Model created in COMSOL Multiphysics 6.4

Surface-to-Surface Radiation with Diffuse and Specular Reflection

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

This tutorial shows how to use the Surface-to-Surface Radiation interface to simulate radiative heat transfer with radiation between diffuse emitters and diffuse-and-specular reflectors. This application is separated in two parts. The first part focuses on a validation test for the radiative heat flux computed from the ray shooting algorithm, wherein results are compared to an analytical solution for two parallel plates at constant temperature. The second part introduces coupling with Heat Transfer in Solids.

Introduction

Opaque surfaces can reflect incident energy in a diffuse and/or specular way. The smoother the opaque surface, the more specular the reflection is expected to be.

Following Ref. 1, the hemispherical reflectivity ρ of an opaque surface reads

$$\rho = \rho^s + \rho^d = 1 - \varepsilon, \tag{1}$$

where ρ^s, ρ^d, and ε are the specular reflectivity, the diffuse reflectivity, and the diffuse emissivity of the surface, respectively.

For an irradiation G, the outgoing energy is the radiosity $\varepsilon E_b + \rho^d G$ plus the energy reflected specularly $\rho^s G$, so that the net radiative heat flux reads

$$q = \varepsilon(G - E_b)$$

with E_b the blackbody emissive power

$$E_b = \sigma T^4,$$

and G is the irradiation

$$G = \int_A J(\mathbf{r}')dF^s_{dA-dA'} + G^s_0.$$

The latter expression depends on the external irradiation G^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 ray shooting algorithm as the surface-to-surface radiation method to model radiative heat transfer. 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.

When **Ray shooting** is selected in the **Radiation Settings** section, the user can adjust three parameters to speed up the simulation and/or to get more accurate results, namely the **Radiation resolution**, the **Tolerance**, and the **Maximum number of adaptations**. These settings are described in the *Settings for the Surface-to-Surface Radiation Interface* section of the *Heat Transfer Module User's Guide*. This benchmark model illustrates the importance of the radiation resolution.

The Opaque Surface feature is used to model surfaces that are diffuse emitters and diffuse-specular reflectors. The emissivity and the specular reflectivity are defined in the **Surface Radiative Properties** section. The diffuse reflectivity is then set from Equation 1.

The model is separated into two parts.

In the first part, the numerical result obtained with the Surface-to-Surface Radiation interface is compared to the exact analytical solution.

This computes the heat flux at the surfaces of two identical infinitely long (out-of-plane in 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 emissivity, $\varepsilon_l = \varepsilon_u = \varepsilon$, and the same probability of specular reflection, such that $\rho_l = \rho_u = \rho$.

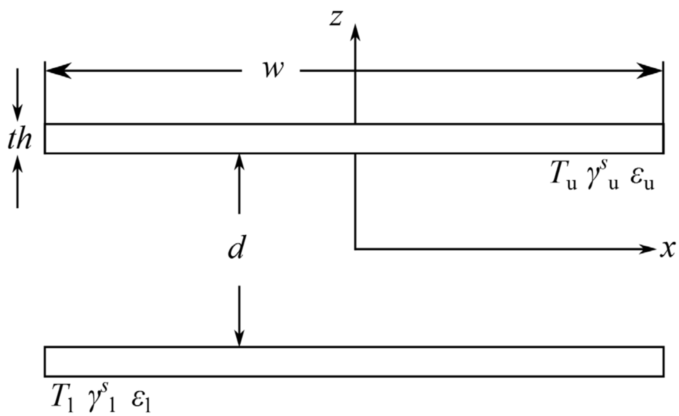

Figure 1: Schematics of the problem. The width of the plates is w = 10 cm, their thickness th = w/20, and the distance between the plates set to d = 10 cm. The temperature, emissivity, and probability of specular reflection are equal for both plates and respectively set to T = 300 K, ε = 0.6, and ρ^s = 0.32.

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):

$$1-(1-\rho^s)\int_{-W/2}^{W/2}\frac{1}{2}\sum_{k=1}\frac{(\rho^s)^{k-1}k^2d\xi'}{(k^2+(\xi-\xi')^2)^{3/2}} =$$

$$\frac{\Psi(\xi)}{\varepsilon}-\frac{\rho^d}{\varepsilon}\left(\int_{-W/2}^{W/2}\Psi(\xi')\frac{1}{2}\sum_{k=1}^{\infty}\frac{(\rho^s)^{k-1}k^2d\xi'}{(k^2+(\xi-\xi')^2)^{3/2}}\right)$$

Here $\xi = x/d$, $W = w/d$, and $\Psi = q/E_b$. The heat flux can be computed using numerical quadrature; a typical solution is presented in Figure 2.

The second part keeps the parallel plates arrangement (same geometry) but changes 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	ρ^s
Lower plate	0.9	0.01
Upper plate	0.6	0.32

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. Both plates simultaneously loss heat by natural convection. 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 numerical solutions (benchmark model). A good agreement is observed; the relative error in the radiative heat flux is below one percent.

When the radiative heat flux computed using the Surface-to-Surface Radiation interface is coupled to the Heat Transfer in Solids interface, the temperature field shown in Figure 3 is obtained. Figure 4 displays the normalized heat flux at the top of the lower plate and at the bottom of the upper plate.

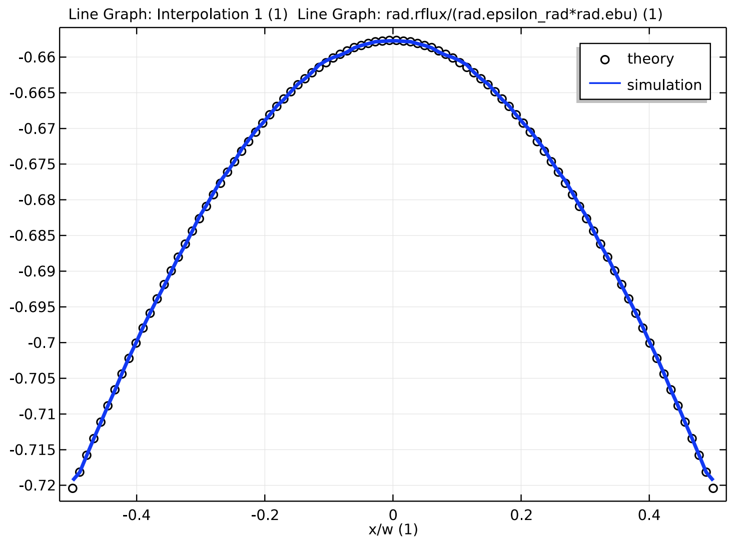

Figure 2: Normalized heat flux at the bottom of the upper plate for T = 300 K, ε = 0.6, ρ^s = 0.32 and w/d = 1. The black circles represent the exact solution obtained from numerical quadrature see Ref. 1.

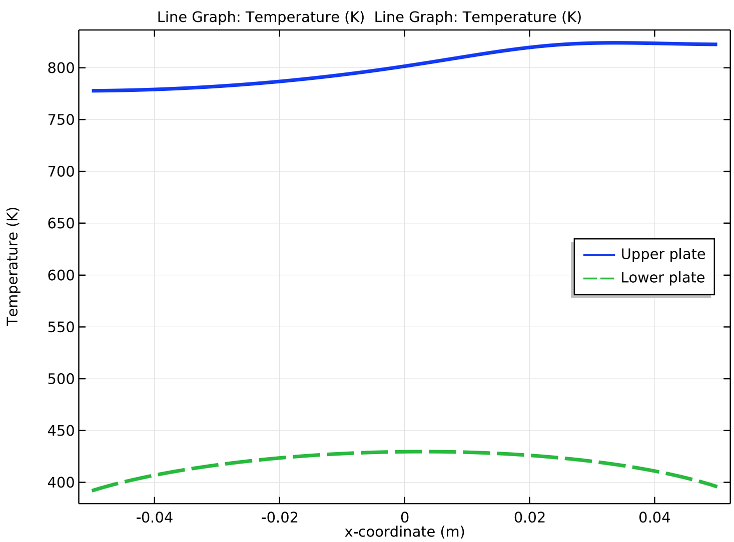

Figure 3: Temperature of the inner surfaces of the plates for the coupled model.

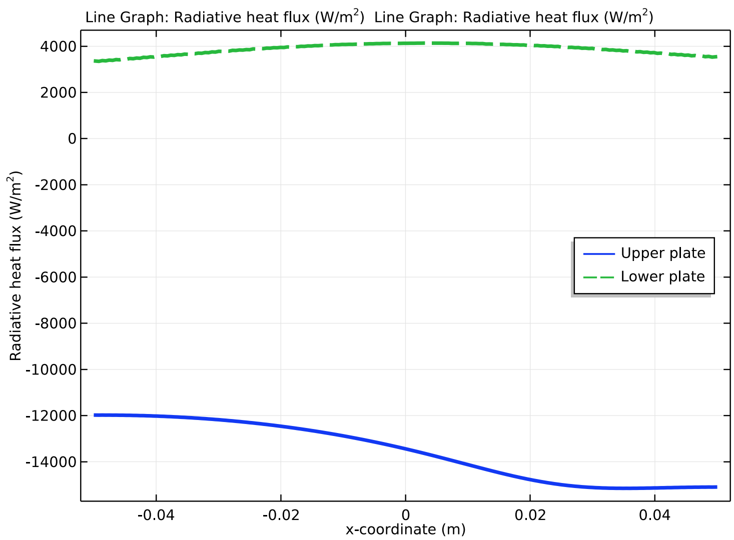

Figure 4: Radiative heat flux at the surface of the plates for the coupled model.

Notes About the COMSOL Implementation

The first study could have been done using lines instead of rectangles to model parallel plates, and even making use of the Symmetry for Surface-to-Surface Radiation feature to reduce the number of degrees of freedom. The Radiation Settings to use in the second study for such a geometry are determined from the first (validation) study.

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_ray_shooting`

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** > **Surface-to-Surface Radiation (rad)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
w	10[cm]	0.1 m	Width of the plates
d	10[cm]	0.1 m	Distance between the plates
th	w/20	0.005 m	Thickness of the plates
T0	300[K]	300 K	Room temperature

GEOMETRY 1

Lower plate

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 w.

4 In the **Height** text field, type th.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **y** text field, type -(d+th)/2.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

8 From the **Show in physics** list, choose **Boundary selection**.

9 In the **Label** text field, type `Lower plate`.

Upper plate

1 Right-click **Lower plate** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **y** text field, type `(d+th)/2`.

4 In the **Label** text field, type `Upper plate`.

DEFINITIONS

Study 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

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

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

5 From the **Selection** list, choose **All boundaries**.

6 In the **Label** text field, type `Study 1`.

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

Name	Expression	Unit	Description
epsilon_mat	0.6		Emissivity
rhod_mat	0.08		Diffuse reflectivity

Interpolation 1 (int1)

1 In the **Definitions** toolbar, click **Interpolation**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `parallel_plates_diffuse_specular_data.txt`.

5 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	1

6 In the **Function** table, enter the following settings:

Function	Unit
reference_solution	1

7 Locate the **Definition** section. In the **Function name** text field, type `reference_solution`.

8 Click the **Show More Options** button in the **Model Builder** toolbar.

9 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

10 Click **OK**.

SURFACE-TO-SURFACE RADIATION (RAD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Surface-to-Surface Radiation (rad)**.

2 In the **Settings** window for **Surface-to-Surface Radiation**, locate the **Radiation Settings** section.

3 Find the **Radiation settings** subsection. From the **Surface-to-surface radiation method** list, choose **Ray shooting**.

The resolution is increased because rays are emitted from the plates in all direction so some of them do not even strike the plates.

4 From the **Radiation resolution** list, choose **256**.

5 Click to expand the **View Factor** section. Find the **Geometry representation** subsection. Select the **High order mesh elements** checkbox. When specular radiation is present it is good practice to do this because it improves the accuracy of the specular radiation evaluation at a slight computational cost increase. Because the geometry of this model contains only flat surfaces, selecting the **High order mesh elements** checkbox has no influence on accuracy but it makes the model ready for geometries with curved surfaces.

The feature Opaque Surface allows to handle mixed diffuse and specular reflection.

Opaque Surface 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Opaque Surface**.

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

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Radiation Direction** section. From the list, choose **Positive normal direction**.

5 Locate the **Model Input** section. In the T text field, type `T0`.

6 Locate the **Ambient** section. In the T_{amb} text field, type `0`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T_{init} text field, type `T0`.

MATERIALS

Plates, boundaries

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 `Plates, boundaries` in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Surface emissivity	epsilon_rad	`epsilon_mat`	1	Basic
Diffuse reflectivity	rho_d_rad	`rhod_mat`	1	Basic

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Size

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

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

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

4 Click **Build All**.

STUDY 1

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

RESULTS

Table 1

1 In the **Results** toolbar, click **Table**.

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

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `parallel_plates_diffuse_specular_data.txt`.

Validation

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

2 In the **Settings** window for **1D Plot Group**, type `Validation` 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 `Dimensionless radiative heat flux`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `x/w`.

7 Select the **y-axis label** checkbox. In the associated text field, type `q/(\varepsilon Eb)`.

Theory

1 Right-click **Validation** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Theory` in the **Label** text field.

3 Select Boundary 5 only.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `reference_solution(x/w)`.

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 **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

9 From the **Color** list, choose **From theme**.

10 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

11 From the **Resolution** list, choose **No refinement**.

12 Click to expand the **Legends** section. Select the **Show legends** checkbox.

13 Find the **Include** subsection. Clear the **Solution** checkbox.

14 Select the **Label** checkbox.

Simulation

1 In the **Model Builder** window, right-click **Validation** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Simulation` in the **Label** text field.

3 Select Boundary 5 only.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `rad.rflux/(rad.epsilon*rad.ebu)`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x/w`.

7 Locate the **Coloring and Style** section. From the **Width** list, choose **3**.

8 Locate the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

9 From the **Resolution** list, choose **No refinement**.

10 Locate the **Legends** section. Select the **Show legends** checkbox.

11 Find the **Include** subsection. Clear the **Solution** checkbox.

12 Select the **Label** checkbox.

Validation

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

ADD PHYSICS

1 In 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 Click the **Add to Component 1** button in the window toolbar.

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

ADD MULTIPHYSICS

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

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

3 In the tree, select **Heat Transfer** > **Radiation** > **Heat Transfer with Surface-to-Surface Radiation**.

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

5 In the **Home** toolbar, click **Add Multiphysics** to close the **Add Multiphysics** window.

MATERIALS

Quartz

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

3 Select Domain 1 only.

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

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	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

Copper

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domain 2 only.

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

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

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	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

DEFINITIONS

Study 2, Upper plate

1 In the **Model Builder** window, right-click **Study 1** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Upper plate**.

4 In the **Label** text field, type `Study 2, Upper plate`.

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

Name	Expression	Unit	Description
epsilon_mat	0.6		Specular reflectivity, upper plate

Study 2, Lower plate

1 Right-click **Study 2, Upper plate** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Lower plate**.

4 In the **Label** text field, type `Study 2, Lower plate`.

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

Name	Expression	Unit	Description
epsilon_mat	0.9		Diffuse reflectivity, lower plate
rhod_mat	0.09		Diffuse reflectivity

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 **Physical Model** section.

3 In the T_{ref} text field, type `T0`.

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 `T0`.

Heat Flux 1

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

2 Click in the **Graphics** window and then press Ctrl+A to select all boundaries.

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

4 From the **Selection** list, choose **All boundaries**.

5 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

6 In the h text field, type `10[W/(m^2*K)]`.

7 In the T_{ext} text field, type `T0`.

Deposited Beam Power 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Deposited Beam Power**.

2 Select Boundary 6 only.

3 In the **Settings** window for **Deposited Beam Power**, locate the **Beam Orientation** section.

4 Specify the **e** vector as

0	x
-1	y

5 Locate the **Beam Profile** section. In the P_0 text field, type `4000[W]`.

6 Specify the **O** vector as

0.025[m]	x
d+w	y

7 In the σ text field, type `0.01[m]`.

STUDY 1, WITHOUT HEAT TRANSFER

1 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

2 In the **Rename Study** dialog, type `Study 1, without Heat Transfer` in the **New label** text field.

3 Click **OK**.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1, without Heat Transfer** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Definitions** > **Study 2, Upper plate** and **Component 1 (comp1)** > **Definitions** > **Study 2, Lower plate**.

5 Right-click and choose **Disable**.

6 In the tree, select **Component 1 (comp1)** > **Heat Transfer in Solids (ht)** and **Component 1 (comp1)** > **Multiphysics** > **Heat Transfer with Surface-to-Surface Radiation 1 (htrad1)**.

7 Right-click and choose **Disable in Model**.

ADD STUDY

1 In 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 **General Studies** > **Stationary**.

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

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

STUDY 2, WITH HEAT TRANSFER

1 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

2 In the **Rename Study** dialog, type `Study 2, with Heat Transfer` in the **New label** text field.

3 Click **OK**.

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Definitions** > **Study 1**.

5 Right-click and choose **Disable**.

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

RESULTS

Temperature

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2, with Heat Transfer/ Solution 2 (sol2)**.

Upper plate

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

2 In the **Settings** window for **Line Graph**, type `Upper plate` in the **Label** text field.

3 Select Boundary 5 only.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `x`.

6 Locate the **Coloring and Style** section. From the **Width** list, choose **3**.

7 Locate the **Legends** section. Select the **Show legends** checkbox.

8 Find the **Include** subsection. Clear the **Solution** checkbox.

9 Select the **Label** checkbox.

Lower plate

1 Right-click **Upper plate** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, type `Lower plate` in the **Label** text field.

3 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

4 Select Boundary 3 only.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

Temperature

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

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Middle right**.

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

Radiative heat flux

1 Right-click **Temperature** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Radiative heat flux` in the **Label** text field.

Upper plate

1 In the **Model Builder** window, expand the **Radiative heat flux** node, then click **Upper plate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `rad.rflux`.

Lower plate

1 In the **Model Builder** window, click **Lower plate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `rad.rflux`.

Radiative heat flux

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

Mean radiative heat flux

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration** > **Line Integration**.

2 In the **Settings** window for **Line Integration**, type `Mean radiative heat flux` in the **Label** text field.

3 Select Boundaries 3 and 5 only.

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

Expression	Unit	Description
`rad.rflux/(w*rad.epsilon*rad.ebu)`	`m`	`Radiative heat flux`

5 Click **Evaluate**.

TABLE 2

Go to the **Table 2** window.

Relative error in radiative heat flux

1 Right-click **Mean radiative heat flux** and choose **Duplicate**.

The radiative heat flux is going to be compared to the analytical solution.

2 In the **Settings** window for **Line Integration**, type `Relative error in radiative heat flux` in the **Label** text field.

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

Expression	Unit	Description
`((rad.rflux/(rad.epsilon*rad.ebu))/reference_solution(x)-1)^2/w`		`Relative error`

4 Click **Evaluate**.

Mean temperature, Lower plate

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration** > **Surface Integration**.

2 In the **Settings** window for **Surface Integration**, type `Mean temperature, Lower plate` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2, with Heat Transfer/Solution 2 (sol2)**.

4 Select Domain 1 only.

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

Expression	Unit	Description
`T/(w*th)`	`m^2*K`	`Temperature`

6 Click **Evaluate**.

Mean temperature, Upper plate

1 Right-click **Mean temperature, Lower plate** and choose **Duplicate**.

2 In the **Settings** window for **Surface Integration**, type `Mean temperature, Upper plate` in the **Label** text field.

3 Select Domain 2 only.

4 Click **Evaluate**.

Study 1, without Heat Transfer/Solution 1, inner faces

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** > **Study 1, without Heat Transfer/Solution 1 (sol1)** and choose **Duplicate**.

3 In the **Settings** window for **Solution**, type `Study 1, without Heat Transfer/ Solution 1, inner faces` in the **Label** text field.

Selection

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

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

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

4 Select Boundaries 3 and 5 only.

Extrusion 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

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

3 From the **Dataset** list, choose **Study 1, without Heat Transfer/Solution 1, inner faces (sol1)**.

4 Locate the **Extrusion** section. In the **z maximum** text field, type `2`.

Surface Radiosity 3D

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

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

Radiosity

1 Right-click **Surface Radiosity 3D** and choose **Surface**.

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

3 In the **Expression** text field, type `rad.J`.

4 In the **Label** text field, type `Radiosity`.

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

6 Click to expand the **Range** section. Select the **Manual color range** checkbox.

7 In the **Minimum** text field, type `285`.

8 In the **Maximum** text field, type `288`.

9 In the **Surface Radiosity 3D** toolbar, click **Plot**.

View 3D 2

1 In the **Model Builder** window, expand the **Results** > **Views** node, then click **View 3D 2**.

2 In the **Settings** window for **View 3D**, locate the **View** section.

3 Clear the **Show grid** checkbox.

Camera

1 In the **Model Builder** window, expand the **View 3D 2** node, then click **Camera**.

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

3 In the **Zoom angle** text field, type `121`.

4 Locate the **Position** section. In the **x** text field, type `0.1`.

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

6 In the **z** text field, type `0.1`.

7 Locate the **Target** section. In the **z** text field, type `0.1`.

8 Locate the **Up Vector** section. In the **x** text field, type `0`.

9 In the **y** text field, type `0`.

10 In the **z** text field, type `1`.

11 Locate the **Center of Rotation** section. In the **z** text field, type `0.1`.

12 Locate the **View Offset** section. In the **x** text field, type `-0.02`.

13 In the **y** text field, type `-0.02`.

Surface Radiosity 3D

Click the **Scene Light** button in the **Graphics** toolbar.