



# 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).

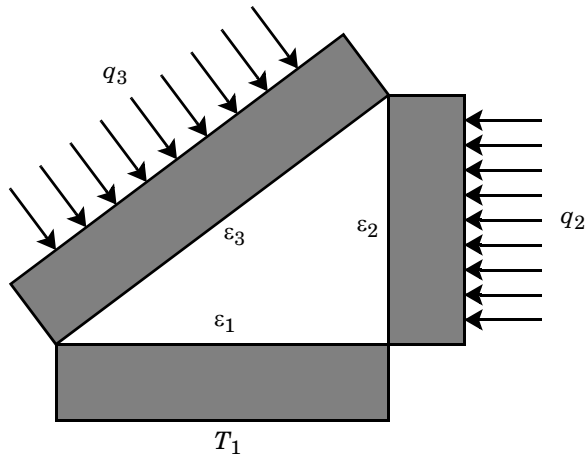


Figure 1: Three domains form a triangular cavity to study heat radiation.

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/m<sup>2</sup> on the outer boundary of the vertical rectangle and  $q_3 = 1000$  W/m<sup>2</sup> 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  $Q_1 = -11,000 \text{ W/m}$  leaves the cavity through the bottom boundary, that the vertical boundary has a temperature of  $T_2 = 641 \text{ K}$ , and that the inclined boundary has a temperature of  $T_3 = 600 \text{ K}$ . 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.

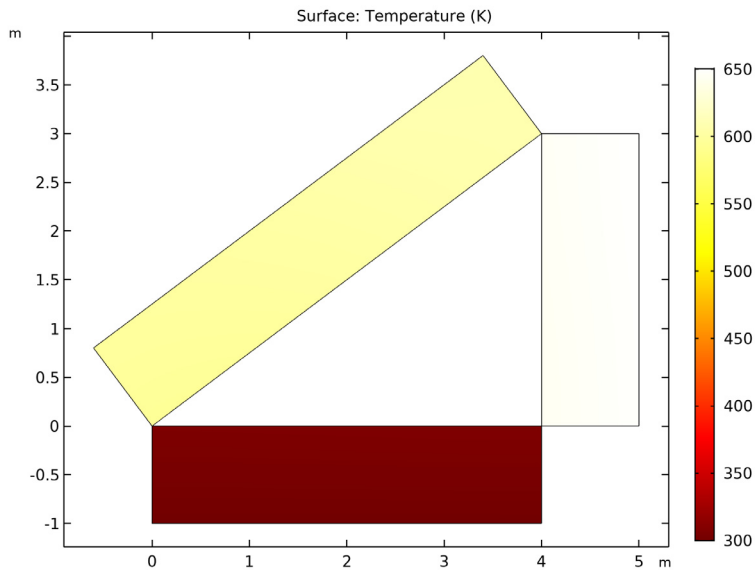


Figure 2: Temperature as a function of position.

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 lower-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.

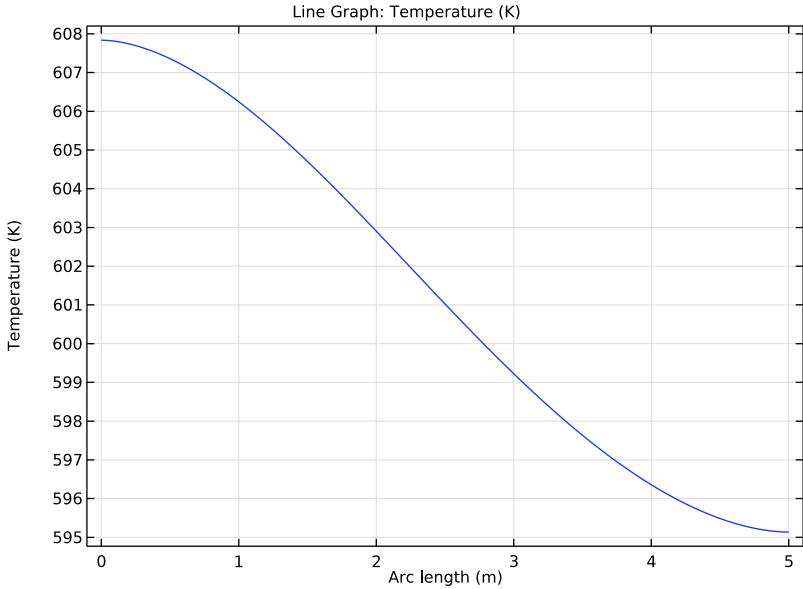


Figure 3: Temperature distribution along the inclined boundary of the cavity.

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*.

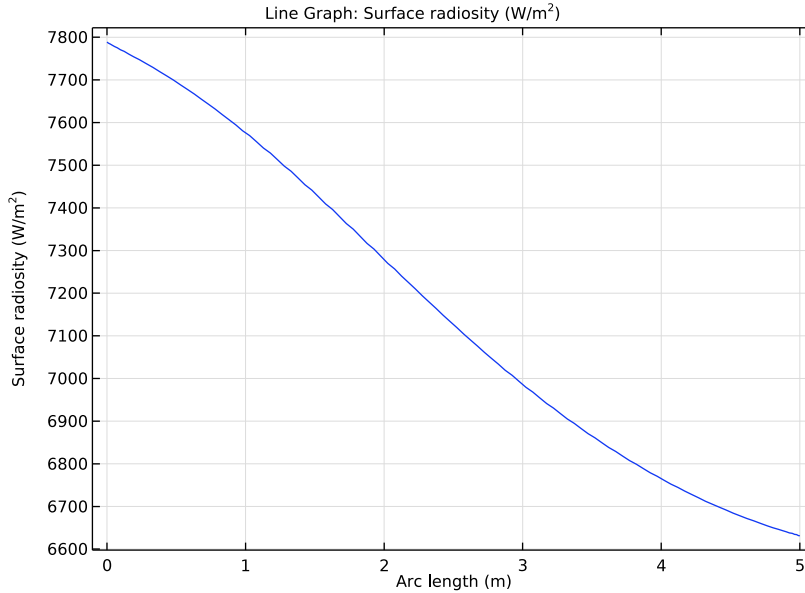
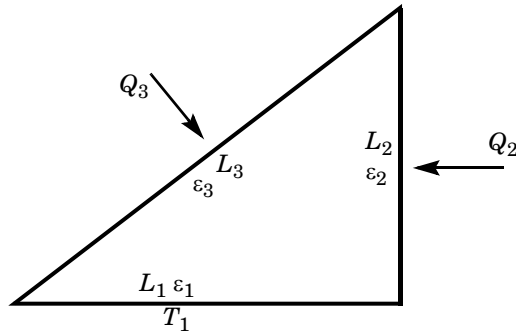


Figure 4: Radiosity along the inclined boundary of the cavity.

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:

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

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

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

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:  $Q_1 = -11,000 \text{ W/m}$ ,  $T_2 = 641 \text{ K}$ , 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**

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**.
- 3 Click **Add**.
- 4 Click **Study**.
- 5 In the **Select Study** tree, select **General 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 In the **Geometry** toolbar, click **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 In the **Geometry** toolbar, click **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 In the **Geometry** toolbar, click **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  $\text{atan}(3/4)$  [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 In the **Geometry** toolbar, click **Build All**.
- 5 Click the **Zoom Extents** button in the **Graphics** toolbar.  
This completes the geometry modeling stage.

## MATERIALS

### Material 1 (mat1)

- 1 In 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	Variable	Value	Unit	Property group
Thermal conductivity	$k_{iso}$ ; $k_{ij} = k_{iso}$ , $k_{ij} = 0$	400	W/(m·K)	Basic
Density	$\rho$	8700	kg/m <sup>3</sup>	Basic
Heat capacity at constant pressure	$C_p$	385	J/(kg·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**, locate the **Initial Values** section.
- 3 In the  $T$  text field, type 300.

Next, define the boundary conditions.

### *Heat Flux 1*

- 1 In 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 In 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 In 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.

## SURFACE-TO-SURFACE RADIATION (RAD)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Surface-to-Surface Radiation (rad)**.
- 2 Select Boundaries 3, 7, and 9 only.

### *Diffuse Surface 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Surface-to-Surface Radiation (rad)** click **Diffuse Surface 1**.
- 2 In the **Settings** window for **Diffuse Surface**, locate the **Ambient** section.
- 3 Find the **Ambient temperature** subsection. In the  $T_{\text{amb}}$  text field, type 300.

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

When the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling is added the default domain opacity is set to **From heat transfer interface** which means that the solid domains are opaque by default while the fluid domains are transparent by default. You can override these default settings by adding one or multiple **Opacity** node(s) under the **Surface-to-Surface Radiation** interface.

#### *Diffuse Surface 2*

- 1 In 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 Find the **Ambient temperature** subsection. In the  $T_{\text{amb}}$  text field, type 300.
- 5 Locate the **Surface Emissivity** section. From the  $\epsilon$  list, choose **User defined**. In the associated text field, type 0.4.

#### *Diffuse Surface 3*

- 1 In 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 Find the **Ambient temperature** subsection. In the  $T_{\text{amb}}$  text field, type 300.
- 5 Locate the **Surface Emissivity** section. From the  $\epsilon$  list, choose **User defined**. In the associated text field, type 0.6.

## **MESH 1**

#### *Edge 1*

- 1 In the **Mesh** toolbar, click **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 **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.

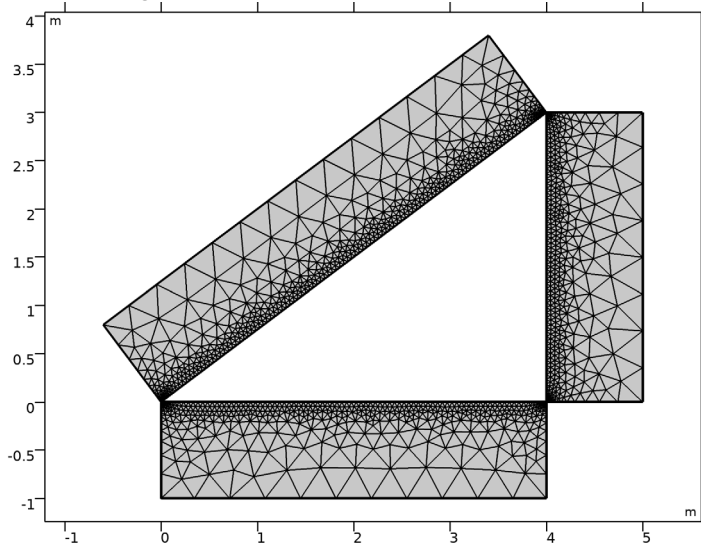
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated text field, type 0.05.

*Size 2*

- 1 In the **Model Builder** window, 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.
- 8 Click **Build Selected**.

*Free Triangular 1*

- 1 In the **Mesh** toolbar, click **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 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 In the **Home** toolbar, click **Compute**.

The computation takes a few 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 In the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.

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

### *Line Graph 1*

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

2 Select **Boundary 3** only.

3 In 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](#)).

### *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 **ID 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 **Component 1>Surface-to-Surface Radiation>Surface radiosity>rad.J - Surface radiosity - W/m<sup>2</sup>**.

3 In 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 In 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 **Component 1>Heat Transfer in Solids>Boundary fluxes>ht.ndflux - Normal conductive heat flux - W/m<sup>2</sup>**.
- 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 In 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 645 K.

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

#### RESULTS

#### Line Integration 3

- 1 In 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.

