

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](#)).

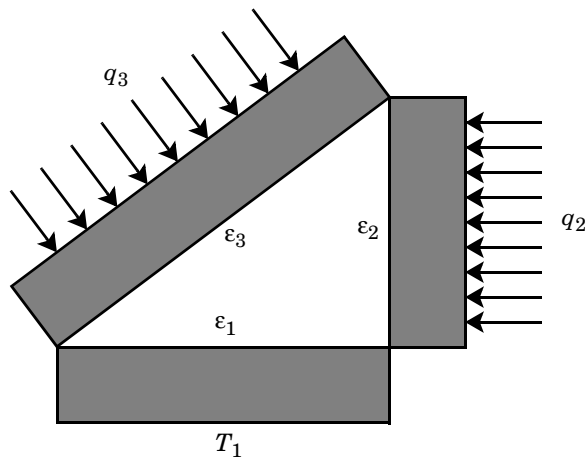


Figure 1: Three domains form a triangular cavity to study heat radiation.

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 $\epsilon_1 = 0.4$ on the bottom boundary, $\epsilon_2 = 0.6$ on the vertical boundary, and $\epsilon_3 = 0.8$ 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 $q_2 = 2000$ W/m² on the outer boundary of the vertical rectangle and $q_3 = 1000$ W/m² 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.

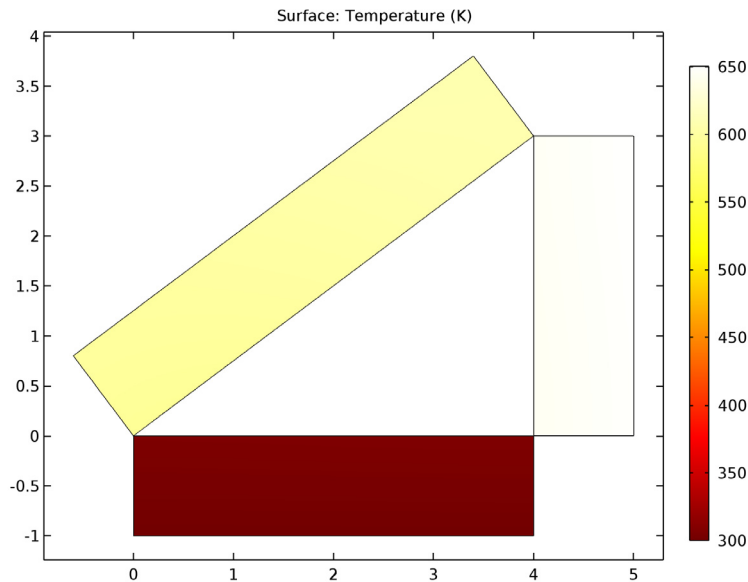


Figure 2: Temperature as a function of position.

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.

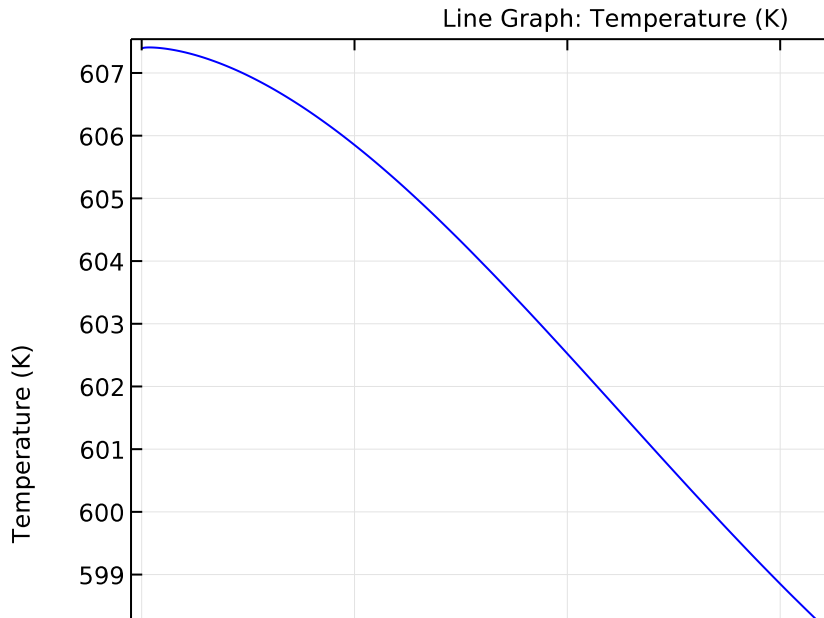


Figure 3: Temperature distribution along the inclined boundary of the cavity.

Figure 4 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*.

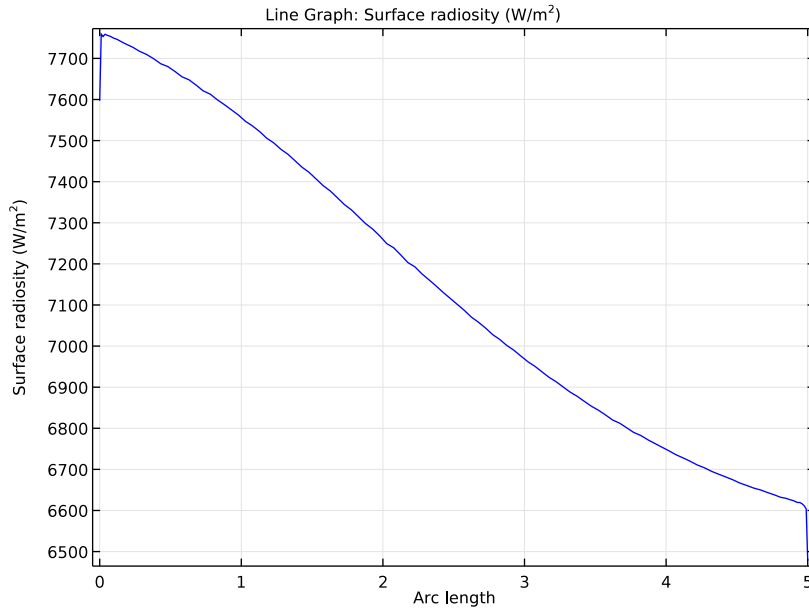


Figure 4: Radiosity along the inclined boundary of the cavity.

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](#).

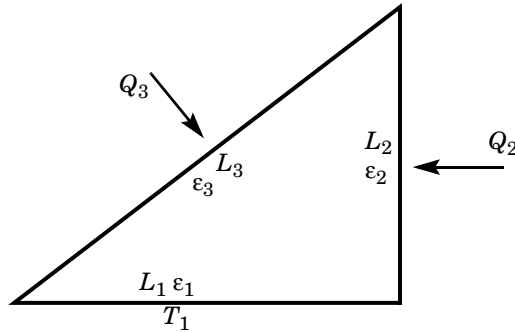


Figure 5: Problem sketch for theoretical analysis of radiation in a triangular cavity.

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:

$$L_1 = 4 \text{ m}, \quad \varepsilon_1 = 0.4$$

$$L_2 = 3 \text{ m}, \quad \varepsilon_2 = 0.6$$

$$L_3 = 5 \text{ m}, \quad \varepsilon_3 = 0.8$$

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: $Q_1 = -11,000 \text{ W/m}$, $T_2 = 641 \text{ K}$, and $T_3 = 600 \text{ K}$.

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

1 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** section.
- 3 In the **Width** text field, type 4.
- 4 Locate the **Position** section. In the **y** text field, type -1.001.

Rectangle 2 (r2)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for Rectangle, locate the **Size** section.
- 3 In the **Height** text field, type 3.
- 4 Locate the **Position** section. In the **x** text field, type 4.001.

Rectangle 3 (r3)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for Rectangle, locate the **Size** section.
- 3 In the **Width** text field, type 5.
- 4 Locate the **Position** section. In the **x** text field, type -0.001.
- 5 In the **y** text field, type 0.001.
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type $\text{atan}(3/4)$ [rad].
- 7 Click the **Build All Objects** button.
- 8 Click the **Zoom Extents** button on the **Graphics** toolbar.

This completes the geometry modeling stage.

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

Heat Transfer in Solids 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Heat Transfer with Surface-to-Surface Radiation (ht)** click **Heat Transfer in Solids 1**.
- 2 In the **Settings** window for Heat Transfer in Solids, locate the **Heat Conduction, Solid** section.
- 3 From the k list, choose **User defined**. In the associated text field, type 400.
- 4 Locate the **Thermodynamics, Solid** section. From the C_p list, choose **User defined**. In the associated text field, type 385.
- 5 From the ρ list, choose **User defined**. In the associated text field, type 8700.

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, locate the **Initial Values** section.
- 3 In the T text field, type 300.

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 5 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 adding or modifying **Opaque** subnodes under the **Heat Transfer in Solids** node.

Diffuse Surface 2

- 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 **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*Edge 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.
- 2 In the **Settings** window for Edge, locate the **Boundary Selection** section.
- 3 From the **Geometric entity level** list, choose **Entire geometry**.

Size 1

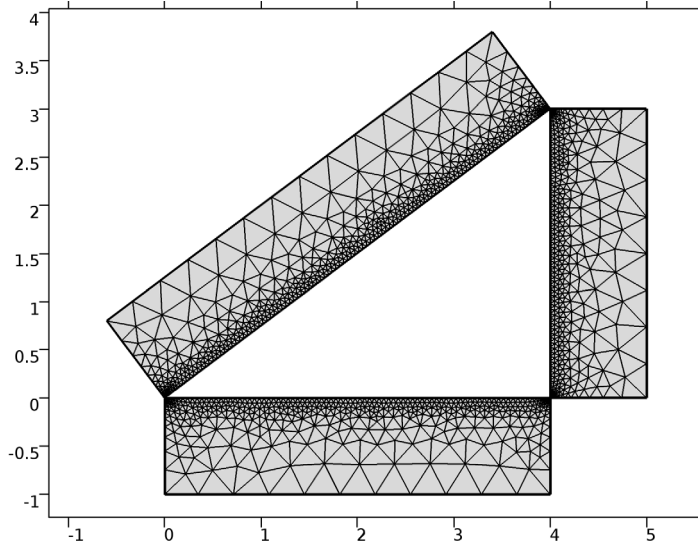
- 1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Size**.
- 2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 3, 6, and 9 only.
- 5 Locate the **Element Size** 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.

Size 2

- 1 Right-click **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 **Point**.
- 4 Select Points 2, 4, 6, and 8–10 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 0.01.

Free Triangular I

- 1 In the **Model Builder** window, right-click **Mesh I** and choose **Free Triangular**.
- 2 In the **Settings** window for Free Triangular, click to expand the **Tessellation** section.
- 3 From the **Method** list, choose **Advancing front**.
- 4 Right-click **Mesh I** and choose **Build All**.



The complete mesh consists of roughly 2700 elements.

STUDY I

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

RESULTS

Temperature (ht)

The computation takes around 10 seconds.

- 1 Click the **Zoom Extents** button on the **Graphics** toolbar.

The first default plot shows the temperature field; compare with [Figure 2](#).

The two additional default plots visualize the temperature field 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.

Temperature Profile vs Arc Length

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

Temperature Profile vs Arc Length 1

- 1 In the **Model Builder** window, right-click **Temperature Profile vs Arc Length** and choose **Duplicate**.
- 2 In the **Settings** window for 1D Plot Group, type Surface Radiosity Profile vs Arc Length in the **Label** text field.

Surface Radiosity Profile vs Arc Length

- 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 **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.

- 1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.
- 2 Select Boundary 6 only.
- 3 In the **Settings** window for Line Integration, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer with Surface-to-Surface Radiation>Boundary fluxes>ht.ndflux - Normal conductive heat flux**.
- 4 Click the **Evaluate** button.

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

Derived Values

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 **Expression** section.

4 In the **Expression** text field, type $T/3$.

5 Click the **Evaluate** button.

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

Derived Values

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 **Expression** section.

4 In the **Expression** text field, type $T/5$.

5 Click the **Evaluate** button.

TABLE

1 Go to the **Table** window.

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

