ/(kg·K)
α_p	Coefficient of thermal expansion	$0.17 \cdot 10^{-3}$ K^{-1}

Prandtl Number

Definition The Prandtl number is the ratio of fluid viscosity to thermal diffusivity. It is defined by:

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

For air at 300 K, $\mathrm{Pr} = 0.71$ (see, for example, A.4 in Ref. 1) and for water at 300 K, $\mathrm{Pr} = 5.8$ (see, for example, A.6 in Ref. 1).

Boundary Layers The Prandtl number also indicates the relative thickness of the outer boundary layer, δ, and the thermal boundary layer, δ_T. In the present case, it is reasonable to estimate the ratio δ/δ_T by the relation (7.32 in Ref. 2)

$$\frac{\delta}{\delta_T} = \sqrt{\mathrm{Pr}} \tag{1}$$

The outer boundary layer is the distance from the wall to the region where the fluid stabilizes. It is different from the momentum boundary layer, δ_M, which measures the distance from the wall to the velocity peak.

Application in this Tutorial With the values given in Table 1, the Prandtl number for water at 290 K, is found to be of the order 1 or 10. According to Equation 1, δ and δ_T should then be of same order of magnitude.

Reynolds Number

Definition The Reynolds number estimates the ratio of inertial forces to viscous forces. It is defined by the formula

$$\mathrm{Re} = \frac{\rho U L}{\mu}$$

where U denotes the typical velocity and L the typical length.

At atmospheric pressure and at 300 K, air and water have the following properties (A.4 and A.6 in Ref. 1).

TABLE 2: THERMOPHYSICAL PROPERTIES OF AIR AND WATER AT 300 K AND ATMOSPHERIC PRESSURE

PARAMETER	DESCRIPTION	AIR	WATER
ρ	Density	1.16 kg/m^3	997 kg/m^3
μ	Dynamic viscosity	$1.85\cdot10^{-5}$ $N\cdot s/m^2$	$8.55\cdot10^{-4}$ $N\cdot s/m^2$

You can thus approximate the Reynolds number by:

$$\mathrm{Re}_{\mathrm{air}} \approx 6 \cdot 10^4 UL \qquad \mathrm{Re}_{\mathrm{water}} \approx 10^6 UL$$

In these relations, U has to be in meters per second and L in meters.

The Reynolds number can be rewritten as the velocity ratio

$$\mathrm{Re} = \frac{U}{\mu/(\rho L)}$$

which compares U to $\mu/(\rho L)$. The latter quantity is homogeneous to a velocity and can be seen as the typical velocity due to viscous forces.

Flow Regime The value of the Reynolds number is used to predict the flow regime. Generally, low values of Re correspond to laminar flow and high values to turbulent flow, with a critical value for the transition regime that depends on the geometry.

As an indication, Reynolds' experiments concerning the flow along a straight and smooth pipe showed that the transition regime in this case occurs when Re is between 2000 and 10^4 (see chapter 1.3 in Ref. 3).

Momentum Boundary Layer The momentum boundary layer thickness can be evaluated, using the Reynolds number, by (5.36 in Ref. 2)

$$\delta_{\mathrm{M}} \approx \frac{L}{\sqrt{\mathrm{Re}}} \tag{2}$$

Application in this Tutorial The typical length L of the model is equal to 10 cm so the Reynolds number is evaluated as

$$\mathrm{Re} \approx 10^5 U$$

where U is still unknown. Estimates of this typical velocity are provided later.

Grashof Number

Definition The Grashof number gives the ratio of buoyant to viscous forces. It is defined by

$$\mathrm{Gr} = \frac{\rho^2 g \alpha_p}{\mu^2} \Delta T L^3$$

where g is the gravity acceleration (equal to 9.81 m.s^{-2}) and ΔT is the typical temperature difference.

At atmospheric pressure and at 300 K, air and water have the following properties (A.4 and A.6 in Ref. 1).

TABLE 3: THERMOPHYSICAL PROPERTIES OF AIR AND WATER AT 300 K AND ATMOSPHERIC PRESSURE

Parameter	**Description**	**Air**	**Water**
ρ	Density	1.16 kg/m^3	997 kg/m^3
μ	Dynamic viscosity	$1.85\cdot10^{-5}$ N.s.m^{-2}	$8.55\cdot10^{-4}$ N·s/m^2
α_p	Coefficient of thermal expansion	$3.34\cdot10^{-3}$ K^{-1}	$2.76\cdot10^{-4}$ K^{-1}

The value of α_p for air was here obtained by the ideal gas approximation:

$$\alpha_p = \frac{1}{T}$$

You can then evaluate the Grashof number by:

$$\mathrm{Gr}_{\mathrm{air}} \approx 10^8 \Delta T L^3 \qquad \mathrm{Gr}_{\mathrm{water}} \approx 4 \cdot 10^9 \Delta T L^3$$

In these relations, ΔT is given in kelvin and L in meters.

The Grashof number can also be expressed as the velocity ratio

$$\mathrm{Gr} = \frac{U_0^2}{(\mu/(\rho L))^2}$$

where U_0 is defined by

$$U_0 = \sqrt{g\alpha_p \Delta T L} \tag{3}$$

This quantity can be considered as the typical velocity due to buoyancy forces.

Flow Regime When buoyancy forces are large compared to viscous forces, the regime is turbulent; otherwise it is laminar. The transition between these two regimes is indicated by the critical order of the Grashof number which is 10^9 (see Figure 7.7 in Ref. 2).

Application in this Tutorial In this tutorial, ΔT is equal to 10 K so the Grashof number is about 10^7 which indicates that a laminar regime is expected.

Table 4 provides the values of the quantities necessary to calculate U_0. This velocity is here of order 10 mm/s.

TABLE 4: THERMOPHYSICAL PROPERTIES OF WATER AT 290 K USED IN THE GRASHOF NUMBER

PARAMETER	DESCRIPTION	VALUE
g	Gravitational acceleration	9.81 m/s^2
α_p	Coefficient of thermal expansion	$0.17\cdot 10^{-3}$ K^{-1}

Rayleigh Number

Definition The Rayleigh number is another indicator of the regime. It is defined by

$$\mathrm{Ra} = \frac{\rho^2 g \alpha_p C_p}{\mu k}\Delta T L^3$$

so it is similar to the Grashof number except that it accounts for the thermal diffusivity, k, given by

$$\kappa = \frac{k}{\rho C_p}$$

Note: The Rayleigh number can be expressed in terms of the Prandtl and the Grashof numbers through the relation $\mathrm{Ra} = \mathrm{Pr}\,\mathrm{Gr}$.

At atmospheric pressure and at 300 K, you can use the approximations of Ra below for air and water (A.4 in Ref. 1)

$$\mathrm{Ra}_{\mathrm{air}} \approx 10^8 \Delta T L^3 \qquad \mathrm{Ra}_{\mathrm{water}} \approx 2\cdot 10^{10}\Delta T L^3$$

In these relations, ΔT is given in kelvin and L in meters.

Using Equation 3, the Rayleigh number can be rewritten as the velocity ratio

$$\mathrm{Ra} = \frac{U_0^2}{(\nu/L)(\kappa/L)}$$

where the ratio κ/L can be seen as a typical velocity due to thermal diffusion.

Flow Regime Like the Grashof number, a critical Rayleigh value indicates the transition between laminar and turbulent flow. For vertical plates, this limit is about 10^9 (9.23 in Ref.

1).

Typical Velocity Because the viscous forces limit the effects of buoyancy, U_0 may give an overestimated typical velocity. Another approach (see 7.25 in Ref. 2) is to use U_1 instead, defined by

$$U_1 = \frac{\kappa}{L}\sqrt{\mathrm{Ra}}$$

that is

$$U_1 = \frac{k}{\rho C_p L}\sqrt{\mathrm{Ra}} \tag{4}$$

or

$$U_1 = \frac{U_0}{\sqrt{\mathrm{Pr}}}$$

This should be a more accurate estimate of U because the fluid's thermal diffusivity and viscosity are used in the calculations. From now on, U_1 is the preferred estimate of U.

Thermal Boundary Layer The Rayleigh number can be used to estimate the thermal boundary layer thickness, δ_T. When Pr is of order 1 or greater, it is approximated by the formula (7.25c in Ref. 2)

$$\delta_\mathrm{T} \approx \frac{L}{\sqrt[4]{\mathrm{Ra}}} \tag{5}$$

Application in this Tutorial Here, Ra is of order 10^8. The laminar regime is confirmed but the Rayleigh number found is near the transition zone. The thermal boundary layer thickness is then found to be of order 1 mm and U_1 of order 10 mm/s.

Synthesis

To prepare the simulation, it is very useful to follow the steps below that give indications of what results to expect. It is important to remember that the quantities computed here are only order of magnitude estimates, which should not be considered with more than one significant digit.

First evaluate the Grashof and Rayleigh numbers. If they are significantly below the critical order of 10^9, the regime is laminar. In this case, Equation 3 or Equation 4 provide estimates of the typical velocity U that you can use to validate the model after performing the simulation.

According to Equation 1, the Prandtl number determines the relative thickness of the thermal boundary layer and the outer layer. Equation 2 and Equation 5 then provide orders of magnitude of the thicknesses. When defining the mesh, refinements have to be done at the boundary layers by, for instance, inserting three to five elements across the estimated thicknesses.

Here, Gr and Ra are 10^7 and 10^8, respectively, and thus below the critical value of 10^9 for vertical plates. A laminar regime is therefore expected but because these values are not significantly below 10^9, convergence is not straightforward. In this regime, the estimates U_0 and U_1 of the typical velocity are both of the order 10 mm/s.

For water at 290 K, Pr is about 10 so δ and δ_{T} are of same orders of magnitude. Here, δ_{T} is of the order 1 mm.

The Reynolds number calculated with U_1 is about 10^3, which confirms that the model is close to the transition regime. Using U_1 and Equation 2, the momentum boundary layer thickness δ_{M} is found to be about 1 mm.

2D MODEL

Figure 3 shows the velocity distribution in the square cavity.

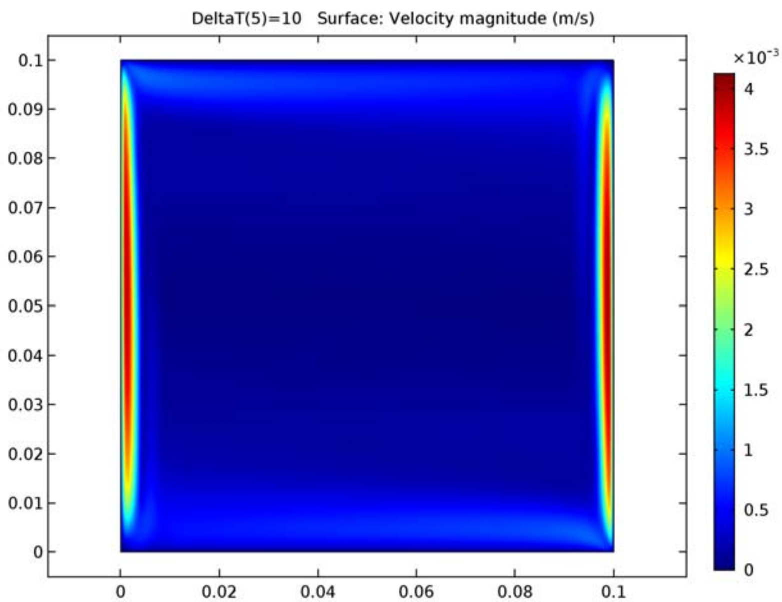


Figure 3: Velocity magnitude for the 2D model.

The regions with faster velocities are located at the lateral boundaries. The maximum velocity is 4.20 mm/s which is in agreement with the estimated typical velocity U_1 of the order 10 mm/s. According to Figure 4, the momentum boundary layer thickness is of order 1 mm, as calculated before.

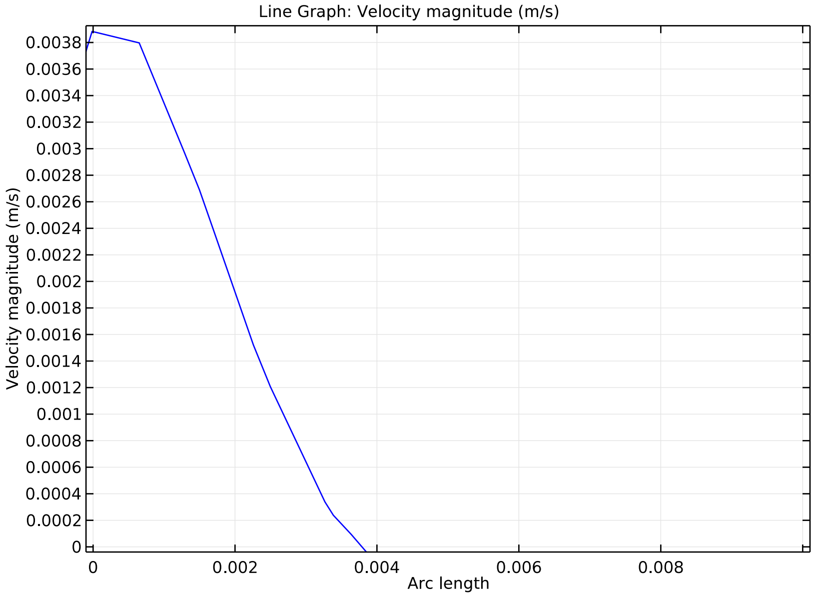


Figure 4: Velocity profile at the left boundary.

Figure 5 shows the temperature field (surface) and velocity field (arrows) of the 2D model.

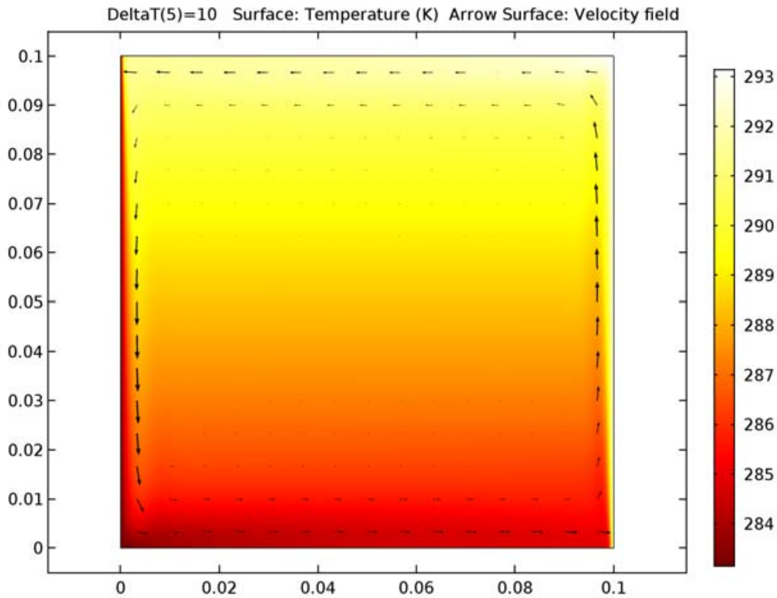


Figure 5: Temperature field (surface plot) and velocity (arrows) for the 2D model.

A large convective cell occupies the whole square. The fluid flow follows the boundaries. As seen in Figure 3, it is faster at the vertical plates where the highest variations of temperature are located. The thermal boundary layer is of the order 1 mm according to Figure 6, which is in agreement with the estimate provided by Equation 5. The outer layer is slightly thicker than the boundary layer.

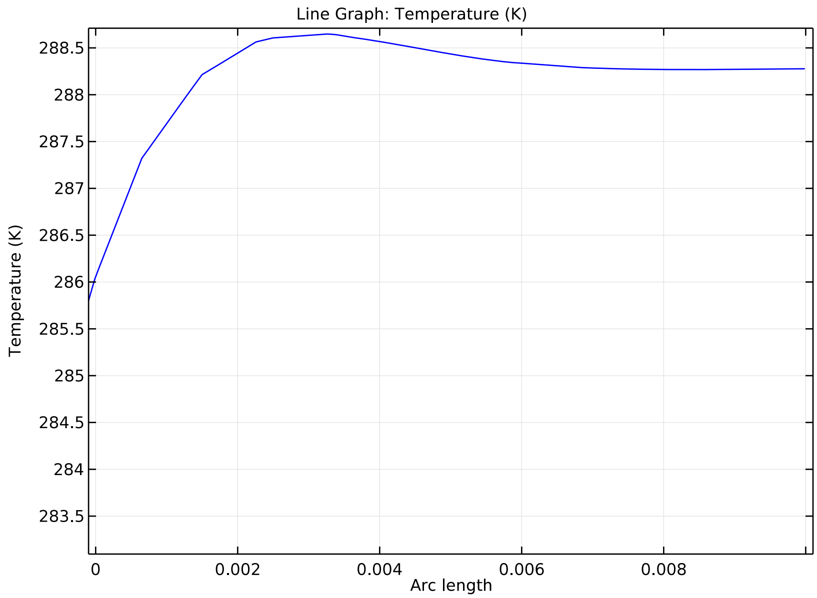


Figure 6: Temperature profile at the left boundary.

Figure 7 shows the correlation between density and temperature.

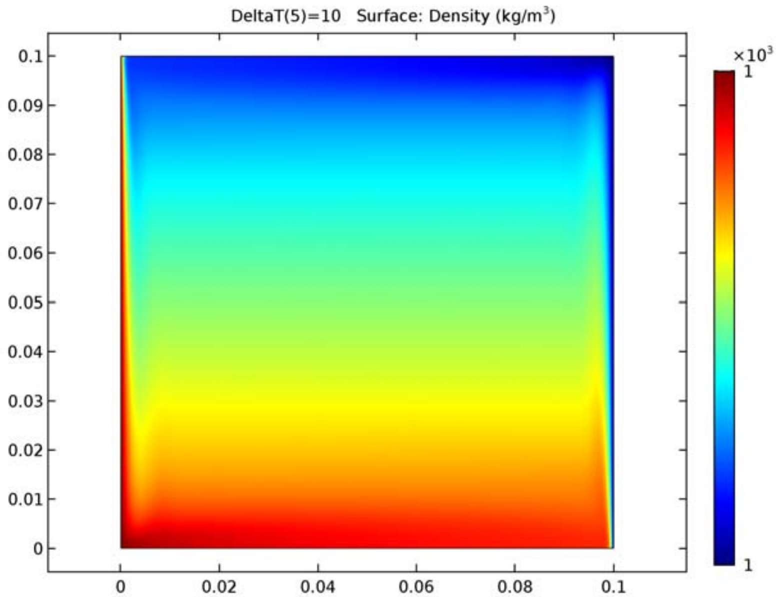


Figure 7: Water density for the 2D model.

As expected, ρ has the same profile as the temperature field. High variations are located at the boundaries, especially at lateral walls, and are responsible for the free convection. The variations are smoother at the center.

3D MODEL

Figure 8 illustrates the velocity plot parallel to the heated plates.

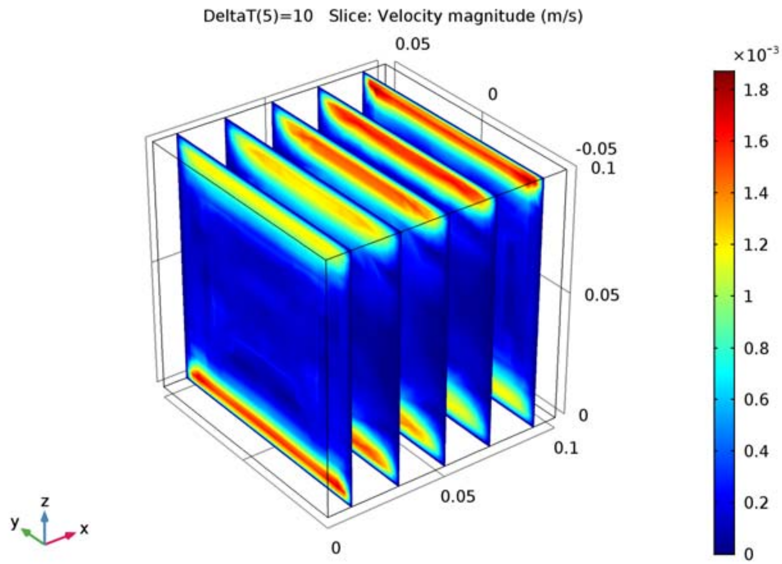


Figure 8: Velocity magnitude field for the 3D model, slices parallel to the heated plates.

A second velocity magnitude field is shown in Figure 9. The plot is close to what was obtained in 2D in Figure 3.

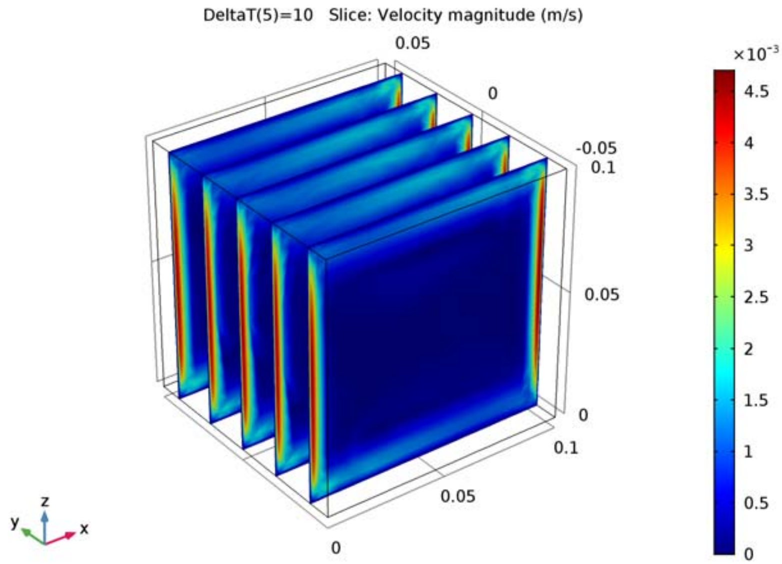


Figure 9: Velocity magnitude field for the 3D model, slices perpendicular to the heated plates.

In Figure 10, velocity arrows are plotted on temperature surface at the middle vertical plane parallel to the plates.

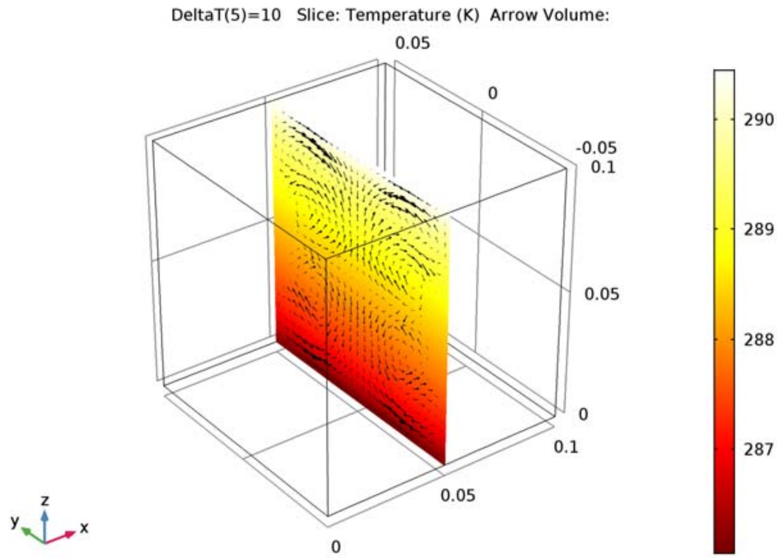
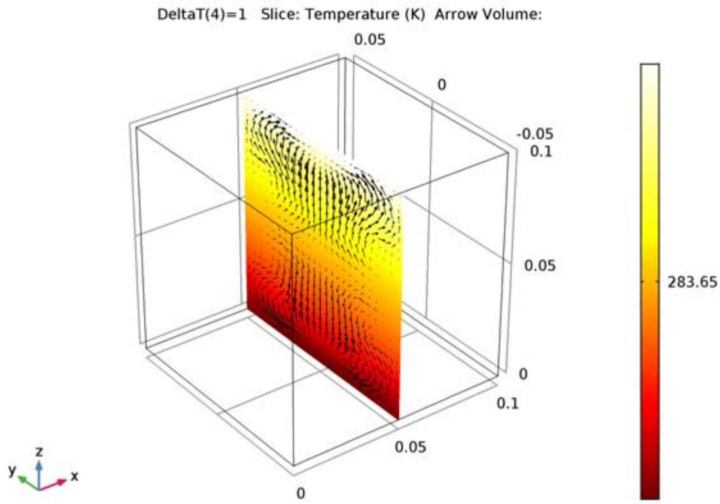


Figure 10: Temperature (surface plot) and velocity (arrows) fields in the cubic cavity, for a temperature difference of 10 K between the vertical plates.

New small convective cells appear on the vertical planes perpendicular to the plates at the four corners. They are more visible at lower Gr values, that is, far from the transition regime. In Figure 11, the temperature difference between the vertical plates is reduced to 1 K and 0.1 K to decrease the Grashof number to 10^5 and 10^6.

Observe the bigger cells at the four corners of the plane.



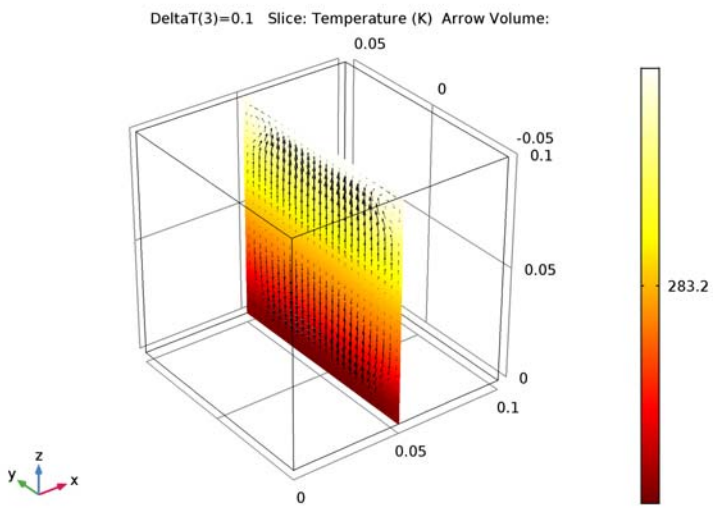


Figure 11: Temperature (surface plot) and velocity (arrows) fields in the cubic cavity, with, for a temperature difference of 1 K (top) and 0.1 K (bottom) between the vertical plates.

Notes About the COMSOL Implementation

The material properties for water are available in the Material Library. The density and dynamic viscosity are functions of the temperature.

At high Gr values, using a good initial condition becomes important in order to achieve convergence. Moreover, a well-tuned mesh is needed to capture the solution, especially the temperature and velocity changes near the walls. Use the Stationary study step's continuation option with ΔT as the continuation parameter to get a solver sequence that uses previous solutions to estimate the initial condition. For this tutorial, it is appropriate to ramp up ΔT from 10^{-3} K to 10 K, which corresponds to a Grashof number range of 10^3–10^7. At $\mathrm{Gr} = 10^3$, the model is easy to solve. The regime is dominated by conduction and viscous effects. At $\mathrm{Gr} = 10^7$, the model becomes more difficult to solve. The regime is greatly influenced by convection and buoyancy.

To get a well-tuned mesh when Gr reaches 10^7, the element size near the prescribed temperature boundaries has to be smaller than the momentum and thermal boundary layer thicknesses, which are of order 1 mm according to Equation 2 and Equation 5. It is recommended to have three to five elements across the layers when using P1 elements (the default setting for fluid flows).

References

1. F.P. Incropera, D.P. DeWitt, T.L. Bergman, and A.S. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th ed., John Wiley & Sons, 2006.

2. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1985.

3. P. A. Davidson, *Turbulence: An Introduction for Scientists and Engineers*, Oxford University Press, 2004.

Application Library path: `Heat_Transfer_Module/Tutorials,`
`_Forced_and_Natural_Convection/buoyancy_water`

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**.
2 In the **Select Physics** tree, select **Fluid Flow>Non-Isothermal Flow>Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
L	10[cm]	0.1 m	Square side length
DeltaT	10[K]	10 K	Temperature difference
Tc	283.15[K]	283.15 K	Low temperature
Th	Tc+DeltaT	293.15 K	High temperature
rho	999[kg/m^3]	999 kg/m³	Density
mu	1.08e-3[N*s/m^2]	0.00108 N·s/(m·m)	Dynamic viscosity
k	0.60[W/(m*K)]	0.6 W/(m·K)	Thermal conductivity
Cp	4.18[kJ/(kg*K)]	4180 J/(kg·K)	Heat capacity
alpha	0.17e-3[1/K]	1.7E-4 1/K	Coefficient of thermal expansion
U0	sqrt(g_const*alpha*DeltaT*L)	0.040831 m/s	Typical velocity due to buoyancy
U1	U0/sqrt(Pr)	0.014885 m/s	Typical velocity estimation
Pr	mu*Cp/k	7.524	Prandtl number
Gr	(U0*rho*L/mu)^2	1.4264E7	Grashof number
Ra	Pr*Gr	1.0733E8	Rayleigh number
Re0	rho*U0*L/mu	3776.8	Reynolds number approximation with U0
Re1	rho*U1*L/mu	1376.9	Reynolds number approximation with U1
eps_t	L/(Ra)^0.25	9.8248E-4 m	Thermal boundary layer thickness
eps_m	L/sqrt(Re1)	0.0026949 m	Momentum boundary layer thickness

The Grashof and Rayleigh numbers should be less than 10^9, indicating that a laminar regime is expected.

GEOMETRY 1

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

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

3 In the **Side length** text field, type L.

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

ADD MATERIAL

1 On 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)

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

LAMINAR FLOW (SPF)

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

2 In the **Settings** window for Laminar Flow, locate the **Physical Model** section.

3 Select the **Include gravity** check box.

4 Find the **Reference values** subsection. Specify the $\mathbf{r}_{\mathrm{ref}}$ vector as

0	x
0.1	y

Pressure Point Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 2 only.

Fixing the pressure at an arbitrary point is necessary to define a well-posed model.

HEAT TRANSFER IN FLUIDS (HT)

Define the initial temperature as the mean value between the high and low temperature values.

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, type `(Tc+Th)/2` in the T text field.

3 In the **Model Builder** window, click **Heat Transfer in Fluids (ht)**.

Temperature 1

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

2 Select Boundary 1 only.

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

4 In the T_0 text field, type `Tc`.

Temperature 2

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

2 Select Boundary 4 only.

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

4 In the T_0 text field, type `Th`.

Now, modify the default mesh size settings to ensure that the mesh satisfies the criterion discussed in the Introduction section.

MESH 1

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

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

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

STUDY 1

Because the Grashof number is near the critical value of around 10^9, the model is highly nonlinear. To achieve convergence, use continuation to ramp up the temperature difference value from 10^{-3} K to 10 K, which corresponds to a Grashof number from 10^3 to 10^7.

Step 1: Stationary

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

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

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

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
DeltaT	`1e-3 1e-2 1e-1 1 10`	K

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

The pseudo stepping time option is generally useful to help the convergence of a stationary flow model. However, a continuation approach is already used here. In this precise model, disabling the pseudo time stepping option improves the convergence. Follow the instructions below to do so.

LAMINAR FLOW (SPF)

On the **Physics** toolbar, click **Heat Transfer in Fluids (ht)** and choose **Laminar Flow (spf)**.

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

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

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

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Velocity (spf)

The first default plot group shows the velocity magnitude as in Figure 3. Notice the high velocities near the lateral walls due to buoyancy effects.

Temperature (ht)

The third default plot shows the temperature distribution. Add arrows of the velocity field to see the correlations between velocity and temperature, as in Figure 5.

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

2 On the **Temperature (ht)** toolbar, click **Plot**.

Arrow Surface 1

1 Right-click **Results>Temperature (ht)** and choose **Arrow Surface**.

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

3 From the **Color** list, choose **Black**.

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

To reproduce the plot of Figure 7, follow the steps below.

2D Plot Group 5

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

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

Surface 1

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

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Material properties>spf.rho - Density**.

3 On the **Density** toolbar, click **Plot**.

In the following steps, the temperature and velocity profiles are plotted near the left boundary in order to estimate the boundary layer thicknesses of the solution.

Cut Line 2D 1

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

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

3 In row **Point 1**, set **y** to `5[cm]`.

4 In row **Point 2**, set **x** to `1[cm]`, and **y** to `5[cm]`.

5 Click **Plot**.

1D Plot Group 6

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

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

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

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

Line Graph 1

1 On the **Temperature at Boundary Layer** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Heat Transfer in Fluids>Temperature>T - Temperature**.

3 On the **Temperature at Boundary Layer** toolbar, click **Plot**.

The thermal boundary layer is around 3 mm.

1D Plot Group 7

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

2 In the **Settings** window for 1D Plot Group, type `Velocity at Boundary Layer` in the **Label** text field.

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

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

Line Graph 1

1 On the **Velocity at Boundary Layer** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude**.

3 On the **Velocity at Boundary Layer** toolbar, click **Plot**.

The momentum boundary layer is around 1 mm and the outer layer between 5 mm and 10 mm.

Now create the 3D version of the model.

ROOT

On the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Recently Used>Laminar Flow**.

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

LAMINAR FLOW 2 (SPF2)

On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Stationary**.

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

STUDY 2

Step 1: Stationary

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

GEOMETRY 2

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for Block, locate the **Size and Shape** section.
3 In the **Width** text field, type `L`.
4 In the **Depth** text field, type `L/2`.
5 In the **Height** text field, type `L`.
6 On the **Geometry** toolbar, click **Build All**.

ADD MATERIAL

1 On 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 (mat2)

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

LAMINAR FLOW 2 (SPF2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Laminar Flow 2 (spf2)**.
2 In the **Settings** window for Laminar Flow, locate the **Physical Model** section.
3 Select the **Include gravity** check box.
4 Find the **Reference values** subsection. Specify the $\mathbf{r}_{\mathrm{ref}}$ vector as

0	x
0	y
0.1	z

Pressure Point Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 4 only.

Symmetry 1

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

2 Select Boundary 2 only.

HEAT TRANSFER IN FLUIDS 2 (HT2)

Initial Values 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Heat Transfer in Fluids 2 (ht2)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `(Tc+Th)/2` in the $T2$ text field.

3 In the **Model Builder** window, click **Heat Transfer in Fluids 2 (ht2)**.

Temperature 1

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

2 Select Boundary 1 only.

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

4 In the T_0 text field, type `Tc`.

Temperature 2

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

2 Select Boundary 6 only.

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

4 In the T_0 text field, type `Th`.

Symmetry 1

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

2 Select Boundary 2 only.

LAMINAR FLOW 2 (SPF2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Laminar Flow 2 (spf2)**.

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

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

MESH 2

To obtain reliable results in a reasonable computational time, create a structured mesh according to the steps below.

1 On the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

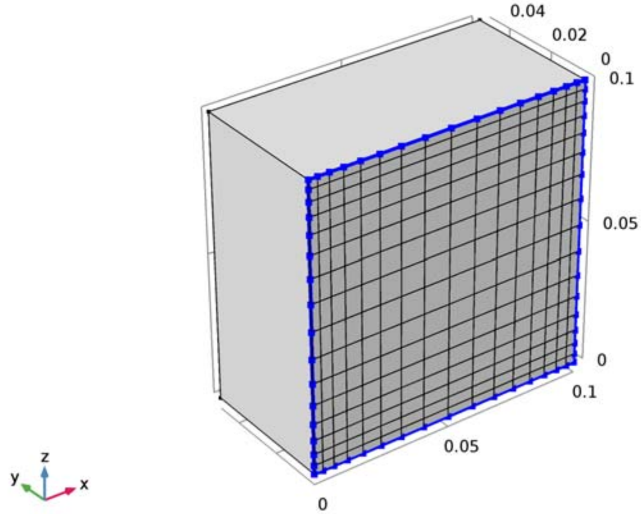
Mapped 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Mapped 1**.
2 Select Boundary 2 only.
3 On the **Mesh** toolbar, click **Distribution**.

Distribution 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2>Mapped 1** click **Distribution 1**.
2 Select Edges 1, 3, 5, and 9 only.
3 In the **Settings** window for Distribution, locate the **Distribution** section.
4 From the **Distribution properties** list, choose **Predefined distribution type**.
5 In the **Number of elements** text field, type 16.
6 In the **Element ratio** text field, type 3.
7 Select the **Symmetric distribution** check box.

8 Click **Build Selected**.



The front face mesh has smaller elements near the edges because large variations in velocity and temperature are expected there.

Now extend the front mesh to the remaining structure.

9 On the **Mesh** toolbar, click **Swept**.

Swept 1

Click **Distribution**.

Distribution 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2>Swept 1** click **Distribution 1**.

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

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 8.

5 In the **Element ratio** text field, type 3.

6 Select the **Reverse direction** check box.

To resolve the boundary layers, use a **Boundary Layers** feature to generate smaller mesh elements near the walls. The thermal boundary layer for the temperature difference of

10 K is approximately 1 mm (see the parameter `eps_t` defined previously). Use this value to define the thickness of the boundary layers.

7 On the **Mesh** toolbar, click **Boundary Layers**.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 Select Boundaries 1 and 3–6 only.

3 In the **Settings** window for Boundary Layer Properties, locate the **Boundary Layer Properties** section.

4 In the **Number of boundary layers** text field, type `5`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `eps_t/5`.

7 Click **Build All**.

STUDY 2

Step 1: Stationary

1 In the **Settings** window for Stationary, locate the **Study Extensions** section.

2 Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
DeltaT	1e-3 1e-2 1e-1 1 10	K

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Laminar Flow (spf)** and **Heat Transfer in Fluids (ht)**.

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

RESULTS

Velocity (spf2)

This default plot group shows the fluid velocity magnitude in only half of the cube. To plot the other half, proceed as follows.

Mirror 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

2 In the **Settings** window for Mirror 3D, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

A new data set containing mirror values is now created. Return to the velocity plot to use this data set.

Velocity (spf2)

1 In the **Model Builder** window, under **Results** click **Velocity (spf2)**.

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

3 From the **Data set** list, choose **Mirror 3D 1**.

4 On the **Velocity (spf2)** toolbar, click **Plot**.

Temperature (ht2)

This default plot group shows the temperature distribution. The mirror data set created previously can be reused here to plot the entire cube.

3D Plot Group 12

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

2 In the **Settings** window for 3D Plot Group, type `Velocity, Front Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Mirror 3D 1**.

Slice 1

1 Right-click **Velocity, Front Plane** and choose **Slice**.

2 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 2>Laminar Flow 2>Velocity and pressure>spf2.U - Velocity magnitude**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

4 On the **Velocity, Front Plane** toolbar, click **Plot**.

This slice view shows the velocity magnitude in the same plane as in the 2D model (Figure 9).

Next, plot arrows of the tangential velocity field in the vertical plane parallel to the plates to reproduce Figure 10.

3D Plot Group 13

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

2 In the **Settings** window for 3D Plot Group, type `Temperature, 10 K Offset` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Mirror 3D 1**.

Slice 1

1 Right-click **Temperature, 10 K Offset** and choose **Slice**.

2 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 2>Heat Transfer in Fluids 2>Temperature>T2 - Temperature**.

3 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

5 On the **Temperature, 10 K Offset** toolbar, click **Plot**.

Temperature, 10 K Offset

In the **Model Builder** window, under **Results** right-click **Temperature, 10 K Offset** and choose **Arrow Volume**.

Arrow Volume 1

1 In the **Settings** window for Arrow Volume, locate the **Expression** section.

2 In the **x component** text field, type `0`.

3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `1`.

4 Find the **y grid points** subsection. In the **Points** text field, type `25`.

5 Find the **z grid points** subsection. In the **Points** text field, type `25`.

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

7 On the **Temperature, 10 K Offset** toolbar, click **Plot**.

Temperature, 10 K Offset

The arrows follow convective cells at the four corners for a temperature difference of 10 K. Follow the steps below to reproduce Figure 11 and to see these cells when the temperature difference is reduced to 1 K and 0.1 K.

Temperature, 10 K Offset 1

1 Right-click **Temperature, 10 K Offset** and choose **Duplicate**.

2 In the **Settings** window for 3D Plot Group, type `Temperature, 1 K Offset` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (DeltaT (K))** list, choose **1**.

4 On the **Temperature, 1 K Offset** toolbar, click **Plot**.

Temperature, 1 K Offset 1

1 Right-click **Results>Temperature, 1 K Offset** and choose **Duplicate**.
2 In the **Settings** window for 3D Plot Group, type `Temperature, 0.1 K Offset` in the **Label** text field.
3 Locate the **Data** section. From the **Parameter value (DeltaT (K))** list, choose **0.1**.
4 On the **Temperature, 0.1 K Offset** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Anisotropic Heat Transfer through Woven Carbon Fibers

Introduction

Composite materials such as carbon-fiber-reinforced polymer (CFRP) have outstanding properties. It is a lightweight material with high stiffness and high temperature tolerance and therefore used in aerospace industry, civil engineering and also for high-end sports goods.

Carbon crystals form flat ribbons. Large numbers of these ribbons are bundled together and woven in different structures as required by the application area. The bundles have anisotropic material properties. For thermal properties this means that the thermal conductivity along the fiber axis is much higher than perpendicular to it.

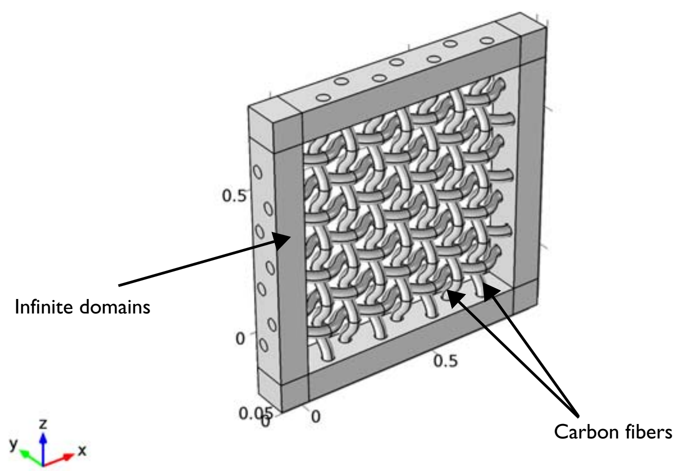


Figure 1: Model geometry: fibers embedded in an epoxy matrix (hidden) with infinite element domain.

In COMSOL Multiphysics, implementing anisotropic properties is straightforward in the global coordinate system, which is set by default. However, in the present case, an anisotropic thermal conductivity needs to be defined along woven fibers where the global coordinate system is inapplicable. The curvilinear coordinate system provides the possibility to create a coordinate system following the curves of a geometry in which anisotropic material properties or anisotropic physics can be defined.

This tutorial shows how to use the curvilinear coordinates interface and how to apply it to define anisotropic thermal conductivity.

Model Definition

The model represents a cutout of a carbon-fiber-reinforced polymer. The geometry used in this case is shown in Figure 1. The fiber bundles have circular cross-sections and are embedded in a matrix made of epoxy.

The "infinite domains" truncate the geometry to model a few fibers only. With the Heat Transfer Module you can assign them as Infinite Element Domains and thus suppress boundary effects. Without the Heat Transfer Module the boundary conditions at the outer side affects the solution-in this case the maximum temperature. Increase the number of fibers to reduce these effects.

MATERIAL PROPERTIES

The material properties are summarized in Table 1.

TABLE 1: MATERIAL PROPERTIES

MATERIAL PROPERTY	EPOXY	CARBON (CORE)	CARBON (INFINITE DOMAIN)
Thermal conductivity	0.2 W/(m·K)	{60,4,4} W/(m·K)	60 W/(m·K)
Density	1200 kg/m³	1500 kg/m³	1500 kg/m³
Heat capacity at constant pressure	1000 J/(kg·K)	1000 J/(kg·K)	1000 J/(kg·K)

Note the syntax of the thermal conductivity for carbon (core). In the general case of an anisotropic thermal conductivity, it is a second order tensor. In the present case, the tensor is diagonal.

$$k = \begin{pmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{pmatrix} = \begin{pmatrix} 60 & 0 & 0 \\ 0 & 4 & 0 \\ 0 & 0 & 4 \end{pmatrix}$$

Note that the conductivity is high in the fibers direction and low in perpendicular direction. The coordinate system used for k must then provide an x-component following the shape of the fibers. The Curvilinear Coordinates interface provides appropriate tools to create such a base vector system.

CURVILINEAR COORDINATES

Three predefined methods and a user-defined method are available to set up a curvilinear coordinate system. Further details can be found in the *COMSOL Multiphysics Reference Guide* in the "Curvilinear Coordinates interface" section. Here you use the diffusion method which solves Laplace's equation resulting in a scalar potential. It is the same as solving the stationary heat transfer equation with temperature boundary conditions resulting in a temperature gradient and forming the first base vector of the new coordinate system. The second base vector is specified manually and the cross-product of both forms the third base vector.

Figure 2 shows the base vector system for a single fiber.

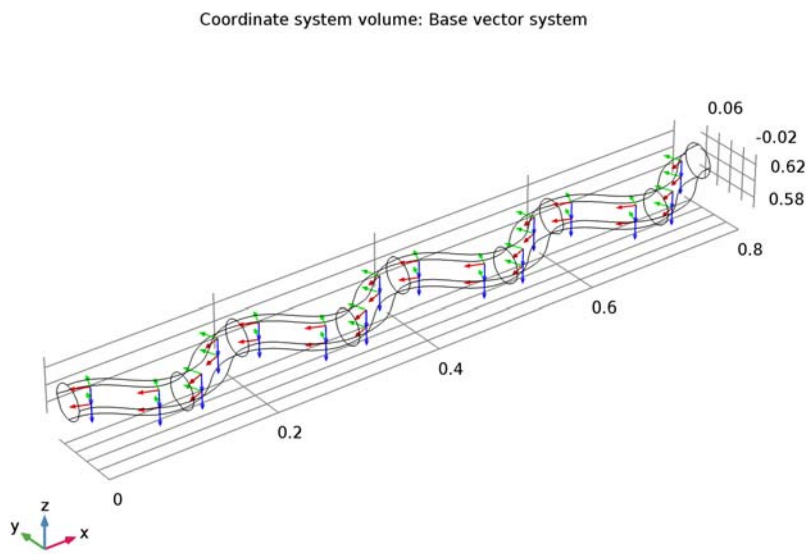


Figure 2: Curvilinear coordinate system from diffusion method.

Alternatively the Flow Method is available, which results in a vector potential. This is equivalent to solving Stokes flow (also known as creeping flow) where the obtained velocity field forms the first base vector. The third option is to choose the Elasticity Method for solving an eigenvalue problem.

If the **Create base vector system** option is selected, the new curvilinear system is available as input for the **Coordinate System Selection** drop-down menu and thus providing new (x, y, z) coordinates.

BOUNDARY CONDITIONS

For the curvilinear coordinates interface, the inlet and outlet boundaries define the direction of the first base vector. The heat transfer analogy consists in setting a high temperature at the inlet and a low temperature at the outlet. All other boundaries are thermally insulated walls.

For the heat transfer interface, a constant temperature boundary condition is set at the outermost walls. A boundary heat source described with a Gaussian pulse in the center of the geometry is applied and a convective cooling boundary condition on both sides.

INFINITE ELEMENTS

To truncate the geometry the Infinite Element Domain feature can be used. Boundary conditions applied to these elements can be imagined as boundary conditions at an infinite distance of the modeling domain. So it does not affect the solution of this particular problem. This works by scaling the width of the domain to be much larger than the original geometry.

Results and Discussion

From the **Curvilinear Coordinates** interface a new coordinate system is obtained as shown in Figure 1. The temperature distribution on the surface shows a high temperature at the center where the maximum of the Gaussian function is located and decreases with increasing distance from the center. The temperature drop to 293 K as specified in the boundary conditions occurs mainly in the Infinite Element Domains.

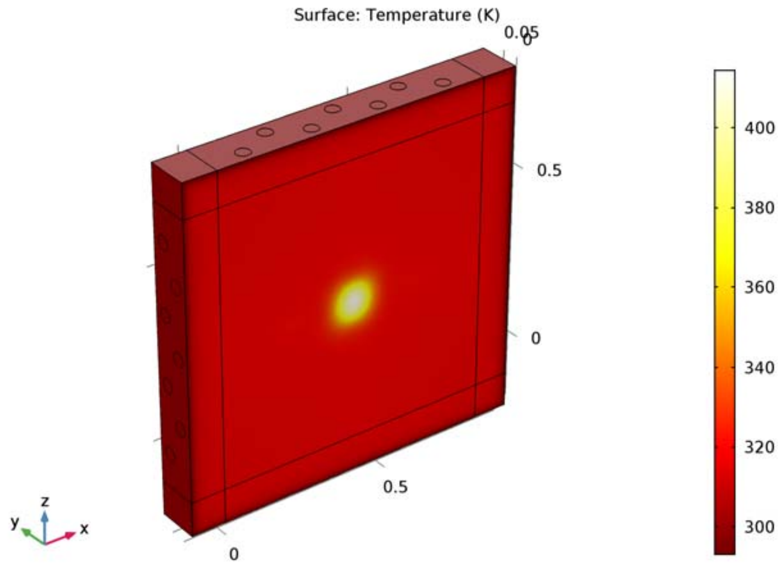


Figure 3: Temperature distribution on the surface.

Figure 4 shows clearly that the heat spreads preferentially along the fiber axis.

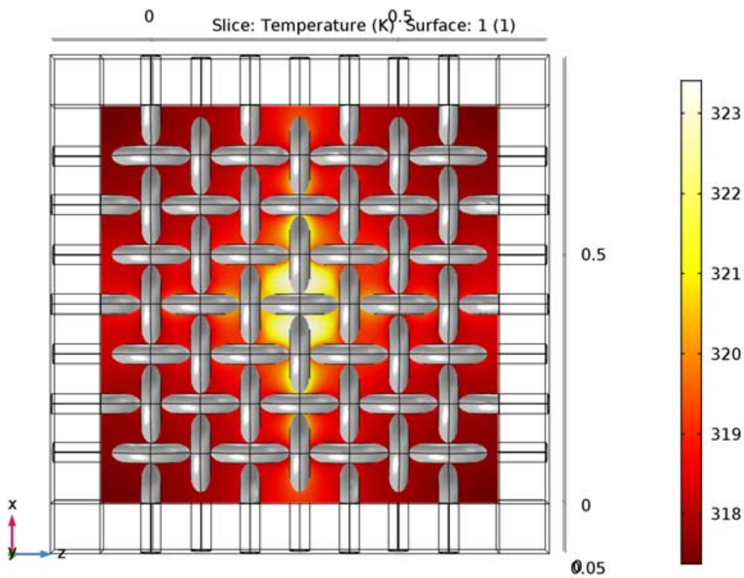


Figure 4: Temperature at the center plane and fiber structure (gray).

Notes About the COMSOL Implementation

This tutorial demonstrates how to use the **Curvilinear Coordinates** interface for defining anisotropic thermal conductivity. Hence, the instructions focus on this part and start with loading the file `carbon_fiber_geom.mph`. The steps needed to create this file are quite complex. This document does not go into details but provides a short summary instead.

The geometry sequence calls geometry subsequences depending on the global parameter q. The subsequences define different cross-sections and can be found under the **Global Definitions** node. Call the elliptical cross-section with $q = 1$ and the rectangular cross-section with $q = 2$.

All subsequent geometry features are based on these subsequences. Inside some of the features, a selection of geometric entities is created automatically by selecting the **Create Selections** option. Instead of selecting objects manually, these selections are used as input in the following geometry node. This approach ensures that all geometry operations adapt and produce the desired geometry automatically, even if a geometry parameter changes.

Selections are also used on the finalized geometry to ensure that physical properties are assigned to the intended entities. These selections are defined under **Component 1>** **Definitions**. They are used to automatically set up boundary and domain conditions as well as the mesh. The resulting model is consistent for any choice of parameters. The extra time needed to set up this kind of geometry sequence and to define selections is regained through an accelerated physics modeling and meshing process.

Application Library path: `Heat_Transfer_Module/Tutorials,_Conduction/carbon_fibers_infinite_elements`

Modeling Instructions

ROOT

Start with loading the model file that contains the geometry and selections used throughout the modeling process.

1. From the **File** menu, choose **Open**.
2. Browse to the application's Application Libraries folder and double-click the file `carbon_fibers_geom.mph`.
3. Click the **Zoom Extents** button on the **Graphics** toolbar.

COMPONENT 1 (COMP1)

Add the **Curvilinear Coordinates** interface for the fibers.

ADD PHYSICS

1. On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Mathematics>Curvilinear Coordinates (cc)**.
4. Click **Add to Component** in the window toolbar.

CURVILINEAR COORDINATES (CC)

1. On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.
2. In the **Model Builder** window, under **Component 1 (comp1)** click **Curvilinear Coordinates (cc)**.
3. In the **Settings** window for Curvilinear Coordinates, locate the **Domain Selection** section.

4 From the **Selection** list, choose **Fibers (Core)**.

5 Locate the **Settings** section. Select the **Create base vector system** check box.

According to the **Curvilinear Coordinates** section, the second basis vector is specified manually. The *y*-direction feels natural.

Coordinate System Settings 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Curvilinear Coordinates (cc)** click **Coordinate System Settings 1**.

2 In the **Settings** window for Coordinate System Settings, locate the **Settings** section.

3 From the **Second basis vector** list, choose **y-axis**.

Diffusion Method 1

1 On the **Physics** toolbar, click **Domains** and choose **Diffusion Method**.

   Wall is the default boundary condition where the normal component of the vector field is zero. The direction of the first basis vector is specified with inlet and outlet boundary conditions.

2 In the **Settings** window for Diffusion Method, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Fibers (Core)**.

Inlet 1

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

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

3 From the **Selection** list, choose **Inlets**.

Diffusion Method 1

In the **Model Builder** window, under **Component 1 (comp1)>Curvilinear Coordinates (cc)** click **Diffusion Method 1**.

Outlet 1

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

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

3 From the **Selection** list, choose **Outlets**.

MESH 1

Now, build a suitable mesh manually. Start with meshing the fibers.

1 On the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.
2 In the **Settings** window for Free Triangular, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Inlets**.
4 On the **Mesh** toolbar, click **Distribution**.

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Free Triangular 1** click **Distribution 1**.
2 In the **Settings** window for Distribution, locate the **Edge Selection** section.
3 From the **Selection** list, choose **Inlet Edges**.
4 Locate the **Distribution** section. In the **Number of elements** text field, type 2.
5 Click **Build Selected**.
6 On the **Mesh** toolbar, click **Swept**.

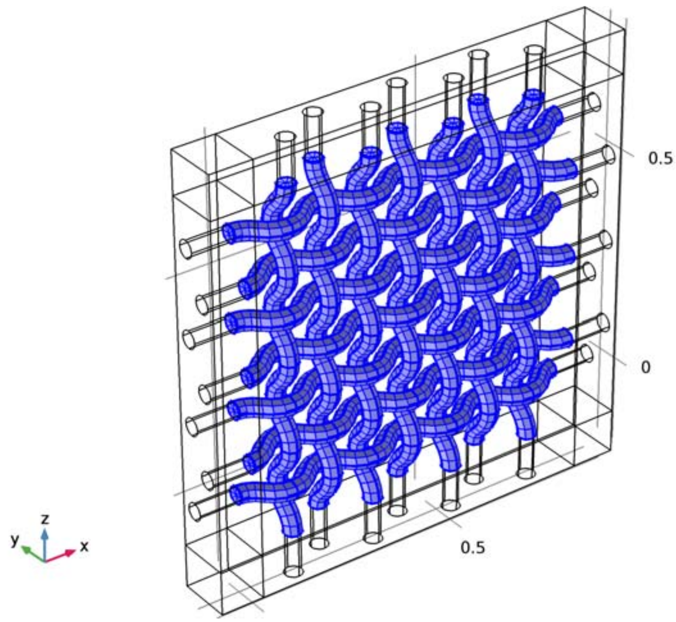
Swept 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Swept 1**.
2 In the **Settings** window for Swept, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Fibers (Core)**.
5 On the **Mesh** toolbar, click **Distribution**.

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Swept 1** click **Distribution 1**.
2 In the **Settings** window for Distribution, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Fibers (Core)**.
4 Locate the **Distribution** section. In the **Number of elements** text field, type 6.

5 Click **Build Selected**.



Convert now the surface elements into triangles in order to use a free tetrahedral mesh for the epoxy domain.

6 On the **Mesh** toolbar, click **Modify** and choose **Elements>Convert**.

Convert 1

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

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

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

4 From the **Selection** list, choose **Fiber Walls**.

5 Click **Build Selected**.

Then, use a triangular mesh at the remaining core boundaries and extrude the resulting boundary mesh into the infinite element domains.

6 On the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Free Triangular 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 2**.

2 In the **Settings** window for Free Triangular, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Epoxy Boundaries (Core)**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 2** and choose **Size**.

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

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

4 Click **Build Selected**.

5 On the **Mesh** toolbar, click **Swept**.

Swept 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Swept 2**.

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

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

4 From the **Selection** list, choose **Infinite Element Domains**.

5 On the **Mesh** toolbar, click **Distribution**.

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Swept 2** click **Distribution 1**.

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

3 In the **Number of elements** text field, type 3.

4 Click **Build Selected**.

5 On the **Mesh** toolbar, click **Free Tetrahedral**.

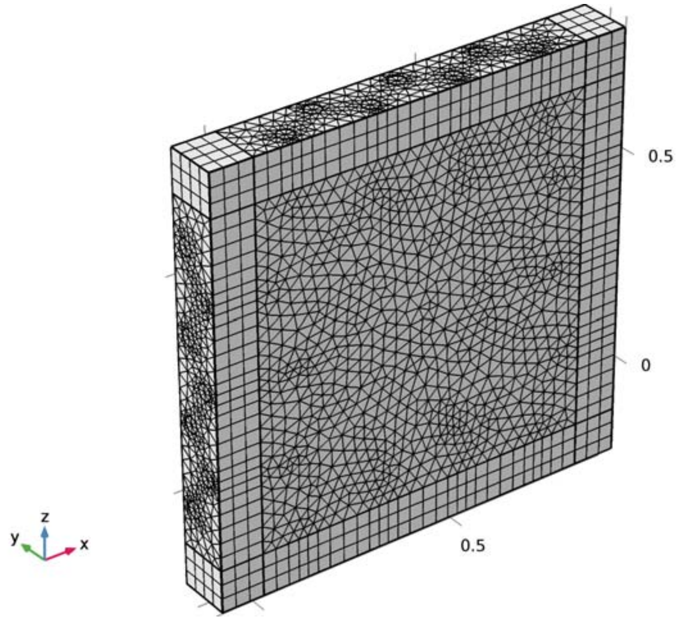
Free Tetrahedral 1

Mesh the remaining parts with a free tetrahedral mesh.

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

2 In the **Settings** window for Free Tetrahedral, click **Build All**.

The final mesh looks like this.



ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

Add a stationary study to compute the new coordinate system with the **Diffusion Method**.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

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

STUDY 1

Step 1: Stationary

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

2 On the **Home** toolbar, click **Compute**.

RESULTS

Vector Field (cc)

The default plots show the coordinate system with volume arrows and a streamlines for the vector field. To create the plot shown in Figure 3, add a selection to the data set. The plot group will then use this subset of the whole geometry only.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.
2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `20, 34, 54, 68, 88, 102, 122, 136` in the **Selection** text field.
6 Click **OK**.

Coordinate System Volume 1

1 In the **Model Builder** window, expand the **Coordinate system (cc)** node, then click **Coordinate System Volume 1**.
2 In the **Settings** window for Coordinate System Volume, locate the **Positioning** section.
3 Find the **x grid points** subsection. In the **Points** text field, type `16`.
4 Find the **y grid points** subsection. In the **Points** text field, type `2`.
5 Find the **z grid points** subsection. In the **Points** text field, type `2`.
6 On the **Coordinate system (cc)** toolbar, click **Plot**.
7 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Now, add the **Heat Transfer in Solids** interface to the component.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

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

HEAT TRANSFER IN SOLIDS (HT)

1 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

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

Infinite Element Domain 1 (ie1)

On the **Definitions** toolbar, click **Infinite Element Domain**.

DEFINITIONS

Infinite Element Domain 1 (ie1)

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

2 From the **Selection** list, choose **Infinite Element Domains**.

In order to apply a heat source right in the center of the model, the **Mass Properties** feature is used. It computes the center of mass, which is automatically the center of the geometry.

3 In the **Model Builder** window, right-click **Definitions** and choose **Mass Properties**.

4 In the **Settings** window for Mass Properties, locate the **Source Selection** section.

5 From the **Selection** list, choose **Fibers (Core)**.

Define a heat source via a local variable, which is defined on the boundary only. Use the mass properties variable for the center of mass to apply the source term exactly in the center.

Variables 1

1 On the **Definitions** toolbar, click **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
Q_in	1e5[W/m^2]*exp(-5e6[1/m^2]*((x-mass1.CMx)^2+(z-mass1.CMz)^2))	W/m²	Boundary heat source

In the next section, you define the materials according to the Material Properties section.

MATERIALS

Material 1 (mat1)

1. On the **Materials** toolbar, click **Blank Material**.
2. In the **Settings** window for Material, type `Epoxy` in the **Label** text field.
3. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	0.2	W/(m·K)	Basic
Density	rho	1200	kg/m³	Basic
Heat capacity at constant pressure	Cp	1000	J/(kg·K)	Basic

Material 2 (mat2)

1. On the **Materials** toolbar, click **Blank Material**.
2. In the **Settings** window for Material, type `Carbon` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Fibers**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	{60, 4, 4}	W/(m·K)	Basic
Density	rho	1500	kg/m³	Basic
Heat capacity at constant pressure	Cp	1000	J/(kg·K)	Basic

Material 3 (mat3)

1. On the **Materials** toolbar, click **Blank Material**.
2. In the **Settings** window for Material, type `Carbon (Infinite Element Domain)` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Fibers (Infinite Element Domain)**.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	60	W/(m·K)	Basic
Density	rho	1500	kg/m³	Basic
Heat capacity at constant pressure	Cp	1000	J/(kg·K)	Basic

Add a second **Heat Transfer in Solids** node for the fibers and choose the curvilinear system as reference system. This way the thermal conductivity is high along the fiber axis and low perpendicular to it.

HEAT TRANSFER IN SOLIDS (HT)

Solid 2

1. On the **Physics** toolbar, click **Domains** and choose **Solid**.
2. In the **Settings** window for Solid, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Fibers (Core)**.
4. Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Curvilinear System (cc) (cc_cs)**.

Set up the boundary conditions, consisting in a heat source, a convective heat flux accounting for cooling, and a fixed temperature at the very outer boundaries of the infinite domain.

Boundary Heat Source 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Boundary Heat Source**.
2. In the **Settings** window for Boundary Heat Source, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Heat Source Boundary**.
4. Locate the **Boundary Heat Source** section. In the Q_b text field, type `Q_in`.

Heat Flux 1

1. On 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 **Cooling Boundaries**.
4. Locate the **Heat Flux** section. Click the **Convective heat flux** button.
5. In the h text field, type `10`.

Temperature 1

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

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

3 From the **Selection** list, choose **Temperature Boundaries**.

The first study was used to compute the curvilinear system. Add a second study to solve for the heat transfer only. Refer to **Study 1** in the **Values of Dependent Variables** section by selecting the solution as input for the variables not solved in this second study. This way the new coordinate system, which is initially unknown by **Study 2**, can be used for the heat transfer computation.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Curvilinear Coordinates (cc)** interface.

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

STUDY 2

Step 1: Stationary

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

2 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

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

4 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

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

7 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The default temperature plot shows the temperature distribution on the surface (Figure 3).

To create Figure 4 follow the steps below.

Study 2/Solution 2 (sol2)

In the **Model Builder** window, under **Results>Data Sets** right-click **Study 2/Solution 2 (sol2)** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **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 **Core Domains**.

Surface 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Surface**.

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

3 From the **Data set** list, choose **Study 2/Solution 2 (2) (sol2)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Fiber Walls**.

3D Plot Group 5

1 On the **Results** toolbar, click **3D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 On the **Temperature at Middle Slice** toolbar, click **Slice**.

Slice 1

1 In the **Model Builder** window, under **Results>Temperature at Middle Slice** click **Slice 1**.

2 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids> Temperature>T - Temperature**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

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

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

6 On the **Temperature at Middle Slice** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Temperature at Middle Slice** click **Surface 1**.

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

3 In the **Expression** text field, type 1.
4 Locate the **Data** section. From the **Data set** list, choose **Surface 1**.
5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
6 From the **Color** list, choose **Gray**.
7 On the **Temperature at Middle Slice** toolbar, click **Plot**.
8 Click the **Go to ZX View** button on the **Graphics** toolbar.
9 Click the **Zoom Extents** button on the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.2a

Radiation in a Cavity

Introduction

The following example shows how to build and solve a radiative heat transfer problem using the Heat Transfer with Surface-to-Surface Radiation interface. This example's simple geometry allows a comparison of results from COMSOL Multiphysics with a theoretical solution.

Model Definition

The geometry consists of three rectangles with lengths 3 m, 4 m, and 5 m, and all have a width of 1 m. The rectangles are situated such that they form a triangular cavity (see Figure 1).

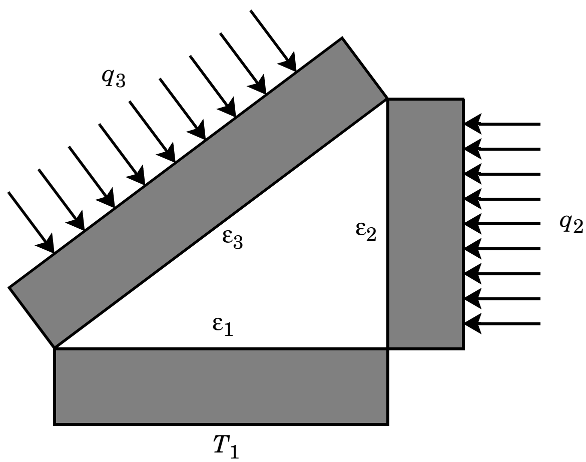


Figure 1: Three domains form a triangular cavity to study heat radiation.

The rectangles are made of copper, which is a good thermal conductor, and they transfer heat internally by conduction. Further, the three inner boundaries that form the cavity can exchange heat only by means of surface-to-surface radiation. The surface emissivities are $\varepsilon_1 = 0.4$ on the bottom boundary, $\varepsilon_2 = 0.6$ on the vertical boundary, and $\varepsilon_3 = 0.8$ on the inclined boundary.

The system holds the temperature on the (outer) lower boundary at $T_1 = 300$ K. Further, the system experiences an inward heat flux of $q_2 = 2000\ \mathrm{W/m^2}$ on the outer boundary of the vertical rectangle and $q_3 = 1000\ \mathrm{W/m^2}$ on the outer boundary of the tilted one. All remaining outer boundaries are insulated.

This example examines the steady state of the cavity's interior boundaries. Specifically, you know the value of the heat flux on the inclined and vertical boundaries, and you want to find the temperature along these boundaries. Conversely, you know the temperature of the bottom boundary and want to determine the heat flux that leaves through it.

Results and Discussion

Figure 2 shows the temperature field resulting from a simulation. If you run the model as described later in this section you can see from the color display that the temperatures are fairly constant within the three rectangles. The theoretical results predict that a heat flux $Q_1 = -11{,}000$ W/m leaves the cavity through the bottom boundary, that the vertical boundary has a temperature of $T_2 = 641$ K, and that the inclined boundary has a temperature of $T_3 = 600$ K. The results obtained are very close to these theoretical values. The section Comparing Results with Theory provides more details on the verification of these results.

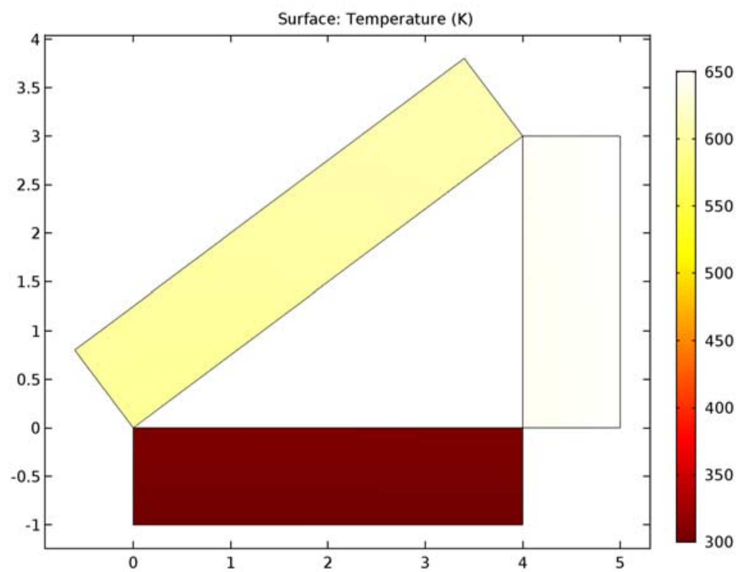


Figure 2: Temperature as a function of position.

Figure 3 details the temperature distribution along the inclined boundary of the cavity. Note that while the temperature is not constant, it varies by only a few percent. The lowest temperature appears on the bottom left, and the highest temperature is on the right, which seems reasonable because that portion of the boundary is located adjacent to the vertical boundary, which has the highest inward heat flux.

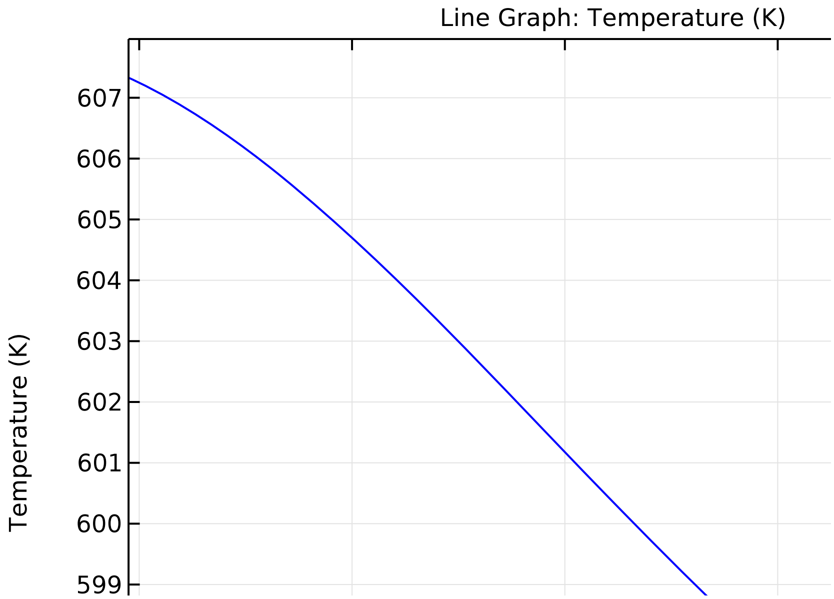


Figure 3: Temperature distribution along the inclined boundary of the cavity.

Figure 3 plots the radiosity along the inclined boundary (in other words, the total heat flux that leaves the boundary into the cavity). The radiosity is the sum of the heat flux the boundary *emits* plus the heat flux it *reflects*.

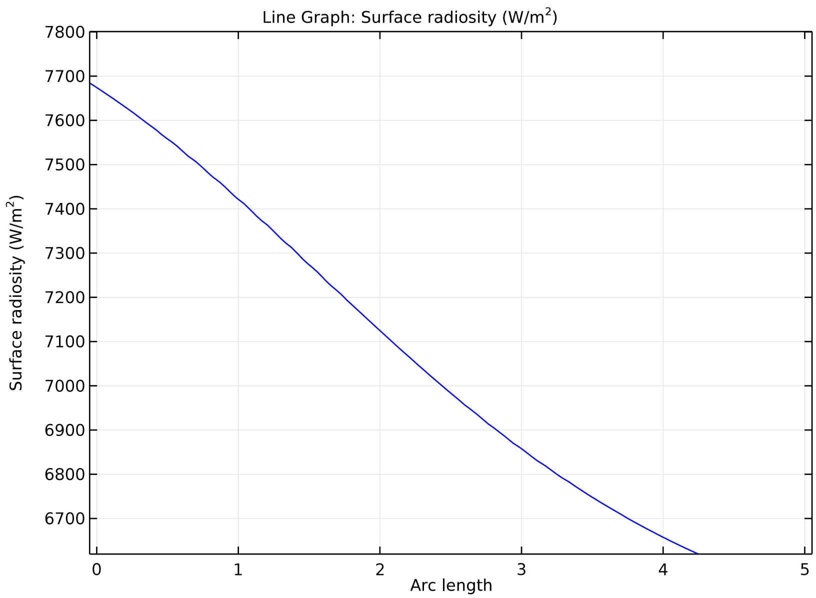


Figure 4: Radiosity along the inclined boundary of the cavity.

Like temperature, the radiosity is fairly constant along the boundary. The small variations have roughly the same distribution as the temperature in Figure 3.

COMPARING RESULTS WITH THEORY

As mentioned at the start of this section, you can compare the results from this tutorial to theoretical values, which are based on the sketch in Figure 5 and whose theory appears in the section “Enclosures with More Than Two Surfaces” in Ref. 1.

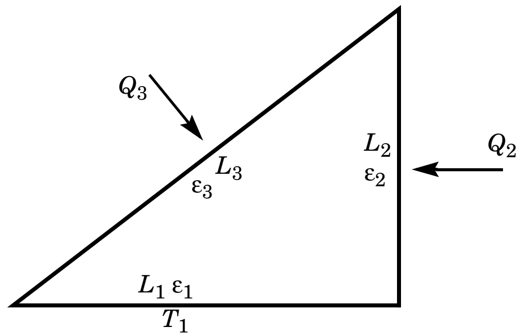


Figure 5: Problem sketch for theoretical analysis of radiation in a triangular cavity.

The theoretical model considers only the three boundaries that form the cavity. On these boundaries, the model either holds the temperature at a constant value or specifies a heat flux. Note that this approach differs somewhat from the model, which sets temperatures and heat fluxes on the rectangles’ *outer* boundaries.

The COMSOL Multiphysics model shows that the temperatures and heat fluxes are nearly equal on the inner and outer boundaries of the rectangles. Therefore, the COMSOL Multiphysics and the theoretical model show good agreement.

Using the notation from Figure 5 with the same assumptions as for the model, you obtain the following lengths and emissivities:

$L_1 = 4$ m, $\varepsilon_1 = 0.4$

$L_2 = 3$ m, $\varepsilon_2 = 0.6$

$L_3 = 5$ m, $\varepsilon_3 = 0.8$

The boundary conditions define either the temperature or the heat flux on each boundary. Now apply the same boundary conditions as in the problem where you set values for T_1, Q_2, and Q_3. You must, however, make a small adjustment for T_1 because the theoretical configuration sets it on the outer boundary and not on the cavity side. T_1 is therefore slightly higher than 300 K in the theoretical analysis:

$$T_1 = 307 \text{ K}$$

$$Q_2 = q_2 L_2 = 2000 \ \frac{\text{W}}{\text{m}} \times 3 \text{ m} = 6000 \text{ W}$$

$$Q_3 = q_3 L_3 = 1000 \ \frac{\text{W}}{\text{m}} \times 5 \text{ m} = 5000 \text{ W}$$

You describe the heat flux from a boundary with the following two equations:

$$Q_i = L_i \frac{\varepsilon_i}{1-\varepsilon_i}(\sigma T_i^{\ 4} - J_i)$$

$$Q_i = L_i \sum_k F_{ik}(J_i - J_k)$$

For a triangular cavity, the following equation gives the view factor between surface i and surface j:

$$F_{ij} = \frac{L_i + L_j - L_k}{2L_i}$$

Substituting this expression for the view factor into the second equation for the heat fluxes results in a linear system of six equations. The six unknowns for this particular problem are J_1, J_2, J_3, Q_1, T_2, and T_3, and you would like to compare the three last to those from the solution. Solving the linear system yields the following values: $Q_1 = -11{,}000$ W/m, $T_2 = 641$ K, and $T_3 = 600$ K.

To compare these results with the model, you must find the total heat flux through the cavity's bottom (horizontal) boundary and also calculate the average temperatures on the other two boundaries.

The following table compares the results from COMSOL Multiphysics with the theoretical values:

QUANTITY	COMSOL MULTIPHYSICS	THEORY
Q_1	-10,965 W/m	-11,000 W/m
T_2	645 K	641 K
T_3	601 K	600 K

The differences are quite small. Also note that the theoretical model includes some simplifications. For example, it assumes that heat fluxes and temperatures are constant along each boundary. It also assumes a constant view factor for each pair of boundaries, whereas COMSOL Multiphysics computes a local view factor at each point of the cavity.

Reference

1. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1993.

Application Library path: `Heat_Transfer_Module/Verification_Examples/cavity_radiation`

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**.
2. In the **Select Physics** tree, select **Heat Transfer>Radiation>Heat Transfer with Surface-to-Surface Radiation (ht)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

The geometry consists of three rectangles positioned such that they create a triangular cavity.

Rectangle 1 (r1)

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

Rectangle 2 (r2)

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

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

3 In the **Height** text field, type `3`.

4 Locate the **Position** section. In the **x** text field, type `4`.

Rectangle 3 (r3)

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

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

3 In the **Width** text field, type `5`.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type `atan(3/4)[rad]`.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

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

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

This completes the geometry modeling stage.

MATERIALS

Material 1 (mat1)

1 On the **Materials** toolbar, click **Blank Material**.

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

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	400	W/(m·K)	Basic
Density	rho	8700	kg/m³	Basic
Heat capacity at constant pressure	Cp	385	J/(kg·K)	Basic

HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION (HT)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer with Surface-to-Surface Radiation (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `300` in the T text field.

Next, define the boundary conditions.

Heat Flux 1

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

2 Select Boundary 2 only.

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

4 In the q_0 text field, type `1000`.

Heat Flux 2

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

2 Select Boundary 12 only.

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

4 In the q_0 text field, type `2000`.

Temperature 1

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

2 Select Boundary 6 only.

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

4 In the T_0 text field, type `300`.

Diffuse Surface 1

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

2 Select Boundary 3 only.

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

4 In the T_{amb} text field, type `300`.

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.8`.

   By default the radiation direction is controlled by the opacity of the domains. The solid parts are automatically defined as opaque. You can change this setting by modifying the **Opacity** subnode under the **Solid** feature.

Diffuse Surface 2

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

2 Select Boundary 7 only.

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

4 In the T_{amb} text field, type `300`.

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.4`.

Diffuse Surface 3

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

2 Select Boundary 9 only.

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

4 In the T_{amb} text field, type `300`.

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.6`.

MESH 1

On the **Mesh** toolbar, click **Edge**.

Edge 1

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

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

3 From the **Geometric entity level** list, choose **Entire geometry**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Size**.

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

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

4 Select Boundaries 3, 7, and 9 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `0.05`.

Edge 1

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

Size 2

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

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

3 Select Points 2, 4, 6, and 8–10 only.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.01`.

7 Click **Build Selected**.

8 On the **Mesh** toolbar, click **Free Triangular**.

Free Triangular 1

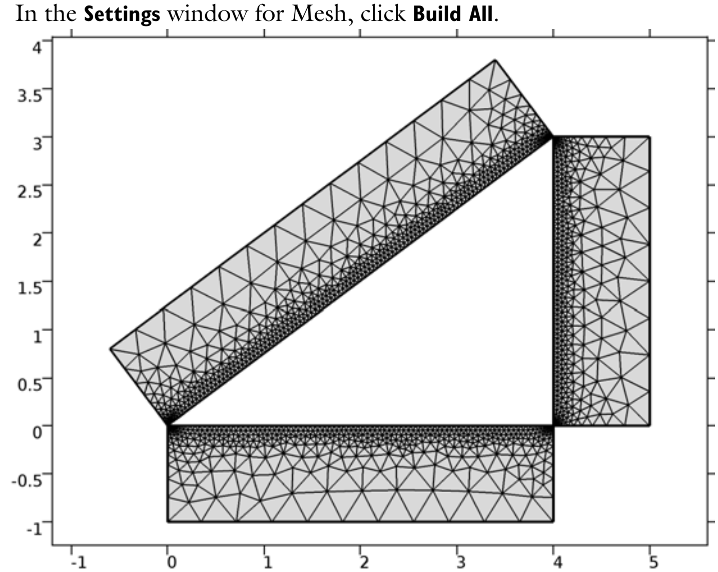
1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.

2 In the **Settings** window for Free Triangular, click to expand the **Tessellation** section.

3 From the **Method** list, choose **Advancing front**.

4 In the **Model Builder** window, click **Mesh 1**.

5 In the **Settings** window for Mesh, click **Build All**.



The complete mesh consists of roughly 2,700 elements.

STUDY 1

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

The computation takes around 10 seconds.

RESULTS

Temperature (ht)

The first default plot shows the temperature field; compare with Figure 2.

The two additional default plots visualize the temperature field isothermal contours, and the radiosity on the boundaries.

Add a 1D plot for the temperature field on the inclined boundary (Figure 3).

1D Plot Group 4

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

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

Line Graph 1

1 On the **Temperature Profile vs Arc Length** toolbar, click **Line Graph**.

2 Select Boundary 3 only.

3 On the **Temperature Profile vs Arc Length** toolbar, click **Plot**.

Temperature Profile vs Arc Length

Add another 1D plot showing the radiosity on the inclined boundary (Figure 4).

1 In the **Model Builder** window, under **Results** right-click **Temperature Profile vs Arc Length** and choose **Duplicate**.

Temperature Profile vs Arc Length 1

In the **Settings** window for 1D Plot Group, type `Surface Radiosity Profile vs Arc Length` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Surface Radiosity Profile vs Arc Length** node, then click **Line Graph 1**.

2 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Heat Transfer with Surface-to-Surface Radiation>Radiation>Surface radiosity>ht.J - Surface radiosity**.

3 On the **Surface Radiosity Profile vs Arc Length** toolbar, click **Plot**.

Derived Values

Determine the conductive heat flux through the cavity's bottom boundary as follows.

Line Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.

2 Select Boundary 7 only.

3 In the **Settings** window for Line Integration, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer with Surface-to-Surface Radiation>Boundary fluxes>ht.ndflux - Normal conductive heat flux**.

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result displayed in the **Table** window should be approximately 11,000 W/m.

Next, find the mean temperature on the vertical boundary of the cavity.

RESULTS

Line Integration 2

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.

2 Select Boundary 9 only.

3 In the **Settings** window for Line Integration, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
T/3	m*K	

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The resulting temperature, read from the table, should be approximately 641 K.

Finally, evaluate the mean temperature on inclined boundary of the cavity.

RESULTS

Line Integration 3

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.
2 Select Boundary 3 only.
3 In the **Settings** window for Line Integration, locate the **Expressions** section.
4 In the table, enter the following settings:

Expression	Unit	Description
T/5	m*K	

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The resulting temperature, read from the table, should be approximately 600 K.

Created in COMSOL Multiphysics 5.2a

Composite Thermal Barrier

Introduction

This example shows how to set up multiple sandwiched thin layers with different thermal conductivities in two different ways.

First, the composite is modeled as a 3D object. In the second approach the Thin Layer boundary condition is used to avoid resolving the thin domains.

The technique is useful when modeling heat transfer through thermal barriers like multilayer coatings.

Model Definition

This tutorial uses a simple geometry as shown in Figure 1. The cylinder has a radius of 2 cm and a height of 4 cm.

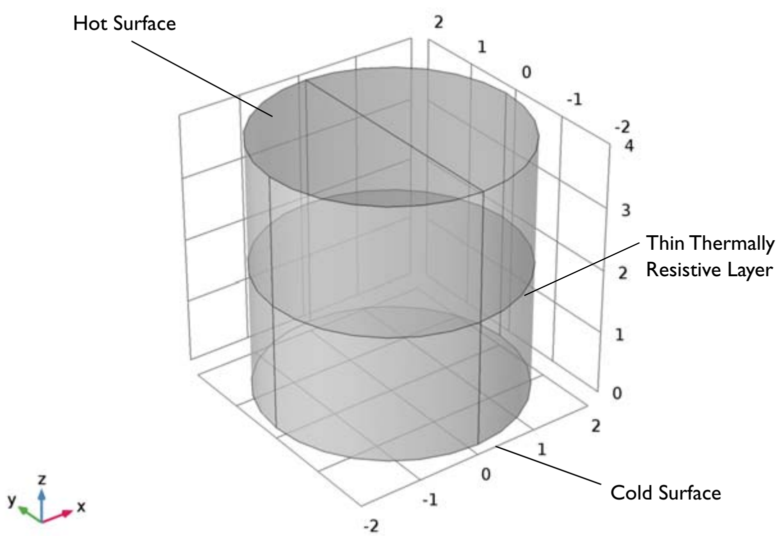


Figure 1: Geometry.

The composite consists of two layers with different thermal conductivities. The first approach resolves each layer as a 3D domain. The height of the layers is about three orders of magnitude smaller than the bulk height. This often requires to build a mesh manually to accurately resolve the thin structure.

COMSOL Multiphysics provides a special boundary condition which is available from the Heat Transfer Module. The second approach uses the Thin Layer feature with resistive property. It simplifies the geometry and thus the mesh. In complex geometries, this boundary condition can reduce the amount of memory and time required for the simulation significantly.

The underlying equation assumes the heat flux through the layer proportional to the temperature difference between upper and lower bulk. It is based on the assumption that the bulk on each side is well stirred so that all resistance against heat transfer is within a thin layer near the wall. Due to the additivity of resistance the flux over the composite can be lumped to

$$-\mathbf{n}_{\mathrm{d}}(-k_{\mathrm{d}}\nabla T_{\mathrm{d}}) = \frac{k_{\mathrm{tot}}}{d_{\mathrm{tot}}}(T_{\mathrm{u}} - T_{\mathrm{d}})$$

$$-\mathbf{n}_{\mathrm{u}}(-k_{\mathrm{u}}\nabla T_{\mathrm{u}}) = \frac{k_{\mathrm{tot}}}{d_{\mathrm{tot}}}(T_{\mathrm{d}} - T_{\mathrm{u}})$$

where the overall thermal conductivity k_{tot} can be calculated as

$$k_{\mathrm{tot}} = \frac{\sum_n d_n}{\sum_n \frac{d_n}{k_n}}$$

MATERIAL PROPERTIES

The cylinder is made of steel. The composite consist of two layers of different ceramics.

TABLE 1: CERAMICS MATERIAL PROPERTIES

PROPERTY	CERAMIC 1	CERAMIC 2
Thermal conductivity	1 W/(m·K)	0.5 W/(m·K)
Density	6000 kg/m^3	5800 kg/m^3
Heat capacity at constant pressure	320 J/(kg·K)	280 J/(kg·K)

BOUNDARY CONDITIONS

The temperature at the bottom is fixed to 20 °C whereas one half of the top boundary is held at 1220 °C (1493 K). All other outer boundaries are perfectly insulated.

Figure 2 shows the temperature distribution in the cylinder. The composite acts as a thermal barrier resulting in a jump of the temperature over the layer.

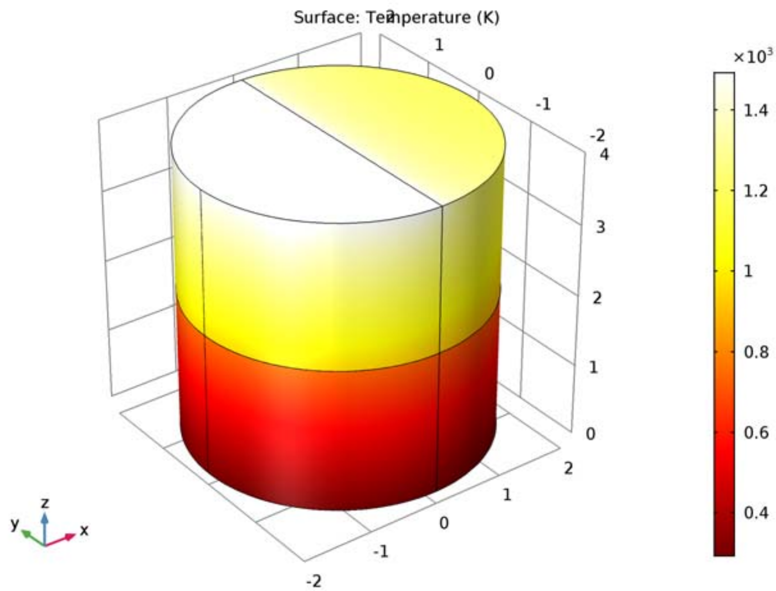


Figure 2: Temperature distribution.

Of interest is if the thin layer boundary condition produces reliable results compared to resolving the thin layers in 3D. This can be done with a comparative line graph as in Figure 3. It shows that the 2D approach, with or without an extra dimension, produces accurate results for the bulk temperatures.

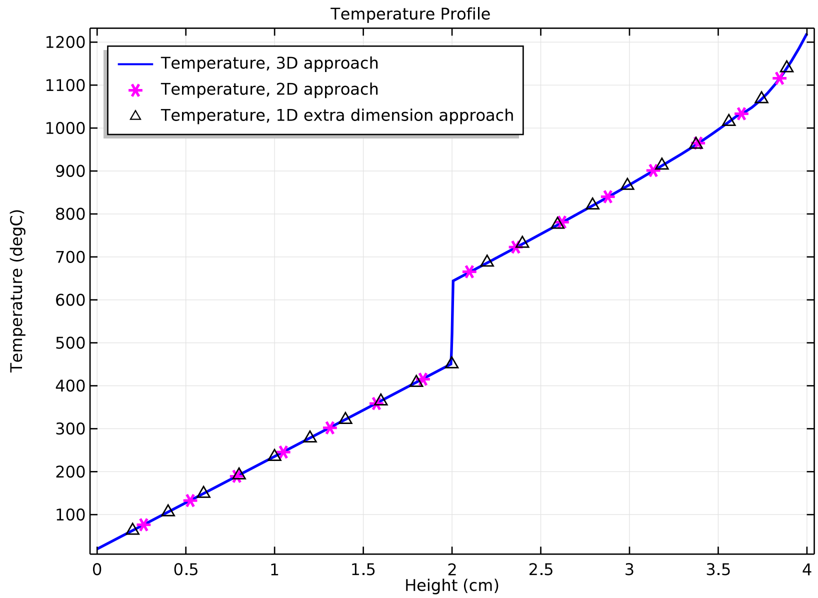


Figure 3: Temperature profile for 3D, 2D and 1D extra dimension approaches.

Another important question for simulating is the influence on the mesh size and on the required RAM.

With the default tetrahedral mesh the number of mesh elements is about 130,000 elements and the meshing algorithm gives some warnings.

With the swept mesh feature you can significantly reduce the number of elements to about 12,800 elements which is only 10%. In complex geometries the swept mesh algorithm is often not applicable. Using the thin layer boundary condition, the number of mesh elements reduces from 12,800 to 9,700 which is about 25% less, even in this simple geometry. You can see the number of mesh elements used in the messages box on the bottom right window.

Notes About the COMSOL Implementation

To compare the results directly, both approaches are handled within one mph-file.

Application Library path: `Heat_Transfer_Module/Tutorials,_Thin_Structure/composite_thermal_barrier`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
d_ceram1	50[um]	5E-5 m	Thickness of layer 1
d_ceram2	75[um]	7.5E-5 m	Thickness of layer 2
T_hot	1220[degC]	1493.2 K	Hot temperature

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for Geometry, locate the **Units** section.
3 From the **Length unit** list, choose **cm**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type `2`.
4 In the **Height** text field, type `4`.
5 On the **Geometry** toolbar, click **Build All**.

Now, create thin cylinders to define the ceramic layers between the two steel domains.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type `2`.
4 In the **Height** text field, type `d_ceram1`.
5 Locate the **Position** section. In the **z** text field, type `2-(d_ceram1+d_ceram2)/2`.
6 On the **Geometry** toolbar, click **Build All**.

Cylinder 3 (cyl3)

1 On the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type `2`.
4 In the **Height** text field, type `d_ceram2`.
5 Locate the **Position** section. In the **z** text field, type `2-(d_ceram1+d_ceram2)/2+ d_ceram1`.
6 On the **Geometry** toolbar, click **Build All**.

Polygon 1 (pol1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.
2 In the **Settings** window for Polygon, locate the **Coordinates** section.
3 In the **x** text field, type `0 0`.
4 In the **y** text field, type `-2 2`.
5 In the **z** text field, type `4 4`.
6 On the **Geometry** toolbar, click **Build All**.

ADD MATERIAL

1 On 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>Steel AISI 4340**.

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

MATERIALS

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Material 2 (mat2)

1 On the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for Material, type `Ceramic 1` in the **Label** text field.

3 Select Domain 2 only.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	1	W/(m·K)	Basic
Density	rho	6000	kg/m³	Basic
Heat capacity at constant pressure	Cp	320	J/(kg·K)	Basic

Material 3 (mat3)

1 On the **Materials** toolbar, click **Blank Material**.

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

3 Select Domain 3 only.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	0.5	W/(m·K)	Basic
Density	rho	5800	kg/m³	Basic
Heat capacity at constant pressure	Cp	280	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

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

2 Select Boundary 3 only.

Temperature 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundary 13 only.
3 In the **Settings** window for Temperature, locate the **Temperature** section.
4 In the T_0 text field, type `T_hot`.

MESH 1

First, mesh the top surface with a free triangular mesh and extrude it in layers through the cylindrical geometry. With a **Distribution** node, specify how many mesh layers are to be created within the domain. Resolve the composite layers with two elements in thickness.

1 On the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.
2 Select Boundaries 13 and 18 only.
3 In the **Settings** window for Free Triangular, click **Build Selected**.
4 On the **Mesh** toolbar, click **Swept**.

Swept 1

Click **Distribution**.

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Swept 1** click **Distribution 1**.
2 Select Domains 2 and 3 only.
3 In the **Settings** window for Distribution, locate the **Distribution** section.
4 In the **Number of elements** text field, type `2`.
5 Click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The following plots are produced by default: temperature profile on the surface as in Figure 2, and isothermal contours.

Surface

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

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

3 From the **Unit** list, choose **degC**.

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

Next, create a temperature profile along the height of the cylinder. You will later compare the graph with the results of the 2D approach.

1D Plot Group 3

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

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

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

4 In the associated text field, type `Height (cm)`.

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

6 In the **Title** text area, type `Temperature Profile`.

7 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 On the **Temperature Profile** toolbar, click **Line Graph**.

2 Select Edges 15, 17, 19, and 21 only.

3 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

4 From the **Unit** list, choose **degC**.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `z`.

7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

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

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Temperature, 3D approach`

11 On the **Temperature Profile** toolbar, click **Plot**.

Create now the second model which uses the **Thin Layer** feature and compare the results to the first approach.

ROOT

1 On the **Home** toolbar, click **Add Component** and choose **3D**.

2 In the **Model Builder** window, click the root node.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

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

ROOT

On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Heat Transfer in Solids (ht)** interface.

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

ROOT

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

GEOMETRY 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `2`.

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

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

Layer name	Thickness (cm)
Layer 1	2

6 Clear the **Layers on side** check box.

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

8 On the **Geometry** toolbar, click **Build All**.

Polygon 1 (pol1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

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

3 In the **x** text field, type `0 0`.

4 In the **y** text field, type `-2 2`.

5 In the **z** text field, type `4 4`.

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

ADD MATERIAL

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

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

3 In the tree, select **Recent Materials>Steel AISI 4340**.

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

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

HEAT TRANSFER IN SOLIDS 2 (HT2)

Thin Layer 1

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

2 Select Boundary 6 only.

3 In the **Settings** window for Thin Layer, locate the **Thin Layer** section.

4 Select the **Multiple layers** check box.

5 Locate the **Heat Conduction** section. From the **Number of layers** list, choose **2**.

6 Find the **Layer 1 (upside)** subsection. From the list, choose **Ceramic 1 (mat2)**.

7 In the d_{s1} text field, type `d_ceram1`.

8 Find the **Layer 2 (downside)** subsection. From the list, choose **Ceramic 2 (mat3)**.

9 In the d_{s2} text field, type `d_ceram2`.

Thin Layer 2

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

2 Select Boundary 6 only.

3 In the **Settings** window for Thin Layer, locate the **Thin Layer** section.

4 From the **Layer type** list, choose **General**.

5 Locate the **Heat Conduction** section. From the **Number of layers** list, choose **2**.

6 Find the **Layer 1 (upside)** subsection. From the list, choose **Ceramic 1 (mat2)**.

7 In the d_{s1} text field, type `d_ceram1`.

8 Find the **Layer 2 (downside)** subsection. From the list, choose **Ceramic 2 (mat3)**.

9 In the d_{s2} text field, type `d_ceram2`.

Temperature 1

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

2 Select Boundary 3 only.

Temperature 2

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

2 Select Boundary 7 only.

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

4 In the T_0 text field, type `T_hot`.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

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

3 Select the **Modify physics tree and variables for study step** check box.

Disable **Thin Layer 2** in this study. First, **Thin Layer 1** is used to show the results with the **Resistive** option. Later, another study will use **Thin Layer 2** with the **General** option.

4 In the **Physics and variables selection** tree, select **Component 2 (comp2)>Heat Transfer in Solids 2 (ht2)>Thin Layer 2**.

5 Click **Disable**.

MESH 2

On the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Free Triangular 1**.

2 Select Boundaries 7 and 10 only.

3 In the **Settings** window for Free Triangular, click **Build Selected**.

4 On the **Mesh** toolbar, click **Swept**.

5 In the **Model Builder** window, click **Mesh 2**.

6 In the **Settings** window for Mesh, click **Build All**.

STUDY 2

On the **Home** toolbar, click **Compute**.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Temperature (ht2)** node, then click **Surface**.

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

3 From the **Unit** list, choose **degC**.

Temperature Profile

In the **Model Builder** window, under **Results** click **Temperature Profile**.

Line Graph 2

1 On the **Temperature Profile** toolbar, click **Line Graph**.

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

3 From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Select Edges 9 and 11 only.

5 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 2>Heat Transfer in Solids 2>Temperature>T2 - Temperature**.

6 Locate the **y-Axis Data** section. From the **Unit** list, choose **degC**.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type z.

9 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

10 From the **Color** list, choose **Magenta**.

11 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

12 In the **Number** text field, type `15`.

13 Locate the **Legends** section. Select the **Show legends** check box.

14 From the **Legends** list, choose **Manual**.

15 In the table, enter the following settings:

Legends
`Temperature, 2D approach`

16 On the **Temperature Profile** toolbar, click **Plot**.

Add now a new study to solve **Component 2** with **Thin Layer 2** instead of **Thin Layer 1**. The **Thin Layer 2** feature would use the **General** option which creates a 1D extra dimension formed by two intervals to represent the two ceramic layers.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Heat Transfer in Solids (ht)** interface.

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

STUDY 3

Step 1: Stationary

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

2 In the **Model Builder** window, under **Study 3** click **Step 1: Stationary**.

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

4 Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 2 (comp2)>Heat Transfer in Solids 2 (ht2)>Thin Layer 1**.

6 Click **Disable**.

7 On the **Home** toolbar, click **Compute**.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Temperature (ht2) 1** node, then click **Surface**.

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

3 From the **Unit** list, choose **degC**.

Temperature Profile

In the **Model Builder** window, under **Results** click **Temperature Profile**.

Line Graph 3

1 On the **Temperature Profile** toolbar, click **Line Graph**.

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

3 From the **Data set** list, choose **Study 3/Solution 3 (5) (sol3)**.

4 Select Edges 9 and 11 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `T2`.

6 From the **Unit** list, choose **degC**.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `z`.

9 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

10 From the **Color** list, choose **Black**.

11 In the **Width** text field, type `2`.

12 Find the **Line markers** subsection. From the **Marker** list, choose **Triangle**.

13 In the **Number** text field, type `20`.

14 Locate the **Legends** section. Select the **Show legends** check box.

15 From the **Legends** list, choose **Manual**.

16 In the table, enter the following settings:

Legends
`Temperature, 1D extra dimension approach`

17 On the **Temperature Profile** toolbar, click **Plot**.

The plot should look like that in Figure 3.

Condensation Detection in an Electronic Device

Introduction

Many systems, for example electronic devices, risk being damaged if exposed to condensation. Given an amount of moisture in the air, condensation occurs when the temperature decreases to reach the dew point. Numerical simulations are useful for obtaining knowledge relevant for preventing the formation of condensation.

Changes in air properties are the primary cause of condensation in some systems. This example simulates the thermodynamical evolution of moist air in an electronic box with the aim of detecting whether condensation occurs when the external environment properties change. The model uses meteorological data for the air temperature, pressure, and relative humidity, measured at New York, JFK station. The property data correspond to average conditions of dry bulb temperature and high conditions of dew point temperature, observed on the 1st of June.

In this simulation, you assume the water vapor concentration to be homogeneous inside the box and equal to the external concentration. Also, the model setup neglects diffusion but considers the external concentration changes during the simulation.

Note: An extension of this application solves for an inhomogeneous concentration computed from the Transport of Diluted Species interface and takes transport and diffusion of the water vapor into account, see Condensation Detection in an Electronic Device with Transport and Diffusion. It requires the Chemical Reaction Engineering Module.

Model Definition

A box with square cross section of side 5 cm is placed in a moist air environment. It contains a heated electronic component and two small slits (1 mm thick) located at the left and right sides. The simulation is in a 2D cross section of the box, which is supposed to be long enough in the orthogonal direction. It is made of aluminum and the electronic component is made of silicon. Figure 1 shows the model geometry.

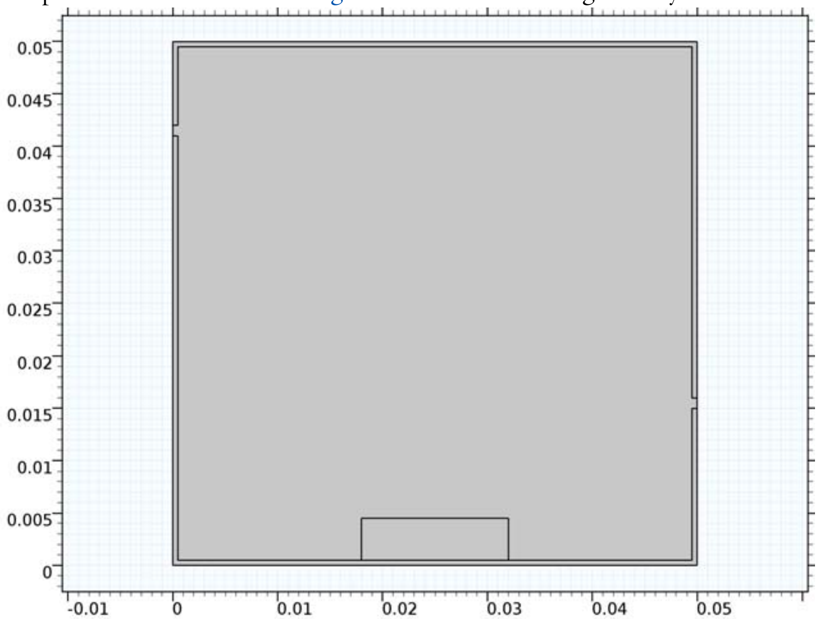


Figure 1: Geometry of the model.

The box is placed in a changing environment. This means that during the simulation, temperature, pressure, and relativc humidity change. Figure 2 shows the temperature and relative humidity as functions of time.

In this simulation, assume the moist air concentration inside the box to be equal to the external concentration.

Outside the box, you apply a convective cooling condition with a heat transfer coefficient h equal to 10 W/(m^2·K) and a time-dependent external temperature equal to the ambient temperature. The central component produces a total power of 1 W during the simulation. At the slit boundaries, set a condition of open boundary to let external moist air freely enter or exit from the box.

The study computes a simulation over one day and the solution is stored every 30 minutes. The goal is to observe if some condensation appears.

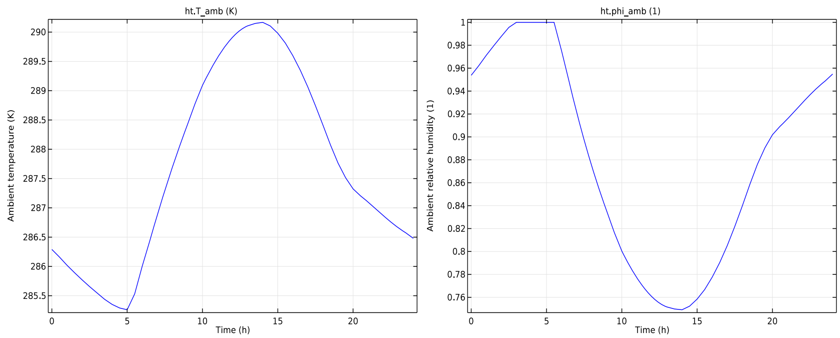


Figure 2: Temperature and relative humidity over the course of a day.

Results and Discussion

Figure 3 shows the temperature and relative humidity profiles at the final time step.

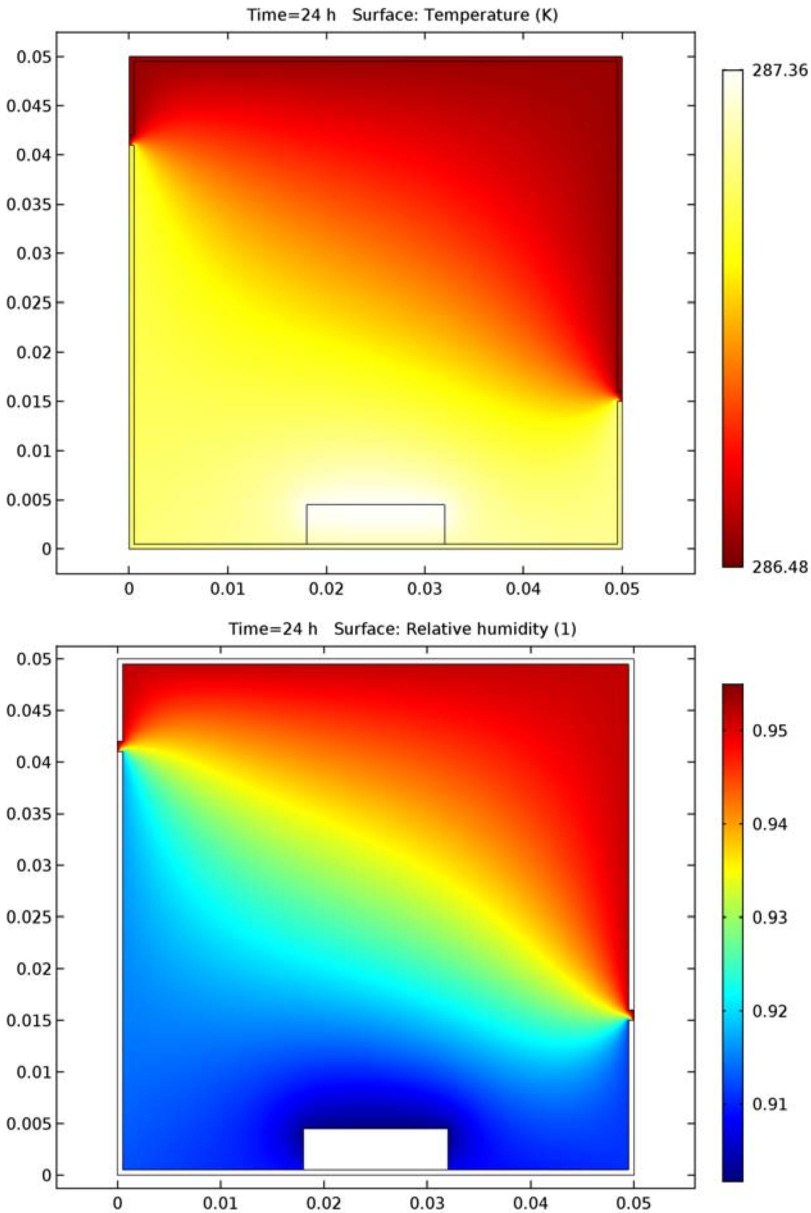


Figure 3: Temperature and relative humidity profiles after 24 hours.

While the temperature gradient is not very large, the power dissipated from the electronic component clearly influences the temperature field: it heats the surrounding air and the walls. Cold air enters through the slits by convection. In addition, the air inside the box is cooled by conduction through the walls. The relative humidity maximum is located where the temperature is the lowest but also where the water vapor concentration is the highest.

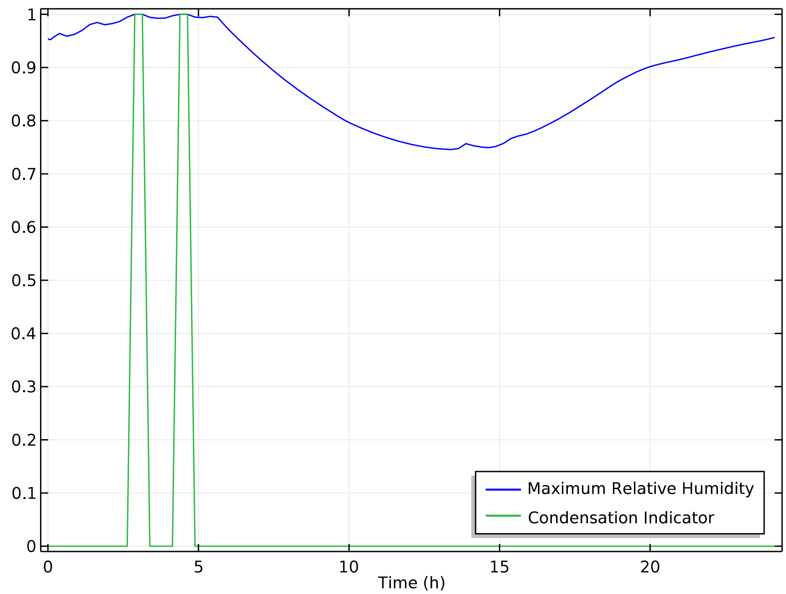


Figure 4: Maximum relative humidity over time inside the box.

Figure 4 represents the evolution of the maximum relative humidity inside the box over the simulation period. This curve reaches a maximum of 100% several times, meaning that condensation occurs. A Boolean condensation indicator is inserted in order to distinguish the exact condensation period. The condensation indicator is set to 1 when condensation is detected (relative humidity equals 1) and to 0 otherwise.

Application Library path: `Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/condensation_electronic_device`

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**.
2 In the **Select Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.
2 In the **Settings** window for Import, locate the **Import** section.
3 Click **Browse**.
4 Browse to the application's Application Libraries folder and double-click the file `condensation_electronic_device.mphbin`.
5 Click **Import**.

   The imported geometry is represented in Figure 1.

MATERIALS

A material is only needed on the solid part as the fluid part is going to be defined at the feature level through the moist air functionality.

ADD MATERIAL

1 On 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>Aluminum**.
4 Click **Add to Component** in the window toolbar.

MATERIALS

Aluminum (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat1)**.

2 Select Domains 1 and 3 only.

ADD MATERIAL

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

2 In the tree, select **Built-In>Silica glass**.

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

MATERIALS

Silica glass (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silica glass (mat2)**.

2 Select Domain 4 only.

3 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

HEAT TRANSFER (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer (ht)**.

2 In the **Settings** window for Heat Transfer, locate the **Ambient Settings** section.

3 From the **Ambient data** list, choose **Meteorological data (ASHRAE 2013)**.

4 Find the **Time** subsection. In the table, enter the following settings:

Hour	Minute	Second
0	0	0

5 Find the **Ambient conditions** subsection. From the **Temperature** list, choose **Low**.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Fluid 1**.

2 Select Domain 2 only.

3 In the **Settings** window for Fluid, locate the **Thermodynamics, Fluid** section.

4 From the **Fluid type** list, choose **Moist air**.

5 From the **Input quantity** list, choose **Relative humidity**.

6 From the ϕ_{ref} list, choose **Ambient relative humidity (ht)**.

7 From the T_{ref} list, choose **Ambient temperature (ht)**.

8 From the p_{ref} list, choose **Ambient absolute pressure (ht)**.

9 Locate the **Domain Selection** section. Click **Create Selection**.

10 In the **Create Selection** dialog box, type `Moist Air` in the **Selection name** text field.

11 Click **OK**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, choose **Ambient temperature (ht)** from the T list.

Heat Flux 1

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

2 Select Boundaries 1, 2, 5, 7, 21, and 23 only.

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

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

5 In the h text field, type `10`.

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

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 4 only.

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

4 Click the **Heat rate** button.

5 In the P_0 text field, type `1`.

Open Boundary 1

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

2 Select Boundaries 3 and 22 only.

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

4 From the T_0 list, choose **Ambient temperature (ht)**.

LAMINAR FLOW (SPF)

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

2 In the **Settings** window for Laminar Flow, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Moist Air**.

Fluid Properties 1

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

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

3 From the μ list, choose **Dynamic viscosity (ht/fluid1)**.

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 In the p text field, type `ht.p_amb-spf.pref`.

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

Open Boundary 1

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

2 Select Boundaries 3 and 22 only.

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

4 In the f_0 text field, type `ht.p_amb-spf.pref`.

DEFINITIONS

Then, two probes are defined in order to get the maximum relative humidity and the condensation indicator at the solver time steps.

Domain Probe 1 (dom1)

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for Domain Probe, locate the **Source Selection** section.

3 Click **Clear Selection**.

4 Select Domain 2 only.

5 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1 (comp1)>Heat Transfer>Moist air>ht.phi - Relative humidity**.

Domain Probe 2 (dom2)

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for Domain Probe, locate the **Source Selection** section.

3 Click **Clear Selection**.

4 Select Domain 2 only.

5 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1 (comp1)>Heat Transfer>Moist air>ht.condInd - Condensation indicator**.

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 From the **Time unit** list, choose **h**.

4 Click **Range**.

5 In the **Range** dialog box, type `0.5` in the **Step** text field.

6 In the **Stop** text field, type `24`.

7 Click **Replace**.

Solution 1 (sol1)

1 On 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>Segregated 1** node, then click **Velocity u, Pressure p**.

4 In the **Settings** window for Segregated Step, click to expand the **Method and termination** section.

5 Locate the **Method and Termination** section. From the **Jacobian update** list, choose **On every iteration**.

   Because the problem is nonlinear, updating the Jacobian on every iteration yields faster resolution.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Time-Dependent Solver 1>Segregated 1** click **Temperature T**.

7 In the **Settings** window for Segregated Step, click to expand the **Method and termination** section.

8 Locate the **Method and Termination** section. From the **Jacobian update** list, choose **On every iteration**.

9 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.

10 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

11 Locate the **Time Stepping** section. Select the **Maximum step** check box.

12 In the associated text field, type `0.25`.

Because the temperature and pressure variations can be quick, forcing a reduced time step helps to capture all curve variations.

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

RESULTS

Temperature (ht)

This default plot represents the temperature profile at the last time step, as shown in the top panel of Figure 3.

Velocity (spf)

This default plot shows the velocity profile at the last time step.

2D Plot Group 6

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

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

3 On the **Relative Humidity** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Relative Humidity** click **Surface 1**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer>Moist air>ht.phi - Relative humidity**.

3 On the **Relative Humidity** toolbar, click **Plot**.

Compare with the relative humidity profile in the bottom panel of Figure 3.

Probe Plot Group 5

Follow the steps below to reproduce the relative humidity evolution shown in Figure 4.

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 5**.

2 In the **Settings** window for 1D Plot Group, type `Maximum Relative Humidity` in the **Label** text field.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Results>Maximum Relative Humidity** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for Table Graph, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Maximum Relative Humidity`
`Condensation Indicator`

Maximum Relative Humidity

1 In the **Model Builder** window, under **Results** click **Maximum Relative Humidity**.

2 In the **Settings** window for 1D Plot Group, click to expand the **Legend** section.

3 From the **Position** list, choose **Lower right**.

4 Click to expand the **Window settings** section. Locate the **Window Settings** section. From the **Plot window** list, choose **Graphics**.

5 On the **Maximum Relative Humidity** toolbar, click **Plot**.

Finally, plot the ambient temperature and relative humidity to reproduce Figure 2.

1D Plot Group 7

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

2 In the **Settings** window for 1D Plot Group, type `Ambient 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 `ht.T_amb (K)`.

Point Graph 1

1 On the **Ambient Temperature** toolbar, click **Point Graph**.

2 Select Point 10 only.

3 In the **Settings** window for Point Graph, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `ht.T_amb`.

5 On the **Ambient Temperature** toolbar, click **Plot**.

This figure should look like the left curve of Figure 2.

1D Plot Group 8

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

2 In the **Settings** window for 1D Plot Group, type `Ambient Relative Humidity` 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 `ht.phi_amb (1)`.

Point Graph 1

1 On the **Ambient Relative Humidity** toolbar, click **Point Graph**.

2 Select Point 10 only.

3 In the **Settings** window for Point Graph, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `ht.phi_amb`.

5 On the **Ambient Relative Humidity** toolbar, click **Plot**.

This figure should look like the right curve of Figure 2.

Modeling a Conical Dielectric Probe for Skin Cancer Diagnosis

Introduction

The response of a millimeter wave with frequencies of 35 GHz and 95 GHz is known to be very sensitive to water content. This model utilizes a low-power 35 GHz Ka-band millimeter wave and its reflectivity to moisture for non-invasive cancer diagnosis. Since skin tumors contain more moisture than healthy skin, it leads to stronger reflections on this frequency band. Hence the probe detects abnormalities in terms of S-parameters at the tumor locations. A circular waveguide at the dominant mode and a conically tapered dielectric probe are quickly analyzed, along with the probe's radiation characteristics, using a 2D axisymmetric model. Temperature variation of the skin and the fraction of necrotic tissue analysis are also performed as well.

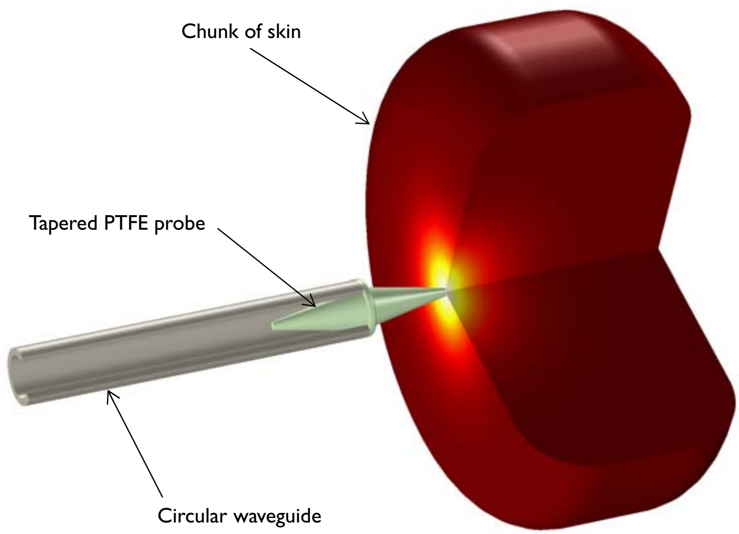


Figure 1: 3D visualization of the 2D axisymmetric model. The probe consists of a circular waveguide and a tapered dielectric rod.

Model Definition

The model consists of a metallic circular waveguide, a tapered PTFE dielectric rod, and a phantom of skin chunk shown in Figure 1. The entire model is enclosed by an air domain which is truncated at its outermost shell with perfectly matched layers (PML) to absorb any radiation directly from the rod or reflected from the skin phantom. One end of the waveguide is terminated with a circular port and excited using the dominant TE_{1m} mode, where m is the azimuthal mode number of this 2D axisymmetric model defined as 1 in the Electromagnetic Waves, Frequency Domain physics interface settings. The other end is connected to a tapered conical PTFE dielectric ($\varepsilon_r = 2.1$) rod. The shape of the rod is symmetrically tapered so the radius is increasing from the inside to the outside of the waveguide, then it is decreasing gradually for the impedance matching between the waveguide and the air domain. There is a ring structure in the middle to support the rod on the rim of the waveguide. The tip of the rod is touching the skin phantom.

The conductivity of the metallic waveguide is assumed to be high enough to neglect any loss and is modeled as perfect electric conductor (PEC). With the given radius of the waveguide and excited TE mode, the cutoff frequency is around 29.3 GHz, which is calculated by

$$f_{c_{ml}} = \frac{c_0 p'_{nm}}{2\pi a}$$

where c_0 is the speed of light, p'_{nm} are the roots of the derivative of the Bessel functions $J_n(x)$, m and n are the mode indices, and a is the radius of the waveguide. The value of p'_{11} is approximately 1.841. The operating frequency of the probe, 35 GHz, is higher than the waveguide cutoff frequency. The excited wave is propagating along the waveguide.

The circular port boundary condition is placed on the interior boundary where the reflection and transmission characteristics are computed automatically in terms of S-parameters. The interior port boundary with PEC backing for one-way excitation requires the slit condition. The port orientation is specified to define the inward direction for the S-parameter calculation.

First, the electromagnetic properties of the model are analyzed without a phantom to check the design validity of the waveguide and dielectric rod. Then, complexity is added, first with a healthy phantom, then a phantom with a skin tumor. See Table 1.

TABLE 1: MATERIAL PROPERTY VARIATION

PROPERTY	PROBE ONLY	WITH A HEALTHY PHANTOM	TUMOR ADDED
Relative permittivity (imaginary part)	0	10	15
Relative permittivity (real part)	1	5	8

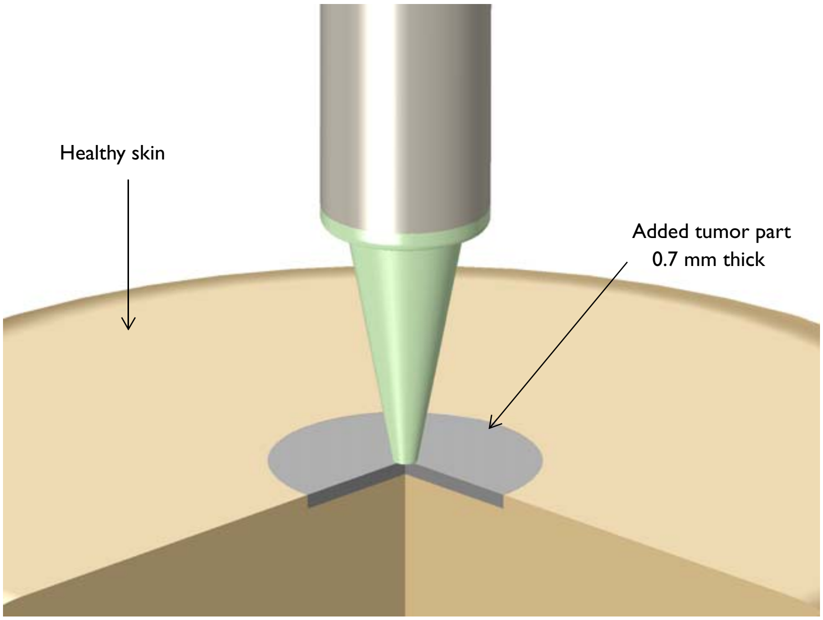


Figure 2: Zoomed 3D visualization of the skin tumor area. The entire probe model is simulated in a 2D axisymmetric space dimension. The measured S-parameters vary due to the different moisture content in each skin phantom.

Though the waveguide excited by low power is expected to be harmless, its effect on necrotic tissue is reviewed by studying Bioheat Transfer as well as temperature, over a 10 minute period.

Results and Discussion

Figure 3 shows the real part of the electric field E_r excited from one end of the waveguide without a phantom. Its radiation pattern is visualized in the Modeling Instructions section.

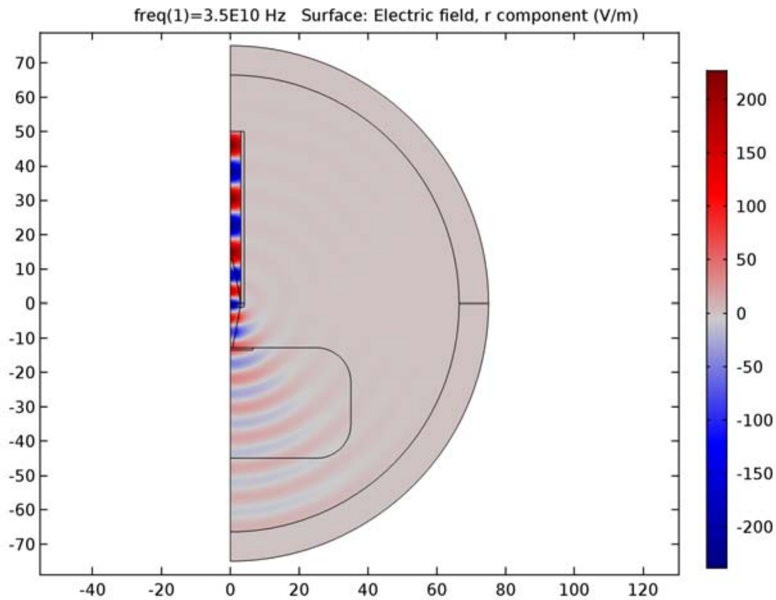


Figure 3: Wave propagation from the dielectric rod plotted with the E_r component of the E-field (probe-only case without a phantom).

Temperature change on the surface of the phantom with the tumor is plotted in Figure 4. Since the input power from the waveguide port is low, 1 mW, the temperature change is within 0.06° even after 10 minutes of millimeter wave exposure. The color difference shows the relatively hotter spot where the temperature is still very close to the initial temperature, 34 °C. Though the temperature analysis for the healthy phantom case is not included, it is easily expected that the temperature variation is less than the case with the tumor because the resistive loss should be lower due to the smaller imaginary part of the permittivity of the healthy skin. So the visualized temperature profile is the worst-case scenario of temperature increase among all three cases. The damaged tissue ratio is visualized in Figure 5. It shows that the effect of the low-power millimeter wave is negligible.

The computed S-parameters indicate more reflection when touching the skin with the tumor due to its higher moisture content, and they are approximately summarized below:

TABLE 2: S-PARAMETER RESPONSE OF THE PROBE

	PROBE ONLY	WITH A HEALTHY PHANTOM	TUMOR ADDED
S_{11}	-29.5 dB	-9.84 dB	-8.97 dB

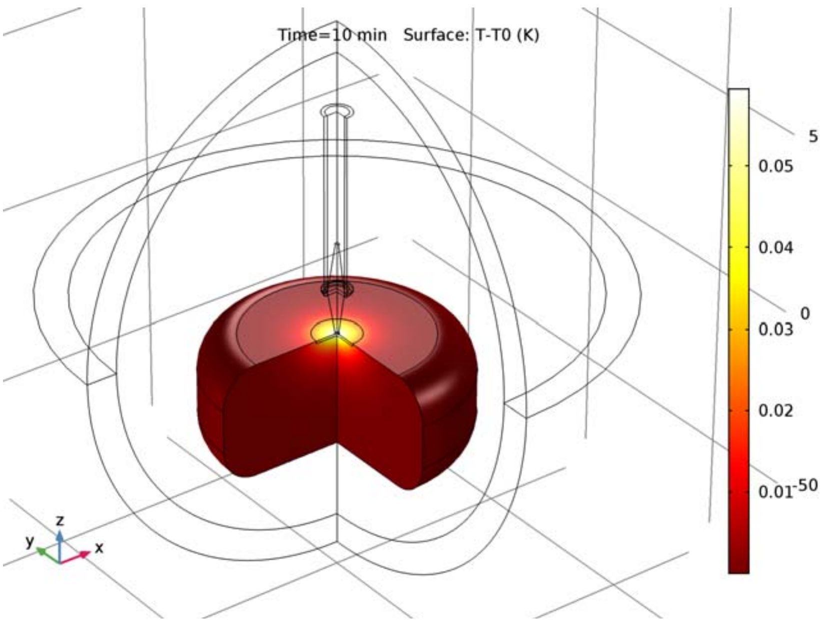


Figure 4: The temperature after 10 minutes. The variation compared to the initial temperature is negligible in the case where the tumor is added at the center of the center top of skin surface.

The modeling instructions show how to access the data plotted in Figure 6 which is not the dependent variable of the Electromagnetic Waves, Frequency Domain, by tweaking the solver settings.

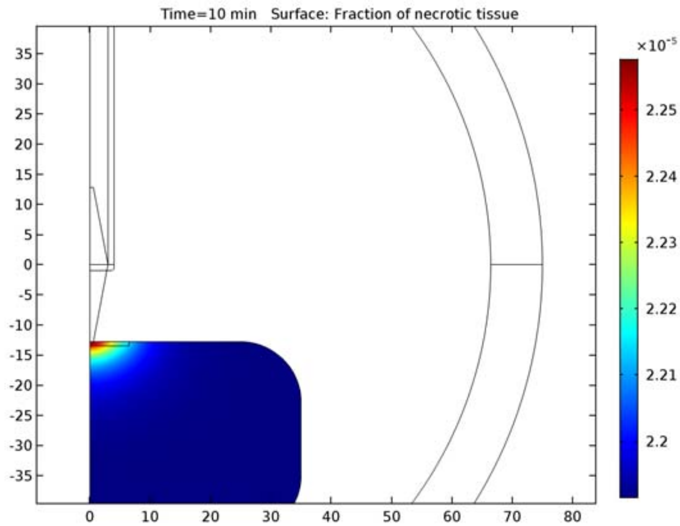


Figure 5: Fraction of necrotic (damaged) tissue is extremely low even after 10 minutes of millimeter wave exposure in the case where the tumor is added at the center of the center top of skin surface.

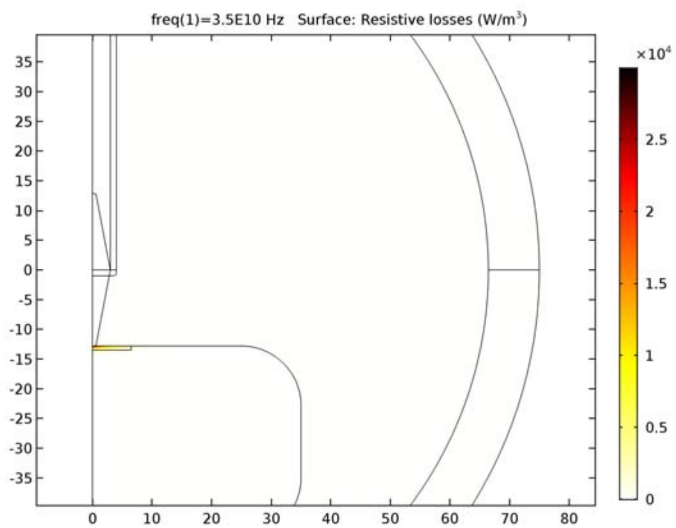


Figure 6: The resistive losses in the case where the tumor is added at the center of the center top of skin surface.

Notes About the COMSOL Implementation

The electromagnetic material properties of skin and tumor at 35 GHz are approximated to show the feasibility of the S-parameter method by detecting the areas with higher moisture content. For any further research, extracting accurate data in the given frequency range is recommended.

Application Library path: `Heat_Transfer_Module/Medical_Technology/conical_dielectric_probe`

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 **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
r1	0.003[m]	0.003 m	Waveguide radius
fc	1.841*c_const/2/pi/r1	2.928E10 1/s	Cutoff frequency
f0	35[GHz]	3.5E10 Hz	Frequency
lda0	c_const/f0	0.0085655 m	Wavelength, free space
l_probe	12.8[mm]	0.0128 m	Tapered probe length
w1_probe	3[mm]	0.003 m	Tapered probe width1
w2_probe	0.58[mm]	5.8E-4 m	Tapered probe width2
T0	34[degC]	307.15 K	Initial skin temperature

GEOMETRY 1

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

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for Circle, locate the **Size and Shape** section.

3 In the **Radius** text field, type 75.

4 In the **Sector angle** text field, type 180.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type 270.

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

Layer name	Thickness (mm)
Layer 1	lda0

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type r1.

4 In the **Height** text field, type `50`.

Rectangle 2 (r2)

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

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

3 In the **Height** text field, type `50`.

4 Locate the **Position** section. In the **r** text field, type `3`.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

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

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **z** to `-l_probe`.

5 In row **2**, set **r** to `w2_probe` and **z** to `-l_probe`.

6 Find the **Added segments** subsection. Click **Add Linear**.

7 Find the **Control points** subsection. In row **2**, set **r** to `w1_probe` and **z** to `0`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **r** to `0`.

10 Find the **Added segments** subsection. Click **Add Linear**.

11 Find the **Control points** subsection. In row **2**, set **z** to `-l_probe`.

Mirror 1 (mir1)

1 Right-click **Bézier Polygon 1 (b1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

3 Select the object **b1** only.

4 In the **Settings** window for Mirror, locate the **Input** section.

5 Select the **Keep input objects** check box.

6 Locate the **Normal Vector to Line of Reflection** section. In the **r** text field, type `0`.

7 In the **z** text field, type `1`.

Rectangle 3 (r3)

1 Right-click **Mirror 1 (mir1)** and choose **Build Selected**.

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

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

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

5 In the **Height** text field, type 1.

6 Locate the **Position** section. In the **z** text field, type -1.

Fillet 1 (fil1)

1 On the **Geometry** toolbar, click **Fillet**.

2 Click the **Select Box** button on the **Graphics** toolbar.

3 On the object **r3**, select Point 2 only.

4 In the **Settings** window for Fillet, locate the **Radius** section.

5 In the **Radius** text field, type 0.5.

Rectangle 4 (r4)

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

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

3 In the **Width** text field, type 35.

4 In the **Height** text field, type 32.2.

5 Locate the **Position** section. In the **z** text field, type -45.

Fillet 2 (fil2)

1 On the **Geometry** toolbar, click **Fillet**.

2 Click the **Select Box** button on the **Graphics** toolbar.

3 On the object **r4**, select Points 2 and 3 only.

4 In the **Settings** window for Fillet, locate the **Radius** section.

5 In the **Radius** text field, type 10.

Rectangle 5 (r5)

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

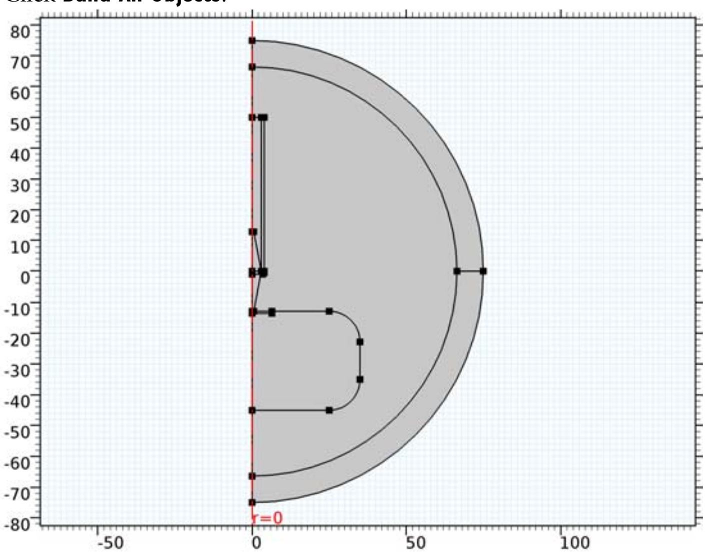
2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type 6.5.

4 In the **Height** text field, type 0.7.

5 Locate the **Position** section. In the **z** text field, type -13.5.

6 Click **Build All Objects**.

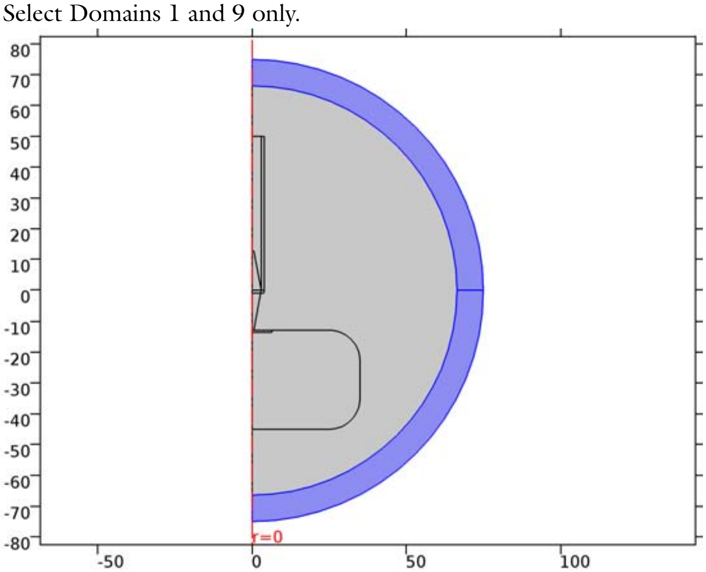


DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 9 only.



ADD MATERIAL

1 On 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>Air**.

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

MATERIALS

Air (mat1)

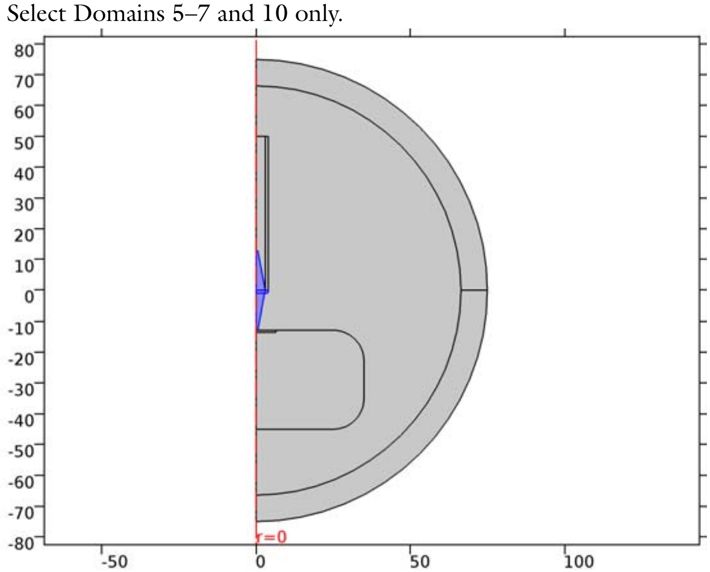
On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Air (mat1)** and choose **Blank Material**.

2 In the **Settings** window for Material, type PTFE in the **Label** text field.

3 Select Domains 5–7 and 10 only.



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

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	2.1	1	Basic

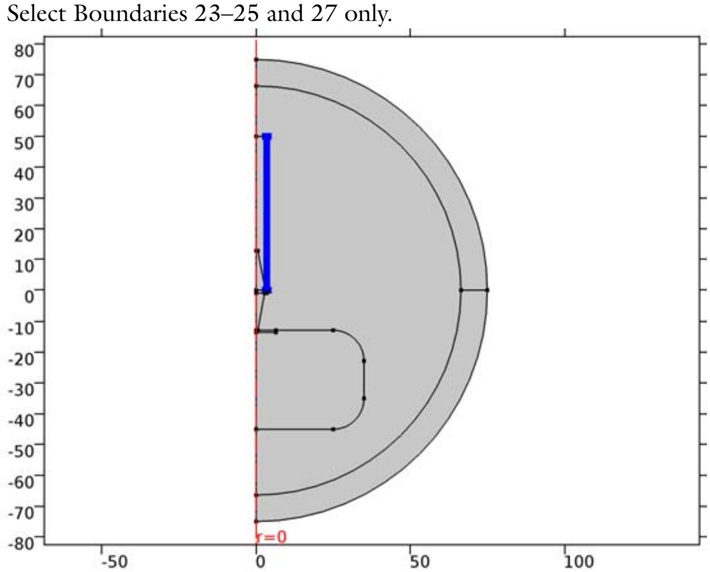
Property	Name	Value	Unit	Property group
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2. In the **Settings** window for Electromagnetic Waves, Frequency Domain, locate the **Out-of-Plane Wave Number** section.
3. In the m text field, type `1`.
4. Locate the **Physics-Controlled Mesh** section. Select the **Enable** check box.
5. In the **Maximum element size in free space** text field, type `lda0/8`.
6. Locate the **Analysis Methodology** section. From the **Methodology options** list, choose **Fast**.

Perfect Electric Conductor 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.
2. Click the **Select Box** button on the **Graphics** toolbar.
3. Select Boundaries 23–25 and 27 only.



Port 1

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

2 Select Boundary 16 only.

3 In the **Settings** window for Port, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Circular**.

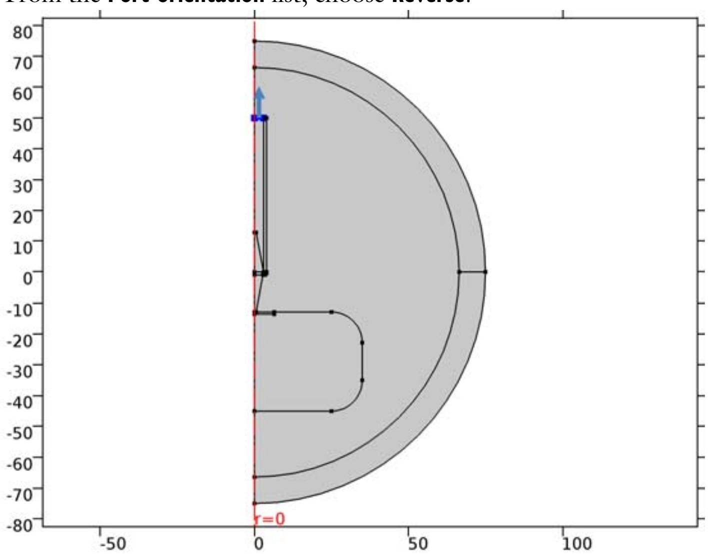
5 From the **Wave excitation at this port** list, choose **On**.

6 In the P_{in} text field, type `1[mW]`.

The input power is `0 dBm`.

7 Select the **Activate slit condition on interior port** check box.

8 From the **Port orientation** list, choose **Reverse**.



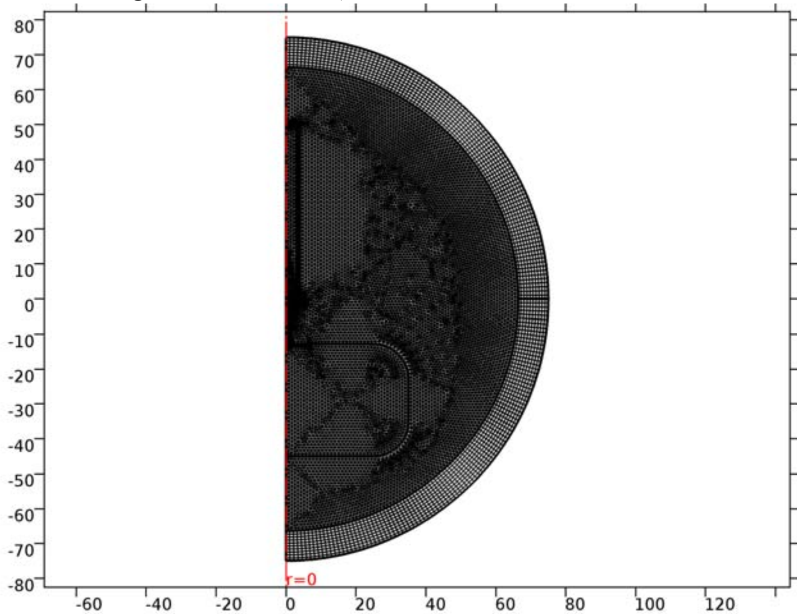
Far-Field Domain 1

On the **Physics** toolbar, click **Domains** and choose **Far-Field Domain**.

MESH 1

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

2 In the **Settings** window for Mesh, click **Build All**.



STUDY 1

Step 1: Frequency Domain

1 In the **Settings** window for Frequency Domain, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `f0`.

3 On the **Home** toolbar, click **Compute**.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Surface**.

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

3 In the **Expression** text field, type `emw.Er`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

5 On the **Electric Field (emw)** toolbar, click **Plot**.

See Figure 3 for the plot of the real part of E_r, showing wave propagation from the input port to the air domain via the tapered dielectric probe.

2D Far Field (emw)

In the **Settings** window for Polar Plot Group, type `Far Field, Polar` in the **Label** text field.

Far Field 1

1 In the **Model Builder** window, expand the **2D Far Field (emw)** node, then click **Results>Far Field, Polar>Far Field 1**.

2 In the **Settings** window for Far Field, locate the **Evaluation** section.

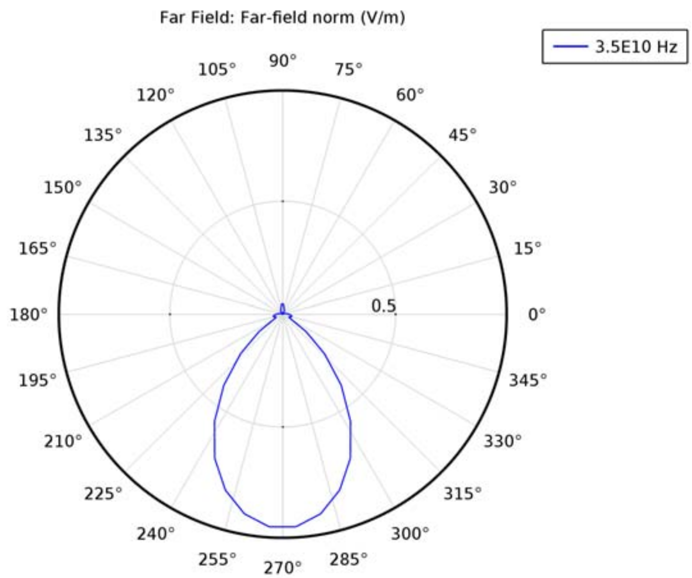
3 Find the **Reference direction** subsection. In the **y** text field, type `1`.

4 In the **z** text field, type `0`.

5 Find the **Normal** subsection. In the **x** text field, type `1`.

6 In the **y** text field, type `0`.

7 On the **Far Field, Polar** toolbar, click **Plot**.

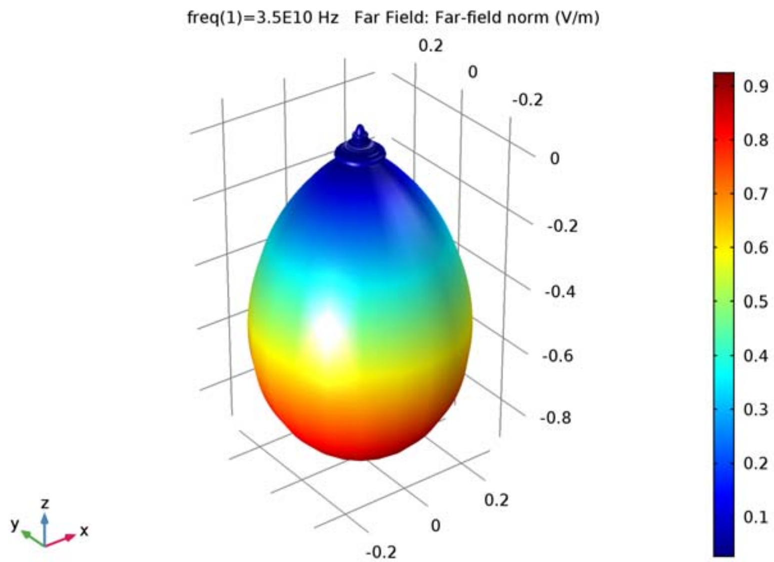


The Far-field pattern on the *yz*-plane shows the radiation from the tapered probe toward the bottom side.

3D Far Field (emw)

1 In the **Model Builder** window, expand the **Results>3D Far Field (emw)** node, then click **3D Far Field (emw)**.

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



The 3D far-field pattern is directed along the z-axis.

S-parameter, S11dB (emw)

1 In the **Model Builder** window, expand the **Results>Derived Values** node, then click **S-parameter, S11dB (emw)**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

The evaluated S-parameter is the input matching property of the circular waveguide without a human body phantom when the dominant mode is excited.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Add another Wave Equation which describe the human body phantom in term of dielectric loss using a complex permittivity.

Wave Equation, Electric 2

1 On the **Physics** toolbar, click **Domains** and choose **Wave Equation, Electric**.

2 Select Domains 3 and 4 only.

3 In the **Settings** window for Wave Equation, Electric, locate the **Electric Displacement Field** section.

4 From the **Electric displacement field model** list, choose **Dielectric loss**.

ADD MATERIAL

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

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

3 In the tree, select **Bioheat>Skin**.

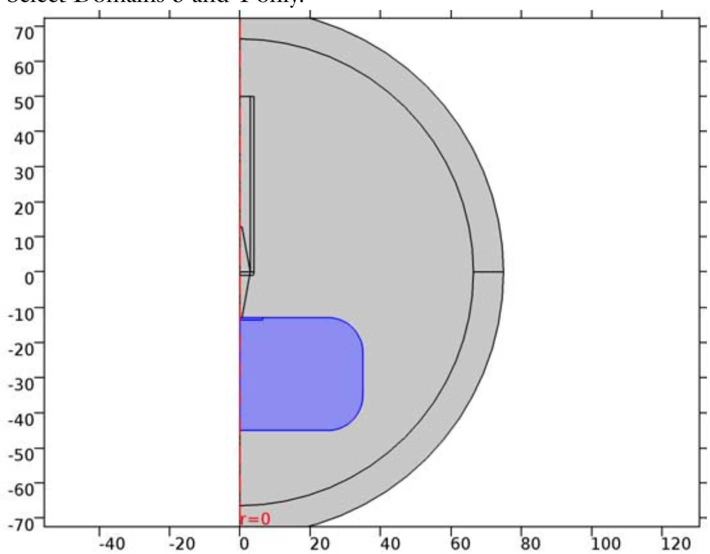
4 Click **Add to Component** in the window toolbar.

MATERIALS

Skin (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Skin (mat3)**.

2 Select Domains 3 and 4 only.



3 In the **Model Builder** window, click **Skin (mat3)**.

4 In the **Settings** window for Material, locate the **Material Contents** section.

5 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity (imaginary part)	epsilonBis	10	1	Dielectric losses
Relative permittivity (real part)	epsilonPrim	5	1	Dielectric losses

Property	Name	Value	Unit	Property group
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

6 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

7 On the **Home** toolbar, click **Compute**.

RESULTS

S-parameter, S11dB (emw)

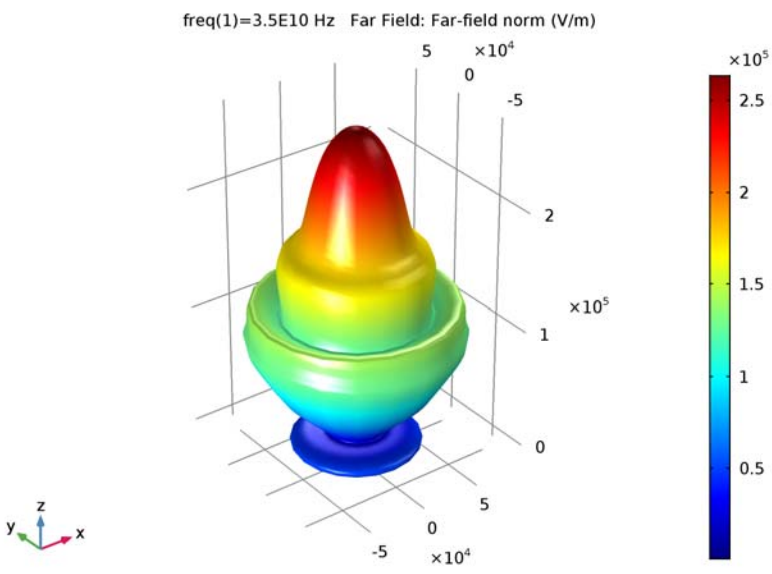
Evaluate the S-parameter assuming the probe is touching a phantom representing a healthy body.

1 In the **Model Builder** window, under **Results>Derived Values** click **S-parameter, S11dB (emw)**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

Far Field, 3D

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



Due to the body, the radiation is reflected back.

MATERIALS

Skin (mat3)

Now, add a tip of tumor skin.

Skin 1 (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Skin (mat3)** and choose **Duplicate**.
2 In the **Settings** window for Material, type `Skin Tumor` in the **Label** text field.
3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.
4 Select Domain 4 only.
5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity (imaginary part)	epsilonBis	15	1	Dielectric losses
Relative permittivity (real part)	epsilonPrim	8	1	Dielectric losses

The effect of millimeter wave radiation on a human body will be investigated using the **Bioheat Transfer** physics interface.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **Heat Transfer>Bioheat Transfer (ht)**.
4 Click **Add to Component** in the window toolbar.

BIOHEAT TRANSFER (HT)

1 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.
2 In the **Model Builder** window, under **Component 1 (comp1)** click **Bioheat Transfer (ht)**.
3 In the **Settings** window for Bioheat Transfer, locate the **Domain Selection** section.
4 Click **Clear Selection**.
5 Select Domains 3 and 4 only.

Biological Tissue 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Biological Tissue 1**.
2 In the **Settings** window for Biological Tissue, locate the **Damaged Tissue** section.
3 Select the **Include damage integral analysis** check box.
4 From the **Damage integral form** list, choose **Energy absorption**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Initial Values 1**.
2 In the **Settings** window for Initial Values, type `T0` in the T text field.

Open Boundary 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.
2 Select Boundaries 4, 8, 19, 29, 30, 37, and 38 only.

MULTIPHYSICS

Electromagnetic Heat Source 1 (emh1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Domain>Electromagnetic Heat Source**.
2 Select Domains 3 and 4 only.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **Custom Studies>Preset Studies for Some Physics Interfaces>Frequency Domain**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Bioheat Transfer (ht)** interface.
5 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Frequency Domain

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.
2 In the **Model Builder** window, under **Study 2** click **Step 1: Frequency Domain**.
3 In the **Settings** window for Frequency Domain, locate the **Study Settings** section.

4 In the **Frequencies** text field, type `f0`.

Time Dependent

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

Step 2: Time Dependent

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

2 From the **Time unit** list, choose **min**.

3 In the **Times** text field, type `range(0,15[s],10)`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Electromagnetic Waves, Frequency Domain (emw)** interface.

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for Study, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

Solution 2 (sol2)

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

2 In the **Model Builder** window, right-click **Solution 2 (sol2)** and choose **Other>Solution Store**.

3 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Solution Store 1 (sol3)** and choose **Move Up**, three times, so it locates right below **Stationary Solver 1**. This allows to access the solution more than the dependent variable in **Frequency Domain** study step.

RESULTS

Revolution 2D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

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

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Click to expand the **Revolution layers** section. Locate the **Revolution Layers** section. In the **Start angle** text field, type `-90`.

5 In the **Revolution angle** text field, type `270`.

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

S-parameter, S11dB (emw)

1 In the **Model Builder** window, expand the **Results** node, then click **Derived Values>S-parameter, S11dB (emw)**.

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

3 From the **Data set** list, choose **Study 2/Solution Store 1 (sol3)**.

4 Click **Evaluate**.

The computed S-parameter shows more reflection on the probe due to the skin tumor.

3D Plot Group 4

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Revolution 2D 2**.

4 From the **Time (min)** list, choose **10**.

Surface 1

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

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

3 In the **Expression** text field, type `T-T0`.

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

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

6 Click the **Zoom In** button on the **Graphics** toolbar.

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

The temperature variation in the skin is shown in Figure 4.

2D Plot Group 5

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Time (min)** list, choose **10**.

Surface 1

1 Right-click **Fraction of Necrotic Tissue** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Bioheat Transfer>Biological tissue>ht.theta_d - Fraction of necrotic tissue**.

3 On the **Fraction of Necrotic Tissue** toolbar, click **Plot**.

4 Click the **Zoom In** button on the **Graphics** toolbar.

The reproduced plot addresses the fraction of necrotic tissue as shown in Figure 5.

2D Plot Group 6

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

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

Resistive Losses

1 Right-click **Resistive Losses** and choose **Surface**.

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

3 From the **Data set** list, choose **Study 2/Solution Store 1 (sol3)**.

Surface 1

1 In the **Model Builder** window, under **Results>Resistive Losses** click **Surface 1**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qrh - Resistive losses**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

4 Select the **Reverse color table** check box.

5 On the **Resistive Losses** toolbar, click **Plot**.

Finish the result analysis by regenerating Figure 6, the resistive losses plot.

Contact Switch

Introduction

A contact switch is used to regulate whether or not an electrical current is passing from a power source into an electrical device. These switches are found in many types of equipment and they are used to control, for example, the power output from a wall socket into a device when it is plugged in; the currents passing across the circuit board of a computer; or the electricity powering a light bulb when the switch is flipped on. Because of their prevalence, simulating contact switches is a fundamental step in designing electronic applications.

The working principle behind a contact switch is simple: two conductive pieces of metal with an electrical voltage difference across them are brought into contact, allowing a current to flow between them. The metallic surfaces of the two components that touch one another are called contacts, and when the connection between the two contacts is broken, the current stops flowing.

The current flow between the two contacts contributes to an increase in temperature in the switch due to the Joule heating effect.

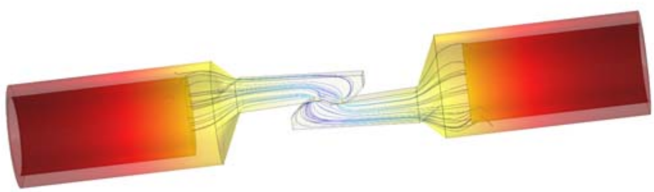


Figure 1: A contact switch.

The heating of the contact switch can change the material properties of the metal as well as the surface area of contact, and therefore is an important effect to consider when modeling the switch. Letting the temperature become too high can even cause the switch to burn out, meaning the switch is no longer functional. Therefore, it is important to analyze its current-carrying capability in order to prevent this from happening. It is also important to consider that when the two metallic pieces come into contact, the surfaces touching each other experience a mechanical pressure or contact pressure. This mechanical pressure on the contacts can alter the electrical and thermal properties of the material locally around the region surrounding the contacts. Therefore, in order to accurately simulate the current-carrying capability and temperature rise in the switch, it is important to take a more comprehensive approach in the simulation and incorporate the effect of contact pressure to compute the electrical and thermal conductance of the contact surfaces.This tutorial illustrates how to implement a multiphysics contact. It models the thermal and electrical behavior of two contacting parts of a switch. The electric current and the heat cross from one part to the other only at the contact surface.

The contact switch device has a cylindrical body and plate hook shapes at the contact area (see Figure 1). There, the thermal and electrical apparent resistances are coupled to the mechanical contact pressure at the interface, which the application solves for.

The initial temperature is equal to the external room temperature. A potential difference between the left and right parts leads to heating through the Joule effect.

Model Definition

The geometry of the switch is shown in Figure 2. Only half of the device is represented due to symmetry considerations.

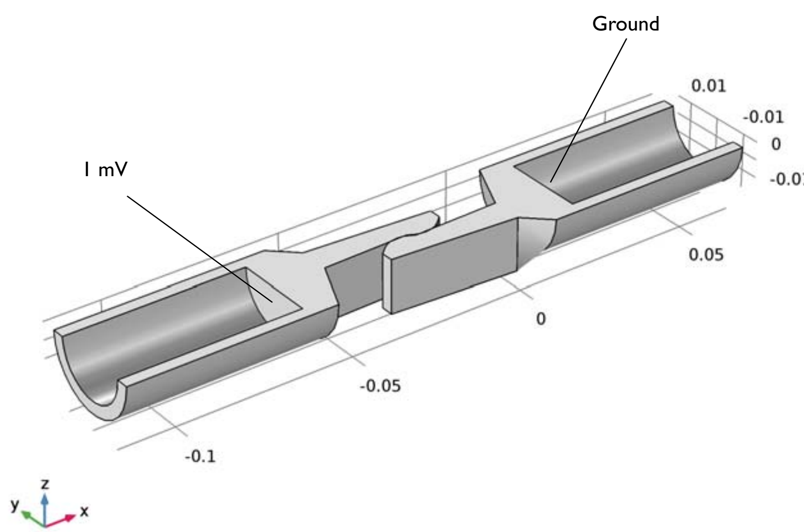


Figure 2: Switch geometry.

The switch is made of copper, with two fixed cylindrical elements and a central region where the contacts are located. On the end of each contact are plate hooks that enable contact between the two pieces. In the simulation, an electric potential of 1 mV is applied to the left side of the switch, while the right side is grounded.

The thermal and electrical contact conductances are assumed to be related only to the contact pressure.

The exposed surfaces of the switch lose heat due to their interaction with air via natural convection. In the simulation, this is modeled by specifying a heat transfer coefficient and the ambient temperature of the surrounding air (a more ambitious simulation might also include the fluid flow of the air). The application first solves for structural contact to obtain the contact pressure on the contact surfaces. These results are then used to compute the electrical and thermal conductance of the contact's surfaces in a Joule heating simulation.

Results and Discussion

Figure 3 shows the electric potential distribution, ranging from the grounded right side to the applied 1 mV on the left.

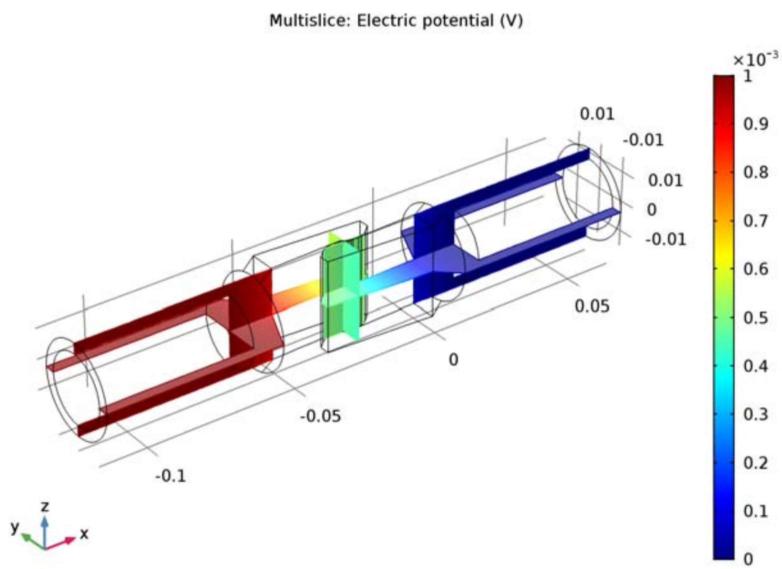


Figure 3: Electric potential profile.

A potential difference across the two components in the switch creates a current flow, which in turn leads to Joule heating. This causes a rise in temperature in the switch. If you leave the switch on for a while, temperature distribution in the switch reaches an equilibrium. Figure 4 shows the temperature distribution in the contact switch. In this example, Joule heating causes the temperature in the switch to rise about 5 K above room temperature, although only a small temperature variation is seen within the switch itself.

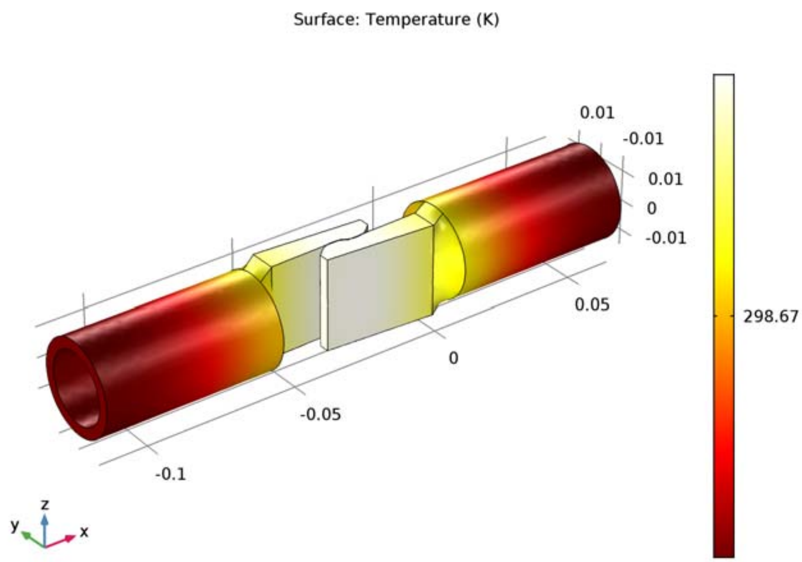


Figure 4: Temperature distribution.

The internal temperature distribution is almost constant. Introducing the effect of electrical and thermal conductance allows us to predict the temperature rise more accurately. The simulation also shows that the switch gets slightly hotter at the contact region.

Finally, Figure 5 plots the temperature distribution at the contact region. Streamlines show the current density.

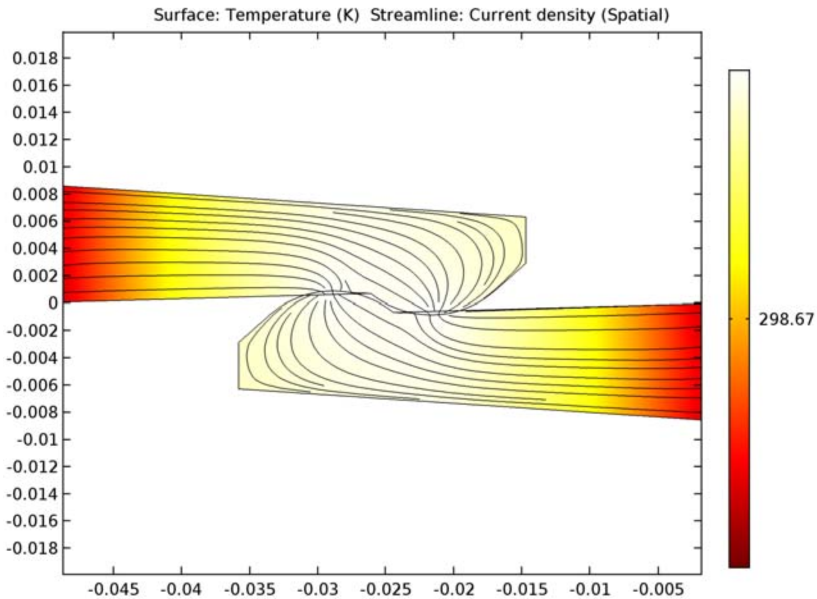


Figure 5: Temperature distribution (volume plot) and current density (streamlines) at the contact region.

Application Library path: `Heat_Transfer_Module/Thermal_Contact_and_Friction/contact_switch`

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 **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer>Electromagnetic Heating>Joule Heating**.

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

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

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

3 Click **Browse**.

4 Browse to the application's Application Libraries folder and double-click the file `contact_switch.mphbin`.

5 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 From the **Pair type** list, choose **Contact pair**.

5 Clear the **Create pairs** check box.

6 On the **Home** toolbar, click **Build All**.

DEFINITIONS

Contact Pair 1 (p1)

1 On the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundaries 12 and 15 only.

3 In the **Settings** window for Pair, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundaries 25 and 28 only.

ADD MATERIAL

1 On 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>Copper**.

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

MATERIALS

Copper (mat1)

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

2 Select Boundaries 4, 5, 34, and 35 only.

Symmetry 1

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

2 Select Boundaries 2 and 22 only.

Contact 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Contact**.

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

3 In the **Pairs** list, select **Contact Pair 1 (p1)**.

4 Locate the **Initial Values** section. In the T_n text field, type `1e7`.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Heat Flux 1

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

2 Select Boundaries 3 and 6–33 only.

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

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

5 In the h text field, type `2`.

Symmetry 1

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

2 Select Boundaries 2 and 22 only.

Pair Thermal Contact 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Pair Thermal Contact**.

2 In the **Settings** window for Pair Thermal Contact, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 1 (p1)**.

4 Locate the **Contact Surface Properties** section. From the p list, choose **Contact pressure (solid/cnt1)**.

ELECTRIC CURRENTS (EC)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

Ground 1

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

2 Select Boundary 34 only.

Electric Potential 1

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

2 Select Boundary 5 only.

3 In the **Settings** window for Electric Potential, locate the **Electric Potential** section.

4 In the V_0 text field, type `1[mV]`.

Pair Electrical Contact 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Pair Electrical Contact**.

2 In the **Settings** window for Pair Electrical Contact, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 1 (p1)**.

4 Locate the **Contact Surface Properties** section. From the p list, choose **Contact pressure (solid/cnt1)**.

MESH 1

On the **Mesh** toolbar, click **Free Tetrahedral**.

Free Tetrahedral 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

Size 1

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

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

3 Select Boundaries 12, 15, 25, and 28 only.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `5e-4`.

Size

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

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

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

4 Click **Build All**.

STUDY 1

Solve the model in two steps. The first step only computes for **Solid Mechanics** while the second solves for Joule Heating (**Electric Currents** and **Heat Transfer in Solids**).

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 box for **Electric Currents (ec)** and **Heat Transfer in Solids (ht)**.

Stationary 2

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

Step 2: Stationary 2

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

2 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

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

RESULTS

Stress (solid)

In this first default plot, the switch is slightly deformed due to the contact pressure. The von Mises stress is located at the switch base and at the contact area.

Follow the next steps to visualize the second and third default plots as in Figure 3 and Figure 4.

Mirror 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

2 In the **Settings** window for Mirror 3D, locate the **Plane Data** section.

3 From the **Plane** list, choose **XY-planes**.

Electric Potential (ec)

1 In the **Model Builder** window, under **Results** click **Electric Potential (ec)**.

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

3 From the **Data set** list, choose **Mirror 3D 1**.

4 On the **Electric Potential (ec)** toolbar, click **Plot**.

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 **Data set** list, choose **Mirror 3D 1**.

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

Data Sets

To observe the temperature and current density only at the contact region (Figure 5), proceed as follows.

Surface 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Surface**.

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

3 From the **x- and y-axes** list, choose **XY-plane**.

4 Select Boundaries 10 and 21 only.

2D Plot Group 5

1 On the **Results** toolbar, click **2D Plot Group**.

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

3 On the **Temperature (Contact Region)** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Temperature (Contact Region)** click **Surface 1**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Temperature>T - Temperature**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

Temperature (Contact Region)

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

2 On the **Temperature (Contact Region)** toolbar, click **Streamline**.

Streamline 1

1 In the **Model Builder** window, under **Results>Temperature (Contact Region)** click **Streamline 1**.

2 In the **Settings** window for Streamline, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electric Currents>Currents and charge>ec.Jx,ec.Jy - Current density (Spatial)**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

4 In the **Separating distance** text field, type `0.02`.

5 On the **Temperature (Contact Region)** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Continuous Casting

Introduction

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

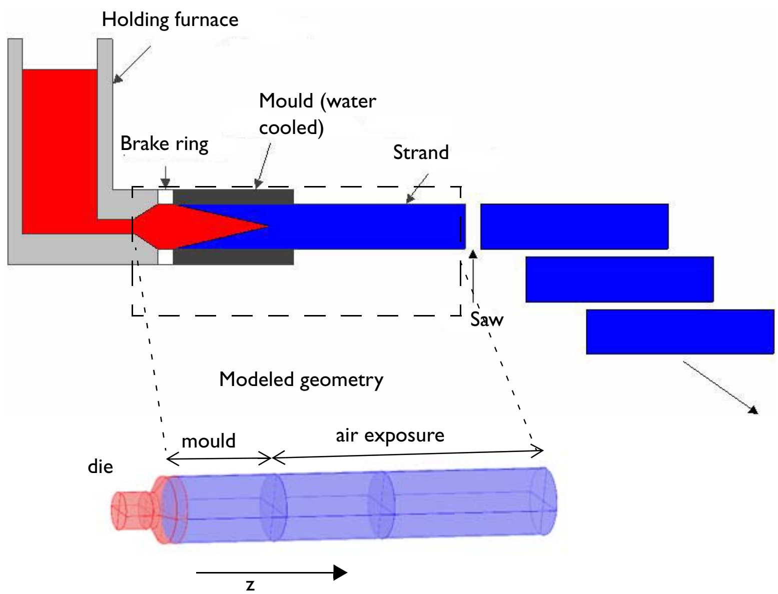


Figure 1: Continuous metal-casting process with a view of the modeled section.

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

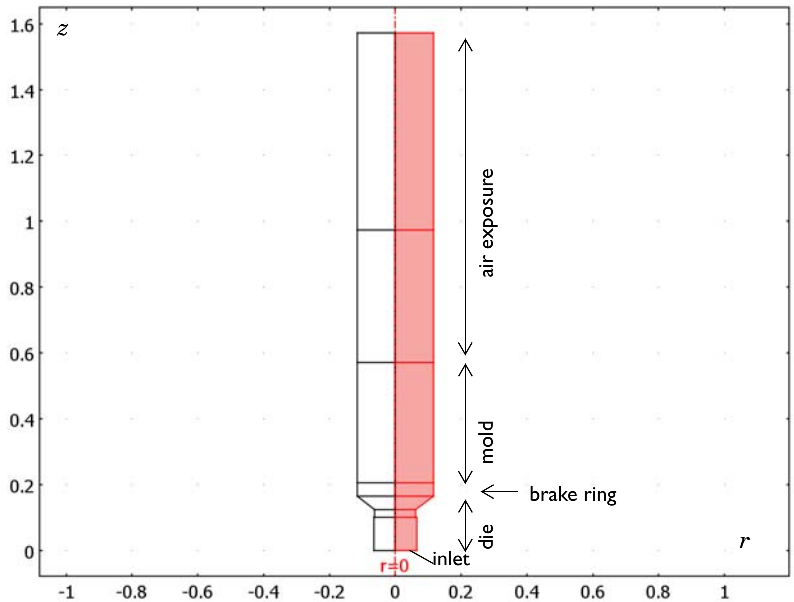


Figure 2: 2D axisymmetric model of the casting process.

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

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

Model Definition

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

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

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

As the melt cools down in the mold, it solidifies. During the phase transition, a significant amount of latent heat is released. The total amount of heat released per unit mass of alloy during the transition is given by the change in enthalpy, ΔH. In addition, the specific heat capacity, C_p, also changes considerably during the transition. The difference in specific heat before and after transition can be approximated by

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

As opposed to pure metals, an alloy generally undergoes a broad temperature transition zone, over several kelvin, in which a mixture of both solid and molten material co-exist in a "mushy" zone. To account for the latent heat related to the phase transition, the Apparent Heat capacity method is used through the Heat Transfer with Phase Change domain condition. The half-width of the transition interval, ΔT, is set to 10 K in this case, and represents half the transition temperature span.

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

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

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

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

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

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

Table 1 reviews the material properties in this model.

TABLE 1: MATERIAL PROPERTIES

PROPERTY	SYMBOL	MELT	SOLID
Density	ρ (kg/m^3)	8500	8500
Heat capacity at constant pressure	C_p (J/(kg·K))	530	380
Thermal conductivity	k (W/(m·K))	200	200
Dynamic viscosity	μ (Ns/m^2)	0.0434	-

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

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

Results and Discussion

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

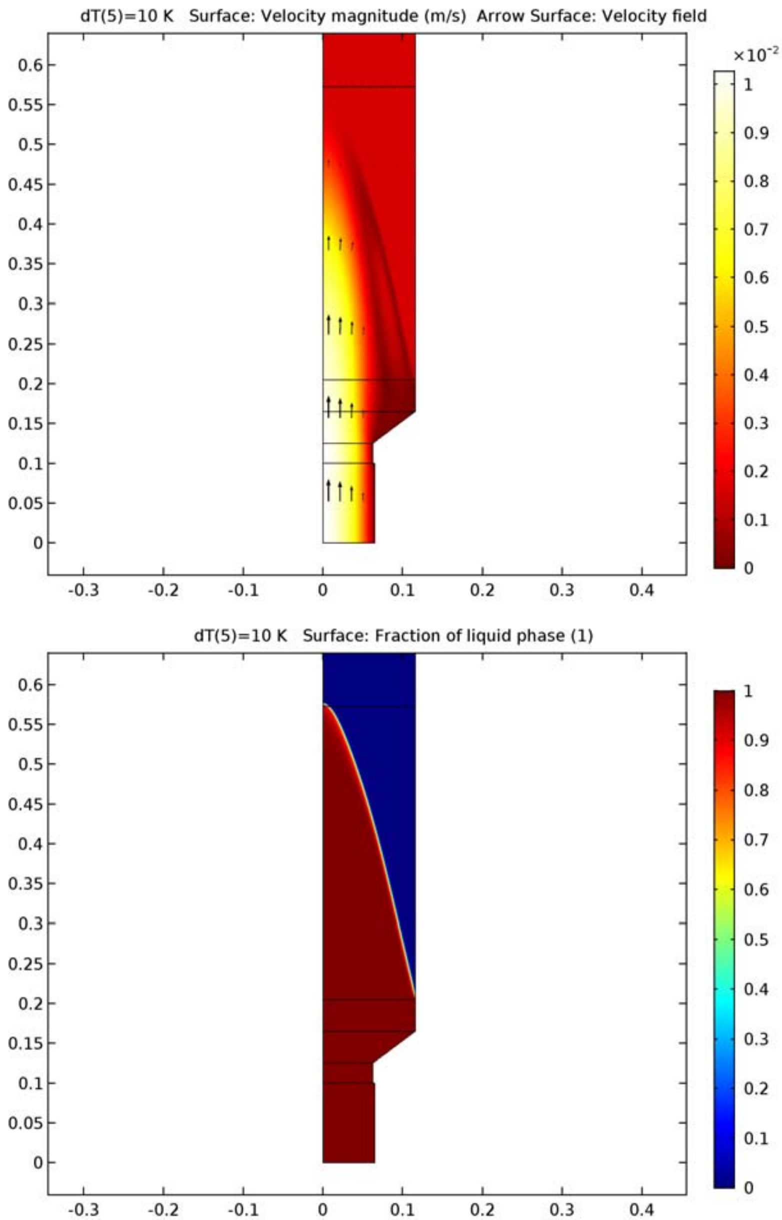


Figure 3: Temperature distribution (top) and fraction of liquid phase (bottom) in the lower part of the cast at a casting rate of 1.6 mm/s.

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

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

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

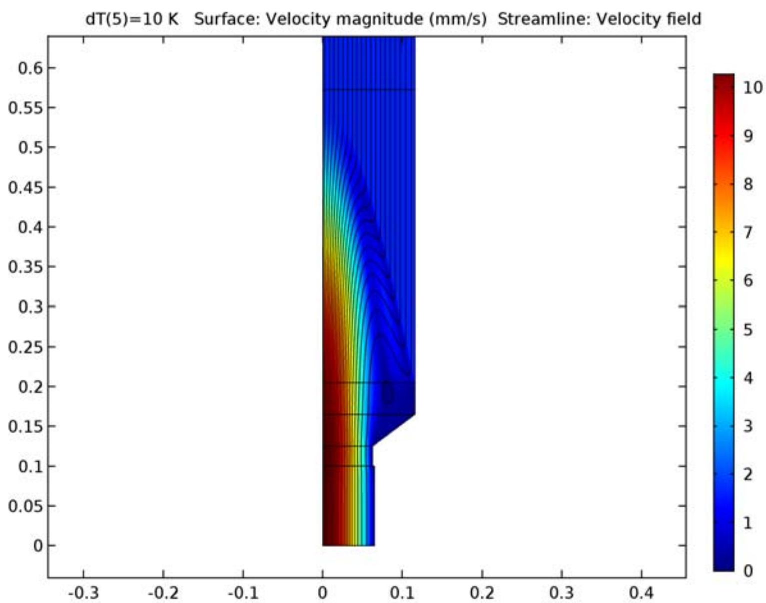


Figure 4: Velocity field with streamlines in the lower part of the process.

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

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

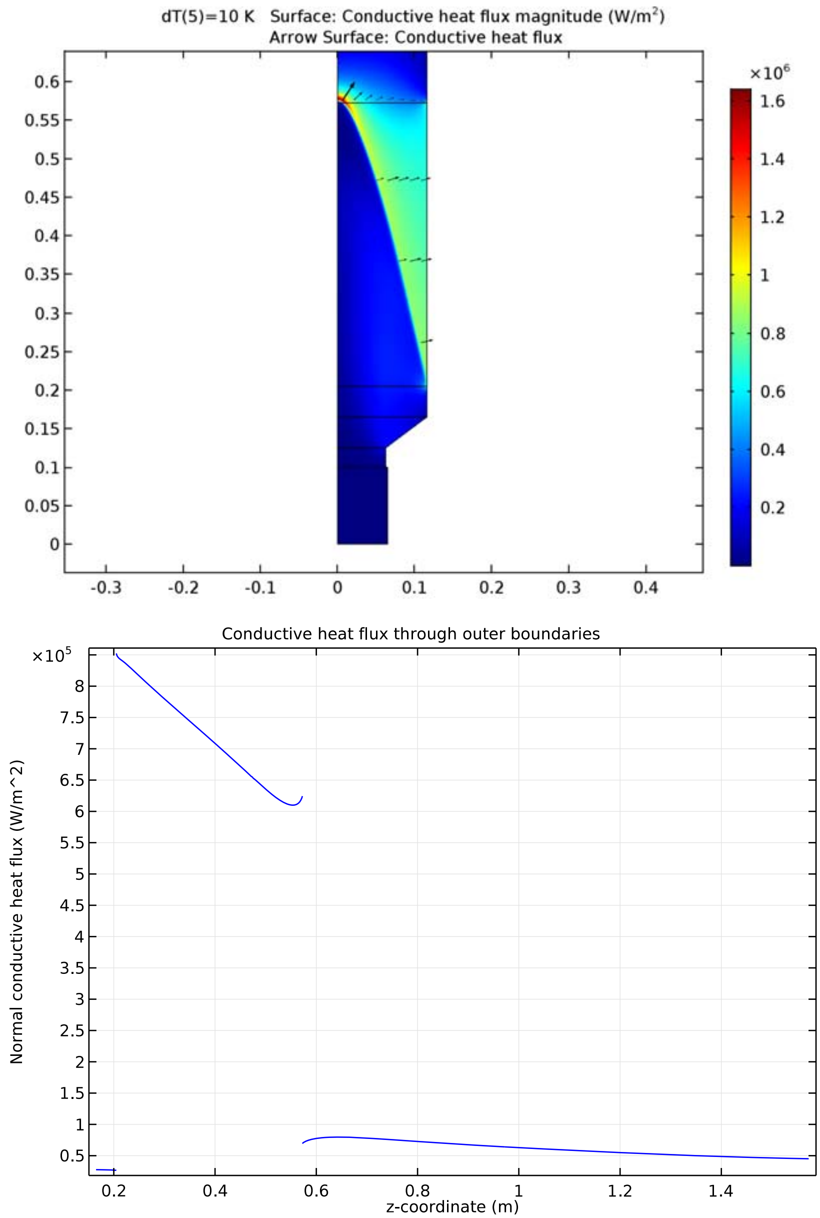


Figure 5: The cooling viewed as conductive heat flux in the domains (top), and through the outer boundary (the cooling zones) after the die (bottom).

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

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

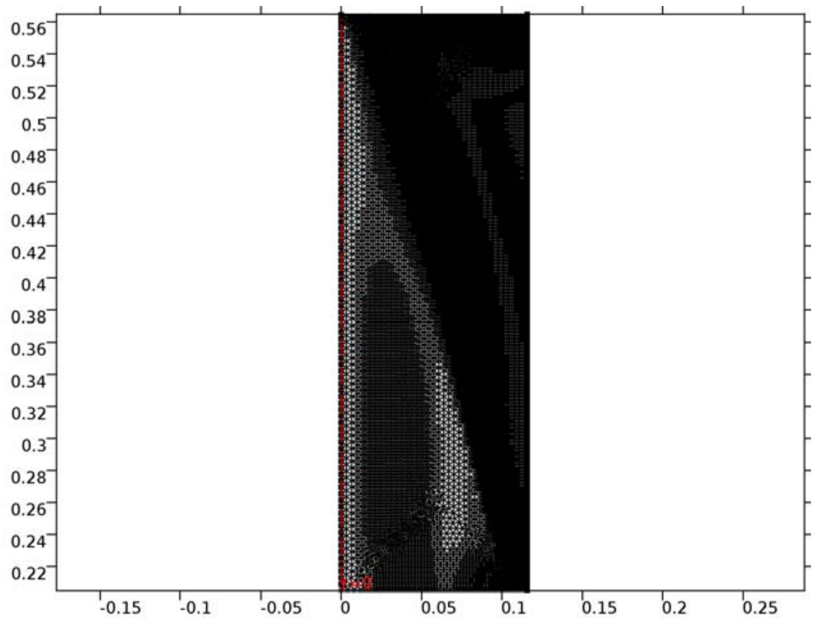


Figure 6: Close-up of the final computational mesh, resulting from the built-in adaptive technique.

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

Reference

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

Application Library path: `Heat_Transfer_Module/Thermal_Processing/continuous_casting`

Modeling Instructions

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

NEW

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>Non-Isothermal Flow>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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

3 Click **Load from File**.

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

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

DEFINITIONS

Variables 1

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

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

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

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

Define the variables by loading the corresponding text file provided.

5 Locate the **Variables** section. Click **Load from File**.

6 Browse to the application's Application Libraries folder and double-click the file `continuous_casting_variables.txt`.

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.065`.

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

5 On the **Geometry** toolbar, click **Build All**.

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `0.0625`.

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

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

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

Rectangle 3 (r3)

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

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

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

4 In the **Height** text field, type `0.04`.

5 Locate the **Position** section. In the **z** text field, type `0.165`.

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

Rectangle 4 (r4)

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

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

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

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

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

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

Rectangle 5 (r5)

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

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

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

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

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

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

Rectangle 6 (r6)

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

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

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

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

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

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

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

Polygon 1 (pol1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Polygon**.

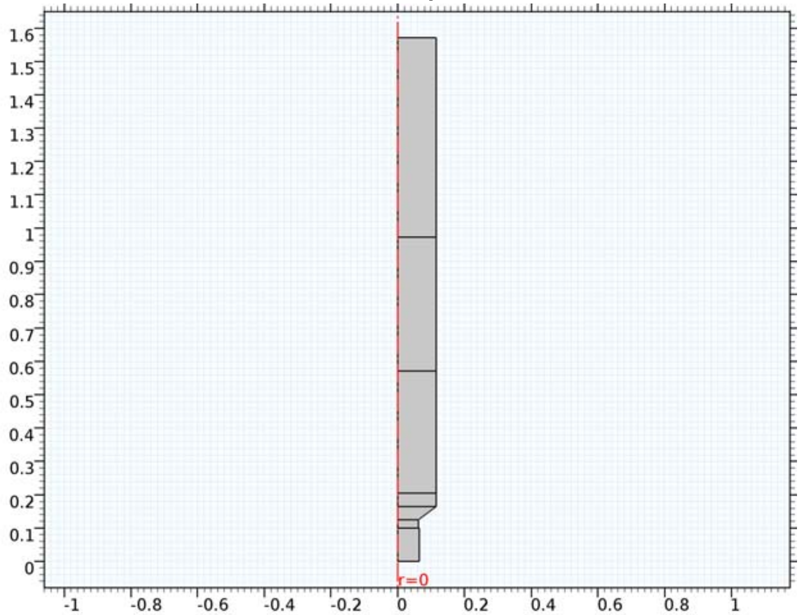
2 In the **Settings** window for Polygon, locate the **Coordinates** section.

3 In the **r** text field, type `0 0 0.11575 0.0625 0`.

4 In the **z** text field, type `0.125 0.165 0.165 0.125 0.125`.

5 On the **Geometry** toolbar, click **Build All**.

6 Click the **Zoom Extents** button on 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.

Material 1 (mat1)

1 On 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	**Name**	**Value**	**Unit**	**Property group**
Dynamic viscosity	mu	0.0434	Pa·s	Basic
Thermal conductivity	k	200	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic

Property	Name	Value	Unit	Property group
Heat capacity at constant pressure	Cp	`Cp_s`	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic

Material 2 (mat2)

1 On the **Materials** toolbar, click **Blank Material**.

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

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

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

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

LAMINAR FLOW (SPF)

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

2 Select Boundary 2 only.

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

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

Outlet 1

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

2 Select Boundaries 15 and 21–23 only.

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

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

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

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

0	r
v_cast	z

Volume Force 1

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

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

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

4 Locate the **Volume Force** section. Specify the $\mathbf{F}$ vector as

Fr	r
Fz	z

HEAT TRANSFER IN FLUIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

2 In the **Settings** window for Heat Transfer in Fluids, locate the **Ambient Settings** 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.

Phase Change Material 1

1 On the **Physics** toolbar, click **Domains** and choose **Phase Change Material**.

2 In the **Settings** window for Phase Change Material, locate the **Domain Selection** section.

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

4 Locate the **Phase Change** section. In the $T_{\mathrm{pc},\,1 \rightarrow 2}$ text field, type T_m.

5 In the $\Delta T_{1 \rightarrow 2}$ text field, type `2*dT`.

The parameter `dT` is multiplied by 2 because it is only the half width of the phase change interval.

6 In the $L_{1 \rightarrow 2}$ text field, type `dH`.

7 Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Solid Metal Alloy (mat1)**.

8 Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Liquid Metal Alloy (mat2)**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `T_in` in the T text field.

Temperature 1

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

2 Select Boundary 2 only.

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

4 In the T_0 text field, type `T_in`.

Heat Flux 1

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

2 Select Boundary 20 only.

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

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

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

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

Heat Flux 2

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

2 Select Boundary 21 only.

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

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

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

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

Heat Flux 3

1. On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. Select Boundaries 22 and 23 only.
3. In the **Settings** window for Heat Flux, locate the **Heat Flux** section.
4. Click the **Convective heat flux** button.
5. In the h text field, type `h_air`.
6. From the T_{ext} list, choose **Ambient temperature (ht)**.

Outflow 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2. Select Boundary 15 only.

Diffuse Surface 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.
2. Select Boundaries 22 and 23 only.
3. In the **Settings** window for Diffuse Surface, locate the **Surface Emissivity** section.
4. From the ε list, choose **User defined**. In the associated text field, type `eps_s`.
5. Locate the **Ambient** section. From the T_{amb} list, choose **Ambient temperature (ht)**.

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for Mesh, locate the **Mesh Settings** section.
3. From the **Element size** list, choose **Finer**.
4. On the **Mesh** toolbar, click **Edit**.

Boundary Layers 1

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

   Deleting the boundary layers is necessary in order to use the adaptive mesh functionality.

Size 1

1. Select Boundaries 16–21 only.
2. In the **Settings** window for Size, locate the **Element Size** section.
3. From the **Predefined** list, choose **Fine**.

4 In the **Model Builder** window, click **Mesh 1**.

5 In the **Settings** window for Mesh, click **Build All**.

STUDY 1

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

1 In the **Settings** window for Study, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

   The default plots are disabled from this study because they will be added from the last study.

Step 1: Stationary

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

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

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

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT	300 100 50 30	K

Stationary 2

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

Step 2: Stationary 2

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

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

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

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

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

Since the fluid flow and heat transfer interfaces are strongly coupled, using a fully coupled solver is expected to be faster. In the following steps, alter the default solver sequence into using the **Fully Coupled** attribute.

Solution 1 (sol1)

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

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

3 Right-click **Stationary Solver 1** and choose **Fully Coupled**.

4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** and choose **Fully Coupled**.

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

MESH 2

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

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

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

The mesh should look like that in Figure 6.

Add a second study for the second parametric study step.

ADD STUDY

1 On the **Study** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

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

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

STUDY 2

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

Step 1: Stationary

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

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

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

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

5 From the **Study** list, choose **Study 1, Stationary 2**.

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

7 Click **Add**.

8 In the table, enter the following settings:

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

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

Again, a fully coupled solver is more robust for this model. Tweak the solver sequence accordingly with the instructions below.

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 4 (sol4)** node.

4 Right-click **Stationary Solver 1** and choose **Fully Coupled**.

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

RESULTS

Velocity (spf)

To reproduce the plot in Figure 4, 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)

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

2 On the **Velocity (spf)** toolbar, click **Streamline**.

Streamline 1

1 In the **Model Builder** window, under **Results>Velocity (spf)** click **Streamline 1**.

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

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

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

Pressure (spf)

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

Velocity (spf) 1

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

Temperature, 3D (ht)

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

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

2D Plot Group 6

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

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

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

4 On the **Fraction of Liquid Phase** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Fraction of Liquid Phase** click **Surface 1**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Definitions> Variables>alpha - Fraction of liquid phase**.

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

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

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

2D Plot Group 7

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

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

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

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

Surface 1

1 In the **Model Builder** window, under **Results>Temperature** click **Surface 1**.

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

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

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

Temperature

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

2 Click **Arrow Surface**.

Arrow Surface 1

1 In the **Model Builder** window, under **Results>Temperature** click **Arrow Surface 1**.

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

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

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

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

Proceed to reproduce the heat flux plots shown in Figure 5.

2D Plot Group 8

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

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

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

4 On the **Conductive Heat Flux** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Conductive Heat Flux** click **Surface 1**.

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

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

Conductive Heat Flux

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

2 Click **Arrow Surface**.

Arrow Surface 1

1 In the **Model Builder** window, under **Results>Conductive Heat Flux** click **Arrow Surface 1**.

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

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

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

5 On 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.

1D Plot Group 9

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

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

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

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

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

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

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

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

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

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

Line Graph 1

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

2 Select Boundaries 20–23 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 **Model>Component 1>Heat Transfer in Fluids>Boundary fluxes>ht.ndflux - Normal conductive heat flux**.

4 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry>Coordinate>z - z-coordinate**.

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

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

Compare the result with the lower plot of Figure 5.

Data Sets

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

Cut Line 2D 1

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

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

3 In row **Point 1**, set **r** to `0.045`, and **z** to `0.42`.

4 In row **Point 2**, set **r** to `0.085`, and **z** to `0.43`.

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

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

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

1D Plot Group 10

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

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

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

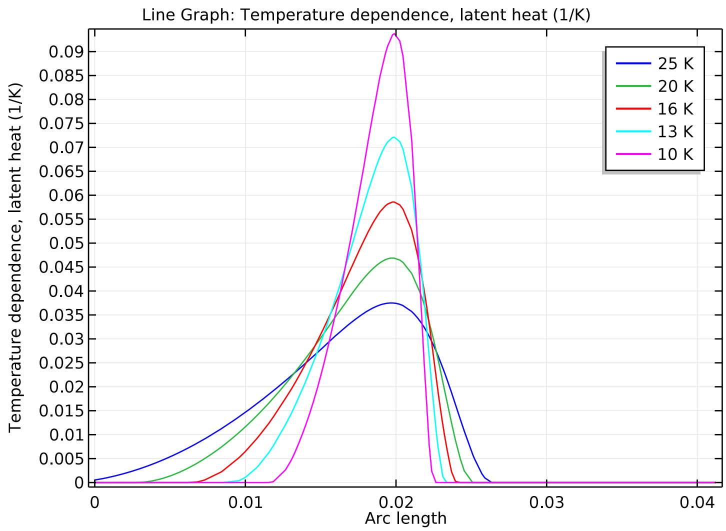
Line Graph 1

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

2 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Definitions>Variables>D - Temperature dependence, latent heat**.

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

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



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

Created in COMSOL Multiphysics 5.2a

Temperature Field in a Cooling Flange

Introduction

Chemical reaction fluids can be cooled using glass flanges. The reaction fluid is passed through the flange and the air surrounding the flange then serves as the coolant. Engineers looking to optimize the cooling performance of such flanges can look to simulation for help.

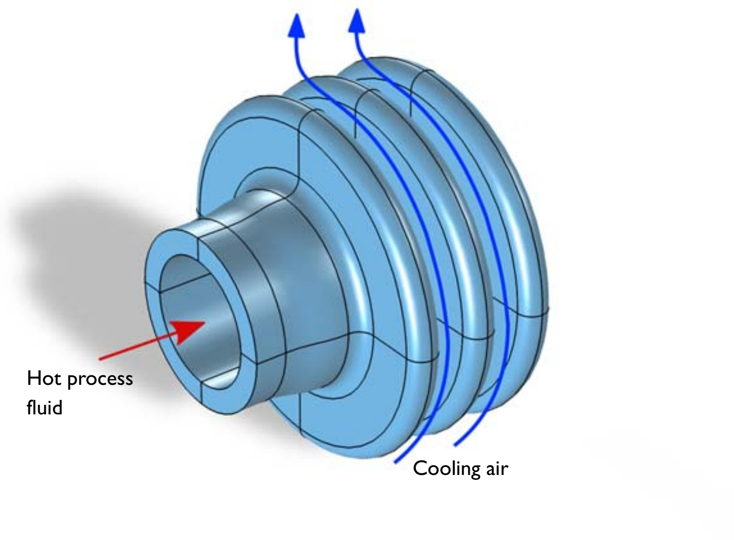


Figure 1: Operating principle of the cooling flange.

First off, the physics to analyze in this case is heat transfer, involving both convection and conduction. Heat transfer occurs via convection to and from the surfaces of the flange, while the flange's interior experiences conduction. In other words, the inside of the tube is heated by the process fluid, and the flange then conducts the heat and transfers it into the air, in turn causing a convective flow due to buoyancy effects. The cooling performance of the flange therefore depends on both convection and conduction.

To simplify the simulation convection cooling is analyzed using the heat transfer coefficient, h. Since this describes the fluid-flow field influence and the convective fluxes, the flow field doesn't have to be computed.

This particular example uses external research data (Ref. 1) for the outer surface heat transfer coefficient, which was originally obtained through semi-empirical data for natural convection around a cylinder. For the tube-facing surface a coefficient for forced convection in a tube is used.

Model Definition

Figure 2 presents a drawing of the modeled geometry.

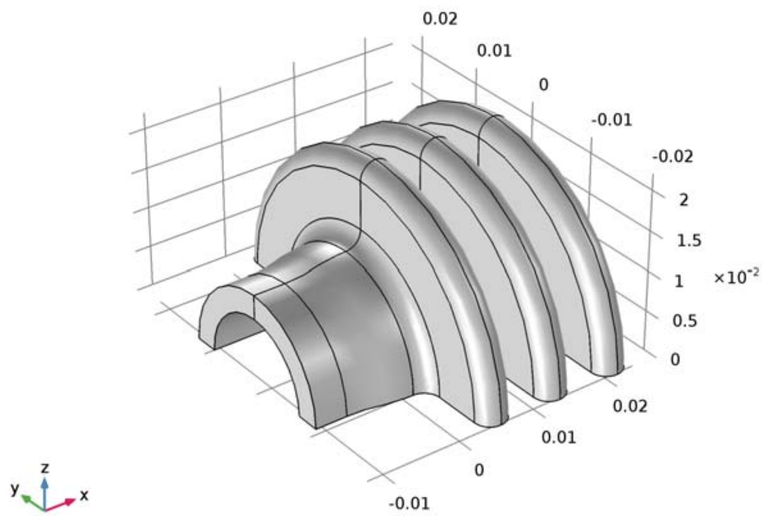


Figure 2: Drawing of the cooling flange.

The glass flange consists of a 4 mm thick pipe and 4 mm thick and 10 mm tall flanges, with a connecting pipe that has a 3 mm thick wall and an inner diameter of 16 mm. Note that during the simulation, the pipe diameter varies while the other dimensions remain fixed.

During operation, the hot process fluid heats the inside of the tube. The flange conducts the heat and transfers it to the surrounding air. As the air is heated, buoyancy effects cause a convective flow.

The heat transfer within the flange is described by the stationary heat equation

$$\nabla \cdot (-k\nabla T) = 0$$

where k is the thermal conductivity (W/(m·K)), and T is the temperature (K). On the flange's exterior boundaries, which face the air and process fluid, the applicable boundary condition is

$$-\mathbf{n} \cdot (-k\nabla T) = h(T_{\mathrm{inf}} - T)$$

where $\mathbf{n}$ is the normal vector of the boundary, h, is the heat transfer coefficient (W/(m^2·K)), and T_{inf} is the temperature of the surrounding medium (K). For this simulation, set T_{inf} to 298 K for the cooling air and to 363 K for the process fluid.

You can approximate the value for the heat transfer coefficient, h, on the process fluid side with a constant value of 15 W/(m^2·K) because the fluid's velocity is close to constant and the model assumes that its temperature decreases only slightly.

The h expression on the air side is more elaborate. Assume that the free-convection process around the flange is similar to that around a cylinder. The heat transfer coefficient for a cylinder is available in the literature (Ref. 1), and you can use the expression

$$h = \frac{k}{L} f(\theta) \mathrm{Gr}^{1/4}$$

where k is the thermal conductivity of air (26.2 mW/(m·K) at 298 K); L is the typical length, which in this case is the outer diameter of the flange (44 mm); and $f(\theta)$ is an empirical coefficient tabulated in Table 1 as a function of the incidence angle θ, which is shown in Figure 3. Finally, Gr is the Grashof number defined as

$$\mathrm{Gr} = \frac{g\alpha_p \Delta T L^3}{\nu^2}$$

where α_p is the coefficient of thermal expansion (1/K), which equals that for an ideal gas, g is the gravitational acceleration (9.81 m/s^2), and ν is the kinematic viscosity (18·10^{-6} m^2·s). For the flange material, use silica glass.

TABLE 1: EMPIRICAL TRANSFER COEFFICIENT VS. INCIDENCE ANGLE.

θ (DEG)	$f(\theta)$
0	0.48
90	0.46
100	0.45
110	0.435
120	0.42
130	0.38

TABLE 1: EMPIRICAL TRANSFER COEFFICIENT VS. INCIDENCE ANGLE.

θ **(DEG)**	$f(\theta)$
140	0.35
150	0.28
160	0.22
180	0.15

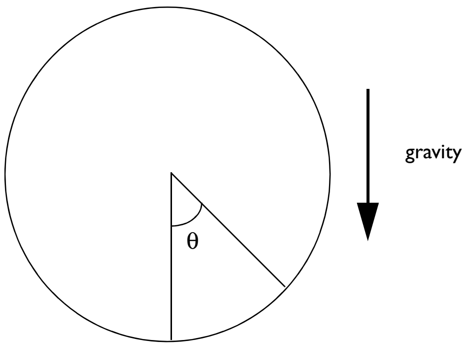


Figure 3: Definition of the incidence angle θ.

Results and Discussion

Figure 4 shows the flange surface temperature at steady state.

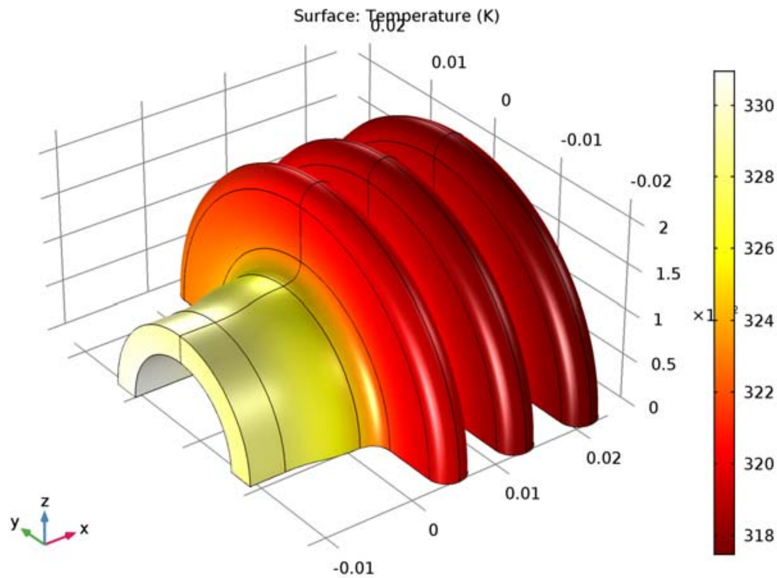


Figure 4: Stationary surface temperature of the flange.

As you can see in the surface temperature plot above to the left, the tube surface temperature is about 13 K higher than the flange shoulders. From the results, we can learn that the heat transfer from the outer surfaces of the flange is pretty efficient; there is a temperature difference of roughly 19 K between the outer flange surface and the air stream.

Figure 5 shows the heat transfer coefficient for the flange's outer walls.

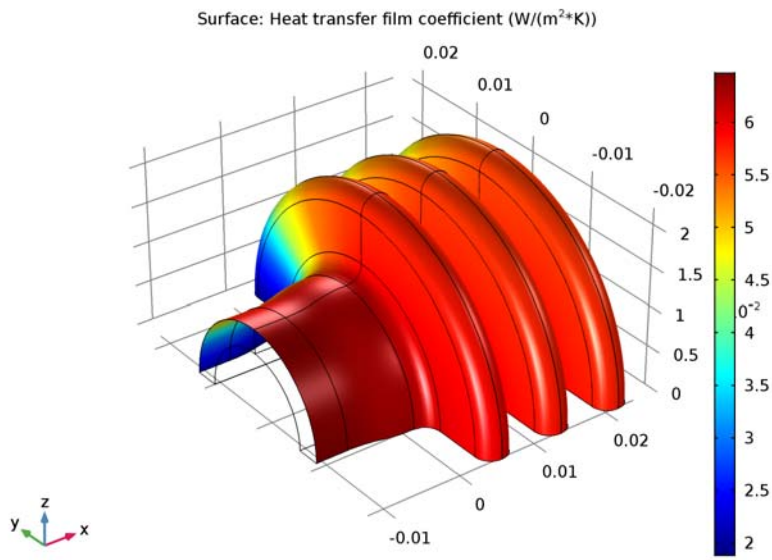


Figure 5: Heat transfer film coefficient, h, for the flange's outer walls.

As you can see, the coefficient decreases significantly along the vertical position of the flange's outer boundary.

Calculating the flange's total cooling power by integrating the normal total heat flux over the outer surfaces gives a value of 0.51 W which correspond to 1.02 W for the full geometry.

The surface temperature plot shown on Figure 4 further gives light to an inefficiency in the current flange design. When it enters the flange the process fluid is 363 K, while the inner surface of the pipe is only 32 K lower. Increasing the tube diameter would improve the heat transfer here, which is performed in a second stage of the modeling by varying the pipe's diameter, but keeping all other factors constant. By plotting in Figure 6 the global cooling power as a function of the inner pipe radius, you can now analyze how altering the radius impacts the performance of the cooling flange.

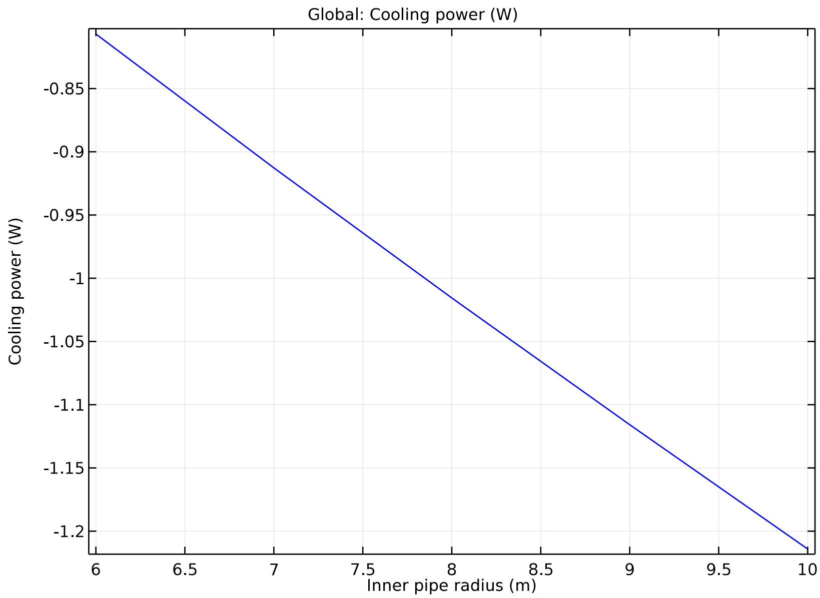


Figure 6: Total cooling power versus inner pipe radius.

Reference

1. B. Sundén, Kompendium i värmeöverföring [Notes on Heat Transfer], Sec. 10-3, Dept. of Heat and Power Engineering, Lund Inst. of Technology, 2003 (in Swedish).

Application Library path: `Heat_Transfer_Module/Thermal_Processing/cooling_flange`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

   Define parameters for relevant temperatures, material properties, and geometric dimensions. The parameters related to the geometry dimensions make it easier to perform parametric studies where you let some of these dimensions vary.

2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
k	26.2[mW/(m*K)]	0.0262 W/(m·K)	Thermal conductivity, air
T_inner	363[K]	363 K	Temperature, process fluid
Hh	15[W/(m^2*K)]	15 W/(m²·K)	Heat transfer coefficient
nu	18e-6[m^2/s]	1.8E-5 m²/s	Kinematic viscosity
r_inner	8[mm]	0.008 m	Inner pipe radius
l1	12[mm]	0.012 m	Pipe length excluding flanges
t1	3[mm]	0.003 m	Pipe thickness
t2	4[mm]	0.004 m	Pipe thickness, flange section
hf	10[mm]	0.01 m	Flange height
wf	4[mm]	0.004 m	Flange width
D	2*(r_inner+t2+hf)	0.044 m	Outer flange diameter

Next, create an interpolation function defined by the data in Table 1.

Interpolation 1 (int1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for Interpolation, locate the **Definition** section.

3 In the **Function name** text field, type `f`.

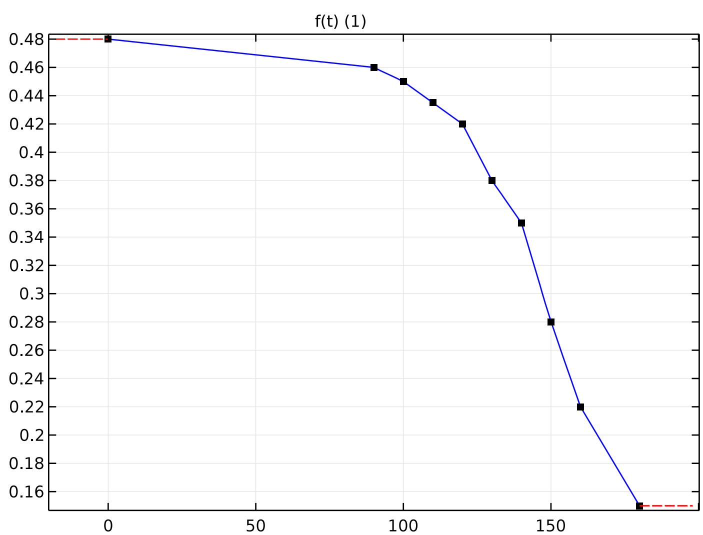
4 In the table, enter the following settings:

t	f(t)
0	0.48
90	0.46
100	0.45
110	0.435
120	0.42
130	0.38
140	0.35
150	0.28
160	0.22
180	0.15

5 Locate the **Units** section. In the **Arguments** text field, type `deg`.

6 In the **Function** text field, type `1`.

7 Click **Plot**.



GEOMETRY 1

In a first geometry modeling stage, create a 2D geometry by following the steps below.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `2*wf`.

4 In the **Height** text field, type `t2`.

5 Locate the **Position** section. In the **yw** text field, type `r_inner`.

6 On the **Work Plane** toolbar, click **Build All**.

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

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

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

3 Find the **Added segments** subsection. Click **Add Quadratic**.

4 Find the **Control points** subsection. In row **1**, set **yw** to `r_inner+t2`.

5 In row **2**, set **xw** to `wf/2`, and **yw** to `r_inner+t2`.

6 In row **3**, set **xw** to `wf/2`, and **yw** to `r_inner+t2+wf/2`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **xw** to `3*wf/2`.

9 Find the **Added segments** subsection. Click **Add Quadratic**.

10 Find the **Control points** subsection. In row **2**, set **yw** to `r_inner+t2`.

11 In row **3**, set **xw** to `2*wf`, and **yw** to `r_inner+t2`.

12 On the **Work Plane** toolbar, click **Build All**.

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

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `wf`.

4 In the **Height** text field, type `hf-wf`.

5 Locate the **Position** section. In the **xw** text field, type `wf/2`.

6 In the **yw** text field, type `r_inner+t2+wf/2`.

7 On the **Work Plane** toolbar, click **Build All**.

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

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for Circle, locate the **Size and Shape** section.

3 In the **Radius** text field, type `wf/2`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **xw** text field, type `wf`.

6 In the **yw** text field, type `r_inner+t2+hf-wf/2`.

7 On the **Work Plane** toolbar, click **Build All**.

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

Array 1 (arr1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

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

4 From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `3`.

6 Locate the **Displacement** section. In the **xw** text field, type `2*wf`.

7 On the **Work Plane** toolbar, click **Build All**.

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

Bézier Polygon 2 (b2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

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

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **yw** to `r_inner`.

5 In row **2**, set **xw** to `-l1`, and **yw** to `r_inner`.

6 Find the **Added segments** subsection. Click **Add Linear**.

7 Find the **Control points** subsection. In row **2**, set **yw** to `r_inner+t1`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **xw** to `-2*l1/3`.

10 Find the **Added segments** subsection. Click **Add Cubic**.

11 Find the **Control points** subsection. In row **2**, set **xw** to `-l1/3`.

12 In row **3**, set **xw** to `-l1/3`, and **yw** to `r_inner+t2`.

13 In row **4**, set **xw** to `0`, and **yw** to `r_inner+t2`.

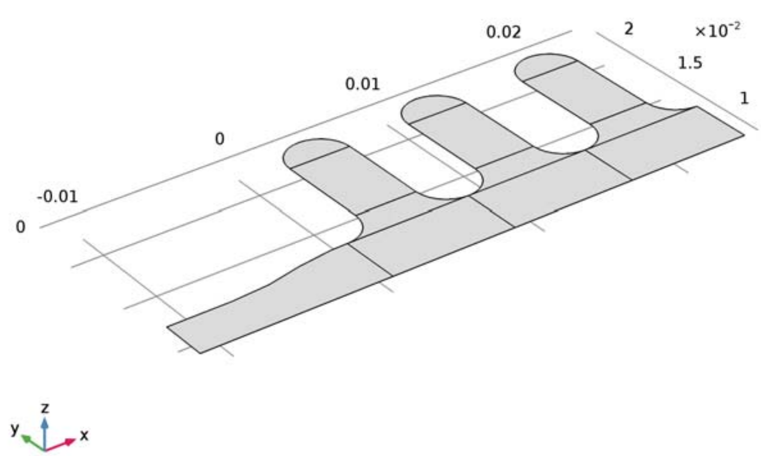
14 On the **Work Plane** toolbar, click **Build All**.

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

16 In the **Model Builder** window, click **Geometry 1**.

17 On the **Home** toolbar, click **Build All**.

You obtain the following 2D geometry.



Next, revolve the embedded 2D geometry to create the 3D model geometry.

Revolve 1 (rev1)

1 On the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for Revolve, locate the **Revolution Angles** section.

3 Click the **Angles** button.

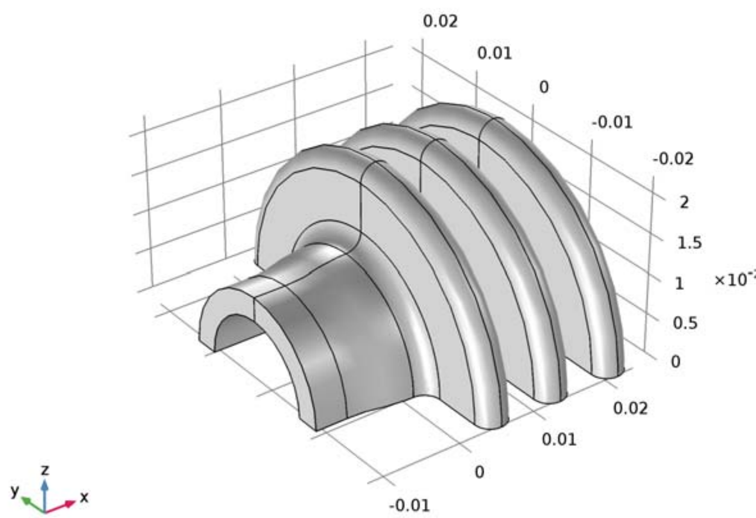
4 In the **End angle** text field, type `180`.

5 Locate the **Revolution Axis** section. Find the **Direction of revolution axis** subsection. In the **xw** text field, type `1`.

6 In the **yw** text field, type `0`.

7 On the **Geometry** toolbar, click **Build All**.

8 Click the **Go to Default 3D View** button on the **Graphics** toolbar.



DEFINITIONS

Variables 1

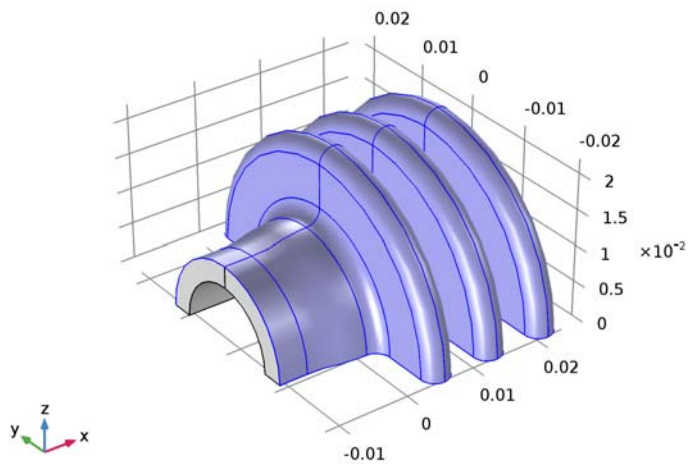
1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

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

4 Select Boundaries 3, 7, 9, 10, 13, 19, 23, 25, 29, 32, 35, 38–42, 45, 51, 55, 57, 61, 64, 67, 70–74, 77, 83, 87, 89, 93, 96, 99, and 102–106 only.

To do this, first copy the list of boundaries from this text, then use the **Paste Selection** button in the **Geometric Entity Selection** section, to paste these numbers.



For use when specifying the boundary condition for the flange's outer surface, create a selection.

5 Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Outer Boundaries` in the **Selection name** text field.

7 Click **OK**.

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

9 In the table, enter the following settings:

Name	Expression	Unit	Description
`alphap`	`1/ht.T_amb`	1/K	`Coefficient of thermal expansion`
`Gr`	`g_const*alphap*(T-ht.T_amb)*D^3/nu^2`		`Grashof number`
`theta`	`atan(y/z)+90[deg]`	rad	`Incidence angle`
`Hc`	`k*f(theta)*Gr^0.25/D`	W/(m²·K)	`Heat transfer film coefficient`

ADD MATERIAL

1 On 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>Silica glass**.

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

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

HEAT TRANSFER IN SOLIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

2 In the **Settings** window for Heat Transfer in Solids, locate the **Ambient Settings** section.

3 In the T_{amb} text field, type `298[K]`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Initial Values 1**.

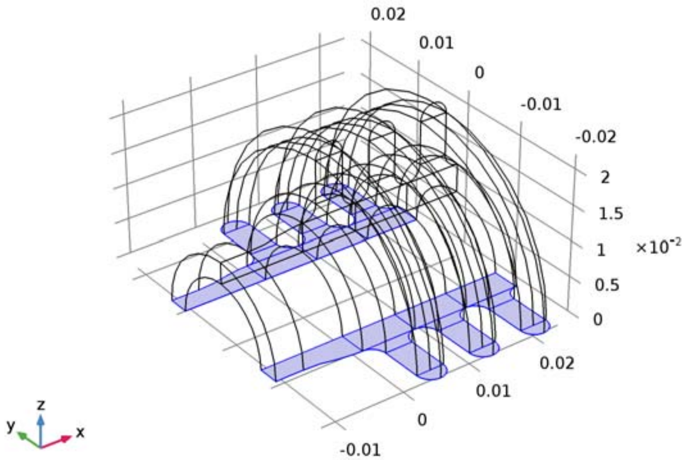
2 In the **Settings** window for Initial Values, type `323.15[K]` in the T text field.

Symmetry 1

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

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

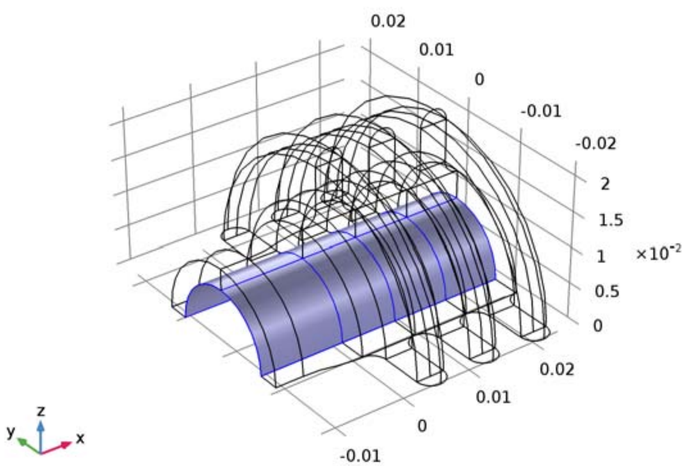
3 Select Boundaries 2, 8, 12, 15, 21, 22, 24, 27, 33, 34, 44, 47, 53, 54, 56, 59, 65, 66, 76, 79, 85, 86, 88, 91, 97, and 98 only.



Heat Flux 1

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

2 Select Boundaries 4, 6, 16, 18, 48, 50, 80, and 82 only.



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

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

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

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

7 Click the **Wireframe Rendering** button on the **Graphics** toolbar to go back to the original view.

Heat Flux 2

1 On 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 **Outer Boundaries**.

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

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

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

MESH 1

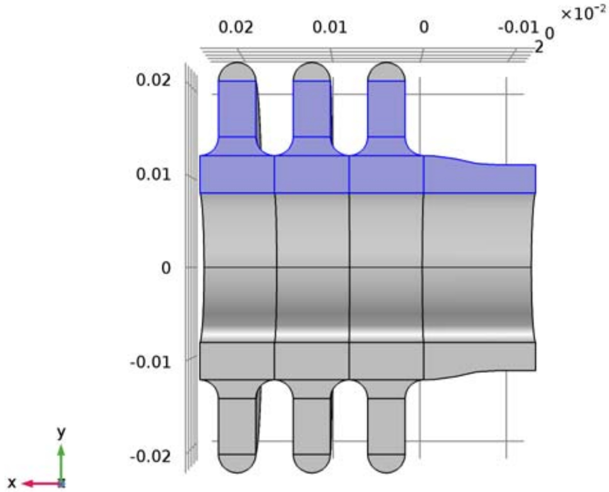
On the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

Mapped 1

1 Click the **Go to XY View** button on the **Graphics** toolbar. Do this twice to see the boundary mesh.

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

3 Select Boundaries 8, 21, 22, 33, 53, 54, 65, 85, 86, and 97 only.



Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Size**.

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

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

4 Click **Build Selected**.

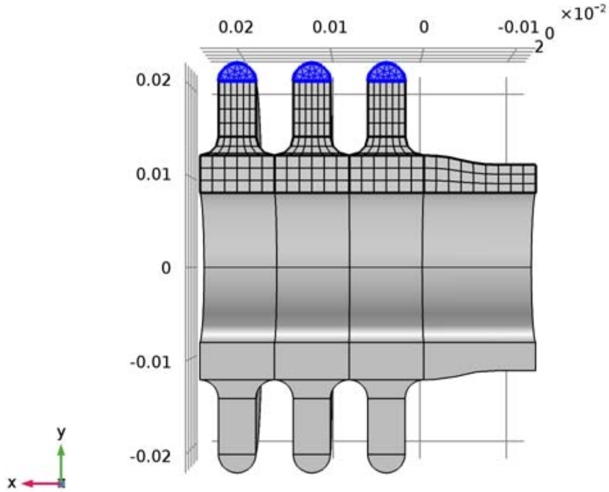
5 On the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Free Triangular 1

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

2 Select Boundaries 34, 66, and 98 only.

3 In the **Settings** window for Free Triangular, click **Build Selected**.



4 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

5 On the **Mesh** toolbar, click **Swept**.

Swept 1

Click **Distribution**.

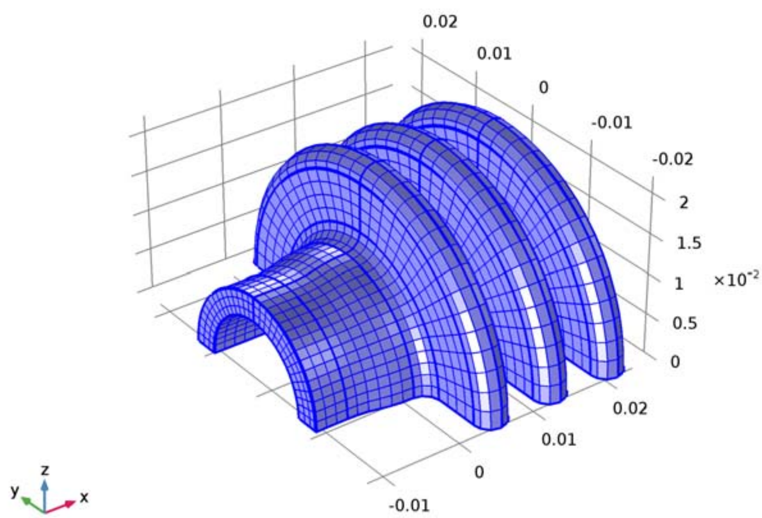
Distribution 1

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

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

3 In the **Number of elements** text field, type 30.

4 Click **Build All**.



STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot group shows the temperature field on the surface; compare with Figure 4.

Delete the second plot group to make a new surface plot of the heat transfer film coefficient.

Isothermal Contours (ht)

1 In the **Model Builder** window, under **Results** right-click **Isothermal Contours (ht)** and choose **Delete**.

2 Click **Yes** to confirm.

3D Plot Group 2

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

2 In the **Settings** window for 3D Plot Group, type `Heat Transfer Film Coefficient` in the **Label** text field.

3 On the **Heat Transfer Film Coefficient** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Heat Transfer Film Coefficient** click **Surface 1**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Definitions> Variables>Hc - Heat transfer film coefficient**.

3 On the **Heat Transfer Film Coefficient** toolbar, click **Plot**.

Compare this plot with that in Figure 5.

Surface Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 In the **Settings** window for Surface Integration, type `Outgoing Heat Flux` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Outer Boundaries**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Boundary fluxes>ht.ntflux - Normal total heat flux**.

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The integrated value, approximately 0.51 W, appears in the **Table** tab below the **Graphics** window. Taking both flange halves into account, the total cooling power of the flange is thus roughly 1 W.

Finally, extend the model by performing a parametric sweep over the inner pipe radius. Before adding a separate study for this purpose, define a variable for the total cooling power.

DEFINITIONS

Integration 1a (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, type `Integration, Outer` in the **Label** text field.

3 In the **Operator name** text field, type `intop_outer`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Outer Boundaries**.

Variables 2

1 On the **Definitions** toolbar, click **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
P_cooling	2*intop_outer(ht.q0)	W	Cooling power

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

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

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

STUDY 2

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
r_inner	range(6,1,10)	mm

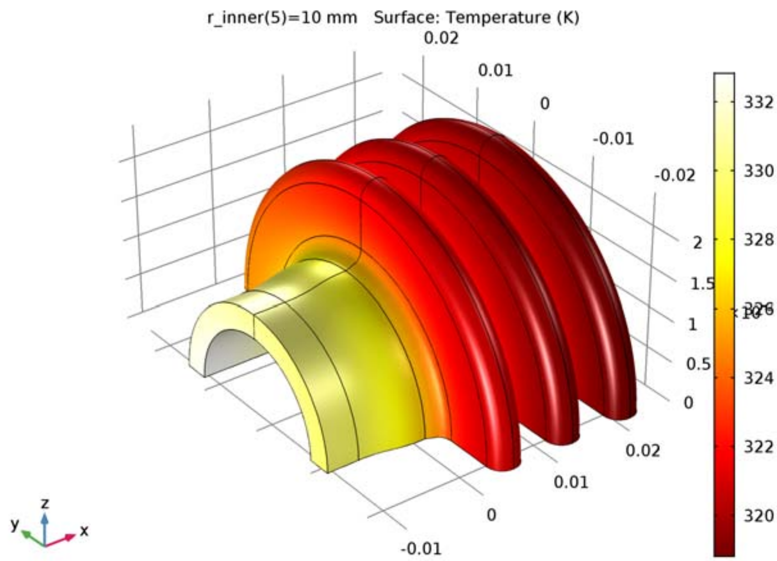
This gives a sweep centered around the original radius value.

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

RESULTS

Temperature (ht) 1

You get a new surface plot of the temperature for the parametric solution. From the Parameter value list you can choose the radius value for which to display the result.



Surface temperature for an inner radius of 10 mm.

Finally, replace the slice plot in the fourth plot group by a graph of the total cooling power versus tube radius.

Isothermal Contours (ht)

1 In the **Model Builder** window, under **Results** right-click **Isothermal Contours (ht)** and choose **Delete**.

2 Click **Yes** to confirm.

1D Plot Group 4

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Parametric Solutions 1 (sol3)**.

Global 1

1 On the **Cooling Power** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Definitions>Variables>P_cooling - Cooling power**.

3 Click to expand the **Legends** section. Clear the **Show legends** check box.

Cooling Power

1 In the **Model Builder** window, under **Results** click **Cooling Power**.

2 In the **Settings** window for 1D Plot Group, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Inner pipe radius (m)`.

5 On the **Cooling Power** toolbar, click **Plot**.

Compare the result with the graph in Figure 6.

Created in COMSOL Multiphysics 5.2a

Cooling and Solidification of Metal

Introduction

This example is a model of a continuous casting process. Liquid metal is poured into a mold of uniform cross section. The outside of the mold is cooled and the metal solidifies as it flows through the mold. When the metal leaves the mold, it is completely solidified on the outside but still liquid inside. The metal then continues to cool and eventually solidify completely, at which point it can be cut into sections. This tutorial simplifies the problem somewhat by not computing the flow field of the liquid metal and assuming there is no volume change during solidification. It is also assumed that the velocity of the metal is constant and uniform throughout the modeling domain. The phase transition from molten to solid state is modeled via the apparent heat capacity formulation. Issues of convergence and mesh refinement are addressed for this highly nonlinear model.

The Continuous Casting application is similar to this one, except that the velocity is computed from the Laminar Flow interface instead of being considered constant and uniform. For a detailed description of the application, see Continuous Casting.

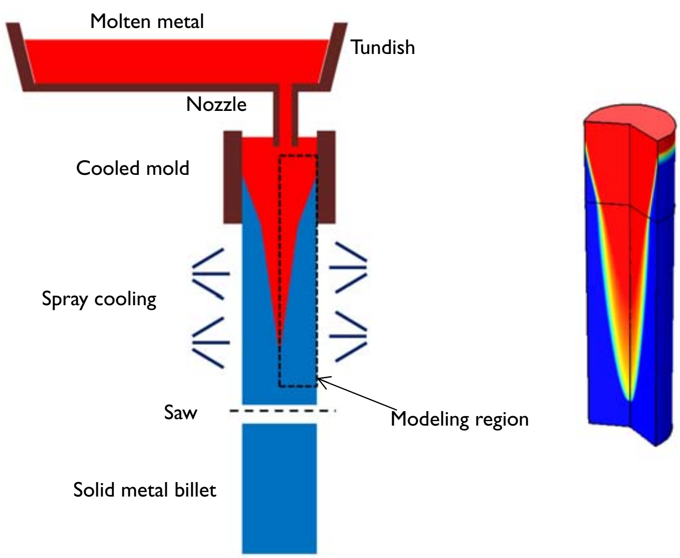


Figure 1: A continuous casting process. The section where the metal is solidifying is being modeled.

Model Overview

The model simplifies the 3D geometry of the continuous casting to a 2D axisymmetric model composed of two rectangular regions: one representing the strand within the mold, and one the spray cooled region outside of the mold, prior to the saw cutoff. In the second section, there is also significant cooling via radiation to the ambient. In this region it is assumed that the molten metal is in a hydrostatic state, that the only motion in the fluid is due to the bulk downward motion of the strand. This simplification allows the assumption of bulk motion throughout the domain.

Since this is a continuous process, the system can be modeled at steady state. The heat transport is described by the equation:

$$\rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot (-k \nabla T) = 0$$

where k and C_p denote thermal conductivity and specific heat, respectively. The velocity, $\mathbf{u}$, is the fixed casting speed of the metal in both liquid and solid states.

As the metal cools down in the mold, it solidifies. During the phase transition, a significant amount of latent heat is released. The total amount of heat released per unit mass of alloy during the transition is given by the change in enthalpy, ΔH. In addition, the specific heat capacity, C_p, also changes considerably during the transition. The difference in specific heat before and after transition can be approximated by

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

As opposed to pure metals, an alloy generally undergoes a broad temperature transition zone, over several Kelvin, in which a mixture of both solid and molten material coexist in a "mushy" zone. To account for the latent heat related to the phase transition, the Apparent Heat capacity method is used through the Heat Transfer with Phase Change domain condition. The objective of the analysis is to make ΔT, the half-width of the transition interval small, such that the solidification front location is well defined.

Table 1 reviews the material properties in this tutorial.

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))	531	380
Thermal conductivity	k (W/(m·K))	150	300

The melting temperature, T_{m}, and enthalpy, ΔH, are 1356 K and 205 kJ/kg, respectively.

This example is a highly nonlinear problem and benefits from taking an iterative approach to finding the solution. The location of the transition between the molten and solid state is a strong function of the casting velocity, the cooling rate in the mold, and the cooling rate in the spray cooled region. A fine mesh is needed across the solidification front to resolve the change in material properties. However, it is not known where this front will be.

By starting with a gradual transition between liquid and solid, it is possible to find a solution even on a relatively coarse mesh. This solution can be used as the starting point for the next step in the solution procedure, which uses a sharper transition from liquid to solid. This is done using the continuation method. Given a monotonic list of values to solve for, the continuation method uses the solution to the last case as the starting condition for the next. Once a solution is found for the smallest desired ΔT, the adaptive mesh refinement algorithm is used to refine the mesh to put more elements around the transition region. This finer mesh is then used to find a solution with an even sharper transition. This can be repeated as needed to get better and better resolution of the location of the solidification front.

In this example, the parameter ΔT is first ramped down from 300 K to 75 K, then the adaptive mesh refinement is used such that a finer mesh is used around the solidification front. The resultant solution and mesh are then used as starting points for a second study, where the parameter ΔT is further ramped down from 50 K to 25 K. The double-dogleg solver is used to find the solution to this highly nonlinear problem. Although it takes more time, this solver converges better in cases when material properties vary strongly with respect to the solution.

Results and Discussion

The solidification front computed with the coarsest mesh, and for $\Delta T = 75$ K, is shown in Figure 2. A wide transition between the molten and solid state is observed. The adaptive mesh refinement algorithm then refines the mesh along the solidification front because this is the region where the results are strongly dependent upon mesh size. This solution, and refined mesh, is used as the starting point for the next solution, which ramps the ΔT parameter down to 25 K. These results are shown in Figure 3.

The point of complete solidification moves slightly as the transition zone is made smaller. As the transition zone becomes smaller, a finer mesh is needed, otherwise the model might not converge. If it is desired to get an even better resolution of the solidification front, the solution procedure used here should be repeated to get an even finer mesh, and further ramp down the ΔT parameter.

The liquid phase fraction is plotted along the r-direction at the line at the bottom of the mold in Figure 4, and Figure 5 shows the liquid fraction along the centerline of the strand.

For smaller values of ΔT, the transition becomes sharper, and the model gives confidence that the metal is completely solidified before the strand is cut.

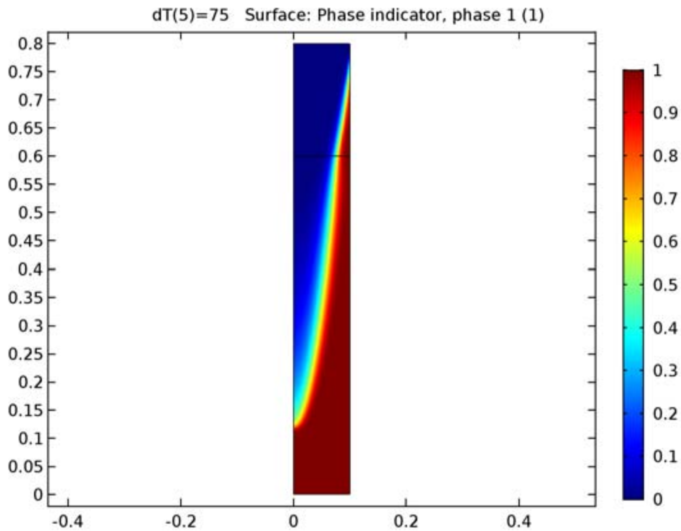


Figure 2: The fraction of liquid phase for ΔT = 75 K shows a gradual transition between the liquid and solid phase.

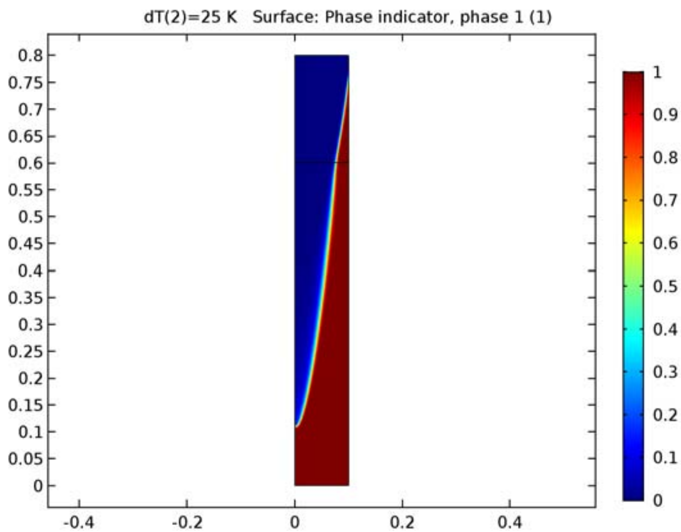


Figure 3: The fraction of liquid phase for ΔT = 25 K shows a sharp transition between the liquid and solid phase.

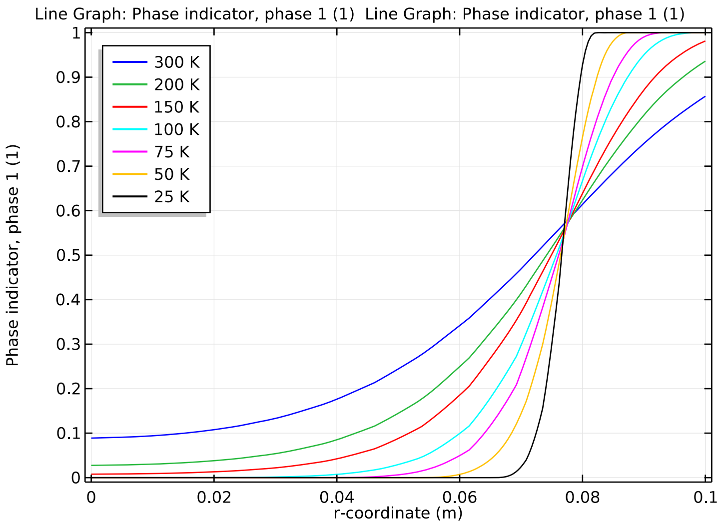


Figure 4: The fraction of liquid phase through the radius for all values of ΔT*. For smaller values of* ΔT*, the transition is sharper.*

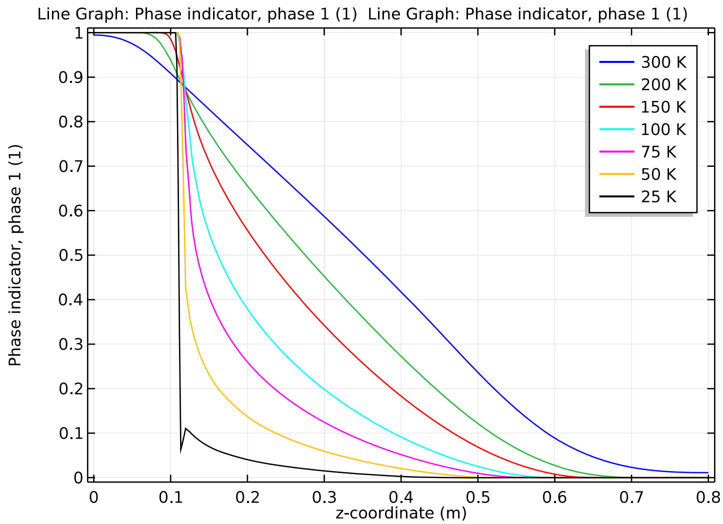


Figure 5: The fraction of liquid phase along the centerline for all values of ΔT. For smaller values of ΔT, the transition is sharper.

Application Library path: `Heat_Transfer_Module/Thermal_Processing/cooling_solidification_metal`

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 **Heat Transfer>Heat Transfer in Fluids (ht)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

First, set up parameters and variables used in the continuous casting process model analysis.

Parameters

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

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

3 Click **Load from File**.

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

GEOMETRY 1

Create two rectangles representing the strand within the mold, and spray cooled region outside of the mold.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.1`.

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

Rectangle 2 (r2)

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

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

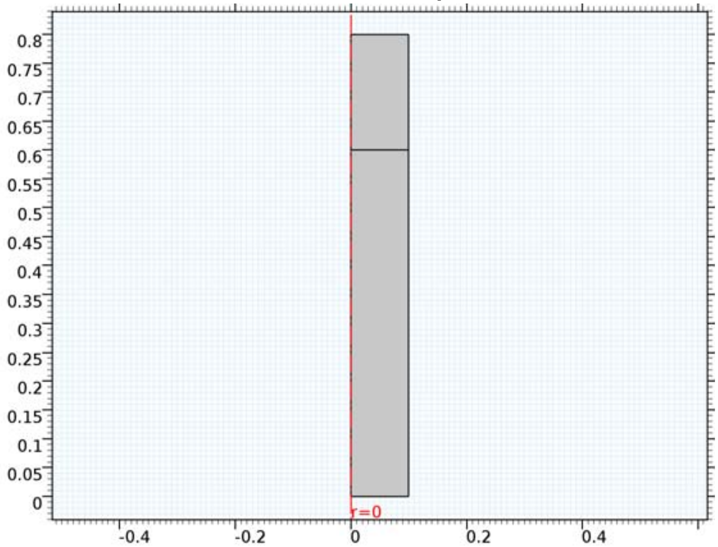
3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **z** text field, type `0.6`.

6 Click **Build All Objects**.

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



MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

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

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

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	300	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	`Cp_s`	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic

Material 2 (mat2)

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

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

3 Select Domains 1 and 2 only.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	150	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	Cp_l	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic

Set up the physics.

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, type `T_in` in the T text field.

Phase Change Material 1

1. On the **Physics** toolbar, click **Domains** and choose **Phase Change Material**.
2. Select Domains 1 and 2 only.
3. In the **Settings** window for Phase Change Material, locate the **Model Inputs** section.
4. Specify the **u** vector as

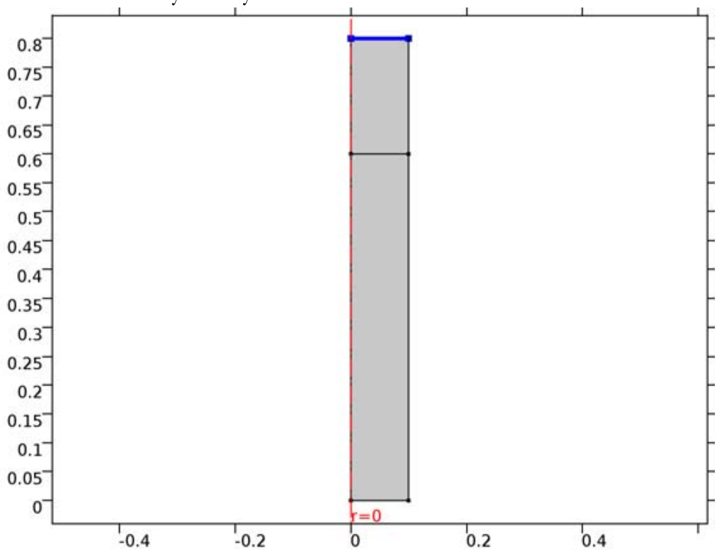
0	r
-v_cast	z

5. Locate the **Phase Change** section. In the $T_{\mathrm{pc},\,1\rightarrow 2}$ text field, type `T_m`.
6. In the $\Delta T_{1\rightarrow 2}$ text field, type `dT`.
7. In the $L_{1\rightarrow 2}$ text field, type `dH`.
8. Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Solid Metal Alloy (mat1)**.
9. Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Liquid Metal Alloy (mat2)**.

Temperature 1

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

2 Select Boundary 5 only.



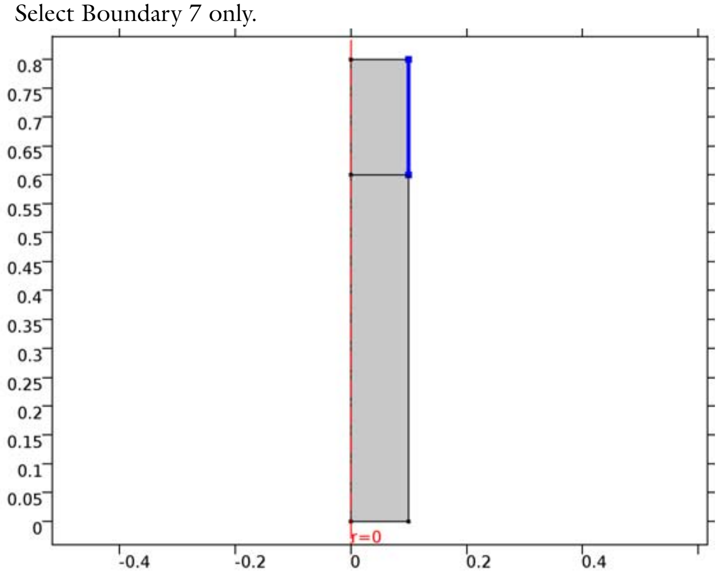
3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `T_in`.

Heat Flux 1

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

2 Select Boundary 7 only.



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

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

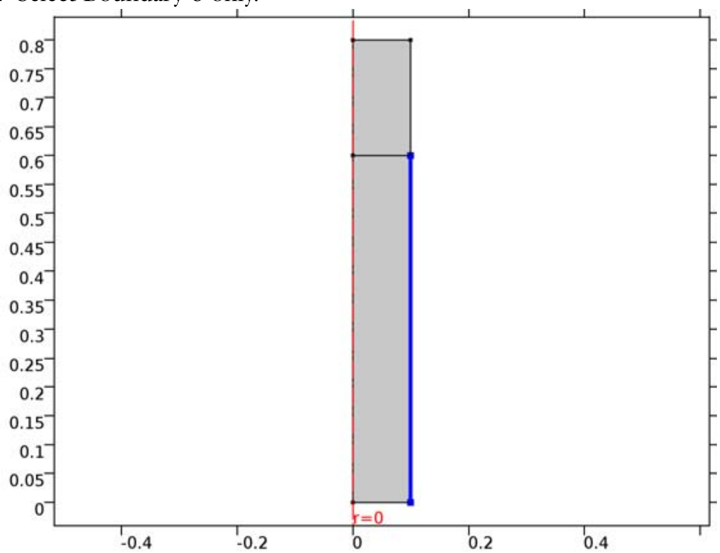
5 In the h text field, type `h_mold`.

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

Heat Flux 2

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

2 Select Boundary 6 only.



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

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

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

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

Diffuse Surface 1

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

2 Select Boundary 6 only.

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

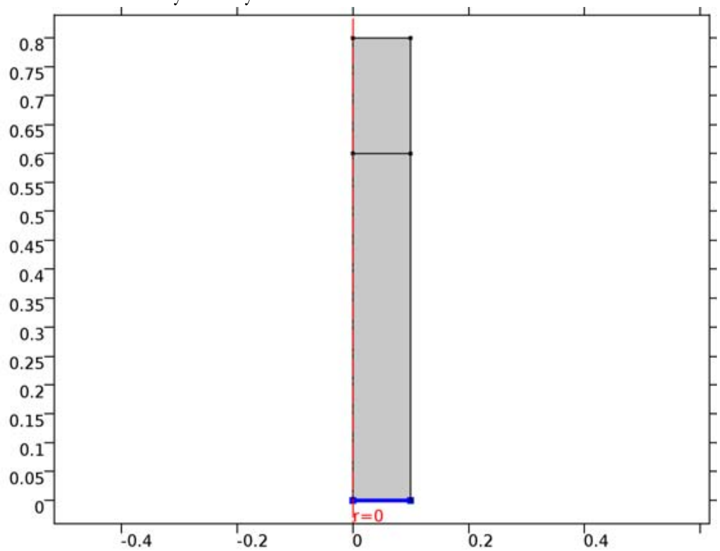
4 From the ε list, choose **User defined**. In the associated text field, type `eps_s`.

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

Outflow 1

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

2 Select Boundary 2 only.



MESH 1

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

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

3 From the **Element size** list, choose **Extremely fine**.

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.

Step 1: Stationary

Set up an auxiliary continuation sweep for the `dT` parameter.

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

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

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

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT	300 200 150 100 75	K

6 Select the **Adaptive mesh refinement** check box.

Solution 1 (sol1)

1 On 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)>Stationary Solver 1** node, then click **Adaptive Mesh Refinement**.

4 In the **Settings** window for Adaptive Mesh Refinement, locate the **Mesh Refinement** section.

5 From the **Refinement method** list, choose **Mesh initialization**.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Fully Coupled 1**.

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

8 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Double dogleg**.

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

RESULTS

2D Plot Group 1

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

2 In the **Settings** window for 2D Plot Group, type `Solid and Liquid Phases (Adaptive Mesh)` in the **Label** text field.

Surface 1

1 Right-click **Solid and Liquid Phases (Adaptive Mesh)** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Fluids>Phase change>ht.theta1 - Phase indicator, phase 1**.

3 On the **Solid and Liquid Phases (Adaptive Mesh)** toolbar, click **Plot**.

The reproduced figure describes the fraction of liquid phase for ΔT = 75 K.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Stationary**.

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

STUDY 2

Step 1: Stationary

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

2 In the **Model Builder** window, click **Study 2**.

3 In the **Settings** window for Study, locate the **Study Settings** section.

4 Clear the **Generate default plots** check box.

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

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

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

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

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

10 From the **Solution** list, choose **Adaptive Mesh Refinement 1 (sol2)**.

11 From the **Parameter value (dT (K))** list, choose **75**.

12 Click to expand the **Mesh selection** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

13 Click **Add**.

14 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT	50 25	K

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for Fully Coupled, locate the **Method and Termination** section.

5 From the **Nonlinear method** list, choose **Double dogleg**.

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

RESULTS

2D Plot Group 2

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

2 In the **Settings** window for 2D Plot Group, type `Solid and Liquid Phases` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

Surface 1

1 Right-click **Solid and Liquid Phases** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Fluids>Phase change>ht.theta1 - Phase indicator, phase 1**.

3 On the **Solid and Liquid Phases** toolbar, click **Plot**.

This shows the fraction of liquid phase for ΔT = 25 K.

1D Plot Group 3

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

2 In the **Settings** window for 1D Plot Group, type `Phase Indicator at Symmetry Axis` in the **Label** text field.

Line Graph 1

1 On the **Phase Indicator at Symmetry Axis** toolbar, click **Line Graph**.

2 Select Boundaries 1 and 3 only.

3 In the **Settings** window for Line Graph, locate the **y-axis data** section.

4 Click **ht.theta1 - Phase indicator, phase 1** in the upper-right corner of the section. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `z`.

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

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

4 On the **Phase Indicator at Symmetry Axis** toolbar, click **Plot**.

Compare the resulting plot with Figure 5 showing the fraction of liquid phase through the centerline for all values of Δ T.

Phase Indicator at Symmetry Axis

In the **Model Builder** window, under **Results** right-click **Phase Indicator at Symmetry Axis** and choose **Duplicate**.

Phase Indicator at Symmetry Axis 1

In the **Settings** window for 1D Plot Group, type `Phase Indicator through Radius` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Phase Indicator through Radius** node, then click **Line Graph 1**.

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

3 Click **Clear Selection**.

4 Select Boundary 4 only.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `r`.

6 On the **Phase Indicator through Radius** toolbar, click **Plot**.

Line Graph 2

1 In the **Model Builder** window, under **Results>Phase Indicator through Radius** click **Line Graph 2**.

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

3 Click **Clear Selection**.

4 Select Boundary 4 only.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `r`.

Phase Indicator through Radius

1 In the **Model Builder** window, under **Results** click **Phase Indicator through Radius**.

2 In the **Settings** window for 1D Plot Group, click to expand the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 On the **Phase Indicator through Radius** toolbar, click **Plot**.

Compare the resulting plot with Figure 4 showing the fraction of liquid phase through the radius for all values of Δ T.

Created in COMSOL Multiphysics 5.2a

Cross-Flow Heat Exchanger

Introduction

This application simulates the fluid flow and heat transfer in a micro-heat exchanger of cross-flow type made of stainless steel. Heat exchangers of this type are found in lab-on-chip devices in biotechnology and microreactors, for example for micro-fuel cells. The application takes into account heat transferred through both convection and conduction. The geometry and material properties are taken from Ref. 1.

Model Definition

Figure 1 shows the heat exchanger's geometry. Notice that the fluid channels have a square cross section rather than the circular cross section more commonly used in micro-heat exchangers. A cross-flow heat exchanger can typically consist of about 20 unit cells. However, because the unit cells are identical except for edge effects in the outer cells, you can restrict the model to a single unit cell.

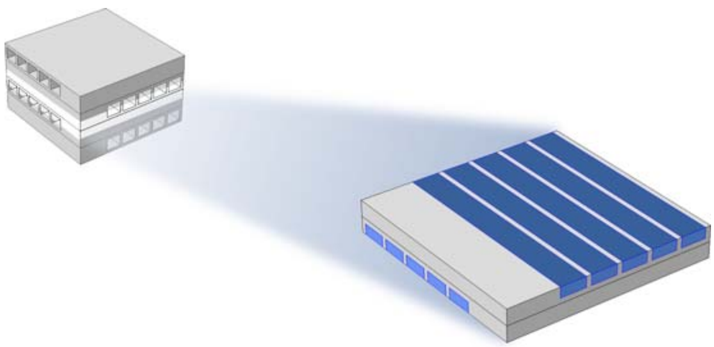


Figure 1: Depiction of the modeled part of the micro-heat exchanger.

Because heat is transferred by convection and conduction, the model uses a Conjugate Heat Transfer interface in the laminar flow regime.

The boundary conditions are insulating for all outer surfaces except for the inlet and outlet boundaries. At the inlets for both cold and hot streams, the temperatures are constant and a laminar inflow profile with an average velocity of 2.5 mm/s is defined.

At the outlet, the heat transport is dominated by convection, which makes the outflow boundary condition suitable. For the flow field, the model applies the outlet boundary condition with a constant pressure.

As shown in Figure 1, you can take advantage of the model's symmetries to model only half of the channel height. Therefore, the symmetry boundary condition applies to the channels.

Results and Discussion

Figure 2 shows the temperature at the channel walls as well as temperature isosurfaces in the device, which clearly reveal the influence of the convective term.

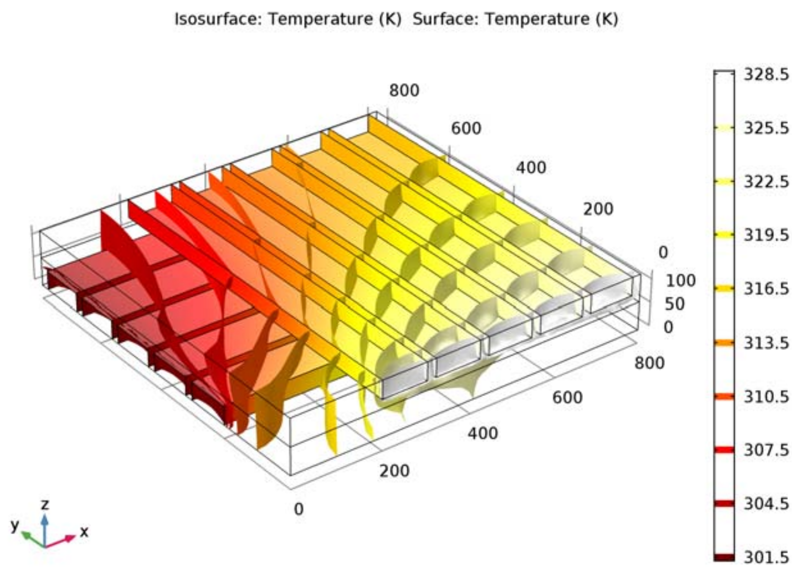


Figure 2: Channel wall temperature and isotherms through the cell geometry.

As can be seen in Figure 3, the temperature differs significantly between the different outlets in both hot and cold streams. This implies that the hot stream is not cooled uniformly.

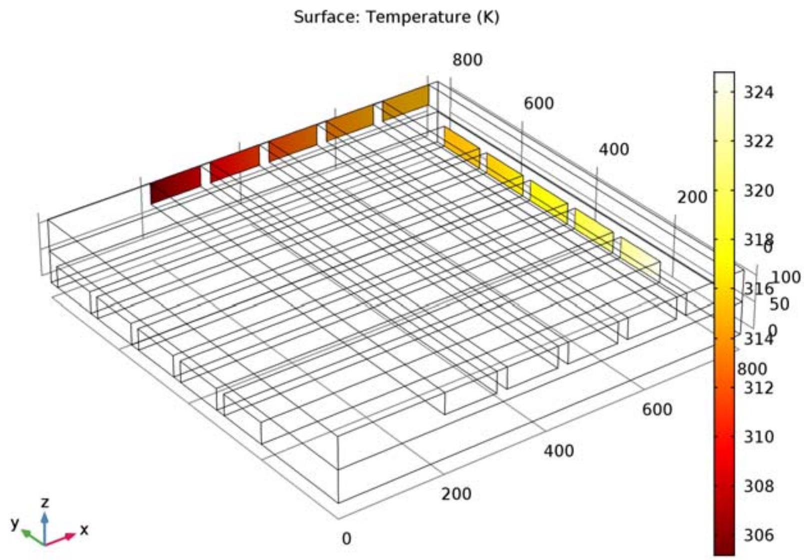


Figure 3: Temperature field at the outlet boundaries.

The flow field in the channels is a typical laminar velocity profile; see Figure 4.

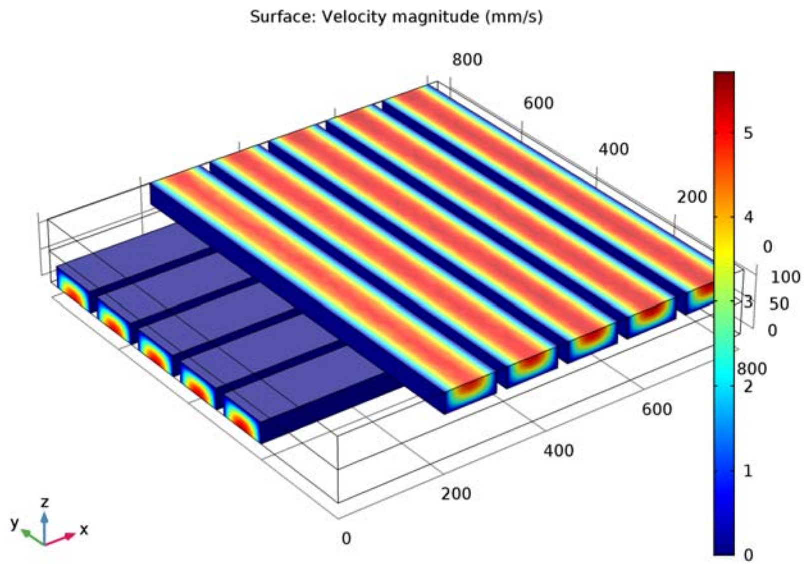


Figure 4: Velocity profile in the channels.

There are several quantities that describe the characteristics and effectiveness of a heat exchanger. The mixing-cup temperature of the fluid leaving the heat exchanger is calculated according to (1.4 in Ref. 2)

$$\langle T\rangle = \frac{\int_{\mathrm{outlet}} \rho C_p T u ds}{\int_{\mathrm{outlet}} \rho C_p u ds} \qquad (1)$$

COMSOL Multiphysics provides built-in variables to easily calculate $\langle T\rangle$. At the upper channels, the outlet mixing-cup temperature is 45 °C. At the lower channels, despite a hotter inlet temperature, a lower value of 38.6 °C is found for the outlet mixing-cup temperature. The maximum pressure drop in the heat exchanger is 5 Pa.

The overall heat transfer coefficient is another interesting quantity. It is a measure of the performance of a heat exchanger design defined as

$$h_{\mathrm{eq}} = \frac{P}{A(T_{\mathrm{hot}} - T_{\mathrm{cold}})} \tag{2}$$

where P is the total exchanged power and A is the surface area through which P flows. In this model, the value of h_{eq} is about 1500 W/(m^2·K).

References

1. W. Ehrfeld, V. Hessel, and H. Löwe, *Microreactors*, John Wiley & Sons, 2000.

2. P.K. Nag, *Heat and Mass Transfer*, 2nd ed., Tata McGraw-Hill, 2007.

Application Library path: `Heat_Transfer_Module/Heat_Exchangers/crossflow_heat_exchanger`

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 **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
T_cold	300[K]	300 K	Temperature, cold stream
T_hot	330[K]	330 K	Temperature, hot stream
u_avg	2.5[mm/s]	0.0025 m/s	Average inlet velocity

GEOMETRY 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for Geometry, locate the **Units** section.
3. From the **Length unit** list, choose **µm**.

First, create the cross-section of one unit cell and extrude it.

Block 1 (blk1)

1. On the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, locate the **Size and Shape** section.
3. In the **Width** text field, type 800.
4. In the **Depth** text field, type 800.
5. In the **Height** text field, type 60.

Block 2 (blk2)

1. Right-click **Block 1 (blk1)** and choose **Build Selected**.
2. On the **Geometry** toolbar, click **Block**.
3. In the **Settings** window for Block, locate the **Size and Shape** section.
4. In the **Width** text field, type 800.
5. In the **Depth** text field, type 100.
6. In the **Height** text field, type 40.
7. Locate the **Position** section. In the **y** text field, type 200.

Array 1 (arr1)

1. Right-click **Block 2 (blk2)** and choose **Build Selected**.
2. On the **Geometry** toolbar, click **Transforms** and choose **Array**.
3. Select the object **blk2** only.
4. In the **Settings** window for Array, locate the **Size** section.
5. From the **Array type** list, choose **Linear**.

6 In the **Size** text field, type `5`.

7 Locate the **Displacement** section. In the **y** text field, type `120`.

8 Locate the **Selections of Resulting Entities** section. Click **New**.

9 In the **New Cumulative Selection** dialog box, type `Channels` in the **Name** text field.

10 Click **OK**.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for Rotate, locate the **Rotation Angle** section.

4 In the **Rotation** text field, type `180`.

5 Locate the **Point on Axis of Rotation** section. In the **z** text field, type `60`.

6 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **Cartesian**.

7 In the **x** text field, type `1`.

8 In the **y** text field, type `1`.

9 In the **z** text field, type `0`.

Keep the existing unit cell by the following step.

10 Locate the **Input** section. Select the **Keep input objects** check box.

11 Click **Build All Objects**.

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

Define several selections that help you throughout the model set-up.

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Upper Inlets` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 41, 48, 55, 62, and 69 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Lower Inlets` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 8, 14, 20, 26, and 32 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Upper Outlets` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 44, 51, 58, 65, and 72 only.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Lower Outlets` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 77–81 only.

Explicit 5

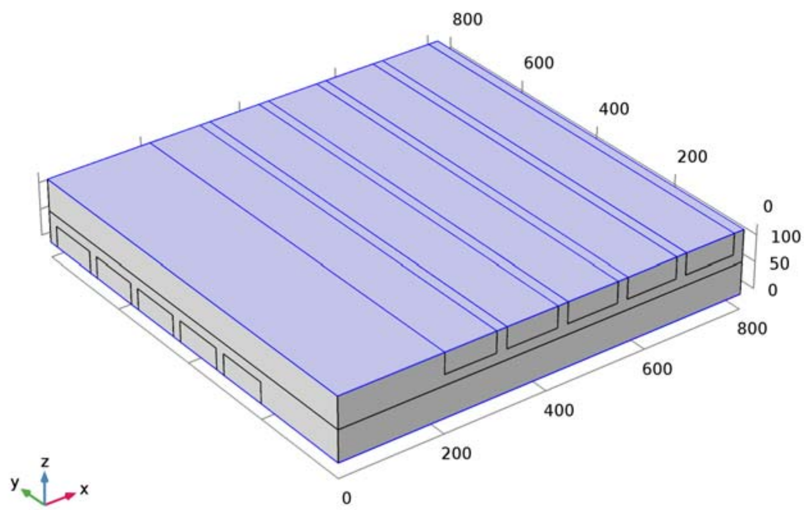
1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Symmetry` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** check box.

Select one of the uppermost and lowermost boundaries, which now automatically will select all uppermost and lowermost boundaries thanks to the continuous tangency.



The next selections are needed to evaluate the equivalent heat transfer coefficient.

Union 1

1. On the **Definitions** toolbar, click **Union**.
2. In the **Settings** window for Union, type `Outlets` in the **Label** text field.
3. Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4. Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
5. In the **Add** dialog box, In the **Selections to add** list, choose **Upper Outlets** and **Lower Outlets**.
6. Click **OK**.

MATERIALS

Define the material properties.

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Stainless Steel` in the **Label** text field.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	15	W/(m·K)	Basic
Density	rho	7800	kg/m³	Basic
Heat capacity at constant pressure	Cp	420	J/(kg·K)	Basic

ADD MATERIAL

1 On 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 (mat2)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat2)**.

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

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

HEAT TRANSFER (HT)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Fluid 1**.

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

3 From the **Selection** list, choose **Channels**.

Temperature 1

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

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

3 From the **Selection** list, choose **Upper Inlets**.

4 Locate the **Temperature** section. In the T_0 text field, type `T_hot`.

Temperature 2

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

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

3 From the **Selection** list, choose **Lower Inlets**.

4 Locate the **Temperature** section. In the T_0 text field, type `T_cold`.

Outflow 1

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

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

3 From the **Selection** list, choose **Upper Outlets**.

Outflow 2

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

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

3 From the **Selection** list, choose **Lower Outlets**.

Symmetry 1

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

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

3 From the **Selection** list, choose **Symmetry**.

So far, the boundary conditions for heat transfer have been specified. Continue with the set up of the flow equation.

LAMINAR FLOW (SPF)

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

2 In the **Settings** window for Laminar Flow, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Channels**.

Because of the different inlet temperatures, the densities for the hot and cold stream vary and produce different velocities when the laminar inflow boundary condition is used. In order to have the same velocity profile on each inlet, define the laminar inflow boundary condition for the hot and cold inlet boundaries separately.

Inlet 1

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

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

3 From the **Selection** list, choose **Upper Inlets**.

4 Locate the **Boundary Condition** section. From the list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} text field, type `u_avg`.

The **Entrance length** value must be large enough so that the flow can reach a laminar profile. For a laminar flow, L_{entr} should be significantly greater than $0.06\mathrm{Re}D$, where Re is the Reynolds number and D is the inlet length scale. In this case, 1 mm is an appropriate value.

6 In the L_{entr} text field, type `1[mm]`.

Inlet 2

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

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

3 From the **Selection** list, choose **Lower Inlets**.

4 Locate the **Boundary Condition** section. From the list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} text field, type `u_avg`.

6 In the L_{entr} text field, type `1[mm]`.

Outlet 1

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

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

3 From the **Selection** list, choose **Upper Outlets**.

Outlet 2

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

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

3 From the **Selection** list, choose **Lower Outlets**.

Symmetry 1

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

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

3 From the **Selection** list, choose **Symmetry**.

After solving the model, the equivalent heat transfer coefficient is evaluated according to Equation 2. To do so, define the following coupling operator.

DEFINITIONS

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, type `Average on Upper Channel Walls` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 40, 42, 45, 47, 49, 52, 54, 56, 59, 61, 63, 66, 68, 70, and 73 only.

   To select more easily these boundaries, use the **Paste** button and insert the list of numbers above in the **Paste Selection** dialog box.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

COMSOL Multiphysics automatically creates four default plots: a temperature plot, an isothermal contour plot, a slice plot for the velocity field, and contour plot for the pressure field. The isothermal contours will be modified, to create the plot shown in Figure 2.

Isothermal Contours (ht)

In the **Model Builder** window, under **Results** right-click **Isothermal Contours (ht)** and choose **Surface**.

Surface 1

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

2 From the **Data set** list, choose **Exterior Walls**.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer>Temperature>T - Temperature**.

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Isosurface**.

5 On the **Isothermal Contours (ht)** toolbar, click **Plot**.

To visualize the velocity field as in Figure 4, follow the steps below:

Slice

1 In the **Model Builder** window, expand the **Velocity (spf)** node.

2 Right-click **Slice** and choose **Delete**.

Velocity (spf)

In the **Model Builder** window, under **Results** right-click **Velocity (spf)** and choose **Surface**.

Surface 1

1 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow> Velocity and pressure>spf.U - Velocity magnitude**.

2 Locate the **Expression** section. From the **Unit** list, choose **mm/s**.

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

Data Sets

To show the temperature on the outlet boundaries only, as in Figure 3, first produce a new data set for the selection built before. Then use this data set for a surface plot of the temperature.

Surface 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Surface**.

2 In the **Settings** window for Surface, type `Outlets` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Outlets**.

3D Plot Group 5

1 On the **Results** toolbar, click **3D Plot Group**.

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

Surface 1

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

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

3 From the **Data set** list, choose **Outlets**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

5 On the **Outlet Temperature** toolbar, click **Plot**.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, type `Mixing-Cup Temperatures` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer>Global>Weighted average temperature>ht.ofl1.Tave - Weighted average temperature**.

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

Expression	Unit	Description
`ht.ofl1.Tave`	`degC`	`Weighted average temperature (Upper Outlets)`
`ht.ofl2.Tave`	`degC`	`Weighted average temperature (Lower Outlets)`

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The mixing-cup temperature at upper outlets is about 38.5°C and at the lower outlets about 45°C.

RESULTS

Derived Values

To calculate the maximum pressure drop proceed as follows:

Surface Maximum 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Maximum>Surface Maximum**.

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

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

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>p - Pressure**.

5 In the **Label** text field, type `Maximum Pressure Drop`.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`p`	`Pa`	`Maximum pressure drop`

7 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The maximum pressure is 5 Pa. The minimum pressure is defined by the outlet boundary conditions and is zero. Thus, the maximum pressure drop is also 5 Pa.

Now, evaluate the equivalent heat transfer coefficient as defined in Equation 2. You can use the integration operators defined previously in **Component 1>Definitions**.

RESULTS

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`aveop1(ht.ntflux)/(T_hot-T_cold)`	`W/(m^2*K)`	

4 In the **Label** text field, type `Heat Transfer Coefficient`.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`aveop1(ht.ntflux)/(T_hot-T_cold)`	`W/(m^2*K)`	`Heat transfer coefficient`

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The equivalent heat transfer coefficient is about $1500 \mathrm{W}/(\mathrm{m}^2 \cdot \mathrm{K})$.

Created in COMSOL Multiphysics 5.2a

Steady-State 2D Axisymmetric Heat Transfer with Conduction

Introduction

The following example illustrates how to build and solve a conductive heat transfer problem using the Heat Transfer interface. The model, taken from a NAFEMS benchmark collection, shows an axisymmetric steady-state thermal analysis. As opposed to the NAFEMS benchmark model, we use the temperature unit Kelvin instead of degrees Celsius for this model.

Model Definition

The modeling domain describes the cross section of a 3D solid as shown in Figure 1.

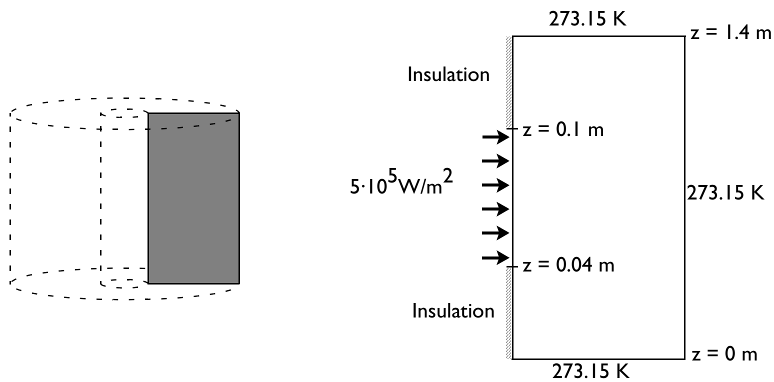


Figure 1: Model geometry and boundary conditions.

You set three types of boundary conditions:·

- Prescribed heat flux
- Insulation/Symmetry
- Prescribed temperature

The governing equation for this problem is the steady-state heat equation for conduction with the volumetric heat source set to zero:

$$\nabla \cdot (-k\nabla T) = 0$$

The thermal conductivity k is 52 W/(m·K).

Results

The plot in Figure 2 shows the temperature distribution.

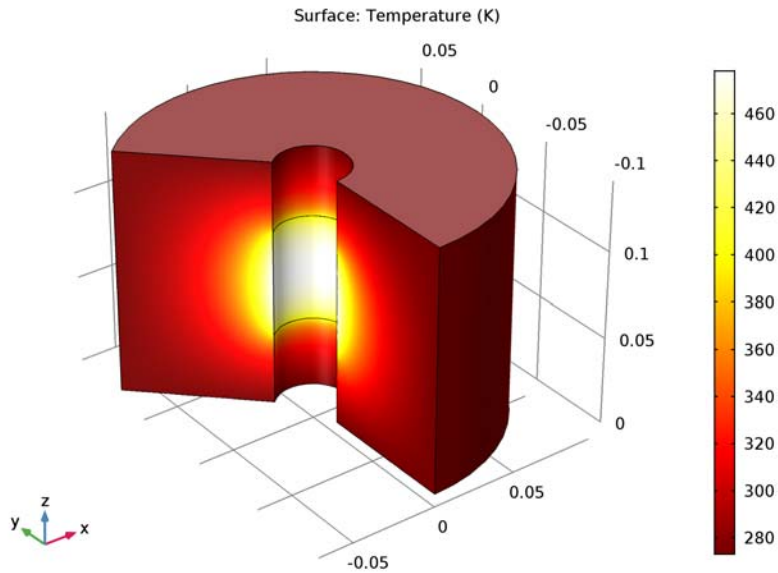


Figure 2: Temperature distribution.

The benchmark result for the target location ($r = 0.04$ m and $z = 0.04$ m) is a temperature of 59.82 °C (332.97 K). The COMSOL Multiphysics model, using a default mesh with about 540 elements, gives a temperature of 332.957 K at the same location.

Reference

1. A.D. Cameron, J.A. Casey, and G.B. Simpson, *NAFEMS Benchmark Tests for Thermal Analysis (Summary)*, NAFEMS, 1986.

Application Library path: `Heat_Transfer_Module/Tutorials,_Conduction/cylinder_conduction`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.08`.

4 In the **Height** text field, type `0.14`.

5 Locate the **Position** section. In the **r** text field, type `0.02`.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **r** text field, type `0.02`.

4 In the **z** text field, type `0.04`.

Point 2 (pt2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **r** text field, type `0.02`.

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

5 On the **Geometry** toolbar, click **Build All**.

HEAT TRANSFER IN SOLIDS (HT)

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.

2 In the **Settings** window for Solid, locate the **Heat Conduction, Solid** section.

3 From the k list, choose **User defined**. In the associated text field, type `52`.

4 Locate the **Thermodynamics, Solid** section. From the C_p list, choose **User defined**. From the ρ list, choose **User defined**.

Temperature 1

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

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

3 In the T_0 text field, type `273.15[K]`.

4 Select Boundaries 2, 5, and 6 only.

Heat Flux 1

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

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

3 In the q_0 text field, type `5e5`.

4 Select Boundary 3 only.

MESH 1

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

2 In the **Settings** window for Mesh, click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

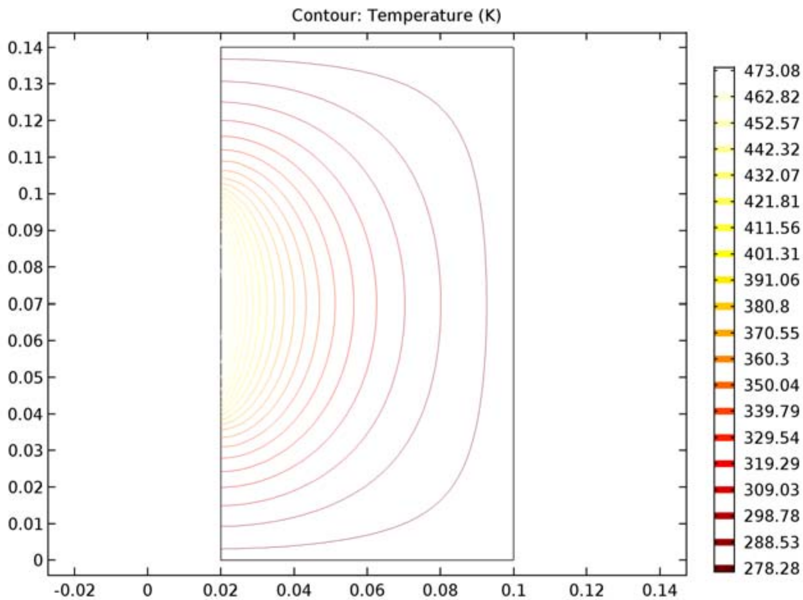
Temperature, 3D (ht)

The first default plot is a revolved 3D plot visualizing the temperature field on the surface; compare with Figure 2.

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

Isothermal Contours (ht)

The second default plot shows a contour plot of the temperature field.



To obtain the temperature value at any point, just click at that point in the Graphics window; The result appears in the Table window at the bottom of the COMSOL Desktop.

Alternatively, you can create a Cut Point data set and Point Evaluation feature as follows.

Cut Point 2D 1

1 On the **Results** toolbar, click **Cut Point 2D**.

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

3 In the **r** text field, type `0.04`.

4 In the **z** text field, type `0.04`.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.

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

3 From the **Data set** list, choose **Cut Point 2D 1**.

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result is approximately 333 K.

Radiative Heat Transfer in Finite Cylindrical Media

This application uses the discrete ordinates method (DOM) to solve a 3D radiative transfer problem in an emitting, absorbing, and linearly anisotropic-scattering finite cylindrical medium. Using the S6 quadrature of DOM leads to accurate results, which are needed in combined modes of heat transfer. The calculated incident radiation and heat fluxes agree well with published results obtained by transformed integral methods (see Ref. 1). In addition, the Heat Transfer Module's DOM formulation easily handles the effects of boundary emission and reflection.

Introduction

There are numerous engineering applications of radiative transfer in absorbing, emitting, and anisotropically scattering media with variable radiation properties. Examples include, among others, coal-fired combustion systems, light-weight fibrous insulations, and heat transfer systems containing small scattering particles. Furthermore, the efficiency of radiative transfer depends on the boundary conditions, for example, the temperature and the emissivity of the surrounding walls, and the target where heat transfer is desired. Studies have shown that radiative transfer is highly sensitive to the wall emissivity. In this study you build a validation model representing a cylinder with homogeneous walls. Then you go on to consider walls with space-dependent emissivity and investigate the effects of albedo and scattering.

Model Definition

In this tutorial you validate the DOM formulation in COMSOL Multiphysics by examining three benchmark cases. You investigate the method's efficiency through parametric analyses by changing the single-scattering albedo, wall emissivity, and linear function. In particular, in the final case, you approximate the scattering phase function by a linear function reflecting highly backward, isotropic, and highly forward scattering.

The model geometry, shown in Figure 1, is a cylinder of radius $R = 0.5$ m and height $L = 1$ m.

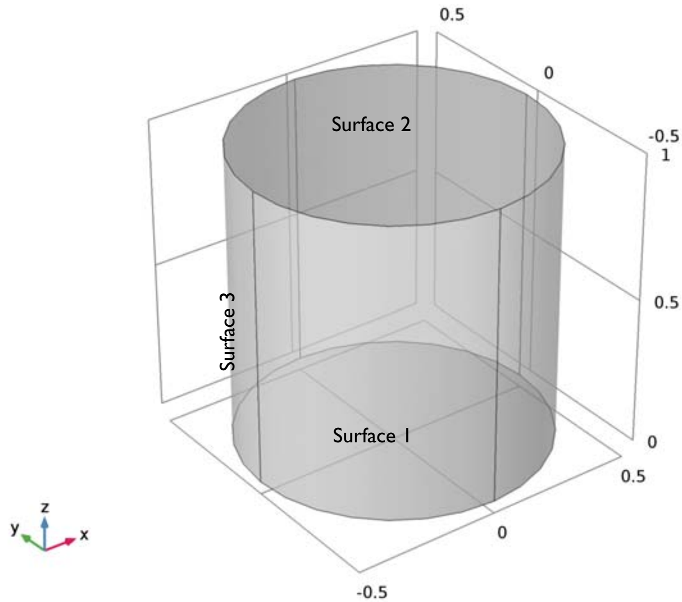


Figure 1: Schematic diagram of the physical model.

These examples use the S6 discrete ordinate method for predicting the heat flux on the enclosure side walls and the incident radiation distribution inside the domain.

VERIFICATION CASE

For the initial study, assume cold boundaries, that is, $I_i = 0$ where $i = 1, 2, 3$ refers to the surface index in Figure 1. Furthermore, assume that the walls diffusively reflect radiation, that is, $\varepsilon_i = 0.5$ for $i = 1, 2, 3$. The medium is at a uniform temperature T such that the blackbody radiation intensity in an arbitrary direction per unit area and solid angle $I_b(T) = \sigma T^4/\pi$ equals $1\ \mathrm{W/(m^2{\cdot}sr)}$.

WALLS WITH VARIABLE EMISSIVITY

Two cases are computed for comparison purposes. These 3D cases represent opaque partial side wall diffuse emission and/or reflection. In both cases, the radiosity on the side walls (Surface 3) varies with the angular coordinate along the full height of the finite cylinder according to $\varepsilon_3 = (1 - y/R)/2$. Both cases also have cold walls at the cylinder top (Surface 1) and bottom (Surface 2).

Case 1 has isotropic scattering function and compares results for different albedos. Case 2 has constant albedo and compares results with highly forward, isotropic, and highly backward scattering function parameterized by the Legendre coefficient a_1.

TABLE 1: NONSTANDARD TEST CASES

CASE	MEDIUM PROPERTIES)
1	albedo = 0.1, 0.5, 0.9 $a_1 = 0$
2	albedo = 0.5 $a_1 = -0.99, 0, 0.99$

THERMAL ANALYSIS

The discrete ordinates method relies on the discrete representation of the directional dependence of the radiation intensity. It involves solving the radiative equation for a set of directions that span the full solid angle range of 4π around a point in space.

The radiation transport equation (RTE) for this type of configuration can be written as

$$\Omega \cdot \nabla I(\Omega, s) = \kappa I_b(T) - \beta I(\Omega, s) + \frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\Omega, \Omega')\phi(\Omega, \Omega')\partial\Omega'$$

where

- $I(\Omega, s)$ is the radiation intensity at a given position s in the direction Ω
- T is the temperature
- κ, β, and σ_s are absorption, extinction, and scattering coefficients, respectively
- $I_b(T)$ is the blackbody radiation intensity
- $\phi(\Omega, \Omega') = 1 + a_1\mu_0$ where $\mu_0 = \Omega \cdot \Omega'$ is the cosine of the scattering angle.

The boundary intensities at the cylinder walls are given by the sum of the effective emitted intensity and the reflected incident intensities in the given direction:

$$I_{bnd}(\Omega) = \varepsilon_w I_b(T) + \frac{\rho_d}{\pi}q_{out} \qquad \text{for all } \Omega \text{ such that } \mathbf{n} \cdot \Omega < 0$$

where

- ε_w is the surface emissivity, which is in the range $[0, 1]$
- $\rho_d = 1 - \varepsilon_w$ is the diffusive reflectivity
- $\mathbf{n}$ is the outward normal vector

- q_{out} is the heat flux striking the wall:

$$q_{\text{out}} = \int_{\mathbf{n} \cdot \mathbf{\Omega}_j > 0} (\mathbf{n} \cdot \Omega) I(\Omega, s) \partial\Omega'$$

The above equations can be discretized in Cartesian coordinates for monochromatic or gray radiation as

$$\Omega_i \cdot \nabla I_i = \kappa I_{\text{b}}(T) - \beta I_i + \frac{\sigma_{\text{s}}}{4\pi} \int_0^{4\pi} I(\Omega, \Omega') \phi(\Omega, \Omega') \partial\Omega'$$

The Sn approximation of the RTE in the direction i can be expressed as

$$\Omega_i \cdot \nabla I_i = \kappa I_{\text{b}}(T) - \beta I_i + \frac{\sigma_{\text{s}}}{4\pi} \sum_{j=1}^{n} w_j I_j \phi(\Omega_j, \Omega_i)$$

For a discrete direction, Ω_i, the values of $\Omega_{i,1}$, $\Omega_{i,2}$, and $\Omega_{i,3}$ define the direction cosines of Ω_i obeying the condition $\Omega_{i,1}{}^2 + \Omega_{i,2}{}^2 + \Omega_{i,3}{}^2 = 1$. The index j in the above equation denotes the direction of incoming radiation contributing to the direction Ω_i.

For a diffuse reflecting surface on a wall boundary, the boundary condition equation is transformed as

$$I_{i,\text{bnd}} = \varepsilon_{\text{w}} I_{\text{b}}(T) + \frac{\rho_{\text{d}}}{\pi} \sum_{\mathbf{n} \cdot \mathbf{\Omega}_j > 0} w_j I_j \mathbf{n} \cdot \Omega_j \qquad \text{for all } \Omega_i \text{ such that } \mathbf{n} \cdot \Omega_i < 0$$

Results and Discussion

The results demonstrate that the DOM procedure for the prediction of radiation is an elegant and accurate method for modeling multidimensional radiative heat transfer in cylindrical geometries.

VERIFICATION CASE

This case treats the effects of the scattering albedo on the incident radiation and heat fluxes. Figure 2 shows the distribution of the net heat flux $q_{\text{r, net}}(R, 0, z)$ versus axial optical thickness. There is good overall agreement of the present work with published literature results (Ref. 2, Ref. 3, and Ref. 4).

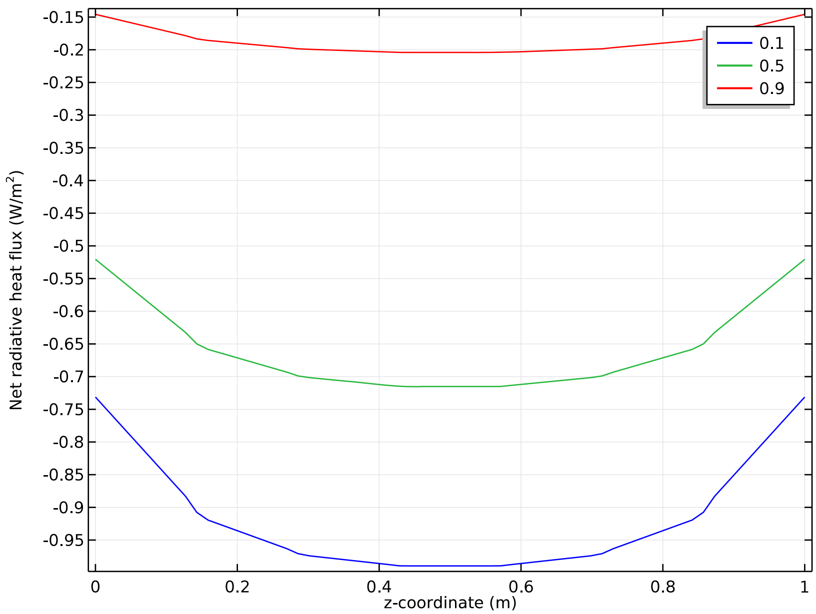


Figure 2: The effects of the scattering albedo on the radial heat flux $q_{\mathrm{r,\ net}}(R, 0, z)$*; for a hot cylindrical medium enclosed by cold walls,* $\varepsilon_1 = \varepsilon_2 = \varepsilon_3 = 0.5$.

The effects of albedo on the distribution of centerline incident radiation in axial direction are shown in Figure 3. The incident radiation is symmetric with respect to $z = L/2$ plane and decrease with increasing scattering albedo. Furthermore, results become more uniform with larger scattering albedo.

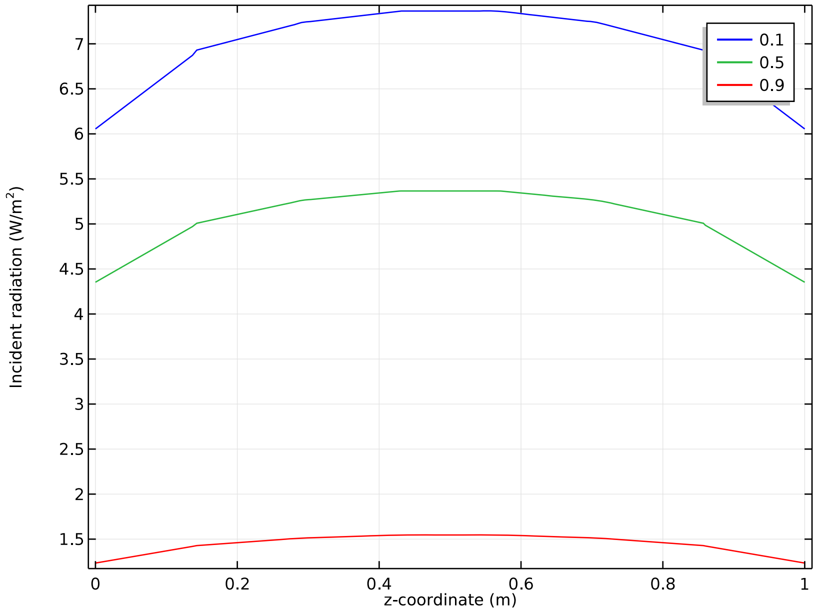


Figure 3: The effects of the scattering albedo on the distribution of centerline incident radiation $G(0, 0, z)$ *for a hot cylindrical medium enclosed by cold walls,* $\varepsilon_1 = \varepsilon_2 = \varepsilon_3 = 0.5$.

Figure 4 displays the distributions of the incident radiation across the mid-plan radius $G(x, 0, L/2)$ with respect to normalized optical thickness x/R.

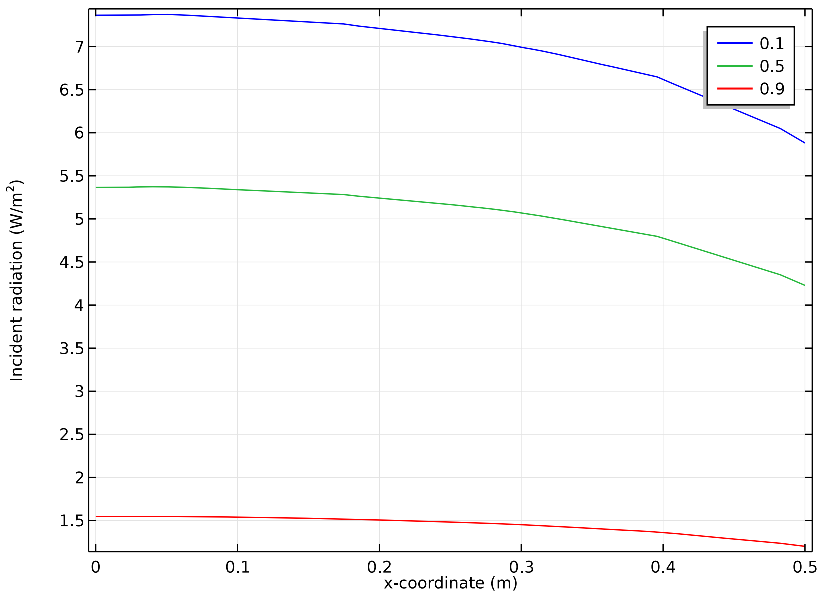


Figure 4: The effects of the scattering albedo on the distributions of the incident radiation $G(x, 0, L/2)$ *with respect to normalized optical thickness* x/R *for a hot cylindrical medium enclosed by cold walls,* $\varepsilon_1 = \varepsilon_2 = \varepsilon_3 = 0.5$.

WALLS WITH VARIABLE EMISSIVITY

The incident radiation at the mid-plane $z = L/2$ at the radial position $R/2$ is shown in Figure 5. At the side surface $R/2$ distance from the cylinder axis, $G(R, 0, L/2)$ changes with the azimuthal angle. The changes in scattering albedo are also illustrated. The smallest albedo makes the biggest change around the azimuthal angle.

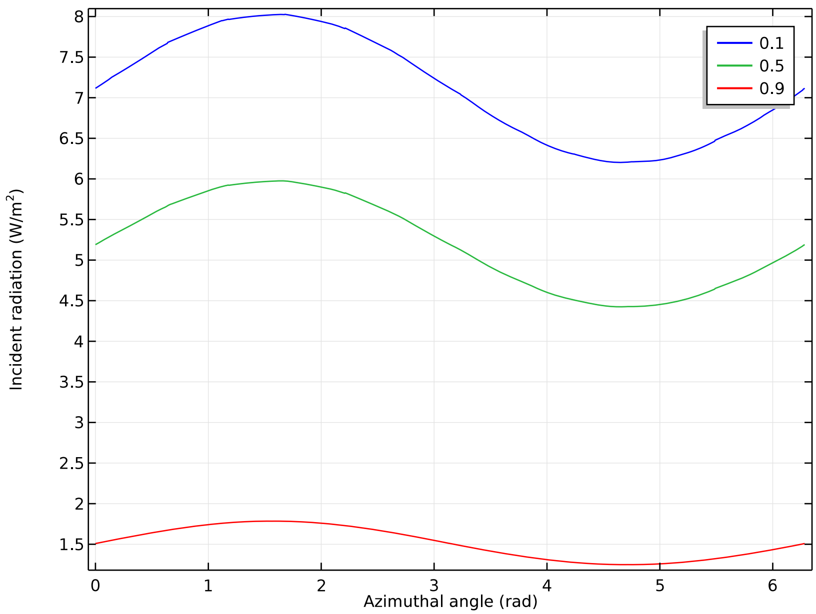


Figure 5: The effects of scattering albedo on the distribution of incident radiation at mid-plane $z = L/2$ at $R/2$ distance from the cylinder axis with respect to an azimuthal coordinate for Case 1.

Figure 6 shows the effect of a nonzero linear anisotropic scattering coefficient a_1. As expected, the differences between isotropic scattering, forward scattering, and backward scattering are most accentuated far from the boundary.

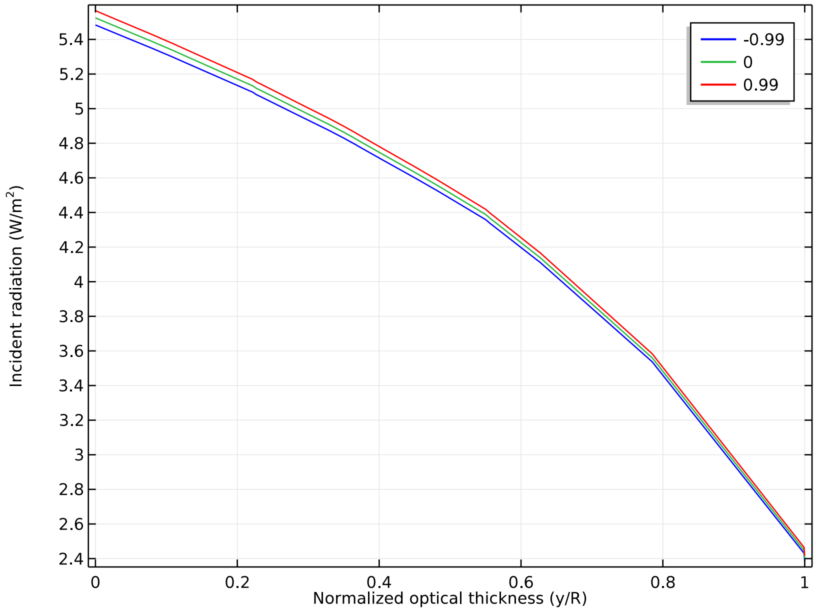


Figure 6: The effects of linear anisotropic scattering coefficient a_1 on the distribution of incident radiation $G(0, y, L/2)$ with respect to normalized optical thickness $-y/R$ for Case 2.

References

1. X.L. Chen and W.H. Sutton, "Radiative Transfer in Finite Cylindrical Media Using Transformed Integral Equations," *J. Quantitative Spectroscopy and Radiative Transfer*, vol. 77, pp. 233–271, 2003.

2. X. Chen, *Transformed Integral Equations of Radiative Transfer and Combined Convection-radiation Heat Transfer Enhancement with Porous Insert*, Doctoral Thesis, University of Oklahoma, 2003.

3. S.T. Thynell and M.N. Ozisik, "Radiation Transfer in Absorbing, Emitting, Isotropically Scattering, Homogeneous Cylindrical Media," *J. Quantitative Spectroscopy and Radiative Transfer*, vol. 38, no. 6, pp. 413–426, 1987.

4. H.Y. Li, M.N. Ozisik, and J.R. Tsai, “Two-dimensional radiation in a cylinder with spatially varying albedo,” *J. Thermophysics and Heat Transfer*, vol. 6, no. 1, pp. 180–182, 1992.

Application Library path: `Heat_Transfer_Module/Verification_Examples/cylinder_participating_media`

Modeling Instructions—Validation Case

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 **Heat Transfer>Radiation>Radiation in Participating Media (rpm)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
T0	(1[W/m^2]*pi/sigma_const)^(1/4)	86.275 K	Body temperature
Tw	0[K]	0 K	Wall temperature
ew	0.5	0.5	Wall emissivity

Name	Expression	Value	Description
omega	0.9	0.9	Single-scattering albedo
sigma_s	omega	0.9	Scattering coefficient
kappa	sigma_s*(1/omega-1)	0.1	Absorption coefficient
R	0.5[m]	0.5 m	Cylinder radius
L	1[m]	1 m	Cylinder length

GEOMETRY 1

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type R.
4 In the **Height** text field, type L.
5 Click **Build All Objects**.

MATERIALS

Add a material to specify the absorption and scattering coefficients inside the cylinder.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for Material, type Domain in the **Label** text field.
3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Absorption coefficient	kappaR	kappa	1/m	Basic
Scattering coefficient	sigmaS	sigma_s	1/m	Basic

Analogously, specify the emissivity of the walls using a material.

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for Material, type Walls in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

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

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

Property	Name	Value	Unit	Property group
Surface emissivity	epsilon_rad	ew	1	Basic

RADIATION IN PARTICIPATING MEDIA (RPM)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Radiation in Participating Media (rpm)**.

2 In the **Settings** window for Radiation in Participating Media, locate the **Participating Media Settings** section.

3 From the P_{index} list, choose **0.3**.

With this performance index value, solving the model requires roughly 2GB of RAM. If your computer has less available memory than that, try a value in the range 0.5 – 1.

4 From the **Discrete ordinates method** list, choose **S6**.

Radiation in Participating Media 1

1 In the **Model Builder** window, expand the **Radiation in Participating Media (rpm)** node, then click **Radiation in Participating Media 1**.

2 In the **Settings** window for Radiation in Participating Media, locate the **Model Inputs** section.

3 In the T text field, type T0.

Opaque Surface 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Radiation in Participating Media (rpm)** click **Opaque Surface 1**.

2 In the **Settings** window for Opaque Surface, locate the **Model Inputs** section.

3 In the T text field, type Tw.

MESH 1

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

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

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

Free Triangular 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundary 4 only.

3 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

In the **Settings** window for Swept, click **Build All**.

STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
omega	0.1 0.5 0.9	

Before solving, switch to an iterative solver to reduce memory consumption.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

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

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node.

5 Right-click **Iterative 1** and choose **Enable**.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** node.

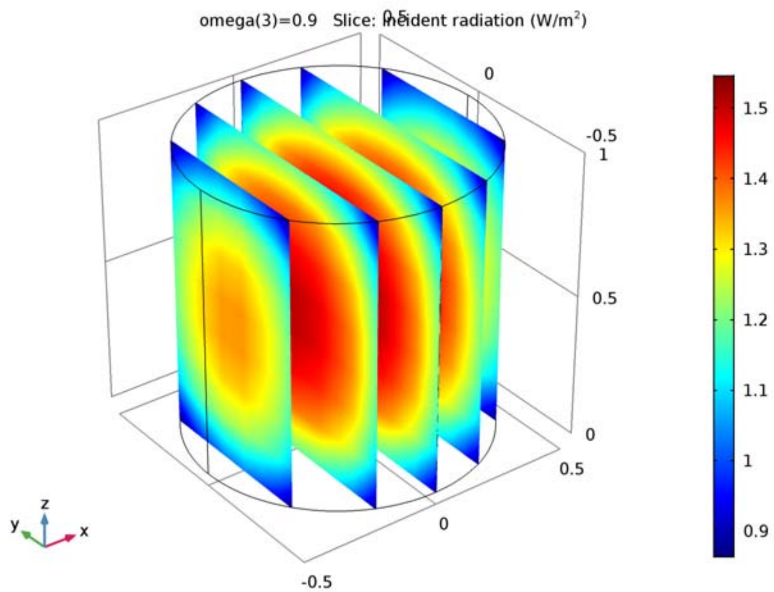
7 Right-click **Incomplete LU** and choose **Enable**.

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

RESULTS

Incident Radiation (rpm)

The first default plot shows the incident radiation distribution in 3D and the second default plot represents the net radiative heat flux distribution in 3D, see figure below.



Add a new 1D Plot to represent the net radiative heat flux along the z-coordinate and compare with Figure 2.

1D Plot Group 3

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

2 In the **Settings** window for 1D Plot Group, type `Net Radiative Heat Flux vs z, 1D` in the **Label** text field.

Line Graph 1

1 On the **Net Radiative Heat Flux vs z, 1D** toolbar, click **Line Graph**.

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

3 Select Edge 12 only.

4 Click the **Transparency** button on the **Graphics** toolbaragain to return to the original state.

5 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 **Model>Component 1>Radiation in Participating Media>Boundary fluxes>rpm.qr_net - Net radiative heat flux**.
6 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry>Coordinate>z - z-coordinate**.
7 Click to expand the **Legends** section. Select the **Show legends** check box.

Finish the plot by adjusting the title and axis labels.

Net Radiative Heat Flux vs z, 1D

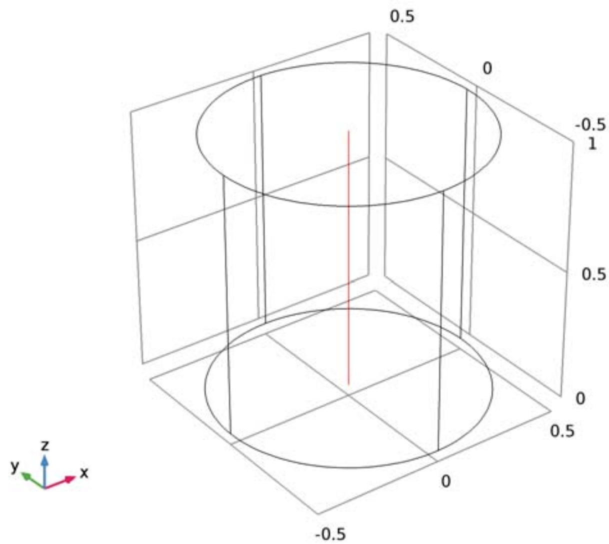
1 In the **Model Builder** window, under **Results** click **Net Radiative Heat Flux vs z, 1D**.
2 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.
3 From the **Title type** list, choose **None**.
4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
5 Select the **y-axis label** check box.
6 On the **Net Radiative Heat Flux vs z, 1D** toolbar, click **Plot**.

Cut Line 3D 1

1 On the **Results** toolbar, click **Cut Line 3D**.
2 In the **Settings** window for Cut Line 3D, locate the **Line Data** section.
3 In row **Point 2**, set **x** to 0 and **z** to L.

4 Click **Plot**.

The Graphics window shows the location of the line in the model geometry.



Add a new 1D Plot to represent the incident radiation along the z-coordinate and compare with Figure 3.

1D Plot Group 4

1. On the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type `Incident Radiation vs z, 1D` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Cut Line 3D 1**.

Line Graph 1

1. On the **Incident Radiation vs z, 1D** toolbar, click **Line Graph**.
2. In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry> Coordinate>z - z-coordinate**.
3. Locate the **Legends** section. Select the **Show legends** check box.

Incident Radiation vs z, 1D

1. In the **Model Builder** window, under **Results** click **Incident Radiation vs z, 1D**.
2. In the **Settings** window for 1D Plot Group, locate the **Title** section.
3. From the **Title type** list, choose **None**.

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

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

6 On the **Incident Radiation vs z, 1D** toolbar, click **Plot**.

Cut Line 3D 2

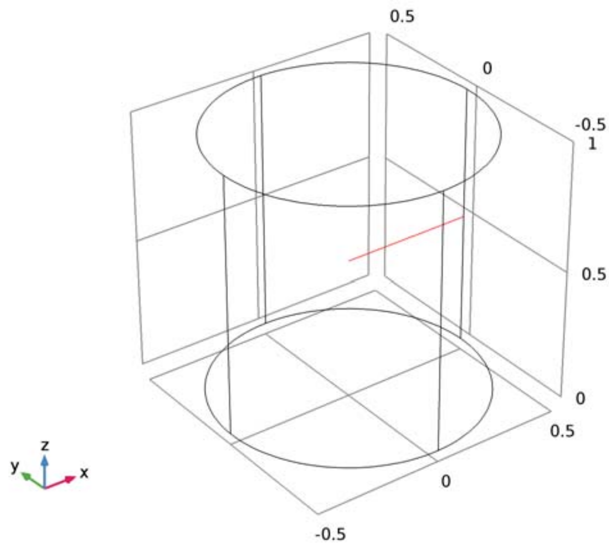
1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for Cut Line 3D, locate the **Line Data** section.

3 In row **Point 1**, set **z** to `L/2`.

4 In row **Point 2**, set **x** to `R` and **z** to `L/2`.

5 Click **Plot**.



Add a new 1D Plot to represent the incident radiation along the x-coordinate and compare with Figure 4.

Incident Radiation vs z, 1D

Right-click **Incident Radiation vs z, 1D** and choose **Duplicate**.

Incident Radiation vs z, 1D 1

1 In the **Settings** window for 1D Plot Group, type `Incident Radiation vs x, 1D` in the **Label** text field.

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

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `x-coordinate (m)`.

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Incident Radiation vs x, 1D** node, then click **Line Graph 1**.

2 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry> Coordinate>x - x-coordinate**.

3 On the **Incident Radiation vs x, 1D** toolbar, click **Plot**.

Modeling Instructions—Case 1

RADIATION IN PARTICIPATING MEDIA (RPM)

Opaque Surface 2

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

2 In the **Settings** window for Opaque Surface, locate the **Model Inputs** section.

3 In the T text field, type `Tw`.

4 Select Boundaries 1, 2, 5, and 6 only.

These are the vertical wall boundaries. To reach all of them, you can rotate the geometry or click either the **Transparency** button or the **Wireframe Rendering** button on the **Graphics** toolbar.

5 Locate the **Wall Settings** section. From the ε list, choose **User defined**. In the associated text field, type `ew*(1-y/R)`.

Now, disable **Opaque Surface 2** in **Study 1** to be able to re run the same **Study 1** configuration.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

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

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Radiation in Participating Media (rpm)>Opaque Surface 2**.

5 Click **Disable**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 In the **Select Study** tree, select **Preset Studies>Stationary**.

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

STUDY 2

Step 1: Stationary

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

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
omega	0.1 0.5 0.9	

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Dependent Variables 1**.

4 In the **Settings** window for Dependent Variables, locate the **General** section.

5 From the **Defined by study step** list, choose **User defined**.

6 Locate the **Initial Values of Variables Solved For** section. From the **Method** list, choose **Solution**.

7 From the **Solution** list, choose **Solution 1 (sol1)**.

8 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node.

9 Right-click **Iterative 1** and choose **Enable**.

10 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Iterative 1** node.

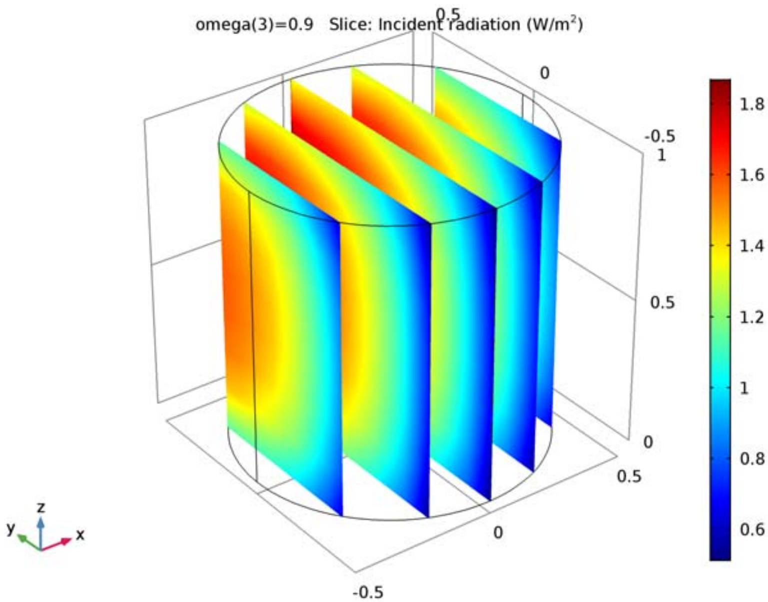
11 Right-click **Incomplete LU** and choose **Enable**.

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

RESULTS

Incident Radiation (rpm) 1

The first default plot shows the incident radiation distribution in 3D and the second default plot represents the net radiative heat flux distribution in 3D, see figure below.



Parameterized Curve 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 3D**.

2 In the **Settings** window for Parameterized Curve 3D, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Parameter** section. In the **Name** text field, type `phi`.

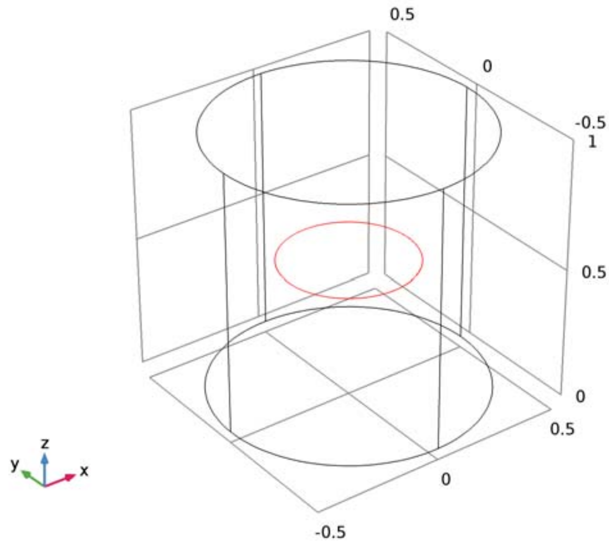
5 In the **Maximum** text field, type `2*pi`.

6 Locate the **Expressions** section. In the **x** text field, type `R*cos(phi)/2`.

7 In the **y** text field, type `R*sin(phi)/2`.

8 In the **z** text field, type `L/2`.

9 Click **Plot**.



Add a new 1D Plot to represent the incident radiation in function of the azimuthal angle and compare with Figure 5.

1D Plot Group 8

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

2 In the **Settings** window for 1D Plot Group, type `Incident Radiation vs Azimuthal Angle, 1D` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Parameterized Curve 3D 1**.

Line Graph 1

1 On the **Incident Radiation vs Azimuthal Angle, 1D** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Legends** section.

3 Select the **Show legends** check box.

Incident Radiation vs Azimuthal Angle, 1D

1 In the **Model Builder** window, under **Results** click **Incident Radiation vs Azimuthal Angle, 1D**.

2 In the **Settings** window for 1D Plot Group, locate the **Title** section.

3 From the **Title type** list, choose **None**.

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

5 In the associated text field, type `Azimuthal angle (rad)`.

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

7 On the **Incident Radiation vs Azimuthal Angle, 1D** toolbar, click **Plot**.

Modeling Instructions—Case 2

GLOBAL DEFINITIONS

Parameters

For Case 2, you need to modify the value of the single-scattering albedo that you defined previously and add a parameter for the Legendre coefficient `a1` in the scattering phase function.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
omega	0.5	0.5	Single-scattering albedo
a1	0.99	0.99	Legendre coefficient

RADIATION IN PARTICIPATING MEDIA (RPM)

Radiation in Participating Media 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Radiation in Participating Media (rpm)** click **Radiation in Participating Media 1**.

2 In the **Settings** window for Radiation in Participating Media, locate the **Scattering** section.

3 From the **Scattering type** list, choose **Linear anisotropic**.

4 In the a_1 text field, type `a1`.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 In the **Select Study** tree, select **Preset Studies>Stationary**.

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

STUDY 3

Step 1: Stationary

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

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.
3 Click **Add**.
4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
a1	-0.99 0 0.99	

Note that the value a1 = 0 gives the same solution as for omega = 0.5 in Case 1.

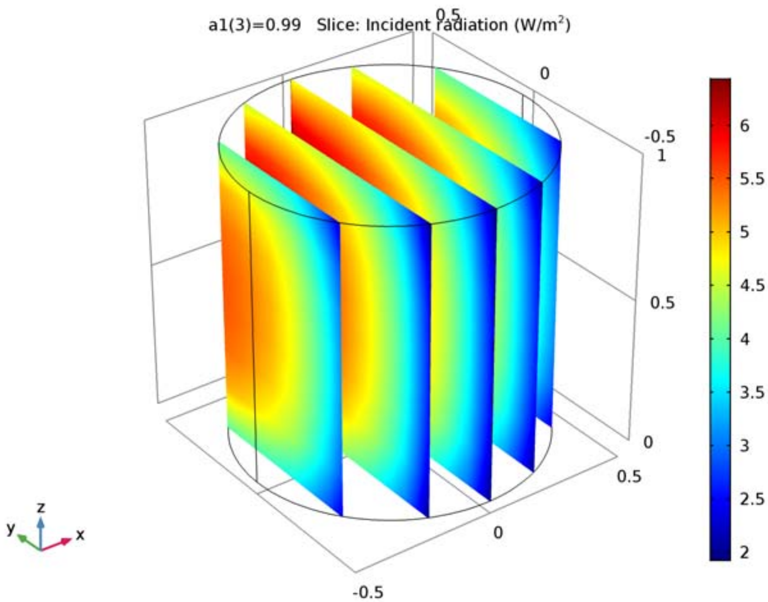
Solution 3 (sol3)

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Study 3>Solver Configurations** node.
3 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Dependent Variables 1**.
4 In the **Settings** window for Dependent Variables, locate the **General** section.
5 From the **Defined by study step** list, choose **User defined**.
6 Locate the **Initial Values of Variables Solved For** section. From the **Method** list, choose **Solution**.
7 From the **Solution** list, choose **Solution 2 (sol2)**.
8 In the **Model Builder** window, expand the **Study 3>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node.
9 Right-click **Iterative 1** and choose **Enable**.
10 In the **Model Builder** window, expand the **Study 3>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1>Iterative 1** node.
11 Right-click **Incomplete LU** and choose **Enable**.
12 On the **Study** toolbar, click **Compute**.

RESULTS

Incident Radiation (rpm) 2

The first default plot shows the incident radiation distribution in 3D and the second default plot represents the net radiative heat flux distribution in 3D, see figure below.



Parameterized Curve 3D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 3D**.

2 In the **Settings** window for Parameterized Curve 3D, locate the **Data** section.

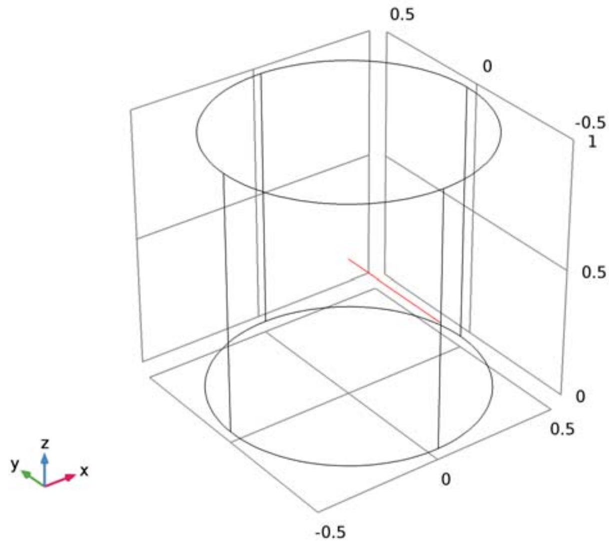
3 From the **Data set** list, choose **Study 3/Solution 3 (sol3)**.

4 Locate the **Expressions** section. In the **y** text field, type `-s*R`.

5 In the **z** text field, type `L/2`.

With the above definition, $s = -y/R$ equals the optical thickness along the negative y-axis for the given constant values of x and z.

6 Click **Plot**.



Add a new 1D Plot to represent the incident radiation in function of the normalized optical thickness and compare with Figure 6.

1D Plot Group 11

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

2 In the **Settings** window for 1D Plot Group, type `Incident Radiation vs Normalized Optical Thickness, 1D` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Parameterized Curve 3D 2**.

Line Graph 1

1 On the **Incident Radiation vs Normalized Optical Thickness, 1D** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `s`.

5 Locate the **Legends** section. Select the **Show legends** check box.

Incident Radiation vs Normalized Optical Thickness, 1D

1 In the **Model Builder** window, under **Results** click **Incident Radiation vs Normalized Optical Thickness, 1D**.

2 In the **Settings** window for 1D Plot Group, locate the **Title** section.

3 From the **Title type** list, choose **None**.

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

5 In the associated text field, type `Normalized optical thickness (y/R)`.

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

7 On the **Incident Radiation vs Normalized Optical Thickness, 1D** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Radiative Heat Transfer in Finite Cylindrical Media—P1 Method

This application uses the P1 approximation to solve a 3D radiative transfer problem in an emitting, absorbing, and linearly anisotropic-scattering finite cylindrical medium. The calculated incident radiation and heat fluxes are compared to the results obtained with the highly accurate S6 discrete ordinates method (see Radiative Heat Transfer in Finite Cylindrical Media). The results obtained with this method show fairly good agreement with published results obtained by transformed integral methods (see Ref. 1) for optically thick media.

Introduction

There are numerous engineering applications of radiative transfer in absorbing, emitting, and anisotropically scattering media with variable radiation properties. Examples include, among others, coal-fired combustion systems, light-weight fibrous insulations, and heat transfer systems containing small scattering particles. Furthermore, the efficiency of radiative transfer depends on the boundary conditions, for example, the temperature and the emissivity of the surrounding walls, and the target where heat transfer is desired. Studies have shown that radiative transfer is highly sensitive to the wall emissivity. In this study you build a validation model representing a cylinder with homogeneous walls. Then you go on to consider walls with space-dependent emissivity and investigate the effects of albedo and scattering.

Model Definition

In this tutorial you validate the P1 formulation in COMSOL Multiphysics by examining three benchmark cases. You investigate the method's efficiency through parametric analyses by changing the single-scattering albedo, wall emissivity, and linear function. In particular, in the final case, you approximate the scattering phase function by a linear function reflecting highly backward, isotropic, and highly forward scattering. The results are compared to the results obtained with the DOM formulation as shown in Radiative Heat Transfer in Finite Cylindrical Media.

The model geometry, shown in Figure 1, is a cylinder of radius $R = 0.5$ m and height $L = 1$ m.

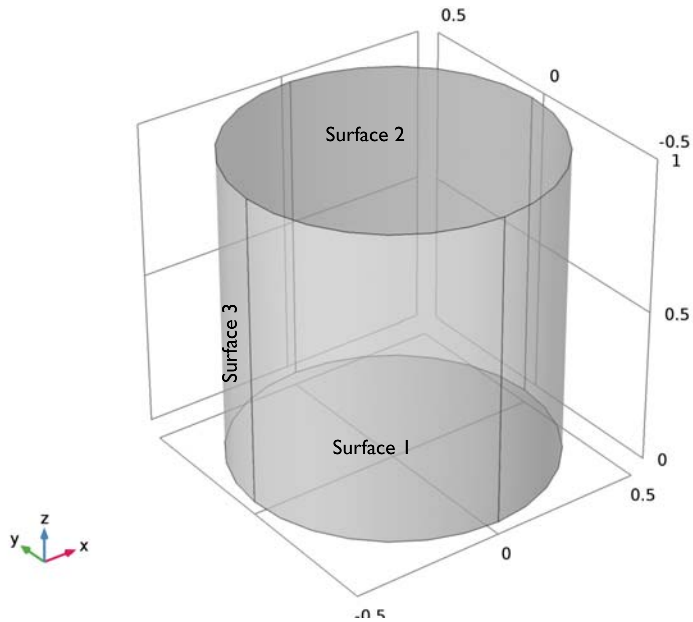


Figure 1: Schematic diagram of the physical model.

These examples use the P1 approximation method for predicting the heat flux on the enclosure side walls and the incident radiation distribution inside the domain.

VERIFICATION CASE

For the initial study, assume cold boundaries, that is, $T = 0$. Furthermore, assume that the walls diffusively reflect radiation, that is, $\varepsilon = 0.5$. The medium is at a uniform temperature T such that the blackbody radiation intensity $I_\mathrm{b}(T) = \sigma T^4/\pi$ is equal to $1\ \mathrm{W/m^2}$.

WALLS WITH VARIABLE EMISSIVITY

Two cases are computed for comparison purposes. These 3D cases represent opaque partial side wall diffuse emission and/or reflection. In both cases, the radiosity on the side walls (Surface 3) varies with the angular coordinate along the full height of the finite cylinder according to $\varepsilon_3 = (1-y/R)/2$. Both cases also have cold walls at the cylinder top (Surface 1) and bottom (Surface 2).

Case 1 has isotropic scattering function and compares results for different albedos. Case 2 has constant albedo and compares results with highly forward, isotropic, and highly backward scattering function parameterized by the Legendre coefficient a_1.

TABLE 1: NONSTANDARD TEST CASES

CASE	MEDIUM PROPERTIES)
1	albedo = 0.1, 0.5, 0.9 $a_1 = 0$
2	albedo = 0.5 $a_1 = -0.99, 0, 0.99$

THERMAL ANALYSIS

The P1 approximation method is the simplest form of the method of spherical harmonics (PN) to describe radiative transport in participating media. The radiation transport equation (RTE) for this type of configuration can be written as

$$\Omega \cdot \nabla I(\Omega, s) = \kappa I_b(T) - \beta I(\Omega, s) + \frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\Omega, \Omega')\phi(\Omega, \Omega')\partial\Omega'$$

where

- $I(\Omega, s)$ is the radiation intensity at a given position s in the direction Ω
- T is the temperature
- κ, β, and σ_s are absorption, extinction, and scattering coefficients, respectively
- $I_b(T)$ is the blackbody radiation intensity
- $\phi(\Omega, \Omega') = 1 + a_1\mu_0$ where $\mu_0 = \Omega \cdot \Omega'$ is the cosine of the scattering angle.

The RTE (an integro-partial differential equation) is transformed into a set of partial differential equations. The remaining task is to solve an additional equation for

$$G = \int_0^{4\pi} I(\Omega)d\Omega$$

which adds a heat source/sink to account for radiative heat transfer.

$$-\nabla \cdot (D_{P1}\nabla G) = Q_r$$

where D_{P1} is a diffusion coefficient and Q_r is the radiative heat source. The boundary condition for opaque surfaces then is

$$-\mathbf{n} \cdot D_{\mathrm{P1}} \nabla G = -q_{\mathrm{r,\,net}}$$

Results and Discussion

The results demonstrate that the P1 method provides a fast alternative to the DOM method. The solution time is only a few seconds whereas it is about 4 hours for the DOM method. The results represent the radiation characteristics well. The numerical values show good agreement for small scattering coefficients (`omega`) and high absorption coefficients (`kappa`). The relation between both values is $\kappa = \omega/(\omega - 1)$.

VERIFICATION CASE

This case examines the effects of the scattering albedo on the incident radiation and heat fluxes. Figure 2 shows the distribution of the net heat flux $q_{\mathrm{r,\,net}}(R, 0, z)$ versus axial optical thickness. There is good agreement with the results obtained from the DOM model. The results for the net radiative heat flux (Figure 2) shows good agreement with distance to the top and bottom boundary. The relative error for small scattering albedo is below 10% and increases with increasing scattering albedo.

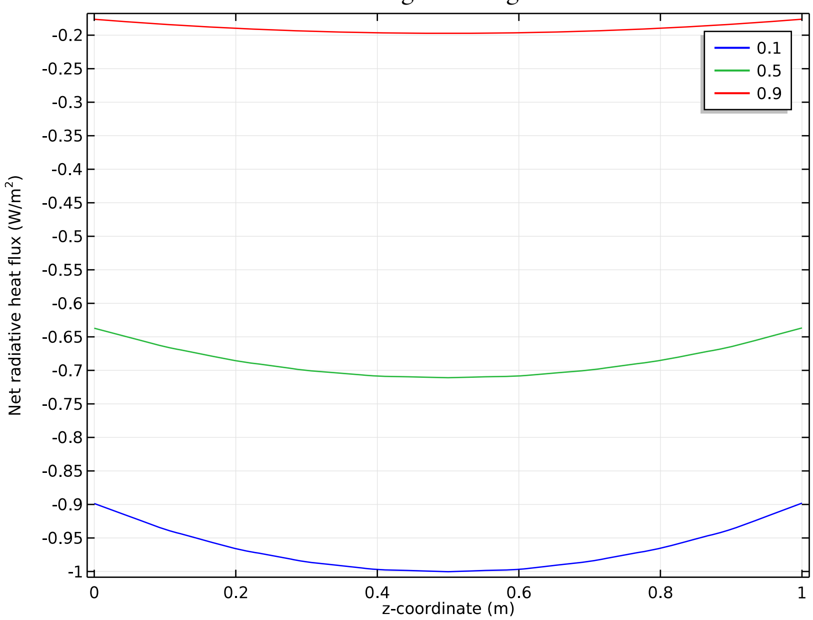


Figure 2: The effects of the scattering albedo on the radial heat flux $q_{\mathrm{r,\,net}}(R, 0, z)$*; for a hot cylindrical medium enclosed by cold walls,* $\varepsilon_1 = \varepsilon_2 = \varepsilon_3 = 0.5$.

The effects of albedo on the distribution of centerline incident radiation in axial direction are shown in Figure 3. The incident radiation is symmetric with respect to $z = L/2$ plane and decrease with increasing scattering albedo. Furthermore, results become more uniform with larger scattering albedo.

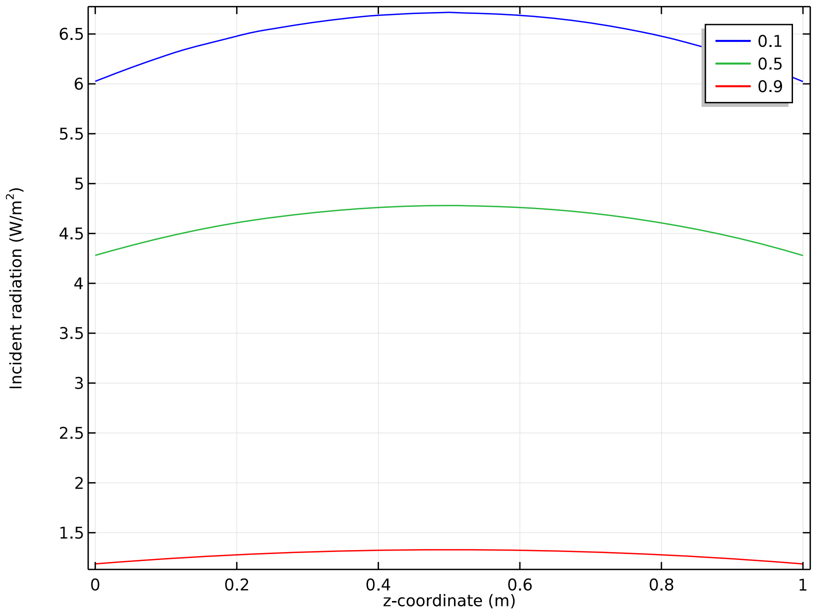


Figure 3: The effects of the scattering albedo on the distribution of centerline incident radiation $G(0, 0, z)$ *for a hot cylindrical medium enclosed by cold walls,* $\varepsilon_1 = \varepsilon_2 = \varepsilon_3 = 0.5$.

Figure 4 displays the distributions of the incident radiation across the mid-plane radius $G(x, 0, L/2)$ with respect to normalized optical thickness x/R.

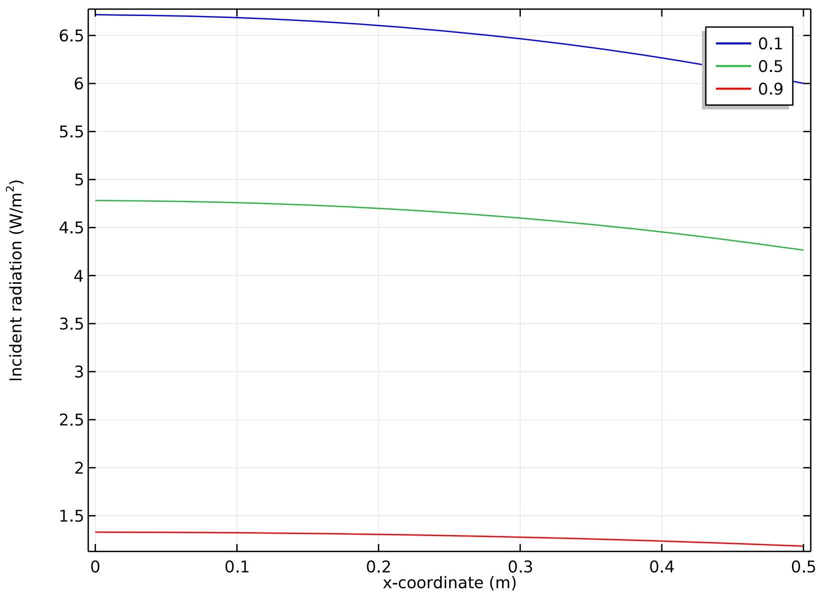


Figure 4: The effects of the scattering albedo on the distributions of the incident radiation $G(x, 0, L/2)$ *with respect to normalized optical thickness* x/R *for a hot cylindrical medium enclosed by cold walls,* $\varepsilon_1 = \varepsilon_2 = \varepsilon_3 = 0.5$.

For the validation case the relative error compared to the DOM model is below 20%.

WALLS WITH VARIABLE EMISSIVITY

The incident radiation at the mid-plane $z = L/2$ at the radial position $R/2$ is shown in Figure 5. At the side surface $R/2$ distance from the cylinder axis, $G(R, 0, L/2)$ changes with the azimuthal angle. The changes in scattering albedo are also illustrated. The smallest albedo makes the biggest change around the azimuthal angle. The numerical error for small scattering albedo again is low (around 10%).

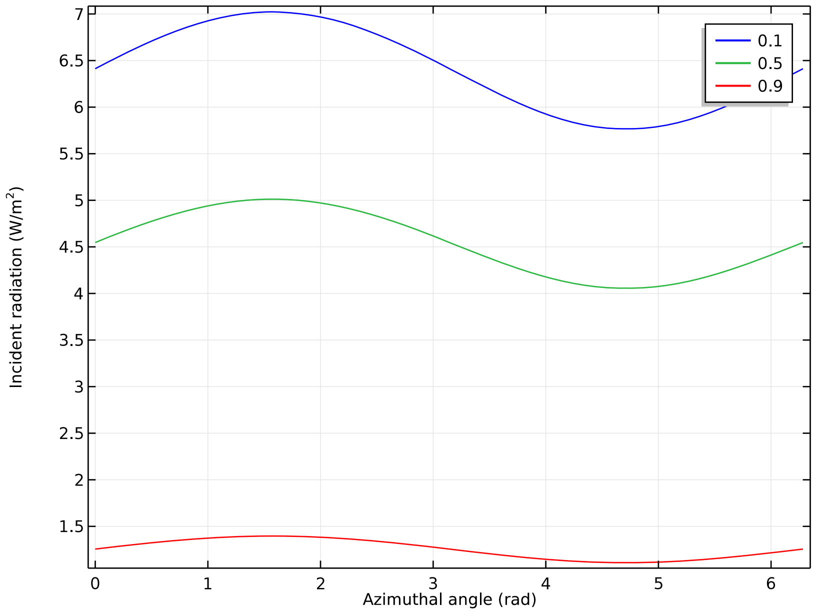


Figure 5: The effects of scattering albedo on the distribution of incident radiation at mid-plane $z = L/2$ *at* $R/2$ *distance from the cylinder axis with respect to an azimuthal coordinate for Case 1.*

Figure 6 shows the effect of a nonzero linear anisotropic scattering coefficient a_1. As expected, the differences between isotropic scattering, forward scattering, and backward scattering are most accentuated far from the boundary.

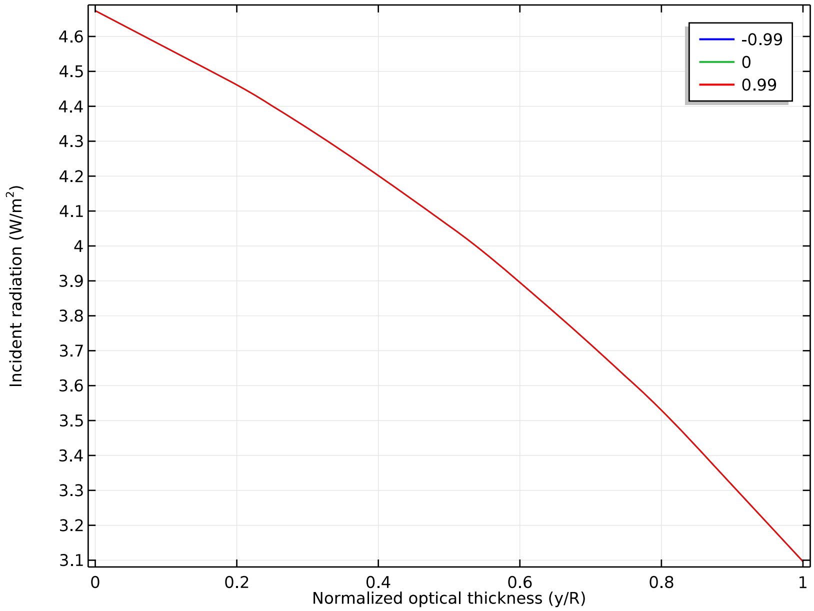


Figure 6: The effects of linear anisotropic scattering coefficient a_1 on the distribution of incident radiation $G(0, y, L/2)$ with respect to normalized optical thickness $-y/R$ for Case 2.

Unlike the discrete ordinates method, the P1 method is computationally inexpensive and gives fast results at the expense of accuracy in most cases. For large optical thickness the results show good agreement with the results obtained from the discrete ordinate method and hence with the results presented in the literature.

References

1. X.L. Chen and W.H. Sutton, "Radiative Transfer in Finite Cylindrical Media Using Transformed Integral Equations," *J. Quantitative Spectroscopy and Radiative Transfer*, vol. 77, pp. 233–271, 2003.

2. X. Chen, *Transformed Integral Equations of Radiative Transfer and Combined Convection-radiation Heat Transfer Enhancement with Porous Insert*, Doctoral Thesis, University of Oklahoma, 2003.

3. S.T. Thynell and M.N. Ozisik, “Radiation Transfer in Absorbing, Emitting, Isotropically Scattering, Homogeneous Cylindrical Media,” *J. Quantitative Spectroscopy and Radiative Transfer*, vol. 38, no. 6, pp. 413–426, 1987.

4. H.Y. Li, M.N. Ozisik, and J.R. Tsai, “Two-dimensional radiation in a cylinder with spatially varying albedo,” *J. Thermophysics and Heat Transfer*, vol. 6, no. 1, pp. 180–182, 1992.

Application Library path: `Heat_Transfer_Module/Verification_Examples/cylinder_participating_media_p1`

Modeling Instructions—Validation Case

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 **Heat Transfer>Radiation>Radiation in Participating Media (rpm)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`T0`	`(1[W/m^2]*pi/ sigma_const)^(1/4)`	86.275 K	`Body temperature`
`Tw`	`0[K]`	0 K	`Wall temperature`
`ew`	`0.5`	0.5	`Wall emissivity`
`omega`	`0.5`	0.5	`Single-scattering albedo`
`sigma_s`	`omega`	0.5	`Scattering coefficient`
`kappa`	`sigma_s*(1/omega-1)`	0.5	`Absorption coeficient`
`R`	`0.5[m]`	0.5 m	`Cylinder radius`
`L`	`1[m]`	1 m	`Cylinder length`

GEOMETRY 1

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `R`.

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

5 Click **Build All Objects**.

MATERIALS

Add a material to specify the absorption and scattering coefficients inside the cylinder.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

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

Property	Name	Value	Unit	Property group
Absorption coefficient	kappaR	`kappa`	1/m	Basic
Scattering coefficient	sigmaS	`sigma_s`	1/m	Basic

Analogously, specify the emissivity of the walls using a material.

Material 2 (mat2)

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

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

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

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

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

Property	Name	Value	Unit	Property group
Surface emissivity	epsilon_rad	`ew`	1	Basic

RADIATION IN PARTICIPATING MEDIA (RPM)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Radiation in Participating Media (rpm)**.

2 In the **Settings** window for Radiation in Participating Media, locate the **Participating Media Settings** section.

3 From the **Radiation discretization method** list, choose **P1 approximation**.

Radiation in Participating Media 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Radiation in Participating Media (rpm)** click **Radiation in Participating Media 1**.

2 In the **Settings** window for Radiation in Participating Media, locate the **Model Inputs** section.

3 In the T text field, type `T0`.

Opaque Surface 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Radiation in Participating Media (rpm)** click **Opaque Surface 1**.

2 In the **Settings** window for Opaque Surface, locate the **Model Inputs** section.

3 In the T text field, type Tw.

STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

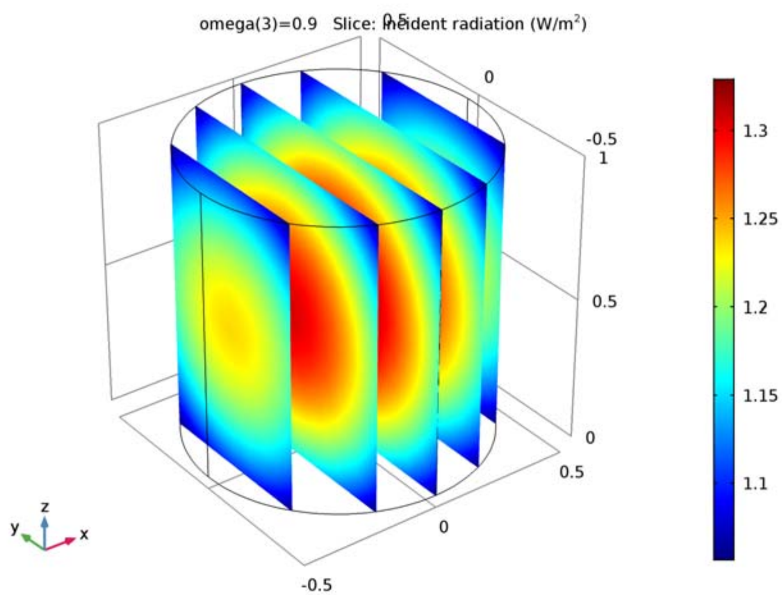
Parameter name	Parameter value list	Parameter unit
omega	0.1 0.5 0.9	

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

RESULTS

Incident Radiation (rpm)

The first default plot shows the incident radiation distribution in 3D and the second default plot represents the net radiative heat flux distribution in 3D, see figure below.



Add a new 1D Plot to represent the net radiative heat flux along the z-coordinate and compare with Figure 2.

1D Plot Group 3

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type `Net Radiative Heat Flux vs z, 1D` in the **Label** text field.

Line Graph 1

1. On the **Net Radiative Heat Flux vs z, 1D** toolbar, click **Line Graph**.
2. Click the **Transparency** button on the **Graphics** toolbar.
3. Select Edge 12 only.
4. Click the **Transparency** button on the **Graphics** toolbaragain to return to the original state.
5. 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 **Model>Component 1>Radiation in Participating Media>Boundary fluxes>rpm.qr_net - Net radiative heat flux**.
6. Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry>Coordinate>z - z-coordinate**.
7. Click to expand the **Legends** section. Select the **Show legends** check box.

Finish the plot by adjusting the title and axis labels.

Net Radiative Heat Flux vs z, 1D

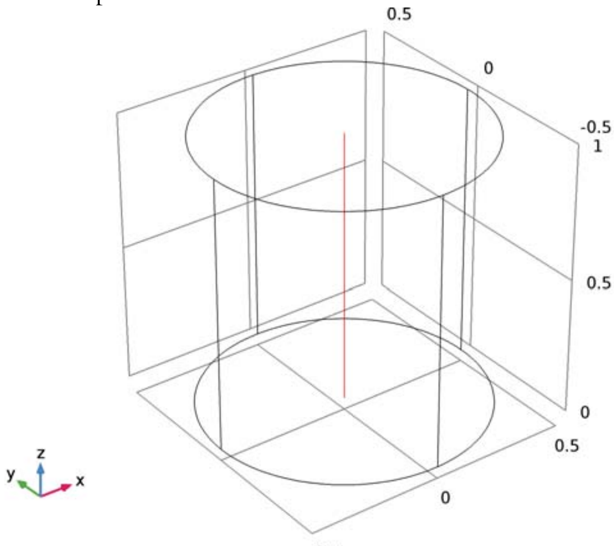
1. In the **Model Builder** window, under **Results** click **Net Radiative Heat Flux vs z, 1D**.
2. In the **Settings** window for 1D Plot Group, click to expand the **Title** section.
3. From the **Title type** list, choose **None**.
4. Locate the **Plot Settings** section. Select the **x-axis label** check box.
5. Select the **y-axis label** check box.
6. On the **Net Radiative Heat Flux vs z, 1D** toolbar, click **Plot**.

Cut Line 3D 1

1. On the **Results** toolbar, click **Cut Line 3D**.
2. In the **Settings** window for Cut Line 3D, locate the **Line Data** section.
3. In row **Point 2**, set **x** to `0`, and **z** to `L`.

4 Click **Plot**.

The Graphics window shows the location of the line in the model geometry.



Add a new 1D Plot to represent the incident radiation along the z-coordinate and compare with Figure 3.

1D Plot Group 4

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

2 In the **Settings** window for 1D Plot Group, type `Incident Radiation vs z, 1D` in the **Label** text field.

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

Line Graph 1

1 On the **Incident Radiation vs z, 1D** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry>Coordinate>z - z-coordinate**.

3 Locate the **Legends** section. Select the **Show legends** check box.

Incident Radiation vs z, 1D

1 In the **Model Builder** window, under **Results** click **Incident Radiation vs z, 1D**.

2 In the **Settings** window for 1D Plot Group, locate the **Title** section.

3 From the **Title type** list, choose **None**.

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

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

6 On the **Incident Radiation vs z, 1D** toolbar, click **Plot**.

Cut Line 3D 2

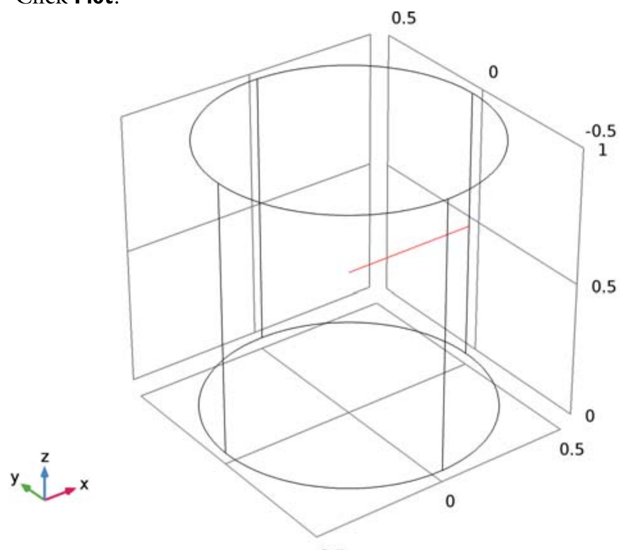
1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for Cut Line 3D, locate the **Line Data** section.

3 In row **Point 1**, set **z** to `L/2`.

4 In row **Point 2**, set **x** to `R`, and **z** to `L/2`.

5 Click **Plot**.



Add a new 1D Plot to represent the incident radiation along the x-coordinate and compare with Figure 4.

Incident Radiation vs z, 1D

Right-click **Incident Radiation vs z, 1D** and choose **Duplicate**.

Incident Radiation vs z, 1D 1

1 In the **Settings** window for 1D Plot Group, type `Incident Radiation vs x, 1D` in the **Label** text field.

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

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `x-coordinate (m)`.

Line Graph 1

1. In the **Model Builder** window, expand the **Results>Incident Radiation vs x, 1D** node, then click **Line Graph 1**.
2. In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry> Coordinate>x - x-coordinate**.
3. On the **Incident Radiation vs x, 1D** toolbar, click **Plot**.

Modeling Instructions—Case 1

RADIATION IN PARTICIPATING MEDIA (RPM)

Opaque Surface 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Opaque Surface**.
2. In the **Settings** window for Opaque Surface, locate the **Model Inputs** section.
3. In the T text field, type `Tw`.
4. Select Boundaries 1, 2, 5, and 6 only.

   These are the vertical wall boundaries. To reach all of them, you can rotate the geometry or click either the **Transparency** button or the **Wireframe Rendering** button on the **Graphics** toolbar.
5. Locate the **Wall Settings** section. From the ε list, choose **User defined**. In the associated text field, type `ew*(1-y/R)`.

Now, disable **Opaque Surface 2** in **Study 1** to be able to re run the same **Study 1** configuration.

STUDY 1

Step 1: Stationary

1. In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.
2. In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.
3. Select the **Modify physics tree and variables for study step** check box.
4. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Radiation in Participating Media (rpm)>Opaque Surface 2**.
5. Click **Disable**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 In the **Select Study** tree, select **Preset Studies>Stationary**.

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

STUDY 2

Step 1: Stationary

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

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

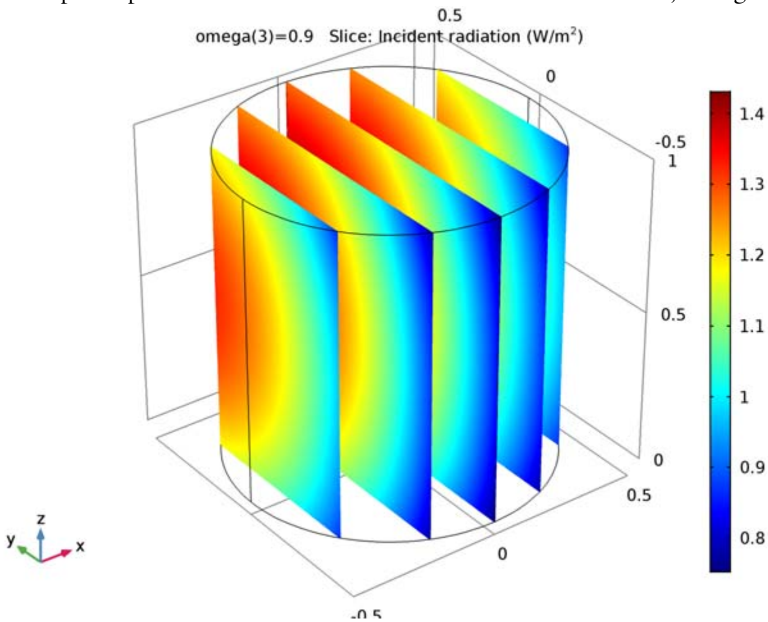
Parameter name	**Parameter value list**	**Parameter unit**
omega	`0.1 0.5 0.9`	

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

RESULTS

Incident Radiation (rpm) 1

The first default plot shows the incident radiation distribution in 3D and the second default plot represents the net radiative heat flux distribution in 3D, see figure below.



Parameterized Curve 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 3D**.

2 In the **Settings** window for Parameterized Curve 3D, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Parameter** section. In the **Name** text field, type `phi`.

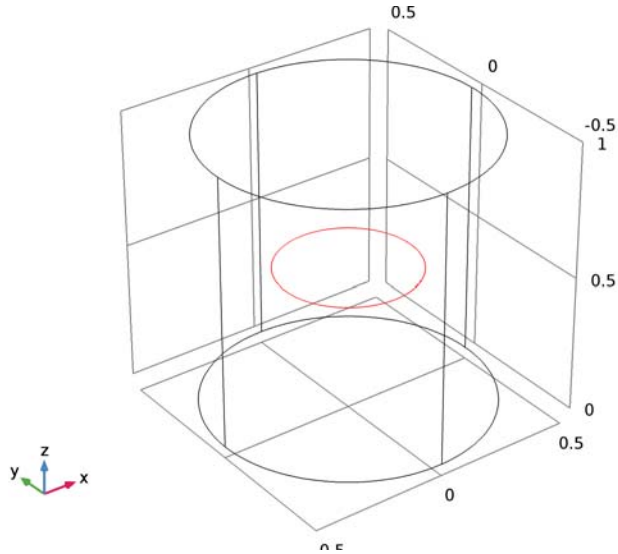
5 In the **Maximum** text field, type `2*pi`.

6 Locate the **Expressions** section. In the **x** text field, type `R*cos(phi)/2`.

7 In the **y** text field, type `R*sin(phi)/2`.

8 In the **z** text field, type `L/2`.

9 Click **Plot**.



Add a new 1D Plot to represent the incident radiation in function of the azimuthal angle and compare with Figure 5.

1D Plot Group 8

1. On the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type `Incident Radiation vs Azimuthal Angle, 1D` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Parameterized Curve 3D 1**.

Line Graph 1

1. On the **Incident Radiation vs Azimuthal Angle, 1D** toolbar, click **Line Graph**.
2. In the **Settings** window for Line Graph, locate the **Legends** section.
3. Select the **Show legends** check box.

Incident Radiation vs Azimuthal Angle, 1D

1. In the **Model Builder** window, under **Results** click **Incident Radiation vs Azimuthal Angle, 1D**.
2. In the **Settings** window for 1D Plot Group, locate the **Title** section.
3. From the **Title type** list, choose **None**.
4. Locate the **Plot Settings** section. Select the **x-axis label** check box.
5. In the associated text field, type `Azimuthal angle (rad)`.

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

7 On the **Incident Radiation vs Azimuthal Angle, 1D** toolbar, click **Plot**.

Modeling Instructions—Case 2

GLOBAL DEFINITIONS

Parameters

For case 2, you need to add a parameter for the Legendre coefficient `a1` in the scattering phase function.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
a1	0.99	0.99	Legendre coefficient

RADIATION IN PARTICIPATING MEDIA (RPM)

Add a new **Radiation in Participating Media** node for **Study 3** only.

Radiation in Participating Media 2

1 On the **Physics** toolbar, click **Domains** and choose **Radiation in Participating Media**.

2 In the **Settings** window for Radiation in Participating Media, locate the **Model Inputs** section.

3 In the T text field, type `T0`.

4 Locate the **Scattering** section. From the **Scattering type** list, choose **Linear anisotropic**.

5 In the a_1 text field, type `a1`.

STUDY 1

Step 1: Stationary

Now, disable **Radiation in Participating Media 2** in **Study 1** and **Study 2** to be able to re run the same configurations for these studies.

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 **Physics and variables selection** tree, select **Component 1 (comp1)>Radiation in Participating Media (rpm)>Radiation in Participating Media 2**.

4 Click **Disable**.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

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

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Radiation in Participating Media (rpm)>Radiation in Participating Media 2**.

5 Click **Disable**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 In the **Select Study** tree, select **Preset Studies>Stationary**.

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

STUDY 3

Step 1: Stationary

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

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
`a1`	`-0.99 0 0.99`	

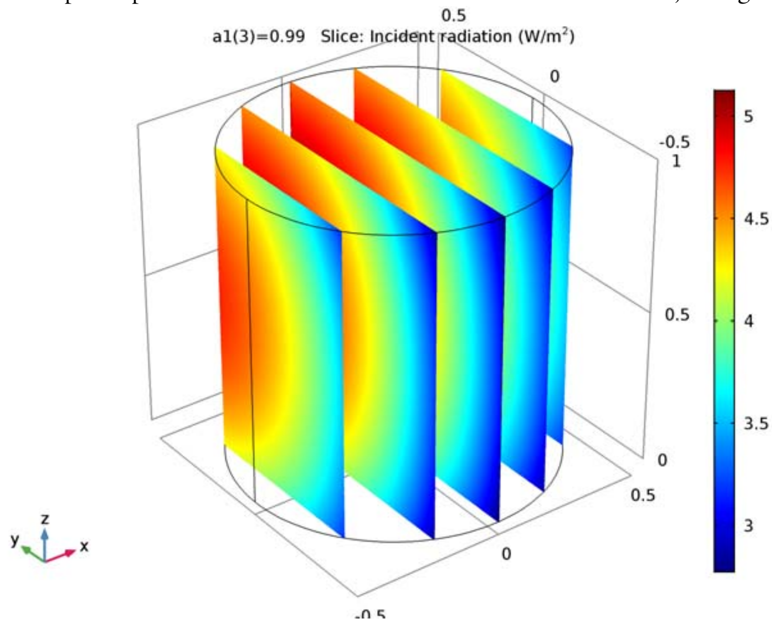
Note that the value `a1` = 0 gives the same solution as for `omega` = 0.5 in case 1.

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

RESULTS

Incident Radiation (rpm) 2

The first default plot shows the incident radiation distribution in 3D and the second default plot represents the net radiative heat flux distribution in 3D, see figure below.

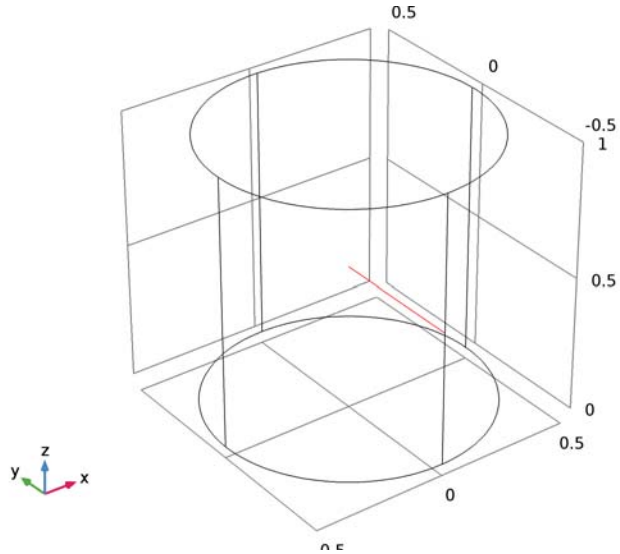


Parameterized Curve 3D 2

1. On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 3D**.
2. In the **Settings** window for Parameterized Curve 3D, locate the **Data** section.
3. From the **Data set** list, choose **Study 3/Solution 3 (sol3)**.
4. Locate the **Expressions** section. In the **y** text field, type `-s*R`.
5. In the **z** text field, type `L/2`.

   With the above definition, $s = -y/R$ equals the optical thickness along the negative y-axis for the given constant values of x and z.

6 Click **Plot**.



Add a new 1D Plot to represent the incident radiation as function of the normalized optical thickness and compare with Figure 6.

1D Plot Group 11

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

2 In the **Settings** window for 1D Plot Group, type `Incident Radiation vs Normalized Optical Thickness, 1D` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Parameterized Curve 3D 2**.

Line Graph 1

1 On the **Incident Radiation vs Normalized Optical Thickness, 1D** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `s`.

5 Locate the **Legends** section. Select the **Show legends** check box.

Incident Radiation vs Normalized Optical Thickness, 1D

1 In the **Model Builder** window, under **Results** click **Incident Radiation vs Normalized Optical Thickness, 1D**.

2 In the **Settings** window for 1D Plot Group, locate the **Title** section.

3 From the **Title type** list, choose **None**.

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

5 In the associated text field, type `Normalized optical thickness (y/R)`.

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

7 On the **Incident Radiation vs Normalized Optical Thickness, 1D** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Disk-Stack Heat Sink

Introduction

This example studies the cooling effects of a disk-stack heat sink on an electronic component. The heat sink shape (see Figure 1) shows several thin aluminum disks piled up around a central hollow column. Such a configuration allows cooling of large surfaces of aluminum fins by air at ambient temperature.

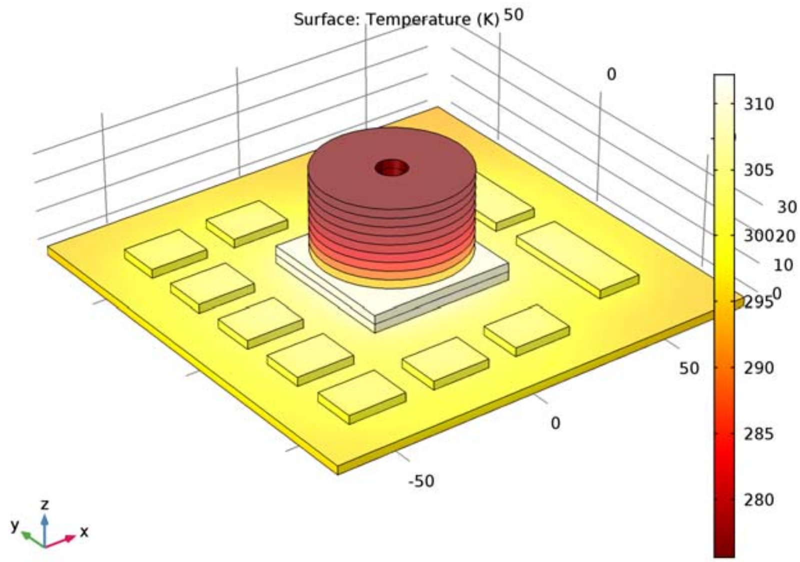


Figure 1: Steady-state surface temperature distribution of the electronic device.

To evaluate the efficiency of the heat sink, this tutorial follows a typical preliminary board-level thermal analysis. First, a simulation of the board with some Integrated Circuits (ICs) is performed. Then, the disk-stack heat sink is added above the main hot electronic component to observe cooling effects. The final part adds a copper layer to the bottom of the board in order to obtain a more uniform temperature distribution and see how it affects the heat transfer in the circuit board.

This exercise highlights a number of important modeling techniques such as combining 3D solids and shells and using thin layer boundary conditions when replacing 3D thin geometries by 2D boundaries.

Model Definition

GEOMETRY

Figure 2 shows that the first studied geometry is made of a circuit board with several ICs on it.

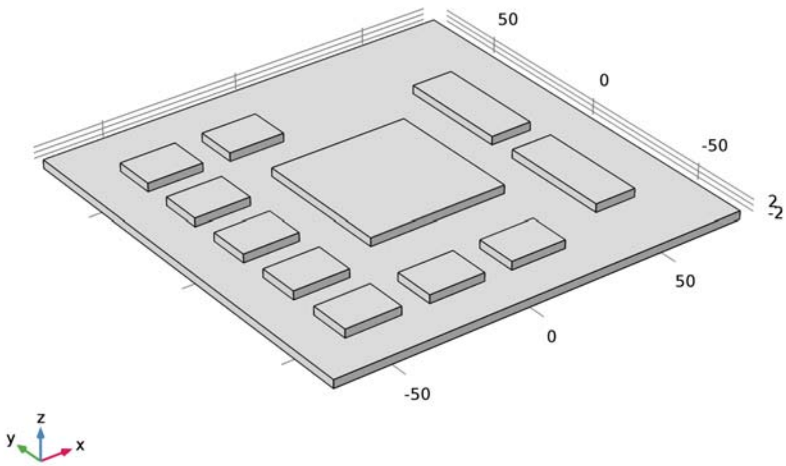


Figure 2: First geometry without heat sink.

This typical board-level thermal analysis determines the temperature profile in and around a high-power chip. The printed-circuit board usually consists of multiple layers of FR4 material (Flame Resistant 4) and copper traces along the board. Hence, the thermal conductivity along the board is much higher than the conductivity through it. It is possible to take several approaches for simulating such a board in COMSOL Multiphysics. This example uses a macro-level approach and assumes a homogeneous PC board with anisotropic thermal material properties. In this case, the heat diffusion through the board and the one lost due to natural convection is insufficient to adequately cool the chip. Hence, a disk-stack heat sink is required to increase the effective cooling area for the chip.

REGARDING MATERIALS

The IC packages and the PC board on which they are mounted must be defined. In reality, these components have very detailed structures and are made of a variety of materials. For a board-level analysis such as the present one, though, it is much simpler to lump all these detailed structures into single homogeneous materials for each component, instead of accounting for the thermal characteristics of a multi-layer PC board, which typically consist of multiple layers of FR4 (insulator) interspersed with layers of copper traces. The thermal result of this construction is that the thermal conductivity along the board is considerably higher than through it. Physical property values depend on the number of layers, how dense the lines are, and how many vias (interconnections between layers) per unit area are present. The numbers in an estimate for a highly layered board which are used to create a strong difference in conductivity between the printed-circuit board plane (x, y) and the orthogonal direction z. Those properties are presented in Table 1. The units are W/(m·K) for thermal conductivity, kg/m^3 for density and J/(kg·K) for heat capacity.

TABLE 1: MATERIALS PROPERTIES

MATERIALS	CONDUCTIVITY	DENSITY	HEAT CAPACITY
Copper	400	8700	385
FR-4	0.3	1900	1369
Aluminum	160	2700	900
IC Packages (Silica Glass)	1.38	2203	703
PC Board (x, y and z directions)	{80, 80, 0.3}	1900	1369

THERMAL CONFIGURATION

In this problem the large central chip dissipates 20 W, the array of smaller chips are 1 W each, and the two elongated chips are 2 W each. The volumetric heat source is calculated by dividing the heat power by the volume of the considered IC.

In this example, you assume that a fan cools the board, and specify a convective heat transfer coefficient for the boundary heat flux. Here, you look for a preliminary sizing calculation and simply assume a convective coefficient, h, of 20 W/(m^2·K). This corresponds to a fan blowing air at approximately 1 m/s on a plate. The air temperature is set to T_0 = 273.15 K during the whole modeling process.

Without a heat sink, the temperature rise in the main chip is higher than the maximum operating temperature. A stacked disk heat sink increases the effective area and therefore cools the chip further. This heat sink consists of a series of thin disks supported by a central hollow column that is mounted to the chip with an aluminum base corresponding to the size of the chip. The heat sink is mounted dry and must therefore account for contact resistance. Figure 3 shows the new geometry with the main chip equipped with the heat sink.

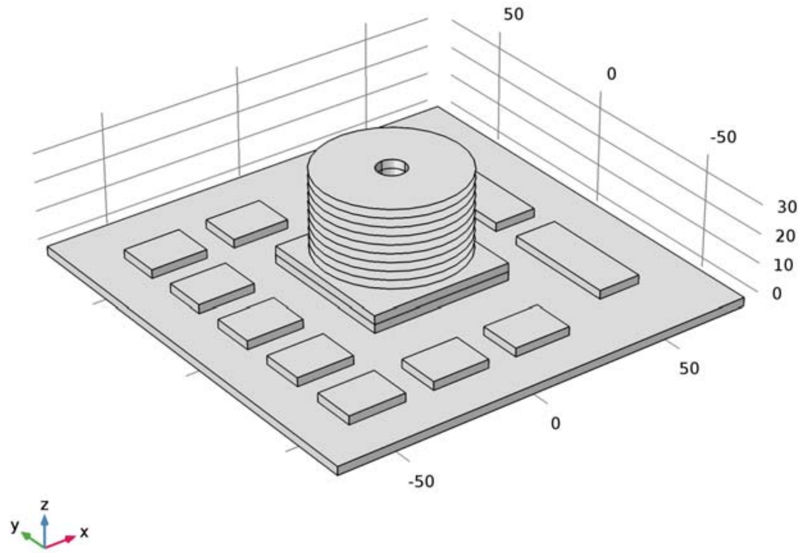


Figure 3: Full geometry of the PC board equipped with the heat sink.

Thermal linkage between the IC and the added heat sink is made using the **Thermal Contact** boundary condition. It provides a heat transfer coefficient at the two surfaces in contact according to (1.9 in Ref. 1):

$$h_{\mathrm{interface}} = h_{\mathrm{constriction}} + h_{\mathrm{gap}} = 1.25 k_{\mathrm{s}} \frac{m}{\sigma}\left(\frac{p}{H_{\mathrm{c}}}\right)^{0.95} + \frac{k_{\mathrm{gap}}}{Y + M_{\mathrm{gap}}}$$

This expression involves two parameters related to the surface microscopic asperities: σ, the average asperities height, and m, the average asperities slope. In this case, σ and m are set to 1 µm and 0.5, respectively. The microhardness of the softer material, H_{c}, is here the hardness of aluminum, equal to 165 MPa. The contact pressure, p, is set to 20 kPa. The thermal conductivity k_{gap} is related to the material in the interstitial gap, here assumed to be air at atmospheric pressure. It is equal to 0.025 W/(m·K).

A design value of 0.3 mm is chosen for the thickness of the fins and the central hollow column.

Finally, the last part explores the possibility of evening out the temperature distribution across the PC board. For instance, add a 0.4 mm layer of copper across the board entire bottom surface. The previous cross section does not suggest much success for this approach. However, it is interesting to check such an analysis for the sake of comparison. In COMSOL Multiphysics, this is easily done using the **Thin Layer** boundary condition.

Results and Discussion

Figure 4 shows the stationary temperature field on the surfaces of the board and chips in Kelvin: the central IC becomes rather hot (337 K) and needs extra cooling.

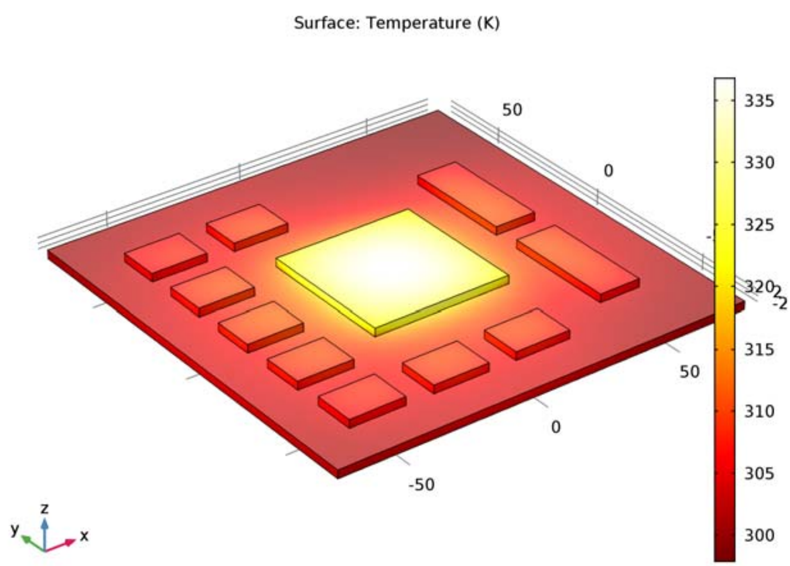


Figure 4: Temperature distribution of the PC board without the heat sink.

As Figure 5 shows, there is a steep thermal gradient between the IC and the heat sink base which is caused by the contact resistance and the significant cooling by the heat sink fins.

The maximum device temperature has now dropped to 313 K, which is 24 K lower than without the heat sink.

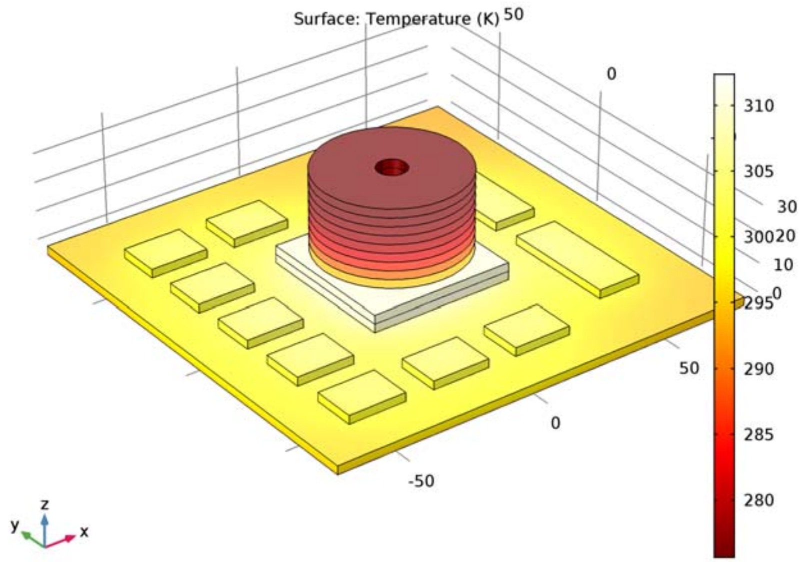


Figure 5: Temperature distribution of the PC board with its heat sink.

Finally, Figure 6 shows that adding a layer of copper at the bottom of the Circuit Board is ineffective. This phenomenon agrees with the fact that the Circuit Board material has a rather poor thermal conductivity along the vertical z-axis (orthogonal to the PC plane).

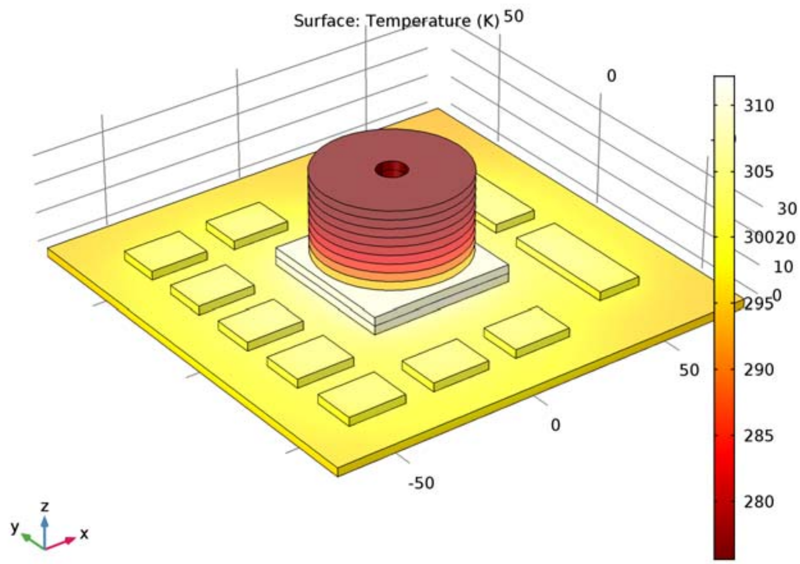


Figure 6: Temperature distribution of the PC board with its heat sink and a layer of copper at bottom

Notes About the COMSOL Implementation

In this application, use the Heat Transfer in Thin Shells interface to model thermal behavior of fins. The number of elements is significantly reduced because, instead of creating a thin 3D geometry, only a 2D layer is meshed.

References

1. A.D. Kraus and A. Bejan, *Heat Transfer Handbook*, John Wiley & Sons, 2003.

Application Library path: `Heat_Transfer_Module/Thermal_Contact_and_Friction/disk_stack_heat_sink`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the application's Application Libraries folder and double-click the file `disk_stack_heat_sink_parameters.txt`.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for Geometry, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for Block, locate the **Size and Shape** section.
3 In the **Width** text field, type `CB_w`.
4 In the **Depth** text field, type `CB_l`.
5 In the **Height** text field, type `CB_t`.
6 Locate the **Position** section. In the **x** text field, type `-CB_w/2`.

7 In the **y** text field, type `-CB_l/2`.

8 In the **z** text field, type `-CB_t`.

Block 2 (blk2)

1 Right-click **Block 1 (blk1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Block**.

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

4 In the **Width** text field, type `IC1_w`.

5 In the **Depth** text field, type `IC1_l`.

6 In the **Height** text field, type `IC1_t`.

7 Locate the **Position** section. In the **x** text field, type `-CB_w/2+IC1_w`.

8 In the **y** text field, type `-CB_l/2+IC1_l`.

Block 3 (blk3)

1 Right-click **Block 2 (blk2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Block**.

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

4 In the **Width** text field, type `IC2_l`.

5 In the **Depth** text field, type `IC2_w`.

6 In the **Height** text field, type `IC2_t`.

7 Locate the **Position** section. In the **x** text field, type `-60`.

8 In the **y** text field, type `-60`.

Copy 1 (copy1)

1 Right-click **Block 3 (blk3)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

3 Select the object **blk3** only.

4 In the **Settings** window for Copy, locate the **Displacement** section.

5 In the **x** text field, type `0 0 0 0 range(30,30,60) 30`.

6 In the **y** text field, type `range(25, 25, 100) 0 0 100`.

Block 4 (blk4)

1 Right-click **Copy 1 (copy1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Block**.

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

4 In the **Width** text field, type `IC3_w`.

5 In the **Depth** text field, type `IC3_l`.

6 In the **Height** text field, type `IC3_t`.

7 Locate the **Position** section. In the **x** text field, type `40`.

8 In the **y** text field, type `-50`.

Copy 2 (copy2)

1 Right-click **Block 4 (blk4)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

3 Select the object **blk4** only.

4 In the **Settings** window for Copy, locate the **Displacement** section.

5 In the **y** text field, type `50`.

6 Click **Build All Objects**.

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `ICs, Type 1` in the **Label** text field.

3 Select Domain 9 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `ICs, Type 2` in the **Label** text field.

3 Select Domains 2–8 and 10 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `ICs, Type 3` in the **Label** text field.

3 Select Domains 11 and 12 only.

Union 1

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for Union, type `ICs` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, select **ICs, Type 1** in the **Selections to add** list.

5 Click **OK**.
6 In the **Settings** window for Union, locate the **Input Entities** section.
7 Under **Selections to add**, click **Add**.
8 In the **Add** dialog box, select **ICs, Type 2** in the **Selections to add** list.
9 Click **OK**.
10 In the **Settings** window for Union, locate the **Input Entities** section.
11 Under **Selections to add**, click **Add**.
12 In the **Add** dialog box, select **ICs, Type 3** in the **Selections to add** list.
13 Click **OK**.

ADD MATERIAL

1 On 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>Silica glass**.
4 Click **Add to Component** in the window toolbar.

MATERIALS

Silica glass (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silica glass (mat1)**.
2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **ICs**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-In>FR4 (Circuit Board)**.
3 Click **Add to Component** in the window toolbar.

MATERIALS

FR4 (Circuit Board) (mat2)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.
2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **FR4 (Circuit Board) (mat2)**.

3 Select Domain 1 only.

Here, the PC board needs to have an orthotropic thermal conductivity to account for conduction induced by several copper tracks in the *xy*-planes of the board.

4 In the **Settings** window for Material, locate the **Material Contents** section.

5 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	{80,80, 0.3}	W/(m·K)	Basic

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, type T0 in the T text field.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for Heat Source, locate the **Domain Selection** section.

3 From the **Selection** list, choose **ICs, Type 1**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type P1.

Heat Source 2

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for Heat Source, locate the **Domain Selection** section.

3 From the **Selection** list, choose **ICs, Type 2**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type P2*8.

Heat Source 3

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for Heat Source, locate the **Domain Selection** section.

3 From the **Selection** list, choose **ICs, Type 3**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `P3*2`.

Heat Flux 1

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

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

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

4 In the h text field, type `htc`.

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

In the followings, select boundaries 1 to 72. For more convenience, use the **Paste Selection** button.

6 Locate the **Boundary Selection** section. Click **Paste Selection**.

7 In the **Paste Selection** dialog box, type `1-72` in the **Selection** text field.

8 Click **OK**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

This is the first temperature distribution. It clearly outlines that the main chip needs more efficient cooling. This is the aim of the next part in which a disk-stack heat sink will be added on the top of the central chip.

GEOMETRY 1

Block 5 (blk5)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `IC1_w`.

4 In the **Depth** text field, type `IC1_l`.

5 In the **Height** text field, type `IC1_t`.

6 Locate the **Position** section. In the **x** text field, type `-CB_w/2+IC1_w`.

7 In the **y** text field, type `-CB_l/2+IC1_l`.

8 In the **z** text field, type `IC1_t`.

Cylinder 1 (cyl1)

1 Right-click **Block 5 (blk5)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Cylinder**.

3 In the **Settings** window for Cylinder, locate the **Object Type** section.

4 From the **Type** list, choose **Surface**.

5 Locate the **Size and Shape** section. In the **Radius** text field, type `i_radius`.

6 In the **Height** text field, type `t_h`.

7 Locate the **Position** section. In the **z** text field, type `IC1_t*2`.

8 Locate the **Selections of Resulting Entities** section. Click **New**.

9 In the **New Cumulative Selection** dialog box, type `Fins` in the **Name** text field.

10 Click **OK**.

Work Plane 1 (wp1)

1 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Work Plane**.

3 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

4 From the **Plane type** list, choose **Face parallel**.

5 On the object **blk5**, select Boundary 4 only.

6 In the **Offset in normal direction** text field, type `air_sp`.

7 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for Circle, locate the **Size and Shape** section.

3 In the **Radius** text field, type `i_radius`.

4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

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

Circle 2 (c2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for Circle, locate the **Size and Shape** section.

3 In the **Radius** text field, type `o_radius`.

4 Right-click **Circle 2 (c2)** and choose **Build Selected**.

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

Difference 1 (dif1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c2** only.

3 In the **Settings** window for Difference, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **c1** only.

Work Plane 1 (wp1)

1 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

3 In the **Settings** window for Work Plane, locate the **Selections of Resulting Entities** section.

4 From the **Contribute to** list, choose **Fins**.

5 In the **Model Builder** window, expand the **Work Plane 1 (wp1)** node.

Array 1 (arr1)

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

3 Select the object **wp1** only.

4 In the **Settings** window for Array, locate the **Size** section.

5 From the **Array type** list, choose **Linear**.

6 In the **Size** text field, type `n_fins`.

7 Locate the **Displacement** section. In the **z** text field, type `air_sp`.

Work Plane 1 (wp1)

1 In the **Settings** window for Array, locate the **Selections of Resulting Entities** section.

2 From the **Contribute to** list, choose **Fins**.

3 Click **Build All Objects**.

ADD MATERIAL

1 On 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>Aluminum**.

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

MATERIALS

Aluminum (mat3)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** node, then click **Component 1 (comp1)>Materials>Aluminum (mat3)**.
2 Select Domain 10 only.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-In>Aluminum**.
3 Click **Add to Component** in the window toolbar.

MATERIALS

Aluminum 1 (mat4)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.
2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum 1 (mat4)**.
3 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
4 From the **Geometric entity level** list, choose **Boundary**.
5 From the **Selection** list, choose **Fins**.

   It is necessary to add a second **Aluminum** material since the first one is used on a different geometry entity level.

HEAT TRANSFER IN SOLIDS (HT)

Heat Flux 1

Add the newly created external boundaries of the heat sink base.

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Heat Flux 1**.
2 In the **Settings** window for Heat Flux, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `49 50 52 64 115` in the **Selection** text field.
5 Click **OK**.

Thermal Contact 1

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

2 In the **Settings** window for Thermal Contact, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `51` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for Thermal Contact, locate the **Thermal Contact** section.

7 From the h_g list, choose **Parallel-plate gap gas conductance**.

8 Locate the **Contact Surface Properties** section. In the p text field, type `20[kPa]`.

9 In the H_c text field, type `165[MPa]`.

10 Click to expand the **Gap properties** section. Locate the **Gap Properties** section. From the k_{gap} list, choose **User defined**.

ADD PHYSICS

1 On the **Physics** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Heat Transfer>Thin Structures>Heat Transfer in Thin Shells (htsh)**.

4 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Temperature** text field, type `T`.

A new physics interface is required here to take into account out-of-plane convective cooling. In the physics interface selection, you have to use the same temperature variable, `T`, to couple the two physics interfaces.

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

HEAT TRANSFER IN THIN SHELLS (HTSH)

1 On the **Physics** toolbar, click **Add Physics** to close the **Add Physics** window.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Thin Shells (htsh)**.

3 In the **Settings** window for Heat Transfer in Thin Shells, locate the **Boundary Selection** section.

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

5 Locate the **Shell Thickness** section. In the d_s text field, type `e_fins`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Thin Shells (htsh)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `T0` in the T text field.

Heat Flux 1

1 On 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 **Fins**.
4 Locate the **Upside Inward Heat Flux** section. Click the **Convective heat flux** button.
5 In the h_{u} text field, type `htc`.
6 In the $T_{\mathrm{ext,\ u}}$ text field, type `T0`.
7 Locate the **Downside Inward Heat Flux** section. Click the **Convective heat flux** button.
8 In the h_{d} text field, type `htc`.
9 In the $T_{\mathrm{ext,\ d}}$ text field, type `T0`.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

This is the temperature profile once the heat sink has been added. The heat sink significantly reduces the average temperature of the main chip.

ADD MATERIAL

1 On 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>Copper**.
4 Click **Add to Component** in the window toolbar.

MATERIALS

Copper (mat5)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.
2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat5)**.
3 Select Domain 1 only.
4 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
5 From the **Geometric entity level** list, choose **Boundary**.
6 Select Boundary 3 only.

HEAT TRANSFER IN SOLIDS (HT)

On the **Physics** toolbar, click **Heat Transfer in Thin Shells (htsh)** and choose **Heat Transfer in Solids (ht)**.

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.
2 Select Boundary 3 only.
3 In the **Settings** window for Thin Layer, locate the **Thin Layer** section.
4 From the **Layer type** list, choose **Thermally thin approximation**.
5 In the d_s text field, type `0.4[mm]`.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

This is the temperature profile once the copper layer has been added. No significant effect due to this modification can be observed.

Derived Values

Finally, check the maximum temperature over the component.

Volume Maximum 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Maximum>Volume Maximum**.
2 In the **Settings** window for Volume Maximum, type `Maximum Temperature` in the **Label** text field.
3 Locate the **Selection** section. From the **Selection** list, choose **All domains**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
T	K	Maximum temperature

5 Click **Evaluate**.

Created in COMSOL Multiphysics 5.2a

Forced Convection Cooling of an Enclosure with Fan and Grille

Introduction

This study simulates the thermal behavior of a computer power supply unit (PSU). Such electronic enclosures typically include cooling devices to avoid electronic components being damaged by excessively high temperatures. In this application, an extracting fan and a perforated grille generate an air flow in the enclosure to cool internal heating.

Air extracted from the enclosure is related to the static pressure (the pressure difference between outside and inside), information that is generally provided by the fan manufacturers as a curve representing fluid velocity as a function of pressure difference.

As shown in Figure 1, the geometry is rather complicated and requires a fine mesh to solve. This results in large computational costs in terms of time and memory.

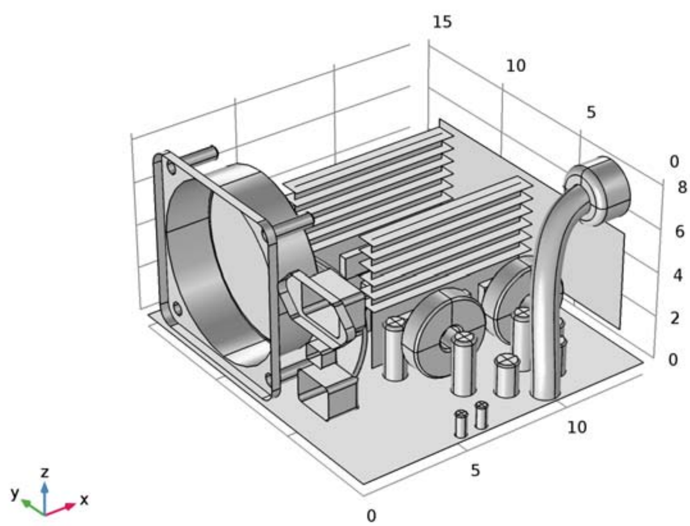


Figure 1: The complete model geometry.

Model Definition

Figure 1 shows the geometry of the PSU. It is composed of a perforated enclosure of 14 cm-by-15 cm-by-8.6 cm and is made of aluminum 6063-T83. Inside the enclosure, only obstacles having a characteristic length of at least 5 mm are represented.

The bottom of the box represents the printed circuit board (PCB). It has an anisotropic thermal conductivity of 10, 10, and 0.36 W/(m·K) along the x-, y-, and z-axes, respectively. Its density is 430 kg/m^3 and its heat capacity at constant pressure is 1100 J/(kg·K). Because the thermal conductivity along the z-axis is relatively low, and that the PCB and the enclosure sides are separated by a thin air layer, it is not necessary to model the bottom wall, nor take into account the cooling on these sides.

The capacitors are approximated by aluminum components. The heat sink fins and the enclosure are made of the same aluminum alloy. The inductors are mainly composed of steel cores and copper coils. The transformers are made of three materials: copper, steel, and plastic. The transistors are modeled as two-domain components: a core made of silicon held in a plastic case. The core is in contact with an aluminum heat sink to allow a more efficient heat transfer. Air speed is assumed to be slow enough for the flow to be considered as laminar.

The simulated PSU consumes a maximum of 230 W. Components have been grouped and assigned to various heat sources as listed in Table 1. The overall heat loss is 41 W, which is about 82% of efficiency.

TABLE 1: HEAT SOURCES OF ELECTRONIC COMPONENTS

Components	**Dissipated heat rate (W)**
Transistor cores	25
Large transformer coil	5
Small transformer coils	3
Inductors	2
Large capacitors	2
Medium capacitors	3
Small capacitors	1

The inlet air temperature is set at 30 °C because it is supposed to come from the computer case in which air has already cooled other components. The inlet boundary is configured with a **Grille** boundary condition. This pressure must describe head loss caused by air entry into the enclosure. The head loss coefficient k_{grille} is represented by the following 6$^{\text{th}}$ order polynomial (Ref. 1):

$$\begin{aligned} k_{\text{grille}} = {} & 12084\alpha^6 - 42281\alpha^5 + 60989\alpha^4 - 46559\alpha^3 \\ & + 19963\alpha^2 - 4618.5\alpha + 462.89 \end{aligned} \tag{1}$$

where α is the opening ratio of the grille.

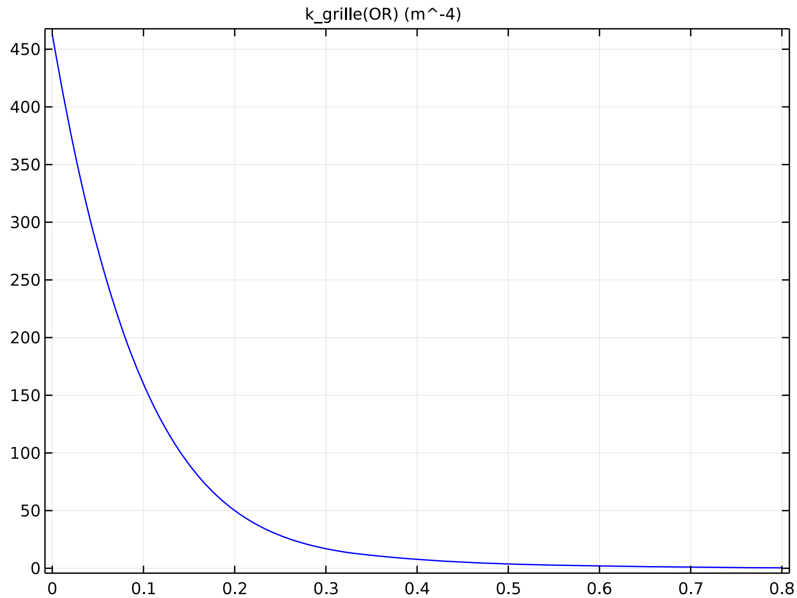


Figure 2: Head loss coefficient as a function of the opening ratio.

The head loss, ΔP (Pa), is given by

$$\Delta P = k_{\text{grille}}\frac{\rho U^2}{2}$$

where ρ is the density (kg/m^3), and U is the velocity magnitude (m/s).

The box, the PCB, the inductor surfaces, and the heat sink fins are configured as thin conductive layers.

Results and Discussion

The most interesting aspect of this simulation is to locate which components are subject to overheating. Figure 3 clearly shows that the temperature distribution is not homogeneous.

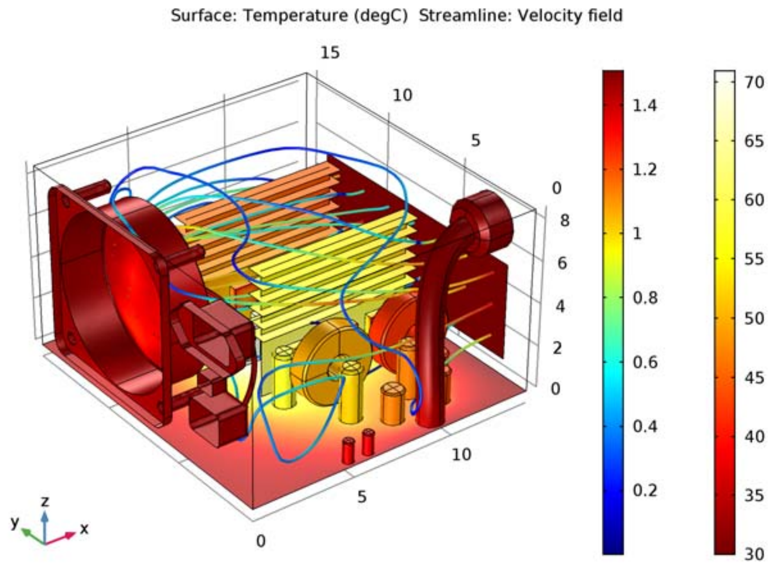


Figure 3: Temperature and fluid velocity fields.

The maximum temperature is about 70 °C and is located at one of the transistor cores. The components furthest away from the air inlet are subject to the highest temperature. Although transistor cores are rather hot, Figure 3 shows that they are significantly cooled by the aluminum heat sinks. The printed circuit board has a significant impact as well by distributing and draining heat off.

On the flow side, air avoids obstacles and tends to go through the upper space of the enclosure. The maximum velocity is about 1.6 m/s.

Figure 4 shows the head loss created by obstacles encountered by air on its path.

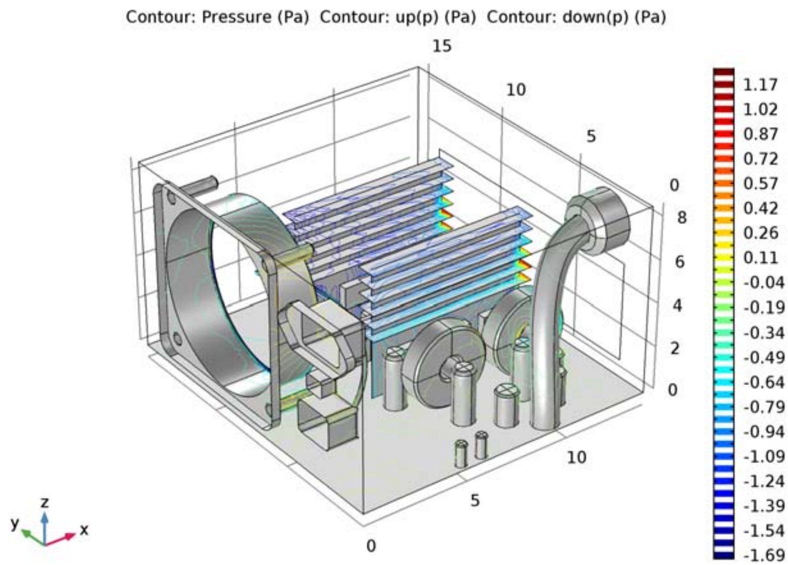


Figure 4: Pressure isosurfaces show the head loss created by electronic components.

As shown in Figure 4, the fluid flow is impacted by obstacles and yields to local head losses.

Notes About the COMSOL Implementation

To model heat sink fins, enclosure walls and circuit board it is strongly recommended to use the **Thin Layer** feature, which is completely adapted for thin geometries and significantly reduces the number of degrees of freedom in the model.

COMSOL Multiphysics provides a useful boundary condition for modeling fan behavior. You just need to provide a few points of a static pressure curve to configure this boundary condition. These data are, most of the time, provided by fan manufacturers.

Reference

1. R.D. Blevins, *Applied Fluid Dynamics Handbook*, Van Nostrand Reinhold, 1984.

Application Library path: `Heat_Transfer_Module/`
`Power_Electronics_and_Electronic_Cooling/electronic_enclosure_cooling`

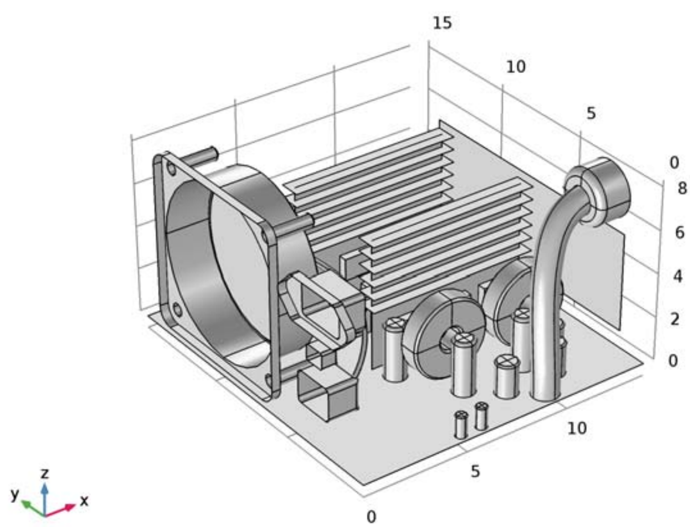
Modeling Instructions

ROOT

The file `electronic_enclosure_cooling_geom.mph` contains a parameterized geometry and prepared selections for the model. Start by loading this file.

1 From the **File** menu, choose **Open**.

2 Browse to the application's Application Libraries folder and double-click the file `electronic_enclosure_cooling_geom.mph`.



GLOBAL DEFINITIONS

Define an analytic function to represent the polynomial expression of the head loss coefficient (Equation 1). This coefficient is a function of the open ratio of the grille.

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for Analytic, type `k_grille` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `12084*OR^6-42281*OR^5+60989*OR^4-46559*OR^3+19963*OR^2-4618.5*OR+462.89`.

4 In the **Arguments** text field, type `OR`.

5 Locate the **Units** section. In the **Arguments** text field, type `1`.

6 In the **Function** text field, type `m^-4`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
OR	0	0.8

8 Click **Plot**.

MATERIALS

In this section, you define the materials of the enclosure and its components. The prepared selections make it easier to select the appropriate domains and boundaries.

ADD MATERIAL

1 On 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>Air**.

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

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

3 From the **Selection** list, choose **Air**.

ADD MATERIAL

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

2 In the tree, select **Built-In>Acrylic plastic**.

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

MATERIALS

Acrylic plastic (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Acrylic plastic (mat2)**.

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

3 From the **Selection** list, choose **Plastic**.

ADD MATERIAL

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

2 In the tree, select **Built-In>Aluminum 6063-T83**.

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

MATERIALS

Aluminum 6063-T83 (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum 6063-T83 (mat3)**.

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

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

4 From the **Selection** list, choose **Aluminum Boundaries**.

ADD MATERIAL

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

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

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

MATERIALS

Steel AISI 4340 (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat4)**.

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

3 From the **Selection** list, choose **Steel Parts**.

ADD MATERIAL

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

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

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

MATERIALS

Aluminum (mat5)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat5)**.

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

3 From the **Selection** list, choose **Capacitors**.

ADD MATERIAL

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

2 In the tree, select **Built-In>Copper**.

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

MATERIALS

Copper (mat6)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat6)**.

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

3 From the **Selection** list, choose **Transformer Coils**.

ADD MATERIAL

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

2 In the tree, select **Built-In>Copper**.

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

MATERIALS

Copper 1 (mat7)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper 1 (mat7)**.

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

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

4 From the **Selection** list, choose **Copper Layers**.

ADD MATERIAL

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

2 In the tree, select **Built-In>FR4 (Circuit Board)**.

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

MATERIALS

FR4 (Circuit Board) (mat8)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **FR4 (Circuit Board) (mat8)**.

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

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

4 From the **Selection** list, choose **Circuit Board**.

Modify the thermal conductivity of the FR4 material to model the anisotropic properties of the printed circuit board.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	{10, 10, 0.3}	W/(m·K)	Basic

ADD MATERIAL

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

2 In the tree, select **Built-In>Silicon**.

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

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Silicon (mat9)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silicon (mat9)**.

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

3 From the **Selection** list, choose **Transistors Silicon Cores**.

The next steps define the boundary conditions of the model.

HEAT TRANSFER (HT)

Set the ambient temperature to be used in intial values of the Heat Transfer interface.

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Heat Transfer (ht)** node, then click **Heat Transfer (ht)**.
2. In the **Settings** window for Heat Transfer, locate the **Ambient Settings** section.
3. In the T_{amb} text field, type `30[degC]`.

Fluid 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Fluid 1**.
2. In the **Settings** window for Fluid, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Air**.

Initial Values 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Initial Values 1**.
2. In the **Settings** window for Initial Values, choose **Ambient temperature (ht)** from the T list.

Now, define the heat rate produced by the different components. Table 1 summarizes the values chosen for each kind of component.

Heat Source 1

1. On the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2. In the **Settings** window for Heat Source, type `Heat Source 1: Transistors` in the **Label** text field.
3. Locate the **Domain Selection** section. From the **Selection** list, choose **Transistors Silicon Cores**.
4. Locate the **Heat Source** section. Click the **Heat rate** button.
5. In the P_0 text field, type `25`.

Heat Source 2

1. On the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2. In the **Settings** window for Heat Source, type `Heat Source 2: Large Transformer Coil` in the **Label** text field.
3. Locate the **Domain Selection** section. From the **Selection** list, choose **Large Transformer Coil**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `5`.

Heat Source 3

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for Heat Source, type `Heat Source 3: Small Transformer Coils` in the **Label** text field.

3 Locate the **Heat Source** section. Click the **Heat rate** button.

4 In the P_0 text field, type `3`.

5 Locate the **Domain Selection** section. From the **Selection** list, choose **Small Transformer Coils**.

Heat Source 4

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for Heat Source, type `Heat Source 4: Inductor` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Inductors**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `2`.

Heat Source 5

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for Heat Source, type `Heat Source 5: Large Capacitors` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Large Capacitors**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `2`.

Heat Source 6

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for Heat Source, type `Heat Source 6: Medium Capacitors` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Medium Capacitors**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `3`.

Heat Source 7

1. On the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2. In the **Settings** window for Heat Source, type `Heat Source 7: Small Capacitors` in the **Label** text field.
3. Locate the **Domain Selection** section. From the **Selection** list, choose **Small Capacitors**.
4. Locate the **Heat Source** section. Click the **Heat rate** button.
5. In the P_0 text field, type `1`.

Temperature 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2. In the **Settings** window for Temperature, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Grille**.
4. Locate the **Temperature** section. From the T_0 list, choose **Ambient temperature (ht)**.

To model heat transfer in thin conductive parts of the enclosure, use the **Thin Layer** boundary condition on thin domains made of aluminum, copper, and FR4.

Thin Layer 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.
2. In the **Settings** window for Thin Layer, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Conductive Layers**.
4. Locate the **Thin Layer** section. From the **Layer type** list, choose **Thermally thin approximation**.
5. In the d_s text field, type `2[mm]`.

Outflow 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2. In the **Settings** window for Outflow, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Fan**.

TURBULENT FLOW, ALGEBRAIC YPLUS (SPF)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, Algebraic yPlus (spf)**.
2. In the **Settings** window for Turbulent Flow, Algebraic yPlus, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Air**.

The thin heat sink fins are represented by interior boundaries in the geometry. An interior wall condition is used to prevent the fluid from flowing through these boundaries.

Interior Wall 1

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

2 In the **Settings** window for Interior Wall, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Fins**.

Fan 1

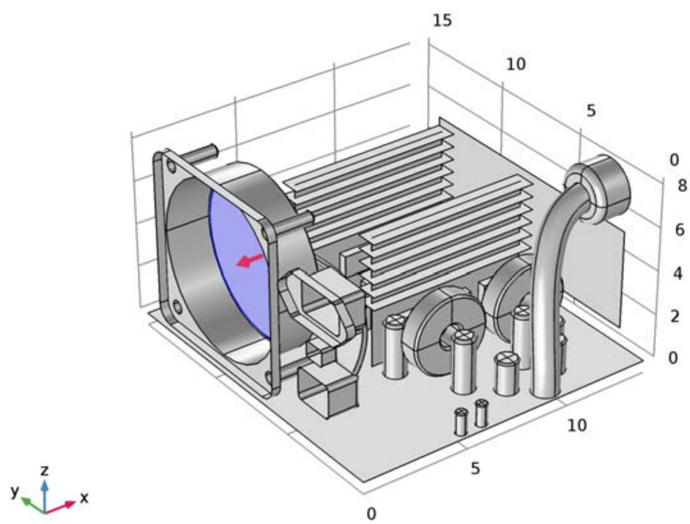
1 On the **Physics** toolbar, click **Boundaries** and choose **Fan**.

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

3 From the **Selection** list, choose **Fan**.

4 Locate the **Flow Direction** section. From the **Flow direction** list, choose **Outlet**.

The **Graphics** window displays an arrow indicating the orientation of the flow through the fan. Compare with the figure below.



Here, the **Fan** condition is set up by loading a data file for the static pressure curve.

5 Locate the **Parameters** section. From the **Static pressure curve** list, choose **Static pressure curve data**.

6 Locate the **Static Pressure Curve Data** section. Click **Load from File**.

7 Browse to the application's Application Libraries folder and double-click the file `electronic_enclosure_cooling_fan_curve.txt`.

8 Locate the **Static Pressure Curve Interpolation** section. From the **Interpolation function type** list, choose **Piecewise cubic**.

The exhaust fan previously defined extracts air entering from an opposite grille. Proceed to create the corresponding boundary condition.

Grille 1

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

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

3 From the **Selection** list, choose **Grille**.

4 Locate the **Parameters** section. From the **Static pressure curve** list, choose **Quadratic loss**.

5 In the *qlc* text field, type `k_grille(OR)*nitf1.rho/2`.

Initial Values 1

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

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

3 Specify the **u** vector as

1	x

MESH 1

You now configure the meshing part. Start by discretizing the surfaces of key components. They would drive the tetrahedral mesh of the whole domain. Boundary layers at walls are added at the end.

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

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

3 From the **Selection** list, choose **Wire Group Surface**.

4 Click to expand the **Advanced settings** section. Locate the **Advanced Settings** section. Select the **Adjust evenly distributed edge mesh** check box.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Size**.

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

3 From the **Predefined** list, choose **Finer**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.5`.

7 Select the **Minimum element size** check box.

8 In the associated text field, type `0.4`.

9 Click **Build Selected**.

Mapped 2

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

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

3 From the **Selection** list, choose **Small Wire Surface**.

4 Locate the **Advanced Settings** section. Select the **Adjust evenly distributed edge mesh** check box.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 2** and choose **Size**.

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

3 From the **Calibrate for** list, choose **Fluid dynamics**.

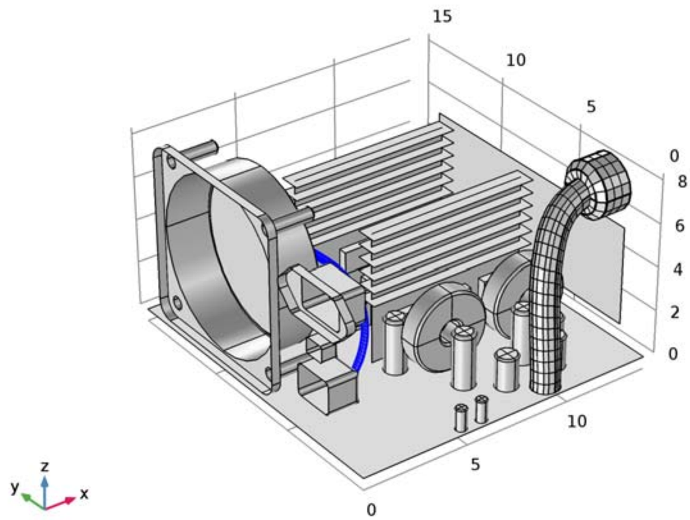
4 From the **Predefined** list, choose **Extra fine**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `0.16`.

8 Click **Build Selected**.



Convert 1

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

2 In the **Settings** window for Convert, click **Build Selected**.

This conversion divides the quadrilateral mesh obtained into triangular elements. This is necessary to make it compatible with the tetrahedral elements created a few steps later.

Free Triangular 1

1 Right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Settings** window for Free Triangular, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Heat Exchange Surface**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **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. Select the **Maximum element size** check box.

5 In the associated text field, type `0.5`.

6 Select the **Minimum element size** check box.

7 In the associated text field, type `0.4`.

8 Select the **Maximum element growth rate** check box.

9 In the associated text field, type `1.05`.

10 Select the **Curvature factor** check box.

11 In the associated text field, type `1`.

12 Select the **Resolution of narrow regions** check box.

13 In the associated text field, type `1`.

14 Click **Build Selected**.

Free Triangular 2

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

2 In the **Settings** window for Free Triangular, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Curved Area**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 2** and choose **Size**.

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

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 Click **Build Selected**.

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

Size

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

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

3 From the **Predefined** list, choose **Coarse**.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for Boundary Layers, locate the **Domain Selection** section.

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

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

5 Click to expand the **Corner settings** section. Locate the **Corner Settings** section. From the **Handling of sharp edges** list, choose **Trimming**.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

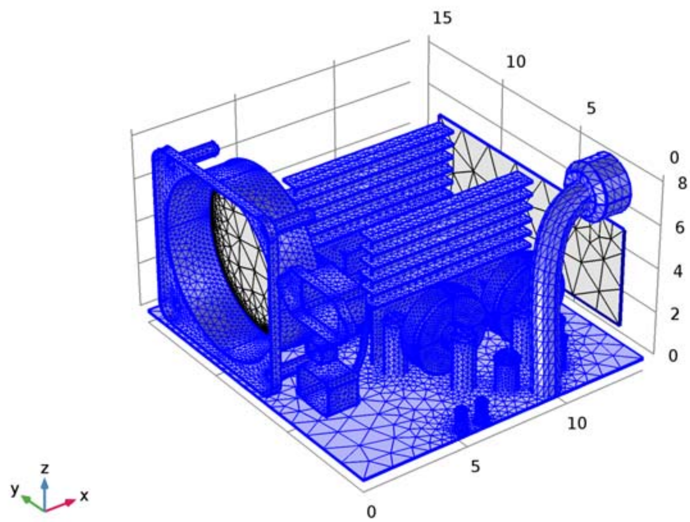
2 In the **Settings** window for Boundary Layer Properties, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type 3.

5 In the **Thickness adjustment factor** text field, type 5.

6 Click **Build All**.



STUDY 1

Now, edit the solver sequence. The default suggestion is a segregated solver. For more robustness, use direct solvers for the temperature group and for the velocity-pressure group.

Solution 1 (sol1)

1 On 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)>Stationary Solver 2>Segregated 1** node, then click **Temperature T**.

4 In the **Settings** window for Segregated Step, locate the **General** section.

5 From the **Linear solver** list, choose **Direct**.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2>Segregated 1** click **Turbulence variables**.

7 In the **Settings** window for Segregated Step, locate the **General** section.

8 From the **Linear solver** list, choose **Direct**.

9 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2>Iterative 1** node, then click **Multigrid 1**.

10 In the **Settings** window for Multigrid, locate the **General** section.

11 In the **Mesh coarsening factor** text field, type `1.5`.

You are now ready to launch the simulation. The computation may take a few hours to complete.

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

RESULTS

Temperature (ht)

The first default plot shows the temperature field. To reproduce the plot in Figure 3 of the temperature and the air velocity, proceed as follows.

Surface

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

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

3 From the **Unit** list, choose **degC**.

Temperature (ht)

In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Streamline**.

Streamline 1

1 In the **Settings** window for Streamline, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Turbulent Flow, Algebraic yPlus>Velocity and pressure>u,v,w - Velocity field**.

2 Locate the **Selection** section. From the **Selection** list, choose **Grille**.

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

Color Expression 1

1 Right-click **Results>Temperature (ht)>Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Turbulent Flow, Algebraic yPlus>Velocity and pressure>spf.U - Velocity magnitude**.

3 On the **Temperature (ht)** toolbar, click **Plot**.

Velocity (spf)

The third default plot group shows the air velocity profile in a slice plot.

Pressure (spf)

This last default plot group shows the pressure field plot as in Figure 4.

Created in COMSOL Multiphysics 5.2a

Evaporation in Porous Media with Large Evaporation Rates

Introduction

Evaporation in porous media is an important process in food and paper industry among others. Many physical effects must be considered: fluid flow, heat transfer and transport of participating fluids and gases. All of these effects are strongly coupled and predefined interfaces can be used to model these effects with COMSOL Multiphysics. Considering an unsaturated porous medium requires adjustments of the predefined equations.

In this model, the changing water saturation inside the porous medium is computed. In contrast to the situation with a stationary liquid phase (see Evaporation in Porous Media with Small Evaporation Rates), where the interfaces provided by COMSOL can be used as they are, the implementation of a multiphase flow requires some changes.

This model and the instructions are based on the corresponding model and focus on the additional steps required to implement multiphase flow in porous media together with evaporation from liquid to gaseous phase.

Model Definition

This model describes a laminar dry air flow through a porous medium containing water vapor and liquid water. The geometry and principle is shown in Figure 1.

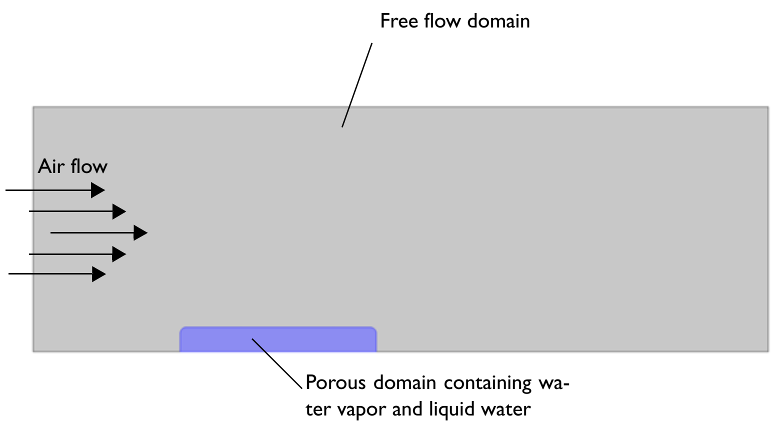


Figure 1: Geometry and principle of the model.

TWO PHASE FLOW IN POROUS MEDIA

The basic principle of modeling two phase flow in porous media is similar for many applications. First, to account for different phases, saturation variables are used that fulfill the following constraint:

$$S_g + S_l = 1 \qquad (1)$$

where the index g is used for the gaseous phase (which in this model is moist air) and the index l is used for the liquid phase (which in this model is water).

Single-phase flow in porous media is described by the Brinkman Equations. With an additional liquid phase, capillary effects also arise and the liquid flow is driven by a pressure gradient and capillary pressure $p_c = p_g - p_l$. How to deal with the latter one depends on the application: sometimes the capillary pressure can be neglected, sometimes it is the driving effect and different approaches exist.

The formulation used in this model follows (Ref. 2), where capillary effects are treated by an additional diffusion term in the transport equation. The Brinkman Equation is used to calculate the flow field $\mathbf{u}_g$ and pressure distribution p_g of moist air in the porous medium. Therefor the porosity ε must take into account that only a fraction of the void space is occupied by the gas phase.

The liquid phase velocity is small compared to the moist air velocity and such Darcy's Law is defined in terms of the gas phase pressure gradient to calculate the water velocity $\mathbf{u}_l$ according to

$$\mathbf{u}_l = -\frac{\kappa_l}{\mu_l}\nabla p_g \tag{2}$$

where κ_l and μ_l are the permeability and viscosity of the liquid phase. It is not necessary to define a second Darcy's law equation, but an additional transport equation for the liquid phase is required.

The boundary conditions for the flow equations and their coupling is the same as in the corresponding model (see Flow properties).

LIQUID PHASE TRANSPORT IN POROUS MEDIA

COMSOL provides the Transport of Diluted Species interface, which solves for a concentration in the very general form:

$$\frac{\partial c}{\partial t} + \nabla \cdot \mathbf{N} = R \tag{3}$$

This interface is used to describe the transport of the liquid phase inside the porous domain, following the ideas about mechanistic formulations from (Ref. 1). In this paper and the references given there the transport of water vapor, liquid water and dry air by convection and diffusion is expressed by flux variables.

It is assumed that no liquid water can leave the porous domain. From now on the index w is used to indicate that the properties are water properties and not a general liquid phase anymore.

The concentration c_w describes the water concentration and is rather an auxiliary variable, since the saturation is crucial for this process. The correlation for water concentration c_w and liquid phase saturation S_l is:

$$S_l = \frac{c_w M_w}{\rho_w \varepsilon}$$

where M_w and ρ_w are the molar mass and density of water, ε is the porosity.

The velocity field in Equation 2 must take into account that the pore space is not fully saturated with water. Additionally the permeability for the liquid phase κ_l depends on the overall permeability of the porous matrix κ and a relative permeability (see Permeability) κ_{rl}, thus:

$$\mathbf{u}_l = -\frac{\kappa\kappa_{rl}}{S_l\varepsilon\mu_w}\nabla p_g$$

The pressure gradient ∇p_g is solved with the Brinkman Equation and μ_w is the viscosity of water. The capillary effect is introduced as the diffusion coefficient D_{cap} which depends on the moisture content (Ref. 2):

VAPOR TRANSPORT IN POROUS MEDIA

The procedure to derive the transport equation for water vapor is similar to the previous section. Starting from the conservation equation (Equation 3) and following the description in (Ref. 1), one has to implement the flux due to convection whereas the velocity field is already known from the Brinkman equation and needs to be applied to the vapor phase. The second transport mechanism is flux due to binary diffusion of water vapor and dry air in the gaseous phase. A common correlation for an effective diffusivity D_{eff} for two components is the Millington and Quirk equation

$$D_{eff} = D_{va}\varepsilon^{4/3}S_g^{10/3}$$

with the vapor-air diffusivity $D_{va} = 2{,}6 \cdot 10^{-5}\frac{\mathrm{m}^2}{\mathrm{s}}$.

Both effects provide the velocity field that is applied to the water vapor transport equation:

$$\mathbf{u} = \frac{\mathbf{u}_g}{S_g\varepsilon} - \frac{M_a D_{eff}}{M_{ma}\rho_{ma}}\nabla\rho_{ma}$$

M_{ma} and ρ_{ma} refer to the moist air molar mass and density.

EVAPORATION

To calculate the amount of water that evaporates into air and to account for the reducing liquid and increasing moist air proportion, the same correlation as in the Evaporation section from Ref. 3 is used:

$$m_{evap} = K(a_w c_{sat} - c)$$

where K (1/s) is the evaporation rate, c_{sat} the vapor concentration under saturation conditions and c the current vapor concentration.

HEAT TRANSFER

The free flow domain contains moist air only and the velocity field from the laminar flow equation is used to describe the heat transferred by convection.

Inside the porous domain the overall velocity field for liquid and gaseous phase contributes to the heat convection term. Averaged thermal properties are required:

$$\begin{aligned} \rho_{\mathrm{tot}} &= S_{\mathrm{g}}\rho_{\mathrm{ma}} + S_{\mathrm{l}}\rho_{\mathrm{w}} \\ C_{\mathrm{p,tot}} &= \frac{S_{\mathrm{g}}\rho_{\mathrm{ma}}C_{\mathrm{p,ma}} + S_{\mathrm{l}}\rho_{\mathrm{w}}C_{\mathrm{p,w}}}{\rho_{\mathrm{tot}}} \\ k_{\mathrm{tot}} &= S_{\mathrm{g}}k_{\mathrm{ma}} + S_{\mathrm{l}}k_{\mathrm{w}} \end{aligned} \tag{4}$$

Then the overall velocity can be expressed as the average of dry air-, water vapor-, and liquid water velocity, which is:

$$\mathbf{u}_{\mathrm{mean}} = \frac{\mathbf{n}_{\mathrm{a}}C_{\mathrm{p,a}} + \mathbf{n}_{v}C_{\mathrm{p,v}} + \mathbf{n}_{\mathrm{w}}C_{\mathrm{p,w}}}{\rho_{\mathrm{tot}}C_{\mathrm{p,tot}}} \tag{5}$$

$\mathbf{n}_{\mathrm{a}}$, $\mathbf{n}_{\mathrm{v}}$, and $\mathbf{n}_{\mathrm{w}}$ are the flux variables for each component (see Ref. 1 for more details). The heat of evaporation is inserted as a source term in the heat transfer equation according to:

$$Q = -H_{\mathrm{vap}} \cdot m_{\mathrm{vap}} \tag{6}$$

where H_{vap} (J/mol) is the latent heat of evaporation.

MOIST AIR PROPERTIES

The properties of moist air are the same as used in the other model (see Thermodynamic properties). Inside the porous domain there is the option to define a single fluid type. Implementing a two phase flow means that the fluid type has to be defined in a way that it accounts for liquid and gaseous phases. Therefore it is necessary to define the moist air properties manually. Also all other material properties are defined manually for sake of simplicity.

PERMEABILITY

The permeability of the porous matrix κ defines the absolute permeability. When two phases are present, the permeability of each phase depends also on the saturation. This is defined by the relative permeabilities κ_{rl} and κ_{rg} for liquid and gaseous phase respectively, so that $\kappa_{\mathrm{l}} = \kappa\kappa_{\mathrm{rl}}$ and $\kappa_{\mathrm{g}} = \kappa\kappa_{\mathrm{rg}}$. The determination of relative permeability curves is often done empirically or experimentally and the form strongly depends on the porous material properties and the liquids themselves.The functions that are used in this model

(Ref. 2) are defined such that they are always positive:

$$\kappa_{rg} = \begin{cases} 1-1{,}1S_l, & S_l < 1/1{,}1 \\ \text{eps}, & S_l \geq 1/1{,}1 \end{cases}$$

$$\kappa_{rl} = \begin{cases} \left(\dfrac{S_l - S_{li}}{1 - S_{li}}\right)^3, & S_l > S_{li} \\ \text{eps}, & S_l \leq S_{li} \end{cases}$$

The variable S_{li} is the irreducible liquid phase saturation, describing the saturation of the liquid phase that will remain inside the porous medium.

Results and Discussion

The temperature (Figure 2) shows significant cooling in the whole domain.

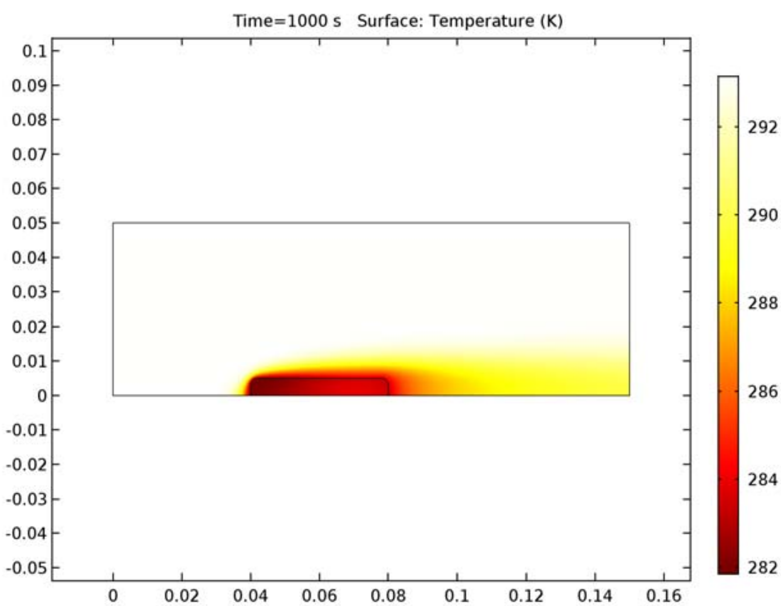


Figure 2: Temperature field after 1000s.

The default pressure contour plot (Figure 3) shows that pressure gradients inside the porous domain dominate over the pressure losses in the free flow domain. The gradients are steepest at the interface which requires a fine mesh resolution.

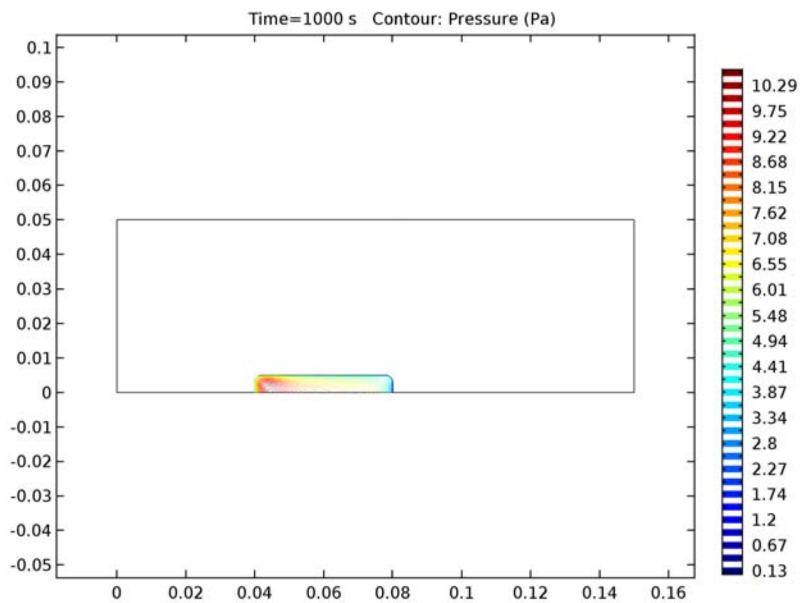


Figure 3: Pressure distribution. Due to the high pressure gradient contour lines are only visible inside the porous domain.

Inside the porous domain the relative humidity is close to 100 % everywhere (Figure 4).

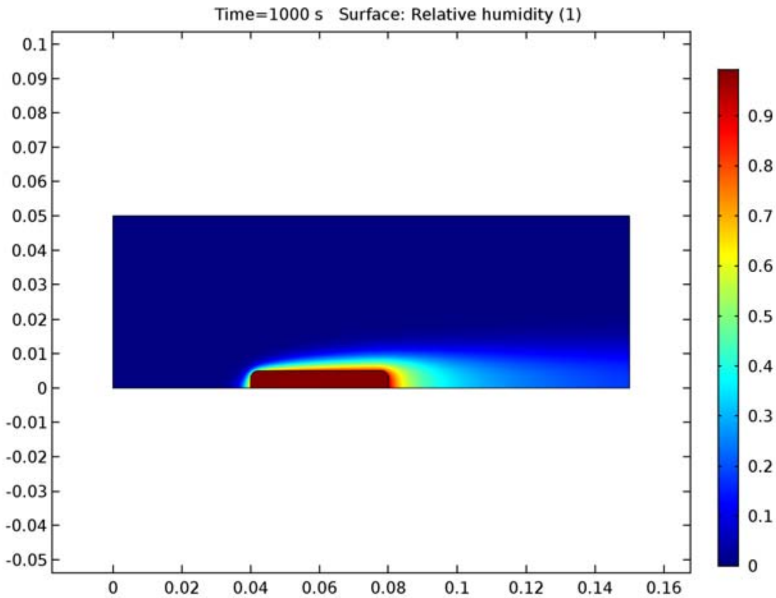


Figure 4: Relative humidity after 1000 s.

Notes About the COMSOL Implementation

Using a proper mesh size is important to resolve the steep gradients at the interface boundaries. Therefore a customized mesh with boundary layers is used.

To get good convergence of the time dependent behavior, first solve the stationary flow equations only. This solution will then be used as initial value for the time dependent study step providing smooth initial conditions in the modeling domain. In addition the default nonlinear solver is changed to be more stable for this strongly coupled problem.

References

1. A.K. Datta, "Porous media approaches to studying simultaneous heat and mass transfer in food processes. I: Problem formulations", *Journal of Food Engineering*, vol. 80, 2007.

2. A.K. Datta, "Porous media approaches to studying simultaneous heat and mass transfer in food processes. II: Property data and representative results, *Journal of Food Engineering*, vol. 80, 2007.

3. Heat_Transfer_Module/Phase_Change/evaporation_porous_media_small_rate

Application Library path: Heat_Transfer_Module/Phase_Change/evaporation_porous_media_large_rate

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**.
2. Click **Done**.

GLOBAL DEFINITIONS

For this model, many parameters and variables are needed. Start by loading all of them from text files. For clarity reasons, the variables are grouped in separate variable sets.

Parameters

1. On the **Home** toolbar, click **Parameters**.
2. In the **Settings** window for Parameters, locate the **Parameters** section.
3. Click **Load from File**.
4. Browse to the application's Application Libraries folder and double-click the file `evaporation_porous_media_large_rate_parameters.txt`.

Variables 1

1. In the **Model Builder** window, right-click **Global Definitions** and choose **Variables**.
2. In the **Settings** window for Variables, type `Air Properties` in the **Label** text field.
3. Locate the **Variables** section. Click **Load from File**.
4. Browse to the application's Application Libraries folder and double-click the file `evaporation_porous_media_large_rate_air.txt`.

Variables 2

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

2 In the **Settings** window for Variables, type `Liquid Water Properties` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

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

Variables 3

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

2 In the **Settings** window for Variables, type `Water Vapor Properties` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

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

Variables 4

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

2 In the **Settings** window for Variables, type `Porous Matrix Properties` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

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

DEFINITIONS

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Variables 5

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

2 In the **Settings** window for Variables, type `Material Properties` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

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

Variables 6

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

2 In the **Settings** window for Variables, type `Porous Medium Velocity Variables` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

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

Variables 7

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

2 In the **Settings** window for Variables, type `Variables` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

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

Define the relative permeabilities for water vapor and liquid water as functions of liquid water saturation. The advantage of using functions is that they can be plotted immediately.

Piecewise 1 (pw1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Piecewise**.

2 In the **Settings** window for Piecewise, type `Relative Permeability, Moist Air` in the **Label** text field.

3 In the **Function name** text field, type `kappa_rma`.

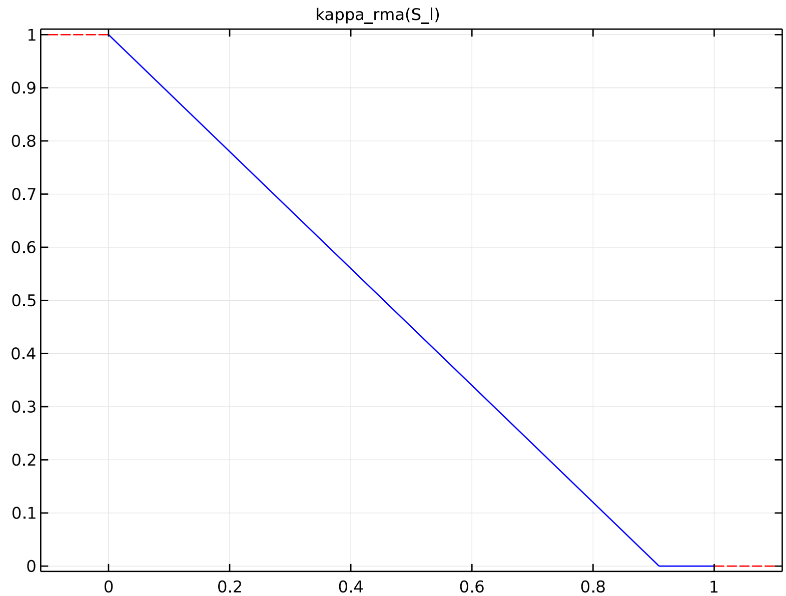
4 Locate the **Definition** section. In the **Argument** text field, type `S_l`.

5 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	1/1.1	1-1.1*S_l
1/1.1	1	eps

To force the saturation to be always positive `eps` is used as minimum value.

6 In the **Settings** window for Piecewise, click **Plot**.



Piecewise 2 (pw2)

1 On the **Home** toolbar, click **Functions** and choose **Global>Piecewise**.

2 In the **Settings** window for Piecewise, type `Relative Permeability, Liquid Phase` in the **Label** text field.

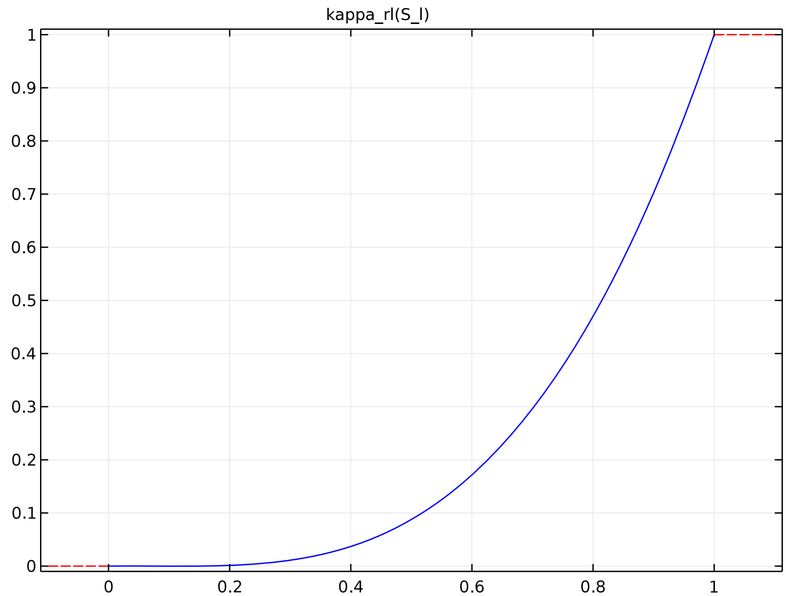
3 In the **Function name** text field, type `kappa_rl`.

4 Locate the **Definition** section. In the **Argument** text field, type `S_l`.

5 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	S_il	eps
S_il	1	((S_l-S_il)/(1-S_il))^3

6 In the **Settings** window for Piecewise, click **Plot**.



All variables are defined. Create the geometry.

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.15`.

4 In the **Height** text field, type `0.05`.

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `0.04`.

4 In the **Height** text field, type `0.005`.

5 Locate the **Position** section. In the **x** text field, type `0.04`.

Fillet 1 (fil1)

1 On the **Geometry** toolbar, click **Fillet**.

2 On the object **r2**, select Points 3 and 4 only.

3 In the **Settings** window for Fillet, locate the **Radius** section.

4 In the **Radius** text field, type `2e-3`.

5 Click **Build All Objects**.

Next, add the physics interfaces and define the domain features that set up the equations to be solved in the domains.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

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

ADD PHYSICS

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

2 In the tree, select **Fluid Flow>Porous Media and Subsurface Flow>Free and Porous Media Flow (fp)**.

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

ADD PHYSICS

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

2 In the tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.

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

ADD PHYSICS

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

2 In the tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.

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

TRANSPORT OF DILUTED SPECIES 2 (TDS2)

On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

HEAT TRANSFER IN FLUIDS (HT)

On the **Physics** toolbar, click **Transport of Diluted Species 2 (tds2)** and choose **Heat Transfer in Fluids (ht)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

2 In the **Settings** window for Heat Transfer in Fluids, locate the **Physical Model** section.

3 Select the **Heat transfer in porous media** check box.

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

Porous Medium 1

1 On the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 Select Domain 2 only.

FREE AND POROUS MEDIA FLOW (FP)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Free and Porous Media Flow (fp)**.

2 In the **Settings** window for Free and Porous Media Flow, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

Porous Matrix Properties 1

1 On the **Physics** toolbar, click **Domains** and choose **Porous Matrix Properties**.

2 Select Domain 2 only.

TRANSPORT OF DILUTED SPECIES (TDS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species (tds)**.

2 In the **Settings** window for Transport of Diluted Species, type `Transport of Diluted Species: Liquid Water` in the **Label** text field.

3 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Concentrations** table, enter the following settings:

cl

4 Select Domain 2 only.

TRANSPORT OF DILUTED SPECIES 2 (TDS2)

On the **Physics** toolbar, click **Transport of Diluted Species (tds)** and choose **Transport of Diluted Species 2 (tds2)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species 2 (tds2)**.

2 In the **Settings** window for Transport of Diluted Species, type `Transport of Diluted Species 2: Water Vapor` in the **Label** text field.

3 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Concentrations** table, enter the following settings:

cv

TRANSPORT OF DILUTED SPECIES 2: WATER VAPOR (TDS2)

Transport Properties 2

1 On the **Physics** toolbar, click **Domains** and choose **Transport Properties**.

2 Select Domain 2 only.

3 In the **Settings** window for Transport Properties, type `Transport Properties 2: Porous Medium` in the **Label** text field.

Continue with the **Heat Transfer in Fluids** interface and set up the multiphysics coupling manually. Also add the heat source term due to evaporation.

HEAT TRANSFER IN FLUIDS (HT)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Fluid 1**.

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

3 From the p_A list, choose **Absolute pressure (fp)**.

4 From the $\mathbf{u}$ list, choose **Velocity field (fp/fp1)**.

5 Locate the **Heat Conduction, Fluid** section. From the k list, choose **User defined**. In the associated text field, type `k_ma`.

This is the equivalent thermal conductivity as defined in the **Variables** node.

6 Locate the **Thermodynamics, Fluid** section. From the **Fluid type** list, choose **Ideal gas**.

7 From the **Gas constant type** list, choose **Mean molar mass**.

8 From the M_n list, choose **User defined**. In the associated text field, type `Mn_ma`.

9 From the C_p list, choose **User defined**. In the associated text field, type `cp_ma`.

Porous Medium 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Porous Medium 1**.

2 In the **Settings** window for Porous Medium, locate the **Model Inputs** section.

3 From the p_A list, choose **Absolute pressure (fp)**.

4 Specify the **u** vector as

u_mean	x
v_mean	y

This is the average velocity for liquid water and water vapor as defined in the **Porous media velocity variables** node.

5 Locate the **Heat Conduction, Fluid** section. From the k list, choose **User defined**. In the associated text field, type `k_tot`.

6 Locate the **Thermodynamics, Fluid** section. From the ρ list, choose **User defined**. In the associated text field, type `rho_tot`.

7 From the C_p list, choose **User defined**. In the associated text field, type `cp_tot`.

8 From the γ list, choose **User defined**. Locate the **Immobile Solids** section. In the θ_p text field, type `1-por`.

9 Locate the **Heat Conduction, Porous Matrix** section. From the k_p list, choose **User defined**. In the associated text field, type `k_p`.

10 Locate the **Thermodynamics, Porous Matrix** section. From the ρ_p list, choose **User defined**. In the associated text field, type `rho_p`.

11 From the $C_{\mathrm{p,\,p}}$ list, choose **User defined**. In the associated text field, type `cp_p`.

12 In the **Model Builder** window, click **Heat Transfer in Fluids (ht)**.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 2 only.

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

4 In the Q_0 text field, type `-H_evap*Mn_l*m_evap`.

FREE AND POROUS MEDIA FLOW (FP)

Continue with the **Free and Porous Media Flow** interface.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Free and Porous Media Flow (fp)** click **Fluid Properties 1**.

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

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

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

Porous Matrix Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Free and Porous Media Flow (fp)** click **Porous Matrix Properties 1**.

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

3 From the ε_p list, choose **User defined**. In the associated text field, type `por*S_ma`.

The pore space is partially filled with liquid water, so that the available space for water vapor depends on the porosity and the saturation.

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

5 In the **Model Builder** window, click **Free and Porous Media Flow (fp)**.

Mass Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Mass Source**.

2 Select Domain 2 only.

3 In the **Settings** window for Mass Source, locate the **Mass Source** section.

4 In the Q_{br} text field, type `Mn_l*m_evap`.

TRANSPORT OF DILUTED SPECIES: LIQUID WATER (TDS)

Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species: Liquid Water (tds)** click **Transport Properties 1**.

2 In the **Settings** window for Transport Properties, locate the **Model Inputs** section.

3 Specify the **u** vector as

-px*kappa_l/(S_l*por*mu_l)	x
-py*kappa_l/(S_l*por*mu_l)	y

4 Locate the **Diffusion** section. In the D_{cl} text field, type `D_cap`.

The velocity field for the transport of liquid water can directly be derived from the vapor flow velocity and must take into account the available pore space and relative permeability.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species: Liquid Water (tds)** click **Initial Values 1**.

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

3 In the *cl* text field, type `S_l_0*por*rho_l/Mn_l`.

4 In the **Model Builder** window, click **Transport of Diluted Species: Liquid Water (tds)**.

Reactions 1

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

2 In the **Settings** window for Reactions, locate the **Reaction Rates** section.

3 In the R_{cl} text field, type `-m_evap`.

4 Select Domain 2 only.

TRANSPORT OF DILUTED SPECIES 2: WATER VAPOR (TDS2)

Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species 2: Water Vapor (tds2)** click **Transport Properties 1**.

2 In the **Settings** window for Transport Properties, locate the **Model Inputs** section.

3 From the **u** list, choose **Velocity field (fp/fp1)**.

4 Locate the **Diffusion** section. In the D_{cv} text field, type `D_va`.

Transport Properties 2: Porous Medium

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species 2: Water Vapor (tds2)** click **Transport Properties 2: Porous Medium**.

2 In the **Settings** window for Transport Properties, locate the **Model Inputs** section.

3 Specify the **u** vector as

`u/(por*S_ma)-Mn_a*D_eff*rho_ma*d(Mn_ma/rho_ma,x)/Mn_ma^2`	x
`v/(por*S_ma)-Mn_a*D_eff*rho_ma*d(Mn_ma/rho_ma,y)/Mn_ma^2`	y

4 Locate the **Diffusion** section. In the D_{cv} text field, type `D_eff*Mn_a/Mn_ma`.

This sets up the water vapor transport according to Vapor Transport in porous media.

5 In the **Model Builder** window, click **Transport of Diluted Species 2: Water Vapor (tds2)**.

Initial Values 2

1 On the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 2 only.

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

4 In the *cv* text field, type `cv_sat_0`.

Reactions 1

1 On the **Physics** toolbar, click **Domains** and choose **Reactions**.
2 Select Domain 2 only.
3 In the **Settings** window for Reactions, locate the **Reaction Rates** section.
4 In the R_{cv} text field, type `m_evap`.

HEAT TRANSFER IN FLUIDS (HT)

Define the boundary conditions for each interface. Again, start with the **Heat Transfer in Fluids** interface. Fix the temperature at the inlet and define the outflow boundary condition.

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, choose **Ambient temperature (ht)** from the T list.
3 In the **Model Builder** window, click **Heat Transfer in Fluids (ht)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundary 1 only.
3 In the **Settings** window for Temperature, locate the **Temperature** section.
4 From the T_0 list, choose **Ambient temperature (ht)**.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2 Select Boundary 9 only.

FREE AND POROUS MEDIA FLOW (FP)

In the **Model Builder** window, under **Component 1 (comp1)** click **Free and Porous Media Flow (fp)**.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2 Select Boundary 1 only.
3 In the **Settings** window for Inlet, locate the **Boundary Condition** section.
4 From the list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} text field, type u0.

Outlet 1

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

2 Select Boundary 9 only.

TRANSPORT OF DILUTED SPECIES 2: WATER VAPOR (TDS2)

In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species 2: Water Vapor (tds2)**.

Concentration 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for Concentration, locate the **Concentration** section.

4 Select the **Species cv** check box.

Outflow 1

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

2 Select Boundary 9 only.

MESH 1

To resolve all effects and get a better convergence for this highly nonlinear problem, build a mesh manually with boundary layers to resolve the interface properly.

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.

Size

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

2 From the **Calibrate for** list, choose **Fluid dynamics**.

3 From the **Predefined** list, choose **Finer**.

Size 1

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

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

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

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

5 Select Boundaries 2–4, 6–8, 10, and 11 only.

6 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

7 From the **Predefined** list, choose **Extremely fine**.

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

Boundary Layer Properties

1 Right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for Boundary Layer Properties, locate the **Boundary Selection** section.

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

4 Select Boundaries 2–4, 6–8, 10, and 11 only.

5 Locate the **Boundary Layer Properties** section. In the **Thickness adjustment factor** text field, type `0.5`.

6 Click **Build All**.

ADD STUDY

Add a stationary study that solves for the fluid flow only without the mass source term. This serves as initial value for the time dependent solution of the whole process.

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

In the table, clear the **Solve for** check box for all interfaces except for the **Free and Porous Media Flow** interface.

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

STUDY 1

Step 1: Stationary

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

2 Select the **Modify physics tree and variables for study step** check box.

3 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Free and Porous Media Flow (fp)>Mass Source 1**.

4 Click **Disable**.

Now the **Mass Source** condition is disabled for the stationary study.

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

Time Dependent

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

Step 2: Time Dependent

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

2 In the **Times** text field, type `range(0,5,1000)`.

The problem is highly nonlinear, so tune some solver settings to get a better convergence.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. Select the **Maximum step** check box.

5 In the associated text field, type `50`.

6 Find the **Algebraic variable settings** subsection. From the **Consistent initialization** list, choose **Off**.

7 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

8 In the **Settings** window for Fully Coupled, click to expand the **Method and termination** section.

9 Locate the **Method and Termination** section. In the **Damping factor** text field, type `0.5`.

10 From the **Jacobian update** list, choose **On every iteration**.

11 In the **Maximum number of iterations** text field, type `15`.

The reduced damping factor forces a smaller hence more stable non-linear solver step. The setting of **Jacobian update** to **On every iteration** will update the matrix after each iteration and hence will also improve stability. This may require more iterations until a stable solution is found. That is why the **Maximum number of iterations** is increased.

12 In the **Settings** window for Fully Coupled, click **Compute**.

RESULTS

2D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Relative Humidity` in the **Label** text field.

3 On the **Relative Humidity** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Relative Humidity** click **Surface 1**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Definitions> Variables>phi - Relative humidity**.

3 On the **Relative Humidity** toolbar, click **Plot**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.2a

Evaporation in Porous Media with Small Evaporation Rates

Introduction

Evaporation in porous media is an important process in food and paper industry among others. Many physical effects must be considered: fluid flow, heat transfer and transport of participating fluids and gases. All of these effects are strongly coupled and can be handled by using the predefined interfaces of COMSOL Multiphysics. When having a liquid and a gaseous phase inside the porous domain, some adjustments of the predefined equations are required. In this tutorial, evaporation is assumed to have a negligible impact on the amount of liquid water, making it possible to describe the transport of water vapor with a convection-diffusion equation.

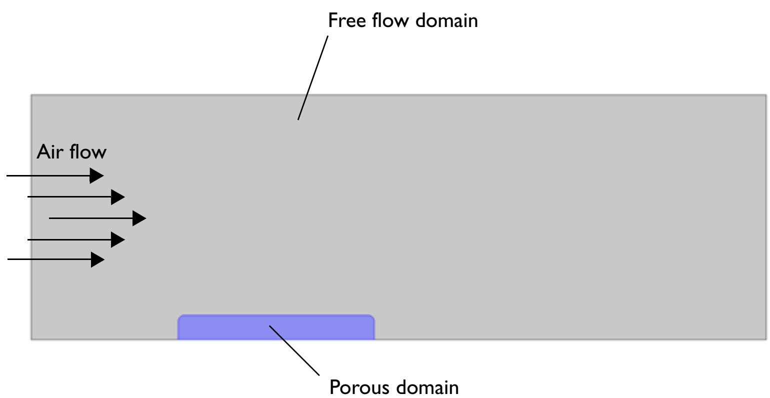


Figure 1: Geometry and principle of the model.

Model Definition

This tutorial describes laminar air flow through a humid porous medium. The air is dry at the inlet and its moisture content increases as it flows through the porous media.

The flow inside the porous medium is described with Brinkman equation. The flow in the surrounding domain is described with the laminar Navier-Stokes equation. The geometry and basic set-up is shown in Figure 1.

THERMODYNAMIC PROPERTIES

The thermodynamic properties of air with water vapor can be described using mixture laws, based on the amount of water vapor and dry air. This is done automatically when choosing Moist air as fluid type and the governing equations can be found in the *Heat Transfer Module User's Guide*. As input term for water vapor, the concentration c (mol/m^3) from the transport equation is used.

FLOW PROPERTIES

To model the fluid flow from the air domain, using the laminar Navier-Stokes equation into the porous domain using Brinkman equation the Laminar Flow Interface is used and extended by enabling porous media domains. This way, the coupling of both flow regimes is done automatically. The resulting velocity field then can be used to model convective heat and species transport.

TRANSPORT PROPERTIES

The fraction of water vapor is small and the Transport of Diluted Species interface is used to describe the transport properties in the free flow and porous domain. In the porous domain, the diffusion coefficient for water vapor into air D_L (m^2/s) needs to be adjusted according to

$$D_e = \frac{\varepsilon_p}{\tau_L} D_L .$$

This describes the effective diffusivity inside a porous medium, depending on its structure, characterized by the dimensionless numbers porosity ε_p and tortuosity τ_L. Here, the Bruggeman correction is used, which is

$$D_e = \varepsilon_p^{3/2} D_L \tag{1}$$

EVAPORATION

Including the evaporation process is done by adding the mass of water vapor that is evaporated as source term in the transport equation. Evaporation takes place, if the concentration of water vapor is below the equilibrium concentration, which is determined by the saturation concentration c_{sat} and water activity a_w, with

$$c_{sat} = \frac{p_{sat}(T)}{RT} \tag{2}$$

with the saturation pressure p_{sat} and the ideal gas constant $R = 8.314$ J/(mol·K). The water activity describes the amount of water that evaporates into air. In general it is a function depending on the water content on dry basis of the surrounding air and the temperature (see for example Ref. 2), but here an approximate value of $a_{\mathrm{w}} = 0.9$ is used because of the moderate variation of water content. Hence the amount of evaporated water is defined as:

$$m_{\mathrm{vap}} = K \cdot (a_{\mathrm{w}} c_{\mathrm{sat}} - c) \tag{3}$$

where K (1/s) is the evaporation rate, and c the current concentration. The evaporation rate depends on the material properties and the process which causes the evaporation. It must be chosen so that the solution is not affected if further increased. This corresponds to assuming that vapor is in equilibrium with the liquid or in other words, the time scale for evaporation is much smaller than the smallest time scale of the transport equations. This is true for pore sizes that are not too large (Ref. 1).

The heat of evaporation is then inserted as source term in the heat transfer equation according to:

$$Q = H_{\mathrm{vap}} \cdot m_{\mathrm{vap}} \tag{4}$$

where H_{vap} (J/mol) is the latent heat of evaporation.

Results and Discussion

After one minute the temperature field shows strong cooling due to evaporation. As seen below, evaporation mainly occurs at the surface of the porous medium. Inside the domain, the temperature gradients remain moderate.

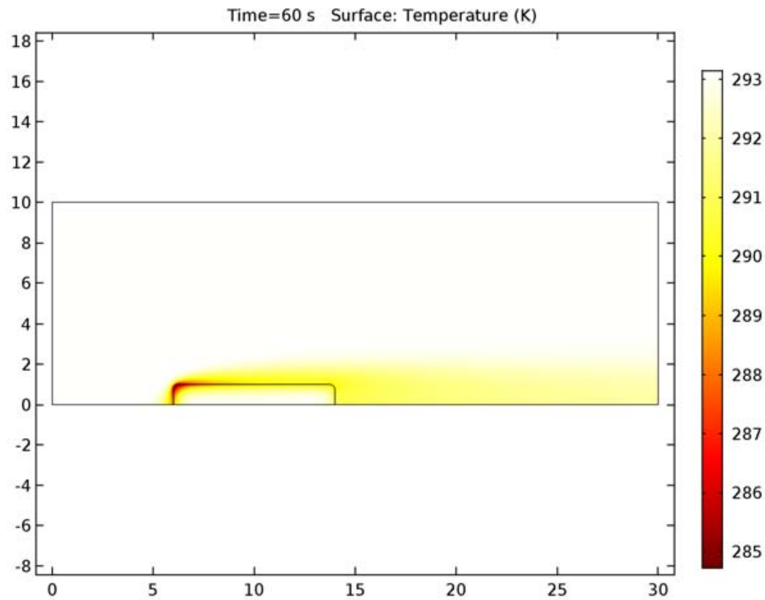


Figure 2: Temperature distribution after 60 s.

The concentration distribution is shown in the next plot.

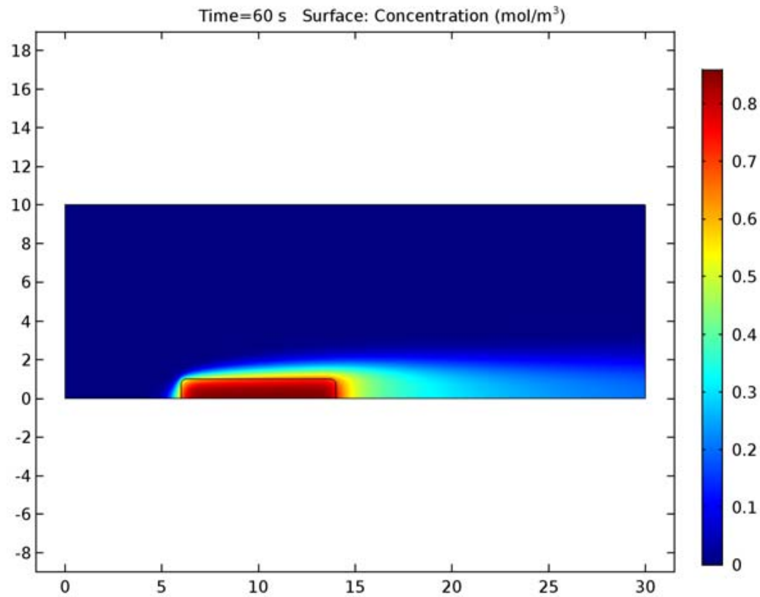


Figure 3: Concentration distribution after 60 s

The pressure field shows steep gradients at the interface boundaries.

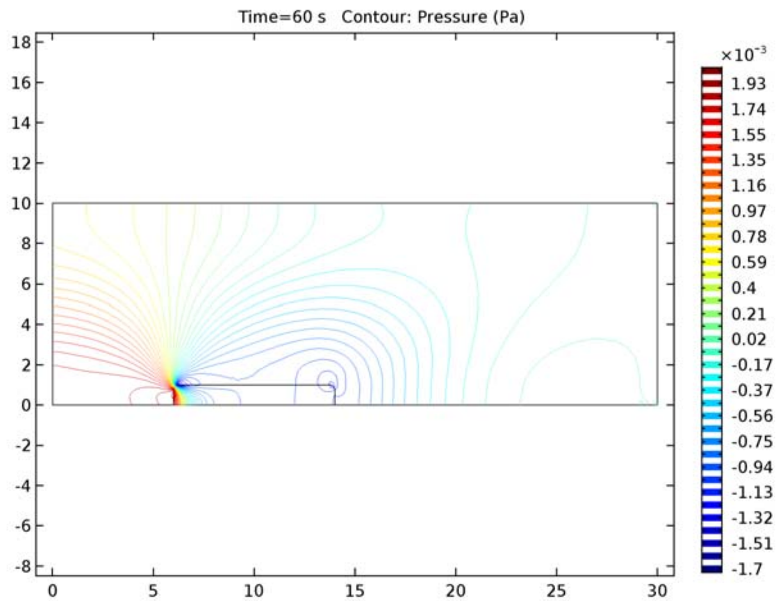


Figure 4: Pressure field combining the pressure from the Laminar Flow interface and the Darcy's Law interface.

Notes About the COMSOL Implementation

Using a proper mesh size is important to resolve the steep gradients at the interface boundaries. Therefore a customized mesh with boundary layers is used.

To get good convergence of the time dependent behavior, first solve the stationary flow equation only. This solution will then be used as initial value for the time dependent study step providing smooth initial conditions in the modeling domain.

Instead of coupling the Transport of Dilutes Species Interface with the flow interface manually, the Reacting Flow in Porous Media interface provides a predefined coupling of both equations. Then, the coupling to the heat transfer interface is the only coupling that has to be done manually. This interface is available with one of the following modules: Batteries and Fuel Cells Module, CFD Module or Chemical Reaction Engineering Module.

Reference

1. A. Halder, A. Dhall and A.K. Datta, "Modeling Transport in Porous Media with Phase Change: Applications to Food Processing", *J. Heat Transfer*, vol. 133, no. 3, 2011.

2. A.K. Datta, "Porous media approaches to studying simultaneous heat and mass transfer in food processes. II: Property data and representative results, *Journal of Food Engineering, Vol. 80, (2007)*

Application Library path: `Heat_Transfer_Module/Phase_Change/evaporation_porous_media_small_rate`

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**.

2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

10 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `30`.

4 In the **Height** text field, type `10`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `8`.

4 Locate the **Position** section. In the **x** text field, type `6`.

Fillet 1 (fil1)

1 On the **Geometry** toolbar, click **Fillet**.

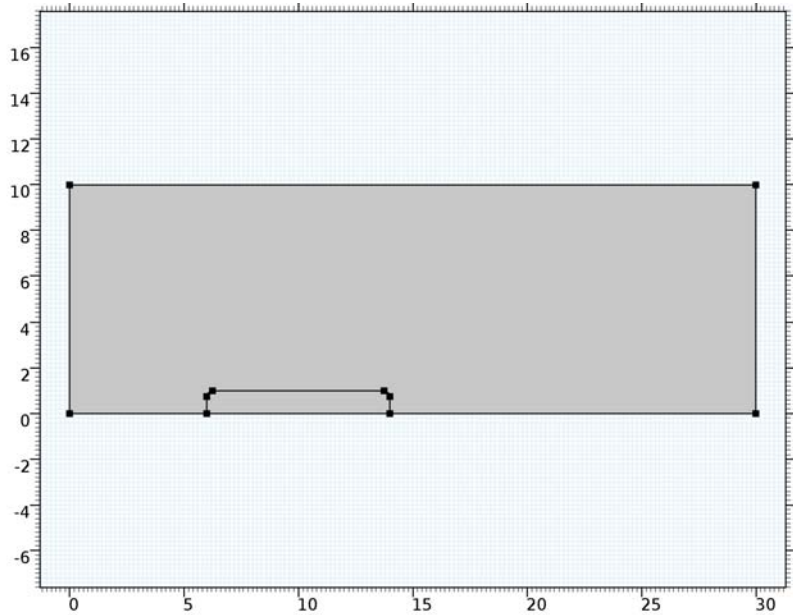
2 On the object **r2**, select Points 3 and 4 only.

3 In the **Settings** window for Fillet, locate the **Radius** section.

4 In the **Radius** text field, type `0.25`.

5 On the **Geometry** toolbar, click **Build All**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.



Start with defining some parameters.

Parameters

On the **Home** toolbar, click **Parameters**.

GLOBAL DEFINITIONS

Parameters

1 In the **Settings** window for Parameters, locate the **Parameters** section.

2 In the table, enter the following settings:

Name	Expression	Value	Description
p0	1[atm]	1.0133E5 Pa	Ambient pressure
T0	20[degC]	293.15 K	Ambient temperature
u0	0.1[m/s]	0.1 m/s	Free stream velocity
D_wa	2.6e-5[m^2/s]	2.6E-5 m²/s	Water-air diffusivity
M_w	0.018[kg/mol]	0.018 kg/mol	Molar mass of water
H_vap	2.454e6[J/kg]*M_w	44172 J/mol	Heat of vaporization

Name	Expression	Value	Description
K_evap	1000[1/s]	1000 1/s	Evaporation rate
a_w	0.9	0.9	Water activity

Set up the different physics interfaces. Start with the heat transfer interface, where you define the thermodynamic properties of moist air for the free flow and porous domain.

HEAT TRANSFER IN FLUIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

2 In the **Settings** window for Heat Transfer in Fluids, locate the **Physical Model** section.

3 Select the **Heat transfer in porous media** check box.

Porous Medium 1

1 On the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

Set the ambient temperature to be used in boundary conditions of the Heat Transfer interface.

2 In the **Settings** window for Heat Transfer in Fluids, locate the **Ambient Settings** section.

3 In the T_{amb} text field, type T0.

Use the velocity from the **Laminar Flow** interface as input. Later it is customized to take the porous properties into account.

4 In the **Model Builder** window, click **Porous Medium 1**.

5 Select Domain 2 only.

6 In the **Settings** window for Porous Medium, locate the **Model Inputs** section.

7 From the p_A list, choose **Absolute pressure (spf)**.

8 From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

Select **Moist Air** as **Fluid type**. The moisture content is evaluated using the concentration from the **Transport of Dilutes Species** interface.

9 Locate the **Thermodynamics, Fluid** section. From the **Fluid type** list, choose **Moist air**.

10 From the **Input quantity** list, choose **Concentration**.

11 Locate the **Model Inputs** section. From the c list, choose **Concentration (tds)**.

12 Locate the **Immobile Solids** section. In the θ_{p} text field, type 0.2.

Similar settings apply for the **Fluid** node.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Fluid 1**.

2 In the **Settings** window for Fluid, locate the **Model Inputs** section.

3 From the p_A list, choose **Absolute pressure (spf)**.

4 From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

5 Locate the **Thermodynamics, Fluid** section. From the **Fluid type** list, choose **Moist air**.

6 From the **Input quantity** list, choose **Concentration**.

7 Locate the **Model Inputs** section. From the c list, choose **Concentration (tds)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 1 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 From the T_0 list, choose **Ambient temperature (ht)**.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 9 only.

The next step is to set up the source term for the heat of vaporization according to Equation 4.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 2 only.

3 In the **Settings** window for Heat Source, locate the **Heat Source** section.

4 In the Q_0 text field, type `-H_vap*m_evap`.

   The variable `m_evap` is defined according to Equation 3 and depends on the saturation concentration (Equation 2) and water activity.

DEFINITIONS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

Variables 1

1 In the **Settings** window for Variables, locate the **Variables** section.

2 In the table, enter the following settings:

Name	Expression	Unit	Description
c_sat	ht.fluid1.fpsat(T)/(R_const*T)	mol/m³	Saturation vapor concentration
m_evap	K_evap*(a_w*c_sat-c)	mol/(m³·s)	Evaporated mass of water vapor

Continue with the **Transport of Diluted Species** interface. Water vapor is produced (m_evap) and saturation concentration is reached in the porous domain. For water-air diffusivity the Bruggeman correction (Equation 1) is used to describe the diffusive transport.

TRANSPORT OF DILUTED SPECIES (TDS)

Transport Properties 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species (tds)** click **Transport Properties 1**.
2. In the **Settings** window for Transport Properties, locate the **Model Inputs** section.
3. From the **u** list, choose **Velocity field (spf)**.
4. Locate the **Diffusion** section. In the D_c text field, type D_wa.

   At the inlet for the free air flow the concentration is set to zero which implies dry air.
5. In the **Model Builder** window, click **Transport of Diluted Species (tds)**.

Concentration 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Concentration**.
2. Select Boundary 1 only.
3. In the **Settings** window for Concentration, locate the **Concentration** section.
4. Select the **Species c** check box.

Outflow 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2. Select Boundary 9 only.

Initial Values 2

1. On the **Physics** toolbar, click **Domains** and choose **Initial Values**.
2. Select Domain 2 only.
3. In the **Settings** window for Initial Values, locate the **Initial Values** section.
4. In the c text field, type ht.porous1.fpsat(T0)/(R_const*T0).

Reactions 1

1. On the **Physics** toolbar, click **Domains** and choose **Reactions**.
2. Select Domain 2 only.
3. In the **Settings** window for Reactions, locate the **Reaction Rates** section.
4. In the R_c text field, type `m_evap`.

Porous Media Transport Properties 1

1. On the **Physics** toolbar, click **Domains** and choose **Porous Media Transport Properties**.
2. Select Domain 2 only.
3. In the **Settings** window for Porous Media Transport Properties, locate the **Model Inputs** section.
4. From the $\mathbf{u}$ list, choose **Velocity field (spf)**.
5. Locate the **Diffusion** section. In the $D_{F,c}$ text field, type `D_wa`.
6. From the **Effective diffusivity model** list, choose **Bruggeman model**.

The remaining task is to set up the flow interface for the free and porous domain.

LAMINAR FLOW (SPF)

The next step enables additional features for the flow interface to account for porous domains also.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2. In the **Settings** window for Laminar Flow, locate the **Physical Model** section.
3. Select the **Enable porous media domains** check box.

Fluid Properties 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Fluid Properties 1**.
2. In the **Settings** window for Fluid Properties, locate the **Fluid Properties** section.
3. From the ρ list, choose **User defined**. In the associated text field, type `ht.rho`.
4. From the μ list, choose **Dynamic viscosity (ht)**.

   This step sets the density to the density that is calculated by the moist air feature within the **Heat Transfer** interface.

   Set up the porous domain properties.
5. In the **Model Builder** window, click **Laminar Flow (spf)**.

Fluid and Matrix Properties 1

1. On the **Physics** toolbar, click **Domains** and choose **Fluid and Matrix Properties**.
2. Select Domain 2 only.
3. In the **Settings** window for Fluid and Matrix Properties, locate the **Fluid Properties** section.
4. From the ρ list, choose **User defined**. In the associated text field, type `ht.rho`.
5. From the μ list, choose **Dynamic viscosity (ht)**.

Inlet 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. Select Boundary 1 only.
3. In the **Settings** window for Inlet, locate the **Boundary Condition** section.
4. From the list, choose **Laminar inflow**.
5. Locate the **Laminar Inflow** section. In the U_{av} text field, type `u0`.

   The **Entrance length** value must be large enough so that the flow can reach a laminar profile. For a laminar flow, L_{entr} should be significantly greater than $0.06\mathrm{Re}D$, where Re is the Reynolds number and D is the inlet length scale. In this case, 30 m is an appropriate value.
6. In the L_{entr} text field, type `30`.

Outlet 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. Select Boundary 9 only.

Specify the porous material properties.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1. In the **Settings** window for Material, type `Porous Matrix` in the **Label** text field.
2. Select Domain 2 only.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	0.21	W/(m·K)	Basic
Density	rho	1528	kg/m³	Basic
Heat capacity at constant pressure	Cp	1650	J/(kg·K)	Basic
Porosity	epsilon	0.4	1	Basic
Permeability	kappa	1e-13	m²	Basic

Steep gradients are present and the default mesh is adjusted to resolve the flow and transport equations with high accuracy.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

Free Triangular 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

Size 1

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Domain**.

3 Select Domain 1 only.

4 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

5 From the **Predefined** list, choose **Finer**.

Free Triangular 1

Right-click **Free Triangular 1** and choose **Size**.

Size 2

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Domain**.

3 Select Domain 2 only.

4 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

5 From the **Predefined** list, choose **Extra fine**.

Boundary Layers 1

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.
2. In the **Settings** window for Boundary Layers, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **All domains**.

Boundary Layer Properties

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.
2. Select Boundaries 2–4, 6–8, 10, and 11 only.
3. In the **Settings** window for Boundary Layer Properties, locate the **Boundary Layer Properties** section.
4. In the **Number of boundary layers** text field, type 5.
5. In the **Thickness adjustment factor** text field, type 2.
6. Click **Build All**.

First, solve the stationary flow equations only to provide a fully developed flow profile for the whole equation system that is solved in a time interval of 1 minute.

STUDY 1

Step 1: Stationary

1. In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.
2. In the table, clear the **Solve for** check box for **Heat Transfer in Fluids (ht)** and **Transport of Diluted Species (tds)**.

Time Dependent

On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

Step 2: Time Dependent

1. In the **Settings** window for Time Dependent, locate the **Study Settings** section.
2. Click **Range**.
3. In the **Range** dialog box, type 0 in the **Start** text field.
4. In the **Step** text field, type 1.
5. In the **Stop** text field, type 60.
6. Click **Replace**.
7. On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

Several default plots are created automatically. The first plot group shows the temperature field as in Figure 2.

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

Concentration (tds)

The concentration distribution (see Figure 3) is computed from the **Transport of Diluted Species** interface.

Pressure (spf)

The last plot group shows the pressure field computed from the **Laminar Flow** interface (Figure 4).

Created in COMSOL Multiphysics 5.2a

Evaporative Cooling of Water

Introduction

This tutorial shows how to simulate cooling of water including evaporative cooling. As an example, a beaker filled with water is used surrounded by an air domain. The airflow transports the water vapor which causes the liquid to cool down. The approach used here neglects volume change of the water inside the beaker. This is a reasonable assumption for problems where the considered time is short compared to the time needed to evaporate a noticeable amount of water.

Model Definition

The model geometry is shown in Figure 1. Symmetry is used to reduce the model size.

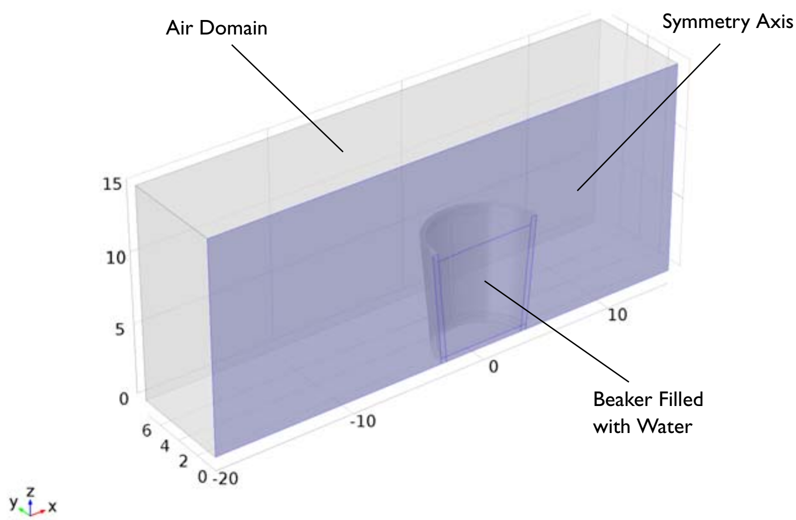


Figure 1: Model geometry, using symmetry.

The beaker is made of glass and contains hot water at 80 °C. The air has an initial temperature of 20 °C and enters the modeling domain with this temperature.

For modeling evaporative cooling, three effects must be taken into account: turbulent flow of the air around the beaker, heat transfer in all domains, and transport of water vapor in the air. This is a real multiphysics problem and this tutorial shows how to set it up.

TURBULENT FLOW

The air flow is modeled with the Turbulent Flow, Low Re k-ε interface, because the Reynolds number is about 1500 and turbulent effects must be considered. In addition they must be taken into account in the transport equations correctly. With the Low Re k-ε turbulence model, the turbulence variables are solved in the whole domain down to the walls and thus provide accurate input values for the transport equations. Assuming that the velocity and pressure field are independent of the air temperature and moisture content. This allows to calculate the turbulent flow field in advance and then use it as input for the heat transfer and species transport equations.

HEAT TRANSFER

The heat transfer inside the beaker and water is due to conduction only. For the air, convection dominates the heat transfer and the turbulent flow field is required. The material properties are determined by the moist air theory.

During evaporation, latent heat is released from the water surface which cools down the water in addition to convective and conductive cooling by the surrounding. This additional heat flux depends on the amount of evaporated water. This means that the fraction of convective and diffusive flux normal to the water surface contributes to the evaporative heat flux.

$$-\mathbf{n}\cdot(-k\nabla T) = H_{\mathrm{vap}}\mathbf{n}\cdot(-D\nabla c + \mathbf{u}c) \tag{1}$$

The latent heat of vaporization H_{vap} is given in kJ/mol.

TRANSPORT OF WATER VAPOR

To obtain the correct amount of water which is evaporated from the beaker into the air, the Transport of Diluted Species interface is used in the air domain. The initial concentration is chosen so that there is an initial relative humidity of 20 %. The source term for water vapor at the water surface is given by the ideal gas law at saturation pressure.

$$c_{\mathrm{vap}} = \frac{p_{\mathrm{sat}}}{R_{\mathrm{g}}T} \tag{2}$$

The transport equation again uses the turbulent flow field as input. Turbulence must also be considered for the diffusion coefficient, by adding the following turbulent diffusivity to the diffusion tensor:

$$\mathbf{D}_{\mathrm{T}} = \frac{\nu_{\mathrm{T}}}{\mathrm{Sc}_{\mathrm{T}}}\mathbf{I} \tag{3}$$

where ν_T is the turbulent kinematic viscosity, Sc_T is the turbulent Schmidt number and $\mathbf{I}$ the unit matrix.

Results and Discussion

The image below shows the temperature field after 20 s with streamlines indicating the flow field.

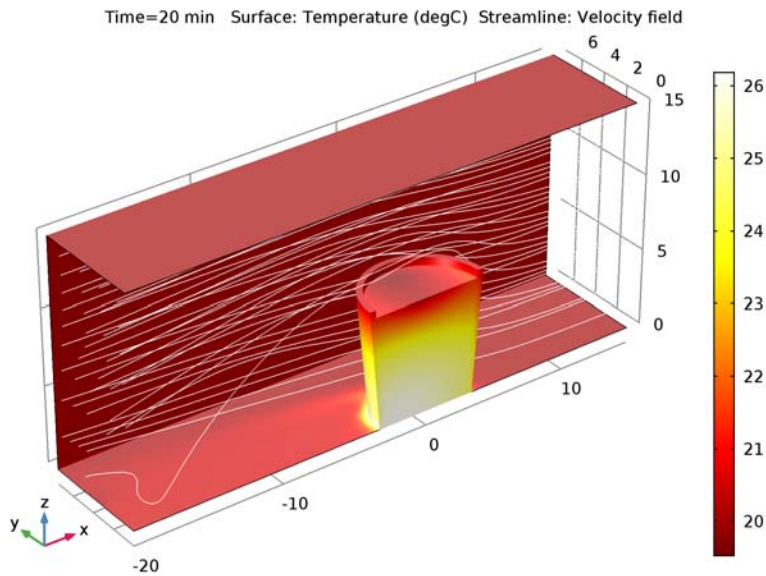


Figure 2: Temperature distribution after 20 min and streamlines indicating the flow field.

Due to convection, conduction, and evaporation the water cools down over time. As shown in Figure 3 the average temperature after 20 minutes is about 61 °C.

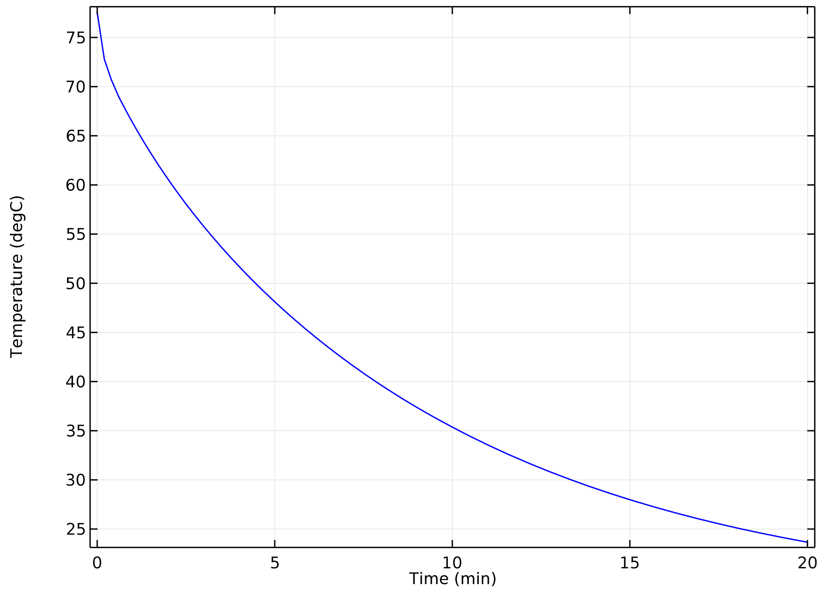


Figure 3: Average water temperature over time.

Figure 4 shows the concentration and relative humidity at the symmetry plane. Close to the water surface, the relative humidity is about 100% as expected. Behind the beaker the relative humidity can become even smaller than 20%. Due to the high temperature, air can absorb a higher amount of water.

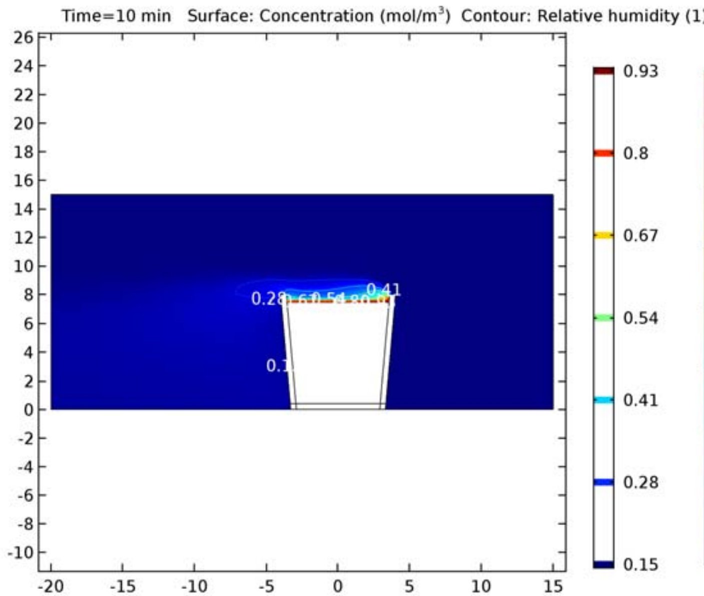


Figure 4: Concentration distribution and contour lines for the relative humidity.

A last study computes for a fictive situation where the evaporation effects are neglected. Figure 5 shows the comparison between average water temperatures with and without evaporation to see the its importance in the cooling process. A difference of 6 K can be observed.

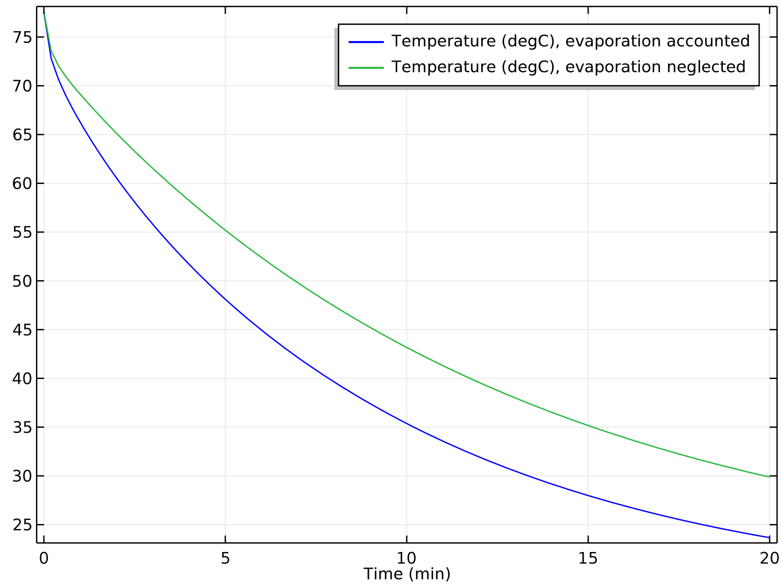


Figure 5: Average water temperature with and without evaporation accounted.

Application Library path: `Heat_Transfer_Module/Phase_Change/evaporative_cooling`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

The first step is to compute the turbulent flow field. After that, the resulting velocity field will be used to compute the transport properties of heat and moisture. To get accurate input values for the turbulent transport equations, the Low-Reynolds k-ε turbulence model is used here.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, Low Re k-ε (spf)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary with Initialization**.
6 Click **Done**.

GEOMETRY 1

Load the geometry sequence from an existing MPH file.

1 On the **Geometry** toolbar, click **Insert Sequence**.
2 Browse to the application's Application Libraries folder and double-click the file `evaporative_cooling_geom_sequence.mph`.

Cone 1 (cone1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Cone 1 (cone1)**.
2 In the **Settings** window for Cone, click **Build All Objects**.
3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

   The flow calculation is done for the air domain only. For now, air is the only material you need.

ADD MATERIAL

1 On 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>Air**.
4 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

TURBULENT FLOW, LOW RE K-ε (SPF)

Since the density variation is not small, the flow can not be regarded as incompressible. Therefore set the flow to be compressible.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, Low Re k-ε (spf)**.

2 In the **Settings** window for Turbulent Flow, Low Re k-ε, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

4 Locate the **Domain Selection** section. Click **Clear Selection**.

5 Select Domain 1 only.

Create a selection from this domain. It can be used later to create new selections or to assign physical properties.

6 Click **Create Selection**.

7 In the **Create Selection** dialog box, type `Air` in the **Selection name** text field.

8 Click **OK**.

Now, define the boundary conditions.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 33 only.

3 In the **Settings** window for Inlet, locate the **Velocity** section.

4 In the U_0 text field, type `2`.

Open Boundary 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundary 1 only.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundary 2 only.

The default **Physics-controlled mesh** is used. It is optimized for flow field calculations using a suitable mesh size and boundary layers according to the settings of the flow interface.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extra coarse**.

STUDY 1

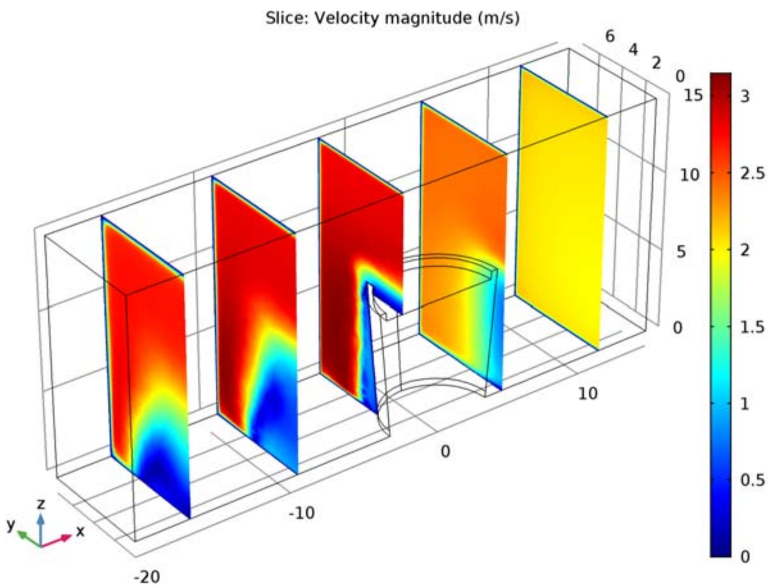
On the **Home** toolbar, click **Compute**.

RESULTS

Velocity (spf)

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The resulting velocity field is shown below:



With this velocity field, the transport equations can be computed. The **Heat Transfer in Fluids** interface together with the **Transport of Diluted Species** interface are used to describe the transport of heat and water vapor and the interaction of both processes.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component** in the window toolbar.

ADD PHYSICS

1 Go to the **Add Physics** window.

2 In the tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

4 Click **Add to Component** in the window toolbar.

The transport equation is active in the air domain only, precisely where the flow equation was solved.

TRANSPORT OF DILUTED SPECIES (TDS)

1 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species (tds)**.

3 In the **Settings** window for Transport of Diluted Species, locate the **Domain Selection** section.

4 From the **Selection** list, choose **Air**.

MATERIALS

Add the materials needed for the heat transfer calculations.

ADD MATERIAL

1 On 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 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat2)**.

2 Select Domain 3 only.

3 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Water` in the **Selection name** text field.

6 Click **OK**.

With this selection and the one for the air domain, it is easy to create the selection for the glass body.

DEFINITIONS

Complement 1

1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for Complement, type `Glass` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, In the **Selections to invert** list, choose **Air** and **Water**.

5 Click **OK**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Glass (quartz)**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Glass (quartz) (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Glass (quartz) (mat3)**.

2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Glass**.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

HEAT TRANSFER IN FLUIDS (HT)

On the **Physics** toolbar, click **Transport of Diluted Species (tds)** and choose **Heat Transfer in Fluids (ht)**.

This switches to the **Heat Transfer in Fluids** interface.

Configure the settings of **Heat Transfer in Fluids 1** for the water domain. To save computational time, the velocity field driven by natural convection is not computed.

Instead, an increased thermal conductivity determined by built-in Nusselt number correlations is defined in the next steps to compensate the missing convective heat flux.

Fluid 1

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

2 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Fluid 1**.

3 In the **Settings** window for Fluid, click to expand the **Equivalent conductivity for convection** section.

4 Locate the **Equivalent Conductivity for Convection** section. Select the **Equivalent conductivity for convection** check box.

5 From the **Nusselt number correlation** list, choose **Vertical rectangular cavity**.

6 In the H text field, type `7.5[cm]`.

7 In the L text field, type `3[cm]`.

8 In the ΔT text field, type `3`.

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, choose **Ambient temperature (ht)** from the T list.

Now, add a second **Heat Transfer in Fluids** node for the air domain. Here, the thermodynamic properties will be determined for moist air. This also enables additional postprocessing variables such as relative humidity.

3 In the **Model Builder** window, click **Heat Transfer in Fluids (ht)**.

Fluid 2

1 On the **Physics** toolbar, click **Domains** and choose **Fluid**.

2 In the **Settings** window for Fluid, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

The humidity is obtained from the concentration distribution calculated within the **Transport of Diluted Species** interface.

4 Locate the **Thermodynamics, Fluid** section. From the **Fluid type** list, choose **Moist air**.

5 From the **Input quantity** list, choose **Concentration**.

6 Locate the **Model Inputs** section. From the c list, choose **Concentration (tds)**.

In the glass body, heat is only transported by conduction.

Then, add a **Heat Transfer in Solids** node for the glass domain.

Solid 1

1 On the **Physics** toolbar, click **Domains** and choose **Solid**.

2 In the **Settings** window for Solid, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Glass**.

The air enters the domain at room temperature. At the outlet, heat is transported by convection.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 33 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 From the T_0 list, choose **Ambient temperature (ht)**.

Open Boundary 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundary 1 only.

3 In the **Settings** window for Open Boundary, locate the **Open Boundary** section.

4 From the T_0 list, choose **Ambient temperature (ht)**.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

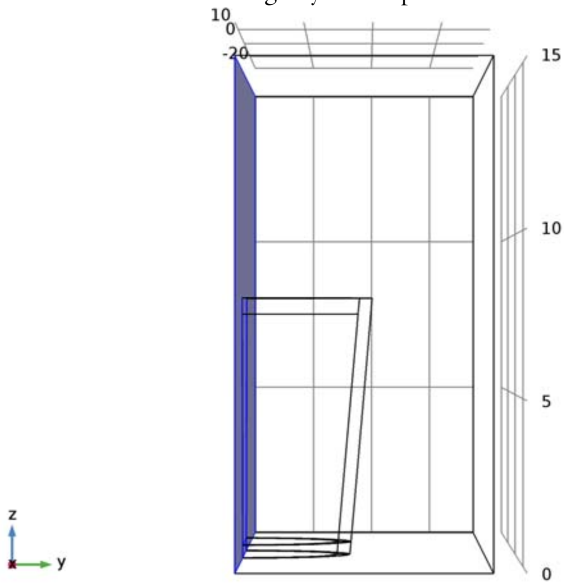
2 Click the **Go to YZ View** button on the **Graphics** toolbar.

3 Click the **Select Box** button on the **Graphics** toolbar.

With this tool, draw a box around all symmetry boundaries, which corresponds to:

4 Select Boundaries 2, 6, 11, 13, 18, 31, and 32 only.

You should see the following in your Graphics window:



The fluid in the beaker has an initial temperature of 80 °C.

Initial Values 2

1 On the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 3 only.

3 In the **Settings** window for Initial Values, locate the **Domain Selection** section.

4 From the **Selection** list, choose **Water**.

5 In the T text field, type `80[degC]`.

The water will cool down over time. To include the heat loss by evaporation, add a heat source at appropriate boundaries according to Equation 1.

Boundary Heat Source 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Heat Source**.

2 Select Boundary 12 only.

3 In the **Settings** window for Boundary Heat Source, locate the **Boundary Heat Source** section.

4 In the Q_b text field, type `-H_vap*c_lm[mol/(s*m^2)]`.

The two variables `H_vap` and `c_lm` in this expression are not defined yet. The Lagrange multiplier, `c_lm`, is the flux corresponding to a prescribed concentration condition at this boundary. This variable will hence be defined later in the **Transport of Diluted Species** physics interface yet to be created. For `H_vap`, use the global parameters to define it according to the steps below.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
M_w	18.015[g/mol]	0.01802 kg/mol	Molar mass of water
H_vap	2454[kJ/kg]*M_w	4.421E4 J/mol	Heat of vaporization

To couple the flow field in the air domain to the heat transport equation, follow the steps below:

MULTIPHYSICS

Non-Isothermal Flow 1 (nitf1)

On the **Physics** toolbar, click **Multiphysics** and choose **Domain>Non-Isothermal Flow**.

The water inside the beaker is assumed to be at rest. To disable the flow coupling there, go back to the heat transfer in fluids node and set the velocity field to zero.

HEAT TRANSFER IN FLUIDS (HT)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Fluid 1**.

2 In the **Settings** window for Fluid, locate the **Model inputs** section.

3 Click **Make All Model Inputs Editable** in the upper-right corner of the section. Locate the **Model Inputs** section. From the T list, choose **Temperature (ht)**.

The next step is to set up the **Transport of Diluted Species** interface which is active in the air domain only. Use the **Flow Coupling** feature from the Multiphysics node and define the diffusion coefficient for water in air.

MULTIPHYSICS

Flow Coupling 1 (fc1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Global>Flow Coupling**.

2 In the **Settings** window for Flow Coupling, locate the **Coupled Interfaces** section.

3 From the **Destination** list, choose **Transport of Diluted Species (tds)**.

TRANSPORT OF DILUTED SPECIES (TDS)

Define the diffusion coefficient for water vapor taking into account turbulent effects according to Equation 3.

Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species (tds)** click **Transport Properties 1**.

2 In the **Settings** window for Transport Properties, locate the **Diffusion** section.

3 In the D_c text field, type `2.6e-5[m^2/s]`.

Turbulent Mixing 1

1 On the **Physics** toolbar, click **Attributes** and choose **Turbulent Mixing**.

2 In the **Settings** window for Turbulent Mixing, locate the **Turbulent Mixing Parameters** section.

3 From the ν_T list, choose **Turbulent kinematic viscosity (spf/fp1)**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
rho_w_max	0.0172[kg/m^3]	0.0172 kg/m³	Maximum amount of water for 100% humidity at 20°C
rel_hum	0.2	0.2	Relative humidity
c0	rel_hum*rho_w_max/M_w	0.191 mol/m³	Initial concentration

TRANSPORT OF DILUTED SPECIES (TDS)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Diluted Species (tds)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, locate the **Initial Values** section.

3 In the c text field, type `c0`.

Concentration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Concentration**.

2 Select Boundary 33 only.

3 In the **Settings** window for Concentration, locate the **Concentration** section.

4 Select the **Species c** check box.

5 In the $c_{0,c}$ text field, type `c0`.

At the water surface the saturation pressure is reached and the saturation concentration is applied at that boundary, according to Equation 2. With the moist air feature, the saturation pressure is calculated automatically and can be accessed directly.

Concentration 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Concentration**.

2 Select Boundary 12 only.

3 In the **Settings** window for Concentration, locate the **Concentration** section.

4 Select the **Species c** check box.

5 In the $c_{0,c}$ text field, type `ht.psat/(R_const*T)`.

In the following steps, use weak constraints for this boundary condition to create the Lagrange multiplier, `c_lm`, that corresponds to the flux. This variable have been used in the expression of the heat source at this same boundary in the **Heat Transfer in Fluids** physics interface.

6 Click to expand the **Constraint settings** section. Locate the **Constraint Settings** section. Select the **Use weak constraints** check box.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 1 only.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundary 2 only.

Since the turbulent flow is already solved, the new mesh can be optimized for the transport equations. A fine mesh at the water-air interface is required.

COMPONENT 1 (COMP1)

Mesh 2

On the **Mesh** toolbar, click **Add Mesh**.

MESH 2

In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Mesh 2** and choose **Free Tetrahedral**.

Free Tetrahedral 1

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

2 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes>Mesh 2** right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

5 Select Boundary 12 only.

6 Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `0.15`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes>Mesh 2** click **Size**.

2 In the **Settings** window for Size, locate the **Element Size** section.

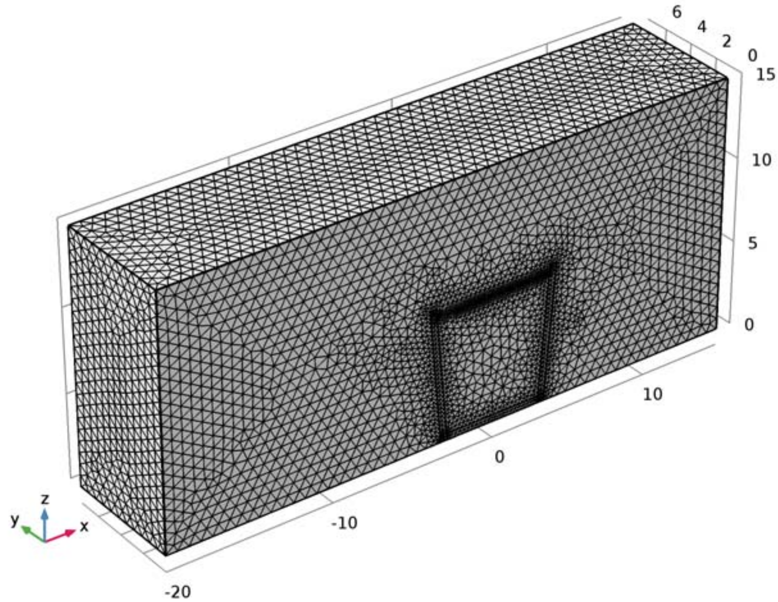
3 From the **Predefined** list, choose **Extremely fine**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Resolution of narrow regions** text field, type `1.5`.

6 Click **Build All**.

7 Click the **Go to Default 3D View** button on the **Graphics** toolbar.



Add a transient study to model the cooling over time.

ADD STUDY

1. On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.
4. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Turbulent Flow, Low Re k-ε (spf)** interface.
5. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Time Dependent

1. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.
2. In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.
3. In the **Settings** window for Time Dependent, locate the **Study Settings** section.

4 From the **Time unit** list, choose **min**.
5 Click **Range**.
6 In the **Range** dialog box, type `12[s]` in the **Step** text field.
7 In the **Stop** text field, type `20`.
8 Click **Replace**.

The second study requires the result from the first one. To make the data from **Study 1** available for **Study 2** follow these steps:

9 In the **Settings** window for Time Dependent, click to expand the **Values of dependent variables** section.
10 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
11 From the **Method** list, choose **Solution**.
12 From the **Study** list, choose **Study 1, Stationary**.

The heat transfer and transport equations are strongly coupled via the moist air feature using the concentration as input. This requires to configure a **Direct** solver in a **Fully Coupled** approach.

Solution 3 (sol3)

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.
3 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.
4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.
5 Right-click **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1** and choose **Fully Coupled**.
6 Right-click **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1>Direct** and choose **Enable**.
7 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

Create the plot shown in Figure 2.

Surface 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Surface**.

2 In the **Settings** window for Surface, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

4 Locate the **Selection** section. From the **Selection** list, choose **All boundaries**.

5 Remove boundaries 1, 2 and 33 from the list.

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 **Data set** list, choose **Surface 2**.

Surface

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 On the **Temperature (ht)** toolbar, click **Streamline**.

Streamline 1

1 In the **Model Builder** window, under **Results>Temperature (ht)** click **Streamline 1**.

2 In the **Settings** window for Streamline, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **On selected boundaries**.

6 Locate the **Selection** section. Select the **Active** toggle button.

7 Select Boundary 33 only.

8 Locate the **Streamline Positioning** section. In the **Number** text field, type 40.

9 Click the **Zoom Extents** button on the **Graphics** toolbar.

10 On the **Temperature (ht)** toolbar, click **Plot**.

Data Sets

To visualize the concentration distribution as in Figure 4, follow the next steps.

Cut Plane 1

1 On the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for Cut Plane, locate the **Plane Data** section.

3 From the **Plane** list, choose **xz-planes**.

4 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

2D Plot Group 8

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Concentration and Relative Humidity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Plane 1**.

4 From the **Time (min)** list, choose **10**.

Surface 1

1 Right-click **Concentration and Relative Humidity** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Transport of Diluted Species>c - Concentration**.

3 On the **Concentration and Relative Humidity** toolbar, click **Plot**.

Concentration and Relative Humidity

In the **Model Builder** window, under **Results** right-click **Concentration and Relative Humidity** and choose **Contour**.

Contour 1

1 In the **Settings** window for Contour, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer in Fluids> Moist air>ht.phi - Relative humidity**.

2 Locate the **Levels** section. In the **Total levels** text field, type `7`.

3 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Tube**.

4 Select the **Radius scale factor** check box.

5 In the associated text field, type `0.025`.

6 Select the **Level labels** check box.

7 In the **Precision** text field, type `2`.

8 From the **Label color** list, choose **White**.

9 On the **Concentration and Relative Humidity** toolbar, click **Plot**.

The relative humidity decreases quickly with the distance to the surface. Due to the high temperature behind the beaker, the relative humidity becomes even lower than 20%.

Derived Values

It is interesting to see how the average temperature decreases with time.

Volume Average 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Average>Volume Average**.
2 In the **Settings** window for Volume Average, type `Average Water Temperature` in the **Label** text field.
3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.
4 Select Domain 3 only.
5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Heat Transfer in Fluids>Temperature>T - Temperature**.
6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`T`	`degC`	`Temperature`

7 Click **Evaluate**.

TABLE

1 Go to the **Table** window.
2 Click **Table Graph** in the window toolbar.

RESULTS

1D Plot Group 9

1 In the **Model Builder** window, under **Results** click **1D Plot Group 9**.
2 In the **Settings** window for 1D Plot Group, type `Average Water Temperature over Time` in the **Label** text field.

Finally, compute how much water is evaporated in the air.

Surface Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.
2 In the **Settings** window for Surface Integration, type `Amount of Evaporated Water` in the **Label** text field.
3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.
4 Select Boundary 12 only.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`2*tds.dflux_cz*M_w`	`kg/s`	

6 Locate the **Data Series Operation** section. From the **Operation** list, choose **Integral**.

7 Click **New Table**.

TABLE

1 Go to the **Table** window.

The factor 2 in the expression is based on the use of a symmetry condition. Within 20 minutes, about 2.5 g of water have been evaporated.

ROOT

Repeat now the previous instructions to create a third study that solves for a fictive model where evaporation is neglected. A comparison with the results returned by **Study 2** would then highlight and quantify the cooling effects of evaporation.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Turbulent Flow, Low Re k-ε (spf)** interface.

5 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Time Dependent**.

6 Click **Add Study** in the window toolbar.

STUDY 3

Step 1: Time Dependent

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.

3 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

4 From the **Time unit** list, choose **min**.

5 Click **Range**.

6 In the **Range** dialog box, type `12[s]` in the **Step** text field.

7 In the **Stop** text field, type 20.

8 Click **Replace**.

9 In the **Settings** window for Time Dependent, locate the **Physics and Variables Selection** section.

10 Select the **Modify physics tree and variables for study step** check box.

11 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Heat Transfer in Fluids (ht)>Boundary Heat Source 1**.

12 Click **Disable**.

By disabling **Boundary Heat Source 1** in this study, evaporation is neglected.

13 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

14 From the **Method** list, choose **Solution**.

15 From the **Study** list, choose **Study 1, Stationary**.

Solution 4 (sol4)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

5 Right-click **Study 3>Solver Configurations>Solution 4 (sol4)>Time-Dependent Solver 1** and choose **Fully Coupled**.

6 Right-click **Study 3>Solver Configurations>Solution 4 (sol4)>Time-Dependent Solver 1> Direct** and choose **Enable**.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Derived Values

Compute the average temperature of water when evaporation is neglected.

Volume Average 2

1 On the **Results** toolbar, click **More Derived Values** and choose **Average>Volume Average**.

2 In the **Settings** window for Volume Average, type `Average Water Temperature, Evaporation Neglected` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 3/Solution 4 (sol4)**.

4 Select Domain 3 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Heat Transfer in Fluids>Temperature>T - Temperature**.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
T	degC	Temperature

7 Click **Table 1 - Average Water Temperature (T)**.

Table Graph 1

1 In the **Model Builder** window, under **Results>Average Water Temperature over Time** click **Table Graph 1**.

2 In the **Settings** window for Table Graph, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Temperature (degC), evaporation accounted
Temperature (degC), evaporation neglected

6 On the **Average Water Temperature over Time** toolbar, click **Plot**.

In this model, evaporation accounts for a decrease of about 6 K at the end of the simulation (see Figure 5).

Created in COMSOL Multiphysics 5.2a

Viscous Heating in a Fluid Damper

Introduction

Fluid dampers are used in military devices for shock isolation and in civil structures for suppressing earthquake-induced shaking and wind-induced vibrations, among many other applications. Fluid dampers work by dissipating the mechanical energy into heat (Ref. 1). This example shows the phenomenon of viscous heating and consequent temperature increase in a fluid damper. Viscous heating is also important in microflow devices, where a small cross-sectional area and large length of the device can generate significant heating and affect the fluid flow consequently (Ref. 2).

Model Definition

The structural elements of a fluid damper are relatively few. Figure 1 depicts a schematic of the fluid damper modeled herein with its main components: damper cylinder housing, piston rod, piston head, and viscous fluid in the chamber. There is a small annular space between the piston head and the inside wall of the cylinder housing. This acts as an effective channel for the fluid. As the piston head moves back and forth inside the damper cylinder, fluid is forced to pass through the annular channel with large shear rate, which leads to significant heat generation. The heat is transferred in both the axial and radial directions. In the radial direction, the heat is conducted through the cylinder house wall and convected to the air outside the damper, which is modeled using the Newton's convective cooling law.

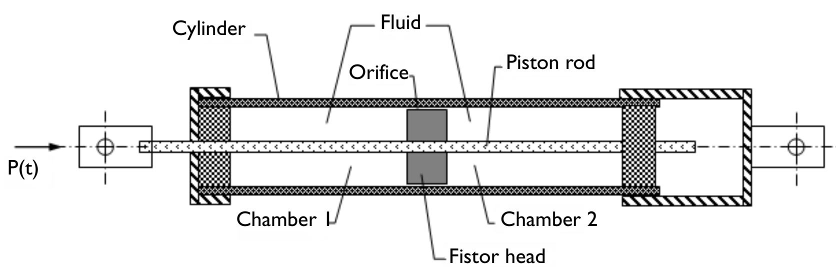


Figure 1: A sketch of a typical fluid damper with its major components

You make use of the axially symmetric nature of the fluid damper and model it in a 2D-axisymmetric geometry as shown in Figure 2. The geometric dimensions and other parameters of the damper are taken according to Ref. 1 to represent the smaller, 15 kip damper experimentally studied therein. Thus, the piston head has a diameter of 8.37 cm, the piston rod diameter is 2.83 cm, and the gap thickness is about 1/100 of the piston head diameter. The damper has the maximum stroke U_0 of 0.1524 m. The damper solid parts are made of steel, and the damping fluid is silicone oil.

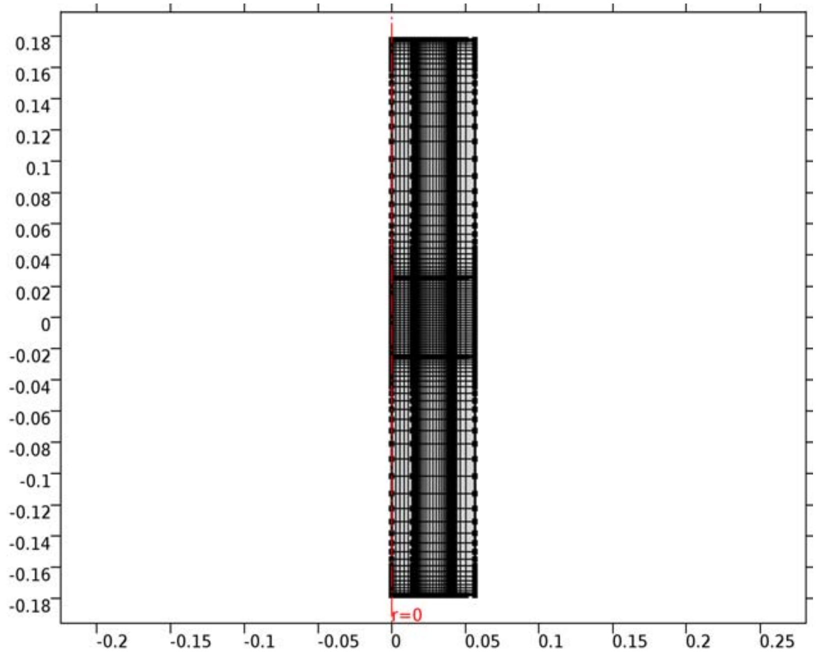


Figure 2: Geometry and mesh. The domains (from left to right) represent: piston rod, piston head and damping fluid space, the damper outer wall.

FLUID FLOW

The fluid flow in the fluid damper is described by the weakly compressible Navier-Stokes equations, solving for the velocity field $\mathbf{u} = (u, w)$ and the pressure p:

$$\rho\frac{\partial\mathbf{u}}{\partial t} + \rho\mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p + \nabla\cdot\left(\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \frac{2}{3}\mu(\nabla\cdot\mathbf{u})\mathbf{I}\right)$$

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0$$

The density is assumed independent of the temperature, while the temperature dependence of the fluid viscosity is taken into account as:

$$\mu = \mu_0 - \alpha(T - T_0) \tag{1}$$

The reference material properties of silicone oil are used.

No Slip wall boundary conditions are applied for both ends of the damper cylinder and on the inner wall of the damper cylinder house. Moving/sliding wall with the given velocity is applied on the boundaries of the piston head and on the piston rod.

CONJUGATE HEAT TRANSFER

The conjugate heat transfer is solved both in the fluid domain and the damper cylinder house wall: heat transfer by convection and conduction in the fluid domain, heat transfer by conduction only in the solid domain, and the temperature field is continuous between the fluid and solid domains. In the fluid domain, the viscous heating is activated and pressure work can be included when the slight compressibility of the damper fluid needs to be considered:

$$\rho C_p\frac{\partial T}{\partial t} + \rho C_p\mathbf{u}\cdot\nabla T + \nabla\cdot\mathbf{q} = -\alpha_p T\frac{\partial p}{\partial t} + \mu\left[\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}} - \frac{2}{3}(\nabla\cdot\mathbf{u})\mathbf{I}\right]:\nabla\mathbf{u} + Q$$

where the first term and second terms on the right-hand side represent the heat source from pressure work and viscous dissipation, respectively. Hence, the problem is a fully coupled fluid-thermal interaction problem.

In the solid domain of the cylinder house wall, this equation reduces to conductive heat transfer equation without any heating source.

The heat flux boundary condition based on the Newton's cooling law is applied on the outside boundaries of the cylinder house wall. The temperature field is continuous between the fluid and solid domains. The ends of the damper connected to the structures outside are kept at constant temperature.

The piston head movement is provided as harmonic oscillations with given amplitude and frequency, $z = a_0\sin(2\pi f)$. The motion is modeled using the arbitrary Lagrangian-Eulerian (ALE) deformed mesh. The ALE method handles the dynamics of the deforming geometry and the moving boundaries with a moving grid. The Navier-Stokes equations for fluid flow and heat equations for temperature variation are formulated in these moving coordinates.

Results and Discussion

The modeled loading has the amplitude of 0.127 m, and the excitation frequency is 0.4 Hz. This represents the long-stroke loading experiment performed in Ref. 1. The loading time period is 40 s.

Note that the simulation results for the temperature are presented in degrees Fahrenheit for the sake of easier comparison with the experimental measurements.

Figure 3 gives the temperature field in the damper at the end the loading. It also shows a typical streamline configuration for the flow induced in the damping fluid.

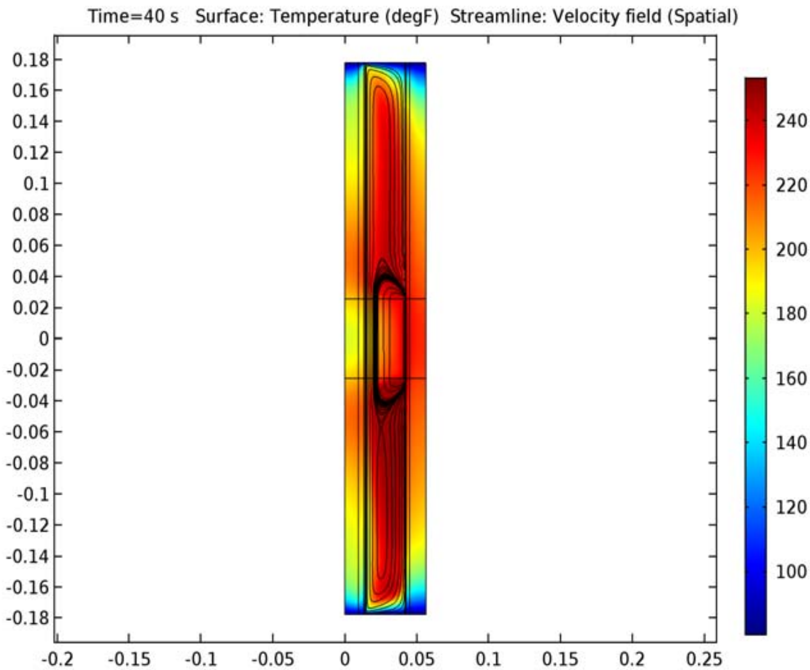


Figure 3: Temperature field in the damper at the end of simulation.

Figure 4 shows the temperature of the inner wall of the damper at the end-of-stroke position $z = U_0$. This corresponds to the internal probe position under experiments performed in Ref. 1. The simulation results show very good agreement with the experimental measurements (see Fig. 9 in Ref. 1).

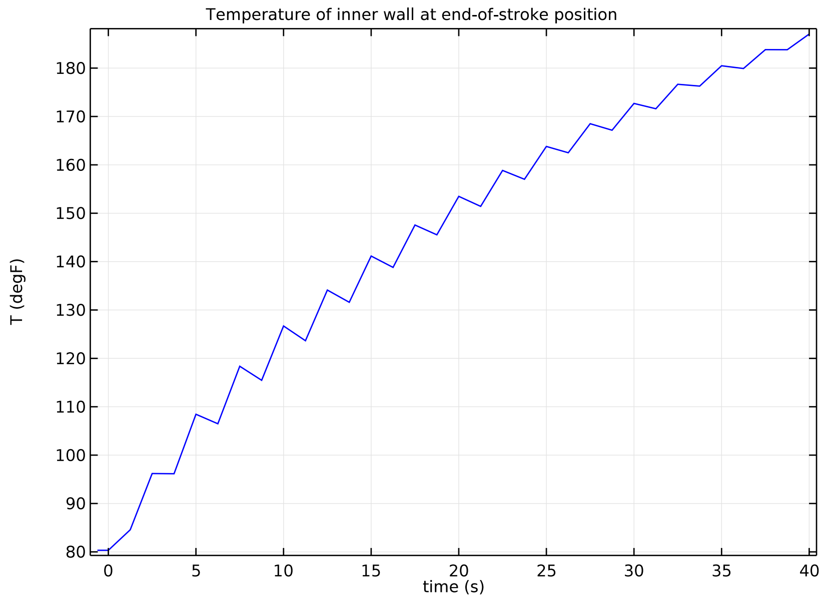


Figure 4: Temperature at the probe position.

Figure 5 shows the temperature variation along the inner wall of the damper after 10 s and 40 s of loading. It clearly shows that the temperature at the probe position does not represent the maximum temperature within the damper. This supports the conclusion drawn in Ref. 1, where the choice of the probe positioning was limited by the construction of the outer shell of the damper. Figure 5 also shows that the temperature near the center of the damper increases by more than 100 degrees already after few loading cycles.

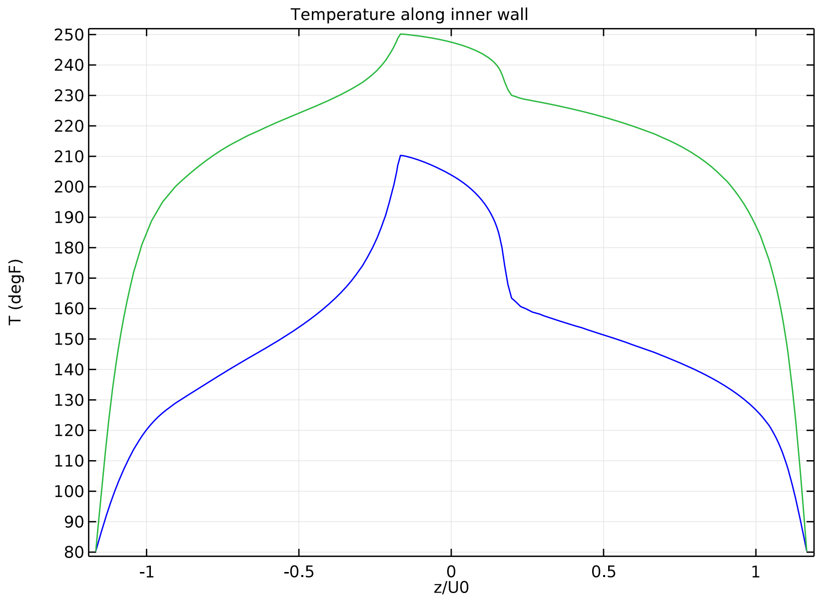


Figure 5: Temperature of the damper inner wall. The probe position corresponds to $z/U_0 = 1$.

Notes About the COMSOL Implementation

You decompose the computational domain into several parts and mesh the domains with mapped meshes to resolve the very thin annular space. For the moving mesh you prescribe the displacement of the mesh in each domain so that their alignment remains unchanged with a zero displacement at the top and the bottom of the damper cylinder housing connecting to the high-performance seal, and the displacement equal to that of the piston head is used for the domain lined up with the piston head. This is achieved by specifying the mesh displacement field as a linear function of the deformed mesh frame coordinate and the reference (material) frame coordinate.

The steel material needed for the damper solid parts is available in the built-in material library. You create a user-defined material for the silicone oil. Such damping fluids are typically characterized by the density, kinematic viscosity at the temperature 25º C, and so-called *viscosity temperature coefficient*, VTC = 1–(viscosity at 98.9º C)/(viscosity at 37.8º C). Using this parameters, you create the linear correlation for the dynamic viscosity given by Equation 1.

References

1. C.J. Black and N. Makris, "Viscous Heating of Fluid Dampers Under Small and Large Amplitude Motions: Experimental Studies and Parametric Modeling," *J. Eng. Mech.*, vol. 133, pp. 566–577, 2007.

2. G.L. Morini, "Viscous Heating in Liquid Flows in Micro-Channels," *Int. J. Heat Mass Transfer*, vol. 48, pp. 3637–3647, 2005.

Application Library path: `Heat_Transfer_Module/Buildings_and_Constructions/fluid_damper`

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 **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.
8. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, click **Parameters**.
2. In the **Settings** window for Parameters, locate the **Parameters** section.
3. Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `fluid_damper_parameters.txt`.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for Variables, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `fluid_damper_variables.txt`.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dr/2`.

4 In the **Height** text field, type `2*Ld`.

5 Locate the **Position** section. In the **z** text field, type `-Ld`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dp/2`.

4 In the **Height** text field, type `2*Ld`.

5 Locate the **Position** section. In the **z** text field, type `-Ld`.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dd/2-Hw`.

4 In the **Height** text field, type `2*Ld`.

5 Locate the **Position** section. In the **z** text field, type `-Ld`.

Rectangle 4 (r4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dd/2`.

4 In the **Height** text field, type `2*Ld`.

5 Locate the **Position** section. In the **z** text field, type `-Ld`.

Rectangle 5 (r5)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dd/2`.

4 In the **Height** text field, type `2*Lp`.

5 Locate the **Position** section. In the **z** text field, type `-Lp`.

6 On the **Geometry** toolbar, click **Build All**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

The model geometry is now complete.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 Select Domains 4 and 6–9 only.

HEAT TRANSFER (HT)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer (ht)**.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Fluid 1**.

2 Select Domains 4 and 6–9 only.

3 In the **Settings** window for Fluid, locate the **Thermodynamics, Fluid** section.

4 From the γ list, choose **User defined**.

ADD MATERIAL

1 On 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>Steel AISI 4340**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

In the following steps, you create a new material for the damping fluid, **Silicone Oil**.

Material 2 (mat2)

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 `Silicon Oil` in the **Label** text field.
3. Select Domains 4 and 6–9 only.

Silicon Oil (mat2)

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Silicon Oil (mat2)** node, then click **Basic (def)**.
2. In the **Settings** window for Property Group, locate the **Output Properties and Model Inputs** section.
3. Find the **Quantities** subsection. In the tree, select **Model Inputs>Temperature**.
4. Click **Add**.
5. Locate the **Local Properties** section. In the **Local properties** table, enter the following settings:

Property	Expression	Unit
nu_25C	0.0125[m^2/s]	m^2/s
VTC	0.6[1]	

6. In the **Model Builder** window, click **Silicon Oil (mat2)**.
7. In the **Settings** window for Material, locate the **Material Contents** section.
8. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	22.5	W/(m·K)	Basic
Density	rho	950	kg/m^3	Basic

Property	Name	Value	Unit	Property group
Heat capacity at constant pressure	Cp	2e3	J/(kg·K)	Basic
Dynamic viscosity	mu	nu_25C*rho*(1-VTC*(T-311[K])/(61[K]))/(1+VTC*0.2107)	Pa·s	Basic

MOVING MESH (ALE)

On the **Physics** toolbar, click **Heat Transfer (ht)** and choose **Moving Mesh (ale)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Moving Mesh (ale)**.

2 In the **Settings** window for Moving Mesh, locate the **Frame Settings** section.

3 From the **Geometry shape order** list, choose **1**.

Prescribed Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Deformation**.

2 Select Domains 2, 5, 8, and 11 only.

3 In the **Settings** window for Prescribed Deformation, locate the **Prescribed Mesh Displacement** section.

4 In the d_z text-field array, type `zp` on the 2nd row.

Prescribed Deformation 2

1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Deformation**.

2 Select Domains 1, 4, 7, and 10 only.

3 In the **Settings** window for Prescribed Deformation, locate the **Prescribed Mesh Displacement** section.

4 In the d_z text-field array, type `zlin1` on the 2nd row.

Prescribed Deformation 3

1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Deformation**.

2 Select Domains 3, 6, 9, and 12 only.

3 In the **Settings** window for Prescribed Deformation, locate the **Prescribed Mesh Displacement** section.

4 In the d_z text-field array, type `zlin2` on the 2nd row.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Wall 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundaries 11 and 13 only.

3 In the **Settings** window for Wall, locate the **Boundary Condition** section.

4 From the **Boundary condition** list, choose **Moving wall**.

5 Specify the $\mathbf{u}_\mathrm{w}$ vector as

`0`	`r`
`d(zp,t)`	`z`

Wall 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundaries 8, 12, and 17 only.

3 In the **Settings** window for Wall, locate the **Boundary Condition** section.

4 From the **Boundary condition** list, choose **Sliding wall**.

5 In the U_w text field, type `d(zp,t)`.

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 In the p text field, type `p0`.

HEAT TRANSFER (HT)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer (ht)**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `T0` in the T text field.

3 In the **Model Builder** window, click **Heat Transfer (ht)**.

Temperature 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2. Select Boundaries 2, 7, 9, 14, 16, 21, 23, and 28 only.
3. In the **Settings** window for Temperature, locate the **Temperature** section.
4. In the T_0 text field, type `T0`.

Heat Flux 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. Select Boundaries 29–31 only.
3. In the **Settings** window for Heat Flux, locate the **Heat Flux** section.
4. Click the **Convective heat flux** button.
5. In the T_{ext} text field, type `T0`.
6. In the h text field, type `hwall`.

LAMINAR FLOW (SPF)

Because the damper is a closed container, you need to pin-point the pressure level within. To achieve that, use the point constraint as follows.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Pressure Point Constraint 1

1. On the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2. Select Point 12 only.
3. In the **Settings** window for Pressure Point Constraint, locate the **Pressure Constraint** section.
4. In the p_0 text field, type `p0`.

MULTIPHYSICS

1. In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Non-Isothermal Flow 1 (nitf1)**.
2. In the **Settings** window for Non-Isothermal Flow, locate the **Flow Heating** section.
3. Select the **Include work done by pressure changes** check box.
4. Select the **Include viscous dissipation** check box.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

Mapped 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

Distribution 1

1 Select Boundaries 23, 25, 27, and 28 only.

2 In the **Settings** window for Distribution, locate the **Distribution** section.

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 4.

5 In the **Element ratio** text field, type 4.

6 From the **Distribution method** list, choose **Geometric sequence**.

7 Select the **Reverse direction** check box.

Mapped 1

Right-click **Mapped 1** and choose **Distribution**.

Distribution 2

1 Select Boundaries 1, 5, 8, 12, 15, 19, 22, 26, 29, and 31 only.

2 In the **Settings** window for Distribution, locate the **Distribution** section.

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 32.

5 In the **Element ratio** text field, type 8.

6 From the **Distribution method** list, choose **Geometric sequence**.

7 Select the **Symmetric distribution** check box.

Mapped 1

Right-click **Mapped 1** and choose **Distribution**.

Distribution 3

1 Select Boundaries 9, 11, 13, and 14 only.

2 In the **Settings** window for Distribution, locate the **Distribution** section.

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 30.

5 In the **Element ratio** text field, type 10.

6 From the **Distribution method** list, choose **Geometric sequence**.

7 Select the **Symmetric distribution** check box.

Mapped 1

Right-click **Mapped 1** and choose **Distribution**.

Distribution 4

1 Select Boundaries 16, 18, 20, and 21 only.

2 In the **Settings** window for Distribution, locate the **Distribution** section.

3 In the **Number of elements** text field, type `8`.

Mapped 1

Right-click **Mapped 1** and choose **Distribution**.

Distribution 5

1 Select Boundaries 3, 10, 17, 24, and 30 only.

2 In the **Settings** window for Distribution, locate the **Distribution** section.

3 In the **Number of elements** text field, type `32`.

Mapped 1

Right-click **Mapped 1** and choose **Distribution**.

Distribution 6

1 Select Boundaries 2, 4, 6, and 7 only.

2 In the **Model Builder** window, click **Mesh 1**.

3 In the **Settings** window for Mesh, click **Build All**.

The mesh is now complete. It should look similar to that shown in Figure 2.

STUDY 1

Step 1: Time Dependent

Start the simulation when the piston is in the lowest position consistent with the steady flow initial conditions.

This will not give constant temperature at $t = 0$, but the overall effect is small.

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

3 In the **Times** text field, type `-0.25/f range(0,tstep,tmax)`.

4 Click to expand the **Results while solving** section. Locate the **Results While Solving** section. Select the **Plot** check box.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

   To control the time-step manually, use the **Generalized alpha** time stepping method.

5 Locate the **Time Stepping** section. From the **Method** list, choose **Generalized alpha**.

6 From the **Steps taken by solver** list, choose **Manual**.

7 In the **Time step** text field, type `tstep/20`.

8 In the **Amplification for high frequency** text field, type `0.5`.

Before computing the solution, set up some plots, including the one to display in the **Graphics** window while solving.

RESULTS

Cut Point 2D 1

On the **Results** toolbar, click **Cut Point 2D**.

Data Sets

1 In the **Settings** window for Cut Point 2D, locate the **Point Data** section.

2 In the **r** text field, type `Dd/2-Hw`.

3 In the **z** text field, type `U0`.

2D Plot Group 1

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Temperature Surface and Velocity Streamlines, 2D` in the **Label** text field.

Temperature Surface and Velocity Streamlines, 2D

1 Right-click **Temperature Surface and Velocity Streamlines, 2D** and choose **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degF**.

4 In the **Expression** text field, type `T`.

5 Right-click **Temperature Surface and Velocity Streamlines, 2D** and choose **Streamline**.

6 In the **Settings** window for Streamline, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer> ht.ur,ht.uz - Velocity field (Spatial)**.

7 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Start point controlled**.

1D Plot Group 2

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Temperature along Inner Wall` in the **Label** text field.

Line Graph 1

On the **Temperature along Inner Wall** toolbar, click **Line Graph**.

Temperature along Inner Wall

1 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

2 In the **Expression** text field, type `T`.

3 Select Boundaries 22, 24, and 26 only.

4 From the **Unit** list, choose **degF**.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `z/U0`.

7 In the **Model Builder** window, click **Temperature along Inner Wall**.

8 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

9 From the **Title type** list, choose **Manual**.

10 In the **Title** text area, type `Temperature along inner wall`.

11 Locate the **Plot Settings** section. Select the **x-axis label** check box.

12 In the associated text field, type `z/U0`.

13 Select the **y-axis label** check box.

14 In the associated text field, type `T (degF)`.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Inner Wall Temperature at End-of-Stroke Position` in the **Label** text field.

Point Graph 1

On the **Inner Wall Temperature at End-of-Stroke Position** toolbar, click **Point Graph**.

Inner Wall Temperature at End-of-Stroke Position

1 In the **Settings** window for Point Graph, locate the **y-Axis Data** section.

2 In the **Expression** text field, type `T`.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **degF**.

5 In the **Model Builder** window, click **Inner Wall Temperature at End-of-Stroke Position**.

6 In the **Settings** window for 1D Plot Group, click to expand the **Axis** section.

7 Locate the **Title** section. From the **Title type** list, choose **Manual**.

8 In the **Title** text area, type `Temperature of inner wall at end-of-stroke position`.

9 Locate the **Plot Settings** section. Select the **x-axis label** check box.

10 In the associated text field, type `time (s)`.

11 Select the **y-axis label** check box.

12 In the associated text field, type `T (degF)`.

STUDY 1

Now return to the **Study 1** branch to compute the solution.

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 **Results While Solving** section.

3 From the **Plot group** list, choose **Inner Wall Temperature at End-of-Stroke Position**.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature Surface and Velocity Streamlines, 2D

During the solution time, a plot of the temperature at the probe position will be displayed and updated following the solver time steps.

When the solution is finished, click the **Zoom Extents** button on the **Graphics** toolbar. This will produce a plot of the temperature field and the flow streamlines within the damper, which should appear similar to that shown in Figure 3.

Temperature along Inner Wall

1 In the **Model Builder** window, under **Results** click **Temperature along Inner Wall**.

2 In the **Settings** window for 1D Plot Group, locate the **Data** section.

3 From the **Time selection** list, choose **From list**.

4 In the **Times (s)** list, choose **10** and **40**.

5 On the **Temperature along Inner Wall** toolbar, click **Plot**.

This will show the temperature distribution along the damper inner wall at times 10 s and 40 s, it should look similar to that shown in Figure 5.

Inner Wall Temperature at End-of-Stroke Position

1 In the **Model Builder** window, under **Results** click **Inner Wall Temperature at End-of-Stroke Position**.

2 On the **Inner Wall Temperature at End-of-Stroke Position** toolbar, click **Plot**.

This will show the temperature variation at the probe position aver the complete loading time period, it should look similar to that shown in Figure 4.

Created in COMSOL Multiphysics 5.2a

Radiative Cooling of a Glass Plate

Introduction

In glass production, the glass melt is cooled down mainly by radiation. To avoid stresses it is important to evenly cool the glass body.

Numerical treatment of radiative heat transfer helps to optimize this cooling process. The governing equation—the Radiative Transfer Equation (RTE)—is an integro-differential equation that requires a lot of computational cost to be solved. Therefore, COMSOL Multiphysics offers three common methods to solve the RTE along with the heat transfer equation.

This tutorial is intended to show the typical set up of all methods computing the heat transfer by radiation inside a gray medium.

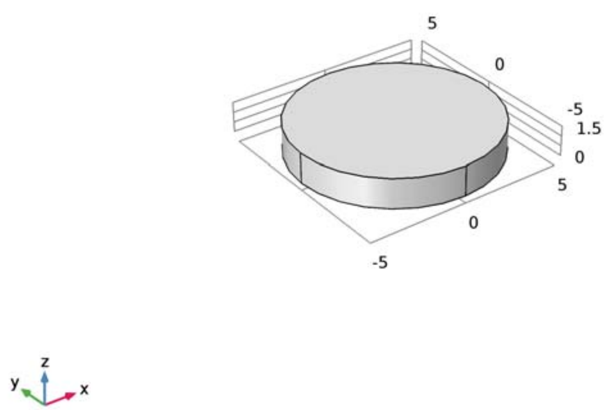


Figure 1: Cylindrical glass plate.

Model Definition

The model geometry is shown in Figure 1. It is a cylinder with of radius $r = 5$ cm and height $h = 1.5$ cm. The radiative cooling starts from an initial temperature of 600 °C due to radiation into an ambient surrounding at 20° C. Convective cooling is neglected, which is reasonable for high temperatures as in this model.

The material properties for the glass body are summarized in Table 1.

TABLE 1: MATERIAL PROPERTIES FOR GLASS

MATERIAL PROPERTY	VALUE
Thermal conductivity	1.2 W/(m·K)
Density	2200 kg/m^3
Heat capacity at constant pressure	850 J/(kg·K)
Scattering coefficient	0

Scattering effects are neglected. For the absorption coefficient a parametric sweep is used to get results for $k = 5, 70, 120$. The boundaries of the glass body have a surface emissivity, ε, equal to 1.

THERMAL ANALYSIS

A detailed explanation about the discrete ordinates method and the P1 method can be found in Radiative Heat Transfer in Finite Cylindrical Media or Radiative Heat Transfer in Finite Cylindrical Media—P1 Method respectively.

The Rosseland approximation is a simplified method that results in an additional nonlinear term for the thermal conductivity. Hence, this method has almost no impact on the computational cost.

For large optical thickness where the integral of the absorption coefficient along a typical path is large, radiation effects only spread at its close surrounding and does not travel far through the medium before being absorbed or scattered. This leads to a diffusion-like equation for the radiative heat flux (Ref. 1):

$$q_{\mathrm{r},\lambda} = -\frac{4\pi}{3\beta_\lambda}\nabla i_{\mathrm{b},\lambda}$$

For a gray medium (after integration over all wave numbers) the radiative heat flux depends on the temperature gradient and can be expressed as:

$$q_{\mathrm{r}} = -k_{\mathrm{r}}\nabla T$$

where k_{r} is a highly nonlinear coefficient, considered as a conductivity, for radiative transfer of the form

$$k_{\mathrm{r}} = \frac{16n^2\sigma T^3}{3\beta_{\mathrm{r}}}$$

with β_r, the Rosseland-mean extinction coefficient, σ the scattering coefficient and n the refractive index. Thus the Rosseland approximation method is also called the diffusion method.

Results and Discussion

The figures below compare the results for low and high absorption coefficient. The P1 method provides a very good approximation for lower absorption coefficients (Figure 2), but with increasing absorption coefficients the results differ increasingly.

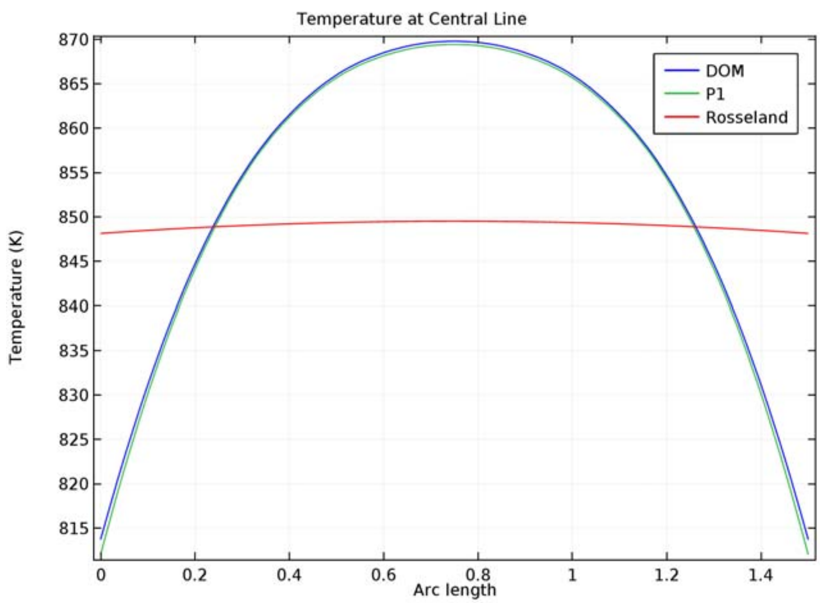


Figure 2: Vertical temperature distribution at the center of the cylinder for $k = 5$.

The Rosseland approximation provides a fast and satisfying solution for the temperature field when very high absorption coefficients or rather high optical thickness is present

(Figure 3).

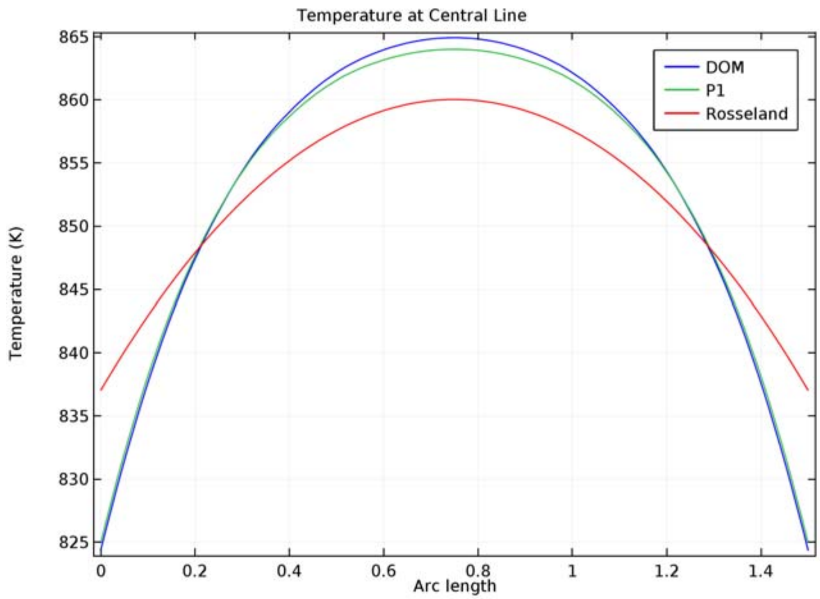


Figure 3: Vertical temperature distribution at the center of the cylinder for k = 120.

Both methods need to be used carefully and it is recommended to validate their applicability. If they can be used, they provide a very fast solution compared to the highly accurate discrete ordinates method.

Reference

1. M.F. Modest, *Radiative Heat Transfer*, 2nd ed., Academic Press, 2003.

Application Library path: `Heat_Transfer_Module/Thermal_Radiation/glass_plate`

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 **Heat Transfer>Radiation>Heat Transfer with Radiation in Participating Media (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
nr	1.45	1.45	Refractive index
T0	600[degC]	873.2 K	Initial temperature
T_amb	20[degC]	293.2 K	Ambient temperature
k	5	5	Absorption coefficient

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for Geometry, locate the **Units** section.
3 From the **Length unit** list, choose **cm**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type 5.
4 In the **Height** text field, type 1.5.
5 Click **Build All Objects**.

Create a user-defined material for the glass body.

Material 1 (mat1)

On the **Materials** toolbar, click **Blank Material**.

MATERIALS

Material 1 (mat1)

1 In the **Settings** window for Material, type `Glass` in the **Label** text field.

2 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	1.2	W/(m·K)	Basic
Density	rho	2200	kg/m³	Basic
Heat capacity at constant pressure	Cp	850	J/(kg·K)	Basic

For radiative heat transfer, the absorption and scattering coefficients are also needed. Add these properties to the material.

3 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Absorption Coefficient**.

4 Click **Add to Material**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Absorption coefficient	kappaR	k	1/m	Basic

6 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Scattering Coefficient**.

7 Click **Add to Material**.

8 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Scattering coefficient	sigmaS	0	1/m	Basic

The boundary condition for radiative cooling requires a surface emissivity. One way to define this coefficient is to add a material for the boundaries.

Material 2 (mat2)

1. On the **Materials** toolbar, click **Blank Material**.
2. In the **Settings** window for Material, type `Glass (Boundaries)` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **All boundaries**.
5. Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Surface Emissivity**.
6. Click **Add to Material**.
7. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Surface emissivity	epsilon_rad	1	1	Basic

Start with the discrete ordinate method to compute the radiative cooling of the glass plate. This method provides the most accurate solution for arbitrary radiation models and is therefore the default method for radiation in participating media. To reduce the memory requirement, you can adjust the performance index for the discrete ordinate method. The method computes the radiative intensities for a number of directions (20 directions by default) and the segregated solver only computes a few directions at once. The performance index determines the number of directions which are calculated at one segregated step and the number of segregated steps accordingly.

HEAT TRANSFER WITH RADIATION IN PARTICIPATING MEDIA (HT)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer with Radiation in Participating Media (ht)**.
2. In the **Settings** window for Heat Transfer with Radiation in Participating Media, locate the **Participating Media Settings** section.
3. From the P_{index} list, choose **0.6**.
4. In the n_{r} text field, type `nr`.

Initial Values 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer with Radiation in Participating Media (ht)** click **Initial Values 1**.
2. In the **Settings** window for Initial Values, type `T0` in the T text field.

   Set the initial temperature to 600 °C.

Radiation in Participating Media 1

1 On the **Physics** toolbar, click **Domains** and choose **Radiation in Participating Media**.

2 Select Domain 1 only.

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 In the **Settings** window for Diffuse Surface, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Ambient** section. In the T_{amb} text field, type `T_amb`.

MESH 1

Build a suitable mesh manually. First, mesh the surface with a free triangular mesh and then add a swept mesh.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundary 4 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.

2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 Select Edges 4, 5, 8, and 11 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

Free Triangular 1

Right-click **Free Triangular 1** and choose **Size**.

Size 2

1 In the **Settings** window for Size, locate the **Element Size** section.

2 From the **Predefined** list, choose **Fine**.

3 Click **Build Selected**.

4 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 1** and choose **Distribution**.

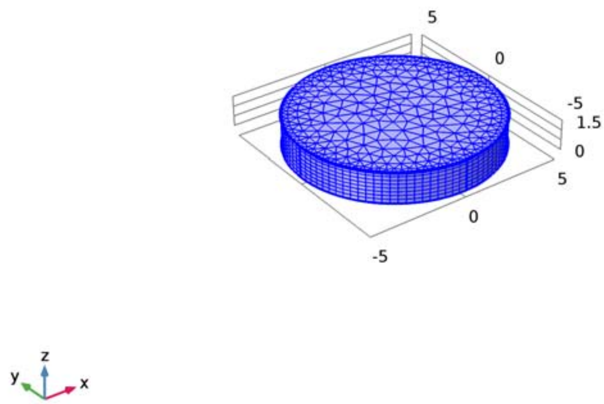
Distribution 1

1 In the **Settings** window for Distribution, locate the **Distribution** section.

2 In the **Number of elements** text field, type `8`.

3 Click **Build Selected**.

The mesh should look like the figure below.



STUDY 1

Next, rename the study node to identify the studies and the related solutions easily.

1 In the **Settings** window for Study, type `Study 1: DOM` in the **Label** text field.

STUDY 1: DOM

Step 1: Time Dependent

The model only compares the results after 10 seconds. To keep the file size small, let COMSOL Multiphysics store only this time step in the file. The computational time step is chosen automatically.

1 In the **Model Builder** window, under **Study 1: DOM** click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

3 In the **Times** text field, type `10`.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
k	5 70 120	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

Next, run the same model but with the P1 method. The only modification to do is to change the **Radiation discretization method** in the **Heat Transfer with Radiation in Participating Media** settings window.

HEAT TRANSFER WITH RADIATION IN PARTICIPATING MEDIA (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer with Radiation in Participating Media (ht)**.

2 In the **Settings** window for Heat Transfer with Radiation in Participating Media, locate the **Participating Media Settings** section.

3 From the **Radiation discretization method** list, choose **P1 approximation**.

To compare the results, a second study is used to compute the same set-up with the P1 method. Add an empty study and copy the settings from the first one.

ADD STUDY

1 On the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Empty Study**.

5 Click **Add Study** in the window toolbar.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for Study, type `Study 2: P1` in the **Label** text field.

3 In the **Model Builder** window under the **Study 1: DOM** node, select both the **Parametric Sweep** and **Step 1: Time Dependent** nodes. Right-click on the selected nodes and choose **Copy**.

STUDY 2: P1

1 In the **Model Builder** window, right-click **Study 2: P1** and choose **Paste Multiple Items**.

2 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht) 1

The same procedure applies for solving with the Rosseland approximation.

HEAT TRANSFER WITH RADIATION IN PARTICIPATING MEDIA (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer with Radiation in Participating Media (ht)**.

2 In the **Settings** window for Heat Transfer with Radiation in Participating Media, locate the **Participating Media Settings** section.

3 From the **Radiation discretization method** list, choose **Rosseland approximation**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

3 Click **Add Study** in the window toolbar.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for Study, type `Study 3: Rosseland` in the **Label** text field.

3 In the **Model Builder** window under the **Study1: DOM** node, select both the **Parametric Sweep** and **Step 1: Time Dependent** nodes. Right-click on the selected nodes and choose **Copy**.

STUDY 3: ROSSELAND

1 In the **Model Builder** window, right-click **Study 3: Rosseland** and choose **Paste Multiple Items**.

2 On the **Study** toolbar, click **Compute**.

RESULTS

Data Sets

To compare the results, add a temperature plot along the center line of the glass plate. Therefore create a cut line data set for each parametric solution.

Cut Line 3D 1

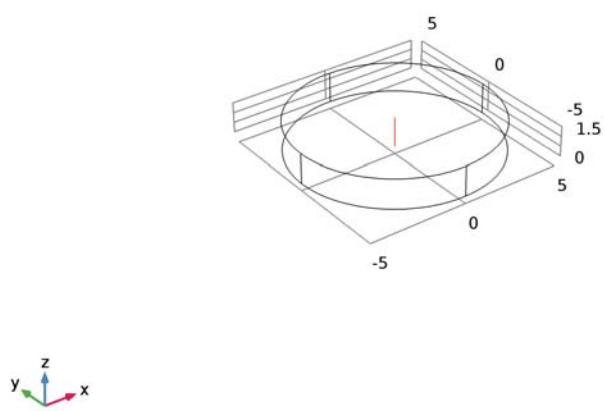
1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for Cut Line 3D, locate the **Data** section.

3 From the **Data set** list, choose **Study 1: DOM/Parametric Solutions 1 (sol2)**.

4 Locate the **Line Data** section. In row **Point 2**, set **x** to `0`, and **z** to `1.5`.

5 Click **Plot**.



Copy the data set twice and assign the solutions from the other studies.

Cut Line 3D 2

1 Right-click **Cut Line 3D 1** and choose **Duplicate**.

2 In the **Settings** window for Cut Line 3D, locate the **Data** section.

3 From the **Data set** list, choose **Study 2: P1/Parametric Solutions 2 (sol7)**.

Cut Line 3D 3

1 Right-click **Results>Data Sets>Cut Line 3D 2** and choose **Duplicate**.

2 In the **Settings** window for Cut Line 3D, locate the **Data** section.

3 From the **Data set** list, choose **Study 3: Rosseland/Parametric Solutions 3 (sol12)**.

Now, add a 1D plot group for the temperature.

1D Plot Group 9

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Temperature at Central Line for k = 5` 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 `Temperature at Central Line`.

Line Graph 1

1 On the **Temperature at Central Line for k = 5** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 3D 1**.

4 From the **Parameter selection (k)** list, choose **First**.

5 From the **Time selection** list, choose **Last**.

6 Click to expand the **Legends** section. Click to collapse the **Legends** section. Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

7 Select the **Show legends** check box.

8 In the table, enter the following settings:

Legends
DOM

9 On the **Temperature at Central Line for k = 5** toolbar, click **Plot**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 3D 2**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
P1

Line Graph 3

1 Right-click **Results>Temperature at Central Line for k = 5>Line Graph 2** and choose **Duplicate**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 3D 3**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Rosseland`

5 On the **Temperature at Central Line for k = 5** toolbar, click **Plot**.

Temperature at Central Line for k = 5

Duplicate the plot group and change the data set to $k = 120$.

1 In the **Model Builder** window, under **Results** right-click **Temperature at Central Line for k = 5** and choose **Duplicate**.

Temperature at Central Line for k = 5.1

In the **Settings** window for 1D Plot Group, type `Temperature at Central Line for k = 120` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Temperature at Central Line for k = 120** node, then click **Line Graph 1**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Parameter selection (k)** list, choose **Last**.

Line Graph 2

1 In the **Model Builder** window, under **Results>Temperature at Central Line for k = 120** click **Line Graph 2**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Parameter selection (k)** list, choose **Last**.

Line Graph 3

1 In the **Model Builder** window, under **Results>Temperature at Central Line for k = 120** click **Line Graph 3**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Parameter selection (k)** list, choose **Last**.

4 On the **Temperature at Central Line for k = 120** toolbar, click **Plot**.

The plots are shown in Figure 2 and Figure 3. For high optical thickness, the Rosseland approximation better represents the overall temperature distribution, whereas the P1 approximation is appropriate for lower optical thickness.

Created in COMSOL Multiphysics 5.2a

Heat Conduction in a Finite Slab

Introduction

This simple example covers the heat conduction in a finite slab, modeling how the temperature varies with time. You first set up the problem in COMSOL Multiphysics and then compare it to the analytical solution given in Ref. 1.

In addition, this example also shows how to avoid oscillations due to a jump between initial and boundary conditions by using a smoothed step function.

Model Definition

The model domain is defined between $x = -b$ and $x = b$. The initial temperature is constant, equal to T_0, over the whole domain; see the figure below. At time $t = 0$, the temperature at both boundaries is lowered to T_1.

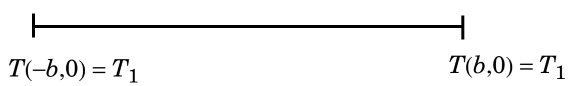


Figure 1: Modeling domain.

To compare the modeling results to the literature (Ref. 1), introduce new dimensionless variables according to the following definitions:

$$\Theta = \frac{T_1 - T}{T_1 - T_0} \qquad \eta = \frac{x}{b} \qquad \tau = \frac{\alpha t}{b^2}$$

The model equation then becomes

$$\frac{\partial \Theta}{\partial \tau} = \frac{\partial^2 \Theta}{\partial \eta^2}$$

with the associated initial condition

$$\tau = 0 \qquad \Theta = 1$$

and boundary conditions

$$\eta = \pm 1 \qquad \Theta = 0$$

The analytical solution of this problem is (see Ref. 1, equation 12.1-31):

$$\Theta = 2\sum_{n=0}^{\infty}\frac{(-1)^n}{\left(n+\frac{1}{2}\right)\pi}\exp\left[-\left(n+\frac{1}{2}\right)^2\pi^2\tau\right]\cos\left(\left(n+\frac{1}{2}\right)\pi\eta\right)$$

To model the temperature decrease at the boundaries use a smoothed step function of time $f(\tau)$.

$$\eta = \pm 1 \qquad \Theta = f(\tau)$$

This method is usually more realistic from a physical point of view than the sudden change in the temperature, and it is also better from a numerical point of view.

Results and Discussion

Figure 2 shows the temperature as a function of position at the dimensionless times $\tau = 0.01, 0.04, 0.1, 0.2, 0.4$, and 0.6. In this plot, the slab's center is situated at $x = 0$ with its end faces located a $x = -1$ and $x = 1$. The temperature profiles shown in the graph are identical to the analytical solution given in Ref. 1.

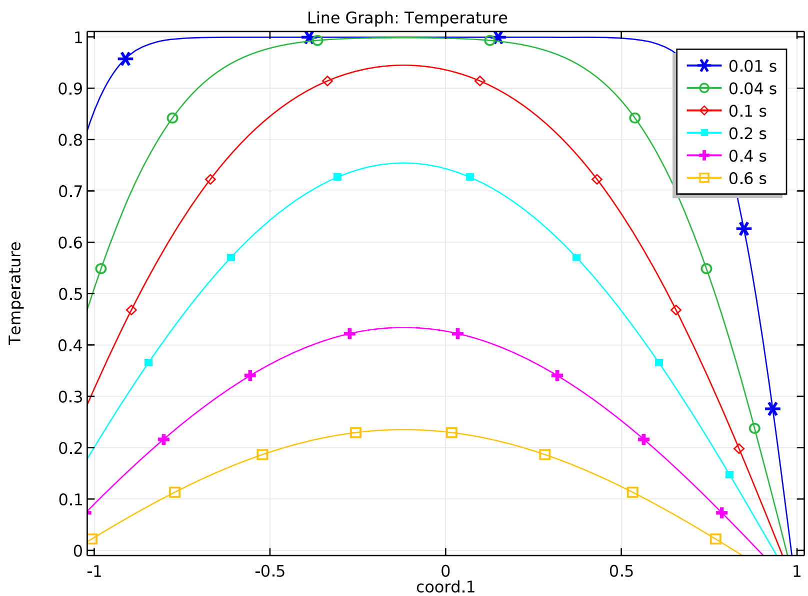


Figure 2: Temperature profiles.

The plot of the L^2 error between the analytical and numerical solutions over time (see Figure 3) confirms this conclusion.

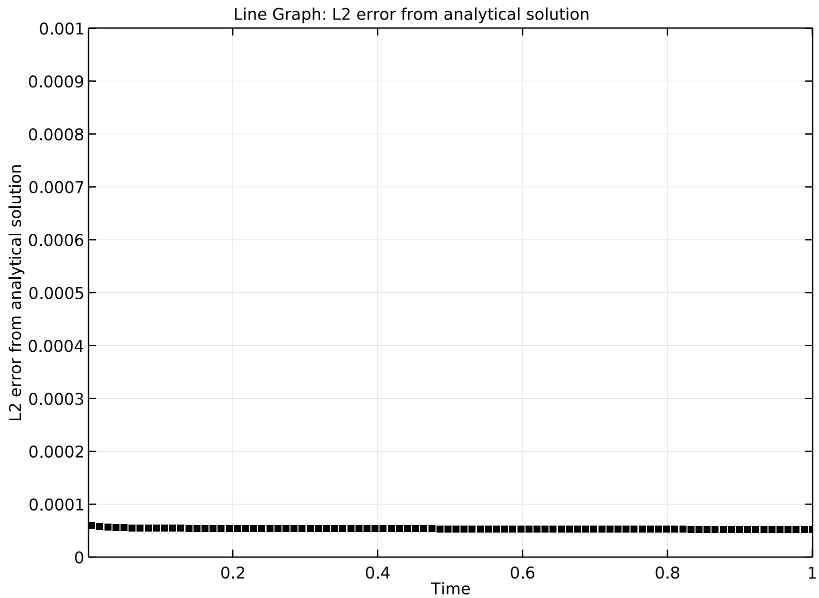


Figure 3: L2 error between analytical and numerical solutions over time.

Reference

1. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, *Transport Phenomena*, 2nd ed., John Wiley & Sons, 2007.

Application Library path: `Heat_Transfer_Module/Tutorials,_Conduction/heat_conduction_in_slab`

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 **1D**.
2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

The Heat Transfer in Solids interface can be used for solving the dimensionless equations. You can switch off the dimensions using the following commands:

COMPONENT 1 (COMP1)

1 In the **Model Builder** window, click **Component 1 (comp1)**.
2 In the **Settings** window for Component, locate the **General** section.
3 From the **Unit system** list, choose **None**.

GEOMETRY 1

Interval 1 (i1)

1 On the **Geometry** toolbar, click **Interval**.
2 In the **Settings** window for Interval, locate the **Interval** section.
3 In the **Left endpoint** text field, type `-1`.
4 Click **Build All Objects**.

DEFINITIONS

Add a step function for use in the boundary conditions.

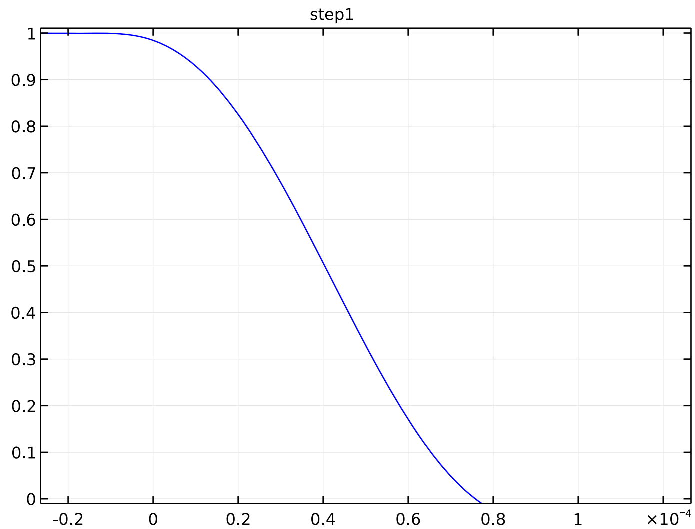
Step 1 (step1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Step**.
2 In the **Settings** window for Step, locate the **Parameters** section.
3 In the **Location** text field, type `5e-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 `1e-4`.

Optionally, you can inspect the shape of the step function.

7 Click **Plot**.



Add an integration operator for the computation of the relative L^2 error between numerical and analytical solutions.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

HEAT TRANSFER IN SOLIDS (HT)

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.

2 In the **Settings** window for Solid, locate the **Heat Conduction, Solid** section.

3 From the k list, choose **User defined**. In the associated text field, type `1`.

4 Locate the **Thermodynamics, Solid** section. From the ρ list, choose **User defined**. In the associated text field, type `1`.

5 From the C_p list, choose **User defined**. In the associated text field, type `1`.

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, type `1` in the T text field.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundaries 1 and 2 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `step1(t)`.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

3 From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for Size, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

Edge 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Edge 1** and choose **Size**.

Size 1

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Boundary**.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `1e-4`.

7 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

2 In the **Times** text field, type `range(0,0.01,1)`.

To make sure that the transition of the boundary temperature from 1 to zero is represented correctly by the transient solver, use the initial time step that is smaller than the transition zone of the step function.

Solution 1 (sol1)

1. On the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
3. In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.
4. Locate the **Time Stepping** section. Select the **Initial step** check box.
5. In the associated text field, type `1e-5`.
6. Select the **Maximum step** check box.
7. In the associated text field, type `1e-3`.
8. On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The default plot shows the temperature distribution along the slab for all time steps. You can compare the computed solution to that of Ref. 1 by plotting the temperature for a given set of output times, as in Figure 2.

1. In the **Model Builder** window, under **Results** click **Temperature (ht)**.
2. In the **Settings** window for 1D Plot Group, locate the **Data** section.
3. From the **Time selection** list, choose **From list**.
4. In the **Times (s)** list, choose **0.01**, **0.04**, **0.1**, **0.2**, **0.4**, and **0.6**.
5. On the **Temperature (ht)** toolbar, click **Plot**.

Line Graph

1. In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Line Graph**.
2. In the **Settings** window for Line Graph, click to expand the **Legends** section.
3. Select the **Show legends** check box.
4. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 On the **Temperature (ht)** toolbar, click **Plot**.

Next plot the relative L^2 error between the numerical and analytical solutions over time.

1D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Relative L2 Error` in the **Label** text field.

Line Graph 1

1 On the **Relative L2 Error** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `sqrt(intop1((T-2*sum((-1)^n/((n+0.5)*pi)*exp(-(n+0.5)^2*pi^2*t)*cos((n+0.5)*pi*x),n,0,1000))^2))/sqrt(intop1(T^2))`.

5 Select the **Description** check box.

6 In the associated text field, type `L2 error from analytical solution`.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `t`.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

10 On the **Relative L2 Error** toolbar, click **Plot**.

As the analytical solution shows oscillations at initial time, change the settings of the graph for a better readability, to get the plot of Figure 3.

Relative L2 Error

1 In the **Model Builder** window, under **Results** click **Relative L2 Error**.

2 In the **Settings** window for 1D Plot Group, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `1e-3`.

5 In the **x maximum** text field, type `1`.

6 In the **y minimum** text field, type `0`.

7 In the **y maximum** text field, type `1e-3`.

8 On the **Relative L2 Error** toolbar, click **Plot**.

Heat Sink

Introduction

This example is intended as a first introduction to simulations of fluid flow and conjugate heat transfer. It shows the following important points:

- How to draw an air box around a device in order to model convective cooling in this box.
- How to set a total heat flux on a boundary using automatic area computation.
- How to display results in an efficient way using selections in data sets.

The application is also described in detail in the book *Introduction to the Heat Transfer Module*. An extension of the application that takes surface-to-surface radiation into account is also available; see Heat Sink with Surface-to-Surface Radiation.

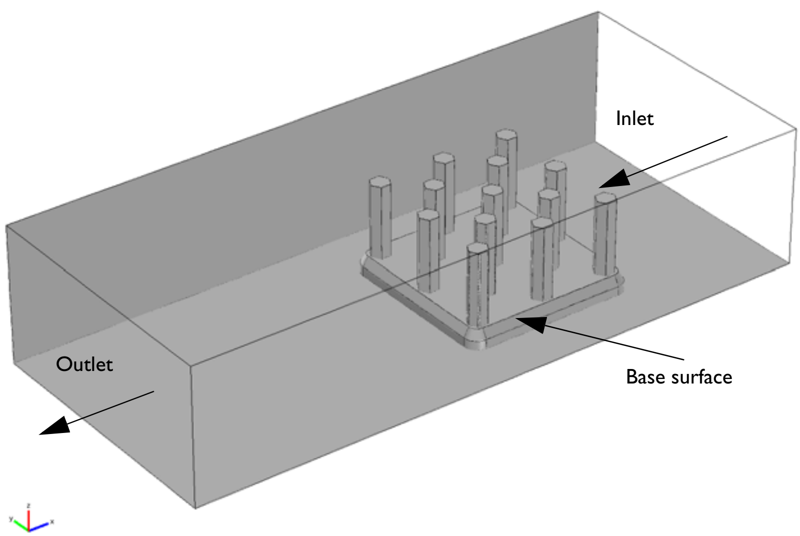


Figure 1: The application setup including channel and heat sink.

Model Definition

The modeled system consists of an aluminum heat sink for cooling of components in electronic circuits mounted inside a channel of rectangular cross section (see Figure 1).

Such a setup is used to measure the cooling capacity of heat sinks. Air enters the channel at the inlet and exits the channel at the outlet. The base surface of the heat sink receives a 1 W heat flux from an external heat source. All other external faces are thermally insulated.

The cooling capacity of the heat sink can be determined by monitoring the temperature of the base surface of the heat sink.

The model solves a thermal balance for the heat sink and the air flowing in the rectangular channel. Thermal energy is transported through conduction in the aluminum heat sink and through conduction and convection in the cooling air. The temperature field is continuous across the internal surfaces between the heat sink and the air in the channel. The temperature is set at the inlet of the channel. The base of the heat sink receives a 1 W heat flux. The transport of thermal energy at the outlet is dominated by convection.

The flow field is obtained by solving one momentum balance for each space coordinate (x, y, and z) and a mass balance. The inlet velocity is defined by a parabolic velocity profile for fully developed laminar flow. At the outlet, the normal stress is equal the outlet pressure and the tangential stress is canceled. At all solid surfaces, the velocity is set to zero in all three spatial directions.

The thermal conductivity of air, the heat capacity of air, and the air density are all temperature-dependent material properties.

You can find all of the settings mentioned above in the predefined multiphysics coupling for Conjugate Heat Transfer in COMSOL Multiphysics. You also find the material properties, including their temperature dependence, in the Material Browser.

Results

In Figure 2, the hot wake behind the heat sink visible in the plot is a sign of the convective cooling effects. The maximum temperature, reached at the heat sink base, is about 380 K.

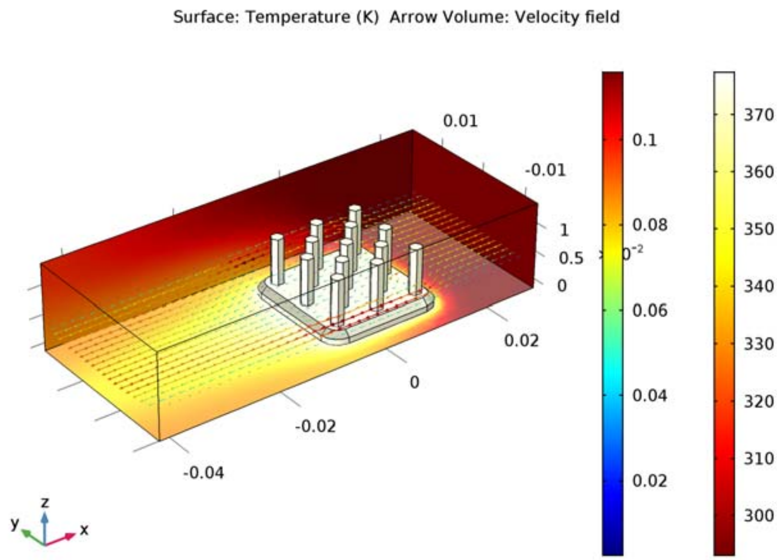


Figure 2: The surface plot shows the temperature field on the channel walls and the heat sink surface, while the arrow plot shows the flow velocity field around the heat sink.

Application Library path: `Heat_Transfer_Module/Tutorials,`
`_Forced_and_Natural_Convection/heat_sink`

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 **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6 Click **Done**.

GEOMETRY 1

The model geometry is available as a parameterized geometry sequence in a separate MPH-file. If you want to build it from scratch, follow the instructions in the section Appendix: Geometry Modeling Instructions. Otherwise load it from file with the following steps.

1 On the **Geometry** toolbar, click **Insert Sequence**.
2 Browse to the application's Application Libraries folder and double-click the file `heat_sink_geom_sequence.mph`.

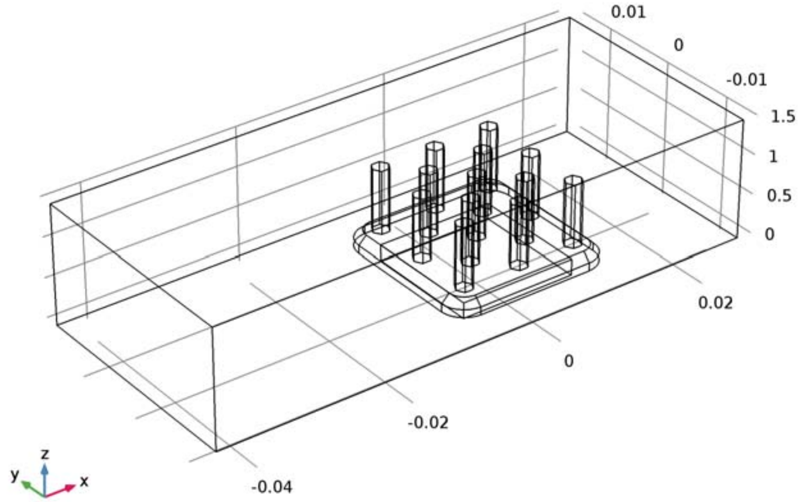
   The application's Application Library folder is shown in the **Application Library path** section immediately before the current section. Note that the path given there is relative to the COMSOL Application Library root, which for a standard installation on Windows is `C:\\Program Files\COMSOL\COMSOL52a\Multiphysics\applications`.

Import 1 (imp1)

1 Click the **Go to Default 3D View** button on the **Graphics** toolbar.
2 On the **Geometry** toolbar, click **Build All**.

   To facilitate face selection in the next steps, use the **Wireframe rendering** option (skip this step if you followed the instructions in the appendix):

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



LAMINAR FLOW (SPF)

Create a selection for the air domain used the physics interfaces to define the fluid.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 Select Domain 1 only.

3 In the **Settings** window for Laminar Flow, locate the **Domain Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Air` in the **Selection name** text field.

6 Click **OK**.

HEAT TRANSFER (HT)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer (ht)**.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Fluid 1**.

2 In the **Settings** window for Fluid, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

MATERIALS

Next, add materials.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Air**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Aluminum 3003-H18**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Aluminum 3003-H18 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum 3003-H18 (mat2)**.

2 Select Domain 2 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Silica glass**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Silica glass (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silica glass (mat3)**.

2 Select Domain 3 only.

3 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Material 4 (mat4)

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 `Thermal Grease` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 34 only.

5 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Thermal Conductivity**.

6 Click **Add to Material**.

7 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`2[W/m/K]`	W/(m·K)	Basic

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`U0`	`5[cm/s]`	0.05 m/s	`Mean inlet velocity`
`P0`	`1[W]`	1 W	`Total power dissipated by the electronics package`

Now define the physical properties of the model. Start with the fluid domain.

LAMINAR FLOW (SPF)

On the **Physics** toolbar, click **Heat Transfer (ht)** and choose **Laminar Flow (spf)**.

The no-slip condition is the default boundary condition for the fluid. Define the inlet and outlet conditions as described below.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 121 only.

3 In the **Settings** window for Inlet, locate the **Boundary Condition** section.

4 From the list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} text field, type `U0`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

3 Select Boundary 1 only.

HEAT TRANSFER (HT)

Thermal insulation is the default boundary condition for the temperature. Define the inlet temperature and the outlet condition as described below.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer (ht)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 121 only.

The ambient temperature is defined in the main node of the Heat Transfer interface. Its default value is 293.15 K which corresponds to the inlet temperature used in this model. It is possible to edit the ambient temperature value or to define it using the meteorological data which gives access to climate data from more than 6000 stations in the world.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 From the T_0 list, choose **Ambient temperature (ht)**.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 1 only.

Next, use the `P0` parameter to define the total heat source in the electronics package.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 3 only.

3 In the **Settings** window for Heat Source, locate the **Heat Source** section.

4 Click the **Heat rate** button.

5 In the P_0 text field, type `P0`.

Finally, add the thin thermal grease layer.

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 Select Boundary 34 only.

3 In the **Settings** window for Thin Layer, locate the **Thin Layer** section.

4 In the d_s text field, type `50[um]`.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extra coarse**.

4 Click **Build All**.

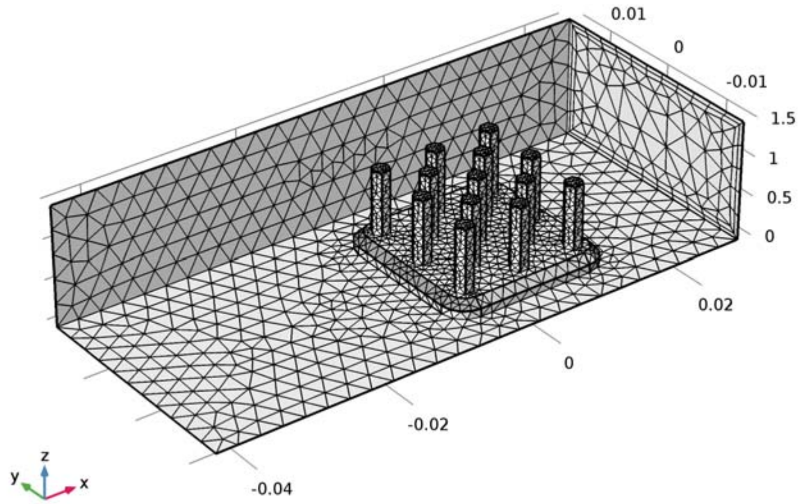
To get a better view of the mesh, hide some of the boundaries.

5 Click the **Click and Hide** button on the **Graphics** toolbar.

6 Click the **Select Boundaries** button on the **Graphics** toolbar.

7 Select Boundaries 1, 2, and 4 only.

The finished mesh should look like that in the figure below.



To achieve more accurate numerical results, this mesh can be refined by choosing another predefined element size. However, doing so requires more computational time and memory.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

Four default plots are generated automatically. The first one shows the temperature on the wall boundaries, the third one shows the velocity magnitude on five parallel slices, and the last one shows the pressure field. Add an arrow plot to visualize the velocity field with temperature field.

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Arrow Volume**.

2 In the **Settings** window for Arrow Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>u,v,w - Velocity field**.

3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `40`.

4 Find the **y grid points** subsection. In the **Points** text field, type `20`.

5 Find the **z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `5[mm]`.

Color Expression 1

1 Right-click **Results>Temperature (ht)>Arrow Volume 1** and choose **Color Expression**.

2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude**.

3 On the **Temperature (ht)** toolbar, click **Plot**.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, type `Energy Balance` in the **Label** text field.

3 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer>Global>Net powers>ht.ntefluxInt - Total net energy rate**.

4 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer>Global>Heat source powers>ht.QInt - Total heat source**.

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

You can verify that the two values match and are close to 1 W in the **Table 1** tab.

Appendix: Geometry Modeling Instructions

ROOT

On the **Home** toolbar, click **Add Component** and choose **3D**.

GLOBAL DEFINITIONS

First define the geometry parameters.

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`L_channel`	`7[cm]`	0.07 m	`Channel length`
`W_channel`	`3[cm]`	0.03 m	`Channel width`
`H_channel`	`1.5[cm]`	0.015 m	`Channel height`
`L_chip`	`1.5[cm]`	0.015 m	`Chip size`
`H_chip`	`2[mm]`	0.002 m	`Chip height`

GEOMETRY 1

Build the geometry in three steps. First, import the heat sink geometry from a file.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

2 In the **Settings** window for Import, locate the **Import** section.

3 Click **Browse**.

4 Browse to the application's Application Libraries folder and double-click the file `heat_sink.mphbin`.

5 Click **Import**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Next, define a work plane containing the bottom surface of the heat sink to draw the imprints of the chip and of the air channel.

Work Plane 1 (wp1)

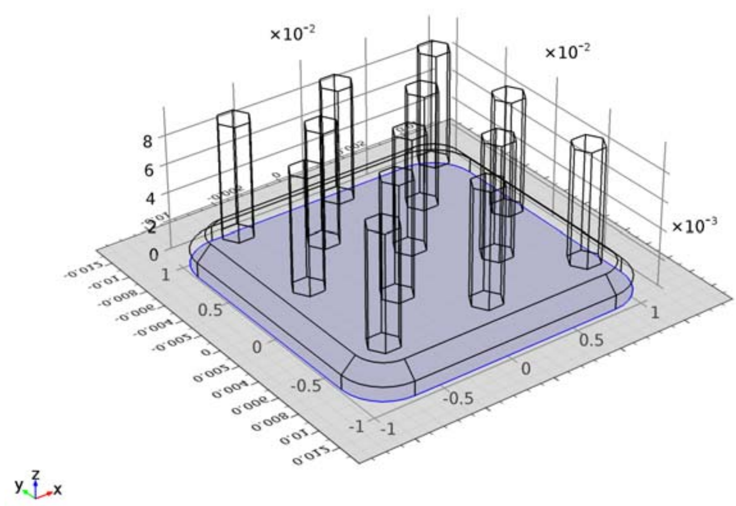
1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Face parallel**.

4 Find the **Planar face** subsection. Select the **Active** toggle button.

5 Select the surface shown in the figure below.



6 Locate the **Unite Objects** section. Clear the **Unite objects** check box.

7 Click **Show Work Plane**.

Square 1 (sq1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for Square, locate the **Size** section.

3 In the **Side length** text field, type `L_chip`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

5 Right-click **Square 1 (sq1)** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `L_channel`.

4 In the **Height** text field, type `W_channel`.

5 Locate the **Position** section. In the **xw** text field, type `-45[mm]`.

6 In the **yw** text field, type `-W_channel/2`.

7 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

8 In the **Model Builder** window, click **Geometry 1**.

Now extrude the chip imprint to define the chip volume.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 Select the object **wp1.sq1** only.

3 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

4 In the table, enter the following settings:

Distances (m)
H_chip

5 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

To finish the geometry, extrude the channel imprint in the opposite direction to define the air volume.

Extrude 2 (ext2)

1 On the **Geometry** toolbar, click **Extrude**.

2 Select the object **wp1.r1** only.

3 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

4 In the table, enter the following settings:

Distances (m)
H_channel

5 Select the **Reverse direction** check box.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

1 On the **Geometry** toolbar, click **Build All**.

The model geometry is now complete.

Created in COMSOL Multiphysics 5.2a

Heat Sink with Surface-to-Surface Radiation

Introduction

This application extends the Heat Sink model by taking surface-to-surface radiation into account. For a detailed description of the application, see Heat Sink. Both applications are also described in detail in the book *Introduction to the Heat Transfer Module*.

Application Library path: `Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/heat_sink_surface_radiation`

Modeling Instructions

ROOT

In this second part you modify and solve the model to study the effects of surface-to-surface radiation between the heat sink and the channel walls.

1. From the **File** menu, choose **Open**.
2. Browse to the application's Application Libraries folder and double-click the file `heat_sink.mph`.

COMPONENT 1 (COMP1)

Now modify the model to include surface-to-surface radiation effects. First you need to enable the surface-to-surface radiation property.

HEAT TRANSFER (HT)

1. In the **Model Builder** window, expand the **Component 1 (comp1)** node, then click **Heat Transfer (ht)**.
2. In the **Settings** window for Heat Transfer, locate the **Physical Model** section.
3. Select the **Surface-to-surface radiation** check box.

   By default, the radiation direction is controlled by the opacity of the domains. The solid parts are automatically defined as opaque while the fluid parts are transparent. You can change these settings by modifying the **Opacity** subnode under **Solid** and **Fluid** features.

   When the **Diffuse Surface** boundary condition defines the radiation direction as opacity controlled (default) the selected boundaries should be located between an opaque and a transparent domain. The exterior is defined as transparent by default. Change this setting to make the exterior opaque and have the radiation direction automatically defined on the channel walls.

4 Locate the **Radiation Settings** section. From the **Exterior radiation** list, choose **Exterior is opaque**.

Now you can add a surface-to-surface boundary condition to the model.

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 Select Boundaries 2–7, 9–30, 36–109, and 111–120 only.

3 In the **Settings** window for Diffuse Surface, locate the **Ambient** section.

4 From the T_{amb} list, choose **Ambient temperature (ht)**.

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.85`.

COMPONENT 1 (COMP1)

Hide the boundaries on the top and fronts to see the interior of the channel and the heat sink.

1 Click the **Click and Hide** button on the **Graphics** toolbar.

2 Click the **Select Boundaries** button on the **Graphics** toolbar.

3 In the **Model Builder** window, click **Component 1 (comp1)**.

4 Select Boundaries 1, 2, and 4 only.

ROOT

In order to keep the previous solution and to be able to compare it with this version of the model, create a new stationary study.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

STUDY 2

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht) 1

The same default plots as before are generated automatically. Modify the temperature plot to compare both cases.

Arrow Volume 1

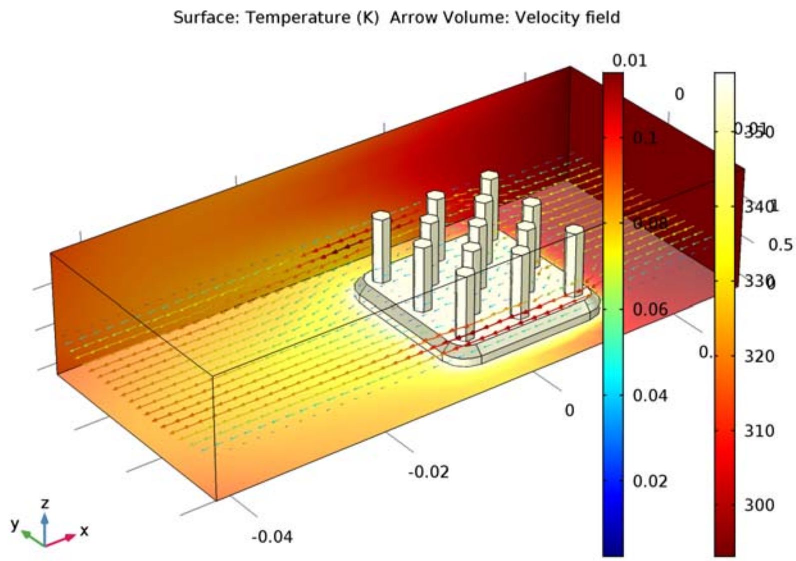
1 In the **Model Builder** window, under **Results** right-click **Temperature (ht) 1** and choose **Arrow Volume**.
2 In the **Settings** window for Arrow Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>u,v,w - Velocity field**.
3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
4 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `40`.
5 Find the **y grid points** subsection. In the **Points** text field, type `20`.
6 Find the **z grid points** subsection. From the **Entry method** list, choose **Coordinates**.
7 In the **Coordinates** text field, type `5[mm]`.

Color Expression 1

1 Right-click **Results>Temperature (ht) 1>Arrow Volume 1** and choose **Color Expression**.
2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude**.

3 On the **Temperature (ht) 1** toolbar, click **Plot**.

The plot in the **Graphics** window should look like that in the figure below.



Created in COMSOL Multiphysics 5.2a

Heating Circuit

Introduction

Small heating circuits find use in many applications. For example, in manufacturing processes they heat up reactive fluids. Figure 1 illustrates a typical heating device for this application. The device consists of an electrically resistive layer deposited on a glass plate. The layer causes Joule heating when a voltage is applied to the circuit. The layer's properties determine the amount of heat produced.

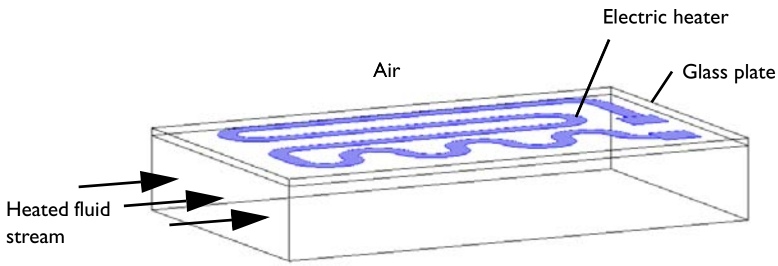


Figure 1: Geometry of a heating device.

In this particular application, you must observe three important design considerations:

- Non-invasive heating
- Minimal deflection of the heating device
- Avoidance of overheating the process fluid

The heater must also work without failure. You achieve the first and second requirements by inserting a glass plate between the heating circuit and the fluid; it acts as a conducting separator. Glass is an ideal material for both these purposes because it is non-reactive and has a low coefficient of thermal expansion.

You must also avoid overheating due to the risk of self-ignition of the reactive fluid stream. Ignition is also the main reason for separating the electrical circuit from direct contact with the fluid. The heating device is tailored for each application, making virtual prototyping very important for manufacturers.

For heating circuits in general, detachment of the resistive layer often determines the failure rate. This is caused by excessive thermally induced interfacial stresses. Once the layer has detached, it gets locally overheated, which accelerates the detachment. Finally, in the worst case, the circuit might overheat and burn. From this perspective, it is also important to study the interfacial tension due to the different thermal-expansion coefficients of the resistive layer and the substrate as well as the differences in temperature.

The geometric shape of the layer is a key parameter to design circuits for proper functioning. You can investigate all of the above-mentioned aspects by modeling the circuit.

This multiphysics example simulates the electrical heat generation, the heat transfer, and the mechanical stresses and deformations of a heating circuit device. The model uses the Heat Transfer in Solids interface of the Heat Transfer Module in combination with the Electric Currents, Shell interface from the AC/DC Module and the Solid Mechanics and Shell interfaces from the Structural Mechanics Module.

Note: This application requires the AC/DC Module, the Heat Transfer Module, and the Structural Mechanics Module.

Model Definition

Figure 2 shows a drawing of the modeled heating circuit.

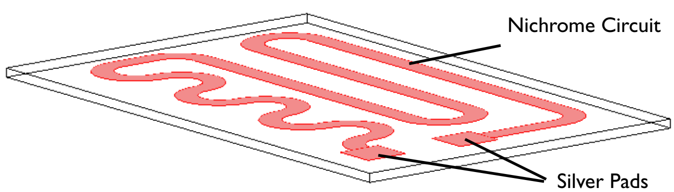


Figure 2: Drawing of the heating circuit deposited on a glass plate.

The device consists of a serpentine-shaped Nichrome resistive layer, 10 μm thick and 5 mm wide, deposited on a glass plate. At each end, it has a silver contact pad measuring 10 mm-by-10 mm-by-10 μm. When the circuit is in use, the deposited side of the glass plate is in contact with surrounding air, and the back side is in contact with the heated fluid. Assume that the edges and sides of the glass plate are thermally insulated.

Table 1 gives the resistor's dimensions.

TABLE 1: DIMENSIONS

OBJECT	LENGTH	WIDTH	THICKNESS
Glass Plate	130 mm	80 mm	2 mm
Pads and Circuit	-	-	10 μm

During operation the resistive layer produces heat. Model the electrically generated heat using the Electric Currents, Shell interface from the AC/DC Module. An electric potential of 12 V is applied to the pads. In the model, you achieve this effect by setting the potential at one edge of the first pad to 12 V and that of one edge of the other pad to 0 V.

To model the heat transfer in the thin conducting layer, use the Thin Layer feature from the Heat Transfer in Solids interface. The heat rate per unit area (measured in W/m^2) produced inside the thin layer is given by

$$q_{\mathrm{prod}} = dQ_{\mathrm{DC}} \tag{1}$$

where $Q_{\mathrm{DC}} = \mathbf{J} \cdot \mathbf{E} = \sigma|\nabla_{\mathbf{t}} V|^2$ (W/m^3) is the power density. The generated heat appears as an inward heat flux at the surface of the glass plate.

At steady state, the resistive layer dissipates the heat it generates in two ways: on its up side to the surrounding air (at 293 K), and on its down side to the glass plate. The glass plate is similarly cooled in two ways: on its circuit side by air, and on its back side by a process fluid (353 K). You model the heat fluxes to the surroundings using heat transfer coefficients, h. For the heat transfer to air, $h = 5\ W/(m^2 \cdot K)$, representing natural convection. On the glass plate's back side, $h = 20\ W/(m^2 \cdot K)$, representing convective heat transfer to the fluid. The sides of the glass plate are insulated.

The model simulates thermal expansion using static structural-mechanics analyses. It uses the Solid Mechanics interface for the glass plate, and the Shell interface for the circuit layer. The equations of these two interfaces are described in the *Structural Mechanics Module User's Guide*. The stresses are set to zero at 293 K. You determine the boundary conditions for the Solid Mechanics interface by fixing one corner with respect to x-, y-, and z-displacements and rotation.

Table 2 summarizes the material properties used in the model.

TABLE 2: MATERIAL PROPERTIES

MATERIAL	E [GPa]	ν	α [1/K]	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]
Silver	83	0.37	1.89e-5	420	10500	230
Nichrome	213	0.33	1e-5	15	9000	20
Glass	73.1	0.17	5.5e-7	1.38	2203	703

Results and Discussion

Figure 3 shows the heat that the resistive layer generates.

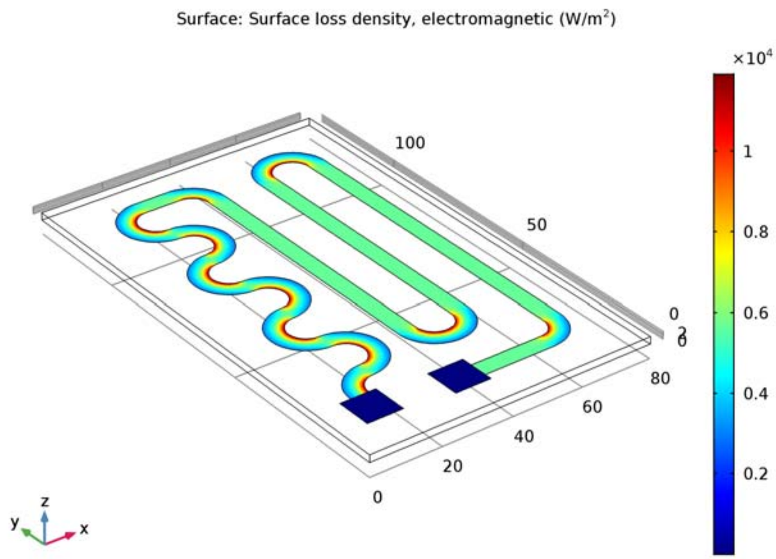


Figure 3: Stationary heat generation in the resistive layer when 12 V is applied.

The highest heating power occurs at the inner corners of the curves due to the higher current density at these spots. The total generated heat, as calculated by integration, is approximately 13.8 W.

Figure 4 shows the temperature of the resistive layer and the glass plate at steady state.

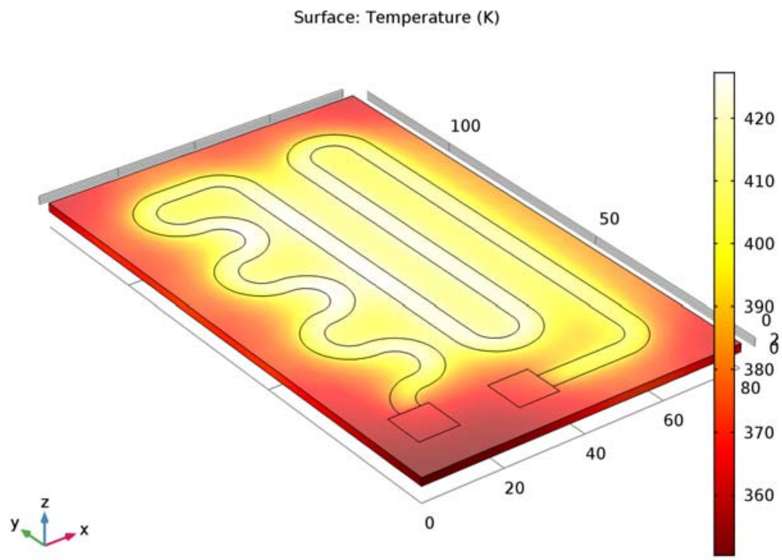


Figure 4: Temperature distribution in the heating device at steady state.

The highest temperature is approximately 428 K, and it appears in the central section of the circuit layer. It is interesting to see that the differences in temperature between the fluid side and the circuit side of the glass plate are quite small because the plate is very thin. Using boundary integration, the integral heat flux on the fluid side evaluates to approximately 8.5 W. This means that the device transfers the majority of the heat it generates—8.5 W out of 13.8 W—to the fluid, which is good from a design perspective, although the thermal resistance of the glass plate results in some losses.

The temperature rise also induces thermal stresses due the materials' different coefficients of thermal expansion. As a result, mechanical stresses and deformations arise in the layer and in the glass plate. Figure 5 shows the effective stress distribution in the device and the resulting deformations. During operation, the glass plate bends towards the air side.

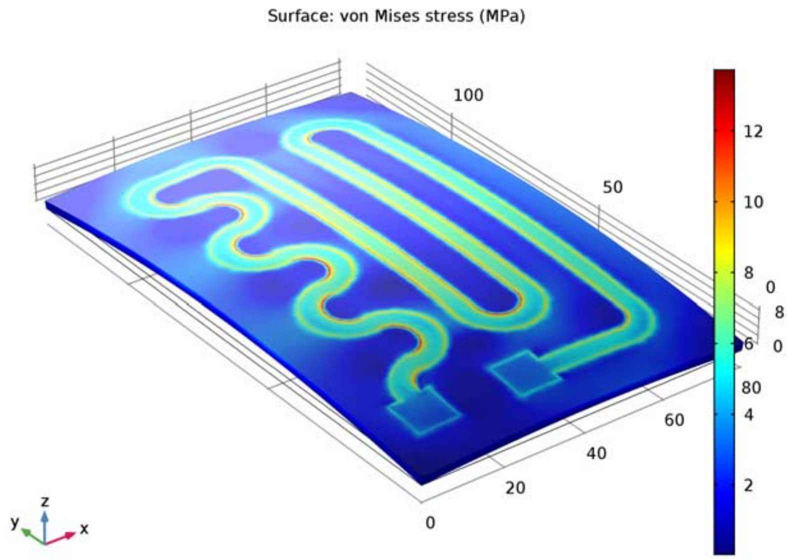


Figure 5: The thermally induced von Mises effective stress plotted with the deformation.

The highest effective stress, approximately 13 MPa, occurs at the inner corners of the curves of the Nichrome circuit. The yield stress for high quality glass is roughly 250 MPa, and for Nichrome it is 360 MPa. This means that the individual objects remain structurally intact for the simulated heating power loads.

You must also consider stresses in the interface between the resistive layer and the glass plate. Assume that the yield stress of the surface adhesion in the interface is in the region of 50 MPa—a value significantly lower than the yield stresses of the other materials in the device. If the effective stress increases above this value, the resistive layer locally detaches from the glass. Once it has detached, heat transfer is locally impeded, which can lead to overheating of the resistive layer and eventually cause the device to fail.

Figure 6 displays the effective forces acting on the adhesive layer during heater operation. As the figure shows, the device experiences a maximum interfacial stress that is an order of magnitude smaller than the yield stress. This means that the device are OK in terms of adhesive stress.

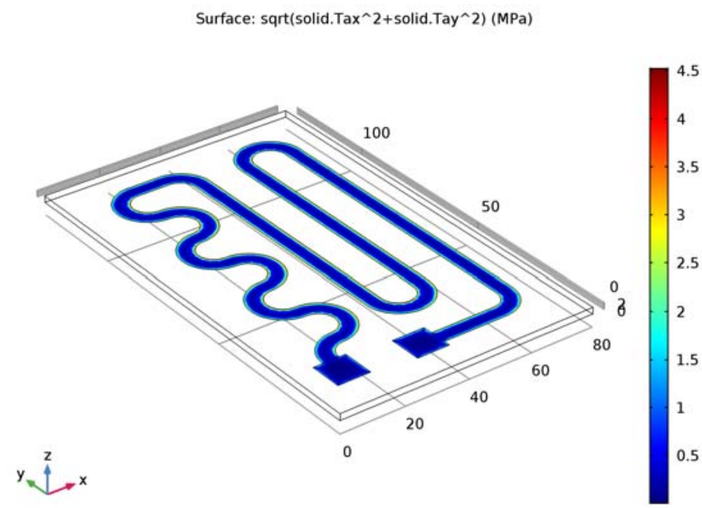


Figure 6: The effective forces in the interface between the resistive layer and the glass plate.

Finally study the device's deflections, shown in Figure 7.

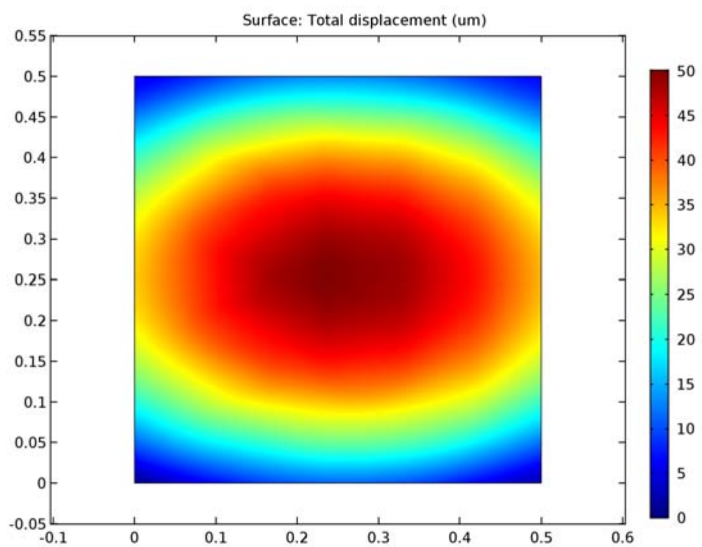


Figure 7: Total displacement on the fluid side of the glass plate.

The maximum displacement, located at the center of the plate, is approximately 50 μm. For high-precision applications, such as semiconductor processing, this might be a significant value that limits the device's operating temperature.

Application Library path: `Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/heating_circuit`

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 **Structural Mechanics>Thermal Stress**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC>Electric Currents, Shell (ecs)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **Structural Mechanics>Membrane (mbrn)**.
7 Click **Add**.
8 Click **Study**.
9 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
10 Click **Done**.

GEOMETRY 1

The **Thermal Stress** interface includes **Heat Transfer in Solids** and **Solid Mechanics**. In the volume, these two interfaces solve for temperature and displacement, respectively. In the shell representing the circuit, the temperature, the electrical potential and displacement are solved by **Heat Transfer In Solids**, **Electric Currents, Shell**, and **Membrane** interfaces, respectively.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
V_in	12[V]	12 V	Input voltage
d_layer	10[um]	1E-5 m	Layer thickness
sigma_silver	6.3e7[S/m]	6.3E7 S/m	Electric conductivity of silver
sigma_nichrome	9.3e5[S/m]	9.3E5 S/m	Electric conductivity of Nichrome
T_air	20[degC]	293.2 K	Air temperature
h_air	5[W/(m^2*K)]	5 W/(m²·K)	Heat transfer film coefficient, air
T_fluid	353[K]	353 K	Fluid temperature
h_fluid	20[W/(m^2*K)]	20 W/(m²·K)	Heat transfer film coefficient, fluid

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type 80.

4 In the **Depth** text field, type 130.

5 In the **Height** text field, type 2.

Work Plane 1 (wp1)

1 Right-click **Block 1 (blk1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Work Plane**.

3 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

4 In the **z-coordinate** text field, type `2`.

5 Click **Show Work Plane**.

Plane Geometry

Click the **Zoom Extents** button on the **Graphics** toolbar.

Square 1 (sq1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for Square, locate the **Size** section.

3 In the **Side length** text field, type `10`.

4 Locate the **Position** section. In the **xw** text field, type `7`.

5 In the **yw** text field, type `10`.

6 Right-click **Square 1 (sq1)** and choose **Build Selected**.

Square 2 (sq2)

1 Right-click **Square 1 (sq1)** and choose **Duplicate**.

2 In the **Settings** window for Square, locate the **Position** section.

3 In the **xw** text field, type `30`.

4 In the **yw** text field, type `8`.

Polygon 1 (pol1)

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Square 2 (sq2)** and choose **Build Selected**.

2 On the **Work Plane** toolbar, click **Primitives** and choose **Polygon**.

3 In the **Settings** window for Polygon, locate the **Coordinates** section.

4 From the **Data source** list, choose **File**.

5 Click **Browse**.

6 Browse to the application's Application Libraries folder and double-click the file `heating_circuit_polygon.txt`.

Fillet 1 (fil1)

1 Right-click **Polygon 1 (pol1)** and choose **Build Selected**.

2 On the **Work Plane** toolbar, click **Fillet**.

3 On the object **pol1**, select Points 2–8, 23–29, 34, 36, 37, 41, and 42 only.

4 In the **Settings** window for Fillet, locate the **Radius** section.

5 In the **Radius** text field, type `10`.

Fillet 2 (fil2)

1 Right-click **Fillet 1 (fil1)** and choose **Build Selected**.

2 On the **Work Plane** toolbar, click **Fillet**.

3 On the object **fil1**, select Points 6–12, 26–31, 37, 40, 43, 46, 49, and 50 only.

4 In the **Settings** window for Fillet, locate the **Radius** section.

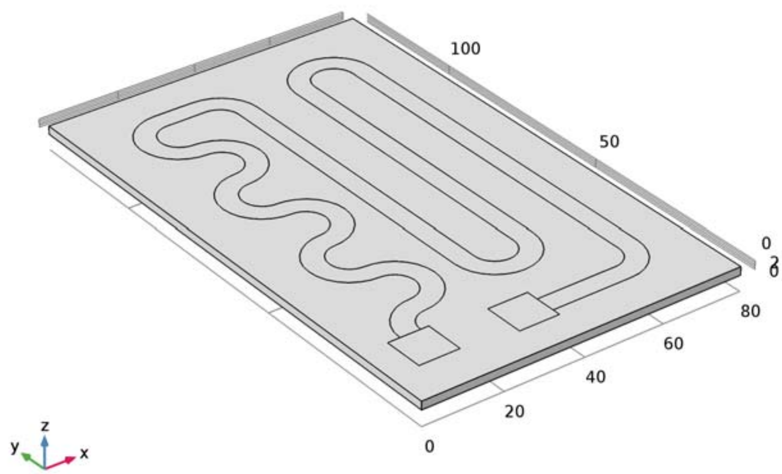
5 In the **Radius** text field, type `5`.

6 On the **Work Plane** toolbar, click **Build All**.

Form Union (fin)

1 On the **Home** toolbar, click **Build All**.

The geometry should look like the figure below.



DEFINITIONS

Add a selection that you can use later when applying boundary conditions and shell physics settings.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Circuit` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 6–8 only.

Before creating the materials for use in this model, it is a good idea to specify which boundaries are to be modeled as conducting shells. Using this information, COMSOL Multiphysics can detect which material properties are needed.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for Thin Layer, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Circuit**.

4 Locate the **Thin Layer** section. From the **Layer type** list, choose **Thermally thin approximation**.

5 In the d_s text field, type `d_layer`.

ELECTRIC CURRENTS, SHELL (ECS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents, Shell (ecs)**.

2 In the **Settings** window for Electric Currents, Shell, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Circuit**.

4 Locate the **Shell Thickness** section. In the d_s text field, type `d_layer`.

MEMBRANE (MBRN)

On the **Physics** toolbar, click **Electric Currents, Shell (ecs)** and choose **Membrane (mbrn)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Membrane (mbrn)**.

2 In the **Settings** window for Membrane, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Circuit**.

4 Locate the **Thickness** section. In the d text field, type `d_layer`.

5 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Displacement field** text field, type `u`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Membrane (mbrn)** click **Linear Elastic Material 1**.

Thermal Expansion 1

1 On the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.
2 In the **Settings** window for Thermal Expansion, locate the **Model Inputs** section.
3 From the T list, choose **Temperature (ht)**.

ADD MATERIAL

1 On 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>Silica glass**.
4 Click **Add to Component** in the window toolbar.
5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Silica glass (mat1)

Now set up the materials.

Material 2 (mat2)

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 `Silver` in the **Label** text field.
3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Circuit**.
5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	420	W/(m·K)	Basic
Density	rho	10500	kg/m³	Basic
Heat capacity at constant pressure	Cp	230	J/(kg·K)	Basic
Electrical conductivity	sigma	`sigma_silver`	S/m	Basic

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	`1`	1	Basic
Young's modulus	E	`83e9`	Pa	Basic
Poisson's ratio	nu	`0.37`	1	Basic
Coefficient of thermal expansion	alpha	`18.9e-6`	1/K	Basic

Material 3 (mat3)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for Material, type `Nichrome` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 7 only.
5. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`15`	W/(m·K)	Basic
Density	rho	`9000`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`20`	J/(kg·K)	Basic
Electrical conductivity	sigma	`sigma_nichrome`	S/m	Basic
Relative permittivity	epsilonr	`1`	1	Basic
Young's modulus	E	`213e9`	Pa	Basic
Poisson's ratio	nu	`0.33`	1	Basic
Coefficient of thermal expansion	alpha	`10e-6`	1/K	Basic

With the materials defined, set up the remaining physics of the model. In the next section, the resistive loss within the circuit is defined as a heat source for the thermal stress physics. The resistive loss is calculated automatically within the **Electric Currents, Shell** physics interface. Add the coupling feature **Boundary Electromagnetic Heat Source** to take the resistive loss into account.

MULTIPHYSICS

Boundary Electromagnetic Heat Source 1 (bemh1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Boundary>Boundary Electromagnetic Heat Source**.
2 In the **Settings** window for Boundary Electromagnetic Heat Source, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Circuit**.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundaries 4 and 6–8 only.
3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.
4 Click the **Convective heat flux** button.
5 In the h text field, type `h_air`.
6 In the T_{ext} text field, type `T_air`.

Heat Flux 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundary 3 only.
3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.
4 Click the **Convective heat flux** button.
5 In the h text field, type `h_fluid`.
6 In the T_{ext} text field, type `T_fluid`.

Next, add constraints to restrain the glass plate movements.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.
2 Select Point 1 only.

Prescribed Displacement 1

1. On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.
2. Select Point 63 only.
3. In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.
4. Select the **Prescribed in y direction** check box.
5. Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

1. On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.
2. Select Point 3 only.
3. In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.
4. Select the **Prescribed in z direction** check box.

Finally, add a voltage and ground.

ELECTRIC CURRENTS, SHELL (ECS)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents, Shell (ecs)**.

Electric Potential 1

1. On the **Physics** toolbar, click **Edges** and choose **Electric Potential**.
2. Select Edge 10 only.
3. In the **Settings** window for Electric Potential, locate the **Electric Potential** section.
4. In the V_0 text field, type `V_in`.

Ground 1

1. On the **Physics** toolbar, click **Edges** and choose **Ground**.
2. Select Edge 43 only.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

Free Triangular 1

Select Boundaries 4 and 6–8 only.

Size 1

1. Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.

2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Circuit**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `2`.

7 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 1** and choose **Distribution**.

Distribution 1

1 In the **Settings** window for Distribution, locate the **Distribution** section.

2 In the **Number of elements** text field, type `3`.

3 Click **Build All**.

STUDY 1

In order to improve the solver's performance, set the segregated solver to calculate temperature, voltage and displacement separately. The best order is `V`, `T`, `u`.

Solution 1 (sol1)

1 On 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)>Stationary Solver 1>Segregated 1** node, then click **Segregated Step 2**.

4 In the **Settings** window for Segregated Step, type `Electric potential V` in the **Label** text field.

5 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1> Segregated 1>Electric potential V** and choose **Move Up**.

6 Locate the **General** section. In the **Variables** list, select **Displacement field (Material) (comp1.u)**.

7 Under **Variables**, click **Delete**.

8 In the **Variables** list, select **Normal strain (comp1.mbrn.unn)**.

9 Under **Variables**, click **Delete**.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 1>Segregated 1** click **Segregated Step 3**.

11 In the **Settings** window for Segregated Step, type `Displacement u` in the **Label** text field.

12 Locate the **General** section. Under **Variables**, click **Add**.

13 In the **Add** dialog box, select **Normal strain (comp1.mbrn.unn)** in the **Variables** list.

14 Click **OK**.

15 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

The default plots show the von Mises stress including the deformation (Figure 5) and the temperature (Figure 4) on the surface of the full 3D geometry, and the electric potential and the von Mises stress on the circuit layer.

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 On the **Stress (solid)** toolbar, click **Plot**.

Surface

1 In the **Model Builder** window, expand the **Results>Stress (mbrn)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 On the **Stress (mbrn)** toolbar, click **Plot**.

Study 1/Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Results>Data Sets** node.

2 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Circuit**.

To generate Figure 3 follow the steps below.

3D Plot Group 6

1 On the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for 3D Plot Group, type `Surface Losses` in the **Label** text field.
3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

1 Right-click **Surface Losses** and choose **Surface**.
2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electric Currents, Shell>Heating and losses>ecs.Qsh - Surface loss density, electromagnetic**.
3 On the **Surface Losses** toolbar, click **Plot**.
4 Click the **Scene Light** button on the **Graphics** toolbar.
5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The following steps generate a plot of the norm of the surface traction vector in the surface plane (see Figure 6):

3D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for 3D Plot Group, type `Interface Stress` in the **Label** text field.
3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

1 Right-click **Interface Stress** and choose **Surface**.
2 In the **Settings** window for Surface, locate the **Expression** section.
3 In the **Expression** text field, type `sqrt(solid.Tax^2+solid.Tay^2)`.
4 From the **Unit** list, choose **MPa**.
5 On the **Interface Stress** toolbar, click **Plot**.

Finally, to obtain Figure 7, proceed as follows:

Surface 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Surface**.
2 Select Boundary 3 only.

2D Plot Group 8

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Displacement, Bottom Boundary` in the **Label** text field.

Surface 1

1 Right-click **Displacement, Bottom Boundary** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>solid.disp - Total displacement**.

3 Locate the **Expression** section. In the **Unit** field, type `um`.

4 On the **Displacement, Bottom Boundary** toolbar, click **Plot**.

Derived Values

To calculate the values for the total generated heat and the integrated heat flux on the fluid side, perform a boundary integration:

Surface Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 Select Boundary 3 only.

3 In the **Settings** window for Surface Integration, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Heat Transfer in Solids>Boundary fluxes>ht.q0 - Inward heat flux**.

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result should be close to 8.5 W.

RESULTS

Surface Integration 2

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 In the **Settings** window for Surface Integration, locate the **Selection** section.

3 From the **Selection** list, choose **Circuit**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Electric Currents, Shell>Heating and losses>ecs.Qsh - Surface loss density, electromagnetic**.

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result should be close to 13.8 W.

Created in COMSOL Multiphysics 5.2a

Free Convection in a Light Bulb

Introduction

This application simulates the non-isothermal flow of argon gas inside a light bulb. The purpose of the model is to show the coupling between energy transport—through conduction, radiation, and convection—and momentum transport induced by density variations in the argon gas.

All three forms of heat transfer are taken into account. First, you have conduction, when a 60 W filament is heated thus transferring heat from the heat source to the light bulb. Then there is convection, which drives a flow inside the bulb transferring the heat from the filament throughout the bulb via the movement of fluids (in this case, argon gas). Finally, there is the radiation portion of the problem, and in this case that includes surface-to-surface and surface-to-ambient radiation. The Heat Transfer Module includes both of these types of radiation, so that you can account for shading and reflections between radiating surfaces, as well as ambient radiation that can be fixed or given by an arbitrary function. The light bulb physics involves both heat transfer and gas flow, which makes this a multiphysics problem and not "just" a heat transfer example.

Note: This application requires the Heat Transfer Module and the Material Library.

Model Definition

A light bulb contains a tungsten filament that is resistively heated when a current is conducted through it. At temperatures around 2000 K the filament starts to emit visible light. To prevent the tungsten wire from burning up, the bulb is filled with a gas, usually argon. The heat generated in the filament is transported to the surroundings through radiation, convection, and conduction. As the gas heats up, density and pressure changes induce a flow inside the bulb.

Figure 1 shows a cross section of the axially symmetric model geometry.

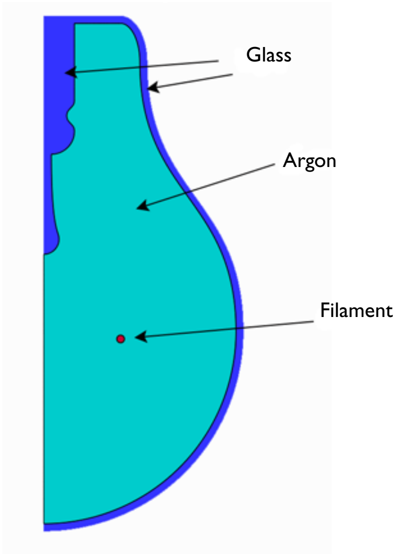


Figure 1: The model geometry.

The filament is approximated with a solid torus, an approximation that implies neglecting any internal effects inside the filament wire.

The equations governing the non-isothermal flow are the Navier-Stokes equation with the gravity forces (see Gravity in the *CFD Module User's Guide*). The density is given by the ideal gas law

$$\rho = \frac{Mp}{RT}$$

where M denotes the molar weight (kg/mol), R the universal gas constant (J/(mol·K)), and T the temperature (K).

The convective and conductive heat transfer are modeled using the heat transfer interface and account for the total light bulb power equal to 60 W.

BOUNDARY CONDITIONS

At the bulb's inner surfaces, radiation is described by surface-to-surface radiation. This means that the mutual irradiation from the surfaces that can be seen from a particular surface and radiation to the surroundings are accounted for. At the outer surfaces of the bulb, radiation is described by surface-to-ambient radiation, which means that there is no reflected radiation from the surroundings (blackbody radiation).

The top part of the bulb, where the bulb is mounted on the cap, is insulated:

$$-\mathbf{n} \cdot (-k\nabla T) = 0$$

Results

The heating inside the bulb has a long and a short time scale from $t = 0$, when the light is turned on. The shorter scale captures the heating of the filament and the gas close to it. The following series of pictures shows the temperature distribution inside the bulb at $t = 2$, 6, and 10 s.

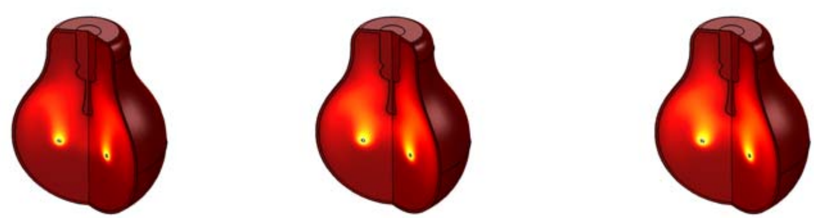


Figure 2: Temperature distribution at t = 2, 6, and 10 s. The color ranges differ between the plots.

When the temperature changes, the density of the gas changes, inducing a gas flow inside the bulb. The following series of pictures shows the velocity field inside the bulb after 2, 6, and 10 s.

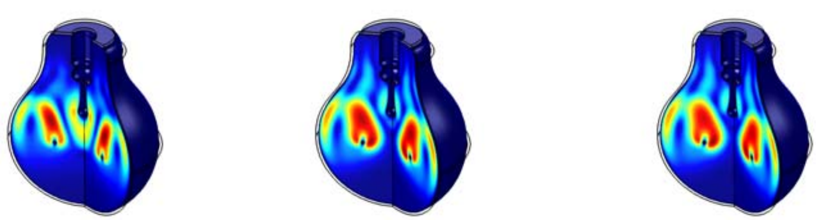


Figure 3: Velocity field after 2, 6, and 10 s. The color ranges differ between the plots.

On the longer time scale, the glass on the bulb's outer side heats up. The following plot shows the temperature distribution in the bulb after 5 minutes.

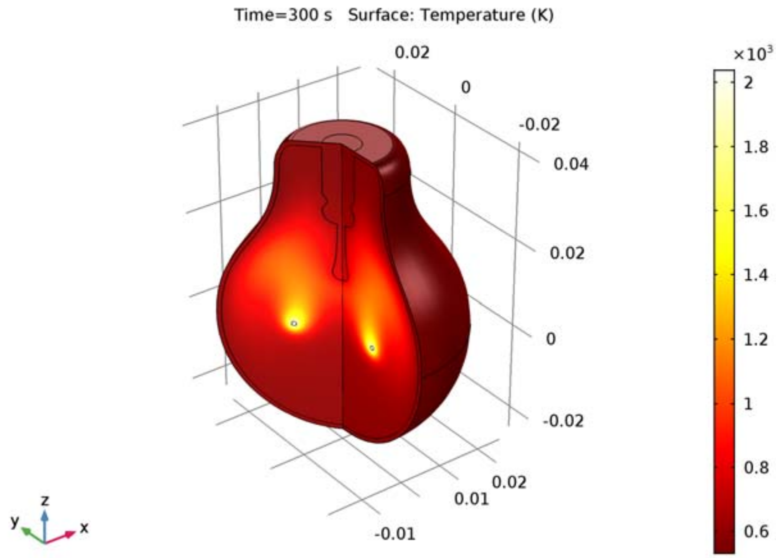


Figure 4: Temperature distribution after 5 minutes.

Figure 5 shows the temperature distribution at a point on the boundary of the bulb at the same vertical level as the filament. This plot shows the slow heating of the bulb. After 5 minutes, the bulb has reached a steady-state temperature of 580 K.

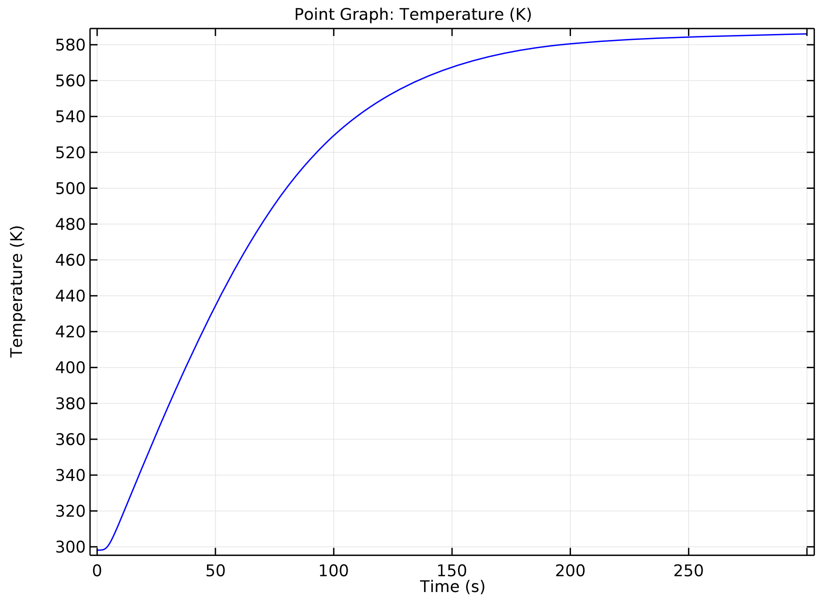


Figure 5: Temperature distribution at a point on the boundary of the bulb at the same vertical level as the filament.

Heat is transported from the boundary of the bulb through both convective heat flux and radiation. The net radiative heat flux leaving the bulb at $t = 300$ s is plotted in Figure 6, as function of the z-coordinate. The top boundaries of the bulb where the bulb is mounted on the cap are excluded from this plot. The distinct bump in the curve occurs at the same vertical level as the filament.

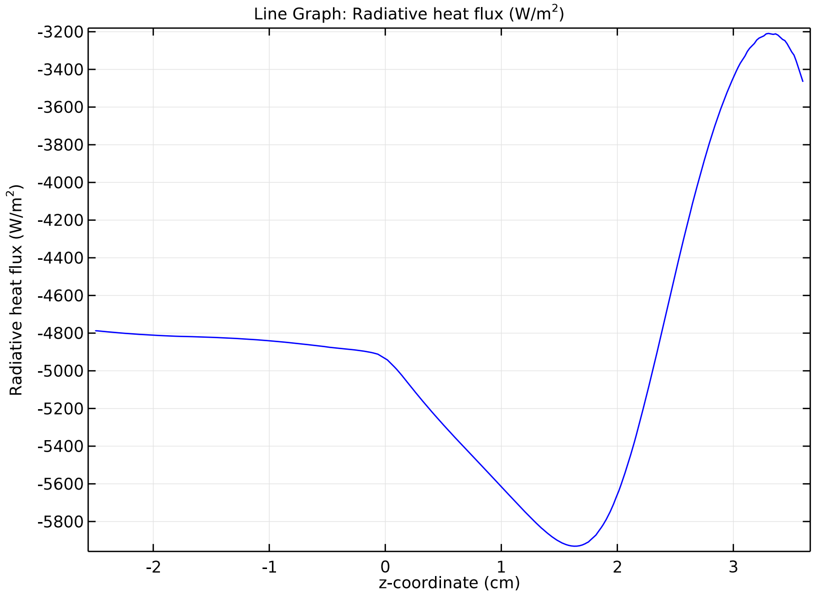


Figure 6: The net radiative heat flux leaving the bulb.

Notes About the COMSOL Implementation

To set up the model, use the Heat Transfer Module's Conjugate Heat Transfer predefined multiphysics coupling. The model uses material from the Material Library to accurately account for temperature-dependent properties over a wide range. The model setup is straightforward and also shows how to create your own material to treat argon as an ideal gas. When working with surface-to-surface radiation in COMSOL, fluid domains are considered as transparent and solid domains as opaque by default, which are the expected properties for this model. The assumption that the glass on the bulb is opaque might seem odd, but it is valid because glass is almost opaque to heat radiation but transparent to radiation in the visible spectrum.

Application Library path: `Heat_Transfer_Module/Thermal_Radiation/light_bulb`

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 **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.
2 In the **Settings** window for Import, locate the **Import** section.
3 Click **Browse**.
4 Browse to the application's Application Libraries folder and double-click the file `light_bulb.mphbin`.
5 Click **Import**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
h0	5[W/(m^2*K)]	5 W/(m²·K)	Heat transfer coefficient
Qf	60[W]	60 W	Heat source in filament

Name	Expression	Value	Description
p0	50[kPa]	5E4 Pa	Initial pressure
rho_glass	2595[kg/m^3]	2595 kg/m³	Density, glass
k_glass	1.09[W/(m*K)]	1.09 W/(m·K)	Thermal conductivity, glass
Cp_glass	750[J/(kg*K)]	750 J/(kg·K)	Heat capacity, glass
eps_glass	0.8	0.8	Surface emissivity, glass
Mw_a	39.94[g/mol]	0.03994 kg/mol	Molar mass, argon

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Material Library>Elements>Tungsten>Tungsten [solid]>Tungsten [solid, Ho et al]**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Tungsten [solid,Ho et al] (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Tungsten [solid,Ho et al] (mat1)**.

2 Select Domain 3 only.

To apply the surface emissivity for tungsten as a material property, you also need to define tungsten as the material for the filament surface.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Material Library>Elements>Tungsten>Tungsten [solid]>Tungsten [solid, Ho et al]**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Tungsten [solid,Ho et al] 1 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Tungsten [solid,Ho et al] 1 (mat2)**.
2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundaries 15–18 only.

Material 3 (mat3)

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 `Glass` in the **Label** text field.
3 Select Domain 1 only.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`k_glass`	W/(m·K)	Basic
Density	rho	`rho_glass`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`Cp_glass`	J/(kg·K)	Basic

You will return to the Materials branch shortly to specify glass at the boundaries of the glass domain and argon as the material for the interior of the bulb. However, first set up the physics to let COMSOL Multiphysics flag what properties you need to specify manually.

LAMINAR FLOW (SPF)

Since the density variation is not small, the flow can not be regarded as incompressible. Therefore set the flow to be compressible.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2 In the **Settings** window for Laminar Flow, locate the **Physical Model** section.
3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.
4 Select Domain 2 only.

Define the pressure reference level in the interface properties.

5 Find the **Reference values** subsection. In the p_{ref} text field, type `p0`.

6 In the **Settings** window for Laminar Flow, locate the **Domain Selection** section.

7 Click **Create Selection**.

8 In the **Create Selection** dialog box, type `Argon` in the **Selection name** text field.

9 Click **OK**.

HEAT TRANSFER (HT)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer (ht)**.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Fluid 1**.

2 In the **Settings** window for Fluid, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Argon**.

4 In the **Model Builder** window, click **Heat Transfer (ht)**.

5 In the **Settings** window for Heat Transfer, locate the **Physical Model** section.

6 Select the **Surface-to-surface radiation** check box.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `25[degC]` in the T text field.

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 Select Boundaries 8, 9, 12–19, and 21–24 only.

3 In the **Settings** window for Diffuse Surface, locate the **Ambient** section.

4 In the T_{amb} text field, type `T`.

   By default, the radiation direction is controlled by the opacity of the domains. The solid parts are automatically defined as opaque while the fluid parts are transparent. You can change these settings by modifying the **Opacity** subnode under the **Solid** and **Fluid** features. For this model, the default settings apply.

Diffuse Surface 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 Select Boundaries 11 and 25 only.

3 In the **Settings** window for Diffuse Surface, locate the **Ambient** section.

4 In the T_{amb} text field, type `25[degC]`.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 3 only.

3 In the **Settings** window for Heat Source, locate the **Heat Source** section.

4 Click the **Heat rate** button.

5 In the P_0 text field, type `Qf`.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 11 and 25 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 In the h text field, type `h0`.

6 In the T_{ext} text field, type `25[degC]`.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for Laminar Flow, locate the **Physical Model** section.

3 Select the **Include gravity** check box.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 4 (mat4)

1 In the **Settings** window for Material, type `Glass (Boundaries)` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

3 Select Boundaries 7–14 and 19–25 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Surface emissivity	epsilon_rad	`eps_glass`	1	Basic

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Material Library>Elements>Argon>Argon [gas]**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Argon [gas] (mat5)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Argon [gas] (mat5)**.

2 Select Domain 2 only.

As you can see, COMSOL Multiphysics warns about required properties that have not been defined yet. Define these as follows.

3 In the **Settings** window for Material, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Ratio of specific heats	gamma	`1.6`	1	Basic
Density	rho	`ht.pA*Mw_a/(R_const*T)`	kg/m³	Basic

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,0.1,1) range(1.5,0.5,20) range(21,3,300)`.

4 Select the **Relative tolerance** check box.

5 In the associated text field, type `0.001`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Dependent Variables 1**.

4 In the **Settings** window for Dependent Variables, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

For a nonlinear time-dependent problem such as this one, a judicious manual scaling of the variables can improve convergence during solving.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Velocity field (comp1.u)**.

7 In the **Settings** window for Field, locate the **Scaling** section.

8 From the **Method** list, choose **Manual**.

9 In the **Scale** text field, type `0.1`.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Surface radiosity (comp1.ht.J)**.

11 In the **Settings** window for Field, locate the **Scaling** section.

12 From the **Method** list, choose **Manual**.

13 In the **Scale** text field, type `1e5`.

14 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Temperature (comp1.T)**.

15 In the **Settings** window for Field, locate the **Scaling** section.

16 From the **Method** list, choose **Manual**.

17 In the **Scale** text field, type `1e3`.

18 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Pressure (comp1.p)**.

19 In the **Settings** window for Field, locate the **Scaling** section.

20 From the **Method** list, choose **Manual**.

21 In the **Scale** text field, type `1e4`.

22 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.

23 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

24 Locate the **Time Stepping** section. From the **Method** list, choose **Generalized alpha**.

25 From the **Steps taken by solver** list, choose **Manual**.

26 In the **Time step** text field, type `0.02*if(t<10,1,10)`.

27 Click to expand the **Advanced** section. On the **Study** toolbar, click **Compute**.

RESULTS

Temperature, 3D (ht)

The first default 3D plot shows the temperature at the end of the simulation interval (Figure 4). Look at the temperature field at different times and compare the resulting series of plots with those in Figure 2.

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

2 In the **Model Builder** window, under **Results** click **Temperature, 3D (ht)**.

3 In the **Settings** window for 3D Plot Group, locate the **Data** section.

4 From the **Time (s)** list, choose **2**.

5 On the **Temperature, 3D (ht)** toolbar, click **Plot**.

Compare with the left panel in Figure 2.

6 From the **Time (s)** list, choose **6**.

7 On the **Temperature, 3D (ht)** toolbar, click **Plot**.

Compare with the middle panel in Figure 2.

8 From the **Time (s)** list, choose **10**.

9 On the **Temperature, 3D (ht)** toolbar, click **Plot**.

Compare with the right panel in Figure 2.

Pressure (spf)

This default plot shows the pressure field in a 2D contour plot. Change the unit to kPa as follows.

1 In the **Model Builder** window, expand the **Pressure (spf)** node, then click **Contour**.

2 In the **Settings** window for Contour, locate the **Expression** section.

3 From the **Unit** list, choose **kPa**.

4 On the **Pressure (spf)** toolbar, click **Plot**.

Velocity (spf) 1

This default plot shows the velocity magnitude in a 3D plot, obtained by a revolution of the 2D axisymmetric data set, at the end of the simulation interval. Now proceed to reproduce the velocity field plots in Figure 3.

Because the velocity magnitude is a quadratic expression in the basic velocity variables it looks less smooth than the temperature plot. You can easily remedy the situation by adjusting the Quality settings.

1 In the **Model Builder** window, expand the **Velocity (spf) 1** node, then click **Surface**.

2 In the **Settings** window for Surface, click to expand the **Quality** section.

3 From the **Resolution** list, choose **Fine**.

4 On the **Velocity (spf) 1** toolbar, click **Plot**. This ensures that the resolution is sufficient.

5 In the **Model Builder** window, click **Velocity (spf) 1**.

6 In the **Settings** window for 3D Plot Group, locate the **Data** section.

7 From the **Time (s)** list, choose **2**.

8 On the **Velocity (spf) 1** toolbar, click **Plot**.

Compare with the left panel in Figure 3.

9 From the **Time (s)** list, choose **6**.

10 On the **Velocity (spf) 1** toolbar, click **Plot**.

Compare with the middle panel in Figure 3.

11 From the **Time (s)** list, choose **10**.

12 On the **Velocity (spf) 1** toolbar, click **Plot**.

Compare with the right panel in Figure 3.

1D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

To visualize the heating of the bulbs surface with time by plotting the temperature at a point at the same vertical level as the filament, follow the steps below.

2 In the **Settings** window for 1D Plot Group, type `Temperature vs Time` in the **Label** text field.

Point Graph 1

On the **Temperature vs Time** toolbar, click **Point Graph**.

Temperature vs Time

1 Select Point 24 only.

2. In the **Settings** window for Point Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Heat Transfer>Temperature>T - Temperature**.
3. On the **Temperature vs Time** toolbar, click **Plot**.

Finally, study the radiative heat flux from the bulb. First plot the radiative heat flux versus the vertical coordinate, z.

1D Plot Group 8

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type `Radiative Heat Flux along z-Coordinate` in the **Label** text field.
3. Locate the **Data** section. From the **Time selection** list, choose **Last**.

Line Graph 1

On the **Radiative Heat Flux along z-Coordinate** toolbar, click **Line Graph**.

Radiative Heat Flux along z-Coordinate

1. Select Boundaries 11 and 25 only.
2. In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Heat Transfer>Boundary fluxes>ht.rflux - Radiative heat flux**.
3. Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry>Coordinate>z - z-coordinate**.
4. Locate the **x-Axis Data** section. From the **Unit** list, choose **cm**.
5. On the **Radiative Heat Flux along z-Coordinate** toolbar, click **Plot**.

Derived Values

You can readily compute the total radiative heat flux from the bulb at steady state as follows.

Line Integration 1

On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.

Derived Values

1. In the **Settings** window for Line Integration, locate the **Data** section.
2. From the **Time selection** list, choose **Last**.
3. Select Boundaries 11 and 25 only.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer>Boundary fluxes>ht.rflux - Radiative heat flux**.

5 Locate the **Integration Settings** section. Select the **Compute surface integral** check box.

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result should be close to 45 W.

Created in COMSOL Multiphysics 5.2a

Heat Conduction with a Localized Heat Source on a Disk

Introduction

This classical verification example solves the steady-state temperature distribution in a plane disk heated by a localized heat source at its center. It shows and compares different ways to define a heat source localized on a small domain by representing it either as a geometrical point or as a small disk.

Both modelings have analytical solutions to which the obtained numerical results can be compared. The results bring guidelines to select the suitable option depending on the ratio of source to surrounding geometry typical size.

Model Definition

The model computes the temperature field on a cork disk of radius $R_{\mathrm{disk}} = 0.1$ m. A fixed temperature $T_0 = 300$ K is set on the disk boundary, and a heat source of total power $P = 1$ W is applied on a small circular area (radius $R_{\mathrm{source}} = 10^{-2}$ m) centered at the origin.

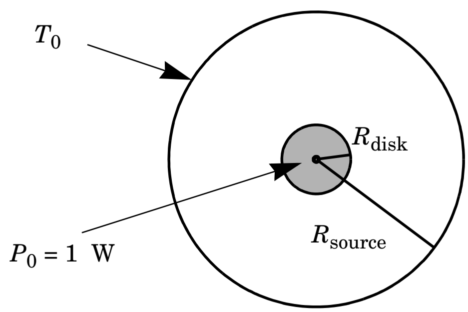


Figure 1: Geometry and boundary conditions.

The model configuration shows an axial symmetry which implies that the temperature profile is only a function of the distance to the disks center. In cylindrical coordinates it means that the temperature profile is a function of r only: $T(r, \theta) = T(r, 0)$. Despite the fact that this tutorial can be set up using a 1D axisymmetric geometry, defining it in 2D makes its extension to non-axial symmetric cases easier. Recall that Cartesian coordinates (x, y) and cylindrical coordinates (r, θ) are related by:

$$r^2 = x^2 + y^2$$

$$\theta = \operatorname{atan}\left(\frac{y}{x + \sqrt{x^2 + y^2}}\right)$$

In this document both coordinate systems are used jointly.

PUNCTUAL HEAT SOURCE MODEL

In order to simplify the geometry and to avoid high aspect ratio when R_{source} is significantly smaller than R_{disk}, represent the source as a punctual source applied on the origin point. This model corresponds to the following equation with a singular source term, with the following formal formulation:

$$\begin{cases} \nabla \cdot (-k\nabla T) = Q\delta & \text{in the disk domain} \\ T = T_0 & \text{on the disk boundary} \end{cases}$$

where k is the thermal conductivity, $Q = P/d_z$ is the volumetric heat source, d_z is the out-of-plane thickness and δ is the Dirac distribution centered at the origin. The solution of this equation is:

$$T(r) = T_0 - \frac{Q}{2\pi k}\ln\left(\frac{r}{R_{\mathrm{disk}}}\right) \tag{1}$$

According to Equation 1, the temperature goes to $\pm\infty$ when approaching the origin ($r = 0$). This singularity is illustrated on Figure 2 where the temperature value increases indefinitely when refining the mesh.

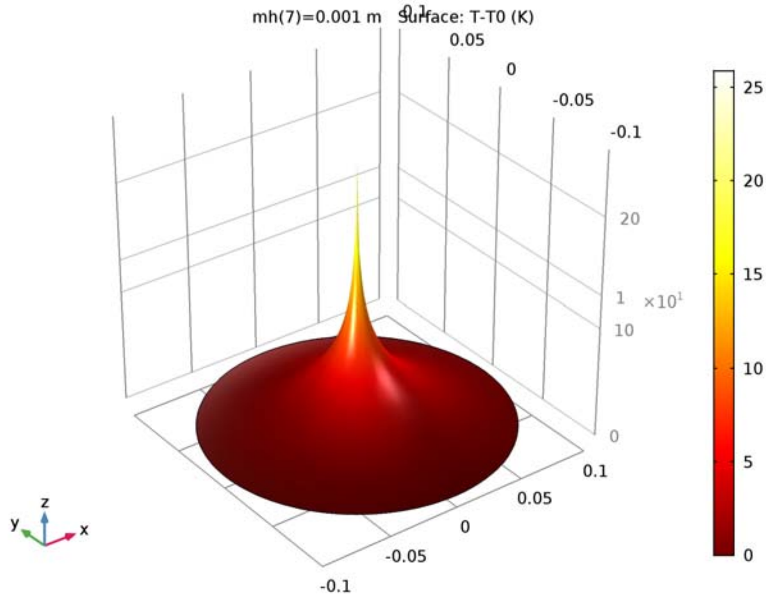


Figure 2: Distribution of relative temperature on the disk with a point heat source at the center.

VOLUME HEAT SOURCE

You can also model the heat source explicitly and apply it on a disk of radius R_{source} around the origin. Then, the formulation of the problem to solve is:

$$\begin{cases} \nabla \cdot (-k\nabla T) = f & \text{in the disk domain} \\ T = T_0 & \text{on the disk boundary} \end{cases}$$

where f is a smoothed heat source distribution defined by

$$f(r) = \begin{cases} \dfrac{Q}{\pi R_{\text{source}}^2} & \text{if } r < R_{\text{source}} \\ 0 & \text{if } r \geq R_{\text{source}} \end{cases}$$

The analytical solution in this case is:

$$T(r) = \begin{cases} T_0 - \dfrac{Q}{2\pi k}\left(\dfrac{1}{2}\left(\dfrac{r^2}{R_{\mathrm{source}}^2} - 1\right) + \ln\left(\dfrac{R_{\mathrm{source}}}{R_{\mathrm{disk}}}\right)\right) & \text{if } r < R_{\mathrm{source}} \\ T_0 - \dfrac{Q}{2\pi k}\ln\left(\dfrac{r}{R_{\mathrm{disk}}}\right) & \text{if } r \geq R_{\mathrm{source}} \end{cases} \quad (2)$$

The spatial extension of the heat source has a smoothing effect that removes the singularity at the origin, as shown on Figure 3 for $R_{\mathrm{source}} = 10^{-2}$ m:

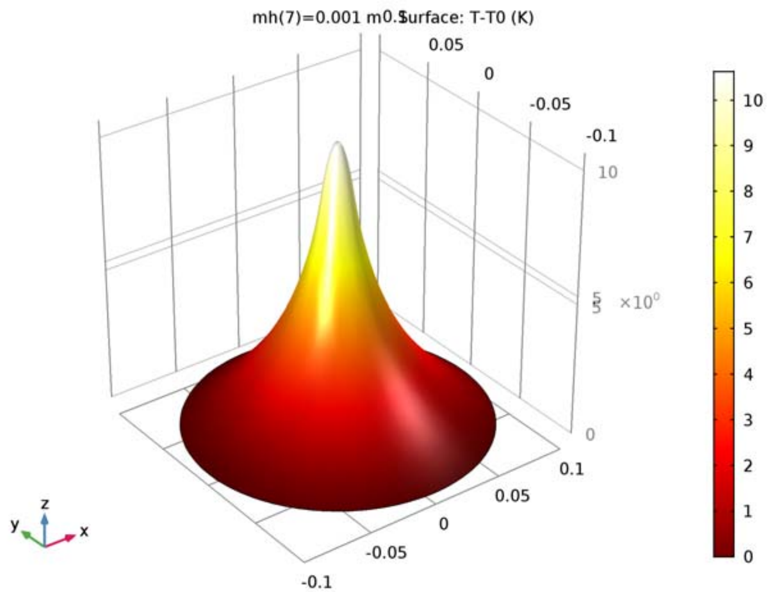


Figure 3: Distribution of relative temperature on the disk with a volume heat source.

Equation 1 and Equation 2 show that the temperature profiles are identical for $r \geq R_{\mathrm{source}}$. The only difference is observed inside the source disk ($r < R_{\mathrm{source}}$), as shown on Figure 4:

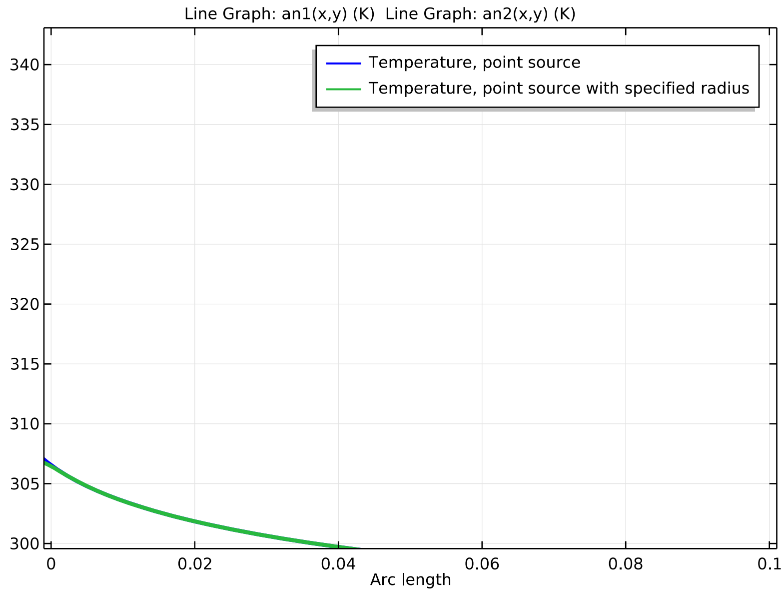


Figure 4: Analytical temperature distribution along a disk radius for punctual vs volume source, for a source radius of 10^{-2} m.

Notes about the COMSOL implementation

COMSOL Multiphysics provides different options to model a localized heat source.

1. The source support can be defined as a geometrical point. In this case, use the **Line Heat Source** (2D and 2D axisymmetric), **Point Heat Source on Axis** (2D axisymmetric) or **Point Heat Source** (3D) features. This leads to a singularity in the temperature field at the point where the source is applied. Numerically, the finer the mesh, the larger the temperature variation. In general, the two alternatives described below should be considered instead of this option, except for cases where a singular source is needed.
2. The heat source definition described above can be modified so that COMSOL Multiphysics accounts for the source size without needing a mesh nor a geometry change. In **Line Heat Source** (2D and 2D axisymmetric), **Point Heat Source on Axis** (2D axisymmetric) or **Point Heat Source** (3D) features, select the **Specify heat source radius** check-box and set the **Heat source radius** to R_{source}. Then the heat source is automatically distributed over a disk in 2D (a torus in 2D axisymmetric or a sphere in 3D) as illustrated by the black disk on the right image of Figure 5, even if the mesh

elements size is larger than R_{source}.

3 If the size of the source is not too small compared to the surrounding geometry details, then a domain representing the heat source can be drawn (see the disk of radius R_{source} on the left image of Figure 5) and a **Heat Source** feature can be defined there. This option can be considered when the increase of the number of mesh elements induced by the geometry change can be afforded.

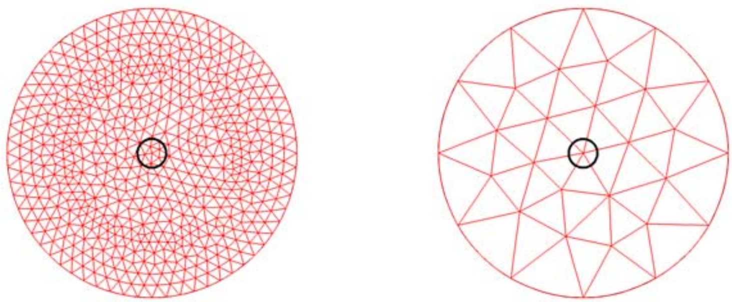


Figure 5: Different mesh configurations: elements size smaller (left) and larger (right) than the source radius. The source domain is delimited by a black circle at the center.

Results and Discussion

In this section, first take advantage of the simple geometrical configuration to verify the accuracy of the different methods by comparing the numerical results with the analytical solutions in the Numerical accuracy of the different methods subsection.

However in many practical cases, the use of the **Heat Source** feature is not an option because of the prohibitive computational cost induced by the meshing of the heat source domain. The accuracy of the **Line Heat Source** feature, with or without the **Specify heat source radius** option is analyzed in the Coarse mesh case subsection.

Finally the results are summarized to define Guidelines for modeling a heat source localized on a small domain.

NUMERICAL ACCURACY OF THE DIFFERENT METHODS

In order to check the accuracy of the **Line Heat Source** feature for a punctual heat source, the mesh is gradually refined around the heat source, by lowering the maximum size `mh` of the elements in the neighborhood of the origin from 10^{-2} m to 10^{-6} m.

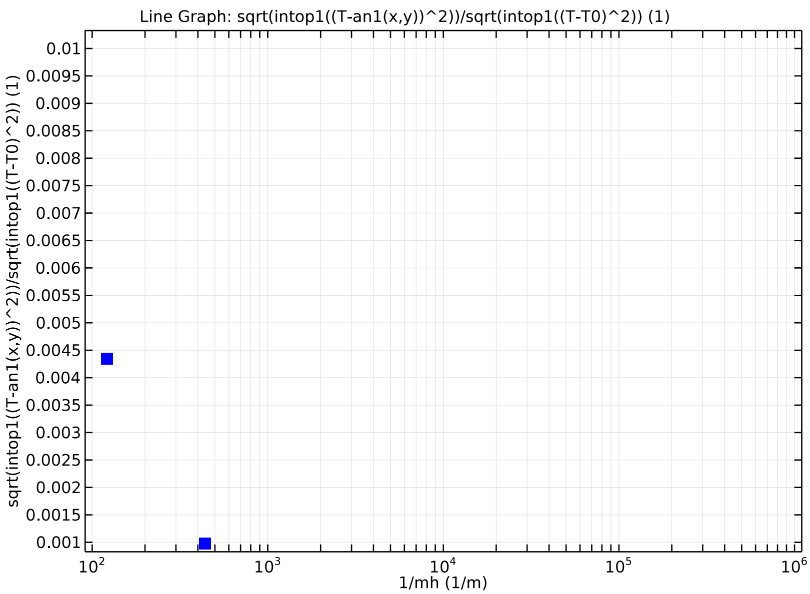


Figure 6: Relative L2 error (punctual analytical vs numerical solution with Line Heat Source feature) as a function of mesh refinement

Figure 6 shows that the relative L2 error diminishes with mesh refinement, which validates the use of the **Line Heat Source** for this kind of problems.

The maximum temperature value obtained with different meshes, shown in Figure 7, illustrates the temperature amplitude increase as the mesh size decreases.

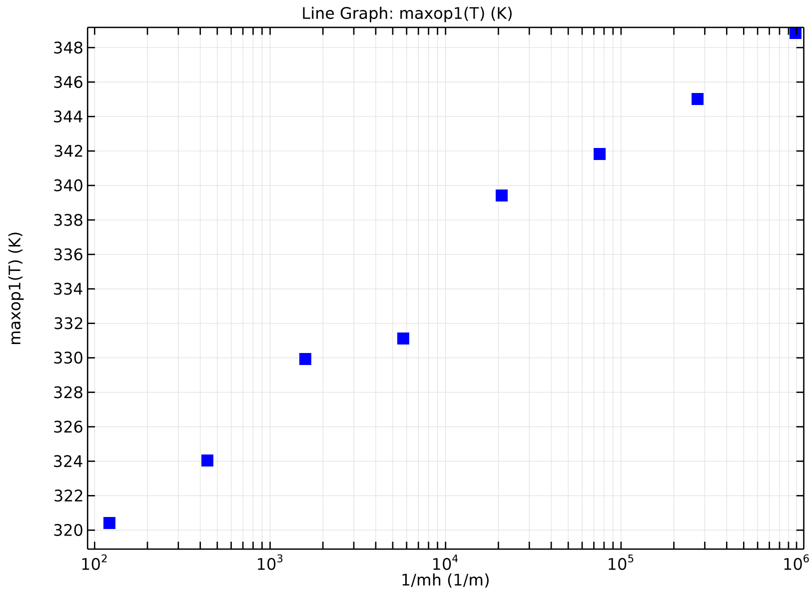


Figure 7: Maximum temperature as a function of mesh refinement (numerical solution for a punctual heat source)

A model with a volume source (R_{source} = 10^{-2} m) is now considered and the convergence of the numerical solution as the mesh is refined is investigated.

First, the **Heat Source** feature is used on the domain of radius R_{source}, which has to be explicitly drawn in the geometry. Figure 8 shows the very good agreement between the computed temperature and the analytical solution.

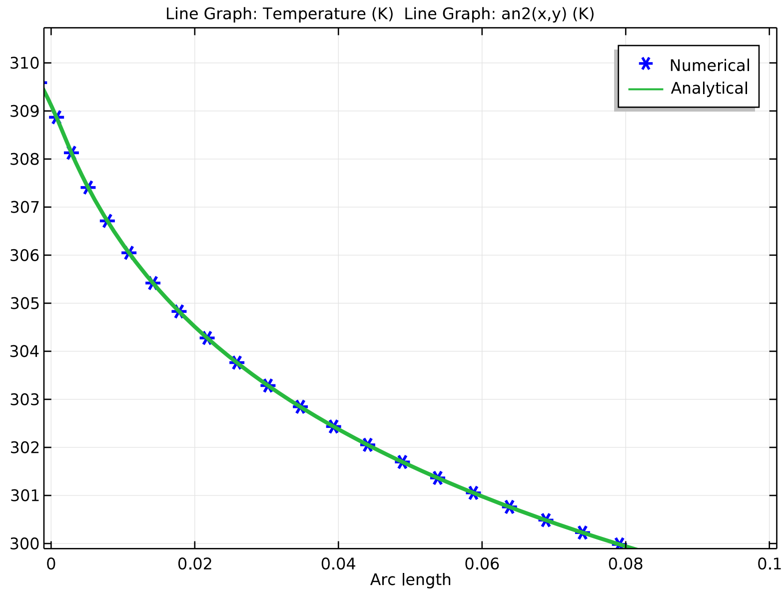


Figure 8: Temperature distribution along a disk radius for a volume source, analytical and numerical computations (Heat Source feature)

The accuracy of the **Line Heat Source** feature with **Specify heat source radius** checkbox selected is verified using comparable mesh configurations on a geometry representing the source as a point.

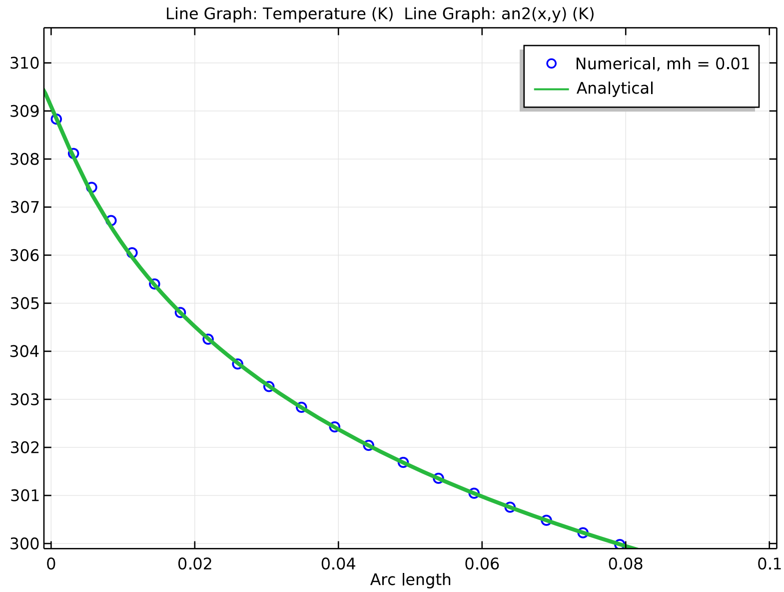


Figure 9: Temperature distribution along a disk radius for a volume source, analytical and numerical solutions (Line Heat Source with Specify heat source radius check-box selected)

Figure 9 shows the very good agreement between the analytical solution and the temperature computed using the **Line Heat Source** feature with **Specify heat source radius** checkbox selected.

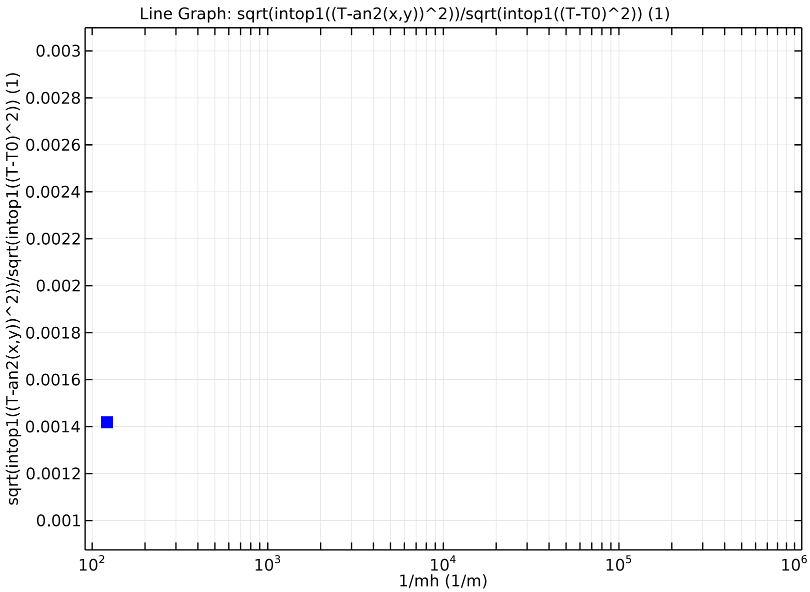


Figure 10: Relative L2 error (volume source analytical vs numerical solution Line Heat Source with Specify heat source radius check-box selected) as a function of mesh refinement

The convergence of the relative L2 error for fine mesh cases (`mh` $\leq 10^{-2}$ m) is shown on Figure 10. This validates the accuracy of the **Line Heat Source** feature with **Specify heat source radius** check-box selected.

COARSE MESH CASE

When the meshing of the heat source domain is not be affordable, the **Heat Source** feature is not applicable any more. Low mesh resolution configurations are now considered to compare the accuracy of the **Line Heat Source** feature, with or without the **Specify heat source radius**.

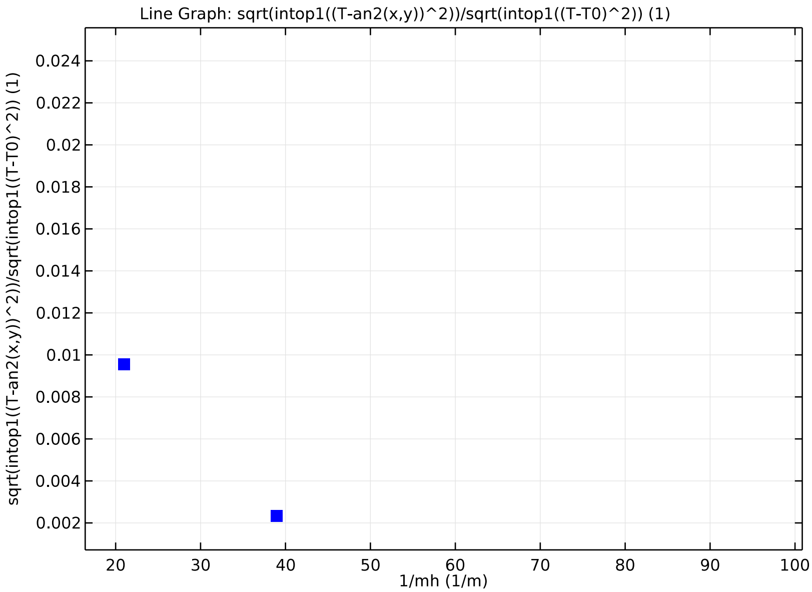


Figure 11: Relative L2 error as a function of mesh size, using Line Heat Source feature with Specify heat source radius check-box selected

The case `mh` = 0.06 m (first point in the top left corner) corresponds to the mesh displayed on the right column of Figure 5, for which the circle of radius R_{source} is much smaller than the mesh element size. Even for this case, the relative L2 error remains in an acceptable range (relative error less than 0.03).

To go further, the error on this very coarse mesh is compared for the two versions of the **Line Heat Source** feature, namely with and without the **Specify heat source radius** checkbox selected.

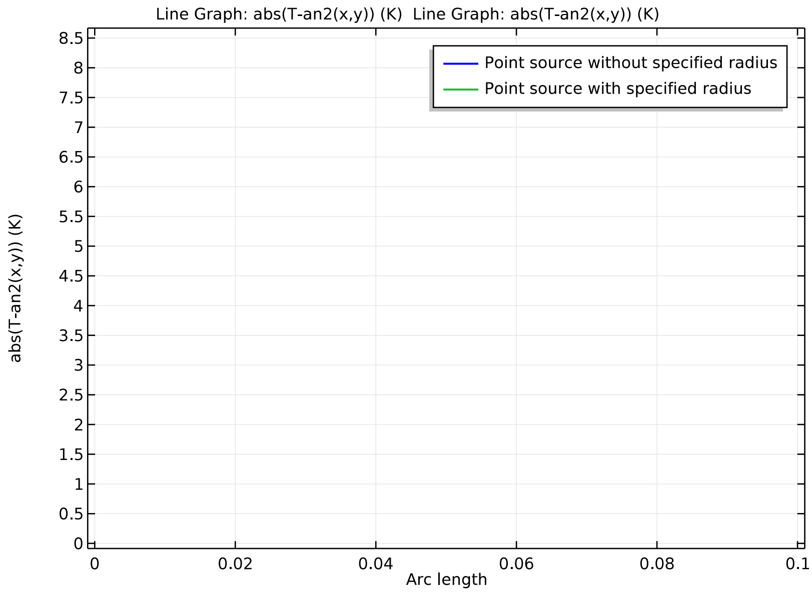


Figure 12: Temperature distribution along a disk radius for the two versions of the Line Heat Source feature (with and without specified radius), on a coarse mesh (mh = 0.06 m)

Figure 12 shows that the error close to the heat source is greatly reduced by selecting the **Specify heat source radius** checkbox.

GUIDELINES

This tutorial brings the following conclusions regarding the modeling of localized heat sources with COMSOL Multiphysics.

If the heat source radius is large enough so that it can be drawn and meshed without prohibitive computational cost, then the **Heat Source** feature is the best option. In other cases, the **Line Heat Source** (2D and 2D axisymmetric), **Point Heat Source on axis** (2D axisymmetric) or **Point Heat Source** (3D) features with **Specify heat source radius** checkbox selected should be preferred. The **Specify heat source radius** checkbox is left unselected only for cases where the source is intended to be singular.

The **Line Heat Source** with specified radius option appears therefore as an accurate alternative to the punctual approach. In particular the temperature at the heat source location converges to a finite value when the mesh is refined.

Application Library path: `Heat_Transfer_Module/Verification_Examples/localized_heat_source`

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**.

2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
R_disk	0.1[m]	0.1 m	Disk radius
R_source	0.01[m]	0.01 m	Source radius
T0	300[K]	300 K	Disk boundary temperature
k_cork	0.042[W/(m*K)]	0.042 W/(m·K)	Thermal conductivity, cork

Name	Expression	Value	Description
cp_cork	1.88[kJ/(kg*K)]	1880 J/(kg·K)	Heat capacity at constant pressure, cork
rho_cork	150[kg/m^3]	150 kg/m³	Density, cork
mh	0.01[m]	0.01 m	Mesh size parameter

Define an analytic function for the solution of the problem with a punctual heat source.

Analytic 1 (an1)

1. On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.
2. In the **Settings** window for Analytic, locate the **Definition** section.
3. In the **Expression** text field, type `-1/(2*pi*k_cork)*log(sqrt(x^2+y^2)/R_disk) + T0`.
4. In the **Arguments** text field, type `x,y`.
5. Locate the **Units** section. In the **Arguments** text field, type `m`.
6. In the **Function** text field, type `K`.
7. Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
x	-R_disk	R_disk
y	-R_disk	R_disk

8. Click **Plot**.

Define an analytic function for the solution of the problem with a volume heat source.

Analytic 2 (an2)

1. On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.
2. In the **Settings** window for Analytic, locate the **Definition** section.
3. In the **Expression** text field, type `if(sqrt(x^2+y^2)>R_source, (-1/(2*pi*k_cork)*log(sqrt(x^2+y^2)/R_disk) + T0), (1/(2*pi*k_cork)*(-(x^2+y^2)/(2*R_source^2)+0.5-log(R_source/R_disk)) + T0))`.
4. In the **Arguments** text field, type `x,y`.

5 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
x	-R_disk	R_disk
y	-R_disk	R_disk

6 Locate the **Units** section. In the **Arguments** text field, type m.

7 In the **Function** text field, type K.

8 Click **Plot**.

The geometry consists of a circle and a point at the origin of the circle. This geometry is suitable for the definition of either a punctual or a volume heat source with the **Line Heat Source** feature.

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for Circle, locate the **Size and Shape** section.

3 In the **Radius** text field, type R_disk.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

2 In the **Settings** window for Point, click **Build All Objects**.

Next, create a new material (Cork) for the disk, and define the needed properties.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, type Cork in the **Label** text field.

2 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	k_cork	W/(m·K)	Basic

Property	Name	Value	Unit	Property group
Density	rho	rho_cork	kg/m³	Basic
Heat capacity at constant pressure	Cp	cp_cork	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 In the **Settings** window for Temperature, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Temperature** section. In the T_0 text field, type T0.

As a first step, consider the case of a punctual heat source. Use the **Line Heat Source** feature with default settings for that. You will disable this branch later when defining a volume heat source.

Line Heat Source 1

1 On the **Physics** toolbar, click **Points** and choose **Line Heat Source**.

2 Select Point 3 only.

3 In the **Settings** window for Line Heat Source, locate the **Line Heat Source** section.

4 Click the **Heat rate** button.

5 In the P_l text field, type 1.

Next, define a parameterized mesh that can be refined around the origin of the disk. This way you can study the effect of mesh refinement on the heat source computation without increasing too much the mesh size.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

3 From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for Size, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarse**.

Free Triangular 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

Size 1

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Point**.

3 Select Point 3 only.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Coarse**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `mh`.

8 Click **Build All**.

Now, define integration and maximum operators on the whole domain, for postprocessing.

DEFINITIONS

Integration 1a (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

Maximum 1 (maxop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Maximum**.

2 In the **Settings** window for Maximum, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for Study, type `Study 1: Point Source` in the **Label** text field.

Parametric Sweep

On the **Study** toolbar, click **Parametric Sweep**.

STUDY 1: POINT SOURCE

Parametric Sweep

Define a parametric sweep on the maximum size of mesh elements in order to study the effects of mesh refinement on the heat source computation.

1 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.
2 Click **Add**.
3 Select `mh` from the list.
4 Click **Range**.
5 In the **Range** dialog box, type `-6` in the **Start** text field.
6 In the **Step** text field, type `0.5`.
7 In the **Stop** text field, type `-2`.
8 From the **Function to apply to all values** list, choose **exp10**.
9 Click **Replace**.
10 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot shows the temperature distribution in a 2D plot for `mh` = 10^{-2}m. Proceed as follows to reproduce the plot of Figure 2 that corresponds to `mh` = 10^{-3}m.

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.
2 In the **Settings** window for 2D Plot Group, type `Temperature - Study 1` in the **Label** text field.
3 Locate the **Data** section. From the **Parameter value (mh (m))** list, choose **0.001**.
4 On the **Temperature - Study 1** toolbar, click **Plot**.

Surface

1 In the **Model Builder** window, expand the **Results>Temperature - Study 1** node, then click **Surface**.
2 In the **Settings** window for Surface, locate the **Expression** section.
3 In the **Expression** text field, type `T-T0`.

Height Expression 1

1 Right-click **Results>Temperature - Study 1>Surface** and choose **Height Expression**.
2 On the **Temperature - Study 1** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Then, proceed to reproduce the plot of Figure 6, by plotting the relative L2 error between numerical and analytical solutions as a function of `1/mh`, in order to check numerical convergence.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `L2 Error from Analytical Solution - Study 1` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1: Point Source/Parametric Solutions 1 (sol2)**.

Line Graph 1

1 On the **L2 Error from Analytical Solution - Study 1** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `sqrt(intop1((T-an1(x,y))^2))/sqrt(intop1((T-T0)^2))`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `1/mh`.

7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

8 In the **Width** text field, type `5`.

9 On the **L2 Error from Analytical Solution - Study 1** toolbar, click **Plot**.

10 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Next, plot the maximum temperature as a function of `1/mh`, as in the plot of Figure 7.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Maximum Temperature - Study 1` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1: Point Source/Parametric Solutions 1 (sol2)**.

Line Graph 1

1 On the **Maximum Temperature - Study 1** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `maxop1(T)`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `1/mh`.

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

8 In the **Width** text field, type `5`.

9 On the **Maximum Temperature - Study 1** toolbar, click **Plot**.

10 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

HEAT TRANSFER IN SOLIDS (HT)

As a second step, consider the case of a volume heat source. Use the **Line Heat Source** feature with the **Specify heat source radius** option selected, and the **Heat source radius** set to a positive value.

Line Heat Source 2

1 On the **Physics** toolbar, click **Points** and choose **Line Heat Source**.

2 Select Point 3 only.

3 In the **Settings** window for Line Heat Source, locate the **Line Heat Source** section.

4 Click the **Heat rate** button.

5 In the P_1 text field, type `1`.

6 Locate the **Heat Source Radius** section. Select the **Specify heat source radius** check box.

7 In the R text field, type `R_source`.

In the next steps, configure **Study 1** to use **Line Heat Source 1** and create a second study that uses **Line Heat Source 2**.

STUDY 1: POINT SOURCE

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1: Point Source** click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)>Line Heat Source 2**.

5 Click **Disable**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Stationary**.

5 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Stationary

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

3 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

4 Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)>Line Heat Source 1**.

6 Click **Disable**.

Again, define a parametric sweep on the maximum size of mesh elements in order to study the effects of mesh refinement on the heat source computation.

7 In the **Model Builder** window, click **Study 2**.

8 In the **Settings** window for Study, type `Study 2: Point Source with Radius` in the **Label** text field.

Parametric Sweep

On the **Study** toolbar, click **Parametric Sweep**.

STUDY 2: POINT SOURCE WITH RADIUS

Parametric Sweep

1 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

2 Click **Add**.

3 Select `mh` from the list.

4 Click **Range**.

5 In the **Range** dialog box, type `-6` in the **Start** text field.

6 In the **Step** text field, type `0.5`.

7 In the **Stop** text field, type `-2`.

8 From the **Function to apply to all values** list, choose **exp10**.

9 Click **Replace**.

10 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot shows the temperature distribution in a 2D plot for mh=10^{-2}m. Proceed as follows to reproduce the plot in Figure 3.

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

2 In the **Settings** window for 2D Plot Group, type `Temperature - Study 2` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (mh (m))** list, choose **0.001**.

4 On the **Temperature - Study 2** toolbar, click **Plot**.

Surface

1 In the **Model Builder** window, expand the **Results>Temperature - Study 2** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 In the **Expression** text field, type `T-T0`.

Height Expression 1

1 Right-click **Results>Temperature - Study 2>Surface** and choose **Height Expression**.

2 On the **Temperature - Study 2** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Plot the analytical solutions of the two problems (punctual and volume heat sources) along a disk radius, to reproduce the plot of Figure 4.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for Cut Line 2D, locate the **Data** section.

3 From the **Data set** list, choose **Study 2: Point Source with Radius/Parametric Solutions 2 (sol13)**.

4 Locate the **Line Data** section. In row **Point 2**, set **x** to `R_disk`.

5 Click **Plot**.

1D Plot Group 7

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Analytical Solutions, Point Source with/without Radius` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

4 From the **Parameter selection (mh)** list, choose **First**.

Line Graph 1

1 On the **Analytical Solutions, Point Source with/without Radius** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `an1(x,y)`.

4 Locate the **Coloring and Style** section. In the **Width** text field, type `3`.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Temperature, point source`

Analytical Solutions, Point Source with/without Radius

In the **Model Builder** window, under **Results** click **Analytical Solutions, Point Source with/without Radius**.

Line Graph 2

1 On the **Analytical Solutions, Point Source with/without Radius** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `an2(x,y)`.

4 Locate the **Coloring and Style** section. In the **Width** text field, type `3`.

5 Locate the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Temperature, point source with specified radius`

8 On the **Analytical Solutions, Point Source with/without Radius** toolbar, click **Plot**.

Next, proceed to reproduce the plot of Figure 9, by plotting the temperature distribution for numerical and analytical solutions along a disk radius, for `mh`$=10^{-2}$m.

1D Plot Group 8

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type `Temperature vs Radius - Study 2` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.
4. From the **Parameter selection (mh)** list, choose **Last**.

Line Graph 1

1. On the **Temperature vs Radius - Study 2** toolbar, click **Line Graph**.
2. In the **Settings** window for Line Graph, locate the **Coloring and Style** section.
3. Find the **Line style** subsection. From the **Line** list, choose **None**.
4. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
5. In the **Number** text field, type `25`.
6. Locate the **Legends** section. Select the **Show legends** check box.
7. From the **Legends** list, choose **Manual**.
8. In the table, enter the following settings:

Legends
`Numerical, mh = 0.01`

Temperature vs Radius - Study 2

In the **Model Builder** window, under **Results** click **Temperature vs Radius - Study 2**.

Line Graph 2

1. On the **Temperature vs Radius - Study 2** toolbar, click **Line Graph**.
2. In the **Settings** window for Line Graph, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `an2(x,y)`.
4. Locate the **Coloring and Style** section. In the **Width** text field, type `3`.
5. Locate the **Legends** section. Select the **Show legends** check box.
6. From the **Legends** list, choose **Manual**.
7. In the table, enter the following settings:

Legends
`Analytical`

8 On the **Temperature vs Radius - Study 2** toolbar, click **Plot**.

Proceed to reproduce the plot of Figure 10, by plotting the relative L2 error between numerical and analytical solutions as a function of mesh size parameter `1/mh`, to check numerical convergence.

1D Plot Group 9

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `L2 Error from Analytical Solution - Study 2` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2: Point Source with Radius/ Parametric Solutions 2 (sol13)**.

Line Graph 1

1 On the **L2 Error from Analytical Solution - Study 2** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `sqrt(intop1((T-an2(x,y))^2))/sqrt(intop1((T-T0)^2))`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `1/mh`.

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

8 In the **Width** text field, type `5`.

9 On the **L2 Error from Analytical Solution - Study 2** toolbar, click **Plot**.

10 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Next, proceed to reproduce the plot of Figure 12, by plotting the absolute value of the error along a disk radius between each numerical solution and the analytical one for a volume heat source, for `mh`=10^{-2}m.

Cut Line 2D 2

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for Cut Line 2D, locate the **Data** section.

3 From the **Data set** list, choose **Study 1: Point Source/Parametric Solutions 1 (sol2)**.

4 Locate the **Line Data** section. In row **Point 2**, set **x** to `R_disk`.

5 Click **Plot**.

1D Plot Group 10

1. On the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type `L1 Error from Analytical Solutions - Study 1 and Study 2` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 2**.
4. From the **Parameter selection (mh)** list, choose **Last**.

Line Graph 1

1. On the **L1 Error from Analytical Solutions - Study 1 and Study 2** toolbar, click **Line Graph**.
2. In the **Settings** window for Line Graph, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `abs(T-an2(x,y))`.
4. Locate the **Coloring and Style** section. In the **Width** text field, type `3`.
5. Locate the **Legends** section. Select the **Show legends** check box.
6. From the **Legends** list, choose **Manual**.
7. In the table, enter the following settings:

Legends
`Point source without specified radius`

L1 Error from Analytical Solutions - Study 1 and Study 2

In the **Model Builder** window, under **Results** click **L1 Error from Analytical Solutions - Study 1 and Study 2**.

Line Graph 2

1. On the **L1 Error from Analytical Solutions - Study 1 and Study 2** toolbar, click **Line Graph**.
2. In the **Settings** window for Line Graph, locate the **Data** section.
3. From the **Data set** list, choose **Cut Line 2D 1**.
4. From the **Parameter selection (mh)** list, choose **Last**.
5. Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(T-an2(x,y))`.
6. Locate the **Coloring and Style** section. In the **Width** text field, type `3`.
7. Locate the **Legends** section. Select the **Show legends** check box.
8. From the **Legends** list, choose **Manual**.
9. In the table, enter the following settings:

Legends
`Point source with specified radius`

10 On the **L1 Error from Analytical Solutions - Study 1 and Study 2** toolbar, click **Plot**.

Now, define a new mesh that is not refined any more at the origin of the disk, but is parameterized to be coarsened instead, with parameter `mh`.

COMPONENT 1 (COMP1)

Mesh 2

On the **Mesh** toolbar, click **Add Mesh**.

MESH 2

1 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

2 From the **Sequence type** list, choose **User-controlled mesh**.

   Use the parameter `mh` to control the maximum and minimum size of the mesh elements.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes>Mesh 2** 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 **Maximum element size** text field, type `mh`.

5 In the **Minimum element size** text field, type `mh`.

6 Click **Build All**.

Define a new study corresponding to the problem with a volume source on a coarse mesh.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Stationary**.

5 Click **Add Study** in the window toolbar.

STUDY 3

Step 1: Stationary

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Model Builder** window, under **Study 3** click **Step 1: Stationary**.

3 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

4 Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)>Line Heat Source 1**.

6 Click **Disable**.

Define a parametric sweep on the size of mesh elements in order to check that the heat source is still well approximated on coarse meshes.

7 In the **Model Builder** window, click **Study 3**.

8 In the **Settings** window for Study, type `Study 3: Point Source with Radius, Coarse Mesh` in the **Label** text field.

Parametric Sweep

On the **Study** toolbar, click **Parametric Sweep**.

STUDY 3: POINT SOURCE WITH RADIUS, COARSE MESH

Parametric Sweep

1 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

2 Click **Add**.

3 Select `mh` from the list.

4 Click **Range**.

5 In the **Range** dialog box, type `0.01` in the **Start** text field.

6 In the **Step** text field, type `0.012`.

7 In the **Stop** text field, type `0.06`.

8 Click **Replace**.

9 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot shows the temperature distribution in a 2D plot for `mh`$=10^{-2}$m. Proceed as follows to reproduce the plots for mesh configurations in Figure 5.

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

2 In the **Settings** window for 2D Plot Group, type `Mesh Resolution - Study 3` in the **Label** text field.

Surface

1 In the **Model Builder** window, expand the **Results>Mesh Resolution - Study 3** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Uniform**.

4 Select the **Wireframe** check box.

Mesh Resolution - Study 3

In the **Model Builder** window, under **Results** right-click **Mesh Resolution - Study 3** and choose **Contour**.

Contour 1

1 In the **Settings** window for Contour, locate the **Expression** section.

2 In the **Expression** text field, type `sqrt(x^2+y^2)`.

3 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `R_source`.

5 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Tube**.

6 From the **Coloring** list, choose **Uniform**.

7 From the **Color** list, choose **Black**.

8 Clear the **Color legend** check box.

9 On the **Mesh Resolution - Study 3** toolbar, click **Plot**.

10 Click the **Zoom Extents** button on the **Graphics** toolbar.

Now change the mesh resolution to a finer configuration.

Mesh Resolution - Study 3

1 In the **Model Builder** window, under **Results** click **Mesh Resolution - Study 3**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Parameter value (mh (m))** list, choose **0.01**.

4 On the **Mesh Resolution - Study 3** toolbar, click **Plot**.

Next, proceed to reproduce the plot of Figure 11, by plotting the relative L2 error between numerical and analytical solutions as a function of `1/mh`, to check numerical convergence.

1D Plot Group 13

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `L2 Error from Analytical Solution - Study 3` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 3: Point Source with Radius, Coarse Mesh/Parametric Solutions 3 (sol24)**.

Line Graph 1

1 On the **L2 Error from Analytical Solution - Study 3** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `sqrt(intop1((T-an2(x,y))^2))/sqrt(intop1((T-T0)^2))`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `1/mh`.

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

8 In the **Width** text field, type `5`.

9 On the **L2 Error from Analytical Solution - Study 3** toolbar, click **Plot**.

As a last step, you can check that the computation of a volume heat source can also be performed by using the **Heat Source** feature. In order to do that, you need to include a smaller circle of radius `R_source` into the geometry, and to change the physics branch. Define a new component to include these changes and keep this step separated from the previous ones.

ROOT

On the **Home** toolbar, click **Add Component** and choose **2D**.

GEOMETRY 2

Import the previously defined geometry, and complete it with the circle of radius `R_source`.

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

2 In the **Settings** window for Import, locate the **Import** section.

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Geometry 1**.

5 Click **Import**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for Circle, locate the **Size and Shape** section.

3 In the **Radius** text field, type `R_source`.

4 Click **Build All Objects**.

This time, use the **Heat Source** feature to apply the source in the domain delimited by the circle of radius `R_source`. This replaces the use of the **Line Heat Source** feature in previous steps.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the following studies:

Studies
Study 1: Point Source
Study 2: Point Source with Radius
Study 3: Point Source with Radius, Coarse Mesh

5 Click **Add to Component** in the window toolbar.

HEAT TRANSFER IN SOLIDS 2 (HT2)

On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundaries 1, 2, 5, and 8 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `T0`.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 2 only.

3 In the **Settings** window for Heat Source, locate the **Heat Source** section.

4 Click the **Heat rate** button.

5 In the P_0 text field, type 1.

Define the same material as before.

MATERIALS

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Blank Material**.

Material 2 (mat2)

1 In the **Settings** window for Material, type `Cork` in the **Label** text field.

2 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	`k_cork`	W/(m·K)	Basic
Density	rho	`rho_cork`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`cp_cork`	J/(kg·K)	Basic

You can visualize the mesh generated by default for this new geometry.

MESH 3

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 3**.

2 In the **Settings** window for Mesh, click **Build All**.

Add a new study for the computation of the volume heat source with the **Heat Source** feature.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Stationary**.

5 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Heat Transfer in Solids (ht)** interface.

6 Click **Add Study** in the window toolbar.

STUDY 4

Step 1: Stationary

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Model Builder** window, click **Study 4**.

3 In the **Settings** window for Study, type `Study 4: Surface Source` in the **Label** text field.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht2)

1 In the **Model Builder** window, under **Results** click **Temperature (ht2)**.

2 In the **Settings** window for 2D Plot Group, type `Temperature - Study 4` in the **Label** text field.

Finally, proceed to reproduce the plot of Figure 8, by plotting the temperature distribution for numerical and analytical solutions along a disk radius.

Cut Line 2D 3

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for Cut Line 2D, locate the **Data** section.

3 From the **Data set** list, choose **Study 4: Surface Source/Solution 30 (8) (sol30)**.

4 Locate the **Line Data** section. In row **Point 2**, set **x** to `R_disk`.

5 Click **Plot**.

1D Plot Group 16

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Temperature vs Radius - Study 4` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 3**.

Line Graph 1

1 On the **Temperature vs Radius - Study 4** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 In the **Number** text field, type `25`.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
Numerical

Temperature vs Radius - Study 4

In the **Model Builder** window, under **Results** click **Temperature vs Radius - Study 4**.

Line Graph 2

1 On the **Temperature vs Radius - Study 4** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `an2(x,y)`.

4 Locate the **Coloring and Style** section. In the **Width** text field, type `3`.

5 Locate the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Analytical

8 On the **Temperature vs Radius - Study 4** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Marangoni Effect

Introduction

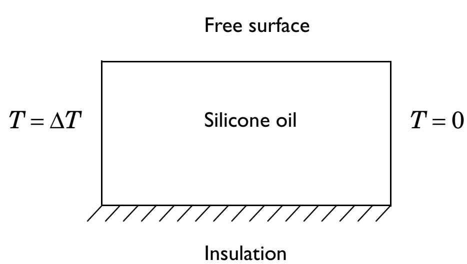
Marangoni convection occurs when the surface tension of an interface (generally liquid-air) depends on the concentration of a species or on the temperature distribution. In the case of temperature dependence, the Marangoni effect is also called thermo-capillary convection. It is of primary importance in the fields of:

- Welding
- Crystal growth
- Electron beam melting of metals

Direct experimental studies are not easy to carry out in these systems because the materials are often metals and temperatures are very high. One possibility is to replace the real system with an experimental setup using a transparent liquid at ambient temperatures.

Model Definition

This tutorial describes the 2D stationary behavior of a vessel filled with silicone oil, for which the thermo-physical properties are known. The aim of the study is to compute the temperature field that induces a flow through the Marangoni effect. The model shows this effect using the simple geometry in the figure below.



GOVERNING EQUATIONS

A stationary momentum balance equation describes the velocity field and the pressure distribution (Navier-Stokes equations, see Incompressible Flow). To include the heating of the fluid, the fluid flow is coupled to an energy balance.

You can use the Boussinesq approximation to include the effect of temperature on the velocity field. In this approximation, variations in temperature produce a buoyancy force (or Archimedes' force) that lifts the fluid as described in Gravity and The Boussinesq Approximation sections in the *CFD Module User's Guide*.

The following equation describes the forces that the Marangoni effect induces on the interface (liquid/air):

$$\eta\frac{\partial u}{\partial y} = \gamma\frac{\partial T}{\partial x} \qquad (1)$$

Here γ is the temperature derivative of the surface tension (N/(m·K)). Equation 1 states that the shear stress on a surface is proportional to the temperature gradient (Ref. 1).

Notes About the COMSOL Implementation

To solve the momentum and energy balance equations, use the predefined Non-Isothermal Flow multiphysics coupling. It automatically couples a Laminar Flow interface for the fluid flow to a Heat Transfer in Fluids interface for the heat transfer by convection and conduction in each direction:

- The Boussinesq approximation means that an expression including temperature acts as a force in the y direction in the momentum balance.
- The convective heat transfer depends on the velocities from the momentum balance.

This means that you must solve the coupled system directly using the nonlinear solver.

To impose the condition that the shear stress is proportional to the temperature gradient on the surface, use the Marangoni Effect multiphysics feature in the Multiphysics node.

Results

The Marangoni effect becomes more pronounced as the temperature difference increases:

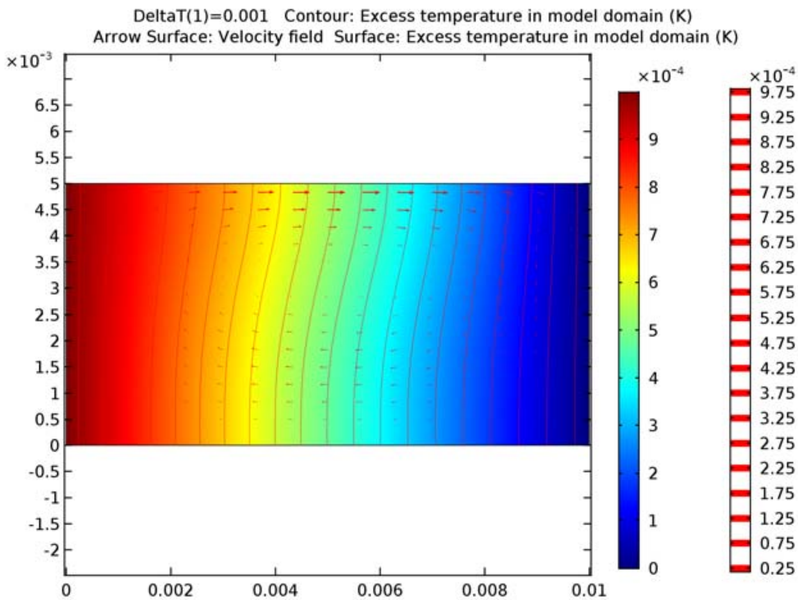


Figure 1: Marangoni convection with a temperature difference of 0.001 K.

For the very low temperature difference of 0.001 K, the temperature field is almost decoupled from the velocity field. Therefore, the temperature decreases almost linearly from left to right.

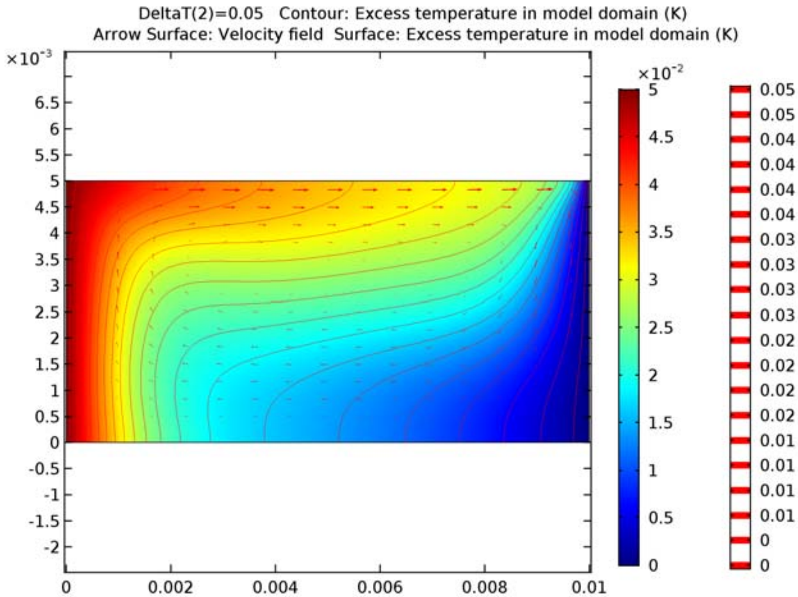


Figure 2: Marangoni convection with a temperature difference of 0.05 K.

For the temperature difference of 0.05 K notice how the Marangoni convection influences the flow of fluid and the distribution of temperature. The temperature is no longer decreasing linearly and you can clearly see the advection of the isotherms caused by the flow.

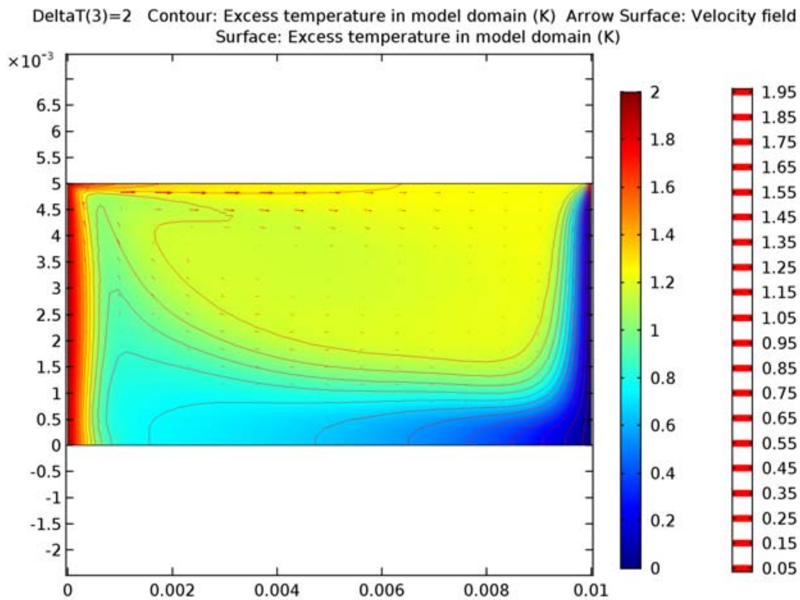


Figure 3: Marangoni convection with a temperature difference of 2 K.

At higher temperature differences (2 K in Figure 3 above), the physical coupling between the temperature and the velocity field is clearly visible. The heat conduction is small compared to the convection, and at the surface the fluid accelerates where the temperature gradient is high.

Reference

1. V.G. Levich, *Physicochemical Hydrodynamics*, Prentice-Hall, N.J., 1962.

Application Library path: `Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/marangoni_effect`

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**.
2 In the **Select Physics** tree, select **Fluid Flow>Non-Isothermal Flow>Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.
3 In the **Width** text field, type `10[mm]`.
4 In the **Height** text field, type `5[mm]`.
5 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the application's Application Libraries folder and double-click the file `marangoni_effect_parameters.txt`.

DEFINITIONS

Variables 1

1 On 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**
`deltaT`	`T-T_right`	K	`Excess temperature in model domain`

This variable is useful when visualizing the model results.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, type `Silicone Oil` in the **Label** text field.

2 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Dynamic viscosity	mu	`mu1`	Pa·s	Basic
Thermal conductivity	k	`k1`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	`Cp1`	J/(kg·K)	Basic
Ratio of specific heats	gamma	`1`	1	Basic

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for Laminar Flow, locate the **Physical Model** section.

3 Select the **Include gravity** check box.

4 From the **Compressibility** list, choose **Incompressible flow**.

5 Find the **Reference values** subsection. In the T_{ref} text field, type `T_ref`.

Here, `T_ref` is the reference temperature at which the material properties are evaluated. It is defined in **Parameters** under **Global Definitions**.

Wall 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundary 3 only.

3 In the **Settings** window for Wall, locate the **Boundary Condition** section.

4 From the **Boundary condition** list, choose **Slip**.

Pressure Point Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 1 only.

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, type `T_right` in the T text field.

3 In the **Model Builder** window, click **Heat Transfer in Fluids (ht)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 4 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `T_right`.

Temperature 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 1 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `T_right+DeltaT`.

MULTIPHYSICS

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Non-Isothermal Flow 1 (nitf1)**.

2 In the **Settings** window for Non-Isothermal Flow, locate the **Material Properties** section.

3 From the **Specify density** list, choose **Custom, linearized density**.

4 In the ρ_{ref} text field, type `rho1`.

5 In the α_p text field, type `alphap1`.

Marangoni Effect 1 (me1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Boundary>Marangoni Effect**.

2 Select Boundary 3 only.

3 In the **Settings** window for Marangoni Effect, locate the **Surface Tension** section.

4 In the σ text field, type `gamma*T`.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

Free Triangular 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

Size 1

1. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Boundary**.
3. Select Boundary 3 only.
4. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6. In the associated text field, type `1e-4`.

Free Triangular 1

Right-click **Free Triangular 1** and choose **Size**.

Size 2

1. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Point**.
3. Select Points 2 and 4 only.
4. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6. In the associated text field, type `2e-5`.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for Size, locate the **Element Size** section.
3. From the **Predefined** list, choose **Extra fine**.
4. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type `1.1`.
6. Click **Build All**.

STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
DeltaT	1e-3 0.05 2	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

To show the temperature field as a surface plot along with overlaid temperature contours and the velocity field using arrows, follow the steps given below.

Isothermal Contours (ht)

1 In the **Model Builder** window, under **Results** click **Isothermal Contours (ht)**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Parameter value (DeltaT)** list, choose **0.001**.

Contour

1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Contour**.

2 In the **Settings** window for Contour, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Definitions> Variables>deltaT - Excess temperature in model domain**.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

4 Right-click **Results>Isothermal Contours (ht)>Contour** and choose **Arrow Surface**.

Isothermal Contours (ht)

In the **Model Builder** window, under **Results** right-click **Isothermal Contours (ht)** and choose **Surface**.

Surface 1

1 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Definitions> Variables>deltaT - Excess temperature in model domain**.

2 On the **Isothermal Contours (ht)** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Isothermal Contours (ht)

The Marangoni effect becomes more pronounced as the temperature difference increases. Visualize this by changing the Parameter value selection.

1 In the **Model Builder** window, under **Results** click **Isothermal Contours (ht)**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Parameter value (DeltaT)** list, choose **0.05**.

4 On the **Isothermal Contours (ht)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

6 From the **Parameter value (DeltaT)** list, choose **2**.

7 On the **Isothermal Contours (ht)** toolbar, click **Plot**.

Microwave Heating of a Cancer Tumor

Introduction

Electromagnetic heating appears in a wide range of engineering problems and is ideally suited for modeling in COMSOL Multiphysics because of its multiphysics capabilities. This example comes from the area of hyperthermic oncology and it models the electromagnetic field coupled to the bioheat equation. The modeling issues and techniques are generally applicable to any problem involving electromagnetic heating.

In hyperthermic oncology, cancer is treated by applying localized heating to the tumor tissue, often in combination with chemotherapy or radiotherapy. Some of the challenges associated with the selective heating of deep-seated tumors without damaging surrounding tissue are:

- Control of heating power and spatial distribution
- Design and placement of temperature sensors

Among possible heating techniques, RF and microwave heating have attracted much attention from clinical researchers. Microwave coagulation therapy is one such technique where a thin microwave antenna is inserted into the tumor. The microwaves heat up the tumor, producing a coagulated region where the cancer cells are killed.

This model computes the temperature field, the radiation field, and the specific absorption rate (SAR)—defined as the ratio of absorbed heat power and tissue density—in liver tissue when using a thin coaxial slot antenna for microwave coagulation therapy. It closely follows the analysis found in Ref. 1. It computes the temperature distribution in the tissue using the bioheat equation.

Note: This application requires the RF Module and the Heat Transfer Module.

Model Definition

Figure 1 shows the antenna geometry. It consists of a thin coaxial cable with a ring-shaped slot measuring 1 mm cut on the outer conductor 5 mm from the short-circuited tip. For hygienic purposes, the antenna is enclosed in a sleeve (catheter) made of PTFE (polytetrafluoroethylene). The following tables give the relevant geometrical dimensions and material data. The antenna operates at 2.45 GHz, a frequency widely used in microwave coagulation therapy.

TABLE 1: DIMENSIONS OF THE COAXIAL SLOT ANTENNA.

PROPERTY	VALUE
Diameter of the central conductor	0.29 mm
Inner diameter of the outer conductor	0.94 mm
Outer diameter of the outer conductor	1.19 mm
Diameter of catheter	1.79 mm

TABLE 2: MATERIAL PROPERTIES.

PROPERTY	INNER DIELECTRIC OF COAXIAL CABLE	CATHETER	LIVER TISSUE
Relative permittivity	2.03	2.60	43.03
Conductivity			1.69 S/m

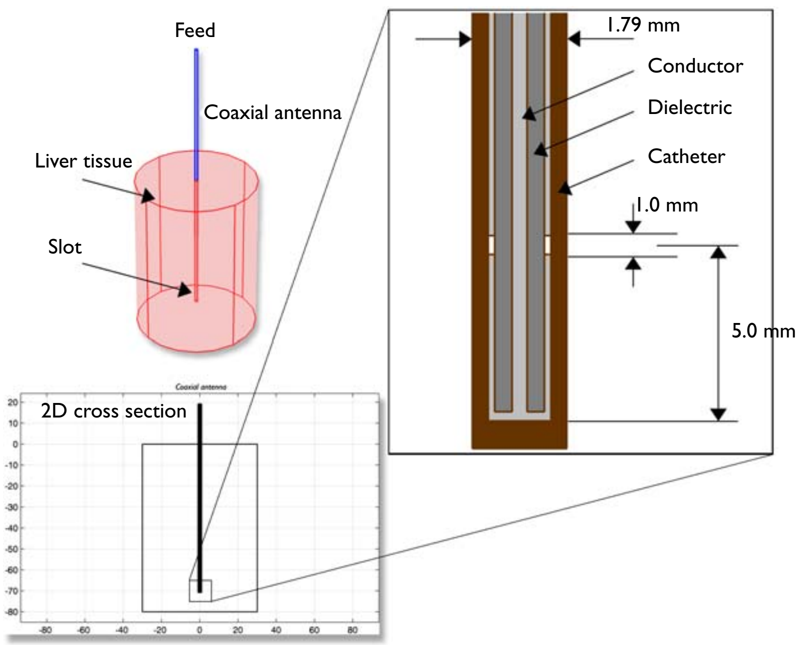


Figure 1: Antenna geometry for microwave coagulation therapy. A coaxial cable with a ring-shaped slot cut on the outer conductor is short-circuited at the tip. A plastic catheter surrounds the antenna.

The model takes advantage of the problem's rotational symmetry, which allows modeling in 2D using cylindrical coordinates as indicated in Figure 2. When modeling in 2D, you can select a fine mesh and achieve excellent accuracy. The model uses a frequency-domain problem formulation with the complex-valued azimuthal component of the magnetic field as the unknown.

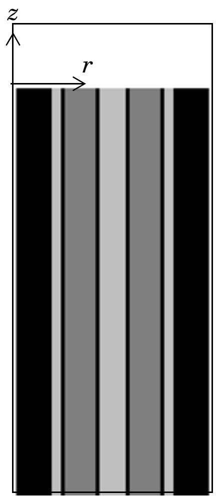


Figure 2: The computational domain appears as a rectangle in the rz-plane.

The radial and axial extent of the computational domain is in reality larger than indicated in Figure 2. This problem does not model the interior of the metallic conductors, and it models metallic parts using boundary conditions, setting the tangential component of the electric field to zero.

DOMAIN AND BOUNDARY EQUATIONS—ELECTROMAGNETICS

An electromagnetic wave propagating in a coaxial cable is characterized by transverse electromagnetic fields (TEM). Assuming time-harmonic fields with complex amplitudes containing the phase information, the appropriate equations are

$$\mathbf{E} = \mathbf{e}_r \frac{C}{r} e^{j(\omega t - kz)}$$

$$\mathbf{H} = \mathbf{e}_\varphi \frac{C}{rZ} e^{j(\omega t - kz)}$$

$$\mathbf{P}_{\mathrm{av}} = \int_{r_{\mathrm{inner}}}^{r_{\mathrm{outer}}} \mathrm{Re}\left(\frac{1}{2}\mathbf{E} \times \mathbf{H}^*\right) 2\pi r dr = \mathbf{e}_z \pi \frac{C^2}{Z} \ln\left(\frac{r_{\mathrm{outer}}}{r_{\mathrm{inner}}}\right)$$

where z is the direction of propagation, and r, φ, and z are cylindrical coordinates centered on the axis of the coaxial cable. $\mathbf{P}_{av}$ is the time-averaged power flow in the cable, Z is the wave impedance in the dielectric of the cable, while r_{inner} and r_{outer} are the dielectric's inner and outer radii, respectively. Further, ω denotes the angular frequency. The propagation constant, k, relates to the wavelength in the medium, λ, as

$$k = \frac{2\pi}{\lambda}$$

In the tissue, the electric field also has a finite axial component whereas the magnetic field is purely in the azimuthal direction. Thus, you can model the antenna using an axisymmetric transverse magnetic (TM) formulation. The wave equation then becomes scalar in H_φ:

$$\nabla \times \left(\left(\varepsilon_r - \frac{j\sigma}{\omega\varepsilon_0}\right)^{-1} \nabla \times H_\varphi\right) - \mu_r k_0{}^2 H_\varphi = 0$$

The boundary conditions for the metallic surfaces are

$$\mathbf{n} \times \mathbf{E} = 0$$

The feed point is modeled using a port boundary condition with a power level set to 10 W. This is essentially a first-order low-reflecting boundary condition with an input field $H_{\varphi 0}$:

$$\mathbf{n} \times \sqrt{\varepsilon}\mathbf{E} - \sqrt{\mu} H_\varphi = -2\sqrt{\mu} H_{\varphi 0}$$

where

$$H_{\varphi 0} = \frac{\sqrt{\dfrac{\mathbf{P}_{av} Z}{\pi r \ln\left(\dfrac{r_{outer}}{r_{inner}}\right)}}}{r}$$

for an input power of $\mathbf{P}_{av}$ deduced from the time-average power flow.

The antenna radiates into the tissue where a damped wave propagates. Because you can discretize only a finite region, you must truncate the geometry some distance from the antenna using a similar absorbing boundary condition without excitation. Apply this boundary condition to all exterior boundaries.

DOMAIN AND BOUNDARY EQUATIONS—HEAT TRANSFER

The bioheat equation describes the time-dependent heat transfer problem as

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = \rho_b C_b \omega_b (T_b - T) + Q_{met} + Q_{ext}$$

where k is the liver's thermal conductivity (W/(m·K)), ρ_b represents the blood density (kg/m^3), C_b is the blood's specific heat capacity (J/(kg·K)), ω_b denotes the blood perfusion rate (1/s), and T_b is the arterial blood temperature (K). Further, Q_{met} is the heat source from metabolism, and Q_{ext} is an external heat source, both measured in W/m^3.

The initial temperature equals T_b in all domains.

This model neglects the heat source from metabolism. The external heat source is equal to the resistive heat generated by the electromagnetic field:

$$Q_{ext} = \frac{1}{2}\mathrm{Re}[(\sigma - j\omega\varepsilon)\mathbf{E} \cdot \mathbf{E}^*]$$

The model assumes that the blood perfusion rate is $\omega_b = 0.0036\ \mathrm{s}^{-1}$, and that the blood enters the liver at the body temperature $T_b = 37$ °C and is heated to a temperature, T. The blood's specific heat capacity is $C_b = 3639$ J/(kg·K).

For a more realistic model, you might consider letting ω_b be a function of the temperature. At least for external body parts such as hands and feet, it is evident that a temperature increase results in an increased blood flow.

This example models the heat-transfer problem only in the liver domain. Where this domain is truncated, it uses insulation, that is

$$\mathbf{n} \cdot \nabla T = 0$$

In addition to the heat transfer equation, this model computes the tissue damage integral. This gives an idea about the degree of tissue injury α during the process, based on the Arrhenius equation:

$$\frac{d\alpha}{dt} = A \exp\left(-\frac{dE}{RT}\right)$$

where A is the frequency factor (s^{-1}) and dE is the activation energy for irreversible damage reaction (J/mol). These two parameters are dependent on the type of tissue. The fraction of necrotic tissue, θ_d, is then expressed by:

$$\theta_d = 1 - \exp(-\alpha)$$

Results and Discussion

Figure 3 shows the resulting steady-state temperature distribution in the liver tissue for an input microwave power of 10 W. The temperature is highest near the antenna. It then decreases with distance from the antenna and reaches 37 °C closer to the outer boundaries of the computational domain. The perfusion of relatively cold blood seems to limit the extent of the area that is heated.

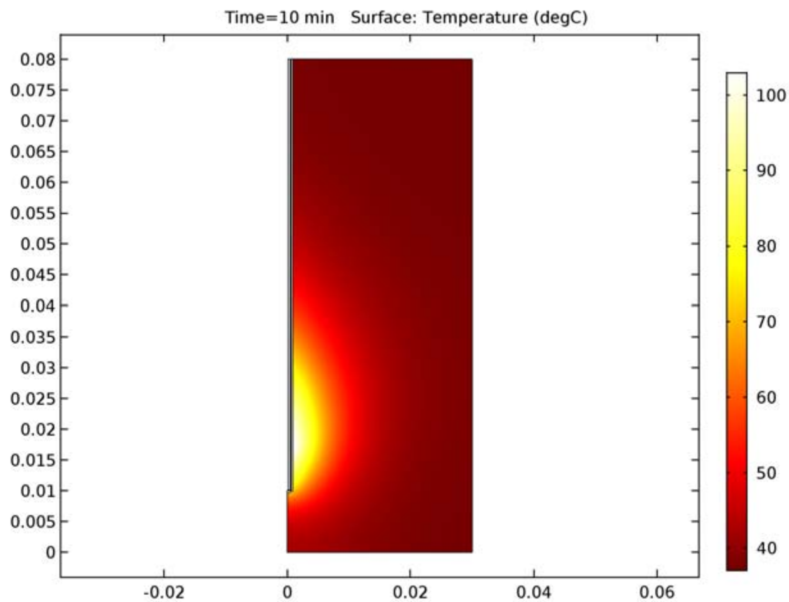


Figure 3: Temperature in the liver tissue.

Figure 4 shows the distribution of the microwave heat source. Clearly the temperature field follows the heat-source distribution quite well. That is, near the antenna the heat source is strong, which leads to high temperatures, while far from the antenna, the heat source is weaker and the blood manages to keep the tissue at normal body temperature.

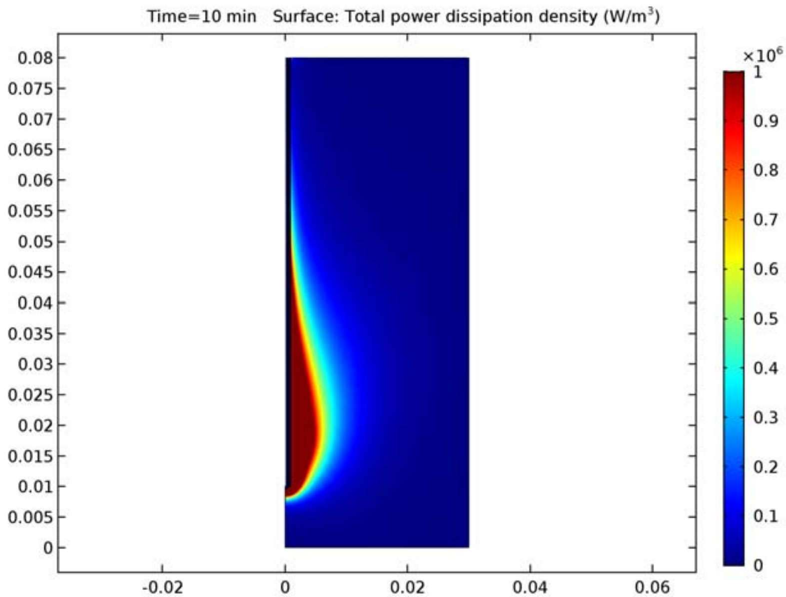


Figure 4: The computed microwave heat-source density takes on its highest values near the tip and the slot. The scale is cut off at 1 W/cm³.

Figure 5 plots the specific absorption rate (SAR) along a line parallel to the antenna and at a distance of 2.5 mm from the antenna axis. The results are in good agreement with those found in Ref. 1.

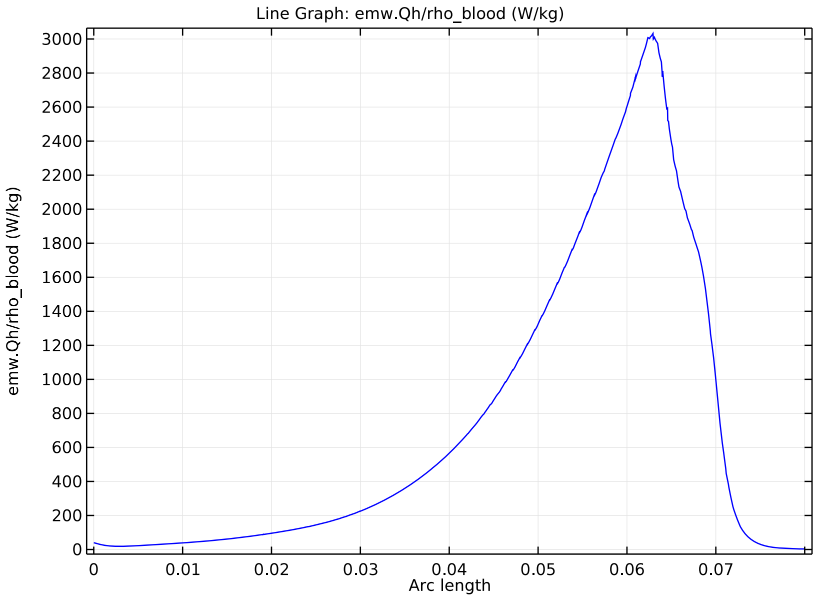


Figure 5: SAR in W/kg along a line parallel to the antenna and at a distance 2.5 mm from the antenna axis. The tip of the antenna is located at 70 mm, and the slot is at 65 mm.

You can visualize the fraction of necrotic tissue in the surface plot of Figure 6.

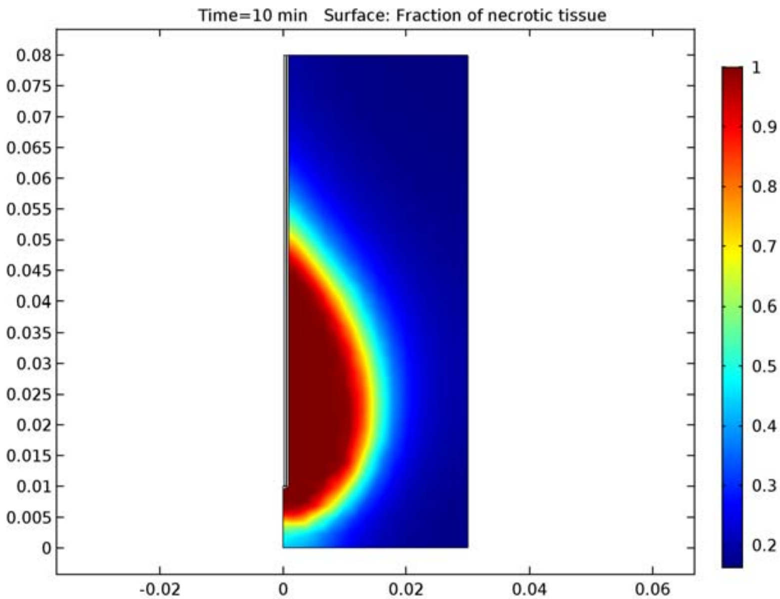


Figure 6: Fraction of necrotic tissue.

Figure 7 shows the fraction of necrotic tissue at four different point of the domain. Observe that necrosis happens faster at the antenna area.

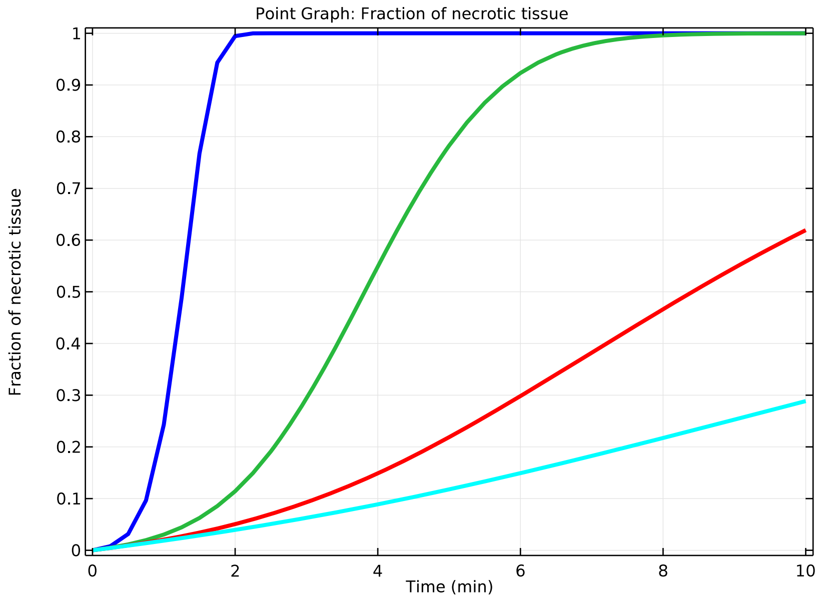


Figure 7: Fraction of necrotic tissue at four points of the domain.

Reference

1. K. Saito, T. Taniguchi, H. Yoshimura, and K. Ito, "Estimation of SAR Distribution of a Tip-Split Array Applicator for Microwave Coagulation Therapy Using the Finite Element Method," *IEICE Trans. Electronics*, vol. E84-C, 7, pp. 948–954, July 2001.

Application Library path: `Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy`

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 **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Heat Transfer>Bioheat Transfer (ht)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Custom Studies>Preset Studies for Some Physics Interfaces>Frequency Domain**.

   The frequency domain study is for the electromagnetic part of the model. You add a transient analysis for the heat transfer part before solving.
8 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.
2 In the **Settings** window for Import, locate the **Import** section.
3 Click **Browse**.
4 Browse to the application's Application Libraries folder and double-click the file `microwave_cancer_therapy.mphbin`.
5 Click **Import**.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

   The relevant material properties and other model data are provided in a text file.
2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `microwave_cancer_therapy_parameters.txt`.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Bioheat>Liver (human)**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Liver (human) (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Liver (human) (mat1)**.

2 Select Domain 1 only.

3 In the **Settings** window for Material, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	`eps_liver`	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	`sigma_liver`	S/m	Basic

The remaining materials take part only in the RF simulation, making any definitions of their thermal properties redundant.

Material 2 (mat2)

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 `Catheter` in the **Label** text field.

3 Select Domain 2 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	`eps_cat`	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Dielectric` in the **Label** text field.

3 Select Domain 3 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	`eps_diel`	1	Basic
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat4)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat4)**.

3 Select Domain 4 only.

The bioheat equation applies only in the liver tissue.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for Electromagnetic Waves, Frequency Domain, click to expand the **Equation** section.

3 From the **Equation form** list, choose **Frequency domain**.

4 From the **Frequency** list, choose **User defined**. In the *f* text field, type `f`.

Because you will use a study with a **Frequency Domain** step for a single frequency followed by a **Time Dependent** step, this is a convenient way to define the frequency in the second study step.

Port 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Port**.

2 Select Boundary 8 only.

3 In the **Settings** window for Port, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Coaxial**.

5 From the **Wave excitation at this port** list, choose **On**.

6 In the P_{in} text field, type `P_in`.

Scattering Boundary Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

2 Select Boundaries 2, 17, 19, and 20 only.

BIOHEAT TRANSFER (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Bioheat Transfer (ht)**.

2 In the **Settings** window for Bioheat Transfer, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

Biological Tissue 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Biological Tissue 1**.

2 In the **Settings** window for Biological Tissue, locate the **Damaged Tissue** section.

3 Select the **Include damage integral analysis** check box.

4 From the **Damage integral form** list, choose **Energy absorption**.

Bioheat 1

1 In the **Model Builder** window, expand the **Biological Tissue 1** node, then click **Bioheat 1**.

2 In the **Settings** window for Bioheat, locate the **Bioheat** section.

3 In the T_b text field, type `T_blood`.

4 In the ρ_b text field, type `rho_blood`.

5 In the $C_{p,\,b}$ text field, type `Cp_blood`.

6 In the ω_b text field, type `omega_blood`.

You have now supplied all the parameters needed for the heat removal by the blood perfusion.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `T_blood` in the T text field.

MULTIPHYSICS

Electromagnetic Heat Source 1 (emh1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Domain>Electromagnetic Heat Source**.

2 In the **Settings** window for Electromagnetic Heat Source, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

This brings the heat created by the electromagnetic waves to the heat transfer simulation.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 **Maximum element size** text field, type `3[mm]`.

Free Triangular 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

Size 1

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Domain**.

3 Select Domain 3 only.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.15[mm]`.

7 Click **Build All**.

The mesh is now reasonably fine everywhere, and especially fine in the coaxial cable, where the wave is created.

STUDY 1

Step 1: Frequency Domain

1 In the **Settings** window for Frequency Domain, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `f`.

3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Bioheat Transfer (ht)** interface.

Add a transient analysis for the heat transfer problem.

Time Dependent

On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

Step 2: Time Dependent

1 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

2 From the **Time unit** list, choose **min**.

3 In the **Times** text field, type `range(0,15[s],10)`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Electromagnetic Waves, Frequency Domain (emw)** interface.

5 On the **Study** toolbar, click **Compute**.

RESULTS

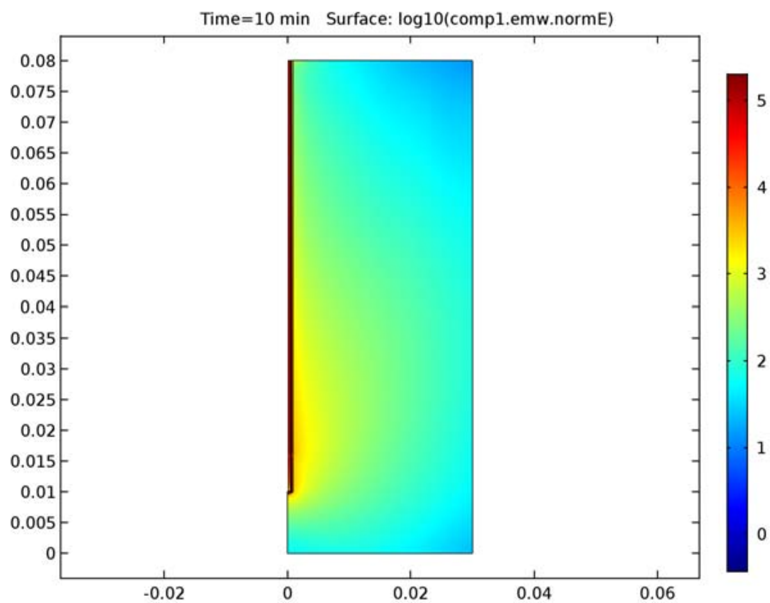
Electric Field (emw)

You have now solved the model first for the electromagnetic wave distribution, then for the temperature distribution resulting from the electromagnetic heating. Such a sequential solution is faster and consumes less memory than a fully coupled analysis, but works only if the material properties do not depend on the temperature.

Surface

The default plot shows the distribution of the electric field norm. The range is dominated by the locally very high values in and in the near vicinity of the coaxial cable. One way to get a more useful picture is to plot the logarithm of the field.

1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Surface**.
2 In the **Settings** window for Surface, locate the **Expression** section.
3 In the **Expression** text field, type `log10(comp1.emw.normE)`.
4 On the **Electric Field (emw)** toolbar, click **Plot**.
5 Click the **Zoom Extents** button on the **Graphics** toolbar.



The local heating power density is an important result of this model. As it is proportional to the electric field squared, this entity is also going to have a very uneven distribution. Manually specifying the range is another option to keep the plot readable.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qh - Total power dissipation density**.
7 On the **Electric Field (emw)** toolbar, click **Plot**.

8 Click to expand the **Range** section. Select the **Manual color range** check box.

9 In the **Maximum** text field, type `1e6`.

10 On the **Electric Field (emw)** toolbar, click **Plot**.

11 Click the **Zoom Extents** button on the **Graphics** toolbar.

Any values greater than 1 MW/m^3 are now displayed as red.

If you divide the power loss density with the density of the liver tissue, you get the SAR. Try plotting this on a vertical line some distance away from the antenna. Take the liver density to be the same as that of blood.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for Cut Line 2D, locate the **Line Data** section.

3 In row **Point 1**, set **r** to `2.5e-3`, and **z** to `0.08`.

4 In row **Point 2**, set **r** to `2.5e-3`.

1D Plot Group 4

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Qh/rho vs Arc Length` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

4 From the **Time selection** list, choose **Last**.

Line Graph 1

1 On the **Qh/rho vs Arc Length** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `emw.Qh/rho_blood`.

4 On the **Qh/rho vs Arc Length** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The plot you just created should look like Figure 5.

Derived Values

To evaluate the total deposited power, integrate the power loss in the liver domain.

Surface Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 In the **Settings** window for Surface Integration, locate the **Data** section.

3 From the **Time selection** list, choose **Last**.

4 Select Domain 1 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qh - Total power dissipation density**.

6 Locate the **Integration Settings** section. Select the **Compute volume integral** check box.

7 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

As shown in the table, the tissue absorbs most of the 10 W input power.

RESULTS

Create a new plot group for the surface plot of the temperature in the tissue (Figure 3).

2D Plot Group 5

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Temperature, 2D` in the **Label** text field.

Surface 1

1 Right-click **Temperature, 2D** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Bioheat Transfer>Temperature>T - Temperature**.

3 Locate the **Expression** section. From the **Unit** list, choose **degC**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

5 On the **Temperature, 2D** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Temperature, 3D (ht)

Click the **Zoom Extents** button on the **Graphics** toolbar.

2D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Generate a plot to show the fraction of necrotic tissue in 2D.

2 In the **Settings** window for 2D Plot Group, type `Damaged Tissue, 2D` in the **Label** text field.

Surface 1

1 Right-click **Damaged Tissue, 2D** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Bioheat Transfer> Biological tissue>ht.theta_d - Fraction of necrotic tissue**.

3 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

4 On the **Damaged Tissue, 2D** toolbar, click **Plot**.

Cut Point 2D 1

1 On the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for Cut Point 2D, locate the **Point Data** section.

3 In the **r** text field, type `ge(0.0050,0.0050,0.02)`.

4 In the **z** text field, type `2e-2`.

5 In the **r** text field, type `range(0.0050,0.0050,0.02)`.

6 Click **Plot**.

1D Plot Group 7

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Temperature, 1D` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.

Point Graph 1

1 On the **Temperature, 1D** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Bioheat Transfer>Temperature>T - Temperature**.

3 On the **Temperature, 1D** toolbar, click **Plot**.

1D Plot Group 8

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Damaged Tissue, 1D` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.

Point Graph 1

1 On the **Damaged Tissue, 1D** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Bioheat Transfer>Biological tissue>ht.theta_d - Fraction of necrotic tissue**.

3 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type 3.

4 On the **Damaged Tissue, 1D** toolbar, click **Plot**.

Mixed Diffuse-Specular Radiation Benchmark

This tutorial shows how to use the Mathematical Particle Tracing interface to simulate radiative heat transfer with mixed diffuse-specular reflection between surfaces in an enclosure. This application is separated in two parts. The first part compares the heat fluxes computed by the Mathematical Particle Tracing interface with the exact solution for two identical infinitely long parallel gray plates under mixed diffuse-specular reflection at constant temperature. The second part couples the Mathematical Particle Tracing interface with the Heat Transfer in Solids interface for the parallel plate geometry but with different characteristics and spatially varying temperatures.

Introduction

Gray surfaces in an enclosure can reflect radiant energy specularly, that is, they reflect light like a mirror. This is particularly true for optically smooth surfaces like clean metals and glassy materials. For these materials the reflectance of a surface can be adequately represented by a combination of a diffuse and a specular component.

Following Ref. 1, the reflectance ρ of a surface can be expressed as

$$\rho = \rho^s + \rho^d = 1 - \varepsilon = 1 - \alpha$$

Where ρ^s and ρ^d are respectively the specular and diffusive reflectance of the surface, and where the symbols ε and α are the emissivity and absorptivity of the surface.

The heat flux q at surfaces in an enclosure is then defined by:

$$q = J - (1 - \rho^s)H$$

Where J is the surface radiosity

$$J = \varepsilon E_b + \rho^d H$$

E_b is the blackbody emissive power

$$E_b = \sigma T^4$$

and H is the irradiance

$$H = \int_A J(\mathbf{r}') dF^s_{dA - dA'} + H^s_0$$

The latter expression depends on the external irradiation H^s_0 and on the differential specular view factor $dF^s_{dA - dA'}$.

For more details on the terminology see Ref. 1.

Model Definition

The model uses the Mathematical Particle Tracing interface to model radiative heat transfer using a discrete transfer method. The heat flux at each boundary element on the surface is computed by sending rays outward from the surface to query the temperatures of other surfaces in the geometry. The following three features are used:

The Inlet feature is used to determine the irradiance of a surface by backward ray tracing. Particles, which represent rays, are released uniformly on the inlet surfaces using a constant velocity. Particles are release uniformly within a hemisphere in velocity space (in 3D) or a semicircle (in 2D) centered about the surface normal.

In 2D the irradiance per ray H_{ij} is defined as

$$H_{ij} = \frac{1}{2}E_b\cos(\theta)d\theta$$

where E_b is the blackbody emissive power of the surface at which the ray arrives and θ is the acute angle between the initial particle velocity vector and the surface normal. The angle $\Delta\theta$ is the plane angle subtended by each ray. The additional factor $1/2$ is used for normalization purposes and compensates for the use of $\cos(\theta)$ to assign weights to different rays based on their angles of incidence; this is validated by the integral

$$\int_{-\pi/2}^{\pi/2}\frac{1}{2}\cos(\theta)d\theta = 1$$

Similarly in 3D, the irradiance per ray is defined as

$$H_{ij} = \frac{1}{\pi}E_b\cos(\theta)d\theta$$

where $\Delta\theta$ is the solid angle subtended by each ray. The validity of the correction factor $1/\pi$ is confirmed by the integral

$$\int_0^{2\pi}\int_0^{\pi/2}\frac{1}{\pi}\cos(\theta)\sin(\theta)(d\theta)d\phi = 1$$

The Nonlocal Accumulator subnode, which can be added to the Inlet node, transmits the value of a variable at the particle's current position and communicates this information back to the mesh element from which the particle was released, where it can be used to change the value of a dependent variable. With the Nonlocal Accumulator it is possible to map the irradiance per ray to the mesh elements from which the rays are initially released. For a mesh element i, the irradiance is

$$H_i = \sum_{j=1}^{N} H_{ij}$$

Where N is the number of rays released from the mesh element i.

The Wall node is used to make particles freeze, reflect diffusely, or reflect specularly at boundaries. The irradiance per ray is only set to a nonzero value when a ray is frozen to the wall, so the study must continue for enough time for all particles to freeze. The time a ray (particle) take to be absorbed depends on the emittance and reflectance of the walls.

The following algorithm is implemented to fulfill the first equation above:

1. Generate a random number $\mathrm{rn1}$ between 0 and 1.
2. If $\mathrm{rn1} > \rho$ the ray is absorbed (Freeze condition).
3. If $\mathrm{rn1} \le \rho$ generation of a second random number $\mathrm{rn2}$ between 0 and 1.
4. If $\mathrm{rn2} < \rho^{\mathrm{S}}/\rho$ the ray undergoes specular reflection (Bounce condition).
5. If $\mathrm{rn2} \ge \rho^{\mathrm{S}}/\rho$ the ray undergoes diffuse reflection with a probability distribution based on Knudsen's cosine law (Diffuse scattering condition).

The model is separated into two parts.

In the first part, we compare the exact analytical solution to the numerical result obtained with the Mathematical Particle Tracing interface.

This computes the heat flux at the surfaces of two identical infinitely long (out-of-plane on Figure 1) parallel plates placed in cold surroundings with mixed diffuse-specular radiation at their surfaces.

The geometry is illustrated in Figure 1. For the benchmark model the lower and upper plates have the same temperature $T_l = T_u = T$, the same emittance $\varepsilon_l = \varepsilon_u = \varepsilon$, and the same probability of specular reflection $\gamma_l = \gamma_u = \gamma$.

Using symmetries, it is possible to determine the heat flux $(q_l = q_u = q)$ on the lower and upper plates using the following analytical solution see Ref. 1.

$$1-(1-\rho^s)\int_{-W/2}^{W/2}\frac{1}{2}\sum_{k=1}^{\infty}\frac{(\rho^s)^{k-1}k^2d\xi'}{(k^2+(\xi-\xi')^2)^{3/2}} =$$

$$\frac{\Psi(\xi)}{\varepsilon}-\frac{\rho^d}{\varepsilon}\left(\int_{-W/2}^{W/2}\Psi(\xi')\frac{1}{2}\sum_{k=1}^{\infty}\frac{(\rho^s)^{k-1}k^2d\xi'}{(k^2+(\xi-\xi')^2)^{3/2}}\right)$$

Where $\xi = x/d$, $W = w/d$ and $\Psi = q/E_b$. The heat flux can be computed using numerical quadrature; a typical solution is presented on Figure 2.

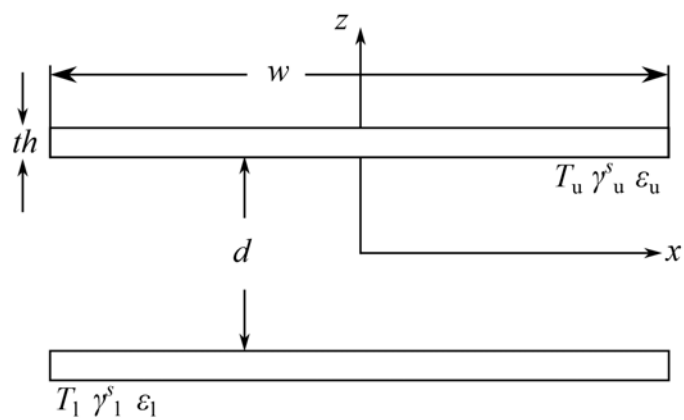


Figure 1: Schematics of the problem. The width of the plates are w = 10 cm, their thickness th = w/20, and the distance between the plates set to d = 10 cm. The temperature, emittance and probability of specular reflection are equal for both plates and respectively set to T = 300 K, ε = 0.6, and γ^s = 0.8.

In the second part, we keep the parallel plates arrangement (same geometry) but change the surface properties and couple the radiation model developed above to the Heat Transfer in Solids interface. Table 1 displays the surface parameters used for this part of the application.

TABLE 1: SURFACE PARAMETERS

Surfaces	ε	γ^s
Surrounding	1	0

TABLE 1: SURFACE PARAMETERS

Surfaces	ε	γ^s
Lower plate	0.9	0.1
Upper plate	0.6	0.8

For this model, the upper plate, made of copper, is heated locally from the top. The lower plate, made of quartz, is heated by the radiation emitted from the upper plate and cooled by natural convection on the bottom surface. The plates' surrounding temperature is set to 300 K.

Results and Discussion

Figure 2 shows a comparison of the normalized heat flux at the plate's surfaces for the exact and ray tracing solutions (benchmark model). A good agreement is observed between the curves. Because the Diffuse scattering wall condition is stochastic in nature, the solutions on the top and bottom surfaces differ slightly from each other and from the exact solution.

When the heat source computed using the Mathematical Particle Tracing interface is coupled to the Heat Transfer in Solids interface, the temperature field shown in Figure 3 is obtained. The temperature on the surface of the bottom plate is plotted in Figure 4. Figure 5 displays the normalized heat flux at the top of the lower plate (blue) and at the bottom of the upper plate (green).

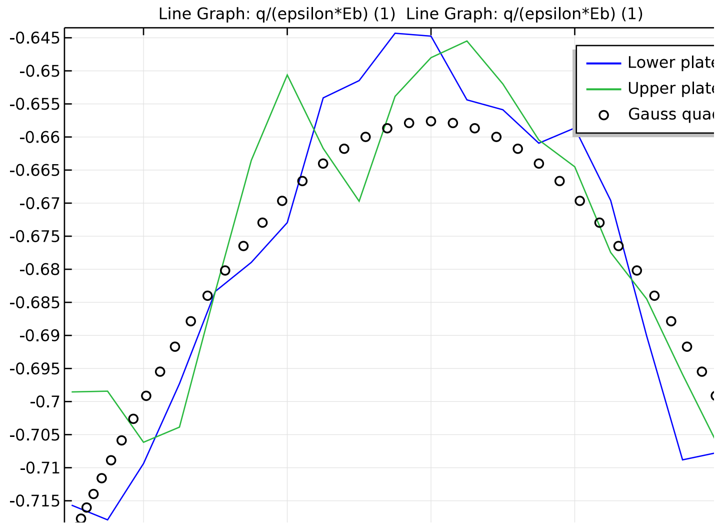


Figure 2: Normalized heat flux at the top of the lower plate (blue) and at the bottom of the upper plate (green) for T = 300 K, ε = 0.6, γ^S = 0.8 and w/d = 1. The black circles represent the exact solution (the same for both surfaces) obtained from numerical quadrature see Ref. 1.

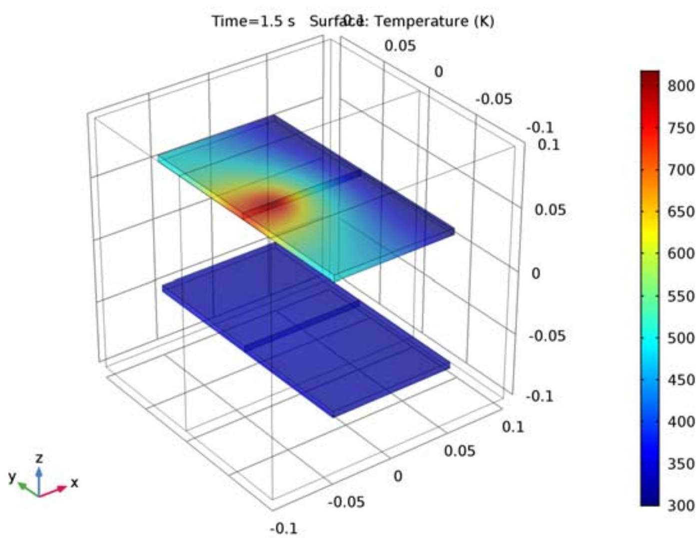


Figure 3: Temperature at the surface of the plates for the coupled model.

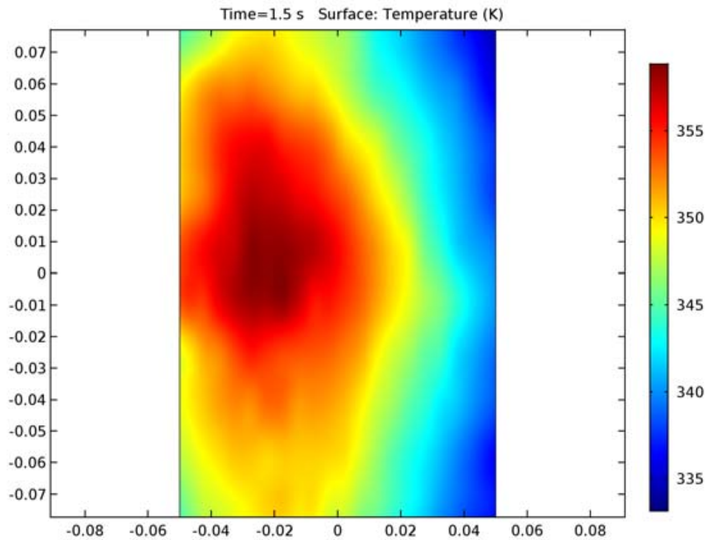


Figure 4: Temperature at the surface of the lower plate for the coupled model.

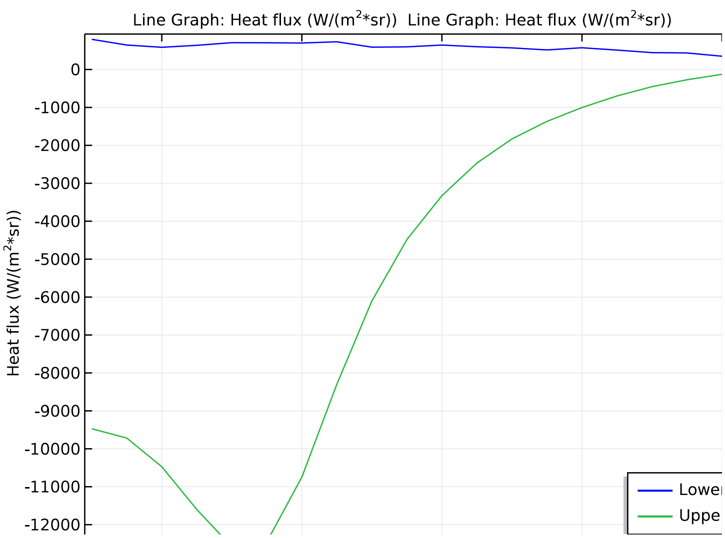


Figure 5: Heat flux at the top of the lower plate (blue) and at the bottom of the upper plate (green) for the coupled model.

Notes About the COMSOL Implementation

This 3D model uses a bounce boundary condition to simulate the effect of infinitely long plates (symmetry conditions).

An auxiliary dependent variable is necessary to define the release angle of each ray.

The second study consists of two study steps, a Stationary study step to compute the temperature and a Time Dependent study step to compute the particle trajectories and radiative heat flux. A self-consistent solution is obtained via an iterative process in which a For-End For loop is used to alternate between the stationary and time-dependent solvers. The loop should be continued until the change in temperature at each successive iteration is negligibly small. For this application, an acceptable self-consistent solution is obtained in three iterations.

Reference

1. M. F. Modest, *Radiative Heat Transfer*, 2nd. ed., Academic Press, 2003.

Application Library path: `Heat_Transfer_Module/Thermal_Radiation/parallel_plates_diffuse_specular`

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 **Mathematics>Mathematical Particle Tracing (pt)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
w	10[cm]	0.1 m	Width of the plates
d	10[cm]	0.1 m	Distance between the plates
l	2*w	0.2 m	Length of the plates
th	w/20	0.005 m	Thickness of the plates
M	20	20	Number of ray bundles along the width of the plates (number of elements)
N	300	300	Number of rays per bundle
Delta_theta	2*pi/N	0.02094	Solid angle subtended per ray
T0	300[K]	300 K	Room temperature

GEOMETRY 1

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `2*w`.

4 In the **Depth** text field, type `l`.

5 In the **Height** text field, type `2*w`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `w`.

4 In the **Depth** text field, type `l`.

5 In the **Height** text field, type `th`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **z** text field, type `-(d+th)/2`.

Block 3 (blk3)

1 Right-click **Block 2 (blk2)** and choose **Duplicate**.

2 In the **Settings** window for Block, locate the **Position** section.

3 In the **z** text field, type `(d+th)/2`.

4 Click the **Transparency** button on the **Graphics** toolbar in order to see the entire geometry.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

Partition the lower upper plate in two sections. The lines created by the partition will be used to display the heat flux across the plates' width.

2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

Partition Objects 1 (par1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Click the **Select Box** button on the **Graphics** toolbar.

3 Click in the **Graphics** window and then press Ctrl+A to select all objects.

4 In the **Settings** window for Partition Objects, locate the **Partition Objects** section.

5 From the **Partition with** list, choose **Work plane**.

6 Click **Build All Objects**.

Define a selection list for the surrounding, lower and upper plate.

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Surrounding` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1–5, 7–9, 32, and 33 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Lower Plate` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 10–13, 18, 20, 21, 26, 28, and 30 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Upper Plate` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 14–17, 22, 24, 25, 27, 29, and 31 only.

Define the model variables.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for Variables, type `Definitions` in the **Label** text field.

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
`H_ij`	`nojac(1/pi*sigma_const*bndenv(Ts)^4*abs(cos(theta_emit))*Delta_theta)`		`Irradiance per ray`
`Eb`	`sigma_const*Ts^4`		`Blackbody emissive power`
`rho`	`1-epsilon`		`Surface reflectance`
`q`	`-epsilon*(Eb-pt.inl1.nacc1.rpi)`		`Heat flux`

Variables 2

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for Variables, type `Surrounding: Study 1` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Surrounding**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
`Ts`	`0[K]`	K	`Temperature of the surrounding`

Name	Expression	Unit	Description
epsilon	1		Emittance of the surrounding
gamma_s	0		Probability of specular reflection

Variables 3

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for Variables, type `Lower Plate: Study 1` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Lower Plate**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
Ts	T0	K	Temperature of the lower plate
epsilon	0.6		Emittance of the lower plate
gamma_s	0.8		Probability of specular reflection

Variables 4

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for Variables, type `Upper Plate: Study 1` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Upper Plate**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
Ts	T0	K	Temperature of the upper plate
epsilon	0.6		Emittance of the upper plate
gamma_s	0.8		Probability of specular reflection

Variables 5

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for Variables, type `Surrounding: Study 2` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Surrounding**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
Ts	T0	K	Temperature of the surrounding
epsilon	1		Emittance of the surrounding
gamma_s	0		Probability of specular reflection

Variables 6

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for Variables, type `Lower Plate: Study 2` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Lower Plate**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
Ts	T		Temperature of the lower plate
epsilon	0.9		Emittance of the lower plate
gamma_s	0.1		Probability of specular reflection

Variables 7

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for Variables, type `Upper Plate: Study 2` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Upper Plate**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
`Ts`	`T`		`Temperature of the upper plate`
`epsilon`	`0.6`		`Emittance of the upper plate`
`gamma_s`	`0.8`		`Probability of specular reflection`

Now define the mathematical particle tracing model. Set the **Maximum number of secondary particles** to zero and avoid allocating unnecessary degrees of freedom to the problem.

MATHEMATICAL PARTICLE TRACING (PT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mathematical Particle Tracing (pt)**.

2 In the **Settings** window for Mathematical Particle Tracing, locate the **Advanced Settings** section.

3 In the **Maximum number of secondary particles** text field, type `0`.

4 Locate the **Domain Selection** section. Click **Clear Selection**.

5 Select Domains 1 and 2 only.

Auxiliary Dependent Variable 1

1 On the **Physics** toolbar, click **Global** and choose **Auxiliary Dependent Variable**.

Add an auxiliary dependent variable that will be used to compute the released angle of the particles.

2 In the **Settings** window for Auxiliary Dependent Variable, locate the **Auxiliary Dependent Variable** section.

3 In the **Field variable name** text field, type `theta_emit`.

4 Locate the **Units** section. Find the **Dependent variable quantity** subsection. From the list, choose **Plane angle (rad)**.

Add an inlet to the surface to which we want to compute the flux, i.e. the lower and upper plate surfaces with the exception of the bottom surface of the lower plate and the top surface of the upper plate.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundaries 10, 13, 14, 16, 18, 21, 22, 24, and 28–31 only.

3 In the **Settings** window for Inlet, locate the **Initial Velocity** section.

4 From the **Initial velocity** list, choose **Constant speed, hemispherical** in order to have an equidistant distribution of rays.

Enter the number of rays per bundle in the **Number of particles in velocity space** field.

5 In the N_{vel} text field, type N.

6 Select the **Specify tangential and normal vector components** check box.

7 Specify the $\mathbf{r}$ vector as

0	t1
0	t2
1	n

Enter the built-in variable for the release angle in the **theta_emit0** field.

8 Locate the **Initial Value of Auxiliary Dependent Variables** section. In the $\mathrm{theta}_e\mathrm{mit}_0$ text field, type `pt.inl1.thetarel`.

Add a nonlocal accumulator to map the computed irradiance per ray to the constant discontinuous Lagrange elements associated to the release sites (in this case at the center of the mesh elements).

Nonlocal Accumulator 1

1 On the **Physics** toolbar, click **Attributes** and choose **Nonlocal Accumulator**.

2 In the **Settings** window for Nonlocal Accumulator, locate the **Accumulator Settings** section.

3 From the **Accumulator type** list, choose **Count**.

4 In the R text field, type `H_ij`.

5 From the **Source geometric entity level** list, choose **Boundaries**.

6 Locate the **Units** section. Find the **Dependent variable quantity** subsection. From the list, choose **Radiative intensity (W/(m^2*sr))**.

Add the bounce boundary condition at the extremities of the domain (symmetry condition). Note that the effect of temperature and emittance of the boundary set in the surrounding variables have no effect of the computation as the walls are, here, purely specular.

Wall 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundaries 2 and 9 only.

3 In the **Settings** window for Wall, locate the **Wall Condition** section.

4 From the **Wall condition** list, choose **Bounce**.

Select the remaining wall boundaries and set the probability of diffuse and specular reflection as well as the probability of absorption of the rays (rho). This wall boundary condition computes the probability of specular reflection (bounce otherwise Knudsen cosine law) if the ray hasn't been absorbed before. The probability of absorption is entered as `rho` in the **Probability** field of the **Primary Particle Condition** section.

Wall 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundaries 1, 3–5, 7, 8, 10, 12–14, 16–18, 20–22, 24, 25, and 28–33 only.

3 In the **Settings** window for Wall, locate the **Wall Condition** section.

4 From the **Wall condition** list, choose **Mixed diffuse and specular reflection**.

5 In the γ_s text field, type `gamma_s`.

6 Locate the **Primary Particle Condition** section. From the **Primary particle condition** list, choose **Probability**.

7 In the γ text field, type `rho`.

To save on computation time, create a simple mesh on one extremity of the domain and sweep it over the entire domain using the swept mesh feature.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

Mapped 1

Select Boundaries 11 and 15 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 16, 18, 21, and 23 only.

3 In the **Settings** window for Distribution, locate the **Distribution** section.

4 In the **Number of elements** text field, type `M`.

Mapped 1

Right-click **Mapped 1** and choose **Distribution**.

Distribution 2

1 Select Edge 19 only.

2 Click the **Zoom Box** button on the **Graphics** toolbar.

3 Select Edges 14, 19, 40, and 43 only.

4 In the **Settings** window for Distribution, locate the **Distribution** section.

5 In the **Number of elements** text field, type `1`.

Free Triangular 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundary 2 only.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

4 Right-click **Mesh 1** and choose **Swept**.

Swept 1

In the **Settings** window for Swept, click **Build All**.

STUDY 1

Generate the default solver sequence and enter a proper maximum time step for the time dependent solver. The particles must travel a distance lower than the distance between the walls for a given time step.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. Select the **Maximum step** check box.

5 In the associated text field, type `0.01`.

Enter 1.5 s as the final time step. This will give enough time for the majority of the particle to freeze.

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 **Times** text field, type `0,1.5`.

Select the boundary definitions for the first study.

4 In the **Model Builder** window, click **Step 1: Time Dependent**.

5 In the **Settings** window for Time Dependent, locate the **Physics and Variables Selection** section.

6 Select the **Modify physics tree and variables for study step** check box.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Surrounding: Study 2**.

8 Click **Disable**.

9 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Lower Plate: Study 2**.

10 Click **Disable**.

11 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Upper Plate: Study 2**.

12 Click **Disable**.

13 In the **Model Builder** window, click **Study 1**.

14 In the **Settings** window for Study, locate the **Study Settings** section.

15 Clear the **Generate default plots** check box.

16 On the **Study** toolbar, click **Compute**.

RESULTS

Load the heat flux data computed by Gauss quadrature. These data represent an exact solution of the model.

Table 1

1 On the **Results** toolbar, click **Table**.

2 In the **Settings** window for Table, type `epsilon=0.6, gamma_s=0.8` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

4 Browse to the application's Application Libraries folder and double-click the file `parallel_plates_diffuse_specular_data.txt`.

5 Click **Update**.

1D Plot Group 1

1 On the **Results** toolbar, click **1D Plot Group**.

Generate the heat flux comparison. By symmetry, the heat flux at the upper and lower plate should be the same.

2 In the **Settings** window for 1D Plot Group, type `Validation` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **Last**.

Line Graph 1

On the **Validation** toolbar, click **Line Graph**.

Validation

1 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

2 In the **Expression** text field, type `q/(epsilon*Eb)`.

3 Locate the **Selection** section. Select the **Active** toggle button.

4 Select Edge 28 only.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x/w`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Lower plate`

10 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

11 Right-click **Line Graph 1** and choose **Duplicate**.

12 In the **Settings** window for Line Graph, locate the **Legends** section.

13 In the table, enter the following settings:

Legends
`Upper plate`

14 Locate the **Selection** section. Select the **Active** toggle button.

15 Click **Clear Selection**.

16 Select Edge 31 only.

17 In the **Model Builder** window, click **Validation**.

Table Graph 1

On the **Validation** toolbar, click **Table Graph**.

Validation

1 In the **Settings** window for Table Graph, locate the **Coloring and Style** section.

2 Find the **Line style** subsection. From the **Line** list, choose **None**.

3 From the **Color** list, choose **Black**.
4 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
5 From the **Positioning** list, choose **In data points**.
6 Click to expand the **Legends** section. Select the **Show legends** check box.
7 From the **Legends** list, choose **Manual**.
8 In the table, enter the following settings:

Legends
Gauss quadrature

9 On the **Validation** toolbar, click **Plot**.

Now add a **Heat Transfer in Solids** interface to the model. Here we are going to couple the radiative heat transfer to the heat transfer in the plates. For this model the top of the upper plate (copper) is heated by a local heat source with one side held at a constant temperature. The bottom of the lower plate (glass) is cooled by convection.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.
5 Click **Add to Component** in the window toolbar.

HEAT TRANSFER IN SOLIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.
2 In the **Settings** window for Heat Transfer in Solids, locate the **Domain Selection** section.
3 In the list, choose **1** and **2**.
4 Click **Remove from Selection**.
5 Select Domains 3–6 only.

Add the material properties to each plate.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, type `Quartz` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

3 Select Domains 3 and 5 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`1.1[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`2200[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`480[J/(kg*K)]`	J/(kg·K)	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Copper` in the **Label** text field.

3 Select Domains 4 and 6 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`400[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`8700[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`385[J/(kg*K)]`	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Initial Values 1

1 In the **Model Builder** window, expand the **Heat Transfer in Solids (ht)** node, then click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `T0` in the T text field.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

Add a fixed temperature on one side of the upper plate.

2 Select Boundaries 29 and 31 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `T0`.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

Add a localized heat flux on the top of the upper plate.

2 Select Boundaries 17 and 25 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 In the q_0 text field, type `5e6[W/m^2]*exp(-((x+0.025[m])^2+y^2)/0.0001[m^2])`.

Heat Flux 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

Add a convective flux on the bottom of the lower plate.

2 Select Boundaries 12 and 20 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 In the h text field, type `10`.

6 In the T_{ext} text field, type `T0`.

Heat Flux 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

Add the radiative heat flux computed by the **Mathematical Particle Tracing** interface.

2 Select Boundaries 10, 13, 14, 16, 18, 21, 22, 24, and 28–31 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 In the q_0 text field, type `q`.

Add a second study to solve the coupled model.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Mathematical Particle Tracing (pt)** interface.

5 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Add Study** in the window toolbar.

Start by adding a stationary study for the **Heat Transfer in Solids** interface only and select the boundary conditions associated to the second study.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

3 Click to collapse the **Study extensions** section. Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Surrounding: Study 1**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Lower Plate: Study 1**.

7 Click **Disable**.

8 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Upper Plate: Study 1**.

9 Click **Disable**.

Then add a time dependent study for the **Mathematical Particle Tracing** only.

Time Dependent

1 On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

Use the same time steps as in **Study 1**.

Step 2: Time Dependent

1 In the **Settings** window for Time Dependent, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Heat Transfer in Solids (ht)** interface.

3 Locate the **Study Settings** section. In the **Times** text field, type `0,1.5`.

Select the boundary definitions for the second study.

4 In the **Model Builder** window, click **Step 2: Time Dependent**.

5 In the **Settings** window for Time Dependent, locate the **Physics and Variables Selection** section.

6 Select the **Modify physics tree and variables for study step** check box.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Surrounding: Study 1**.

8 Click **Disable**.

9 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Lower Plate: Study 1**.

10 Click **Disable**.

11 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions>Upper Plate: Study 1**.

12 Click **Disable**.

Generate the default study sequence for the two steps defined above.

13 In the **Model Builder** window, click **Study 2**.

14 In the **Settings** window for Study, locate the **Study Settings** section.

15 Clear the **Generate default plots** check box.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

Add a for loop and move the generated sequence in the loop. The purpose of the for loop is to match the temperature given by the **Heat Transfer in Solids** interface to the surface temperature used by the radiative model.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 Right-click **Solution 2 (sol2)** and choose **Programming>For**.

Three loops are expected to be sufficient to match the radiative flux with the **Heat Transfer in Solids** interface with a negligible error.

4 In the **Settings** window for For, locate the **General** section.

5 In the **Number of iterations** text field, type 3.

Move the **For** node on top of the study sequence. This will include the sequence in the loop.

6 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>For 1** and choose **Move Up**.

7 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>For 1** and choose **Move Up**.

8 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>For 1** and choose **Move Up**.

9 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>For 1** and choose **Move Up**.

10 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>For 1** and choose **Move Up**.

11 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>For 1** and choose **Move Up**.

12 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>For 1** and choose **Move Up**.

Generate the initial radiative heat flux for the **Heat Transfer in Solids** interface computation.

13 Right-click **Solution 2 (sol2)** and choose **Compile Equations**.

14 In the **Settings** window for Compile Equations, locate the **Study and Step** section.

15 From the **Use study step** list, choose **Step 2: Time Dependent**.

Move the **Compile Equations: Stationary** node on top of the sequence above the **For** node.

16 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

17 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

18 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

19 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

20 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

21 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

22 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

23 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

24 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Compile Equations: Stationary (3)** and choose **Move Up**.

25 Right-click **Solution 2 (sol2)** and choose **Dependent Variables**.

26 In the **Settings** window for Dependent Variables, type `Dependent Variables 0` in the **Label** text field.

Move the **Dependent Variables** node in between the **Compile Equations** node and the **For** node.

27 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

28 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

29 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

30 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

31 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

32 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

33 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

34 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

35 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 0** and choose **Move Up**.

36 Locate the **General** section. From the **Defined by study step** list, choose **User defined**.

37 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)** click **Compile Equations: Time Dependent (3)**.

38 In the **Settings** window for Compile Equations, type `Compile Equations: Time Dependent 0` in the **Label** text field.

Use the initial solution given by the **Mathematical Particle Tracing** interface as **Values of variables not solved for** for the **Heat Transfer in Solids** interface computation.

39 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)** click **Dependent Variables 1**.

40 In the **Settings** window for Dependent Variables, locate the **General** section.

41 From the **Defined by study step** list, choose **User defined**.

42 Locate the **Values of Variables Not Solved For** section. From the **Method** list, choose **Solution**.

43 From the **Solution** list, choose **Solution 2 (sol2)**.

Use the solution given by the **Heat transfer in solids** interface as **Values of variables not solved for** for the **Mathematical Particle Tracing** interface computation.

44 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)** click **Dependent Variables 2**.

45 In the **Settings** window for Dependent Variables, locate the **General** section.

46 From the **Defined by study step** list, choose **User defined**.

47 Locate the **Initial Values of Variables Solved For** section. From the **Method** list, choose **Initial expression**.

48 From the **Solution** list, choose **Zero**.

49 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)** click **Time-Dependent Solver 1**.

50 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

51 Locate the **Time Stepping** section. Select the **Maximum step** check box.

52 In the associated text field, type `0.01`.

53 Click **Compute**.

Generate a plot of the surface temperatures.

RESULTS

3D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Right-click **3D Plot Group 2** and choose **Surface**.

5 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Temperature>T - Temperature**.

6 On the **3D Plot Group 2** toolbar, click **Plot**.

Create a surface dataset to display the temperature on the top of the lower plate.

Surface 1

On the **Results** toolbar, click **More Data Sets** and choose **Surface**.

Data Sets

1 In the **Settings** window for Surface, locate the **Parameterization** section.

2 From the **x- and y-axes** list, choose **xy-plane**.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Select Boundaries 13 and 21 only.

2D Plot Group 3

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Surface**.

3 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Temperature>T - Temperature**.

4 On the **2D Plot Group 3** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Generate a figure displaying heat flux across the width of the plates.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Time selection** list, choose **Last**.

5 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

On the **1D Plot Group 4** toolbar, click **Line Graph**.

1D Plot Group 4

1 Select Edge 28 only.

2 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `q`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `x`.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Lower plate`

9 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

10 Right-click **Line Graph 1** and choose **Duplicate**.

11 In the **Settings** window for Line Graph, locate the **Legends** section.

12 In the table, enter the following settings:

Legends
`Upper plate`

13 Locate the **Selection** section. Select the **Active** toggle button.

14 Click **Clear Selection**.

15 Select Edge 31 only.

16 On the **1D Plot Group 4** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Sun's Radiation Effect on Two Coolers Placed Under a Parasol

Introduction

A warm sunny day at the beach can be more enjoyable with a steady supply of frosty beverages. This example considers two Styrofoam coolers containing cold beverage cans sitting on a sandy beach. A parasol provides partial shade for one of the coolers over the course of the day. The difference in beverage temperature over time is computed. This tutorial demonstrates the usage of the external radiation source boundary condition, as well as how to model structures exposed to ambient conditions.

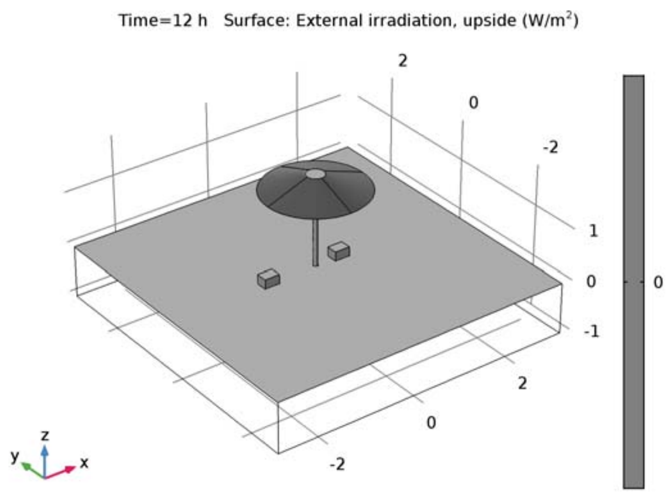


Figure 1: A parasol provides shade on the beach. Two Styrofoam coolers contain beverage cans that should remain as cold as possible.

Model Definition

A system of a parasol and two coolers is modeled as shown in Figure 1. The coolers, made of Styrofoam, contain six beverage cans each. The beverage cans are represented by water-filled cylinders with walls modeled as thin thermally resistive layers of aluminum. Because a slit is defined for the temperature on the walls, the temperature can differ between the inner and outer faces. This is used to define initial conditions that are discontinuous between the exterior and interior can surfaces. Because aluminum has higher thermal conductivity than the surrounding materials, the thin thermally resistive layer condition behaves like a continuity condition as soon as the initial temperature difference vanishes. The initial can temperature is 1 °C. The spacing between the cans and the cooler walls is small, so the model neglects free convection inside the cooler for simplicity.

The parasol primarily provides shade but otherwise has no significant thermal effect on the beverage temperature. For this reason, it is not too important to have a high fidelity model of the parasol. It is only the shadow cast by the parasol that contributes to the beverage temperature profile. The material used for the parasol is acrylic plastic.

The primary source of heat in this model is the solar irradiation, which is included using the External Radiation Source feature. This feature uses the longitude, latitude, time zone, time of year, and time of day to compute the direction of the incident solar radiation over the simulation time. A sunny location at the equator in the middle of the Pacific ocean is chosen for this analysis. Assuming no cloud cover, the solar flux at the surface is about 1000 W/m^2. All of the ambient surfaces of the model are included in the solar loading calculation, and shadowing effects are included.

The temperature of the sun is about 5800 K, and it emits primarily short-wavelength infrared and visible light at wavelengths shorter than 2.5 microns. The fraction of this short-wavelength solar radiation that is absorbed by the various materials is quantified by the solar absorptivity. Because the surfaces are at a much lower temperature, they reradiate in the long-wavelength infrared band, at wavelengths above 2.5 microns, and the fraction of reradiated energy is quantified by the surface emissivity. The solar and ambient wavelength dependence of emissivity model is used to account for differing emissivities in different wavelength bands.

There are three ambient temperature conditions in this model. First, the ground at 1 m below the sand surface is assumed to be at a constant temperature of 27 °C throughout the day, corresponding to the average water temperature at this location.

The second ambient condition is the surrounding air temperature. There exists a combination of free and forced convection, due to wind, from all exposed surfaces to the ambient air, the temperature of which is assumed to vary sinusoidally through the day. In this application, the Convective Heat Flux boundary condition uses a bulk heat transfer coefficient of 20 $W/(m^2 \cdot K)$ for all exposed surfaces.

The third boundary condition is the radiative view factor to ambient. The gray body radiative view factors are computed between all exposed faces in the model, and radiative heat transfer is computed between these faces. However, these computed view factors do not sum to unity. There is a significant view factor to surrounding regions that is not modeled; this is the residual view factor. The temperature of the ambient is the same as the ambient air temperature.

Results and Discussion

Figure 2 plots the temperature profile at 4 p.m. Observe the decrease of temperature where the parasol shade stands.

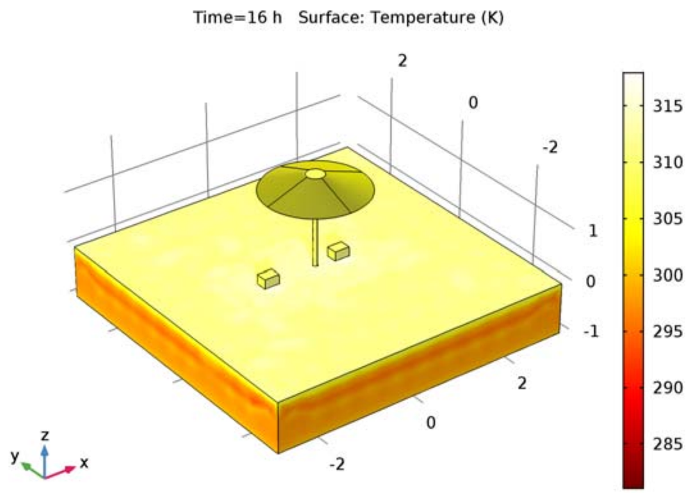


Figure 2: Temperature distribution.

Figure 3 plots the temperature of the beverage inside two of the cans. This shows clearly the advantage of placing the cooler in the shade. At 2 p.m., the parasol shade starts to leave the cooler corresponding to the green curve, which is responsible for the sudden variation in the temperature increase at that moment.

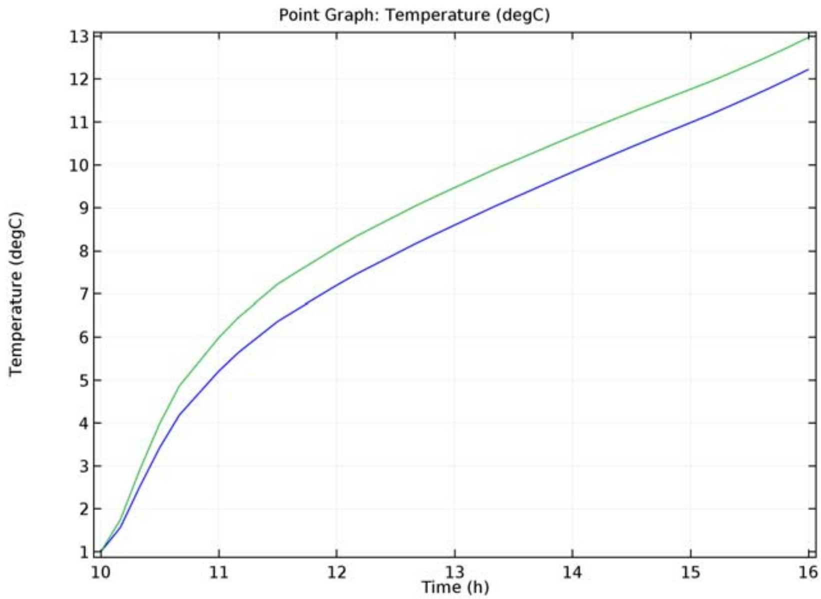


Figure 3: Beverage temperature over time inside of the two coolers at the left side of the parasol (blue curve) and at the right side (green curve).

Reference

1. F.P. Incropera, D.P. DeWitt, T.L. Bergman, and A.S. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th ed., John Wiley & Sons, 2006.

Application Library path: `Heat_Transfer_Module/Thermal_Radiation/parasol`

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 **Heat Transfer>Radiation>Heat Transfer with Surface-to-Surface Radiation (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

Define an analytic function for the time-dependent ambient temperature.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
Tavg	27[degC]	300.15 K	Average ambient temperature
dT	3[K]	3 K	Half diurnal temperature variation
dateDay	1	1	Day
dateMonth	1	1	Month
dateYear	2012	2012	Year

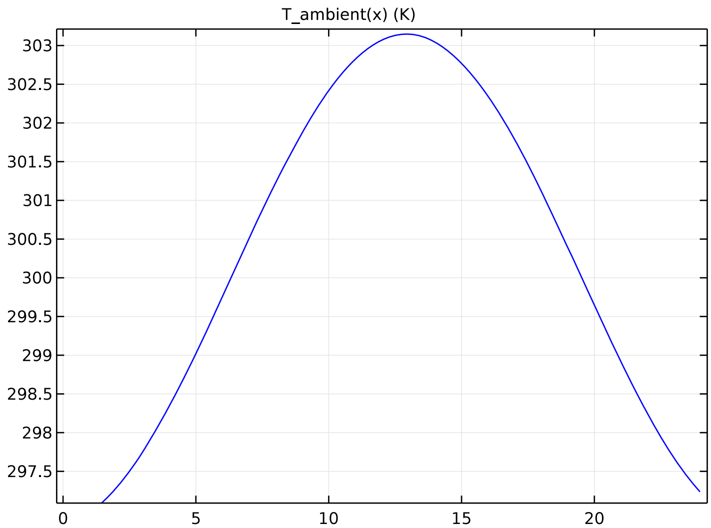
Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.
2 In the **Settings** window for Analytic, type `T_ambient` in the **Function name** text field.
3 Locate the **Definition** section. In the **Expression** text field, type `Tavg[1/K]+dT[1/K]*cos(2*pi*(x-14)/24)`.
4 Locate the **Units** section. In the **Arguments** text field, type `h`.
5 In the **Function** text field, type `K`.

6 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
x	0	24*3600

7 Click **Plot**.



GEOMETRY 1

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type 6.

4 In the **Depth** text field, type 6.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **z** text field, type -0.5.

7 Right-click **Block 1 (blk1)** and choose **Build Selected**.

This first block corresponds to a large region of sand. The next two blocks are the styrofoam coolers on the sand.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.3`.

4 In the **Depth** text field, type `0.22`.

5 In the **Height** text field, type `0.18`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `0.5`.

8 In the **z** text field, type `0.09`.

9 Right-click **Block 2 (blk2)** and choose **Build Selected**.

Block 3 (blk3)

1 Right-click **Block 2 (blk2)** and choose **Duplicate**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.26`.

4 In the **Depth** text field, type `0.18`.

5 In the **Height** text field, type `0.14`.

6 Right-click **Component 1 (comp1)>Geometry 1>Block 3 (blk3)** and choose **Build Selected**.

In the next few steps, you create two six-pack cans by building one cylinder that is duplicated in two 3x2 arrays. Because the cans are located inside the two styrofoam coolers, you need to enable the **Wireframe Rendering** option to see them.

7 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.03`.

4 In the **Height** text field, type `0.125`.

5 Locate the **Position** section. In the **x** text field, type `0.42`.

6 In the **y** text field, type `0.04`.

7 In the **z** text field, type `0.02`.

Array 1 (arr1)

1 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

3 Select the object **cyl1** only.

4 In the **Settings** window for Array, locate the **Size** section.

5 In the **x size** text field, type 3.

6 In the **y size** text field, type 2.

7 Locate the **Displacement** section. In the **x** text field, type 0.08.

8 In the **y** text field, type -0.08.

Copy 1 (copy1)

1 Right-click **Array 1 (arr1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

3 Select the objects **blk2**, **arr1(3,2,1)**, **arr1(1,1,1)**, **arr1(2,2,1)**, **arr1(2,1,1)**, **blk3**, **arr1(3,1,1)**, and **arr1(1,2,1)** only.

For more convenience, use the **Select Box** button to select the above-mentioned objects.

4 In the **Settings** window for Copy, locate the **Displacement** section.

5 In the **x** text field, type -1.5.

6 Right-click **Copy 1 (copy1)** and choose **Build Selected**.

Now, create the parasol.

Cone 1 (cone1)

1 On the **Geometry** toolbar, click **Cone**.

2 In the **Settings** window for Cone, locate the **Size and Shape** section.

3 In the **Height** text field, type 0.3.

4 From the **Specify top size using** list, choose **Angle**.

5 In the **Semi-angle** text field, type 70.

6 Locate the **Position** section. In the **z** text field, type 1.5.

7 Right-click **Cone 1 (cone1)** and choose **Build Selected**.

Cone 2 (cone2)

1 Right-click **Cone 1 (cone1)** and choose **Duplicate**.

2 In the **Settings** window for Cone, locate the **Position** section.

3 In the **z** text field, type 1.4.

Difference 1 (dif1)

1 Right-click **Component 1 (comp1)>Geometry 1>Cone 2 (cone2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

3 Select the object **cone1** only.

4 In the **Settings** window for Difference, locate the **Difference** section.

5 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

6 Select the object **cone2** only.

Cylinder 2 (cyl2)

1 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Cylinder**.

3 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

4 In the **Radius** text field, type `0.05`.

5 In the **Height** text field, type `1.7`.

Form Union (fin)

1 Right-click **Cylinder 2 (cyl2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Build All**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

The following selection gathers the boundaries of the twelve cans.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Beverage Can Walls` in the **Label** text field.

3 Select Domains 4–7, 9, 10, and 14–19 only.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

The next selection is for the irradiated surfaces.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Irradiated Surfaces` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 4, 6, 7, 9, 10, 38–40, 56–62, 65–72, 74, 75, and 118 only.

ADD MATERIAL

1 On 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.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat2)**.

2 Select Domains 3 and 13 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Acrylic plastic**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Acrylic plastic (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Acrylic plastic (mat3)**.

2 Select Domains 8 and 11 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Aluminum**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Aluminum (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat4)**.

2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Beverage Can Walls**.

Material 5 (mat5)

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 `Styrofoam` in the **Label** text field.

3 Select Domains 2 and 12 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	0.05	W/(m·K)	Basic
Density	rho	200	kg/m³	Basic
Heat capacity at constant pressure	Cp	1300	J/(kg·K)	Basic

Material 6 (mat6)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Sand` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	0.3	W/(m·K)	Basic
Density	rho	1500	kg/m³	Basic
Heat capacity at constant pressure	Cp	800	J/(kg·K)	Basic

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer with Surface-to-Surface Radiation (ht)**.

2 In the **Settings** window for Heat Transfer with Surface-to-Surface Radiation, locate the **Radiation Settings** section.

3 From the **Wavelength dependence of emissivity** list, choose **Solar and ambient**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer with Surface-to-Surface Radiation (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `T_ambient(t)` in the T text field.

Initial Values 2

1 On the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domains 2–7, 9, 10, and 12–19 only.

3 In the **Settings** window for Initial Values, type `1[degC]` in the T text field.

External Radiation Source 1

1 On the **Physics** toolbar, click **Global** and choose **External Radiation Source**.

2 In the **Settings** window for External Radiation Source, locate the **External Radiation Source** section.

3 From the **Source position** list, choose **Solar position**.

4 From the **Location defined by** list, choose **City**.

5 From the list, choose **Caracas, Venezuela**.

6 In the **Date** table, enter the following settings:

Day	Month	Year
dateDay	dateMonth	dateYear

7 In the **Local time** table, enter the following settings:

Hour	Minute	Second
0	0	0

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 3 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `27[degC]`.

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 In the **Settings** window for Diffuse Surface, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Irradiated Surfaces**.

4 Locate the **Ambient** section. In the T_{amb} text field, type `T_ambient(t)`.

5 Locate the **Surface Emissivity** section. Find the **Solar spectral band** subsection. From the ε_{B1} list, choose **User defined**. In the associated text field, type `0.2`.

6 Find the **Ambient spectral band** subsection. From the ε_{B2} list, choose **User defined**. In the associated text field, type `0.8`.

Diffuse Surface 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 In the **Settings** window for Diffuse Surface, locate the **Ambient** section.

3 In the T_{amb} text field, type `T_ambient(t)`.

4 Locate the **Surface Emissivity** section. Find the **Solar spectral band** subsection. From the ε_{B1} list, choose **User defined**. In the associated text field, type `0.94`.

5 Find the **Ambient spectral band** subsection. From the ε_{B2} list, choose **User defined**. In the associated text field, type `0.76`.

6 Select Boundary 4 only.

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for Thin Layer, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Beverage Can Walls**.

4 Locate the **Thin Layer** section. In the d_s text field, type `300[um]`.

Heat Flux 1

1 On 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 **Irradiated Surfaces**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `20`.

6 In the T_{ext} text field, type `T_ambient(t)`.

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 From the **Time unit** list, choose **h**.

4 Click **Range**.

5 In the **Range** dialog box, type `10` in the **Start** text field.

6 In the **Step** text field, type `10[min]`.

7 In the **Stop** text field, type `16`.

8 Click **Replace**.

The study starts at 10:00 a.m. and ends at 4:00 p.m. with timesteps of 10 minutes.

9 On the **Home** toolbar, click **Compute**.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

The first default plot shows the temperature distribution as in Figure 2.

Isothermal Contours (ht)

The second default plot shows the isothermal contours.

Radiosity (ht)

This default plot shows the surface radiosity. Proceed to plot the external irradiation at 2:00 p.m. and see the parasol shade as in Figure 1.

1 In the **Model Builder** window, under **Results** click **Radiosity (ht)**.

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Time (h)** list, choose **12**.

Surface

1 In the **Model Builder** window, expand the **Radiosity (ht)** node, then 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 **Model>Component 1>Heat Transfer with Surface-to-Surface Radiation>Radiation>External irradiation>ht.G_extB1u - External irradiation, upside**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayPrint**.

4 On the **Radiosity (ht)** toolbar, click **Plot**.

Next, observe the temperature of the beverages as in Figure 3.

1D Plot Group 4

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type `Temperature in the Coolers` in the **Label** text field.

Point Graph 1

1. On the **Temperature in the Coolers** toolbar, click **Point Graph**.
2. Select Points 41 and 126 only.

   For more convenience, you can click the **Paste Selection** button and paste the point numbers.
3. In the **Settings** window for Point Graph, locate the **y-Axis Data** section.
4. From the **Unit** list, choose **degC**.
5. On the **Temperature in the Coolers** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Phase Change

Introduction

This example demonstrates how to model a phase change and predict its impact on a heat transfer analysis. When a material changes phase, for instance from solid to liquid, energy is added to the solid. Instead of creating a temperature rise, the energy alters the material's molecular structure. Heat consumed or released by a phase change affects fluid flow, magma movement and production, chemical reactions, mineral stability, and many other earth-science applications.

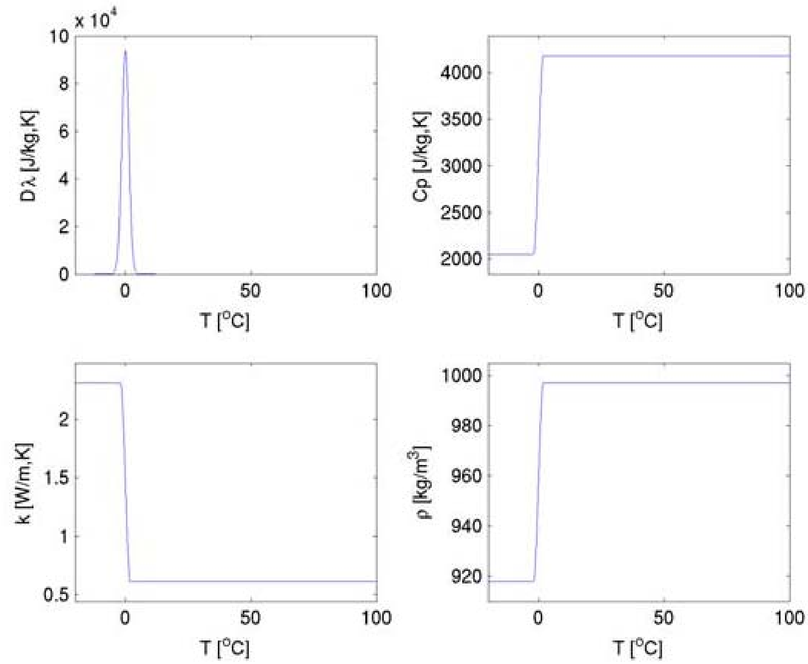


Figure 1: Material properties as functions of temperature.

This 1D example uses the Phase Change Material feature from the Heat Transfer Module to examine transient temperature transfer in a rod of ice that heats up and changes to water. In particular, the model demonstrates how to handle material properties that vary as a function of temperature.

This model proceeds as follows. First, estimate the ice-to-water phase change using the transient conduction equation with the latent heat of fusion. Next, compare the first solution to estimates that neglect latent heat. Finally, run additional simulations to evaluate impacts of the temperature interval over which the phase change occurs.

Model Definition

This example describes the ice-to-water phase change along a 1-cm rod of ice. At its left end the rod is insulated, and at the other temperature is maintained at 80 °C. Values for thermal properties depend on the phase. They are presented in Table 1, at 265 K for ice and 300 K for water.

TABLE 1: MATERIAL PROPERTIES OF ICE AND WATER

MATERIAL PROPERTY	ICE (AT 265 K)	WATER (AT 300 K)
Density	918 kg/m^3	997 kg/m^3
Heat capacity at constant pressure	2052 J/(kg·K)	4179 J(kg·K)
Thermal conductivity	2.31 W/(m·K)	0.613 W/(m·K)

The latent heat of fusion, l_m, is 333.5 kJ/kg and the rod is initially at –20 °C.

During the ice-to-water phase change, the density is modified, resulting in a volume compression. The material coordinates express all transformations in the initial coordinate system, when ice occupies all the domain. Assuming that there is no mixing in the liquid phase, the conduction equation in material coordinates can be used. It simplifies the model since you do not need to calculate the velocity field resulting from density variations during phase change. The conduction equation in material coordinates reads

$$\rho C_{eq}\frac{\partial T}{\partial t} + \nabla \cdot (-k_{eq}\nabla T) = Q \tag{1}$$

where ρ (kg/m^3) is the density, C_{eq} (J/kg·K) is the effective heat capacity at constant pressure, k_{eq} is the effective thermal conductivity (W/m·K), T is temperature (K), and Q is a heat source (W/m^3).

The material properties ρ and k_{eq} of water must be in material coordinates. Because values given in Table 1 come from measurements, they correspond to spatial coordinates. Hence, conversion into material coordinates is necessary. In 1D models, you just have to multiply by the ratio of densities, ρ_{ratio}:

$$\rho_{ratio} = \frac{\rho_{ice}}{\rho_{water}}$$

Note: With this transformation, the density of water, ρ, in material coordinates is $\rho = \rho_{water}\rho_{ratio} = \rho_{ice}$. This is consistent with conservation of mass because the integral of ρ over the geometry domain remains constant in time.

The boundary conditions for this model are

- thermal insulation at $x = 0$;
- fixed temperature at $x = 0.01$; the fixed temperature creates a temperature discontinuity at the starting time. You can thus replace T_{hot} by a smoothed step function T_{right} that increases the temperature from T_0 to T_{hot} in 0.1 s.

Results and Discussion

Figure 2 shows images of the temperature distribution in time, predicted with latent heat. The system is solid ice at $t = 0$, and water content increases with time.

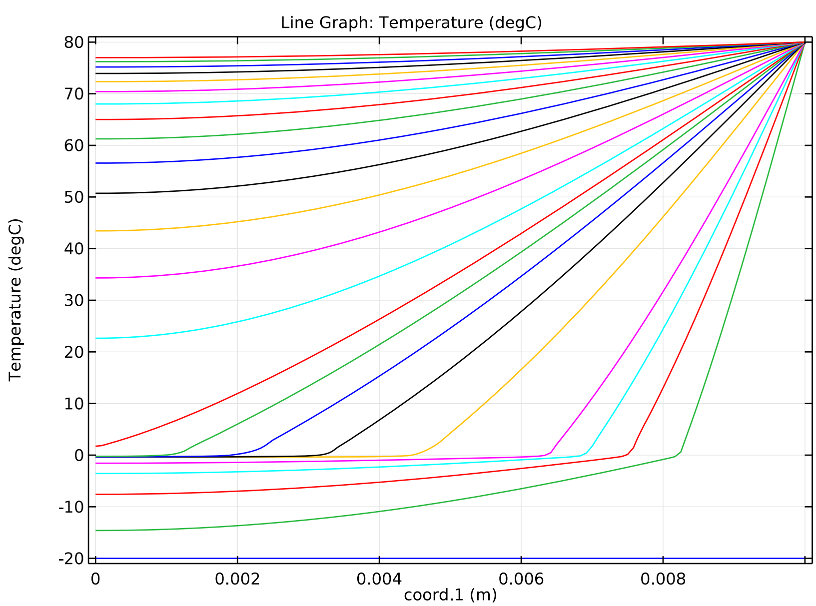


Figure 2: Temperature estimates with latent heat at t = 0 s, 15 s, 30 s, 45 s, 60 s, 2 min, 3 min, 4 min, …, 20 min.

The distributions all level out around the 0 °C temperature point because not all of the energy is going toward a temperature rise; some is being absorbed to change the molecular structure and change the phase.

The solution in Figure 3 shows temperature estimates for the simulation without latent heat.

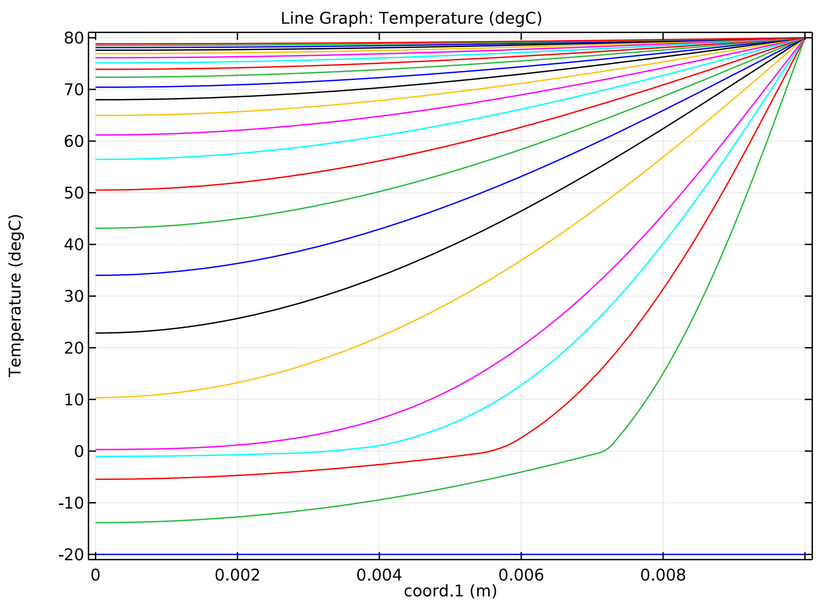


Figure 3: Temperature estimates without latent heat at t = 0 s, 15 s, 30 s, 45 s, 60 s, 2 min, 3 min, 4 min, …, 20 min.

A change of profile also occurs at 0 °C but is less visible. Because latent heat is not accounted for, this change is here due to the different thermophysical properties of water below and above 0 °C.

Figure 4 shows results for different solid-to-liquid intervals at three times. The smaller the interval, the sharper the bend in the temperature profile at zero temperature, T. In the simulations, narrowing the temperature interval to a step change, for example, comes at a large computational cost. In the figure, the results for the wide and narrow pulses compare closely.

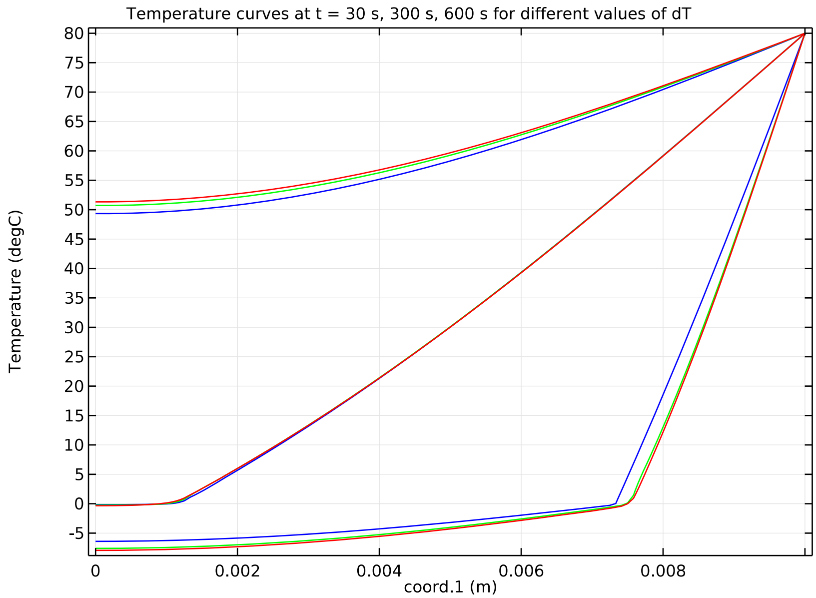


Figure 4: Temperature estimates for different temperature intervals for latent heat consumption. Estimates are for dT intervals of 0.5 K (blue), 1 K (green), and 2 K (red) at t = 30 s (three curves at bottom), 5 min (three curves at middle), and 10 min (three curves on top).

References

1. S.E. Ingebritsen and W.E. Sanford, *Groundwater in Geologic Processes*, Cambridge University Press, 1998.

2. N.H. Sleep and K. Fujita, *Principles of Geophysics*, Blackwell Science, 1997.

3. D.L. Turcotte and G. Schubert, *Geodynamics: Applications of Continuum Physics to Geological Problems*, 2nd ed., Cambridge University Press, 2002.

Application Library path: `Heat_Transfer_Module/Phase_Change/phase_change`

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 **1D**.
2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

   The **Heat Transfer in Fluids** interface with its **Phase Change Material** feature solves for the temperature and automatically calculates the equivalent conductivity and the equivalent specific heat capacity.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

Interval 1 (i1)

1 On the **Geometry** toolbar, click **Interval**.
2 In the **Settings** window for Interval, locate the **Interval** section.
3 In the **Right endpoint** text field, type `0.01`.

Form Union (fin)

1 Right-click **Interval 1 (i1)** and choose **Build Selected**.
2 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.
3 Click the **Zoom Extents** button on the **Graphics** toolbar.

GLOBAL DEFINITIONS

The following steps describe how the model parameters are defined.

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
T_trans	0[degC]	273.15 K	Transition temperature
dT	1[K]	1 K	Transition interval
lm	333.5[kJ/kg]	3.335E5 J/kg	Latent heat of fusion
T_0	-20[degC]	253.15 K	Initial temperature of the rod
T_hot	80[degC]	353.15 K	Temperature of hot water
rho_ice	918[kg/m^3]	918 kg/m³	Density of ice
rho_water	997[kg/m^3]	997 kg/m³	Density of water
rho_ratio	rho_ice/rho_water	0.92076	Ratio of densities

Step 1 (step1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Step**.

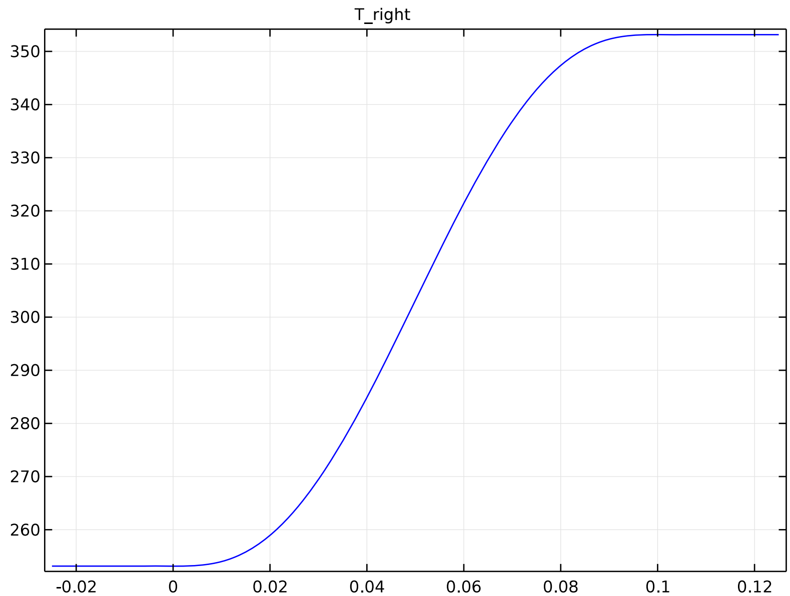
2 In the **Settings** window for Step, type `T_right` in the **Function name** text field.

3 Locate the **Parameters** section. In the **Location** text field, type `0.05`.

4 In the **From** text field, type `T_0`.

5 In the **To** text field, type `T_hot`.

6 Click to expand the **Smoothing** section. Click **Plot**.



MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, type `Ice` in the **Label** text field.

2 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	2.31	W/(m·K)	Basic
Density	rho	rho_ice	kg/m³	Basic
Heat capacity at constant pressure	Cp	2052	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Water` in the **Label** text field.

3 Select Domain 1 only.

Because the model is solved in material coordinates, the water density and thermal conductivity are converted.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`0.613[W/(m*K)]*rho_ratio`	W/(m·K)	Basic
Density	rho	`rho_water*rho_ratio`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`4179`	J/(kg·K)	Basic
Ratio of specific heats	gamma	`1`	1	Basic

HEAT TRANSFER IN FLUIDS (HT)

Phase Change Material 1

1 On the **Physics** toolbar, click **Domains** and choose **Phase Change Material**.

2 Select Domain 1 only.

3 In the **Settings** window for Phase Change Material, locate the **Phase Change** section.

4 In the $\Delta T_{1 \rightarrow 2}$ text field, type `dT`.

5 In the $L_{1 \rightarrow 2}$ text field, type `lm`.

6 Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Ice (mat1)**.

7 Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Water (mat2)**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `T_0` in the T text field.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 2 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `T_right(t[1/s])`.

MESH 1

Follow the steps below to generate a relatively fine mesh of 120 elements.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edge**.

Edge 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Edge 1** and choose **Distribution**.

Distribution 1

1 In the **Settings** window for Distribution, locate the **Distribution** section.

2 In the **Number of elements** text field, type `120`.

3 Click **Build Selected**.

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 Click **Range**.

4 In the **Range** dialog box, type `15` in the **Step** text field.

5 In the **Stop** text field, type `60`.

6 Click **Replace**.

7 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

8 Click **Range**.

9 In the **Range** dialog box, type `120` in the **Start** text field.

10 In the **Step** text field, type `60`.

11 In the **Stop** text field, type `1200`.

12 Click **Add**.

13 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

14 Select the **Relative tolerance** check box.

15 In the associated text field, type `0.001`.

16 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

All the parameter values in this model have a time unit of seconds, so the output time you enter here gives a total simulation time of 20 minutes. Different output intervals can be generated by adding other range commands as it is done above. Within the first minute, solution data is stored every 15 seconds, whereas for the remaining simulation period, the data is only stored every 60 seconds.

A line plot of the temperature distribution along the rod for all times is automatically produced. To generate Figure 2, you only need to change the temperature unit.

Line Graph

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Line Graph**.
2 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.
3 From the **Unit** list, choose **degC**.
4 On the **Temperature (ht)** toolbar, click **Plot**.

Phase Change Without Latent Heat

To analyze the impact of the latent heat terms on the phase change model, it is useful to estimate temperatures using the same approach but without the latent heat term. Therefore, latent heat `lm` is just set to zero. To keep the original value of 333.5 kJ/kg, introduce the parameter `lm_original`.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
lm	0	0	Latent heat of fusion
lm_original	333.5[kJ/kg]	3.335E5 J/kg	Latent heat of fusion, original

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

5 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Time Dependent

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

3 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

4 Click **Range**.

5 In the **Range** dialog box, type `60` in the **Stop** text field.

6 In the **Step** text field, type `15`.

7 Click **Replace**.

8 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

9 Click **Range**.

10 In the **Range** dialog box, type `120` in the **Start** text field.

11 In the **Stop** text field, type `1200`.

12 In the **Step** text field, type `60`.

13 Click **Add**.

14 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

15 Select the **Relative tolerance** check box.

16 In the associated text field, type `0.001`.

17 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht) 1

1 In the **Model Builder** window, under **Results** click **Temperature (ht) 1**.

2 In the **Settings** window for 1D Plot Group, type `Temperature, No Latent Heat` in the **Label** text field.

To generate Figure 3, you only need to change the units in the automatically generated temperature plot.

Line Graph

1 In the **Model Builder** window, expand the **Results>Temperature, No Latent Heat** node, then click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

3 From the **Unit** list, choose **degC**.

4 On the **Temperature, No Latent Heat** toolbar, click **Plot**.

To be able to keep track of the different studies, rename the data sets containing the solutions of **Study 1** and **Study 2**.

Study 1/Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

2 In the **Settings** window for Solution, type `Solution 1, lm Included` in the **Label** text field.

Study 2/Solution 2 (sol2)

1 In the **Model Builder** window, under **Results>Data Sets** click **Study 2/Solution 2 (sol2)**.

2 In the **Settings** window for Solution, type `Solution 2, lm Excluded` in the **Label** text field.

Phase Change for Varying Transition Intervals

Solutions to the phase change problem vary with the range in temperatures `dT` over which you assume that the phase transition occurs. To visualize the impact of different transition widths sample results from the original simulation and compare those estimates to results from simulations with varying `dT` values.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
lm	lm_original	3.335E5 J/kg	Latent heat of fusion

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

5 Click **Add Study** in the window toolbar.

STUDY 3

Step 1: Time Dependent

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.

3 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

4 Click **Range**.

5 In the **Range** dialog box, type 60 in the **Stop** text field.

6 In the **Step** text field, type 15.

7 Click **Replace**.

8 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

9 Click **Range**.

10 In the **Range** dialog box, type 120 in the **Start** text field.

11 In the **Stop** text field, type 1200.

12 In the **Step** text field, type 60.

13 Click **Add**.

14 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

15 Select the **Relative tolerance** check box.

16 In the associated text field, type 0.001.

Follow the steps below to calculate the temperature distribution of the rod for different values of the transition interval by just adding a parametric sweep to the study node. In this example, use the values 0.1 K, 0.5 K, and 2.5 K for dT.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
`dT`	`0.5 1 2`	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht) 1

Again, the temperature distribution along the rod for all time steps and `dT`-values is produced automatically. You can modify this plot to generate Figure 4 by following the steps below.

1 In the **Model Builder** window, under **Results** click **Temperature (ht) 1**.

2 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Temperature curves at t = 30 s, 300 s, 600 s for different values of dT`.

Line Graph

1 In the **Model Builder** window, expand the **Temperature (ht) 1** node, then click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Data set** list, choose **Study 3/Parametric Solutions 1 (sol4)**.

4 From the **Parameter selection (dT)** list, choose **First**.

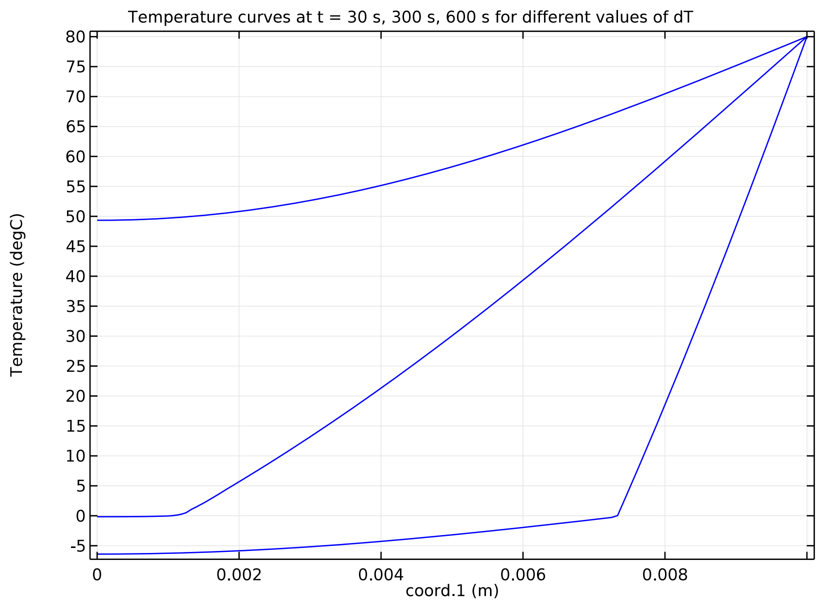
5 From the **Time selection** list, choose **Interpolated**.

6 In the **Times (s)** text field, type `30 300 600`.

7 Locate the **y-Axis Data** section. From the **Unit** list, choose **degC**.

8 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

9 On the **Temperature (ht) 1** toolbar, click **Plot**.



Line Graph 2

1 Right-click **Results>Temperature (ht) 1>Line Graph** and choose **Duplicate**.

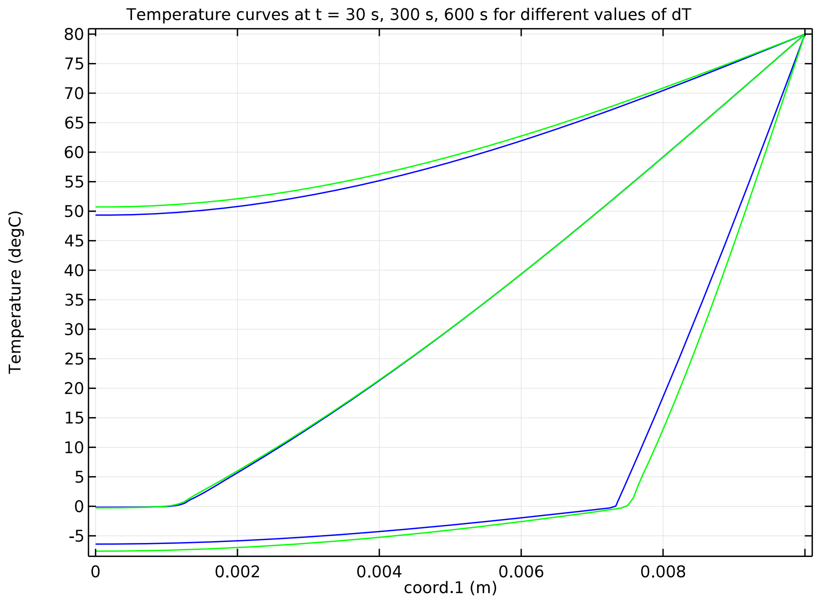
2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Parameter selection (dT)** list, choose **From list**.

4 In the **Parameter values (dT)** list, select **1**.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

6 On the **Temperature (ht) 1** toolbar, click **Plot**.



Line Graph

Right-click **Line Graph** and choose **Duplicate**.

Line Graph 3

1 In the **Settings** window for Line Graph, locate the **Data** section.

2 From the **Parameter selection (dT)** list, choose **Last**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

4 On the **Temperature (ht) 1** toolbar, click **Plot**.

Temperature (ht) 1

1 In the **Model Builder** window, under **Results** click **Temperature (ht) 1**.

2 In the **Settings** window for 1D Plot Group, type `Temperature, Varying dT` in the **Label** text field.

Power Transistor

Introduction

Transistors are building blocks of electronic appliances, and can be found in radios, computers, and calculators, to name a few. When working with electrical systems you typically have to deal with heat transfer; electric heating is often an unwanted result of current conduction.

This example simulates a system consisting of a small part of a circuit board containing a power transistor and the copper pathways connected to the transistor. The purpose of the simulation is to estimate the operating temperature of the transistor, which can be substantially higher than room temperature due to undesired electric heating.

Transistors are semiconductor devices used to switch or amplify electronic signals. There are different types of transistors, ranging in size depending on how they are packaged. Power transistors carry and dissipate more power and therefore come in larger packages. These packages can be attached to a heat sink for better cooling and to avoid overheating of the system.

The heat sink would then be attached to the transistor via the copper plate located behind the ceramic piece (shown in Figure 1 to the left). While it's often important to construct a way to cool electronic systems, such as in the case of components in hybrid cars, each system has its own acceptable operating temperature range. What determines the maximum and minimum temperature limits include the semiconductor material properties, the transistor type, the design of the device, and so forth. There is a conventional temperature range, however, which is thought to be between -55 °C and 125 °C.

Model Definition

Figure 1 shows the model geometry used in the simulation. The power transistor is mounted on the circuit board using through-hole technology. The solder in the holes give mechanical support and electronic contact between the copper routes and the transistor pins.

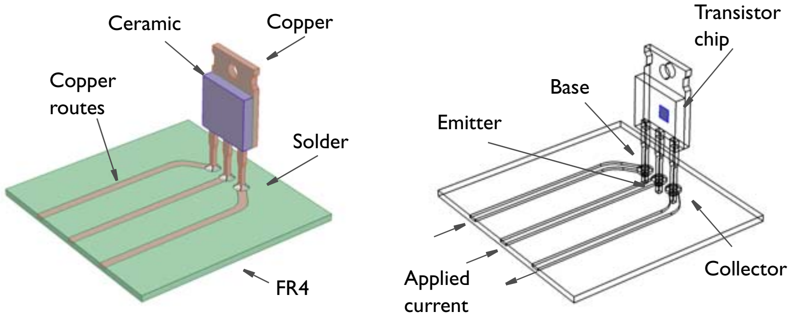


Figure 1: Model geometry and position of transistor chip.

The transistor chip itself is a very thin structure represented by an internal surface in Figure 1. The chip is connected to the pins but this connections are assumed to have negligible effects on heat transfer.

The transistor package front part is made of ceramics while the back part, which could be clamped to a heat sink, is made of copper. The transistor chip and the front part of the package have matching thermal properties. The copper pins are soldered to the circuit board by the solder material 60Sn-40Pb (60 % tin and 40 % lead). The circuit board is made of FR4.

Current conduction and Joule heating take place in the copper routes, in the solders, and in the pins. In these parts, the physics of heat transfer and heat production due to Joule heating are fully coupled to the conduction of electric current. In all other parts of the transistor, only heat transfer and heat production take place.

The transistor chip itself is represented by an internal boundary with an internal production of heat corresponding to 0.9 W. Cooling through convection takes place at all external boundaries with a heat transfer coefficient of 5.0 $W/(m^2 \cdot K)$. This value of the heat transfer coefficient corresponds to the worst case scenario when the fan is switched off. The ambient temperature is 293.15 K.

Current enters the circuit board at the left vertical boundaries of the copper routes connected to the base, emitter, and collector in Figure 1. The value of the current at the boundary of the route connected to the emitter is 0.12 A. The value of the current at the boundary of the route connected to the collector is 0.11988 A. The difference in absolute current between the emitter and collector currents corresponds to the current at the boundary of the route connected to the base, which is 0.12 mA.

Results and Discussion

Figure 2 below shows the temperature distribution in the device. The maximum temperature is about 354 K or 81 °C. This is well within the acceptable operation temperature range for the transistor, which implies that attaching it to a heat sink is not needed in this case.

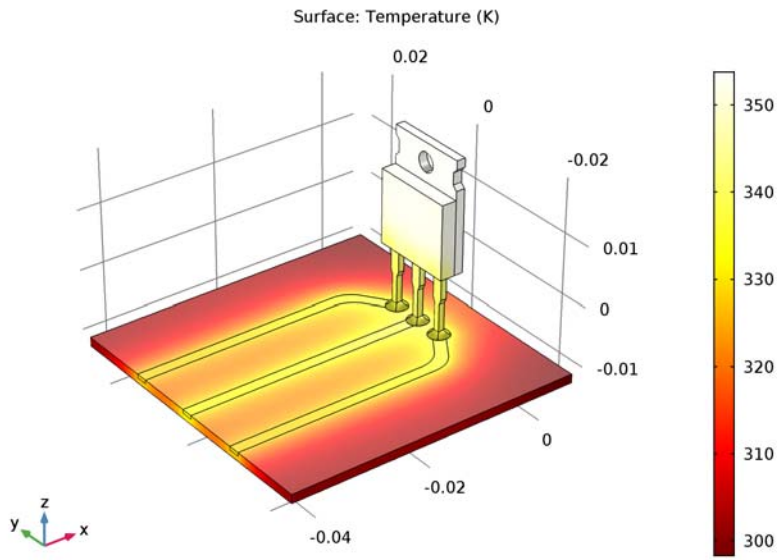


Figure 2: Temperature distribution.

Also worth noting is that electric heating, or Joule heating as it is also referred to, hardly influences the temperature of the copper routes at the distance from the transistor modeled above. That's most likely due to copper's high conductivity; some of the heat produced in the transistor chip is conducted away from the device via the copper routes. Figure 3 shows the temperature along the copper routes connected to the base and the collector respectively. The current density in the base is 1/1000 of that in the collector but the temperature in the copper routes connected to the base and collector is almost identical.

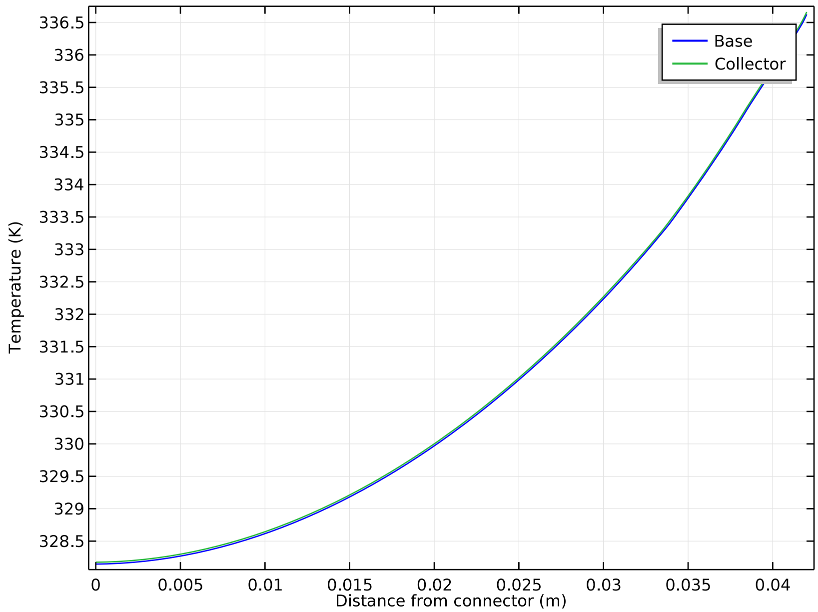


Figure 3: Temperature along the copper routes connected to the base and collector.

The fact that the Joule heating effect does not increase temperature in the copper routes leads to the conclusion that the higher temperature in these routes is due to coppers high conductivity. The copper routes conduct some of the heat produced in the transistor chip away from this device. The circuit board has a poor thermal conductivity and is therefore not heated to the same extent as the copper routes.

Notes About the COMSOL Implementation

You can find all the material properties for this application in COMSOL's Material Library. Furthermore, the ready-made physics interface for Joule heating sets up all model formulations that you need for the simulation: Electric Current and Heat Transfer in Solids are added with the corresponding Joule Heating coupling features in the Multiphysics node.

The Joule heating interface is by default available for all materials in the model. However, the circuit board material and the package material do no conduct electric current. For this reason, you have to edit the selection of Electric current physics to remove non-conducting domains. On the circuit board material, only heat transfer physics is calculated. By removing the non-conductive parts of the device to the list of Electric Current physics, Electromagnetic Heat Source and Boundary Electromagnetic Heat Source are automatically not applicable on these domains.

Application Library path: `Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/power_transistor`

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 **Heat Transfer>Electromagnetic Heating>Joule Heating**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

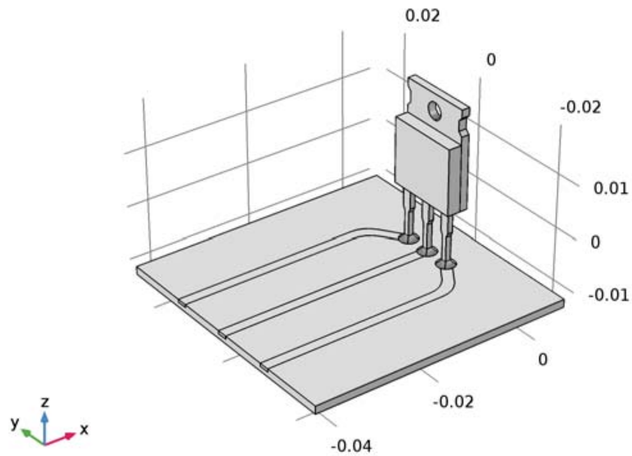
1 On the **Home** toolbar, click **Import**.

2 In the **Settings** window for Import, locate the **Import** section.

3 Click **Browse**.

4 Browse to the application's Application Libraries folder and double-click the file `power_transistor.mphbin`.

5 Click **Build All Objects**.



GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
j_CE	1e5[A/m^2]	1E5 A/m²	Current density, collector and emitter routes
Q_h	1e5[W/m^2]	1E5 W/m²	Boundary heat source strength
h_coeff	5[W/(m^2*K)]	5 W/(m²·K)	Heat transfer coefficient

ADD MATERIAL

1 On 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>Copper**.

4 Click **Add to Component** in the window toolbar.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>FR4 (Circuit Board)**.

3 Click **Add to Component** in the window toolbar.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Silica glass**.

3 Click **Add to Component** in the window toolbar.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Solder, 60Sn-40Pb**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Solder, 60Sn-40Pb (mat4)

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Copper (mat1)

1 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat1)**.

3 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

4 Click **Clear Selection**.

5 Select Domains 2–4 and 9–12 only.

FR4 (Circuit Board) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **FR4 (Circuit Board) (mat2)**.

2 Select Domain 1 only.

Silica glass (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silica glass (mat3)**.

2 Select Domain 7 only.

Solder, 60Sn-40Pb (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Solder, 60Sn-40Pb (mat4)**.

2 Select Domains 5, 6, and 8 only.

3 In the **Settings** window for Material, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	1	1	Basic

ELECTRIC CURRENTS (EC)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

2 Select Domains 2–6 and 8–11 only.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundaries 69, 87, and 105 only.

Normal Current Density 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Current Density**.

2 Select Boundary 10 only.

3 In the **Settings** window for Normal Current Density, locate the **Normal Current Density** section.

4 In the J_n text field, type `(1-1e-3)*j_CE`.

Normal Current Density 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Current Density**.

2 Select Boundary 5 only.

3 In the **Settings** window for Normal Current Density, locate the **Normal Current Density** section.

4 In the J_n text field, type `-j_CE`.

Normal Current Density 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Current Density**.

2 Select Boundary 15 only.

3 In the **Settings** window for Normal Current Density, locate the **Normal Current Density** section.

4 In the J_n text field, type `1e-3*j_CE`.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Heat Flux 1

1 On 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. Click the **Convective heat flux** button.

5 In the h text field, type `h_coeff`.

Boundary Heat Source 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Heat Source**.

2 Select Boundary 142 only.

3 In the **Settings** window for Boundary Heat Source, locate the **Boundary Heat Source** section.

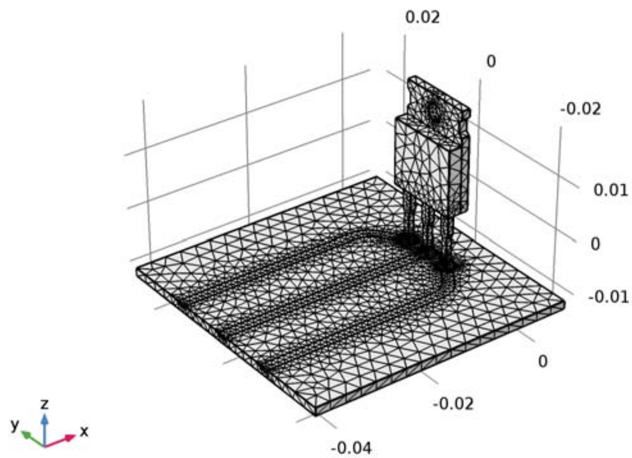
4 In the Q_b text field, type `Q_h`.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

Free Tetrahedral 1

1 In the **Settings** window for Free Tetrahedral, click **Build All**.



STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Electric Potential (ec)** node.

2 Right-click **Multislice 1** and choose **Delete**.

Electric Potential (ec)

In the **Model Builder** window, under **Results** right-click **Electric Potential (ec)** and choose **Surface**.

Surface 1

On the **Electric Potential (ec)** toolbar, click **Plot**.

Temperature (ht)

The second default plot shows the temperature. Add an arrow plot of the total heat flux.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Arrow Surface**.

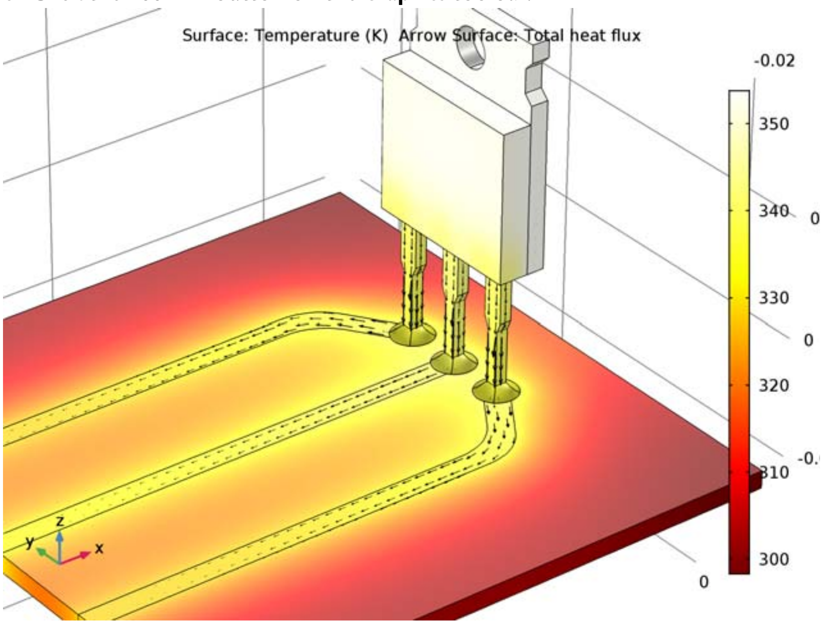
2 In the **Settings** window for Arrow Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Domain fluxes>ht.tfluxx,...,ht.tfluxz - Total heat flux**.

3 Locate the **Coloring and Style** section. In the **Number of arrows** text field, type `5e3`.

4 From the **Color** list, choose **Black**.

5 On the **Temperature (ht)** toolbar, click **Plot**.

6 Click the **Zoom In** button on the **Graphics** toolbar.



7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Finally, reproduce the plot in Figure 3 by following the steps outlined below.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Temperature along Copper Routes` in the **Label** text field.

Line Graph 1

1 On the **Temperature along Copper Routes** toolbar, click **Line Graph**.

2 Select Edges 28, 41, and 53 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 **Model>Component 1>Heat Transfer in Solids>Temperature>T - Temperature**.

4 Click to expand the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`Base`

7 On the **Temperature along Copper Routes** toolbar, click **Plot**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for Line Graph, locate the **Selection** section.

3 Select the **Active** toggle button.

4 Select Edges 14, 38, and 50 only.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Collector`

6 On the **Temperature along Copper Routes** toolbar, click **Plot**.

Temperature along Copper Routes

1 In the **Model Builder** window, under **Results** click **Temperature along Copper Routes**.

2 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Distance from connector (m)`.

6 On the **Temperature along Copper Routes** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Shell-and-Tube Heat Exchanger

Introduction

Shell-and-tube heat exchangers are one of the most widely used type of heat exchanger in the processing industries (65% of the market according to Ref. 1) and are commonly found in oil refineries, nuclear power plants, and other large-scale chemical processes. Additionally, they can be found in many engines and are used to cool hydraulic fluid and oil. In this application, two separated fluids at different temperatures flow through the heat exchanger: one through the tubes (tube side) and the other through the shell around the tubes (shell side). Several design parameters and operating conditions influence the optimal performance of a shell-and-tube heat exchanger.

The main purpose of this tutorial is to show the basic principles for setting up a heat exchanger model. It can also serve as a starting point for more sophisticated applications, such as parameter studies or adding additional effects like corrosion, thermal stress, and vibration.

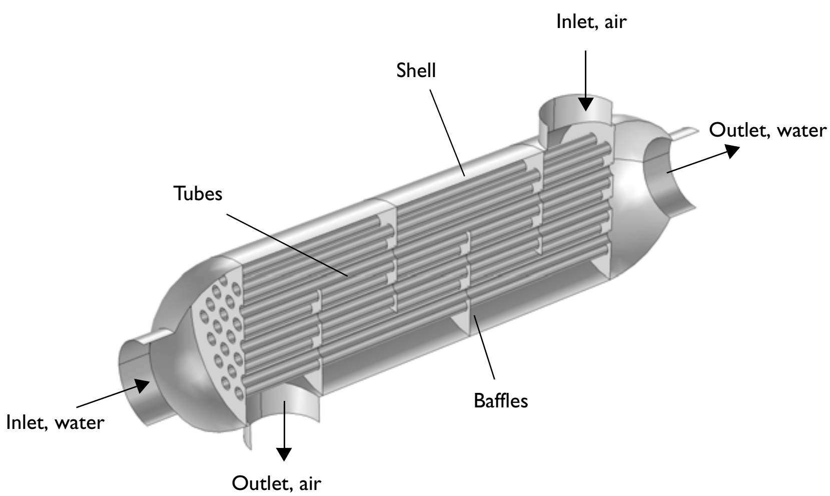


Figure 1: Geometry of the shell-and-tube heat exchanger.

Model Definition

The concept used to design a shell-and-tube heat exchanger is examined by exploring the working model of a straight, cross-flow, one pass shell-and-tube heat exchanger. The geometry of such a model is shown in Figure 1.

The heat exchanger is made of structural steel. In this example, two fluids pass through the heat exchanger. The first fluid, in this case water, flows through the tubes, while the second fluid, air, circulates within the shell of the heat exchanger but outside of the tubes. Both of these fluids have different starting temperatures when entering the heat exchanger, however after circulating within it, the fluids are brought closer to an equilibrium temperature. The baffles introduce some cross-flow to the air and such increasing the area of heat exchange. Another advantage is that baffles reduce vibration due to the fluid motion.

This model uses the Non-Isothermal Flow predefined multiphysics coupling configured with the k-ε turbulence model. It takes advantage of symmetries to model only one half of the heat exchanger, thereby reducing model size and computational costs.

BOUNDARY CONDITIONS

All heat exchanger walls including the baffles are modeled as shells in 3D. This requires special boundary conditions for the flow and heat transport equations.

The interior wall boundary condition for the flow separates the fluids from each other and is also used to describe the baffles. On both sides, it applies the wall functions needed for simulating walls with the k-ε turbulence model.

To account for the in-plane heat flux in the shell, the **Thin Layer** boundary condition is applied. This boundary condition simulates heat transfer in thin shell structures. Here, the shell is supposed made of steel and with a 5 mm thickness.

Water enters the tube side with a velocity of 0.1 m/s and a temperature of 80 °C. Air enters the shell side with a velocity of 1 m/s and a temperature of 5 °C. At both inlets, the recommended values for the turbulence length scale is 0.07 L, where L is the channel radius (see Inlet Values for the Turbulence Length Scale and Turbulent Intensity). For the water inlet, this radius is equal to 5 cm and for the air inlet, it is 4.5 cm.

Beside the symmetry plane, all remaining exterior boundaries are thermally insulated walls.

Results and Discussion

An important criterion for estimating the accuracy of a turbulence model is the wall resolution. Hence, COMSOL Multiphysics creates a plot of the wall resolution by default. The value for δ_w^+ has to be 11.06, which corresponds to the distance from the wall where the logarithmic layer meets the viscous sublayer. Furthermore, the wall lift-off δ_w has to be small compared to the dimension of the geometry. On interior walls you have δ_w for the upside and downside of the wall. To visualize the upside and downside directions, use an arrow surface plot with the components `unx`, `uny`, and `unz` for the up direction and `dnx`, `dny`, and `dnz` for the down direction.

Figure 2 shows the upside wall lift-off. This is the wall lift-off inside the tubes where the probably most critical area in terms of mesh resolution is located. It is about 10% of the tube radius, which is sufficient.

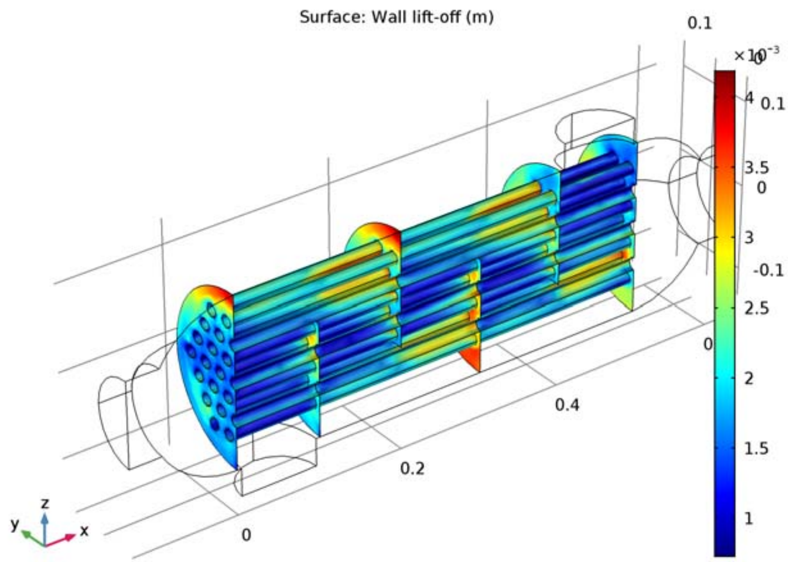


Figure 2: Wall lift-off for the tubes.

The tube side velocity shows a uniform distribution in the tubes. Before water enters the tubes, recirculation zones are present. The streamline colors represent the temperature and you can see that the temperatures at both outlets are close to each other.

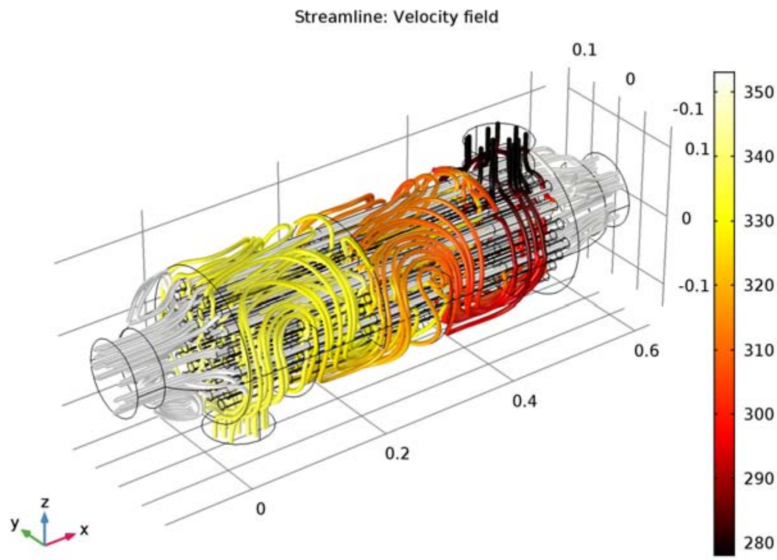


Figure 3: Velocity streamlines.

Figure 4 shows the temperature distribution on the heat exchanger boundaries.

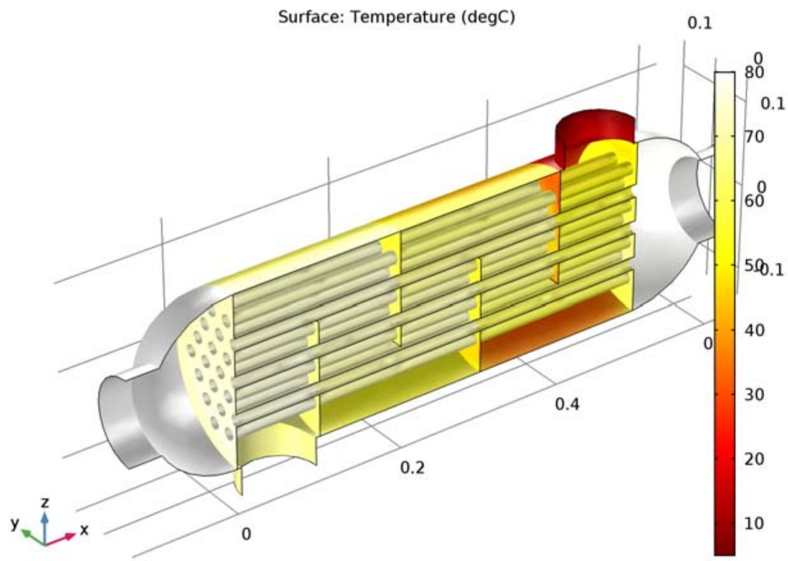


Figure 4: Temperature at the heat exchanger boundaries.

There are several quantities that describe the characteristics and effectiveness of a heat exchanger. One is the equivalent heat transfer coefficient given by

$$h_{\mathrm{eq}} = \frac{P}{A(T_{\mathrm{hot}} - T_{\mathrm{cold}})} \tag{1}$$

where P is the total exchanged power and A is the surface area through which P flows. In this model the value of h_{eq} is 5.4 W/(m^2·K).

The pressure drop is about 33 Pa on the tube side and 12 Pa on the shell side.

Notes About the COMSOL Implementation

Solve the model using a physics-controlled mesh. For flow applications this means that COMSOL Multiphysics automatically generates a mesh sequence where the mesh size depends on whether the flow is laminar or turbulent and where a boundary layer mesh is applied to all no slip walls. Even if the coarsest mesh size is used, the mesh is still fine enough to resolve the flow pattern and thus the temperature distribution well.

Nevertheless, this application requires about 10 GB RAM. Alternatively, you can set up a coarser mesh manually, but keep in mind that this can lead to lower accuracy.

The first part of the modeling process is the preprocessing. This includes defining parameters, preparing the geometry, and defining relevant selections. You can skip this part by loading the file `shell_and_tube_heat_exchanger_geom.mph`. However, we recommend that you take a look at these steps at least once. Especially when developing models intended for optimization and sophisticated analyses, these steps can significantly simplify and accelerate the modeling process.

Defining parameters beforehand enables setting up a parametric study immediately, also for multiple parameter sets. In addition, this provides a fast overview of the operating conditions. In the Modeling Instructions, several selections are also created. Once defined, they are available in every step of the modeling process. If you want to change from concurrent to countercurrent heat exchanger, you only need to redefine the selections.

Application Library path: `Heat_Transfer_Module/Heat_Exchangers/shell_and_tube_heat_exchanger`

Reference

1. H. S. Lee, *Thermal Design*, John Wiley & Sons, 2010.

Modeling Instructions

ROOT

The file `shell_and_tube_heat_exchanger_geom.mph` contains a parameterized geometry and prepared selections for the model. Start by loading this file.

1 From the **File** menu, choose **Open**.

2 Browse to the application's Application Library folder and double-click the file `shell_and_tube_heat_exchanger_geom.mph`.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Water, liquid**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Water, liquid (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat2)**.

2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Water Domain**.

The heat exchanger itself is made of structural steel. Apply this material to the walls of the heat exchanger.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Structural steel**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Structural steel (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Structural steel (mat3)**.

2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Walls**.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

TURBULENT FLOW, K-ε (SPF)

Since the density variation is not small, the flow can not be regarded as incompressible. Therefore set the flow to be compressible.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ε (spf)**.

2 In the **Settings** window for Turbulent Flow, k-ε, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

The next step is to set the boundary conditions. At first the boundary conditions for the flow equations are applied. These are the inlet and outlet boundary conditions, symmetry as well as the interior walls which separate the air from the water domain. The default wall boundary condition then applies to the outer boundaries.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for Inlet, type `Inlet 1: Water` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet, Water**.

4 Locate the **Turbulence Conditions** section. In the L_T text field, type `0.07*5[cm]`.

This is an estimation for the turbulence length scale as explained in the Model Definition section.

5 Locate the **Velocity** section. In the U_0 text field, type `u_water`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for Outlet, type `Outlet 1: Water` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet, Water**.

4 Locate the **Pressure Conditions** section. Select the **Normal flow** check box.

Inlet 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for Inlet, type `Inlet 2: Air` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet, Air**.

4 Locate the **Turbulence Conditions** section. In the L_T text field, type `0.07*4.5[cm]`.

This is an estimation for the turbulence length scale as explained in the Model Definition section.

5 Locate the **Velocity** section. In the U_0 text field, type `u_air`.

Outlet 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for Outlet, type `Outlet 2: Air` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet, Air**.

4 Locate the **Pressure Conditions** section. Select the **Normal flow** check box.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for Symmetry, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

Interior Wall 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Wall**.

2 In the **Settings** window for Interior Wall, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

The boundary conditions that set up the heat transfer equation are the temperatures at the inlets and the outflow at the outlets, the symmetry and for all walls the highly conductive layer feature accounts for the heat conduction through the shell.

HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 In the **Settings** window for Temperature, type `Temperature 1: Water` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet, Water**.

4 Locate the **Temperature** section. In the T_0 text field, type `T_water`.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 In the **Settings** window for Outflow, type `Outflow 1: Water` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet, Water**.

Temperature 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 In the **Settings** window for Temperature, type `Temperature 2: Air` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet, Air**.

4 Locate the **Temperature** section. In the T_0 text field, type `T_air`.

Outflow 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 In the **Settings** window for Outflow, type `Outflow 2: Air` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet, Air**.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for Symmetry, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for Thin Layer, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

4 Locate the **Thin Layer** section. From the **Layer type** list, choose **Thermally thin approximation**.

5 In the d_s text field, type `0.005[m]`.

The physics is now defined. For evaluating the equivalent heat transfer coefficient according to Equation 1 directly after solving the model, you need to define the following component coupling operator. It can also be defined and evaluated after computing the simulation, but in this case you need to choose **Update Solution** (after right-clicking on the **Study 1** node) to make this operator available without running the model again.

DEFINITIONS

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, type `Average Operator on Water-Air Interface` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Water-Air Interface**.

The heat exchanger properties and operating conditions are well defined and the model is ready to solve. For a first estimation of the heat exchanger performance a coarse mesh is satisfying. The solution is obtained very fast and provides qualitatively good results. Reliable quantitative results require a good resolution especially of the wall regions.

The default physics-controlled mesh is a good starting point and can be customized to reduce computational costs. Here you reduce the number of boundary layers and scale the geometry for the meshing sequence in the x-direction. This results in an anisotropic mesh that is suitable for the minor changes of the flow field and temperature field in the x-direction.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extremely coarse**.

4 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

2 In the **Settings** window for Free Tetrahedral, click to expand the **Scale geometry** section.

3 Locate the **Scale Geometry** section. In the **x-direction scale** text field, type `0.5`.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 In the **Settings** window for Boundary Layer Properties, locate the **Boundary Layer Properties** section.

3 In the **Number of boundary layers** text field, type `3`.

4 Click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Velocity (spf)

A slice plot of the velocity field, a surface plot of the wall resolution and a surface plot of the temperature are generated by default. You can either customize these plots or create new plot groups to visualize the results.

The wall resolution indicates the accuracy of the flow close to the walls where the wall functions are applied. The variable `spf.d_w_plus` should be 11.06 and the wall lift-off `spf.delta_w` needs to be significantly smaller than the dimension of the geometry. On interior boundaries, these variables are available for the upside and downside of the wall indicated by `spf.d_w_plus_u/d` or `spf.delta_w_u/d`, respectively. The critical regions in terms of the wall resolution are in the tubes.

Wall Resolution (spf)

Delete the **Upside Wall Lift-Off** and **Downside Wall Lift-Off** from the **Wall Resolution (spf)** plot group.

1 In the **Model Builder** window, expand the **Results>Wall Resolution (spf)** node, then click **Wall Lift-Off**.

2 In the **Settings** window for Surface, locate the **Data** section.

3 From the **Data set** list, choose **Interior Walls**.

4 Locate the **Expression** section. In the **Expression** text field, type `spf.delta_w_u`.

5 On the **Wall Resolution (spf)** toolbar, click **Plot**.

   Figure 4 shows the temperature distribution on all wall boundaries. The default temperature plot uses the **Surface 1** data set created automatically and contains exterior walls only. It is easy to change it by using the selection created at the beginning.

Data Sets

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **All Walls**.

2 In the **Settings** window for Surface, locate the **Selection** section.

3 From the **Selection** list, choose **Walls**.

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 **Data set** list, choose **All Walls**.

4 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

5 In the **Settings** window for Surface, locate the **Expression** section.

6 From the **Unit** list, choose **degC**.

7 On the **Temperature (ht)** toolbar, click **Plot**.

In Figure 3, the streamlines are plotted for the full 3D geometry. Even if only one half of the heat exchanger is modeled the solution can be mirrored to obtain a full 3D view of the results. To do so, follow the steps below:

Mirror 3D 1

On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

Data Sets

1 In the **Settings** window for Mirror 3D, locate the **Plane Data** section.

2 From the **Plane** list, choose **zx-planes**.

3D Plot Group 6

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, type `Velocity, Streamlines` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Mirror 3D 1**.

Velocity, Streamlines

1 Right-click **Velocity, Streamlines** and choose **Streamline**.

2 In the **Settings** window for Streamline, locate the **Streamline Positioning** section.

3 In the **Points** text field, type `100`.

4 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

5 Right-click **Results>Velocity, Streamlines>Streamline 1** and choose **Color Expression**.

6 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Fluids>Temperature>T - Temperature**.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

Evaluate the equivalent heat transfer coefficient by using the component coupling operators defined previously.

Global Evaluation 1

On the **Results** toolbar, click **Global Evaluation**.

Derived Values

1 In the **Settings** window for Global Evaluation, type `Heat Transfer Coefficient` in the **Label** text field.

2 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`aveop1(nitf1.qwf_u)/(T_water-T_air)`	`W/(m^2*K)`	

The variable `nitf1.qwf_u` efficiently computes the heat flux through the wall separating water and air domains, based on the wall function definition.

3 Click **Evaluate**.

To evaluate the pressure drop, the average inlet pressures are computed.

Surface Average 1

On the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

Derived Values

1. In the **Settings** window for Surface Average, type `Inlet Pressure, Water` in the **Label** text field.
2. Locate the **Selection** section. From the **Selection** list, choose **Inlet, Water**.
3. Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Turbulent Flow, k-ε>Velocity and pressure>p - Pressure**.
4. Click **Evaluate**.

Surface Average 2

On the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

Derived Values

1. In the **Settings** window for Surface Average, type `Inlet Pressure, Air` in the **Label** text field.
2. Locate the **Selection** section. From the **Selection** list, choose **Inlet, Air**.
3. Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Turbulent Flow, k-ε>Velocity and pressure>p - Pressure**.
4. Click **Evaluate**.

TABLE

1. Go to the **Table** window.

   The tube side pressure drop is 33 Pa and the shell side pressure drop is 12 Pa.

Shell Conduction

Introduction

The following example illustrates how to build and solve a model using the Heat Transfer in Thin Shells interface. This example is a 2D NAFEMS benchmark (Ref. 1), which was transformed to 3D.

Model Definition

Figure 1 describes the 2D benchmark example.

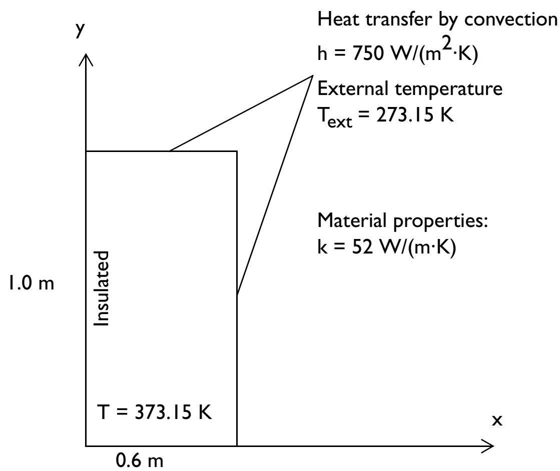


Figure 1: A 2D benchmark example for a thin conductive shell.

The 3D model bends this plate so that it becomes a quarter of a cylinder (Figure 2).

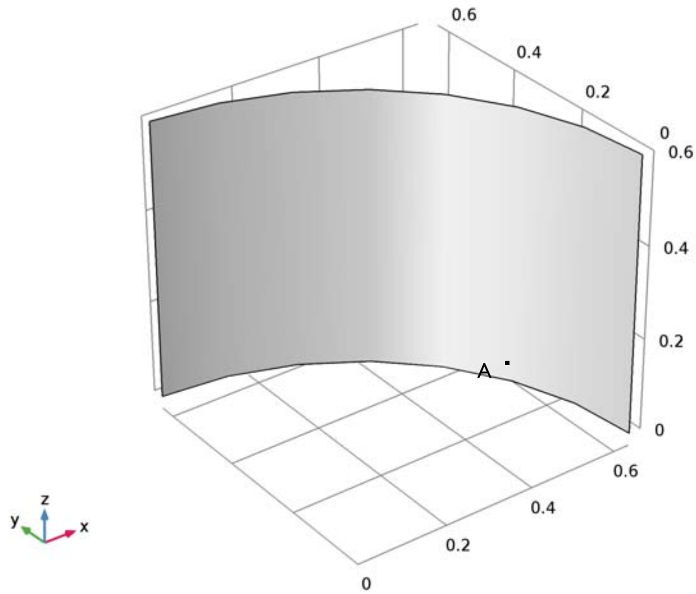


Figure 2: The 3D geometry based on the 2D model.

Results

The temperature at point A in Figure 2 (291.40 K) is in agreement with that from the NAFEMS benchmark (Ref. 1). Figure 3 shows the temperature distribution.

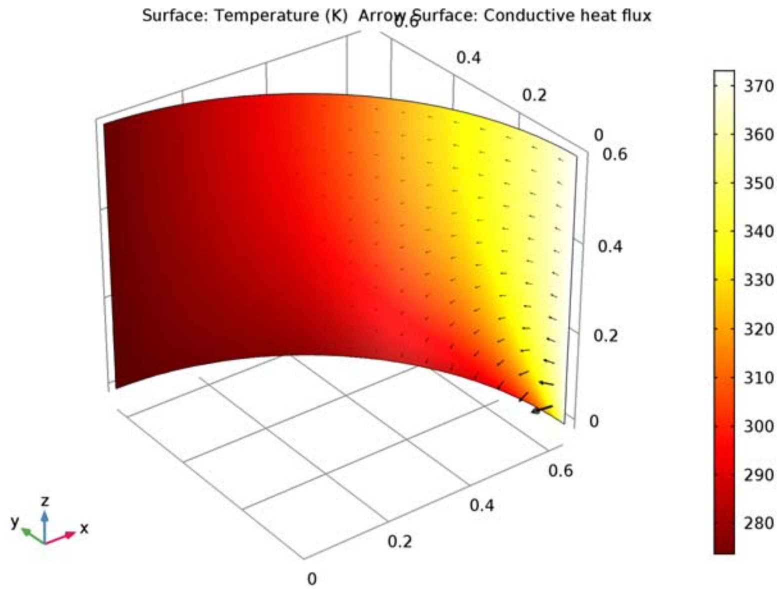


Figure 3: The resulting temperature field of the 3D model.

Reference

1. J.A. Casey and G.B Simpson, "Two-dimensional Steady State," *Benchmark Tests for Thermal Analysis*, NAFEMS, Test 10, p. 2.9, 1986.

Application Library path: `Heat_Transfer_Module/Tutorials,_Thin_Structure/shell_conduction`

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 **Heat Transfer>Thin Structures>Heat Transfer in Thin Shells (htsh)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
T_edge	373.15[K]	373.15 K	Edge temperature
T_ext	273.15[K]	273.15 K	External temperature
ht	750[W/(m^2*K)]	750 W/(m²·K)	Heat transfer coefficient

GEOMETRY 1

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Object Type** section.

3 From the **Type** list, choose **Surface**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `2/pi`.

5 In the **Height** text field, type `0.6`.

6 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **cyl1**, select Boundaries 1–3 only.

3 Right-click **Component 1 (comp1)>Geometry 1>Delete Entities 1 (del1)** and choose **Build Selected**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Point**.

2 In the **Settings** window for Point, locate the **Point** section.

3 In the **x** text field, type `(2/pi)*cos(pi*18/180)`.

4 In the **y** text field, type `(2/pi)*sin(pi*18/180)`.

This step embeds the point where you compare the calculated solution with the benchmark.

5 Right-click **Point 1 (pt1)** and choose **Build Selected**.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	52	W/(m·K)	Basic
Density	rho	8800	kg/m³	Basic
Heat capacity at constant pressure	Cp	420	J/(kg·K)	Basic

HEAT TRANSFER IN THIN SHELLS (HTSH)

Temperature 1

1 On the **Physics** toolbar, click **Edges** and choose **Temperature**.

2 Select Edge 5 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `T_edge`.

Heat Flux 1

1. On the **Physics** toolbar, click **Edges** and choose **Heat Flux**.
2. Select Edges 1, 2, and 4 only.
3. In the **Settings** window for Heat Flux, locate the **Heat Flux** section.
4. Click the **Convective heat flux** button.
5. In the h text field, type `ht`.
6. In the T_{ext} text field, type `T_ext`.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

Mapped 1

1. Select Boundary 1 only.
2. In the **Settings** window for Mapped, click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (htsh)

The default plot is the surface plot of the temperature and the arrow plot of the conductive heat flux; compare with Figure 3.

1. Click the **Zoom Extents** button on the **Graphics** toolbar.

Derived Values

Follow the steps below to obtain the temperature at the benchmark verification point.

Point Evaluation 1

1. On the **Results** toolbar, click **Point Evaluation**.
2. Select Point 3 only.
3. In the **Settings** window for Point Evaluation, click **Evaluate**.

TABLE

1. Go to the **Table** window.

   The result shown in the **Table** window below the **Graphics** window should be approximately 290.4 K.

Created in COMSOL Multiphysics 5.2a

Heat Transfer in a Surface-Mount Package for a Silicon Chip

Introduction

All integrated circuits—especially high-speed devices—produce heat. In today's dense electronic system layouts heat sources are many times placed close to heat-sensitive ICs. Designers of printed-circuit boards often need to consider the relative placement of heat-sensitive and heat-producing devices, so that the sensitive ones do not overheat.

One type of heat-generating device is a voltage regulator, which can produce several watts of heat and reach a temperature higher than 70 °C. If the board design places such a device close to a surface-mounted package that contains a sensitive silicon chip, the regulator's heat could cause reliability problems and failure due to overheating.

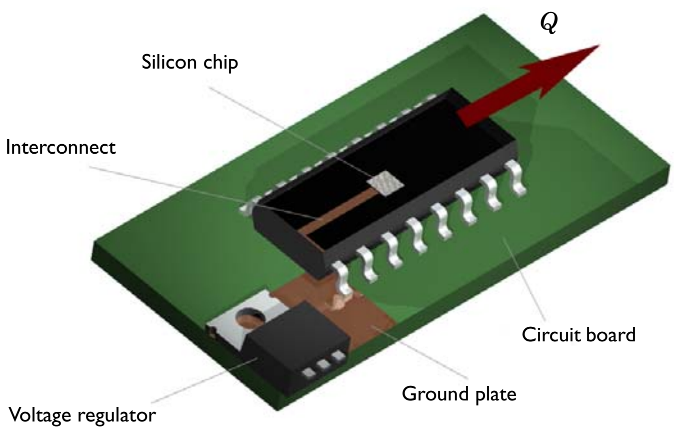


Figure 1: Layout of the simulated silicon device, its package, and a voltage regulator. The chip and the voltage regulator are connected through a ground plate, a pin, and the interconnect.

This simulation investigates the thermal situation for a silicon chip in a surface-mount package placed on a circuit board close to a hot voltage regulator. The chip is subjected to heat from the regulator and from internally generated heat.

Model Definition

The model is based on a SMD IC and voltage regulator layout as in Figure 1. The silicon chip sits in the center of the package and dissipates its heat to the surrounding environments. The chip also connects to a ground plane through an interconnect and one of the pins. A heat generating voltage regulator is placed on the same ground plane. This means that the voltage regulator may affect the silicon chip by the conducted heat and this may lead to overheating of the chip.

Heat transfers through the mounted package to the surroundings through conduction according to:

$$\nabla \cdot (-k\nabla T) = Q$$

The heat source, Q, is negligible in the circuit board, pins and package, while in the chip, this model sets that parameter to a value equivalent to 20 mW. The conductivities of the components are chosen to be similar to:

- silicon, for the chip
- aluminum, for the pins
- FR4, for the PC board
- copper, for the ground plane and interconnect
- an arbitrary plastic, for the chip package

Heat dissipates from all air-exposed surfaces through forced heat convection, which is modeled using a heat transfer coefficient, h:

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\mathrm{inf}} - T)$$

The voltage regulator is simulated by setting a fixed temperature at that surface. The thin conducting layers of the ground plane and interconnect within the package is modeled using a 2D shell approximation, according to:

$$\nabla_{\mathbf{t}} \cdot (-d_{\mathrm{s}} k \nabla_{\mathbf{t}} T) = 0$$

where d_{s} is the layer's thickness, and $\nabla_{\mathbf{t}}$ represents the nabla operator projected onto the direction of the plane. The model uses a Heat Transfer interface to describe the 3D heat transfer as well as the 2D shell heat transfer.

Results and Discussions

Figure 2 illustrates the temperature distribution through the thickness. Being a good conductor, the interconnect delivers heat to the outer edge of the package, which gives the fairly constant temperature distribution around the interconnect.

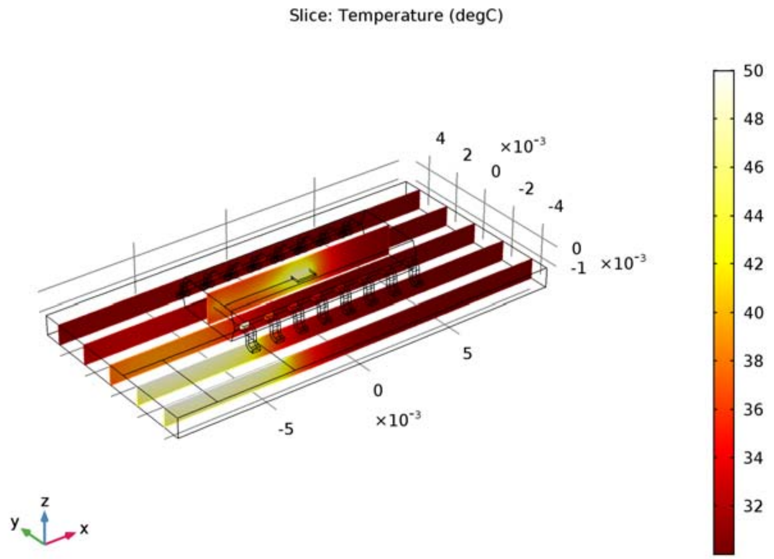


Figure 2: Slice plot of the temperature through the circuit board, interconnect, chip, and package. The effect of the interconnect is evident by its ability to conduct heat from the chip to the outer parts of the package.

An alternative view is achieved by using the transparency feature in the visualization tools of COMSOL Multiphysics. This results in a transparent 3D view of the temperature distribution, as shown in Figure 3. In that figure you can see the temperature distribution around the chip and along the interconnect.

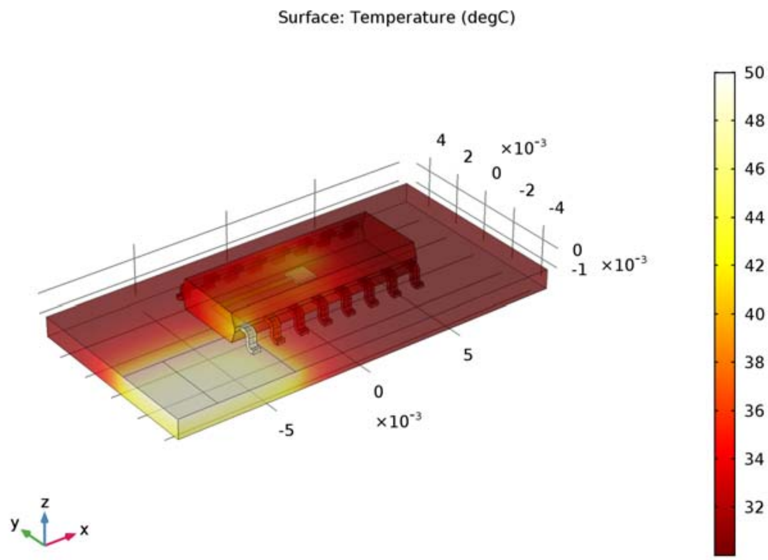


Figure 3: Boundary plot of the temperature created with the assistance of the transparency tool in COMSOL Multiphysics. This view also gives the temperature distribution on the chip and along the interconnect.

To get a closer look at the stationary temperature of the silicon chip, plot the temperature at the bottom boundary of the chip.

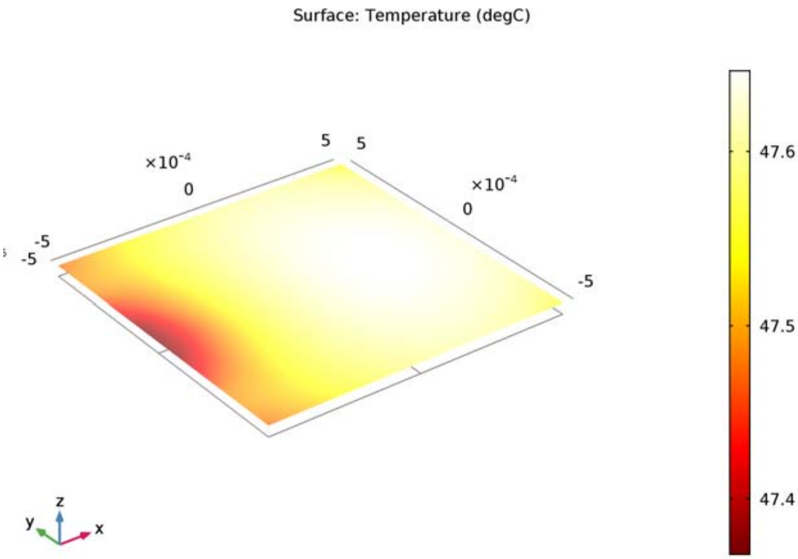


Figure 4: Temperature distribution on the bottom surface of the silicon chip.

The simulation predicts a maximum temperature of the silicon device of 47.7 °C. This means that the device does not overheat in the present configuration.

Application Library path: `Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/surface_mount_package`

Notes About the COMSOL Implementation

This tutorial uses the Heat Transfer interface from the Heat Transfer Module and, in particular, its Thin Layer feature. These are thin layers that conduct heat well so you need not define them in 3D. The two layers that have this definition are:

- The interconnect between the chip and the grounded pin.
- The ground plate that is also thermally connected to the temperature constraint coming from the voltage regulator.

While the numerical method considers these two modeling domains as interior boundaries, the model still includes a thickness to take the 3D heat flux into account.

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

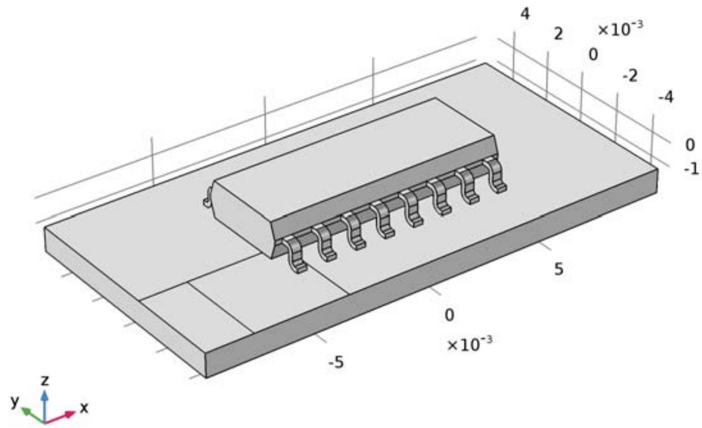
The 3D workspace is now ready. While you can create the 3D geometry with the built-in CAD tools of COMSOL Multiphysics, it is still provided here in the form of a CAD-file ready to be imported.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.
2 In the **Settings** window for Import, locate the **Import** section.
3 From the **Source** list, choose **COMSOL Multiphysics file**.
4 Click **Browse**.
5 Browse to the application's Application Libraries folder and double-click the file `surface_mount_package.mphtxt`.
6 Click **Import**.

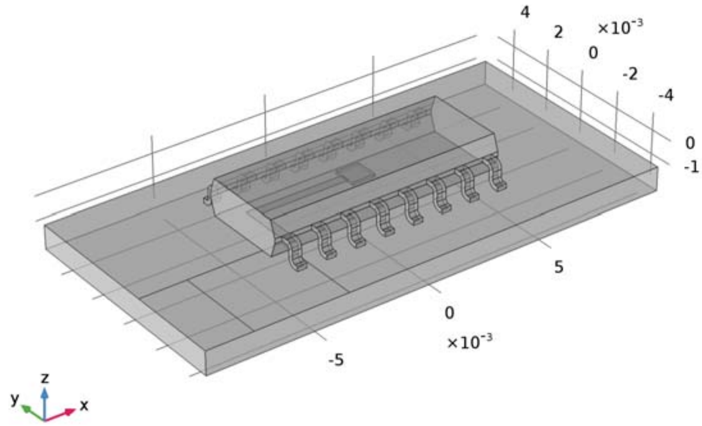
7 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The geometry in the **Graphics** window should now look like that in the figure below.



The imported geometry is complete. In particular, it includes the interconnect between the pin and the chip as well as the ground plate and the temperature surface resulting from the voltage regulator. To see the interconnect and the chip, you need to turn on transparency.

8 Click the **Transparency** button on the **Graphics** toolbar.



9 Click the **Transparency** button on the **Graphics** toolbar again to remove the transparency.

DEFINITIONS

Define a selection that contains all exterior boundaries. You will use it when defining the boundary conditions.

Explicit 1

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for Explicit, type `Exterior Boundaries` in the **Label** text field.
3. Locate the **Input Entities** section. Select the **All domains** check box.
4. Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

Now, define the domain settings including material properties, element order, heat source, and initial values.

MATERIALS

To define material properties for the model domains, use three predefined materials from the **Material Browser** and one custom material.

ADD MATERIAL

1. On 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>Aluminum**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Aluminum (mat1)

The first material you add to a model applies to all domains by default. Keep this setting and use overrides for the domains of different materials.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>FR4 (Circuit Board)**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

FR4 (Circuit Board) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **FR4 (Circuit Board) (mat2)**.

2 Select Domain 1 only.

Material 3 (mat3)

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 `Plastic` in the **Label** text field.

3 Select Domain 2 only.

Plastic (mat3)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Plastic (mat3)** node, then click **Basic (def)**.

2 In the **Settings** window for Property Group, locate the **Output Properties and Model Inputs** section.

3 Find the **Quantities** subsection. In the tree, select **Output Properties>Density**.

4 Click **Add**.

5 Find the **Output properties** subsection. In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Density	rho	2700	kg/m³	1x1

6 Find the **Quantities** subsection. In the tree, select **Output Properties>Heat Capacity at Constant Pressure**.

7 Click **Add**.

8 Find the **Output properties** subsection. In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Heat capacity at constant pressure	Cp	900	J/(kg·K)	1x1

9 Find the **Quantities** subsection. In the tree, select **Output Properties>Thermal Conductivity**.

10 Click **Add**.

11 Find the **Output properties** subsection. In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Thermal conductivity	k ; kii = k, kij = 0	0.2	W/(m·K)	3x3

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Silicon**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Silicon (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silicon (mat4)**.

2 Select Domain 11 only.

This completes the materials settings.

HEAT TRANSFER IN SOLIDS (HT)

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 11 only.

3 In the **Settings** window for Heat Source, locate the **Heat Source** section.

4 In the Q_0 text field, type `2e8`.

This completes the domain settings. Now, configure to the boundary conditions.

Heat Flux 1

1 On 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 **Exterior Boundaries**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `50`.

6 In the T_{ext} text field, type `30[degC]`.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 4 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `50[degC]`.

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 Select Boundary 7 only.

3 In the **Settings** window for Thin Layer, locate the **Thin Layer** section.

4 From the **Layer type** list, choose **Thermally thin approximation**.

5 In the d_{s} text field, type `1e-4`.

6 Locate the **Heat Conduction** section. From the k_{s} list, choose **User defined**. In the associated text field, type `400`.

Thin Layer 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 Select Boundary 31 only.

3 In the **Settings** window for Thin Layer, locate the **Thin Layer** section.

4 From the **Layer type** list, choose **Thermally thin approximation**.

5 In the d_{s} text field, type `5e-6`.

6 Locate the **Heat Conduction** section. From the k_s list, choose **User defined**. In the associated text field, type 400.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.

Size 1

1. In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Boundary**.
3. Select Boundaries 4 and 7 only.
4. Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.
5. In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for Size, locate the **Element Size** section.
3. From the **Predefined** list, choose **Fine**.
4. Click **Build All**.

   The mesh should consist of around 30,000 elements.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

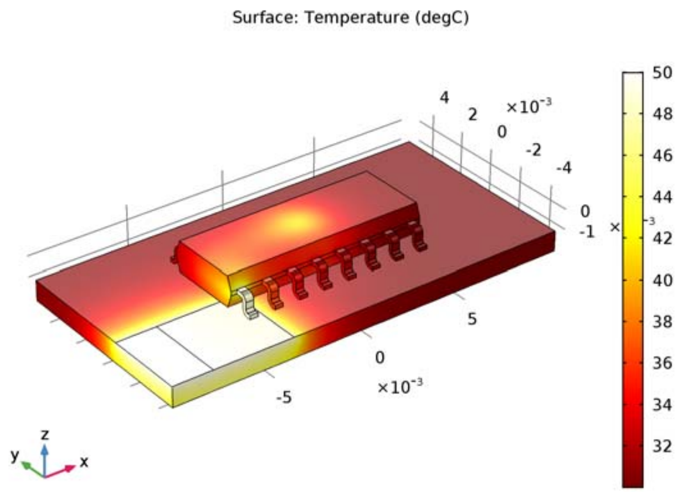
By default, you get surface and isosurface plots for the temperature. Note that the temperature is displayed in kelvin, which is the default temperature unit in the SI system.

To get a surface temperature plot in degrees Celsius, simply change the unit for the first default plot group.

Surface

1. In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.
2. In the **Settings** window for Surface, locate the **Expression** section.
3. From the **Unit** list, choose **degC**.
4. On the **Temperature (ht)** toolbar, click **Plot**.

5 Click the **Go to Default 3D View** button on the **Graphics** toolbar.



To see the chip as well, turn on transparency.

6 Click the **Transparency** button on the **Graphics** toolbar.

Compare the resulting plot to that in Figure 3.

7 Click the **Transparency** button on the **Graphics** toolbar again to remove the transparency.

Reproduce the plot in Figure 2 with the following steps.

3D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, type `Temperature, Slices` in the **Label** text field.

Slice 1

1 Right-click **Temperature, Slices** and choose **Slice**.

2 In the **Settings** window for Slice, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

6 On the **Temperature, Slices** toolbar, click **Plot**.

Compare the result to that in Figure 2.

To visualize the temperature distribution on the silicon chip's bottom surface, follow the steps given below.

Study 1/Solution 1 (2) (sol1)

On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

Selection

1 On the **Results** toolbar, click **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 Boundary 137 only.

3D Plot Group 4

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, type `Temperature, Chip Surface` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

Surface 1

1 Right-click **Temperature, Chip Surface** and choose **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

5 On the **Temperature, Chip Surface** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the resulting plot to that in Figure 4.

Created in COMSOL Multiphysics 5.2a

Thermo-Mechanical Analysis of a Surface-Mounted Resistor

Introduction

The drive for miniaturizing electronic devices has resulted in today's extensive use of surface-mount electronic components. An important aspect in electronics design and the choice of materials is a product's durability and lifetime. For surface-mount resistors and other components producing heat it is a well-known problem that temperature cycling can lead to cracks propagating through the solder joints, resulting in premature failure (Ref. 1). For electronics in general there is a strong interest in changing the soldering material from lead- or tin-based solder alloys to other mixtures.

The following multiphysics example models the heat transport and structural stresses and deformations resulting from the temperature distribution using the Heat Transfer in Solids and Solid Mechanics interfaces.

Note: This application requires either the Structural Mechanics Module or the MEMS Module.

Model Definition

Figure 1 shows a photograph of a surface-mount resistor together with a diagram of it on a printed circuit board (PCB).

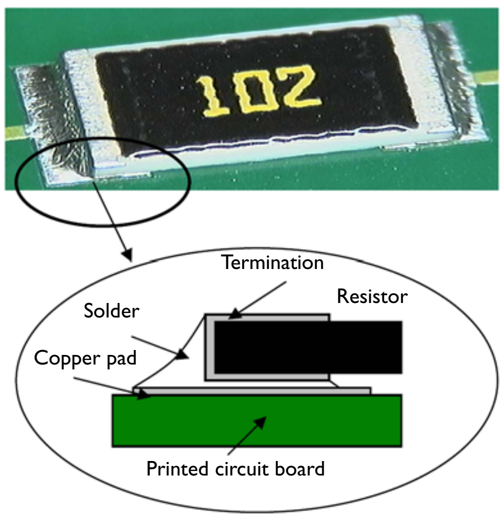


Figure 1: A photo and diagram of a typical surface-mounted resistor soldered to a PCB.

Table 1 shows the dimensions of the resistor and other key components in the model including the PCB.

TABLE 1: COMPONENT DIMENSIONS

COMPONENT	LENGTH	WIDTH	HEIGHT
Resistor (Alumina)	6 mm	3 mm	0.5 mm
PCB (FR4)	16 mm	8 mm	1.6 mm
Cu pad	2 mm	3 mm	35 µm
Termination (Silver)	0.5 mm	3 mm	25 µm
Stand-off (gap to PCB)	-	-	105 µm

The simulation makes use of the symmetry so that it needs to include only half of the component (Figure 2). The modeling of the PCB is terminated a distance away from the resistor, in order to reduce effects of the boundary conditions.

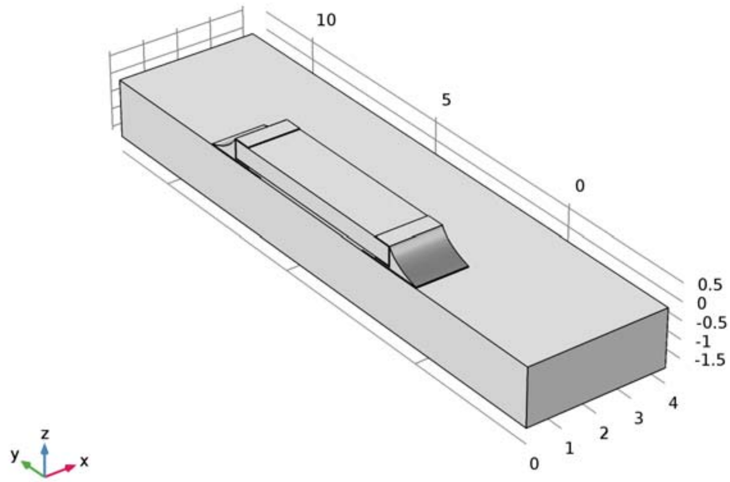


Figure 2: The simulation models only one half of the resistor.

In operation, the resistor dissipates 0.2 W of power as heat. Conduction to the PCB and convection to the surrounding air provide cooling. In this model, the heat transfer occurs through conduction in the subdomains. The model simplifies the surface cooling and describes it using a heat transfer coefficient, h, in this case set to $10\ \mathrm{W/(m^2{\cdot}K)}$; the surrounding air temperature, T_{inf}, is at 300 K. The resulting heat transfer equation and boundary condition (included in the model using the Heat Transfer interface) are

$$\nabla\cdot(-k\nabla T) = Q$$

$$-\mathbf{n}\cdot(-k\nabla T) = h(T_{\mathrm{inf}} - T)$$

where k is the thermal conductivity, and Q is the heating power per unit volume of the resistor (equal to $22.2\ \mathrm{MW/m^3}$ corresponding to 0.2 W in total).

The model handles thermal expansion using a static structural analysis using the Structural Mechanics interface (a description of the corresponding equations is available in the *Structural Mechanics Module User's Guide*). The thermal and mechanical material properties in this model are taken from the material library The data for the solid materials are temperature independent, and the reference values are shown in the table below.

TABLE 2: MATERIAL PROPERTIES

MATERIAL	E (GPa)	ν	α (ppm)	k (W/(m·K))	ρ (kg/m^3)	C_p (J/(kg·K))
Silver	83	0.37	18.9	420	10500	230
Alumina	300	0.222	8.0	27	3900	900
Cu	110	0.35	17	400	8700	385
Fr4	22	0.28	18	0.3	1900	1369
60Sn-40Pb	10	0.4	21	50	9000	150

Air has temperature-dependent and pressure-dependent properties in the built-in material library. Because the temperature is a variable of the problem, it is automatically used. The pressure is by default set to 1 atm.

The stresses are zero at room temperature, 293 K. The boundary condition for the Solid Mechanics interface is that the cuts in the PCB do not rotate and have no net force normal to the cut.

Results and Discussion

The isosurfaces in Figure 3 show the temperature distribution at steady state. The highest temperature occurs in the center of the resistor. The circuit board also heats up significantly.

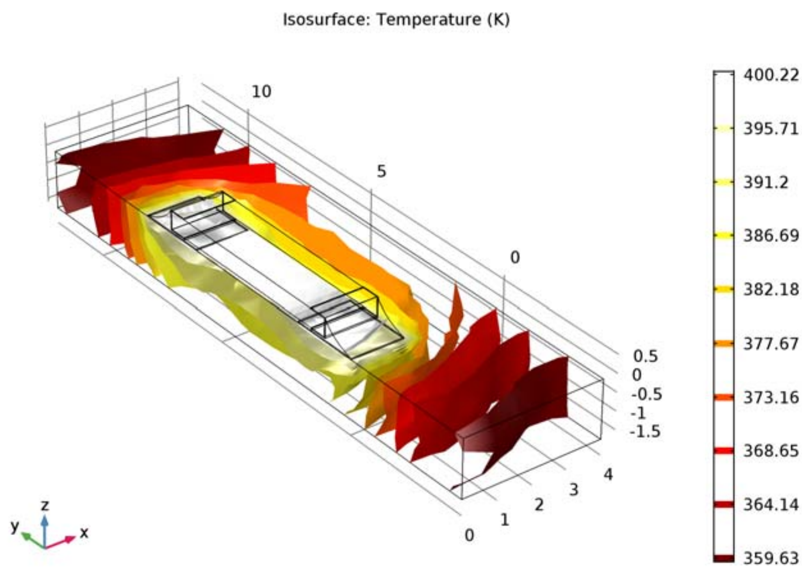


Figure 3: Temperature distribution in the resistor and the circuit board at steady state.

Thermal stresses appear as a result of the temperature increase; they arise from the materials' different expansion coefficients and from the bending of the PCB. Figure 4 plots the effective stress (von Mises) together with the resulting deformation of the assembly.

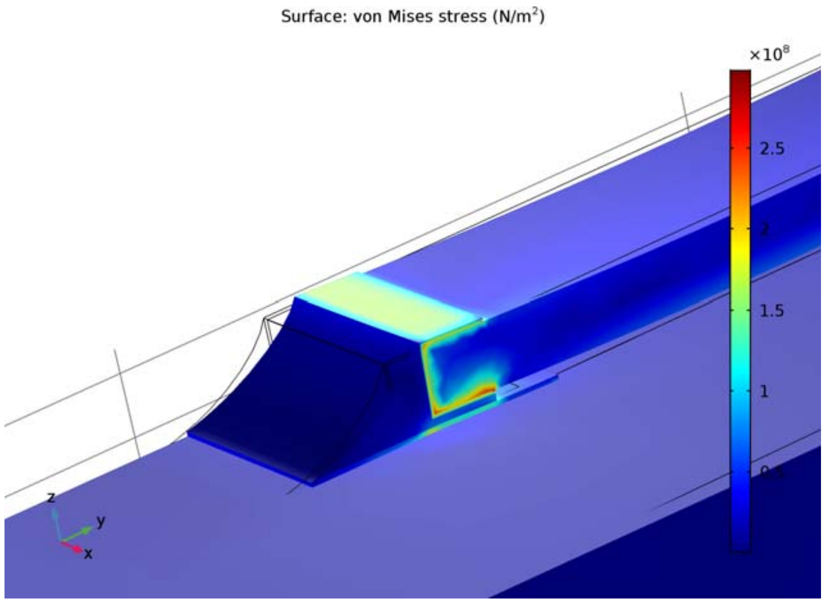


Figure 4: The thermally induced distribution of von Mises effective stress together with the deformation (magnified).

The highest stresses seem to occur in the termination material. It is interesting to compare these effective stresses to the yield stress and thereby investigate whether or not the material is irreversibly deformed. In that case the solder is the weak point. Figure 5 the stress in the solder alone.

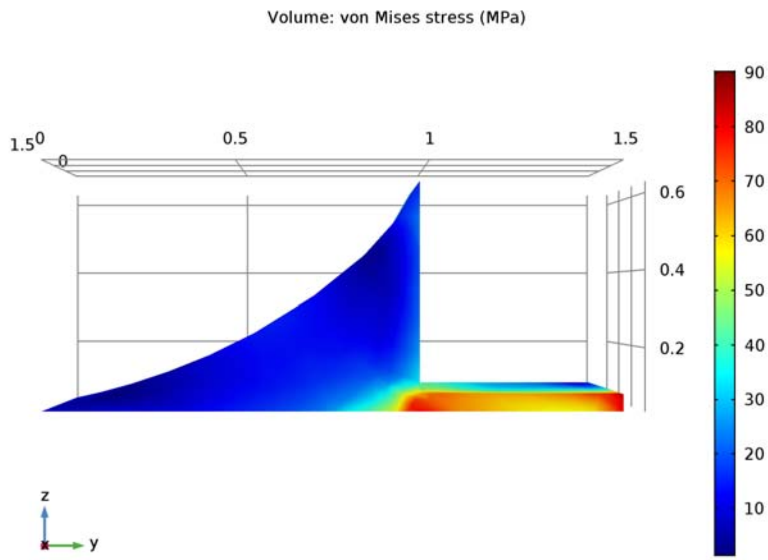


Figure 5: Close-up of the von Mises effective stresses in the solder joints.

The yield stress for solder is approximately 220 MPa. The highest effective stress appears to be about 60 percent of this value. The structure does not get permanently deformed directly when heated. However, it is possible that the solder displays creep strains over time because of the combination of fairly high stress levels and elevated temperatures.

Notes About the COMSOL Implementation

In this example, the Thermal Stress interface automatically adds and couples the Solid Mechanics interface and the Heat Transfer interface. This is done by the two predefined multiphysics features Thermal Expansion and Temperature Coupling.

Build the geometry as an assembly of the bottom plate and the resistor to make it possible to mesh the parts independently. Use continuity conditions for the temperature and the displacements to connect the top and bottom parts of the geometry.

It is assumed that the temperature gradient in the normal direction on the cuts in the PCB is zero. The mechanical boundary conditions on the cuts must not impose a general state of compressive stress due to thermal expansion. At the same time, the restraint from the part of the PCB that is not modeled means that there is no rotation of the cross section.

To obtain this effect, the entire cut must have the same (but unknown) displacement in the direction normal to the cut.

You can achieve this by using the roller constraint at the left cut of the plate that prevents the boundary to move in the direction normal to the cut. For remaining two cuts, introduce two new degrees of freedom (named `uface` and `vface`) in the model and prescribe the displacements of the cut faces to them.

You solve the problem sequentially using two stationary study steps. The heat transfer problem is nonlinear because the air has temperature-dependent properties. The structural problem, on the other hand, is linear. For the structural analysis, use a memory-efficient iterative solver to make it possible to solve the problem also on computers with limited memory.

References

1. H. Lu, C. Bailey, M. Dusek, C. Hunt, and J. Nottay, "Modeling the Fatigue Life of Solder Joints of Surface Mount Resistors," EMAP, 2000.

2. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 1, Pergamon Press, appendix, 1990.

Application Library path: `Heat_Transfer_Module/Thermal_Stress/surface_resistor`

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 **Structural Mechanics>Thermal Stress**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
T_air	300[K]	300 K	Air temperature
h_air	10[W/(m^2*K)]	10 W/(m²·K)	Heat transfer coefficient
Psource	0.2[W]/2	0.1 W	Heat dissipated by the resistor on the half geometry
p0	1[atm]	1.013E5 Pa	Air pressure
T0	80[degC]	353.2 K	Initial temperature guess

GEOMETRY 1

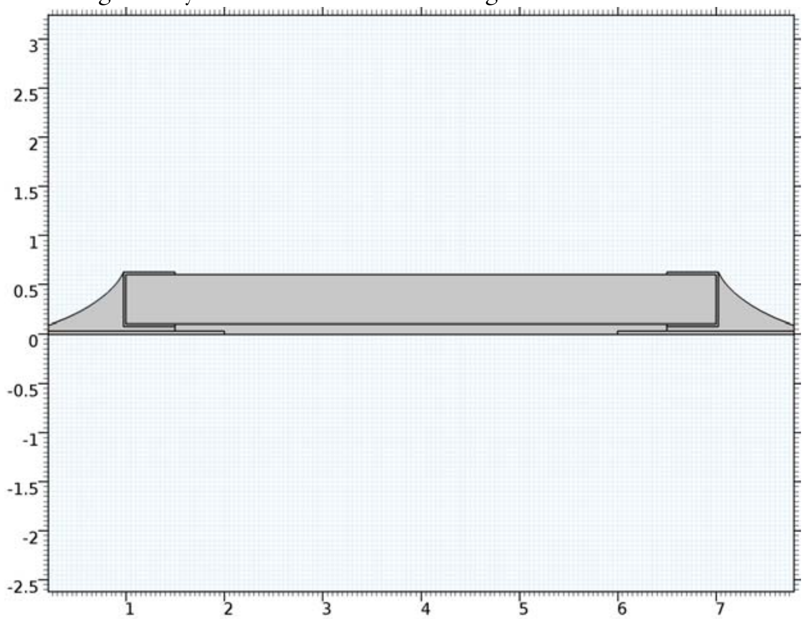
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Create the geometry. To simplify this step, insert a prepared 2D geometry sequence. On the Geometry toolbar, point to Import/Export and choose Insert Sequence. Browse to the application's Application Library folder and double-click the file `surface_resistor.mph`.

The 2D geometry should now look as in the figure below.



Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 Select the object **wp1** only.

3 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

4 In the table, enter the following settings:

Distances (mm)
1.5

5 Click **Build All Objects**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `4`.

4 In the **Depth** text field, type `16`.

5 In the **Height** text field, type `1.6`.

6 Locate the **Position** section. In the **y** text field, type `-4`.

7 In the **z** text field, type `-1.6`.

8 Click **Build All Objects**.

9 Click the **Zoom Extents** button on the **Graphics** toolbar.

Now, create an imprint of the resistor's bottom boundary on the printed circuit board to make a pair with matching parts.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Select the **Create imprints** check box.

5 On the **Geometry** toolbar, click **Build All**.

The completed geometry is shown in Figure 2.

ADD MATERIAL

1 On 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>FR4 (Circuit Board)**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

FR4 (Circuit Board) (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **FR4 (Circuit Board) (mat1)**.

2 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Select Domain 1 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Alumina**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Alumina (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Alumina (mat2)**.

2 Select Domain 5 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Copper**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Copper (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat3)**.

2 Select Domains 2 and 7 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Material Library>Elements>Silver>Silver [solid]**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Silver [solid] (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silver [solid] (mat4)**.

2 Select Domains 4 and 9 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Solder, 60Sn-40Pb**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Solder, 60Sn-40Pb (mat5)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Solder, 60Sn-40Pb (mat5)**.

2 Select Domains 3 and 8 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat6)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat6)**.

2 Select Domain 6 only.

3 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domains 1–5 and 7–9 only.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 10, 13, 16, 20, 33, 37, and 40 only.

Continuity 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Continuity**.

2 In the **Settings** window for Continuity, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Identity Pair 1 (ap1)**.

Roller 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Roller**.

2 Select Boundary 8 only.

Create the special boundary conditions on the cuts in the PCB by introducing new degrees of freedom, `uface` and `vface`, for the normal displacements.

3 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Global Equations 1

1 On the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for Global Equations, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	**f(u,ut,utt,t) (1)**	**Initial value (u_0) (1)**	**Initial value (u_t0) (1/s)**	**Description**
`uface`		0	0	
`vface`		0	0	

4 Locate the **Units** section. Find the **Dependent variable quantity** subsection. From the list, choose **Displacement field (m)**.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Point 1 only.

3 In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundary 2 only.

3 In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

5 In the u_{0y} text field, type `vface`.

Prescribed Displacement 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundary 9 only.

3 In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 In the u_{0x} text field, type `uface`.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Fluid 1

1 On the **Physics** toolbar, click **Domains** and choose **Fluid**.

2 Select Domain 6 only.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 5 only.

3 In the **Settings** window for Heat Source, locate the **Heat Source** section.

4 Click the **Heat rate** button.

5 In the P_0 text field, type `Psource`.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 3, 4, 11, 14, 19, 29, 30, 44, 46, and 49–58 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 In the h text field, type `h_air`.

6 In the T_{ext} text field, type `T_air`.

Next, add the continuity condition on the identity pair to couple the domains together.

Continuity 1

1 On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Continuity**.

2 In the **Settings** window for Continuity, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Identity Pair 1 (ap1)**.

Because the material properties are temperature dependent, the solution converges better if you supply an initial guess of the temperature.

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, type `T0` in the T text field.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

Free Triangular 1

Select Boundaries 10, 13, 16, 20, 24, 33, 37, and 40 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.
2 In the **Settings** window for Size, locate the **Element Size** section.
3 From the **Predefined** list, choose **Fine**.
4 Click the **Custom** button.
5 Locate the **Element Size Parameters** section. Select the **Minimum element size** check box.
6 In the associated text field, type `0.1`.
7 Click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.
2 In the **Settings** window for Swept, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 2–9 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.
2 In the **Settings** window for Distribution, locate the **Distribution** section.
3 In the **Number of elements** text field, type `10`.
4 Click **Build All**.
5 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

Free Tetrahedral 1

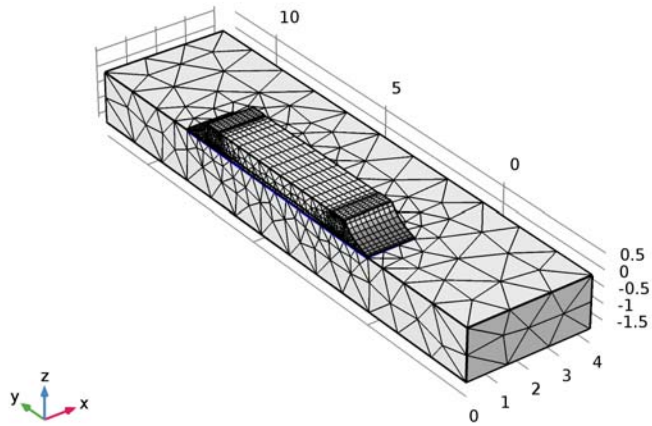
In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

Size 1

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2 From the **Geometric entity level** list, choose **Boundary**.
3 Select Boundaries 5–7 only.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

5 Click **Build All**.



STUDY 1

Because the heat transfer problem is independent of the displacements, use the first stationary study step to find the temperature distribution and the second stationary step to solve for the displacements.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then 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 box for the **Solid Mechanics** interface.

Stationary 2

On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

Step 2: Stationary 2

1 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Heat Transfer in Solids** interface.

3 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 1, Stationary**.

Solution 1 (sol1)

1 On 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)>Stationary Solver 1** node.

4 Right-click **Direct** and choose **Enable**.

Set up an iterative solver that can significantly save on the memory needed for the computations.

5 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Stationary Solver 2**.

6 In the **Settings** window for Stationary Solver, locate the **General** section.

7 From the **Linearity** list, choose **Linear**.

8 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** node.

9 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** and choose **Iterative**.

10 In the **Settings** window for Iterative, click to expand the **Error** section.

11 In the **Factor in error estimate** text field, type `2000`.

12 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2>Iterative 1** and choose **Multigrid**.

13 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

The first plot shows the effective stress together with the resulting deformation. Modify it to reproduce Figure 4.

1 In the **Model Builder** window, expand the **Stress (solid)** node.

Deformation

1 In the **Model Builder** window, expand the **Results>Stress (solid)>Surface 1** node, then click **Deformation**.

2 In the **Settings** window for Deformation, locate the **Expression** section.

3 In the **Y component** text field, type `v-vface`.

4 On the **Stress (solid)** toolbar, click **Plot**.

Stress (solid)

Hold down the left mouse button and drag in the Graphics window to rotate the geometry so that you see the opposite side of the resistor, which is where the largest stresses occur. Similarly, use the right mouse button to translate the geometry and the middle button to zoom.

Now, study the stresses in the solder.

Study 1/Solution 1 (3) (sol1)

On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 3 only.

3D Plot Group 4

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, type `Stress in Solder Joint` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

Volume 1

1 Right-click **Stress in Solder Joint** and choose **Volume**.

2 In the **Settings** window for Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Stress>solid.mises - von Mises stress**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

4 On the **Stress in Solder Joint** toolbar, click **Plot**.

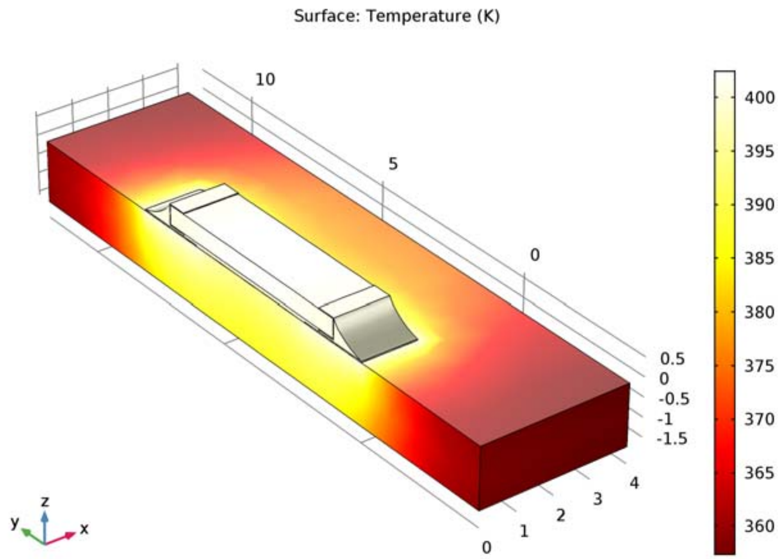
5 Click the **Go to YZ View** button on the **Graphics** toolbar.

Compare the resulting plot with that in Figure 5.

Temperature (ht)

The second default plot group shows the temperature on the modeled geometry's surface.

1 Click the **Go to Default 3D View** button on the **Graphics** toolbar.



Isothermal Contours (ht)

The third default plot shows the isosurfaces (Figure 3).

Created in COMSOL Multiphysics 5.2a

Thermal Bridges in Building Construction—3D Iron Bar Through Insulation Layer

Introduction

The European standard EN ISO 10211:2007 for thermal bridges in building constructions provides four test cases—two 2D and two 3D—for validating a numerical method (Ref. 1). If the values obtained by a method conform to the results of all these four cases, the method is classified as a *three-dimensional steady-state high precision method*.

COMSOL Multiphysics successfully passes all the test cases described by the standard. This document presents an implementation of the second 3D model (Case 4).

This tutorial studies the heat conduction in a thermal bridge made up of an iron bar and an insulation layer that separates a hot internal side from a cold external side. The iron bar is embedded in the insulation layer as shown in Figure 1. After solving the model, the heat flux between the internal and external sides and the maximum temperature on the external wall are calculated, and the results are compared to the expected values.

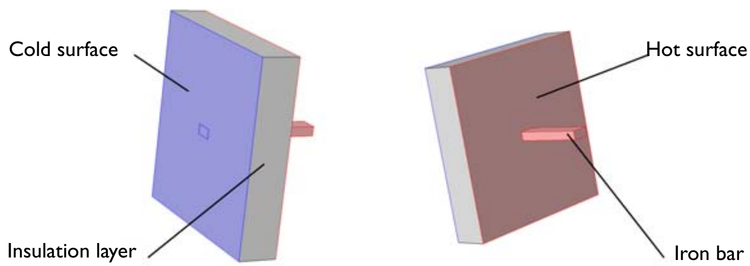


Figure 1: Back side (left) and front side (right) views of the iron bar embedded in an insulation layer, ISO 10211:2007 test case 4. Colored regions correspond to internal and external boundaries.

Model Definition

The geometry is illustrated above in Figure 1. The square insulation layer, with a low thermal conductivity k of 0.1 W/(m·K), has a cold and a hot surface. The iron bar has a higher thermal conductivity, 50 W/(m·K). Its boundaries are mainly located in the hot environment but one of them reaches the cold side.

Cold and hot surfaces are subject to convective heat flux. The ISO 10211:2007 standard specifies the values of the thermal resistance, R, which is related to the heat transfer coefficient, h, according to

$$h = \frac{1}{R}$$

Notes About the COMSOL Implementation

Compared to the rest of the structure, the dimensions of the intersection between the iron bar and the insulation layer are relatively small but the temperature gradients are large. Therefore, the element size is reduced in this area to give sufficient accuracy. To save computational time, this refinement is not applied on the remaining mesh.

Results and Discussion

In Figure 2, the temperature profile shows the effects of the thermal bridge where the heat variations are most pronounced.

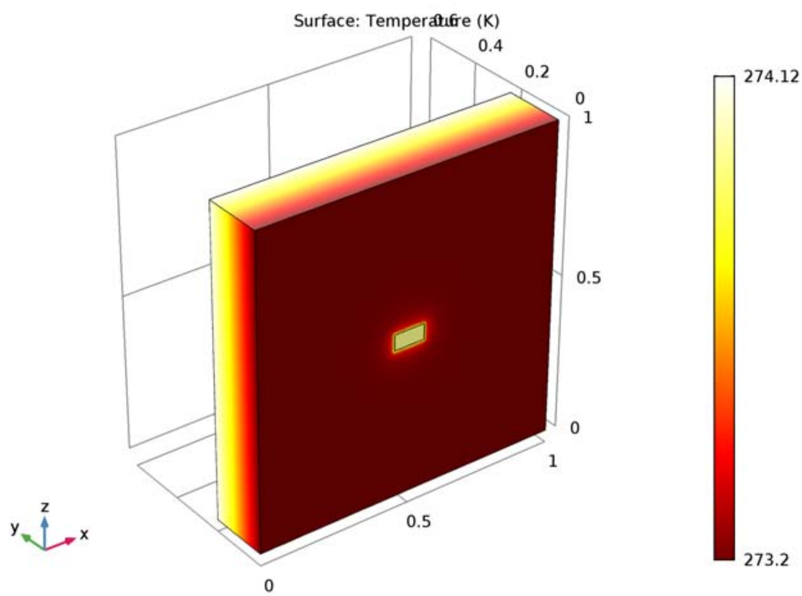


Figure 2: Temperature distribution of ISO 10211:2007 test case 4.

Table 1 compares the numerical results of COMSOL Multiphysics with the expected values provided by EN ISO 10211:2007 (Ref. 1).

TABLE 1: COMPARISON BETWEEN EXPECTED VALUES AND COMPUTED VALUES

MEASURED QUANTITY	EXPECTED VALUE	COMPUTED VALUE	DIFFERENCE
Highest temperature on external side	0.805 °C	0.8015 °C	$4.3\cdot10^{-3}$ °C
Heat flux	0.540 W	0.5415 W	0.28%

The maximum permissible differences to pass this test case are $5\cdot10^{-3}$ °C for temperature and 1% for heat flux. The measured values are completely coherent and meet the validation criteria.

Reference

1. European Committee for Standardization, *EN ISO 10211, Thermal bridges in building construction – Heat flows and surface temperatures – Detailed calculations (ISO 10211:2007)*, Appendix A, pp. 30–36, 2007.

Application Library path: `Heat_Transfer_Module/Buildings_and_Constructions/thermal_bridge_3d_iron_bar`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

Define the geometrical parameters.

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`w1`	`1[m]`	1 m	`Insulation layer width`
`d1`	`0.2[m]`	0.2 m	`Insulation layer depth`
`h1`	`1[m]`	1 m	`Insulation layer height`
`w2`	`0.1[m]`	0.1 m	`Iron bar width`
`d2`	`0.6[m]`	0.6 m	`Iron bar depth`
`h2`	`50[mm]`	0.05 m	`Iron bar height`

GEOMETRY 1

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `w1`.

4 In the **Depth** text field, type `d1`.

5 In the **Height** text field, type `h1`.

6 Right-click **Block 1 (blk1)** and choose **Build Selected**.

Create the iron bar at the center of the insulation layer.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `w2`.

4 In the **Depth** text field, type `d2`.

5 In the **Height** text field, type `h2`.

6 Locate the **Position** section. In the **x** text field, type `w1/2-w2/2`.

7 In the **z** text field, type `h1/2-h2/2`.

8 Right-click **Block 2 (blk2)** and choose **Build Selected**.

9 Click the **Wireframe Rendering** button on the **Graphics** toolbar to get a better view of the interior parts.

To remove an unnecessary internal boundary proceed as follows.

Ignore Faces 1 (igf1)

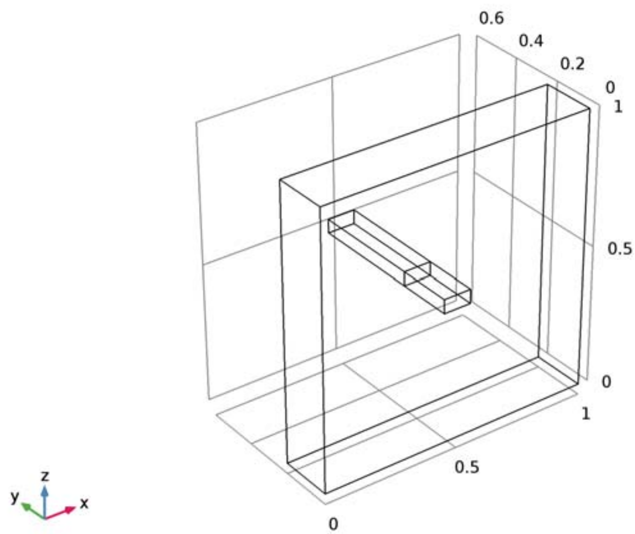
1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Faces**.

2 On the object **fin**, select Boundary 11 only.

Note that you can create the selection by clicking the **Paste Selection** button and typing the indices in the dialog box that opens.

3 Right-click **Ignore Faces 1 (igf1)** and choose **Build Selected**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.



DEFINITIONS

Create a set of selections for use when setting up the physics. First, select the boundaries inside the region $y \geq 0.1$.

Box 1

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for Box, type `Internal` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **y minimum** text field, type `0.1`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Next, select all boundaries inside the region $y \leq 0.1$.

Box 2

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for Box, type `External` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **y maximum** text field, type `0.1`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, type `Insulation` in the **Label** text field.

2 Select Domain 1 only.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	0.1	W/(m·K)	Basic
Density	rho	500	kg/m³	Basic
Heat capacity at constant pressure	Cp	1700	J/(kg·K)	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Iron` in the **Label** text field.

3 Select Domain 2 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	50	W/(m·K)	Basic
Density	rho	7800	kg/m³	Basic
Heat capacity at constant pressure	Cp	460	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Heat Flux 1

1 On 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 **Internal**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `1/0.10`.

6 In the T_{ext} text field, type `1[degC]`.

Heat Flux 2

1 On 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 **External**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `1/0.10`.

6 In the T_{ext} text field, type `0[degC]`.

MESH 1

Because the greatest variations of temperature are expected at the iron bar's external boundary, refine the mesh in this region. Use default settings in the remaining domains.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

Free Tetrahedral 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

Size 1

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Domain**.

3 Select Domain 2 only.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

5 Click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot group shows the temperature distribution; compare with Figure 2.

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

Derived Values

Follow the steps below to calculate the maximum temperature on the external surface and the heat flux between the internal and external sides. Compare the values with the expected results listed in Table 1.

Surface Maximum 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Maximum>Surface Maximum**.

2 In the **Settings** window for Surface Maximum, locate the **Selection** section.

3 From the **Selection** list, choose **External**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
T	degC	Temperature

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The displayed value should be close to 0.805 °C.

RESULTS

Surface Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 In the **Settings** window for Surface Integration, locate the **Selection** section.

3 From the **Selection** list, choose **External**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Boundary fluxes>ht.q0 - Inward heat flux**.

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The measured flux should be close to 0.540 W.

Created in COMSOL Multiphysics 5.2a

Thermal Bridges in Building Construction—3D Structure Between Two Floors

Introduction

The European standard EN ISO 10211:2007 for thermal bridges in building constructions provides four test cases—two 2D and two 3D—for validating a numerical method (Ref. 1). If the values obtained by a method conform to the results of all these four cases, the method is classified as a *three-dimensional steady-state high precision method*.

COMSOL Multiphysics successfully passes all the test cases described by the standard and is hence classified as a *three-dimensional steady-state high precision method*. This document presents an implementation of the first 3D model (Case 3).

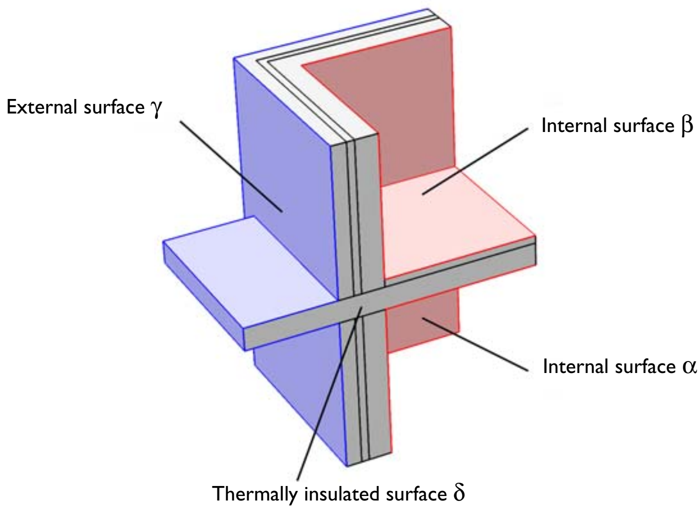


Figure 1: Geometry and boundary conditions of ISO 10211:2007 test case 3.

This tutorial studies the heat conduction in a building structure separating two floors from the external environment. The structure's surfaces are divided into four parts:

- the lower level, α;
- the upper level, β;
- the outside, γ;
- and the remaining thermally insulated surfaces, δ.

The values of interest for validation are the lowest temperatures at surfaces α and β, and the heat fluxes through α, β, and γ.

Model Definition

Figure 1 illustrates the geometry. The external surface is at 0 °C and the interior surface is at 20 °C. Four materials with distinct thermal conductivities k are used in the structure. The horizontal block separating the two floors has the highest thermal conductivity (2.5 W/(m·K)). It crosses the wall, thereby creating a thermal bridge in the structure.

The surfaces α, β, and γ are subject to convective heat flux. The ISO 10211:2007 standard specifies the values of the thermal resistance, R, which is related to the heat transfer coefficient, h, according to

$$h = \frac{1}{R}$$

Results and Discussion

Figure 2 shows the temperature profile. The heat losses are greater near the thermal bridge formed by the horizontal block that crosses the wall.

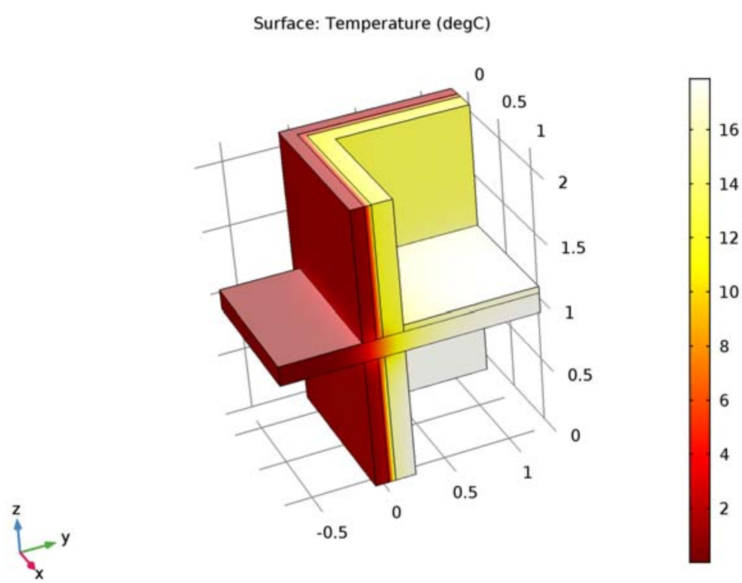


Figure 2: Temperature distribution of ISO 10211:2007 test case 3.

The numerical results of COMSOL Multiphysics are compared with the expected values provided by ISO 10211:2007 (Ref. 1) in Table 1.

TABLE 1: COMPARISON BETWEEN EXPECTED VALUES AND COMPUTED VALUES

MEASURED QUANTITY	EXPECTED VALUES	COMPUTED VALUES	DIFFERENCE
Minimum temperature on α	11.32 °C	11.318 °C	0.002 °C
Minimum temperature on β	11.11 °C	11.107 °C	0.003 °C
Heat flux through α	46.09 W	46.196 W	0.23%
Heat flux through β	13.89 W	13.92 W	0.22%
Heat flux through γ	59.98 W	60.116 W	0.23%

The maximum permissible differences to pass this test case are 0.1 °C for temperature and 1% for heat flux. The measured values are completely consistent and meet the validation criteria.

As shown in Figure 3 and Figure 4, the minimum temperature of the surfaces α and β are located at their respective corners.

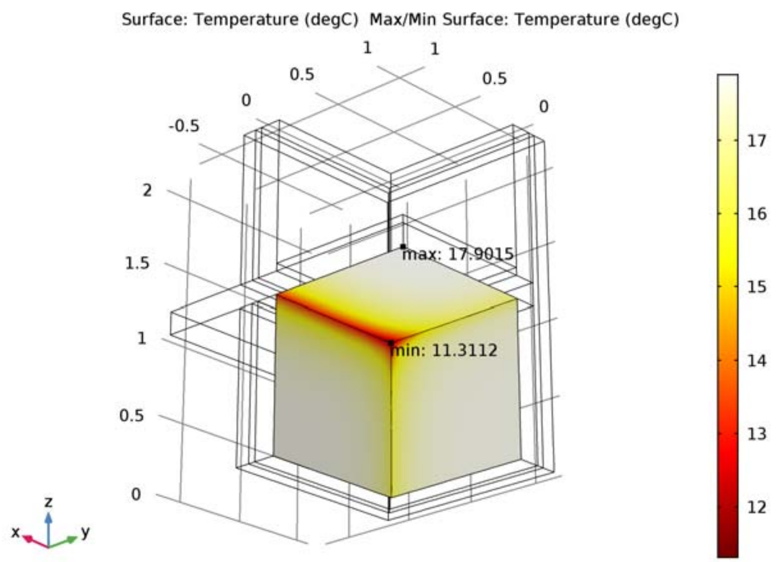


Figure 3: Minimum and maximum temperatures on surface α, ISO 10211:2007 test case 3.

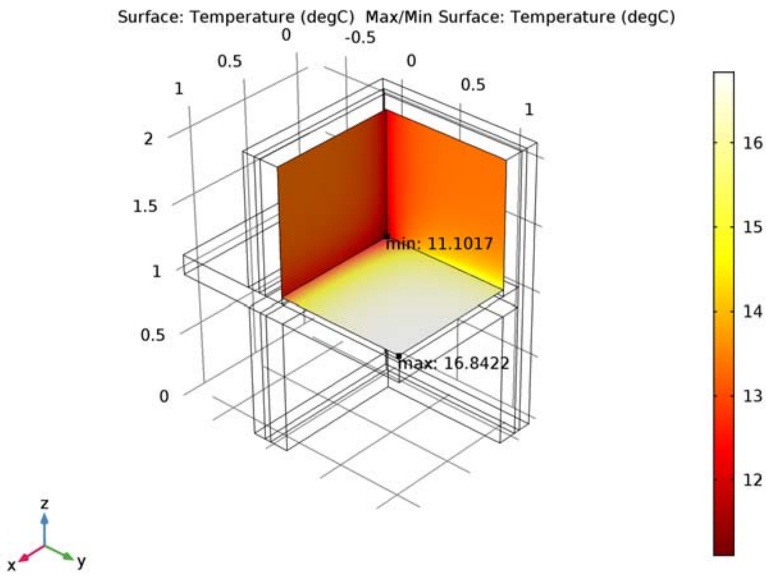


Figure 4: Minimum and maximum temperatures on surface β, *ISO 10211:2007 test case 3.*

Reference

1. European Committee for Standardization, *EN ISO 10211, Thermal bridges in building construction – Heat flows and surface temperatures – Detailed calculations (ISO 10211:2007)*, Appendix A, pp. 30–36, 2007.

Application Library path: `Heat_Transfer_Module/Buildings_and_Constructions/thermal_bridge_3d_two_floors`

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Define the geometrical parameters.

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
wall_t1	5[cm]	0.05 m	Insulation layer thickness
wall_t2	30[cm]	0.3 m	Wall thickness
rect1	1.2[m]	1.2 m	Wall first rectangle basis
rect2	1.3[m]	1.3 m	Wall second rectangle basis
rect_shift	-10[cm]	-0.1 m	Shift for the rectangles
wall_h	2.15[m]	2.15 m	Wall height
blk_w	1.15[m]	1.15 m	Rectangle horizontal block width
blk_d	1.9[m]	1.9 m	Rectangle horizontal block depth
blk_h	0.15[m]	0.15 m	Rectangle horizontal block height
blk_shiftx	5[cm]	0.05 m	Rectangle horizontal block x-shift
blk_shifty	-0.7[m]	-0.7 m	Rectangle horizontal block y-shift
blk_shiftz	1[m]	1 m	Rectangle horizontal block z-shift
sq_l	1[m]	1 m	Square horizontal block side length

Name	Expression	Value	Description
sq_h	5[cm]	0.05 m	Square horizontal block height
sq_shift	0.2[m]	0.2 m	Square horizontal block shift

GEOMETRY 1

To build the walls separating the external and internal surfaces, create the cross section geometry and extrude it.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, click **Show Work Plane**.

Plane Geometry

The first two rectangles below correspond to the insulation layer of the wall.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `rect1`.

4 In the **Height** text field, type `wall_t1`.

5 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `wall_t1`.

4 In the **Height** text field, type `rect1`.

5 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Union 1 (uni1)

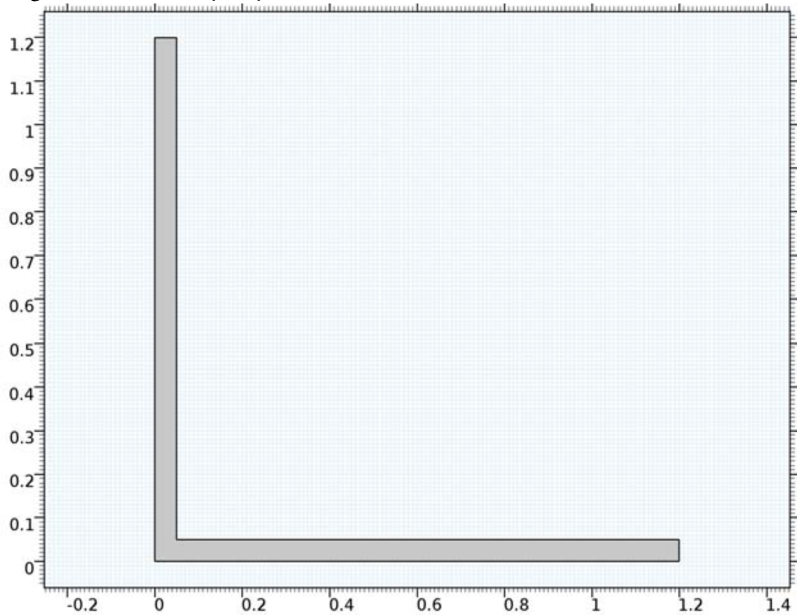
1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for Union, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 Right-click **Union 1 (uni1)** and choose **Build Selected**.



Plane Geometry

Build the remaining layers of the walls in a similar manner.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `rect2`.

4 In the **Height** text field, type `wall_t2`.

5 Locate the **Position** section. In the **xw** text field, type `rect_shift`.

6 In the **yw** text field, type `rect_shift`.

7 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

Rectangle 4 (r4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `wall_t2`.

4 In the **Height** text field, type `rect2`.

5 Locate the **Position** section. In the **xw** text field, type `rect_shift`.

6 In the **yw** text field, type `rect_shift`.

7 Right-click **Rectangle 4 (r4)** and choose **Build Selected**.

Union 2 (uni2)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **r4** and **r3** only.

3 In the **Settings** window for Union, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 Right-click **Union 2 (uni2)** and choose **Build Selected**.

6 In the **Model Builder** window, click **Geometry 1**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

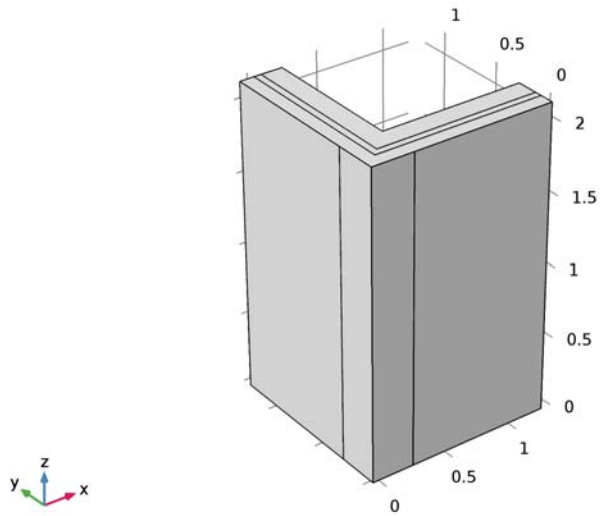
2 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (m)
`wall_h`

4 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.



Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

This block separates the two floors of the structure.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Width** text field, type `blk_w`.

4 In the **Depth** text field, type `blk_d`.

5 In the **Height** text field, type `blk_h`.

6 Locate the **Position** section. In the **x** text field, type `blk_shiftx`.

7 In the **y** text field, type `blk_shifty`.

8 In the **z** text field, type `blk_shiftz`.

9 Right-click **Block 1 (blk1)** and choose **Build Selected**.

To remove unnecessary edges, you need to remove the parts of the walls that intersect this block. To do so, use the boolean operation **Difference** to subtract a copy of the block from the walls. Begin by creating the copy.

10 Right-click **Block 1 (blk1)** and choose **Duplicate**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **ext1** only.

The object labeled `ext1` is made up of the walls previously obtained by extrusion.

3 In the **Settings** window for Difference, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **blk1** only.

Block 3 (blk3)

1 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Block**.

This block corresponds to the floor of the inside upper level.

3 In the **Settings** window for Block, locate the **Size and Shape** section.

4 In the **Width** text field, type `sq_l`.

5 In the **Depth** text field, type `sq_l`.

6 In the **Height** text field, type `sq_h`.

7 Locate the **Position** section. In the **x** text field, type `sq_shift`.

8 In the **y** text field, type `sq_shift`.

9 In the **z** text field, type `blk_shiftz+blk_h`.

Ignore Edges 1 (ige1)

1 Right-click **Block 3 (blk3)** and choose **Build Selected**.

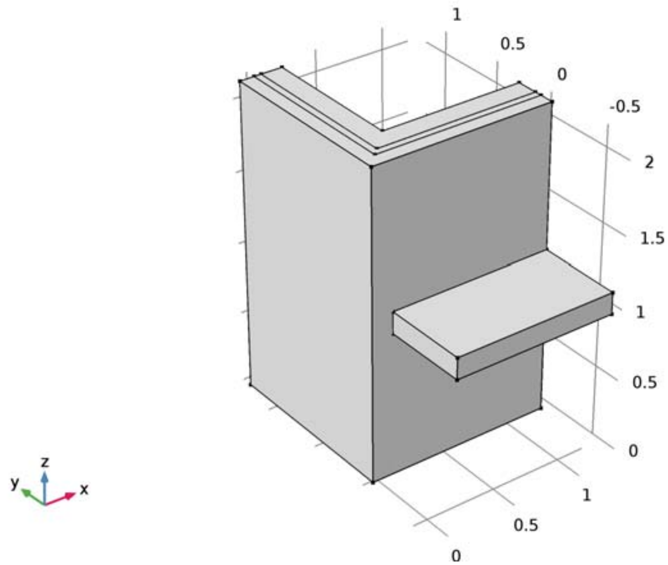
2 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

In the first steps of the geometry sequence, six unused vertical edges were created on the walls. They are responsible for unnecessary constraints on the mesh and they generate extra boundaries by splitting some faces. For these reasons, follow the instructions below to remove them.

3 On the object **fin**, select Edges 6, 17, 33, 38, 60, and 63 only.

To reach the edges, click the **Wireframe Rendering** button on the **Graphics** toolbar. Note that you can make the selection by clicking the **Paste Selection** button and typing the indices in the dialog box that opens.

4 Right-click **Ignore Edges 1 (ige1)** and choose **Build Selected**.



MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, type `Interior Wall` in the **Label** text field.

2 Select Domains 4 and 5 only.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	0.7	W/(m·K)	Basic
Density	rho	1700	kg/m³	Basic
Heat capacity at constant pressure	Cp	800	J/(kg·K)	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Isolation` in the **Label** text field.

3 Select Domain 2 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	0.04	W/(m·K)	Basic
Density	rho	200	kg/m³	Basic
Heat capacity at constant pressure	Cp	1000	J/(kg·K)	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Exterior Wall` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	1	W/(m·K)	Basic
Density	rho	2000	kg/m³	Basic
Heat capacity at constant pressure	Cp	1000	J/(kg·K)	Basic

Material 4 (mat4)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Horizontal Structure` in the **Label** text field.

3 Select Domain 3 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	2.5	W/(m·K)	Basic
Density	rho	5000	kg/m³	Basic
Heat capacity at constant pressure	Cp	600	J/(kg·K)	Basic

Material 5 (mat5)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Floor` in the **Label** text field.

3 Select Domain 6 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	1	W/(m·K)	Basic
Density	rho	1000	kg/m³	Basic
Heat capacity at constant pressure	Cp	800	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 33–35 only.

3 In the **Settings** window for Heat Flux, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `alpha` in the **Selection name** text field.

6 Click **OK**.

7 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

8 Click the **Convective heat flux** button.

9 In the h text field, type `1/0.2`.

10 In the T_{ext} text field, type `20[degC]`.

Heat Flux 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 39–41 only.

3 In the **Settings** window for Heat Flux, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `beta` in the **Selection name** text field.

6 Click **OK**.

7 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

8 Click the **Convective heat flux** button.

9 In the h text field, type `1/0.2`.

10 In the T_{ext} text field, type `15[degC]`.

Heat Flux 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 1, 2, and 11–14 only.

3 In the **Settings** window for Heat Flux, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `gamma` in the **Selection name** text field.

6 Click **OK**.

7 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

8 Click the **Convective heat flux** button.

9 In the h text field, type `1/0.05`.

10 In the T_{ext} text field, type `0[degC]`.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot group shows the temperature distribution.

Surface

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 On the **Temperature (ht)** toolbar, click **Plot**.

Derived Values

Follow the steps below to find the minimum temperatures of α and β as well as the heat flux of α, β, and γ.

Surface Minimum 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Minimum>Surface Minimum**.

2 In the **Settings** window for Surface Minimum, locate the **Selection** section.

3 From the **Selection** list, choose **alpha**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
T	degC	Temperature

5 Click **Evaluate**.

Derived Values

The displayed value should be close to 11.32 °C.

Surface Minimum 2

1 On the **Results** toolbar, click **More Derived Values** and choose **Minimum>Surface Minimum**.

2 In the **Settings** window for Surface Minimum, locate the **Selection** section.

3 From the **Selection** list, choose **beta**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
T	degC	Temperature

5 Click **Evaluate**.

Derived Values

The displayed value should be close to 11.11 °C.

Surface Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 In the **Settings** window for Surface Integration, locate the **Selection** section.

3 From the **Selection** list, choose **alpha**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Boundary fluxes>ht.q0 - Inward heat flux**.

5 Click **Evaluate**.

Derived Values

The displayed value should be close to 46.09 W.

Surface Integration 2

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.
2 In the **Settings** window for Surface Integration, locate the **Selection** section.
3 From the **Selection** list, choose **beta**.
4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Boundary fluxes>ht.q0 - Inward heat flux**.
5 Click **Evaluate**.

Derived Values

The displayed value should be close to 13.89 W.

Surface Integration 3

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.
2 In the **Settings** window for Surface Integration, locate the **Selection** section.
3 From the **Selection** list, choose **gamma**.
4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Boundary fluxes>ht.q0 - Inward heat flux**.
5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

   The displayed value should be close to 59.98 W.

RESULTS

Temperature (ht)

To plot the location of the minimum temperature on α, follow the instructions below.

Study 1/Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Results>Data Sets** node.
2 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **alpha**.

The resulting data set is restricted to the surfaces α.

3D Plot Group 3

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, type `Minimum Temperature on Alpha` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

1 Right-click **Minimum Temperature on Alpha** and choose **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

Minimum Temperature on Alpha

In the **Model Builder** window, under **Results** click **Minimum Temperature on Alpha**.

Max/Min Surface 1

1 On the **Minimum Temperature on Alpha** toolbar, click **More Plots** and choose **Max/Min Surface**.

2 In the **Settings** window for Max/Min Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 On the **Minimum Temperature on Alpha** toolbar, click **Plot**.

TABLE

1 Go to the **Table** window.

As shown in Figure 3, the minimum temperature is at the corner of α.

RESULTS

Study 1/Solution 1 (2) (sol1)

Now plot the location of the minimum temperature on β.

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Study 1/Solution 1 (2) (sol1)** and choose **Duplicate**.

Selection

1 In the **Model Builder** window, expand the **Study 1/Solution 1 (3) (sol1)** node, then click **Selection**.

2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **beta**.

The resulting data set is restricted to the surfaces β.

Minimum Temperature on Alpha

In the **Model Builder** window, under **Results** right-click **Minimum Temperature on Alpha** and choose **Duplicate**.

Minimum Temperature on Alpha 1

1 In the **Settings** window for 3D Plot Group, type `Minimum Temperature on Beta` in the **Label** text field.

2 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.

3 On the **Minimum Temperature on Beta** toolbar, click **Plot**.

Thermoelectric Leg

Introduction

A thermocouple is made of two different conductors (legs) in contact with each other at one point (junction). When a temperature difference is established between the two legs, then a voltage is established across the junction. Therefore a thermocouple properly calibrated is a temperature sensor and can convert temperature gradients into electric currents. In this validation example, we verify the response of one leg when a current is passed through the device. A cooling effect, known as the Peltier effect, is expected.

Model Definition

The component is 1-by-1-by-6 mm, as shown in Figure 1. The core of the device, the thermoelectric part, is made of bismuth telluride (Bi_2Te_3). It is capped by two thin copper electrodes, 0.1 mm thick.

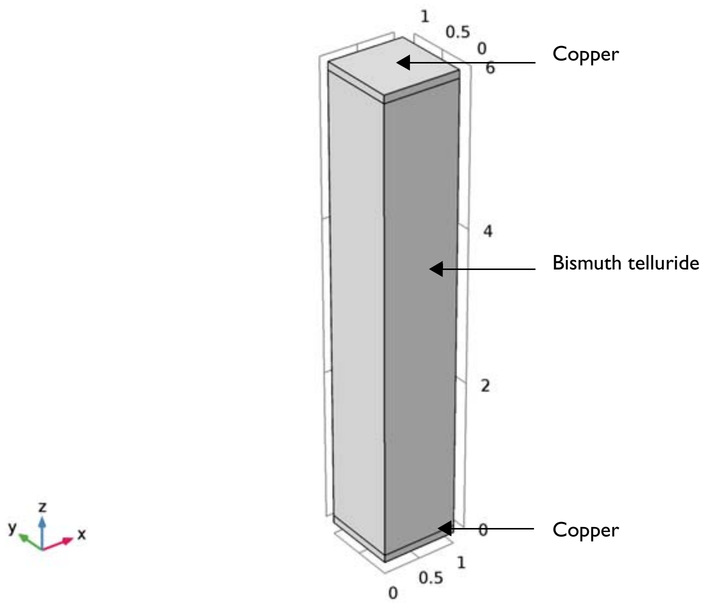


Figure 1: Thermoelectric leg geometry.

The material properties are available in COMSOL Multiphysics Material Library. However, since the properties for bismuth telluride slightly differ from these from the original benchmark, values from Ref. 1 are used in this application.

TABLE 1: MATERIAL PROPERTIES FOR BISMUTH TELLURIDE

Property	Value
Thermal conductivity	1.6 W/(m·K)
Density	7740 kg/m³
Heat capacity at constant pressure	154.4 J/(kg·K)
Electrical conductivity	1.1e5 S/m
Relative permittivity	1
Seebeck coefficient	2e-4 V/K

In addition Seebeck coefficient for copper, $6.5 \cdot 10^{-6}$ V/K, is also taken from Ref. 1.

The bottom electrode surface is held at 0 °C while the top electrode and the lateral surfaces are thermally insulated.

The bottom electrode is electrically grounded at 0 V. The total inward electric current through the top electrode is 0.7 A. The lateral surfaces are electrically insulated.

Results and Discussion

The current circulating in the thermoelectric device is responsible for the cooling effect shown in Figure 2. The temperature field is in complete agreement with the results from Ref. 1.

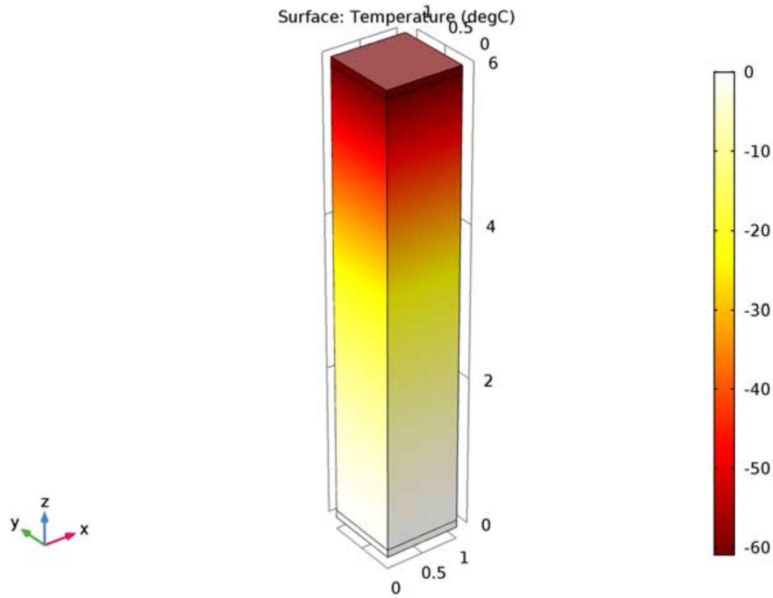


Figure 2: Temperature field on the thermoelectric leg surface.

Figure 3 shows the isothermal surfaces and the heat flux which is in the same direction as the electric current (from the top to the bottom).

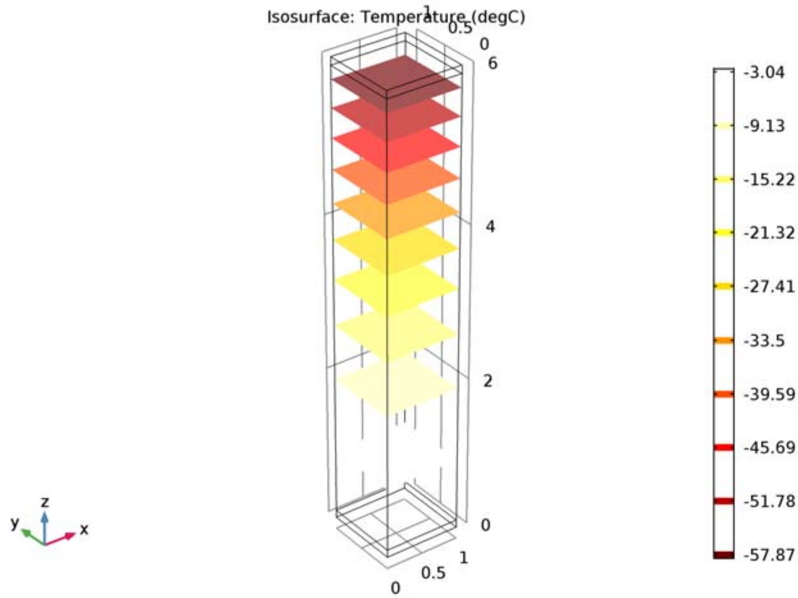


Figure 3: Isothermal surfaces in the thermoelectric leg.

The top level electrode reaches an electric potential of around 49.1 mV due to the inward current density set on this boundary. This corresponds to the value presented in Ref. 1.

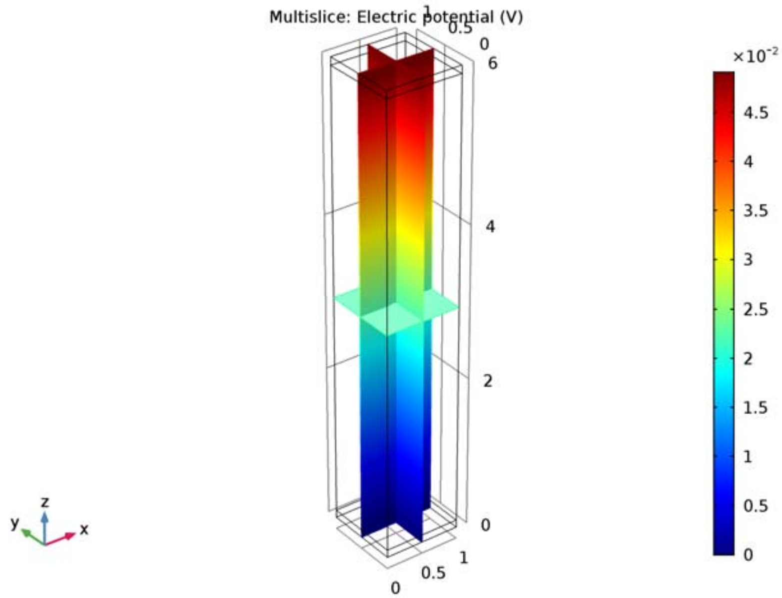


Figure 4: Electric potential in the thermoelectric leg.

Reference

1. M. Jaegle, Multiphysics Simulation of Thermoelectric Systems, "Modeling of Peltier-Cooling and Thermoelectric Generation," *Proc. COMSOL Conf. 2008 Hanover*, 2008.

Application Library path: `Heat_Transfer_Module/Verification_Examples/thermoelectric_leg`

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 **Heat Transfer>Thermoelectric Effect**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for Block, locate the **Size and Shape** section.

3 In the **Height** text field, type `6`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`0.1`

5 Find the **Layer position** subsection. Select the **Top** check box.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Now define the parameters that will be used for the model. The inward current density, J_0, corresponds to a total current of 0.7 A through a 1x1 mm square.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`T0`	`0[degC]`	273.15 K	`Temperature reference`
`J0`	`0.7[A]/(1[mm])^2`	7E5 A/m²	`Inward current density`

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1. In the **Settings** window for Material, type `Bismuth Telluride - Bi2Te3` in the **Label** text field.
2. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`1.6`	W/(m·K)	Basic
Density	rho	`7740`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`154.4`	J/(kg·K)	Basic
Electrical conductivity	sigma	`1.1e5`	S/m	Basic
Relative permittivity	epsilonr	`1`	1	Basic
Seebeck coefficient	S	`2e-4`	V/K	Basic

ADD MATERIAL

1. On 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>Copper**.
4. Click **Add to Component** in the window toolbar.

MATERIALS

Copper (mat2)

1. On the **Home** toolbar, click **Add Material** to close the **Add Material** window.
2. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat2)**.
3. Select Domains 1 and 3 only.

4 In the **Settings** window for Material, locate the **Material Contents** section.

5 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Seebeck coefficient	S	6.5e-6	V/K	Basic

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

3 Select Boundary 3 only.

4 In the **Settings** window for Temperature, locate the **Temperature** section.

5 In the T_0 text field, type `T0`.

ELECTRIC CURRENTS (EC)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundary 3 only.

Normal Current Density 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Current Density**.

2 Select Boundary 10 only.

3 In the **Settings** window for Normal Current Density, locate the **Normal Current Density** section.

4 In the J_n text field, type `J0`.

Due to the geometrical properties, replacing the default tetrahedral mesh by a hexahedral sweep mesh is more suited.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Swept**.

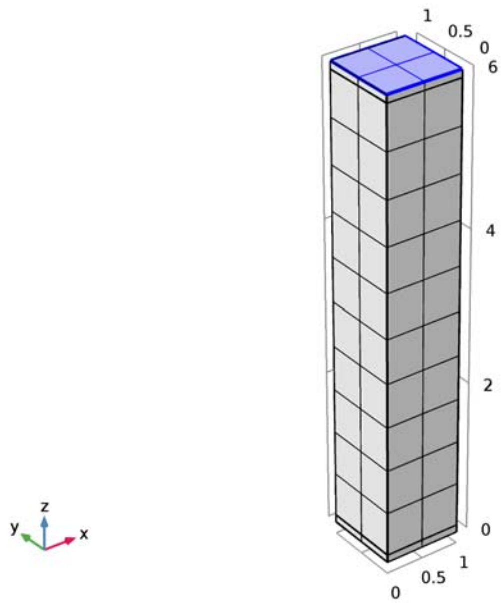
Swept 1

1 In the **Settings** window for Swept, click to expand the **Source faces** section.

2 Select Boundary 10 only.

Now visualize the mesh and compare it with the figure below.

3 Click **Build All**.



STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot shows the temperature field; compare with Figure 2.

Surface

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 On the **Temperature (ht)** toolbar, click **Plot**.

Isothermal Contours (ht)

The second default plot, corresponding to Figure 3, shows the isothermal surfaces.

Isosurface

1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Isosurface**.
2 In the **Settings** window for Isosurface, locate the **Expression** section.
3 From the **Unit** list, choose **degC**.
4 On the **Isothermal Contours (ht)** toolbar, click **Plot**.

Electric Potential (ec)

The third default plot group shows the electric potential distribution as in Figure 4.

Created in COMSOL Multiphysics 5.2a

Tin Melting Front

Introduction

This example demonstrates how to model phase transition by a moving boundary interface according to the Stefan problem. It is adapted from the benchmark study in Ref. 1.

A square cavity containing both solid and liquid tin is submitted to a temperature difference between left and right boundaries. Fluid and solid parts are solved in separate domains sharing a moving melting front (see Figure 1). The position of this boundary through time is calculated according to the Stefan energy balance condition.

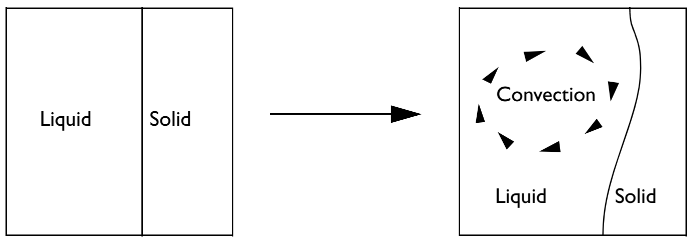


Figure 1: Square cavity with moving phase interface

In the melt, motion generated by natural convection is expected due to the temperature gradient. This motion, in turn, influences the front displacement.

Model Definition

The geometry presented in Figure 2 shows a square of side length 10 cm filled with pure tin. The left and right boundaries are maintained at 508 K and 503 K, respectively.

Because the fusion temperature of pure tin is 505 K, both liquid and solid phases co-exist in the square.

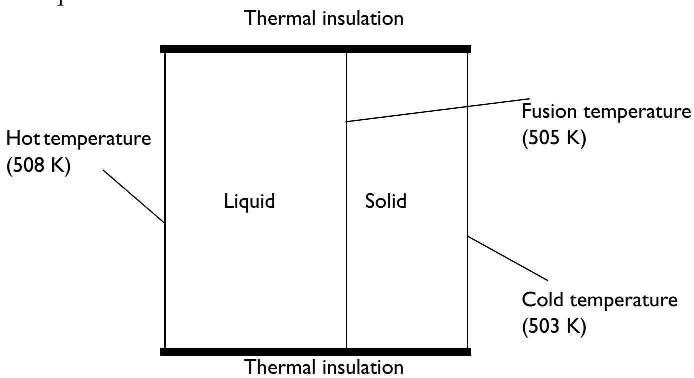


Figure 2: Geometry and boundary conditions at starting time.

The initial temperature distribution is assumed to vary linearly in the horizontal direction as shown in Figure 3.

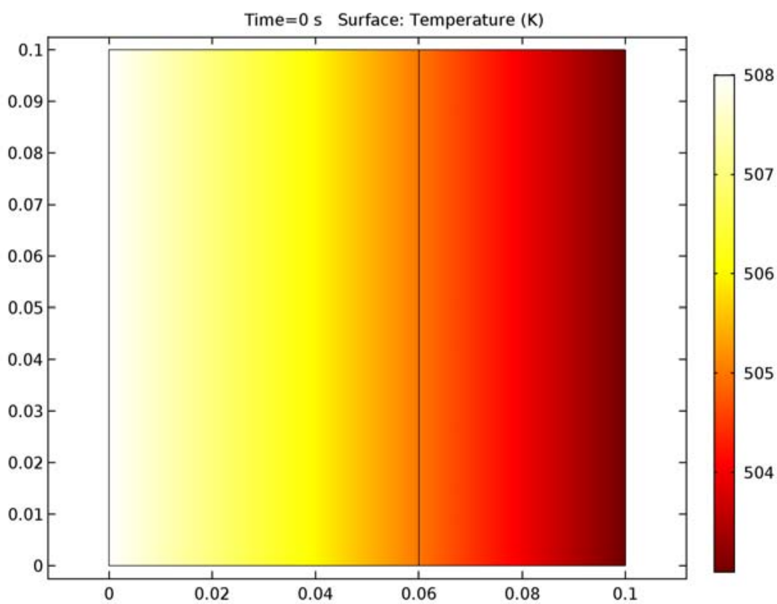


Figure 3: Initial temperature profile.

The melting front is the vertical line located at $x = 6$ cm where the temperature is 505 K.

The liquid part on the left is governed by the Navier-Stokes equations in the Boussinesq approximation as described in Gravity and The Boussinesq Approximation sections in the *CFD Module User's Guide*, with $T_{\mathrm{ref}} = T_{\mathrm{f}}$ the fusion temperature of tin. The reduced pressure formulation is used to enhance the accuracy of the buoyancy forces since they are relatively small compared to the other terms in the momentum balance.

As the metal melts, the solid-liquid interface moves toward the solid side. The energy balance at this front is expressed by

$$\rho_0 \Delta H \mathbf{v} \cdot \mathbf{n} = (\Phi_{\mathrm{l}} - \Phi_{\mathrm{s}}) \cdot \mathbf{n} \tag{1}$$

where ΔH is the latent heat of fusion, equal to 60 kJ/kg, $\mathbf{v}$ (m/s) is the front velocity vector, $\mathbf{n}$ is the normal vector at the front, and Φ_{l} and Φ_{s} (W/m^2) are the heat fluxes coming from the liquid and solid sides, respectively (see Figure 4).

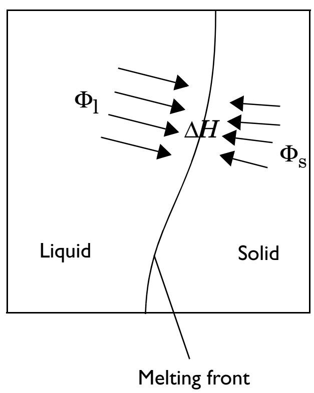


Figure 4: Heat fluxes at the melting front

Table 1 reviews the material properties of tin (Ref. 2) used in this model.

TABLE 1: MATERIAL PROPERTIES OF TIN

PARAMETER	DESCRIPTION	VALUE
ρ_0	Density	7500 kg/m^3
C_p	Heat capacity	200 J/(kg·K)
k	Thermal conductivity	60 W/(m·K)
α_p	Coefficient of thermal expansion	$2.67 \cdot 10^{-4}$ K^{-1}
ν	Kinematic viscosity	$8.0 \cdot 10^{-7}$ m^2/s

TABLE 1: MATERIAL PROPERTIES OF TIN

PARAMETER	DESCRIPTION	VALUE
T_f	Fusion temperature	505 K
ΔH	Latent heat of fusion	60 kJ/kg

Results and Discussion

Figure 5 shows the velocity profile in the fluid domain. The convective cell due to buoyancy increases the melting speed at the upper part of the cavity.

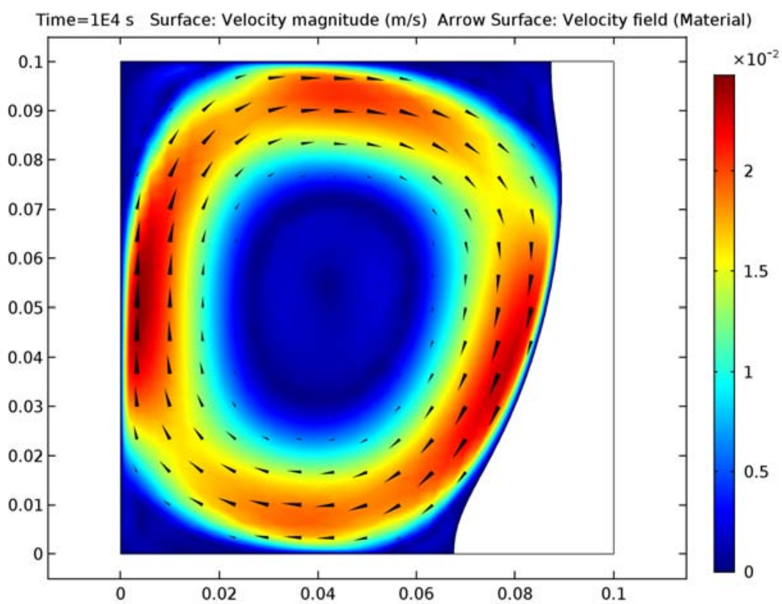


Figure 5: Velocity profile in the fluid at the end of the simulation.

At the end of the simulation, the melting front does not move anymore because balance between left and right adjacent fluxes has been reached.

In Figure 6, the temperature profile is represented jointly by a heat flux arrow plot.

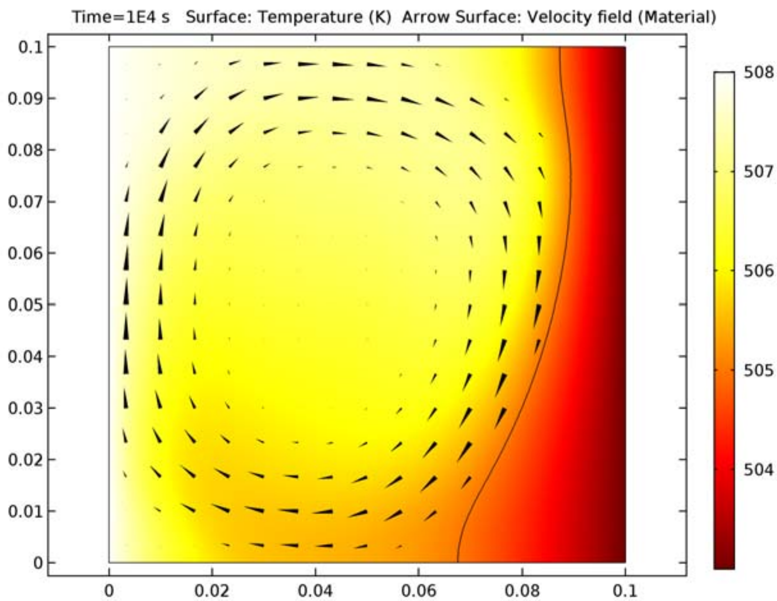


Figure 6: Temperature profile at the end of the simulation.

Notes About the COMSOL Implementation

The quantities Φ_l and Φ_s, illustrated in Figure 4, can be computed using the `up` and `down` operators. The components of $\Phi_l - \Phi_s$ would then be given by `up(ht.tfluxx)-down(ht.tfluxx)` and `up(ht.tfluxy)-down(ht.tfluxy)`. However, this method evaluates the temperature gradient which may lead to imprecisions due to the mesh discretization. Instead, the quantity $(\Phi_l - \Phi_s) \cdot \mathbf{n}$, involved in Equation 1, is more precisely evaluated through the Lagrange multiplier for temperature, `T_lm`. This variable is available when weak constraints are enabled in the region of interest, as it is the case here with the fixed temperature constraint at the melting front. For more information about weak constraints, refer to the section Weak Constraint in the *COMSOL Multiphysics Reference Manual*.

To handle the melting front movement, a mesh deformation is necessary. During such a transformation, matter from solid tin is removed while the same amount of liquid tin is added to the fluid. The appropriate tool for deforming the mesh without reflecting any expansion or contraction effects to the material properties is the Deformed Geometry interface.

References

1. F. Wolff and R. Viskanta, “Solidification of a Pure Metal at a Vertical Wall in the Presence of Liquid Superheat,” *Int.J. Heat and Mass Transfer*, vol. 31, no. 8, pp. 1735–1744, 1988.

2. V. Alexiades, N. Hannoun, and T.Z. Mai, “Tin Melting: Effect of Grid Size and Scheme on the Numerical Solution,” *Proc. 5th Mississippi State Conf. Differential Equations and Computational Simulations*, pp. 55–69, 2003.

Application Library path: `Heat_Transfer_Module/Phase_Change/tin_melting_front`

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**.

2 In the **Select Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
k_Sn	60[W/(m*K)]	60 W/(m·K)	Thermal conductivity
Cp_Sn	200[J/(kg*K)]	200 J/(kg·K)	Specific heat capacity
alpha_Sn	2.67e-4[1/K]	2.67E-4 1/K	Coefficient of thermal expansion
nu_Sn	8e-7[m^2/s]	8E-7 m^2/s	Kinematic viscosity
rho_Sn	7500[kg/m^3]	7500 kg/m^3	Density
DelH	60[kJ/kg]	60000 J/kg	Latent heat of fusion
Tf	505[K]	505 K	Melting point
Th	508[K]	508 K	Hot wall temperature
Tc	503[K]	503 K	Cold wall temperature
p_ref	100[Pa]	100 Pa	Gauge pressure

GEOMETRY 1

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.
2 In the **Settings** window for Square, locate the **Size** section.
3 In the **Side length** text field, type 0.1.
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.06

5 Select the **Layers to the left** check box.
6 Clear the **Layers on bottom** check box.
7 Click **Build All Objects**.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, type `Tin (Solid)` in the **Label** text field.

2 Select Domain 2 only.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`k_Sn`	W/(m·K)	Basic
Density	rho	`rho_Sn`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`Cp_Sn`	J/(kg·K)	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Tin (Liquid)` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Geometric Entity Selection** section. Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Tin (liquid)` in the **Selection name** text field.

6 Click **OK**.

Before defining the material properties of liquid tin, indicate which is the fluid domain to let COMSOL Multiphysics flag what properties you need to specify.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for Laminar Flow, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Tin (liquid)**.

HEAT TRANSFER (HT)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer (ht)**.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Fluid 1**.

2 In the **Settings** window for Fluid, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Tin (liquid)**.

MATERIALS

Tin (Liquid) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Tin (Liquid) (mat2)**.

2 In the **Settings** window for Material, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`k_Sn`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	`Cp_Sn`	J/(kg·K)	Basic
Ratio of specific heats	gamma	`1.4`	1	Basic
Dynamic viscosity	mu	`rho_Sn*nu_Sn`	Pa·s	Basic

HEAT TRANSFER (HT)

Initial Values 1

Define the initial temperature as a function of `Xg`, the first coordinate on the undeformed geometry.

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `Th-Xg/0.1[m]*(Th-Tc)` in the T text field.

LAMINAR FLOW (SPF)

On the **Physics** toolbar, click **Heat Transfer (ht)** and choose **Laminar Flow (spf)**.

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 In the p text field, type `p_ref`.

4 In the **Model Builder** window, click **Laminar Flow (spf)**.

5 In the **Settings** window for Laminar Flow, locate the **Physical Model** section.

6 From the **Compressibility** list, choose **Incompressible flow**.

7 Select the **Include gravity** check box.

8 Select the **Use reduced pressure** check box.

9 Find the **Reference values** subsection. In the T_{ref} text field, type `Tf`.

HEAT TRANSFER (HT)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer (ht)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 1 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `Th`.

Temperature 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 4 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `Tf`.

Show advanced physics options as follows to enable weak constraints on the melting front. This creates the Lagrange multiplier for temperature, which evaluates the heat flux jump between the adjacent liquid and solid domains more accurately.

5 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

6 Click to expand the **Constraint settings** section. Locate the **Constraint Settings** section. Select the **Use weak constraints** check box.

Temperature 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 7 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `Tc`.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Pressure Point Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 1 only.

3 In the **Settings** window for Pressure Point Constraint, locate the **Pressure Constraint** section.

4 In the p_0 text field, type `p_ref`.

The model only contains information about the pressure gradient and estimates the pressure field up to a constant. To define this constant, you arbitrarily fix the pressure at a point.

MULTIPHYSICS

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Non-Isothermal Flow 1 (nitf1)**.

2 In the **Settings** window for Non-Isothermal Flow, locate the **Material Properties** section.

3 From the **Specify density** list, choose **Custom, linearized density**.

4 In the ρ_{ref} text field, type `rho_Sn`.

5 In the α_p text field, type `alpha_Sn`.

DEFORMED GEOMETRY (DG)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Deformed Geometry (dg)**.

In the **Model Builder** window, under **Component 1 (comp1)** click **Deformed Geometry (dg)**.

Free Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.

2 In the **Settings** window for Free Deformation, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

Prescribed Mesh Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundaries 1 and 7 only.

3 In the **Settings** window for Prescribed Mesh Displacement, locate the **Prescribed Mesh Displacement** section.

4 Clear the **Prescribed y displacement** check box.

Prescribed Mesh Displacement 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Deformed Geometry (dg)** click **Prescribed Mesh Displacement 1**.
2. In the **Settings** window for Prescribed Mesh Displacement, locate the **Prescribed Mesh Displacement** section.
3. Clear the **Prescribed x displacement** check box.

Prescribed Normal Mesh Velocity 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Normal Mesh Velocity**.
2. Select Boundary 4 only.
3. In the **Settings** window for Prescribed Normal Mesh Velocity, locate the **Normal Mesh Velocity** section.
4. In the v_{n} text field, type `T_lm[W/m^2]/(rho_Sn*DelH)`.

   The variable `T_lm` is the Lagrange multiplier for temperature.

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for Mesh, locate the **Mesh Settings** section.
3. From the **Element size** list, choose **Fine**.
4. Click **Build All**.

STUDY 1

Step 1: Time Dependent

1. In the **Settings** window for Time Dependent, locate the **Study Settings** section.
2. Click **Range**.
3. In the **Range** dialog box, type `100` in the **Step** text field.
4. In the **Stop** text field, type `10000`.
5. Click **Replace**.
6. In the **Settings** window for Time Dependent, locate the **Study Settings** section.
7. Select the **Relative tolerance** check box.
8. In the associated text field, type `1e-3`.

Solution 1 (sol1)

1. On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
3 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.
4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.
5 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

This default plot shows the temperature profile. To reproduce Figure 3, add arrows of the heat flux field to see the relation between temperature and velocity.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Arrow Surface**.
2 In the **Settings** window for Arrow Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>u,v - Velocity field (Material)**.
3 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.
4 From the **Color** list, choose **Black**.
5 On the **Temperature (ht)** toolbar, click **Plot**.

Velocity (spf)

This default plot shows the velocity profile in the fluid region. To reproduce Figure 5, add arrows of the velocity field to visualize the convective flow direction.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Velocity (spf)** and choose **Arrow Surface**.
2 In the **Settings** window for Arrow Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Laminar Flow>Velocity and pressure>u,v - Velocity field (Material)**.
3 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.
4 From the **Color** list, choose **Black**.
5 On the **Velocity (spf)** toolbar, click **Plot**.

Finally, plot the mesh deformation as follows.

2D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Mesh Deformation` in the **Label** text field.

Mesh 1

1 Right-click **Mesh Deformation** and choose **Mesh**.

2 In the **Settings** window for Mesh, locate the **Color** section.

3 From the **Element color** list, choose **None**.

4 From the **Wireframe color** list, choose **Blue**.

Filter 1

1 Right-click **Results>Mesh Deformation>Mesh 1** and choose **Filter**.

2 In the **Settings** window for Filter, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `dom==1`.

This logical expression restricts the plot to the fluid domain.

4 On the **Mesh Deformation** toolbar, click **Plot**.

Mesh 1

Right-click **Mesh 1** and choose **Duplicate**.

Mesh 2

1 In the **Settings** window for Mesh, locate the **Color** section.

2 From the **Wireframe color** list, choose **Red**.

Filter 1

1 In the **Model Builder** window, expand the **Mesh 2** node, then click **Filter 1**.

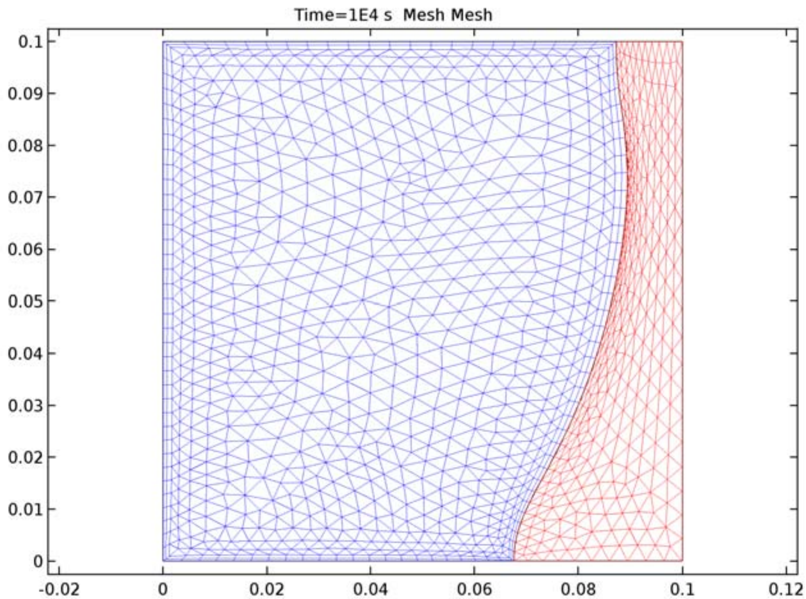
2 In the **Settings** window for Filter, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `dom==2`.

This logical expression filters the solid domain.

4 On the **Mesh Deformation** toolbar, click **Plot**.

The plot should look like that in the figure below.



Created in COMSOL Multiphysics 5.2a

Hepatic Tumor Ablation

Introduction

One method for removing cancerous tumors from healthy tissue is to heat the malignant tissue to a critical temperature that kills the cancer cells. This example accomplishes the localized heating by inserting a four-armed electric probe through which an electric current runs. Equations for the electric field for this case appear in the Electric Currents interface, and this example couples them to the bioheat equation, which models the temperature field in the tissue. The heat source resulting from the electric field is also known as *resistive heating* or *Joule heating*. The original model comes from S. Tungjitkusolmun and others (Ref. 1), but we have made some simplifications. For instance, while the original uses RF heating (with AC currents), the COMSOL Multiphysics model approximates the energy with DC currents.

This medical procedure removes the tumorous tissue by heating it above 45 °C to 50 °C. Doing so requires a local heat source, which physicians create by inserting a small electric probe. The probe is made of a trocar (the main rod) and four electrode arms as shown in Figure 1. The trocar is electrically insulated except near the electrode arms.

An electric current through the probe creates an electric field in the tissue. The field is strongest in the immediate vicinity of the probe and generates resistive heating, which dominates around the probe's electrode arms because of the strong electric field.

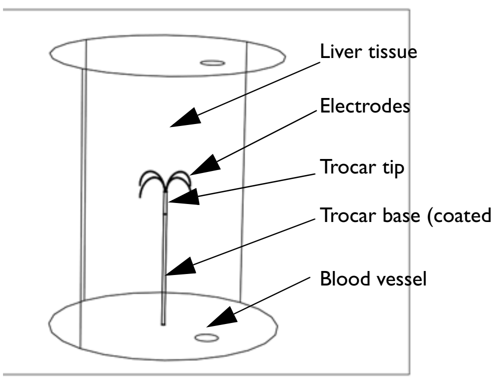


Figure 1: Cylindrical modeling domain with the four-armed electric probe in the middle, which is located next to a large blood vessel.

Model Definition

This tutorial uses the Bioheat Transfer interface, the Electric Currents interface and a multiphysics feature, Electromagnetic Heat Source, to implement a transient analysis.

The standard temperature unit in COMSOL Multiphysics is kelvin (K). This tutorial uses the Celsius temperature scale, which is more convenient for models involving the bioheat equation.

The model approximates the body tissue with a large cylinder and assumes that its boundary temperature remains at 37 °C during the entire procedure. The tumor is located near the center of the cylinder and has the same thermal properties as the surrounding tissue. The model locates the probe along the cylinder's center line such that its electrodes span the region where the tumor is located. The geometry also includes a large blood vessel.

HEAT TRANSFER

The bioheat equation governs heat transfer in the tissue

$$\delta_{ts}\rho C\frac{\partial T}{\partial t} + \nabla\cdot(-k\nabla T) = \rho_b C_b \omega_b (T_b - T) + Q_{met} + Q_{ext}$$

where δ_{ts} is a time-scaling coefficient; ρ is the tissue density (kg/m^3); C is the tissue's specific heat (J/(kg·K)); and k is its thermal conductivity (W/(m·K)). On the right side of the equality, ρ_b gives the blood's density (kg/m^3); C_b is the blood's specific heat (J/(kg·K)); ω_b is its perfusion rate (1/s); T_b is the arterial blood temperature (K); while Q_{met} and Q_{ext} are the heat sources from metabolism and spatial heating, respectively (W/m^3).

In this example, the bioheat equation also models heat transfer in various parts of the probe with the appropriate values for the specific heat, C (J/(kg·K)), and thermal conductivity, k (W/(m·K)). For these parts, all terms on the right-hand side are zero.

The model next sets the boundary conditions at the outer boundaries of the cylinder and at the walls of the blood vessel to a temperature of 37 °C. Assume heat flux continuity on all other boundaries.

The initial temperature equals 37 °C in all domains.

In addition to the heat transfer equation this model provides a calculation of the tissue damage integral. This gives an idea about the degree of tissue injury α during the process, based on the Arrhenius equation:

$$\frac{d\alpha}{dt} = A\exp\left(-\frac{dE}{RT}\right)$$

where A is the frequency factor (s^{-1}) and dE is the activation energy for irreversible damage reaction (J/mol). These two parameters are dependent on the type of tissue. The fraction of necrotic tissue, θ_d, is then expressed by:

$$\theta_d = 1 - \exp(-\alpha)$$

ELECTRIC CURRENT

The governing equation for the Electric Currents interface is

$$-\nabla \cdot (\sigma \nabla V - \mathbf{J}^e) = Q_j$$

where V is the potential (V), σ the electrical conductivity (S/m), $\mathbf{J}^e$ an externally generated current density (A/m^2), Q_j the current source (A/m^3).

In this model both $\mathbf{J}^e$ and Q_j are zero. The governing equation therefore simplifies into:

$$-\nabla \cdot (\sigma \nabla V) = 0 .$$

The boundary conditions at the cylinder's outer boundaries is ground (0 V potential). At the electrode boundaries the potential equals 22 V. Assume continuity for all other boundaries.

The boundary conditions for the Electric Currents interface are:

$V = 0$	on the cylinder wall
$V = V_0$	on the electrode surfaces
$\mathbf{n} \cdot (J_1 - J_2) = 0$	on all other boundaries

The boundary conditions for the bioheat equation are:

$T = T_b$	on the cylinder wall and blood-vessel wall
$\mathbf{n} \cdot (k_1 \nabla T_1 - k_2 \nabla T_2) = 0$	on all interior boundaries

The model solves the above equations with the given boundary conditions to obtain the temperature field as a function of time.

Results and Discussion

The model shows how the temperature increases with time in the tissue around the electrode.

The slice plot in Figure 2 illustrates the temperature field at the end of the procedure.

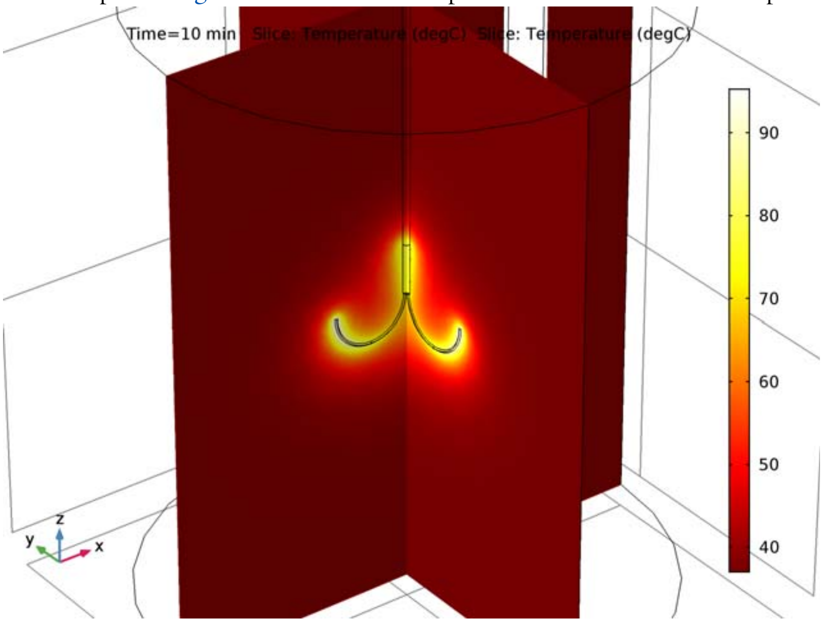


Figure 2: Temperature field at time = 10 minutes.

Figure 3 shows the temperature at the tip of one of the electrode arms. The temperature rises quickly until it reaches a steady-state temperature of about 90 °C.

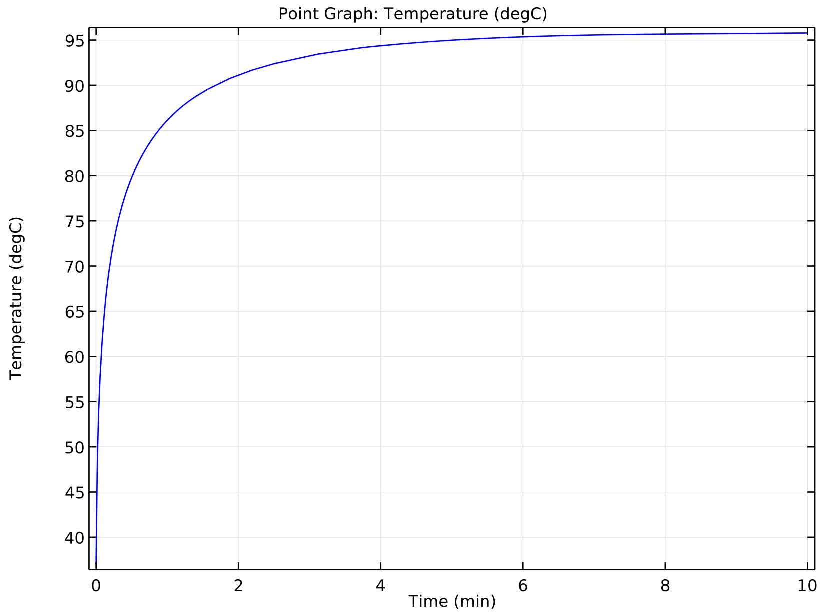


Figure 3: Temperature versus time at the tip of one of the electrode arms.

It is also interesting to visualize the region where cancer cells die, that is, where the temperature has reached at least 50 °C. You can visualize this area with an isosurface for that temperature; Figure 4 shows one after 10 minutes.

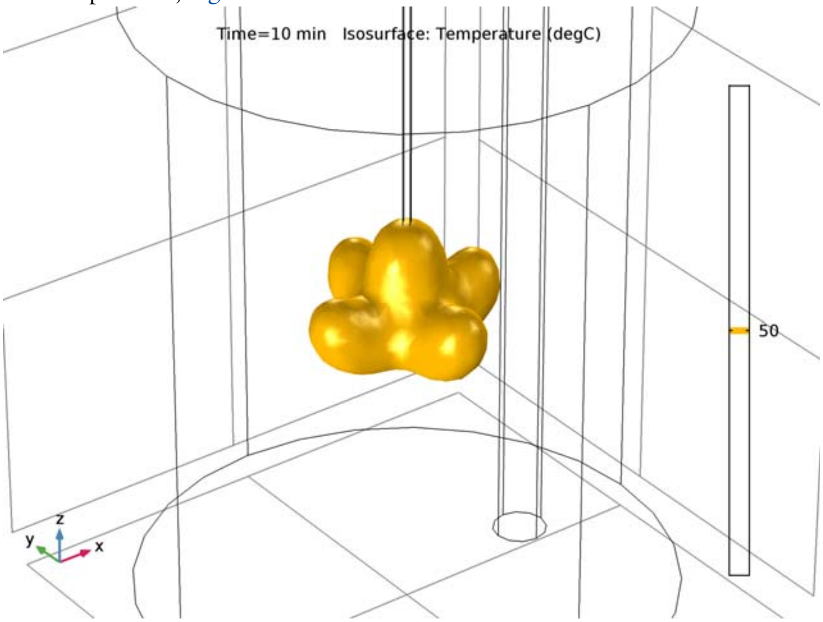


Figure 4: Visualization of the region that has reached 50°C after 10 minutes.

In addition to the previous figure, you can visualize the fraction of necrotic tissue in the slice plot of Figure 5.

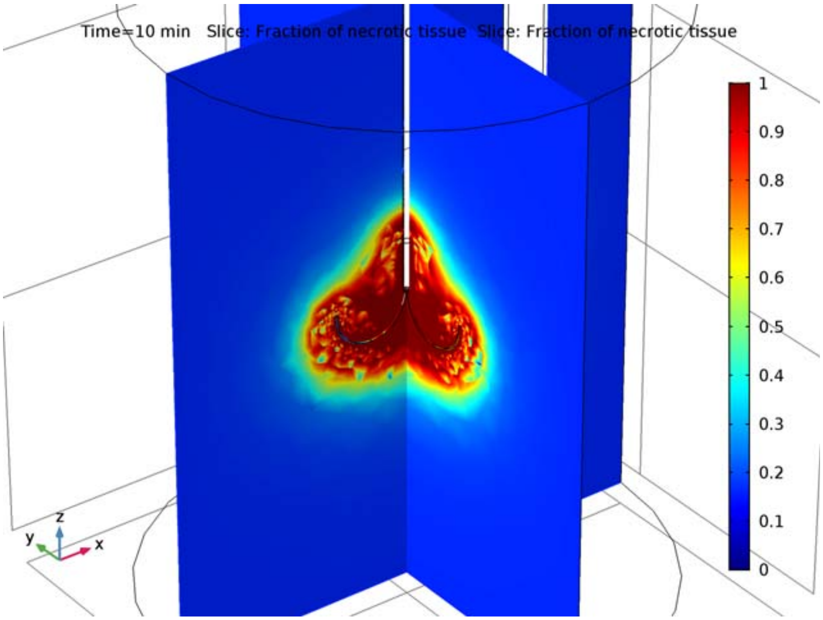


Figure 5: Fraction of necrotic tissue.

Finally, Figure 6 shows the fraction of necrotic tissue at three different points above the electrode arm. Observe that necrosis happens faster next to the electrode and the trocar tip.

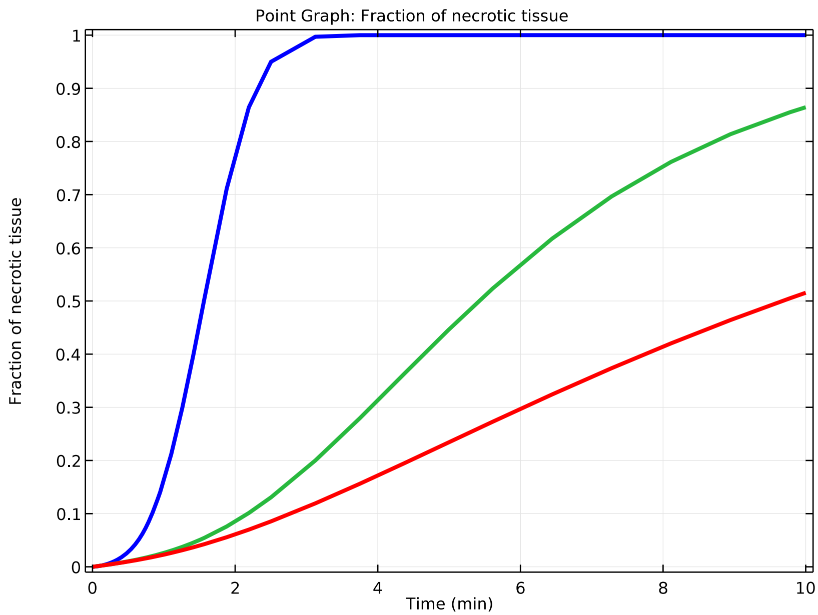


Figure 6: Fraction of necrotic tissue at three points above the electrode arm.

Reference

1. S. Tungjitkusolmun, S. Tyler Staelin, D. Haemmerich, J.Z. Tsai, H. Cao, J.G. Webster, F.T. Lee, Jr., D.M. Mahvi, and V.R. Vorperian, "Three-Dimensional Finite Element Analyses for Radio-Frequency Hepatic Tumor Ablation," *IEEE Transactions on Biomedical Engineering*, vol. 49, no. 1, 2002.

Application Library path: `Heat_Transfer_Module/Medical_Technology/tumor_ablation`

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 **AC/DC>Electric Currents (ec)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Heat Transfer>Bioheat Transfer (ht)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.
8 Click **Done**.

GLOBAL DEFINITIONS

First, define the global parameters of the model and the geometry.

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
rho_b	1000[kg/m^3]	1000 kg/m³	Density, blood
c_b	4180[J/(kg*K)]	4180 J/(kg·K)	Heat capacity, blood
omega_b	6.4e-3[1/s]	0.0064 1/s	Blood perfusion rate
T_b	37[degC]	310.15 K	Arterial blood temperature
T0	37[degC]	310.15 K	Initial and boundary temperature
V0	22[V]	22 V	Electric voltage
xc_v	26[mm]	0.026 m	Vessel cylinder center x coordinate
a_time	10[min]	600 s	Ablation time

GEOMETRY 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for Geometry, locate the **Units** section.
3. From the **Length unit** list, choose **mm**.

Cylinder 1 (cyl1)

1. On the **Geometry** toolbar, click **Cylinder**.
2. In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3. In the **Radius** text field, type `0.9144`.
4. In the **Height** text field, type `60`.
5. Locate the **Position** section. In the **z** text field, type `60`.
6. Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	10

7. Clear the **Layers on side** check box.
8. Select the **Layers on bottom** check box.

Torus 1 (tor1)

1. Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.
2. On the **Geometry** toolbar, click **Torus**.
3. In the **Settings** window for Torus, locate the **Size and Shape** section.
4. In the **Major radius** text field, type `7.5`.
5. In the **Minor radius** text field, type `0.2667`.
6. In the **Revolution angle** text field, type `180`.
7. Locate the **Position** section. In the **x** text field, type `8`.
8. In the **z** text field, type `60`.
9. Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.
10. Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

Rotate 1 (rot1)

1. Right-click **Torus 1 (tor1)** and choose **Build Selected**.
2. On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.
3. Select the object **tor1** only.
4. In the **Settings** window for Rotate, locate the **Rotation Angle** section.

5 Click **Range**.

6 In the **Range** dialog box, type `0` in the **Start** text field.

7 In the **Step** text field, type `90`.

8 In the **Stop** text field, type `270`.

9 Click **Replace**.

10 Right-click **Rotate 1 (rot1)** and choose **Build Selected**.

11 Click the **Zoom Extents** button on the **Graphics** toolbar.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `5`.

4 In the **Height** text field, type `120`.

5 Locate the **Position** section. In the **x** text field, type `xc_v`.

6 Right-click **Cylinder 2 (cyl2)** and choose **Build Selected**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Cylinder 3 (cyl3)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `50`.

4 In the **Height** text field, type `120`.

5 On the **Geometry** toolbar, click **Build All**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Select the **All domains** check box.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Liver Tissue` in the **Label** text field.

3 Select Domain 1 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Blood Vessel` in the **Label** text field.

3 Select Domain 8 only.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Electrodes` in the **Label** text field.

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

4 Select Domains 2 and 5–7 only.

Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Trocar Tip` in the **Label** text field.

3 Select Domain 3 only.

Explicit 6

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Trocar Base` in the **Label** text field.

3 Select Domain 4 only.

Union 1

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for Union, type `Trocar` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, In the **Selections to add** list, choose **Electrodes**, **Trocar Tip**, and **Trocar Base**.

5 Click **OK**.

Union 2

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for Union, type `Tissue and Trocar` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, In the **Selections to add** list, choose **Liver Tissue** and **Trocar**.

5 Click **OK**.

Adjacent 1

1 On the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for Adjacent, type `Tissue and Trocar, Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Tissue and Trocar** in the **Input selections** list.

5 Click **OK**.

Adjacent 2

1 On the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for Adjacent, type `Trocar Tip and Electrodes, Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, In the **Input selections** list, choose **Electrodes** and **Trocar Tip**.

5 Click **OK**.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Bioheat>Liver (human)**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Liver (human) (mat1)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Liver (human) (mat1)**.

3 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

4 From the **Selection** list, choose **Liver Tissue**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`0.333[S/m]`	S/m	Basic
Relative permittivity	epsilonr	`1`	1	Basic

Material 2 (mat2)

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 `Blood` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Blood Vessel**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`0.667[S/m]`	S/m	Basic
Relative permittivity	epsilonr	`1`	1	Basic
Thermal conductivity	k	`0.543[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`rho_b`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`c_b`	J/(kg·K)	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Electrodes` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Electrodes**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`1e8[S/m]`	S/m	Basic
Relative permittivity	epsilonr	`1`	1	Basic
Thermal conductivity	k	`18[W/(m*K)]`	W/(m·K)	Basic

Property	Name	Value	Unit	Property group
Density	rho	6450[kg/m^3]	kg/m³	Basic
Heat capacity at constant pressure	Cp	840[J/(kg*K)]	J/(kg·K)	Basic

Material 4 (mat4)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Trocar Tip` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Trocar Tip**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	4e6[S/m]	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic
Thermal conductivity	k	71[W/(m*K)]	W/(m·K)	Basic
Density	rho	21500[kg/m^3]	kg/m³	Basic
Heat capacity at constant pressure	Cp	132[J/(kg*K)]	J/(kg·K)	Basic

Material 5 (mat5)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Trocar Base` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Trocar Base**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	1e-5[S/m]	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic
Thermal conductivity	k	0.026[W/(m*K)]	W/(m·K)	Basic

Property	Name	Value	Unit	Property group
Density	rho	`70[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`1045[J/(kg*K)]`	J/(kg·K)	Basic

ELECTRIC CURRENTS (EC)

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

   At the time scale of the tumor ablation process, the electric field is stationary. Change the equation form accordingly.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

3 In the **Settings** window for Electric Currents, click to expand the **Equation** section.

4 From the **Equation form** list, choose **Stationary**.

   To reduce the size of the computation problem, select a lower element order.

5 Click to expand the **Discretization** section. From the **Electric potential** list, choose **Linear**.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 In the **Settings** window for Ground, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior Boundaries**.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for Electric Potential, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Trocar Tip and Electrodes, Exterior Boundaries**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `V0`.

BIOHEAT TRANSFER (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Bioheat Transfer (ht)**.

2 In the **Settings** window for Bioheat Transfer, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Tissue and Trocar**.

Biological Tissue 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Biological Tissue 1**.

2 In the **Settings** window for Biological Tissue, locate the **Damaged Tissue** section.

3 Select the **Include damage integral analysis** check box.

4 From the **Damage integral form** list, choose **Energy absorption**.

Bioheat 1

1 In the **Model Builder** window, expand the **Biological Tissue 1** node, then click **Bioheat 1**.

2 In the **Settings** window for Bioheat, locate the **Bioheat** section.

3 In the T_b text field, type `T_b`.

4 In the $C_{p,\,b}$ text field, type `c_b`.

5 In the ω_b text field, type `omega_b`.

6 In the ρ_b text field, type `rho_b`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, type `T0` in the T text field.

3 In the **Model Builder** window, click **Bioheat Transfer (ht)**.

Solid 1

1 On the **Physics** toolbar, click **Domains** and choose **Solid**.

2 In the **Settings** window for Solid, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Trocar**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 In the **Settings** window for Temperature, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Tissue and Trocar, Exterior Boundaries**.

4 Locate the **Temperature** section. In the T_0 text field, type `T_b`.

MULTIPHYSICS

Electromagnetic Heat Source 1 (emh1)

1 On the **Physics** toolbar, click **Multiphysics** and choose **Domain>Electromagnetic Heat Source**.

2 In the **Settings** window for Electromagnetic Heat Source, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

Free Tetrahedral 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

Size 1

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2 From the **Geometric entity level** list, choose **Domain**.
3 Select Domains 2 and 5–7 only.
4 Locate the **Element Size** section. Click the **Custom** button.
5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6 In the associated text field, type `0.38`.
7 Select the **Minimum element size** check box.
8 In the associated text field, type `0.35`.

Free Tetrahedral 1

Right-click **Free Tetrahedral 1** and choose **Size**.

Size 2

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
2 From the **Geometric entity level** list, choose **Domain**.
3 Select Domains 3 and 4 only.
4 Locate the **Element Size** section. Click the **Custom** button.
5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6 In the associated text field, type `1.3`.
7 Select the **Minimum element size** check box.
8 In the associated text field, type `1.1`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 **Resolution of narrow regions** text field, type `0`.

5 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

2 From the **Time unit** list, choose **min**.

3 In the **Times** text field, type `range(0,a_time/4,a_time)`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. Select the **Initial step** check box.

5 In the associated text field, type `0.01[s]`.

6 Select the **Maximum step** check box.

7 In the associated text field, type `50[s]`.

Add probes to save the fraction of necrotic tissue and the temperature over time at some specified points.

DEFINITIONS

Domain Point Probe 1

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

2 In the **Settings** window for Domain Point Probe, locate the **Point Selection** section.

3 In row **Coordinates**, set **x** to `-4`.

4 In row **Coordinates**, set **z** to `65`.

5 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

6 In the **Settings** window for Point Probe Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1 (comp1)>Bioheat Transfer>Biological tissue>ht.theta_d - Fraction of necrotic tissue**.

7 Click to expand the **Table and window settings** section. Locate the **Table and Window Settings** section. Click **Plot window**.

8 In the **Model Builder** window, right-click **Domain Point Probe 1** and choose **Duplicate**.
9 In the **Settings** window for Domain Point Probe, locate the **Point Selection** section.
10 In row **Coordinates**, set **x** to `-12`.
11 Right-click **Domain Point Probe 1** and choose **Duplicate**.
12 In the **Settings** window for Domain Point Probe, locate the **Point Selection** section.
13 In row **Coordinates**, set **x** to `-20`.

Domain Point Probe 4

1 On the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.
2 In the **Settings** window for Domain Point Probe, locate the **Point Selection** section.
3 In row **Coordinates**, set **x** to `-0.2667`.
4 In row **Coordinates**, set **y** to `15.5`.
5 In row **Coordinates**, set **z** to `60`.
6 In the **Model Builder** window, expand the **Domain Point Probe 4** node, then click **Point Probe Expression 4 (ppb4)**.
7 In the **Settings** window for Point Probe Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1 (comp1)>Bioheat Transfer>Temperature>T - Temperature**.
8 Locate the **Table and Window Settings** section. Click **Plot window**.
9 Locate the **Expression** section. From the **Table and plot unit** list, choose **degC**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Electric Potential (ec)

The first default plot shows the electric potential on slices.

Temperature (ht)

The second default plot shows the temperature at the final time.

To reproduce the two-slice plot of the temperature at 10 minutes shown in Figure 2, proceed as follows.

Before adding slices, delete the default **Surface** node.

Surface

In the **Model Builder** window, expand the **Temperature (ht)** node.

Temperature (ht)

1 Right-click **Surface** and choose **Delete**.

. Click **Yes** to confirm.

Slice 1

1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Slice**.

2 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Bioheat Transfer> Temperature>T - Temperature**.

3 Locate the **Expression** section. From the **Unit** list, choose **degC**.

4 Locate the **Plane Data** section. In the **Planes** text field, type 1.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

Temperature (ht)

Right-click **Temperature (ht)** and choose **Slice**.

Slice 2

1 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Bioheat Transfer> Temperature>T - Temperature**.

2 Locate the **Expression** section. From the **Unit** list, choose **degC**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

4 In the **Planes** text field, type 1.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

6 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

7 On the **Temperature (ht)** toolbar, click **Plot**.

8 Click the **Zoom In** button on the **Graphics** toolbar.

Isothermal Contours (ht)

The third default plot shows isothermal contours for the final time.

To reproduce Figure 4, modify this plot group as follows:

Isosurface

1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Isosurface**.

2 In the **Settings** window for Isosurface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `50`.

Isothermal Contours (ht)

1 In the **Model Builder** window, under **Results** click **Isothermal Contours (ht)**.

2 On the **Isothermal Contours (ht)** toolbar, click **Plot**.

Probe Plot Group 4

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 4**.

2 In the **Settings** window for 1D Plot Group, type `Damaged Tissue, 1D` in the **Label** text field.

3 On the **Damaged Tissue, 1D** toolbar, click **Plot**.

Generate plots to show the fraction of necrotic tissue.

Probe Plot Group 5

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 5**.

2 In the **Settings** window for 1D Plot Group, type `Temperature at One Electrode Tip` in the **Label** text field.

3 On the **Temperature at One Electrode Tip** toolbar, click **Plot**.

3D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, type `Damaged Tissue, 3D` in the **Label** text field.

Slice 1

1 Right-click **Damaged Tissue, 3D** and choose **Slice**.

2 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Bioheat Transfer> Biological tissue>ht.theta_d - Fraction of necrotic tissue**.

3 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

Slice 2

1 Right-click **Results>Damaged Tissue, 3D>Slice 1** and choose **Duplicate**.
2 In the **Settings** window for Slice, locate the **Plane Data** section.
3 From the **Plane** list, choose **zx-planes**.
4 In the **Planes** text field, type 1.
5 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.
6 On the **Damaged Tissue, 3D** toolbar, click **Plot**.

Natural Convection Cooling of a Vacuum Flask

Introduction

This example solves a pure conduction problem and a free-convection problem in which a vacuum flask holding hot coffee dissipates thermal energy. The main interest is to calculate the flask's cooling power; that is, how much heat it loses per unit time.

Figure 1: Schematic picture of the flask.

The coffee has an initial temperature of 90 °C and cools down over time. The observation period is 10 h. This tutorial compares two different approaches to model natural convection cooling:

- Using heat transfer coefficients to describe the thermal dissipation
- Modeling the convective flow of air outside the flask to describe the thermal dissipation

The first approach describes the outside heat flux using a heat transfer coefficient function from the Heat Transfer Coefficients library included with the Heat Transfer Module. This results in a rather simple model that predicts the stationary cooling well and produces accurate results for temperature distribution and cooling power.

The second approach solves for both the total energy balance and the flow equations of the outside cooling air. This application produces detailed results for the flow field around the flask as well as for the temperature distribution and cooling power. However, it is more complex and requires more computational resources than the first version.

Model Definition

Figure 2 shows the model geometry.

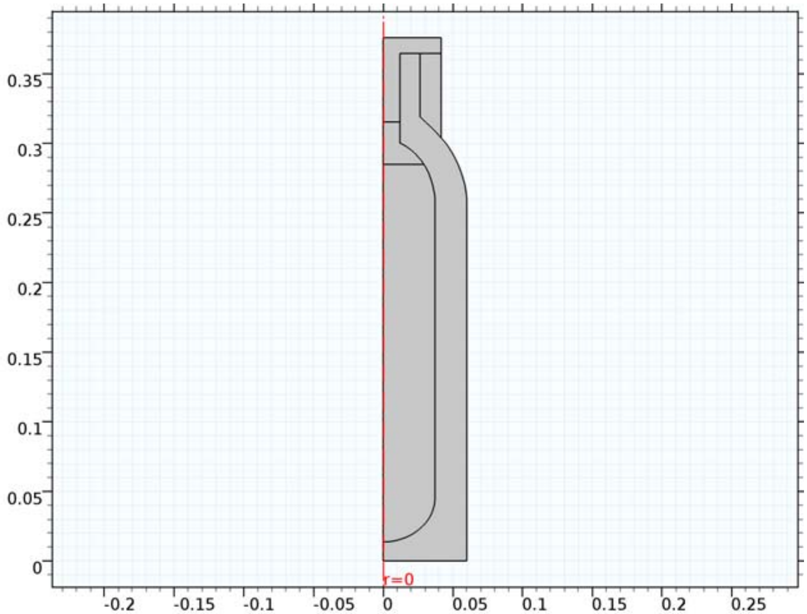


Figure 2: The 2D-axisymmetric representation of the vacuum flask.

CONTROL VOLUME

For the first approach, the model does not include a control volume around the flask to represent the domain of the surrounding air. Instead, it uses a heat transfer coefficient correlation for vertical and horizontal plates.

For the second approach, the model uses a control volume around the flask to represent the domain of the surrounding air. Choosing an appropriate control volume for natural convection models is difficult. Your choice strongly influences the model, the mesh, the convergence, and especially the flow behavior. The real-world air domain surrounding the flask is the entire room or atmosphere in which the flask is placed. Making the rectangle as large as the external room would result in a very large model requiring a supercomputer to solve. At the other extreme, if you make the control volume too small, the solution is affected by the imposed artificial boundary conditions, and there can also be a truncation of flow eddies, making convergence difficult.

An appropriate truncation should resolve the flow field around the flask but avoid modeling a large surrounding. One way to approach this task is to start with a small control volume, set up and solve the model, then expand the control volume, solve the model again, and see if the results change. This example uses a sufficiently large control volume by truncating the air domain at $r = 0.1$ m and $z = 0.5$ m. The boundary condition at the boundaries that are open to large volumes can handle both entering and leaving fluid. The entering fluid has the temperature of the surroundings whereas the leaving fluid has an unknown temperature that results from the cooling effects of the flow field.

MATERIAL PROPERTIES

Next consider the materials that make up the flask model. The flask contains coffee that has almost exactly the same material properties as water. The screw stopper and insulation ring are made of nylon. The flask bottle consists of stainless steel, and the filling material between the inner and outer walls is a plastic foam. The material library includes all materials used in this model except the foam, which you specify manually. Table 1 provides a list of standard foam's thermal properties.

TABLE 1: FOAM MATERIAL PROPERTIES

PROPERTY	VALUE
Conductivity	0.03 W/(m·K)
Density	60 kg/m^3
Heat Capacity	200 J/(kg·K)

HEAT TRANSFER PHYSICS

This example assumes the hot liquid (coffee) to have a uniform temperature distribution that changes only with time. This is a reasonable approximation since the observation period is long and effects of spatially varying temperatures in the liquid are small. The Heat Transfer Module provides the isothermal domain feature. It solves only an additional ordinary differential equation of the form:

$$mC_p\frac{dT}{dt} = Q$$

where m is the total mass of the domain, which can be prescribed or is calculated automatically from the material properties. The source term Q is calculated from the adjacent entities.

The walls of the flask, that are made of steel are modeled as thin conductive layers, so that their thickness does not need to be resolved by the mesh.

Approach 1—Loading a Heat Transfer Coefficient Function

This application uses a simplified approach and solves the time-dependent thermal-conduction equation making use of a heat transfer coefficient, h, to describe the natural convection cooling on the outside surfaces of the flask. This approach is very powerful in many situations, especially if the main interest is not the flow behavior but rather its cooling power. By using the appropriate h correlations, you can generally arrive at accurate results at a very low computational cost. In addition, many correlations are valid for the entire flow regime, from laminar to turbulent flow. This makes it possible to approach the problem directly without predicting whether the flow is laminar or turbulent.

BOUNDARY CONDITIONS

Vertical boundaries along the axis of symmetry have a symmetry condition (zero gradients, set by COMSOL Multiphysics automatically); the bottom is modeled as perfectly insulated (zero flux). The flask surfaces are exposed to air and are cooled by convection. The use of a thin layer feature models the thickness of the steel shell.

The only remaining energy-balance boundary condition is for the flask surface. In the first approach a convective heat-transfer coefficient together with the ambient temperature, 25 °C, describes the heat flux.

CONVECTIVE HEAT TRANSFER COEFFICIENT

The outer surfaces dissipate heat via natural convection. This loss is characterized by the convective heat transfer coefficient, h, which in practice you often determine with empirical handbook correlations. Because these correlations depend on the surface temperature, T_{surface}, engineers must estimate T_{surface} and then iterate between h and T_{surface} to obtain a converged value for h. Most of these correlations require tedious computations and property interpolations that make this iterative process quite unpleasant and labor intensive.

A typical handbook correlation (see Ref. 1) for h for the case of natural convection in air on a vertical heated wall $\mathrm{Ra}_L \le 10^9$ is

$$h = \frac{k \cdot \overline{\mathrm{Nu}_L}}{L}$$

$$\overline{\mathrm{Nu}_L} = 0.68 + \frac{0.670\mathrm{Ra}_L^{1/4}}{\left[1 + \left(\frac{0.492}{\mathrm{Pr}}\right)^{9/16}\right]^{4/9}}$$

where Ra_L and Pr are the Rayleigh and Prandtl dimensionless numbers. A similar relation involving Nusselt numbers holds for inclined and horizontal planes (see The Heat Transfer Coefficients in the *Heat Transfer Module User's Guide* for details).

COMSOL Multiphysics handles these types of nonlinearities internally and adds much convenience to such computations, so there is no need to iterate.

The Heat Transfer Module provides heat transfer coefficient functions that you can access easily in the Convective Heat Flux feature.

Approach 2—Modeling the External Flow

Another approach for simulating the cooled flask is to produce a model that computes the convective velocity field around the flask in detail. Before proceeding with a simulation of this kind, it is a good idea to try to estimate the Rayleigh number because that number influences the choice between assuming laminar flow and applying a turbulence model.

The Rayleigh number describes the ratio between buoyancy and viscous forces in free convection problems. It is defined as

$$\mathrm{Ra} = \frac{g\alpha_p \Delta T h^3}{\kappa\nu}$$

with g as the gravity (SI unit: $\mathrm{m/s^2}$), κ the thermal diffusivity (SI unit: $\mathrm{m^2/s}$), ΔT the Temperature difference (SI unit: K), h the height of the convective object (SI unit: m), α_p the coefficient of thermal expansion (SI unit: 1/K), and ν the kinematic viscosity (SI unit: $\mathrm{m^2/s}$).

The model's length scale is the length of the heated fluid's flow path, in this example 0.5 m. Notice that this value increases if the modeled flow domain is extended in the direction of the flow. ΔT is about 15 K (assuming that the flask surface temperature is 15 °C above the ambient temperature). Together with the material properties of air at atmospheric pressure and T about 25 °C the result is below $1 \cdot 10^9$, which indicates that the flow is still laminar rather than turbulent. Thus, it makes sense to model the flow using a physics interface for laminar flow.

BUOYANCY-DRIVEN FLOW

To model non-isothermal buoyancy-driven flow, the following example implements a buoyancy force in the fluid. It provides a generalized Navier-Stokes formulation that takes varying density into account as well as the energy equation.

The buoyancy forces are included using the Gravity feature described in the Gravity section in the *CFD Module User's Guide*.

BOUNDARY CONDITIONS

When solving flow problems numerically, your engineering intuition is crucial in setting good boundary conditions. In this problem, the warm flask drives vertical air currents along its walls, and they eventually join in a thermal plume above the top of the flask. Air is pulled from the surroundings toward the flask where it eventually feeds into the vertical flow.

The open boundary condition is a boundary condition for the heat and flow equation and can handle incoming flow with ambient temperature and leaving flow with a-priori unknown temperature.

You would expect this flow to be quite weak and therefore do not anticipate any significant changes in dynamic pressure.

Flow Boundary Conditions

- On the top and right boundaries, the normal stress is zero as an open boundary with ambient temperature of 25 °C.
- On the upper-left boundary, the flow domain coincides with the axis of symmetry where the Axial Symmetry condition is applied automatically.
- All other boundaries (the flask surface and the bottom horizontal line) are walls where you use the No slip condition.

Thermal Boundary Conditions

- The top and right boundaries are the exit and entrance of the flow domain respectively where convection dominates; accordingly, use an open boundary condition.
- Again, the top left is described by axial symmetry, which is set by default.
- Assume that the bottom is perfectly insulated.

All other boundaries (the flask surface) have continuity in temperature and flux by default.

Results and Discussion

Figure 3 shows the temperature distribution in both the flask and the surrounding air. However, the temperature results in the solid parts are close to identical for the case of modeling with a heat transfer coefficient.

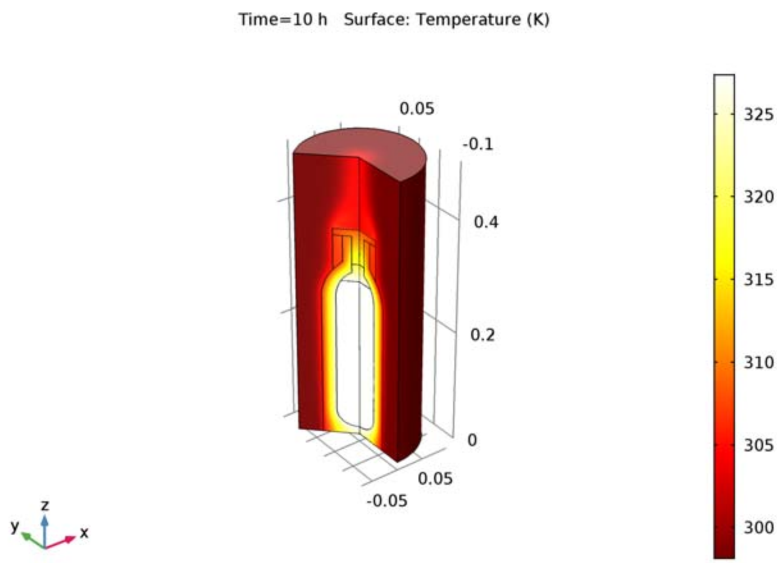


Figure 3: Temperature results for the model including the fluid flow.

One objective of the model is to predict the coffee temperature over time. The plot below shows the results of both approaches and one can see that both results produce almost exactly the same curve.

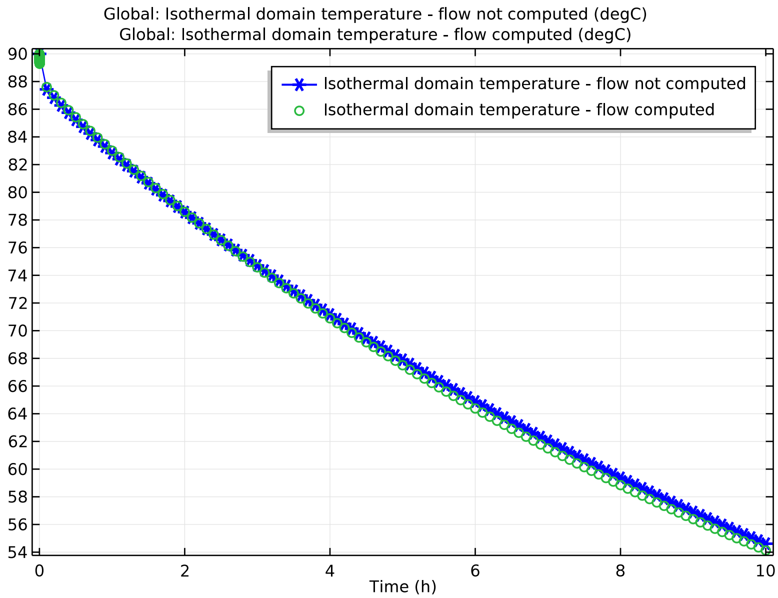


Figure 4: Isothermal domain temperature over time for both approaches.

A second question concerns how the cooling power is distributed on the flask surface. The heat transfer coefficient represents this property. Figure 5 shows a comparison of the predicted distribution of h along the height of the flask between the two models.

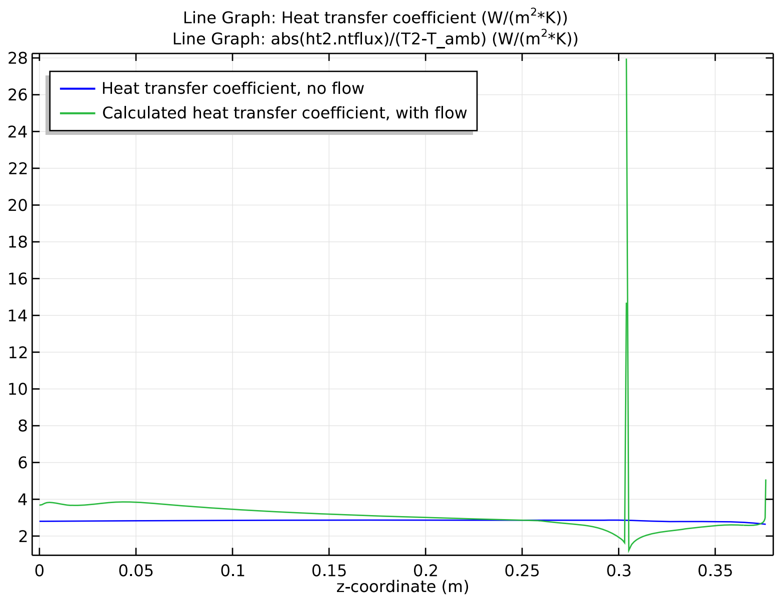


Figure 5: Heat transfer coefficient along the vertical flask walls. Blue line: modeling approach using the heat transfer coefficient library, green line: modeling approach including the fluid flow.

Figure 6 depicts the flow of air around the flask calculated from the flow model. This fluid flow model does a better job at describing local cooling power.

One interesting result is the vortex formed above the lid. It reduces the cooling in this region.

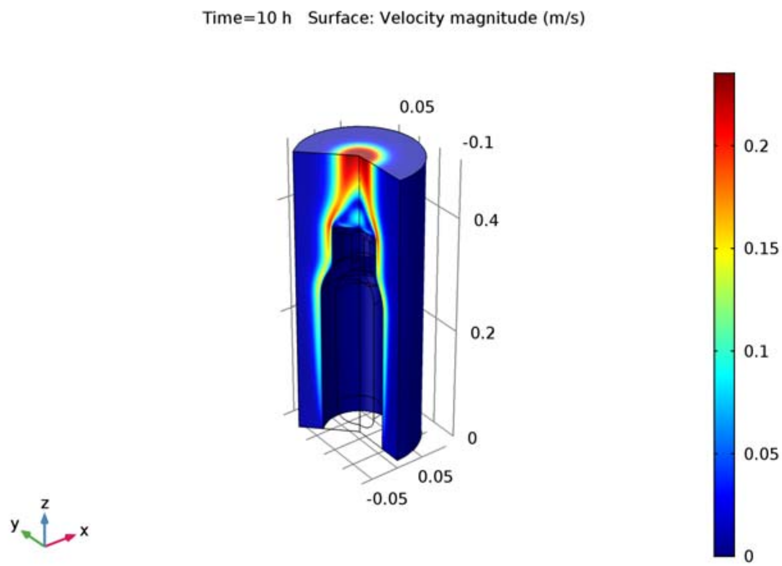


Figure 6: Fluid velocity for air around the flask.

CONCLUSIONS

Using the Convective Heat Flux feature you can easily obtain simulation results. The predicted heat transfer coefficient is in the same range as the results from the model that includes the correlations, and the total cooling power is almost identical.

However, the predefined heat transfer coefficients do not predict the local effects of air flow surrounding the flask. For this purpose, a flow model is more accurate. This means that you can use this type of model to create and calibrate functions for heat transfer coefficients for your geometries. Once calibrated, the functions allow you to use the first approach when solving large-scale and time-dependent models.

Application Library path: `Heat_Transfer_Module/Tutorials,` `_Forced_and_Natural_Convection/vacuum_flask`

Reference

1. F. Incropera, D. Dewitt, T. Bergman, and A. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th ed., John Wiley & Sons, 2007.

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 **Heat Transfer>Heat Transfer in Solids (ht)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1 On the **Geometry** toolbar, click **Insert Sequence**.
2 Browse to the application's Application Libraries folder and double-click the file `vacuum_flask_geom_sequence.mph`.

Bézier Polygon 1 (b1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Bézier Polygon 1 (b1)**.
2 In the **Settings** window for Bézier Polygon, click **Build All Objects**.

   You should now see the geometry shown in Figure 2.

DEFINITIONS

In the following section you define selections which will be needed during the model set up, for example the boundaries that represent the steel shell of the flask and the boundaries that are convectively cooled by the surrounding air.

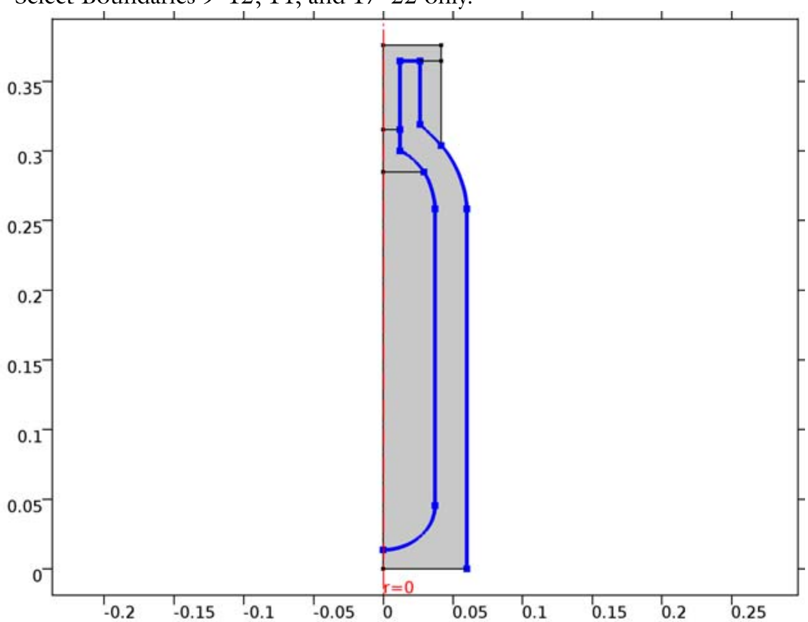
Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Shell` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 9–12, 14, and 17–22 only.



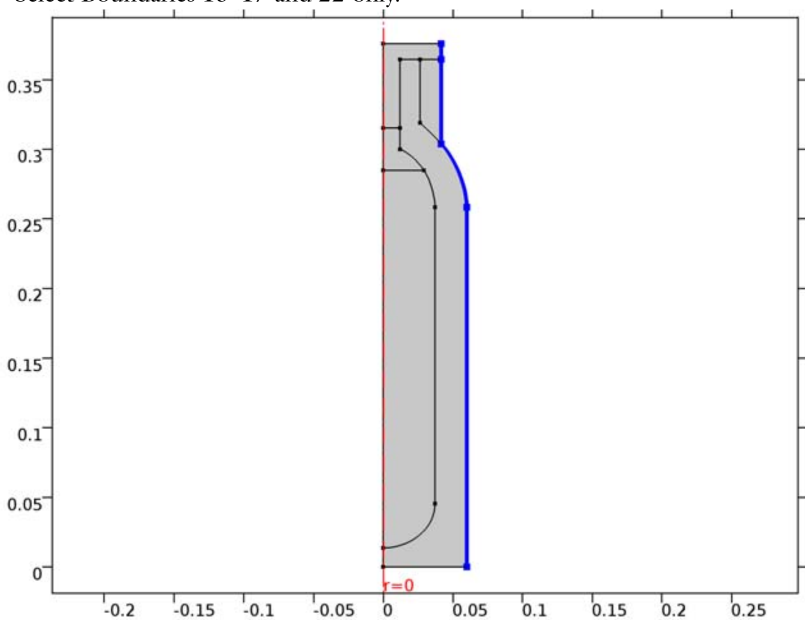
Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Flask, Vertical Walls` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 15–17 and 22 only.



GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
T_amb	25[degC]	298.15 K	Temperature of surrounding air
T_coffee	90[degC]	363.15 K	Coffee temperature
d_shell	0.5[mm]	5E-4 m	Steel-shell thickness
p_amb	1[atm]	1.0133E5 Pa	Ambient pressure

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

Leave the default geometric entity selection; subsequent materials that you add will override air as the material for the domains where it does not apply.

The properties of coffee are almost the same as for water.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Water, liquid**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Water, liquid (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat2)**.

2 Select Domain 2 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Nylon**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Nylon (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Nylon (mat3)**.

2 Select Domains 4 and 5 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Steel AISI 4340**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Steel AISI 4340 (mat4)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat4)**.

3 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Shell**.

Material 5 (mat5)

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 `Foam` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	`0.03`	W/(m·K)	Basic
Density	rho	60	kg/m³	Basic
Heat capacity at constant pressure	Cp	200	J/(kg·K)	Basic

HEAT TRANSFER IN SOLIDS (HT)

Assuming that the coffee temperature is uniform and depends on the time only, the domain can be defined as isothermal domain with the initial coffee temperature.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

2 In the **Settings** window for Heat Transfer in Solids, locate the **Physical Model** section.

3 Select the **Isothermal domain** check box.

Define the ambient temperature used in boundary conditions and intial values.

4 Locate the **Ambient Settings** section. In the T_{amb} text field, type `T_amb`.

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, choose **Ambient temperature (ht)** from the T list.

Isothermal Domain 1

1 On the **Physics** toolbar, click **Domains** and choose **Isothermal Domain**.

2 Select Domain 2 only.

For the **Isothermal Domain Interface** boundary condition use a heat flux condition that describes a good heat transmission from the coffee to the thin conductive shell boundary.

Isothermal Domain Interface 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Isothermal Domain Interface 1**.

2 In the **Settings** window for Isothermal Domain Interface, locate the **Isothermal Domain Interface** section.

3 From the **Interface type** list, choose **Convective heat flux**.

4 In the h text field, type `100`.

Initial Values 2

1 On the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 2 only.

3 In the **Settings** window for Initial Values, type `T_coffee` in the T text field.

The steel walls of the flask are represented by a special boundary condition for highly conductive layers:

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for Thin Layer, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Shell**.

4 Locate the **Thin Layer** section. From the **Layer type** list, choose **Thermally thin approximation**.

5 In the d_s text field, type `d_shell`.

To allow for cooling to the surrounding, add heat flux conditions that use appropriate heat transfer coefficients from a library.

Heat Flux 1

1 On 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 **Flask, Vertical Walls**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 From the **Heat transfer coefficient** list, choose **External natural convection**.

6 In the L text field, type `height`.

7 From the T_{ext} list, choose **Ambient temperature (ht)**.

Heat Flux 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 8 only.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 From the **Heat transfer coefficient** list, choose **External natural convection**.

6 From the list, choose **Horizontal plate, upside**.

7 In the L text field, type `radius`.

8 From the T_{ext} list, choose **Ambient temperature (ht)**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

3 From the **Time unit** list, choose **h**.

4 In the **Times** text field, type `range(0,0.1,10)`.

Constrict the initial time step to capture the strong gradient between the initial coffee temperature and the initial flask temperature.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. Select the **Initial step** check box.

5 In the associated text field, type `10[s]`.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature, 3D (ht)

A 3D temperature plot and an isothermal contour plot are produced by default. To display the temperatures in degrees Celsius, you can edit these existing plots:

Surface

1 In the **Model Builder** window, expand the **Temperature, 3D (ht)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 On the **Temperature, 3D (ht)** toolbar, click **Plot**.

Contour

1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Contour**.

2 In the **Settings** window for Contour, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 On the **Isothermal Contours (ht)** toolbar, click **Plot**.

Approach 2 - Modeling the External Flow

The second modeling approach is done within the same MPH-file. This way the results from both approaches can be compared directly. Add a second model as follows:

1 On the **Home** toolbar, click **Add Component** and choose **2D Axisymmetric**.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component** in the window toolbar.

HEAT TRANSFER 2 (HT2)

On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

5 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Heat Transfer in Solids (ht)** interface.

   This way, the new study solves for the coupled heat transfer and laminar flow interfaces only.

6 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Time Dependent

On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GEOMETRY 2

The flask's geometry is already present in **Component 1**. Import the geometry sequence from above as follows:

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

2 In the **Settings** window for Import, locate the **Import** section.

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Geometry 1**.

5 Click **Import**.

In this approach, you model the fluid flow explicitly, so you need to add a flow domain to the model.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.1[m]`.

4 In the **Height** text field, type `0.5[m]`.

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

Define the same selections as before, which you can use during the model setup and for comparing the results of this approach to the first one.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Shell` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 11–14, 16, 19, and 22–26 only.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for Explicit, type `Flask, Vertical Walls` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 17–19 and 26 only.

MATERIALS

You have added the materials in Model 1 already, you can now choose them from the **Recent Materials** folder in the **Add Material** window easily.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Recent Materials>Air**.

4 Click **Add to Component** in the window toolbar.

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Recent Materials>Water, liquid**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Water, liquid (mat7)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials** node, then click **Component 2 (comp2)>Materials>Water, liquid (mat7)**.

2 Select Domain 2 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Recent Materials>Nylon**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Nylon (mat8)

1 In the **Model Builder** window, under **Component 2 (comp2)>Materials** click **Nylon (mat8)**.

2 Select Domains 4 and 6 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Recent Materials>Steel AISI 4340**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Steel AISI 4340 (mat9)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 2 (comp2)>Materials** click **Steel AISI 4340 (mat9)**.

3 In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Shell**.

Material 10 (mat10)

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type `Foam` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	0.03	W/(m·K)	Basic
Density	rho	60	kg/m³	Basic
Heat capacity at constant pressure	Cp	200	J/(kg·K)	Basic

After setting up the physics interfaces, the warning for missing material, marked with red crosses in the materials node, will disappear.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Laminar Flow (spf)**.

2 Select Domain 5 only.

3 In the **Settings** window for Laminar Flow, locate the **Physical Model** section.

4 From the **Compressibility** list, choose **Weakly compressible flow**.

5 Select the **Include gravity** check box.

6 Locate the **Domain Selection** section. Click **Create Selection**.

7 Locate the **Physical Model** section. Find the **Reference values** subsection. In the T_{ref} text field, type `T_amb`.

8 In the **Create Selection** dialog box, type `Air` in the **Selection name** text field.

9 Click **OK**.

HEAT TRANSFER 2 (HT2)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer 2 (ht2)**.

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Heat Transfer 2 (ht2)**.

2 In the **Settings** window for Heat Transfer, locate the **Ambient Settings** section.

3 In the T_{amb} text field, type `T_amb`.

Fluid 1

The interface provides nodes for the solid and fluid domain by default and the remaining step is to assign the surrounding air domain to the **Fluid Properties** node. Because the density depends on the temperature and the pressure, choose the calculated pressure as input for this material property.

1 In the **Model Builder** window, under **Component 2 (comp2)>Heat Transfer 2 (ht2)** click **Fluid 1**.

2 In the **Settings** window for Fluid, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

To get a good initial guess for the solver, set the initial value for the temperature to the ambient temperature.

Initial Values 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Heat Transfer 2 (ht2)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, choose **Ambient temperature (ht2)** from the $T2$ list.

LAMINAR FLOW (SPF)

On the **Physics** toolbar, click **Heat Transfer 2 (ht2)** and choose **Laminar Flow (spf)**.

In the **Model Builder** window, under **Component 2 (comp2)** click **Laminar Flow (spf)**.

Open Boundary 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundaries 10 and 21 only.

HEAT TRANSFER 2 (HT2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Heat Transfer 2 (ht2)**.

2 In the **Settings** window for Heat Transfer, locate the **Physical Model** section.

3 Select the **Isothermal domain** check box.

Isothermal Domain 1

1 On the **Physics** toolbar, click **Domains** and choose **Isothermal Domain**.

2 Select Domain 2 only.

Isothermal Domain Interface 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Heat Transfer 2 (ht2)** click **Isothermal Domain Interface 1**.

2 In the **Settings** window for Isothermal Domain Interface, locate the **Isothermal Domain Interface** section.

3 From the **Interface type** list, choose **Convective heat flux**.

4 In the h text field, type `100`.

Initial Values 2

1 On the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 2 only.

3 In the **Settings** window for Initial Values, type `T_coffee` in the $T2$ text field.

Solid 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Heat Transfer 2 (ht2)** click **Solid 1**.

2 In the **Settings** window for Solid, locate the **Model inputs** section.

3 Click **Make All Model Inputs Editable** in the upper-right corner of the section. Locate the **Model Inputs** section. From the T list, choose **Temperature (ht2)**.

Thin Layer 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for Thin Layer, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Shell**.

4 Locate the **Thin Layer** section. From the **Layer type** list, choose **Thermally thin approximation**.

5 In the d_s text field, type `d_shell`.

Open Boundary 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundaries 10 and 21 only.

3 In the **Settings** window for Open Boundary, locate the **Open Boundary** section.

4 In the T_0 text field, type `T_amb`.

MESH 2

Use a finer mesh to get a good resolution of the flow field.

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

2 In the **Settings** window for Mesh, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

STUDY 2

Large differences of the initial values `T_coffee` and `T_amb` can cause numerical instabilities if time steps become too large. Forcing the solver to use small time steps in the beginning until the strong gradients are blurred out helps to overcome this problem. To do so, add two time dependent solver steps in one study. Set up the time intervals as follows:

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

3 From the **Time unit** list, choose **h**.

4 In the **Times** text field, type `range(0,0.1,10)[s] range(0.1,0.1,10)`.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for Time-Dependent Solver, locate the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Strict**.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature, 3D (ht2)

Change the default temperature plot to use degrees Celsius as quantity unit (as in Figure 3).

Surface

1 In the **Model Builder** window, expand the **Temperature, 3D (ht2)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Velocity (spf) 1

The velocity field is automatically shown in its dedicated plot group (see Figure 6).

Create a 2D plot of the velocity streamlines.

2D Plot Group 8

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Velocity, Streamlines` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 From the **Time (h)** list, choose **10**.

Streamline 1

1 Right-click **Velocity, Streamlines** and choose **Streamline**.

2 In the **Settings** window for Streamline, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 2>Laminar Flow>Velocity and pressure>u,w - Velocity field**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.

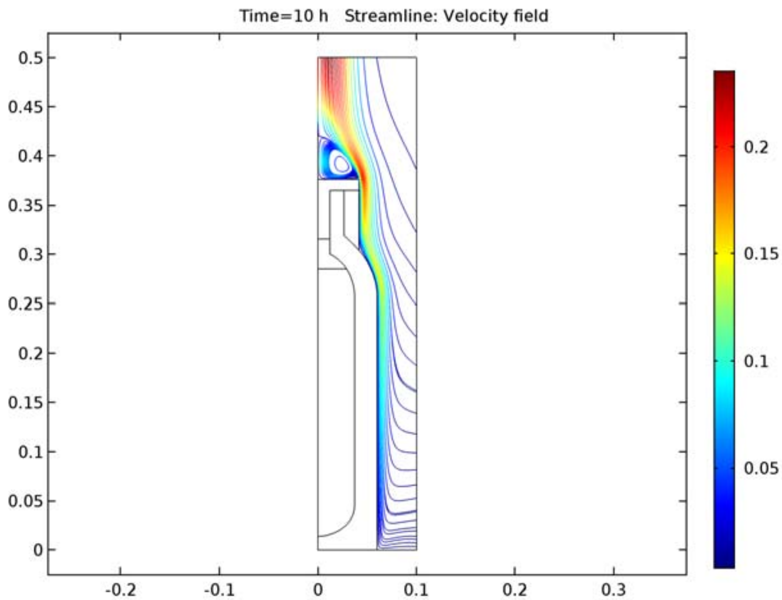
Color Expression 1

1 Right-click **Results>Velocity, Streamlines>Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 2>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude**.

3 On the **Velocity, Streamlines** toolbar, click **Plot**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.



To compare both approaches, evaluate the coffee temperature over time and use a 1D Plot to visualize the results. The solutions are stored under the **Data Sets** node and for each plot you can choose which data set should be used.

1D Plot Group 9

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, type `Coffee Temperature vs Time` in the **Label** text field.

Global 1

1. On the **Coffee Temperature vs Time** toolbar, click **Global**.
2. In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>Temperature>ht.id1.T - Isothermal domain temperature**.
3. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`ht.id1.T`	`degC`	`Isothermal domain temperature - flow not computed`

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 From the **Positioning** list, choose **In data points**.

Coffee Temperature vs Time

In the **Model Builder** window, under **Results** click **Coffee Temperature vs Time**.

Global 2

1 On the **Coffee Temperature vs Time** toolbar, click **Global**.

2 In the **Settings** window for Global, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 2>Heat Transfer 2>Temperature>ht2.id1.T - Isothermal domain temperature**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`ht2.id1.T`	`degC`	`Isothermal domain temperature - flow computed`

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 From the **Positioning** list, choose **In data points**.

9 On the **Coffee Temperature vs Time** toolbar, click **Plot**.

To compare the heat transfer coefficients for the two modeling approaches (Figure 4), do the following:

1D Plot Group 10

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Heat Transfer Coefficient` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **Last**.

4 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 On the **Heat Transfer Coefficient** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Selection** section.

3 From the **Selection** list, choose **Flask, Vertical Walls**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `ht.hf1.h`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `z`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Heat transfer coefficient, no flow`

Heat Transfer Coefficient

In the **Model Builder** window, under **Results** click **Heat Transfer Coefficient**.

Line Graph 2

1 On the **Heat Transfer Coefficient** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 From the **Time selection** list, choose **Last**.

5 Locate the **Selection** section. From the **Selection** list, choose **Flask, Vertical Walls**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(ht2.ntflux)/(T2-T_amb)`.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `z`.

9 Locate the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
`Calculated heat transfer coefficient, with flow`

Heat Transfer Coefficient

1 In the **Model Builder** window, under **Results** click **Heat Transfer Coefficient**.

2 On the **Heat Transfer Coefficient** toolbar, click **Plot**.

Geometry Modeling Instructions

If you wish to create the geometry yourself, follow these steps.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
height	380[mm]	0.38 m	Flask height
radius	40[mm]	0.04 m	Bottleneck radius

GEOMETRY 1

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **r** to `1.5*radius`.

5 Find the **Added segments** subsection. Click **Add Linear**.

6 Find the **Control points** subsection. In row **2**, set **z** to `0.68*height`.

7 Find the **Added segments** subsection. Click **Add Quadratic**.

8 Find the **Control points** subsection. In row **2**, set **z** to `0.751*height`.

9 In row **3**, set **r** to `1.04*radius`.

10 In row **3**, set **z** to `0.80*height`.

11 Find the **Added segments** subsection. Click **Add Quadratic**.

12 Find the **Control points** subsection. In row **2**, set **r** to `0.88*radius`.

13 In row **2**, set **z** to `0.82*height`.

14 In row **3**, set **r** to `0.66*radius`.

15 In row **3**, set **z** to `0.84*height`.

16 Find the **Added segments** subsection. Click **Add Linear**.

17 Find the **Control points** subsection. In row **2**, set **z** to `0.96*height`.

18 Find the **Added segments** subsection. Click **Add Linear**.

19 Find the **Control points** subsection. In row **2**, set **r** to `0.3*radius`.

20 Find the **Added segments** subsection. Click **Add Linear**.

21 Find the **Control points** subsection. In row **2**, set **z** to `0.83*height`.

22 Find the **Added segments** subsection. Click **Add Linear**.

23 Find the **Control points** subsection. In row **2**, set **z** to `0.79*height`.

24 Find the **Added segments** subsection. Click **Add Quadratic**.

25 Find the **Control points** subsection. In row **2**, set **r** to `0.56*radius`.

26 In row **2**, set **z** to `0.78*height`.

27 In row **3**, set **r** to `0.73*radius`.

28 In row **3**, set **z** to `0.75*height`.

29 Find the **Added segments** subsection. Click **Add Quadratic**.

30 Find the **Control points** subsection. In row **2**, set **r** to `0.93*radius`.

31 In row **2**, set **z** to `0.72*height`.

32 In row **3**, set **r** to `0.93*radius`.

33 In row **3**, set **z** to `0.68*height`.

34 Find the **Added segments** subsection. Click **Add Linear**.

35 Find the **Control points** subsection. In row **2**, set **z** to `0.12*height`.

36 Find the **Added segments** subsection. Click **Add Quadratic**.

37 Find the **Control points** subsection. In row **2**, set **z** to `0.036*height`.

38 In row **3**, set **r** to `0*radius`.

39 In row **3**, set **z** to `0.036*height`.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `1.04*radius`.

4 In the **Height** text field, type `0.16*height`.

5 Locate the **Position** section. In the **z** text field, type `0.83*height`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.74*radius`.

4 In the **Height** text field, type `0.13*height`.

5 Locate the **Position** section. In the **r** text field, type `0.3*radius`.

6 In the **z** text field, type `0.83*height`.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

3 In the **Settings** window for Difference, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **r2** only.

Bézier Polygon 2 (b2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Quadratic**.

4 Find the **Control points** subsection. In row **1**, set **r** to `1.04*radius`.

5 In row **1**, set **z** to `0.80*height`.

6 In row **2**, set **r** to `0.88*radius`.

7 In row **2**, set **z** to `0.82*height`.

8 In row **3**, set **r** to `0.66*radius`.

9 In row **3**, set **z** to `0.84*height`.

10 Find the **Added segments** subsection. Click **Add Linear**.

11 Find the **Control points** subsection. In row **2**, set **z** to `0.96*height`.

12 Find the **Added segments** subsection. Click **Add Linear**.

13 Find the **Control points** subsection. In row **2**, set **r** to `1.04*radius`.

14 Find the **Added segments** subsection. Click **Add Linear**.

15 Find the **Control points** subsection. In row **2**, set **z** to `0.80*height`.

Bézier Polygon 3 (b3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **r** to `0.3*radius`.

5 In row **1**, set **z** to `0.83*height`.

6 In row **2**, set **r** to `0.3*radius`.

7 In row **2**, set **z** to `0.79*height`.

8 Find the **Added segments** subsection. Click **Add Quadratic**.

9 Find the **Control points** subsection. In row **2**, set **r** to `0.56*radius`.

10 In row **2**, set **z** to `0.78*height`.

11 In row **3**, set **r** to `0.73*radius`.

12 In row **3**, set **z** to `0.75*height`.

13 Find the **Added segments** subsection. Click **Add Linear**.

14 Find the **Control points** subsection. In row **2**, set **r** to `0*radius`.

15 Find the **Added segments** subsection. Click **Add Linear**.

16 Find the **Control points** subsection. In row **2**, set **z** to `0.83*height`.

Bézier Polygon 4 (b4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **z** to `0.75*height`.

5 In row **2**, set **z** to `0.036*height`.

Convert to Solid 1 (csol1)

On the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

Form Union (fin)

Click **Build All**.

Created in COMSOL Multiphysics 5.2a

View Factor Computation

The computation of view factors is central when performing heat transfer simulations using the radiosity method to account for surface-to-surface radiation. This benchmark demonstrates how to compute geometrical view factors for two concentric spheres that emit and receive radiation from each other. It compares simulation results to exact analytical values.

Introduction

The surface-to-surface radiation method in the Heat Transfer module relies on the radiosity method. When surface-to-surface radiation is activated, operators are available to compute the view factors between diffuse surfaces that irradiate each other. For some standard configurations one can determine the view factors analytically, but in engineering applications this is rarely possible. A benchmark example that compares exact analytical values with simulation results shows the accuracy of the numerical method. Here, two concentric spheres are used, for which the analytical view factors are known.

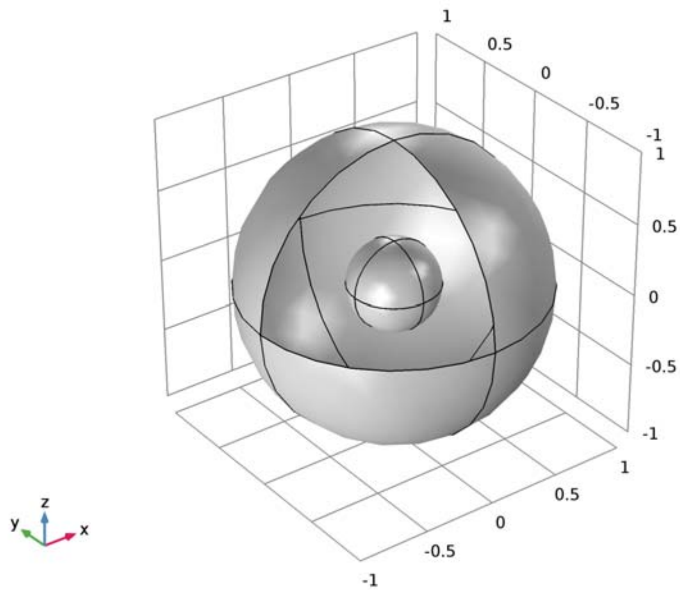


Figure 1: Model geometry.

Model Definition

The model consists of two concentric radiating spheres (Figure 1) acting as perfect emitters (surface emissivities both equal to 1). The radiation direction of each sphere is such that they irradiate each other.

ANALYTICAL VIEW FACTORS

A detailed introduction on view factors is given in (Ref. 1). An arbitrary view factor $F_{1 \to 2}$ is defined as the proportion of radiation leaving surface S_1 and intercepting surface S_2, that is

$$F_{1 \to 2} = \frac{\text{Radiation leaving } S_1 \text{ and intercepting } S_2}{\text{Total radiation leaving } S_1}$$

If the radiosity is the same on each surface, it only depends on the geometrical configuration and thus can be calculated from geometrical properties. For two concentric spheres, labeled ext for the outer sphere and int for the inner sphere, the view factors are:

$$F_{\mathrm{ext} \to \mathrm{ext}} = 1 - \left(\frac{R_{\mathrm{int}}}{R_{\mathrm{ext}}}\right)^2$$

$$F_{\mathrm{ext} \to \mathrm{int}} = \left(\frac{R_{\mathrm{int}}}{R_{\mathrm{ext}}}\right)^2$$

$$F_{\mathrm{int} \to \mathrm{int}} = 0$$

$$F_{\mathrm{int} \to \mathrm{ext}} = 1$$

Refer to Table 1 for numerical values of these quantities.

VIEW FACTOR COMPUTATION IN COMSOL MULTIPHYSICS

The view factor computation is performed at each evaluation point on the boundaries. A boundary has an *up* and a *down* side and both can be exposed to radiation. In this model the outer sphere radiates inwards, which is the downward direction and the inner sphere radiates outwards which is the upward direction. Two operators are available for evaluation of the mutual surface irradiation. These are:

- `radopu(`*expression_upside*`,`*expression_downside*`)`
- `radopd(`*expression_upside*`,`*expression_downside*`)`

Both are available on all boundaries where surface-to-surface features are active. The `radopu(,)` and `radopd(,)` operators perform the computation on the upside and downside of the boundary where they are evaluated. The *expression_upside* and *expression_downside* arguments are the radiosity expression on the upside and downside of the boundaries that irradiate the boundary where the operators are evaluated.

To compute the geometrical view factor, e.g. $F_{\mathrm{ext} \to \mathrm{int}}$, in COMSOL Multiphysics, the following integration needs to be defined:

$$F_{\mathrm{ext} \to \mathrm{int}} = \frac{\int_{S_{\mathrm{int}}} \mathrm{radopu}(0, \mathrm{ext})ds}{A_{\mathrm{ext}}}$$

where S_{int} and S_{ext} denote interior and exterior surfaces, A_{int} and A_{ext} are corresponding surface areas.

The integration at the numerator is defined over the inner sphere, which radiates only on the upside of its surface. Hence, `radopd(,)` evaluates to zero and the integrand reduces to `radopu(0,`*expression_upside*`)`. Moreover since the outer sphere radiates only on the downside of its surface the *expression_downside* argument should be `ext`.

Results and Discussion

The view factors computed with COMSOL Multiphysics are listed in Table 1 together with analytical values and absolute errors.

TABLE 1: VIEW FACTOR COMPUTATION

VIEW FACTOR	ANALYTICAL VALUE	COMPUTED VALUE	ERROR
$F_{\mathrm{int} \to \mathrm{int}}$	0	0	0
$F_{\mathrm{int} \to \mathrm{ext}}$	1	1	$3.25 \cdot 10^{-5}$
$F_{\mathrm{ext} \to \mathrm{ext}}$	0.91	0.91	$2.93 \cdot 10^{-6}$
$F_{\mathrm{ext} \to \mathrm{int}}$	0.09	0.09	$5.35 \cdot 10^{-10}$

In this table, analytical and computed values are slightly different but rounded to hundredths. The very small errors are listed in the last column and reflect accurate results of the view factors provided by the simulation.

Reference

1. Michael F. Modest, *Radiative Heat Transfer*, 3rd ed., Academic Press, 2013

Application Library path: `Heat_Transfer_Module/Verification_Examples/view_factor`

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**.

You are only interested in the view factors, hence Surface-to-Surface Radiation is the only needed interface.

2 In the **Select Physics** tree, select **Heat Transfer>Radiation>Surface-to-Surface Radiation (rad)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Define the radii of the spheres and the analytical values for the view factors as parameters.

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
r_int	0.3[m]	0.3 m	Radius, interior sphere
r_ext	1[m]	1 m	Radius, exterior sphere

Name	Expression	Value	Description
F_ext_ext	1-(r_int/r_ext)^2	0.91	Analytical view factor from exterior sphere to exterior sphere
F_ext_int	(r_int/r_ext)^2	0.09	Analytical view factor from exterior sphere to interior sphere
F_int_int	0	0	Analytical view factor from interior sphere to interior sphere
F_int_ext	1	1	Analytical view factor from interior sphere to exterior sphere

GEOMETRY 1

No domain information are required, define the geometry as surface objects.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for Sphere, locate the **Object Type** section.

3 From the **Type** list, choose **Surface**.

4 Locate the **Size** section. In the **Radius** text field, type `r_int`.

Create a selection of this sphere to easily access its entities throughout the modeling process.

5 Locate the **Selections of Resulting Entities** section. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Inner sphere` in the **Name** text field.

7 Click **OK**.

Sphere 2 (sph2)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for Sphere, locate the **Object Type** section.

3 From the **Type** list, choose **Surface**.

4 Locate the **Size** section. In the **Radius** text field, type `r_ext`.

5 Locate the **Selections of Resulting Entities** section. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Outer sphere` in the **Name** text field.

7 Click **OK**.

8 In the **Settings** window for Sphere, click **Build All Objects**.

9 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Create a new view which hides one of the front boundaries so that you can look inside the outer sphere.

DEFINITIONS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

Hide for Geometry 1

On the **View 2** toolbar, click **Hide for Geometry**.

View 2

1 In the **Settings** window for Hide for Geometry, locate the **Selection** section.

2 From the **Geometric entity level** list, choose **Boundary**.

3 On the object **sph2**, select Boundary 2 only.

GEOMETRY 1

Add a material to the boundaries to define their emissivities.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, type `Blackbody` in the **Label** text field.

2 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Surface Emissivity**.

3 Click **Add to Material**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Surface emissivity	epsilon_rad	1	1	Basic

SURFACE-TO-SURFACE RADIATION (RAD)

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 In the **Settings** window for Diffuse Surface, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inner sphere**.

4 Locate the **Radiation Settings** section. From the **Radiation direction** list, choose **Positive normal direction**.

Diffuse Surface 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 In the **Settings** window for Diffuse Surface, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outer sphere**.

4 Locate the **Radiation Settings** section. From the **Radiation direction** list, choose **Negative normal direction**.

Define new variables and integration operators to evaluate the view factors directly after running the study. First, define variables that are used to identify the surfaces.

DEFINITIONS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

Variables 1

1 In the **Settings** window for Variables, type `Identifiers, Inner Sphere` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

3 From the **Selection** list, choose **Inner sphere**.

4 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
ext	0		Exterior surface indicator
int	1		Interior surface indicator

Variables 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for Variables, type `Identifiers, Outer Sphere` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Outer sphere**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
ext	1		Exterior surface indicator
int	0		Interior surface indicator

Now, define integration coupling operators for both spheres.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, type `Integration, Inner Sphere` in the **Label** text field.

3 In the **Operator name** text field, type `intop_int`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Inner sphere**.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for Integration, type `Integration, Outer Sphere` in the **Label** text field.

3 In the **Operator name** text field, type `intop_ext`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Outer sphere**.

Adjust the mesh size for both spheres manually to get approximately the same number of elements on each sphere. This is an efficient mesh for view factor computation.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

Free Triangular 1

1 In the **Settings** window for Free Triangular, locate the **Boundary Selection** section.

2 From the **Selection** list, choose **All boundaries**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.

2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Inner sphere**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `r_int/5`.

Free Triangular 1

Right-click **Free Triangular 1** and choose **Size**.

Size 2

1 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

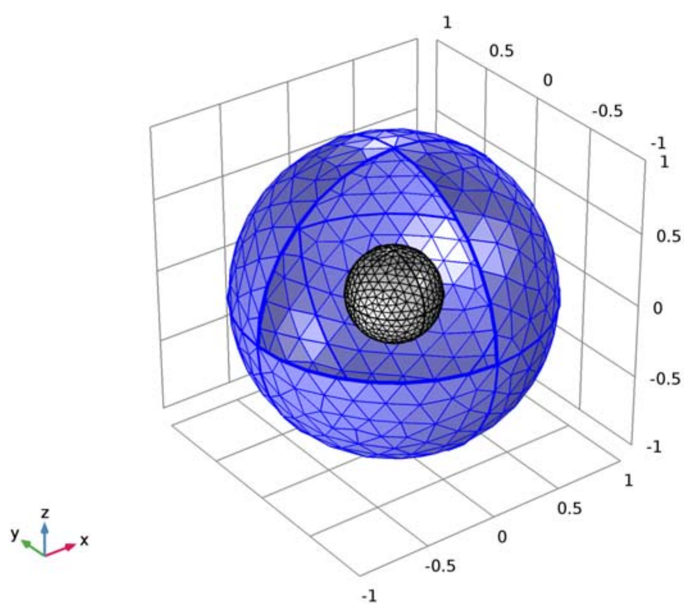
2 From the **Selection** list, choose **Outer sphere**.

3 Locate the **Element Size** section. Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `r_ext/5`.

6 Click **Build All**.



The view factors are computed automatically, before the actual study runs. To evaluate the geometrical view factors, it is sufficient to obtain the initial values.

STUDY 1

1 In the **Settings** window for Study, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

Study 1

On the **Study** toolbar, click **Get Initial Value**.

RESULTS

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, type `View Factors` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`intop_int(comp1.rad.radopu(int,0))/intop_int(1)`	1	`Interior to interior view factor`
`intop_ext(comp1.rad.radopd(int,0))/intop_int(1)`	1	`Interior to exterior view factor`
`intop_ext(comp1.rad.radopd(0,ext))/intop_ext(1)`	1	`Exterior to exterior view factor`
`intop_int(comp1.rad.radopu(0,ext))/intop_ext(1)`	1	`Exterior to interior view factor`

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, type `Absolute Error, View Factors` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`abs(intop_int(comp1.rad.radopu(int,0))/intop_int(1)-F_int_int)`	1	`Absolute error, interior to interior view factor`
`abs(intop_ext(comp1.rad.radopd(int,0))/intop_int(1)-F_int_ext)`	1	`Absolute error, interior to exterior view factor`

Expression	Unit	Description
abs(intop_ext(comp1.rad.radopd(0,ext))/intop_ext(1)-F_ext_ext)	1	Absolute error, exterior to exterior view factor
abs(intop_int(comp1.rad.radopu(0,ext))/intop_ext(1)-F_ext_int)	1	Absolute error, exterior to interior view factor

4 On the **Results** toolbar, click **Evaluate All**.

TABLE

1 Go to the **Table** window.

The results are displayed in the **Table** window.

Created in COMSOL Multiphysics 5.2a

Glazing Influence on Thermal Performances of a Window

Introduction

During the design of a building, environmental issues have gained considerable influence in the entire project. One of the first concerns is to improve thermal performances. In this process, simulation software provide key tools for modeling thermal losses and performances in the building

The international standard ISO 10077-2:2012 (Ref. 1) deals with thermal performances of windows, doors, and shutters. It provides computed values of the thermal characteristics of frame profiles in order to validate a simulation software.

COMSOL Multiphysics successfully passes the entire benchmark. This document describes two test cases of ISO 10077-2:2012 related to the glazing influence on thermal performances of a window. Other test cases from this standard are available in the following applications:

- Thermal Performances of Windows
- Thermal Performances of Roller Shutters

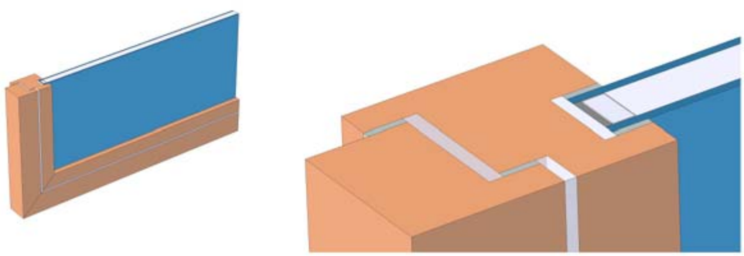


Figure 1: Geometry of the window and cross-section view.

Model Definition

On each test case, a window section separates a hot internal side from a cold external side. The window frame is the same but in the first application, an insulation panel replaces the traditional glazing. This traditional glazing is tackled in the second application. After solving a model, two quantities are calculated and compared to the normative values:

- The thermal conductance between internal and external sides
- The thermal transmittance of the frame is calculated

AIR CAVITIES

A window frame contains many cavities. The purpose is to ensure thermal insulation. According to the standard, cavities are modeled in different ways depending on their shapes.

The heat flow rate in cavities is represented by an equivalent thermal conductivity k_{eq}, which includes the heat flow by conduction, convection, and radiation. It also depends on the geometry of the cavity and on the adjacent materials. The definition of k_{eq} is detailed in the next paragraphs.

Cavities are divided into three types:

- *unventilated cavities*, completely closed or connected either to the exterior or to the interior by a slit with a width not exceeding 2 mm;
- *slightly ventilated cavities*, connected either to the exterior or to the interior by a slit greater than 2 mm but not exceeding 10 mm;
- *well-ventilated cavities*, corresponding to a configuration not covered by one of the two preceding types, it is assumed that the whole surface is exposed to the environment so that boundary conditions are applied to (see the Boundary conditions section below for more information).

Unventilated Rectangular Cavity

For an unventilated rectangular cavity, the equivalent thermal conductivity is defined by:

$$k_{\mathrm{eq}} = \frac{d}{R}$$

where d is the cavity dimension in the heat flow rate direction, and R is the cavity thermal resistance given by:

$$R = \frac{1}{h_{\mathrm{a}} + h_{\mathrm{r}}}$$

Here, h_{a} is the convective heat transfer coefficient, and h_{r} is the radiative heat transfer coefficient. These coefficients are defined by:

$$h_{\mathrm{a}} = \begin{cases} \dfrac{C_1}{d} & \text{if } b \le 5 \text{ mm} \\ \max\left(\dfrac{C_1}{d}, C_2(\Delta T/(1\mathrm{K}))^{1/3}\right) & \text{otherwise} \end{cases}$$

$$h_r = 4\sigma T_m^3 EF$$

where:

- $C_1 = 0.025\ \mathrm{W/(m{\cdot}K)}$
- $C_2 = 0.73\ \mathrm{W/(m^2{\cdot}K)}$
- ΔT is the maximum surface temperature difference in the cavity
- $\sigma = 5.67{\cdot}10^{-8}\ \mathrm{W/(m^2{\cdot}K^4)}$ is the Stefan-Boltzmann constant
- T_m is the average temperature on the boundaries of the cavity
- E is the intersurface emittance, defined by:

$$E = \frac{1}{\dfrac{1}{\varepsilon_1} + \dfrac{1}{\varepsilon_2} - 1}$$

- ε_1 and ε_2 are the surface emissivities (both are equal to 0.90 in this model)
- F is the view factor of the rectangular section, defined by:

$$F = \frac{1}{2}\left(1 - \frac{d}{b} + \sqrt{1 + \left(\frac{d}{b}\right)^2}\right)$$

- d is the cavity dimension in the heat flow rate direction
- b is the cavity dimension perpendicular to the heat flow rate direction

Slightly Ventilated Rectangular Cavities

For a slightly ventilated cavity, the equivalent thermal conductivity is twice that of an unventilated cavity of the same size.

BOUNDARY CONDITIONS

The heat flux conditions for internal and external sides are given by the Newton's law of cooling:

$$-\mathbf{n} \cdot (-k\nabla T) = h(T_{ext} - T)$$

where T_{ext} is the exterior temperature ($T_{ext} = T_i = 20$ °C for the internal side and $T_{ext} = T_e = 0$ °C for the external side). The standard defines thermal surface resistance, R_s, which is related to the heat transfer coefficient, h, by:

$$h = \frac{1}{R_s}$$

Internal and external thermal surface resistances are not equal. Moreover, on boundaries linked to the internal side, an increased thermal resistance is used for the edges. Figure 2 explains how to determine boundaries where it should be applied.

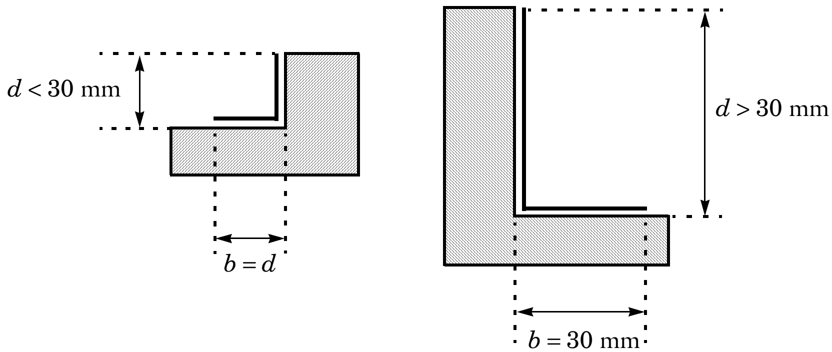


Figure 2: Protected boundaries.

If d is greater than 30 mm, b is set to 30 mm. Otherwise, $b = d$ is chosen. Furthermore, two boundaries are considered as adiabatic: the boundary in contact with the wall and the end of the insulation panel or glazing.

DESCRIPTION OF THE TWO APPLICATIONS

Figure 3 and Figure 4 depict the geometry of each application but only a part of the insulation panel or glazing is represented. Unventilated cavities are red-numbered while slightly ventilated cavities are green-numbered. Boundaries with an increased thermal resistance are represented with bold black lines. Adiabatic boundaries in contact with the wall are represented with a striped rectangle.

Application 1: Wood Frame with an Insulation Panel

The first application studies the heat conduction in the wood frame section with an insulation panel. The frame section is made of two wood blocks with a low thermal conductivity of 0.13 W/(m·K). In order to make the contact between these two blocks and to waterproof the window, two ethylene propylene diene monomer (EPDM) gaskets are used. Two other EPDM blocks are arranged on both sides of the insulation panel. The insulation panel has a very low thermal conductivity of 0.035 W/(m·K).

Two cavities are completely closed and are considered as *unventilated*. The third one is considered as *slightly ventilated*.

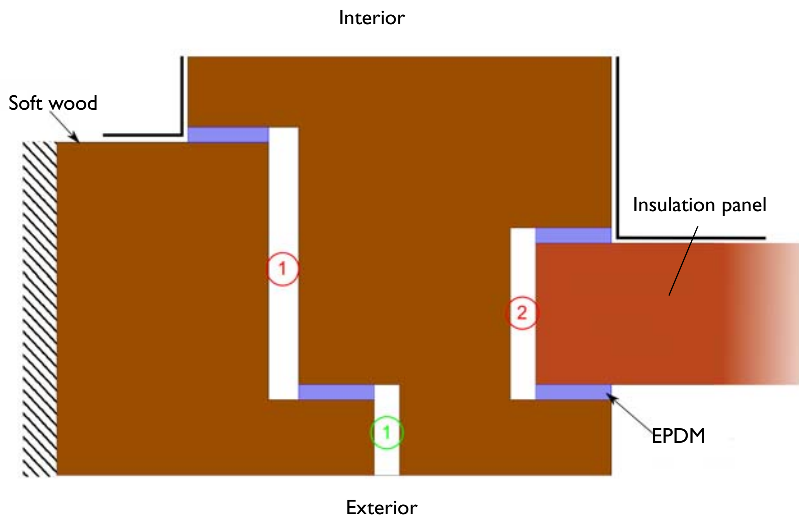


Figure 3: Geometry of the wood frame with an insulation panel.

Application 2: Wood Frame with a Traditional Glazing

The glazing is made of two glass panels with a thermal conductivity of 1.00 W/(m·K). On the frame side of the glazing, a structure made of aluminum, polysulfide, and silica gel is used to block the glass blocks. Their thermal conductivities are 160 W/(m·K), 0.40 W/(m·K), and 0.13 W/(m·K), respectively. The space between the glass panels is filled with a gas whose thermal conductivity is 0.034 W/(m·K) (so this space is not considered as a traditional air cavity).

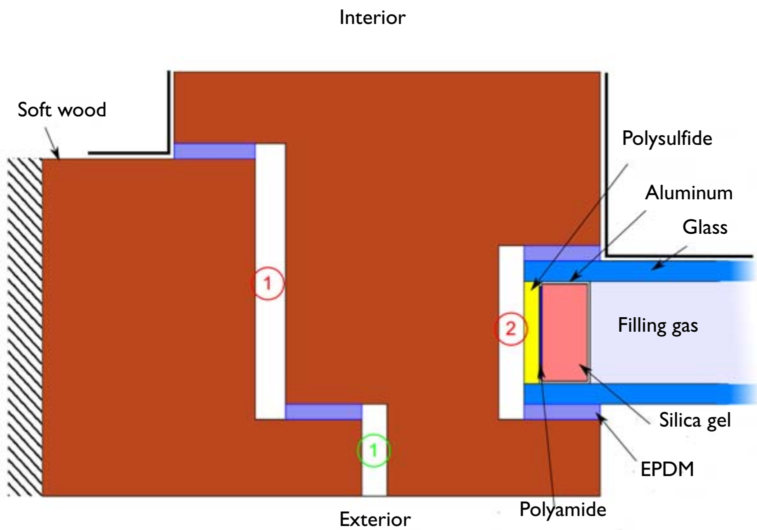


Figure 4: Geometry of the wood frame with a glazing.

Results and Discussion

TEMPERATURE PROFILES

Figure 5 and Figure 6 show the temperature profiles for each application.

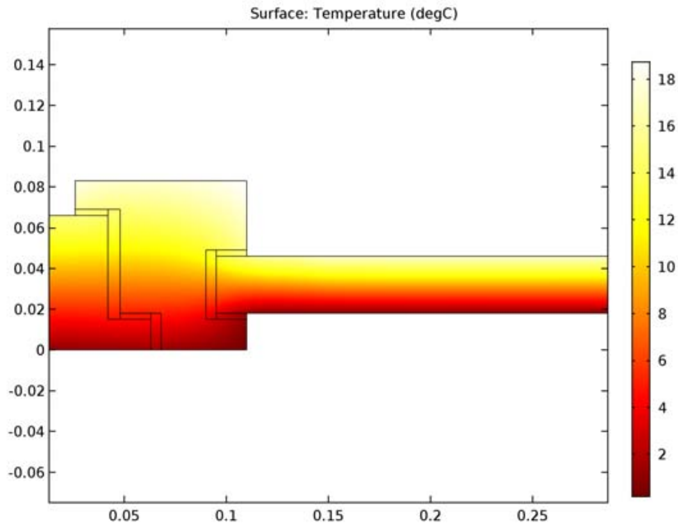


Figure 5: Temperature profile with the insulation panel.

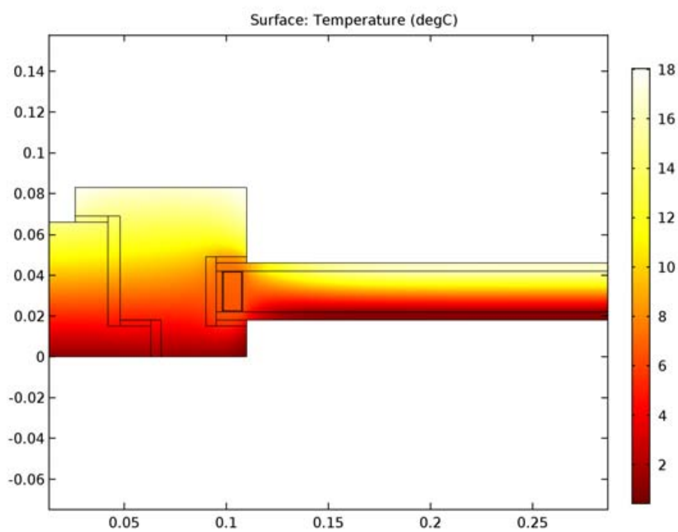


Figure 6: Temperature distribution with glazing.

QUANTITIES OF INTEREST

The quantities of interest are the following:

- The thermal conductance of the entire section L^{2D} given by:

$$L^{2D} = \frac{\phi}{T_e - T_i}$$

  where ϕ is the heat flow rate through the window (in W/m), $T_e = 0$ °C is the external temperature and $T_i = 20$ °C is the internal temperature.
- The thermal transmittance of the frame U_f defined by:

$$U_f = \frac{L^{2D} - U_p b_p}{b_f}$$

  where b_p is the visible width of the panel expressed in meters, b_f is the projected width of the frame section expressed in meters and U_p is the thermal transmittance of the central area of the panel expressed in W/(m²·K).
- The linear thermal transmittance of the frame Ψ defined by:

$$\Psi = L^{2D} - U_f b_f - U_g b_g$$

  where b_g is the visible width of the glazing expressed in meters, U_g is the thermal transmittance of the central area of the glazing expressed in W/(m²·K).

Here, Ψ describes the additional heat flow caused by the interaction of the frame and the glass edge, including the effect of the spacer. The thermal transmittance U_g is provided, equal to 1.3 W/(m²·K).

Table 1 and Table 2 compare the numerical results of COMSOL Multiphysics with the expected values provided by ISO 10077-2:2012.

TABLE 1: COMPARISON BETWEEN EXPECTED AND COMPUTED VALUES OF QUANTITIES IN APPLICATION 1

QUANTITY	EXPECTED VALUE	COMPUTED VALUE	RELATIVE ERROR
L^{2D} (W/(m·K))	0.346	0.345	0.29%
U_f (W/(m²·K))	1.36	1.38	1.47%

TABLE 2: COMPARISON BETWEEN EXPECTED AND COMPUTED VALUES OF QUANTITIES IN APPLICATION 2

QUANTITY	EXPECTED VALUE	COMPUTED VALUE	RELATIVE ERROR
L^{2D} (W/(m·K))	0.481	0.483	0.42%
Ψ (W/(m²·K))	0.084	0.085	1.19%

The maximum permissible differences to pass this test case are 3% for the thermal conductance and 5% for the (linear) thermal transmittance. The measured values are completely coherent and meet the validation criteria.

Reference

1. European Committee for Standardization, *ISO 10077-2:2012, Thermal performance of windows, doors and shutters – Calculation of thermal transmittance – Part 2: Numerical method for frames*, 2012.

Application Library path: `Heat_Transfer_Module/Buildings_and_Constructions/window_and_glazing_thermal_performances`

Modeling Instructions

ROOT

Start by opening the following prepared file. It already contains global definitions, geometries, local variables, selections, operators and material properties.

1. From the **File** menu, choose **Open**.
2. Browse to the application's Application Libraries folder and double-click the file `window_and_glazing_thermal_performances_preset.mph`.

Window with Insulation Panel

WINDOW WITH INSULATION PANEL (COMP1)

In the **Model Builder** window, expand the **Window with Insulation Panel (comp1)** node.

DEFINITIONS

Variables 1

Define the thermal conductance of the section for the postprocessing part as follows.

1. In the **Model Builder** window, expand the **Window with Insulation Panel (comp1)> Definitions** node, then click **Variables 1**.
2. In the **Settings** window for Variables, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
L2D	int_internal(ht.ntflux /(Te-Ti))	W/(m·K)	Thermal conductance of the frame

Note that the heat flow rate through the internal and through the external boundaries are equal (in absolute value) because other boundaries are considered adiabatic.

4 In the **Model Builder** window, collapse the **Definitions** node.

DEFINITIONS

In the **Model Builder** window, collapse the **Window with Insulation Panel (comp1)>Definitions** node.

HEAT TRANSFER IN SOLIDS (HT)

Fluid 1

1 On the **Physics** toolbar, click **Domains** and choose **Fluid**.

2 Select Domains 4, 6, and 7 only.

Heat Flux 1

1 On 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 **Exterior Side**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `1/Rse`.

6 In the T_{ext} text field, type `Te`.

Heat Flux 2

1 On 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 **Interior Side (Flat Area)**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `1/Rsi_n`.

6 In the T_{ext} text field, type `Ti`.

Heat Flux 3

1 On 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 **Interior Side (Corner Area)**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `1/Rsi_p`.

6 In the T_{ext} text field, type `Ti`.

Open Boundary 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundary 19 only.

3 In the **Settings** window for Open Boundary, locate the **Open Boundary** section.

4 In the T_0 text field, type `Te`.

5 In the **Model Builder** window, collapse the **Heat Transfer in Solids (ht)** node.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, collapse the **Window with Insulation Panel (comp1)>Heat Transfer in Solids (ht)** node.

STUDY 1

The heat flow rate through the interior (or exterior) side of the section needs to be determined to calculate the thermal conductance of the section. In order to have enough precision in this value, the default relative tolerance of the solver has already been modified to 10^{-6}. To access to this value, expand the **Solver 1** node and click on the **Stationary Solver 1** node. In the **Stationary Solver** settings window, locate the **General** section.

1 On the **Home** toolbar, click **Compute**.

RESULTS

Derived Values

A **Global Evaluation** node is added in order to calculate the thermal conductance of the section and the thermal transmittance of the frame.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, type `Thermal Properties, Window with Insulation Panel` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
L2D	W/(m*K)	Thermal Conductance of the Section (L2D)
(L2D-Up*bp)/bf	W/(m^2*K)	Thermal Transmittance of the Frame (Uf)

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The results should be close to the expected values in Table 1.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Results>Temperature (ht)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 On the **Temperature (ht)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The current plot group shows the temperature distribution; compare with Figure 5.

The same simulation method is applied to the other benchmark. The instructions below describe the steps to achieve the calculations.

WINDOW WITH INSULATION PANEL (COMP1)

In the **Model Builder** window, collapse the **Window with Insulation Panel (comp1)** node.

Window with Glazing

WINDOW WITH GLAZING (COMP2)

In the **Model Builder** window, expand the **Window with Glazing (comp2)** node.

DEFINITIONS

Variables 2

1 In the **Model Builder** window, expand the **Window with Glazing (comp2)>Definitions** node, then click **Variables 2**.

2 In the **Settings** window for Variables, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`L2D`	`int_internal(ht2.ntflux/(Te-Ti))`	W/(m·K)	`Thermal conductance of the frame`

4 In the **Model Builder** window, collapse the **Definitions** node.

DEFINITIONS

In the **Model Builder** window, collapse the **Window with Glazing (comp2)>Definitions** node.

HEAT TRANSFER IN SOLIDS 2 (HT2)

On the **Physics** toolbar, click **Heat Transfer in Solids (ht)** and choose **Heat Transfer in Solids 2 (ht2)**.

In the **Model Builder** window, under **Window with Glazing (comp2)** click **Heat Transfer in Solids 2 (ht2)**.

Fluid 1

1 On the **Physics** toolbar, click **Domains** and choose **Fluid**.

2 Select Domains 4, 6, 7, and 16 only.

Heat Flux 1

1 On 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 **Exterior Side**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `1/Rse`.

6 In the T_{ext} text field, type `Te`.

Heat Flux 2

1 On 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 **Interior Side (Flat Area)**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `1/Rsi_n`.

6 In the T_{ext} text field, type `Ti`.

Heat Flux 3

1 On 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 **Interior Side (Corner Area)**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `1/Rsi_p`.

6 In the T_{ext} text field, type `Ti`.

Open Boundary 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundary 19 only.

3 In the **Settings** window for Open Boundary, locate the **Open Boundary** section.

4 In the T_0 text field, type `Te`.

5 In the **Model Builder** window, collapse the **Heat Transfer in Solids 2 (ht2)** node.

HEAT TRANSFER IN SOLIDS 2 (HT2)

In the **Model Builder** window, collapse the **Window with Glazing (comp2)>Heat Transfer in Solids 2 (ht2)** node.

STUDY 2

On the **Home** toolbar, click **Compute**.

RESULTS

Derived Values

A **Global Evaluation** node is added in order to calculate the thermal conductance of the section and the linear thermal transmittance of the frame.

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, type `Thermal Properties, Window with Glazing` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (4) (sol2)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
L2D	W/(m*K)	Thermal Conductance of the Section (L2D)
L2D-Uf*bf-Ug*bg	W/(m*K)	Linear Thermal Transmittance of the Frame (psi)

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The results should be close to the expected values in Table 2.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Results>Temperature (ht2)** node, then click **Surface**.

2 In the **Settings** window for Surface, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 On the **Temperature (ht2)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The current plot group shows the temperature distribution; compare with Figure 6.