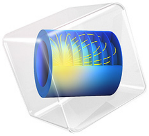
Created in COMSOL Multiphysics 5.5



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 $\mathrm{rn1} \leq \rho$ 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 $\mathrm{rn2} \geq \rho^{\mathrm{s}}/\rho$ 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.

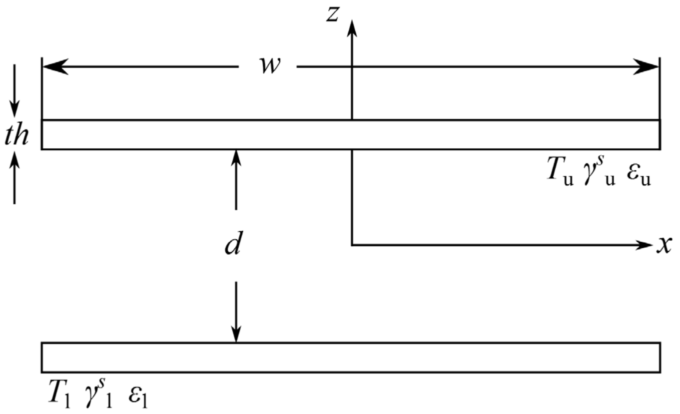


Figure 1: Schematics of the problem. The width of the plates are w = 10 cm, their thickness th = w/20, and the distance between the plates set to d = 10 cm. The temperature, emittance and probability of specular reflection are equal for both plates and respectively set to T = 300 K, ε = 0.6, and γ^s = 0.8.

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	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).

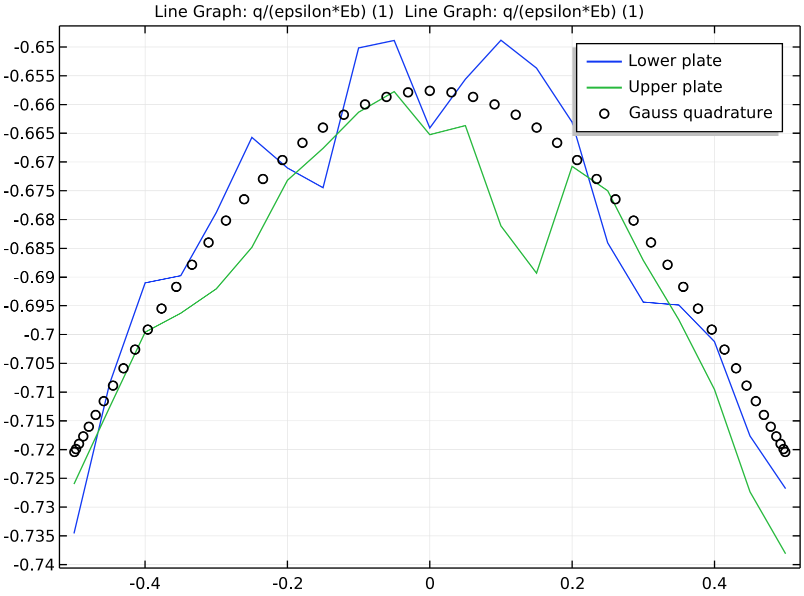


Figure 2: Normalized heat flux at the top of the lower plate (blue) and at the bottom of the upper plate (green) for $T = 300$ *K,* $\varepsilon = 0.6$*,* $\gamma^s = 0.8$ *and* $w/d = 1$*. The black circles represent*

the exact solution (the same for both surfaces) obtained from numerical quadrature see Ref. 1.

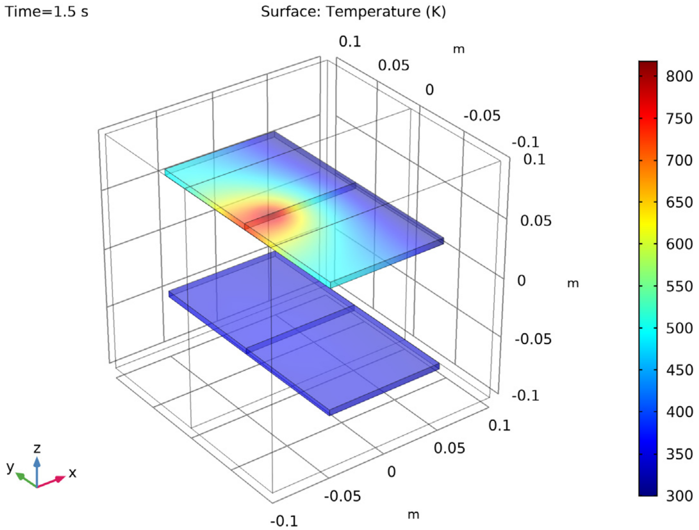


Figure 3: Temperature at the surface of the plates for the coupled model.

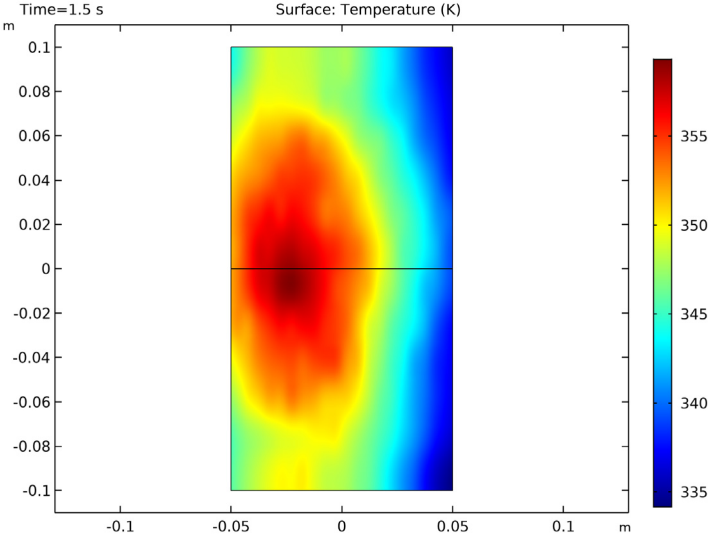


Figure 4: Temperature at the surface of the lower plate for the coupled model.

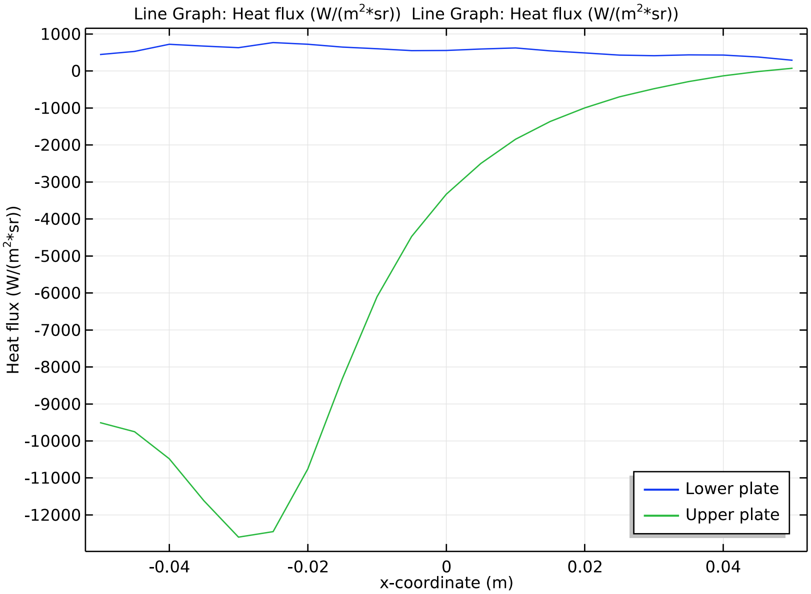


Figure 5: Heat flux at the top of the lower plate (blue) and at the bottom of the upper plate

(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 **General Studies>Time Dependent**.

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`
`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`	0.020944	`Solid angle subtended per ray`
`T0`	`300[K]`	300 K	`Room temperature`

GEOMETRY 1

Block 1 (blk1)

1 In 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 In 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 in the **Graphics** toolbar in order to see the entire geometry.

Work Plane 1 (wp1)

1 In 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 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

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

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

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

Define a selection list for the surrounding, lower and upper plate.

DEFINITIONS

Explicit 1

1 In 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 In 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 In 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 **Definitions** toolbar, click **Local 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 In the **Definitions** toolbar, click **Local 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`
`epsilon`	`1`		`Emittance of the surrounding`
`gamma_s`	`0`		`Probability of specular reflection`

Variables 3

1 In the **Definitions** toolbar, click **Local 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 In the **Definitions** toolbar, click **Local 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 In the **Definitions** toolbar, click **Local 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 In the **Definitions** toolbar, click **Local 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 In the **Definitions** toolbar, click **Local 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 **Particle Release and Propagation** 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 In 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. Click **Select Quantity**.

5 In the **Physical Quantity** dialog box, type `planeangle` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Plane angle (rad)**.

8 Click **OK**.

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 In 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 **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 $\mathrm{theta}_e mit_0$ 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 In 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. Click **Select Quantity**.

7 In the **Physical Quantity** dialog box, type `radiativeintensity` in the text field.

8 Click **Filter**.

9 In the tree, select **Transport>Radiative intensity (W/(m^2*sr))**.

10 Click **OK**.

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 In 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 has not been absorbed before. The probability of absorption is entered as `rho` in the **Probability** field of the **Primary Particle Condition** section.

Wall 3

1. In 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

Mapped 1

1. In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.
2. Select Boundaries 11 and 15 only.

Distribution 1

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

Distribution 2

1. In the **Mesh** toolbar, click **Distribution**.
2. Select Edges 14, 19, 40, and 43 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `1`.

Free Triangular 1

1. In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.
2. Select Boundary 2 only.

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

Swept 1

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

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

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.

STUDY 1

In the **Study** toolbar, click **Show Default Solver**.

Solution 1 (sol1)

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

2 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

3 From the **Maximum step constraint** list, choose **Constant**.

4 In the **Maximum step** 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, 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 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Surrounding: Study 2**.

6 Click **Disable**.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Lower Plate: Study 2**.

8 Click **Disable**.

9 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Upper Plate: Study 2**.

10 Click **Disable**.

11 In the **Model Builder** window, click **Study 1**.

12 In the **Settings** window for **Study**, locate the **Study Settings** section.

13 Clear the **Generate default plots** check box.

14 In 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 In 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 model's Application Libraries folder and double-click the file `parallel_plates_diffuse_specular_data.txt`.

5 Click **Update**.

1D Plot Group 1

1 In 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

1 In the **Validation** 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 `q/(epsilon*Eb)`.

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**.

Line Graph 2

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

2 In the **Settings** window for **Line Graph**, locate the **Legends** section.

3 In the table, enter the following settings:

Legends
`Upper plate`

4 Locate the **Selection** section. Select the **Active** toggle button.

5 Click **Clear Selection**.

6 Select Edge 31 only.

Validation

In the **Model Builder** window, click **Validation**.

Table Graph 1

1 In the **Validation** toolbar, click **Table Graph**.

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

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

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

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

6 From the **Positioning** list, choose **In data points**.

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
`Gauss quadrature`

10 In 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. 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. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.
5. Click **Add to Component 1** in the window toolbar.
6. In the **Model Builder** window, click **Component 1 (comp1)**.
7. In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

HEAT TRANSFER IN SOLIDS (HT)

1. In the **Settings** window for **Heat Transfer in Solids**, locate the **Domain Selection** section.
2. In the list, choose **1** and **2**.
3. Click **Remove from Selection**.
4. Select Domains 3–6 only.

Add the material properties to each plate.

MATERIALS

Material 1 (mat1)

1. In the **Materials** toolbar, click **Blank Material**.
2. In the **Settings** window for **Material**, type `Quartz` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. Click **Clear Selection**.
4. Select Domains 3 and 5 only.

5 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

Material 2 (mat2)

1 In the **Materials** toolbar, click **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	**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

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 `T0`.

Temperature 1

1 In 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. In 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. In 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. In 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. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Mathematical Particle Tracing (pt)** interface.
4. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.
5. Click **Add Study** in the window toolbar.
6. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

Start by adding a stationary study for the **Heat Transfer in Solids** interface only and select the boundary conditions associated to the second study.

1. In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.
2. Click to collapse the **Study Extensions** section. Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
3. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Surrounding: Study 1**.
4. Click **Disable**.
5. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Lower Plate: Study 1**.
6. Click **Disable**.
7. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Upper Plate: Study 1**.
8. Click **Disable**.

   Then add a time dependent study for the **Mathematical Particle Tracing** only.

Time Dependent

1. In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

   Use the same time steps as in **Study 1**.
2. In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.
3. In the table, clear the **Solve for** check box for the **Heat Transfer in Solids (ht)** interface.
4. Locate the **Study Settings** section. In the **Times** text field, type `0,1.5`.

   Select the boundary definitions for the second study.
5. Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
6. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Surrounding: Study 1**.
7. Click **Disable**.
8. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Lower Plate: Study 1**.
9. Click **Disable**.

10 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Definitions> Upper Plate: Study 1**.

11 Click **Disable**.

Generate the default study sequence for the two steps defined above.

12 In the **Model Builder** window, click **Study 2**.

13 In the **Settings** window for **Study**, locate the **Study Settings** section.

14 Clear the **Generate default plots** check box.

15 In the **Study** toolbar, click **Show Default Solver**.

Solution 2 (sol2)

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.

1 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

2 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.

3 In the **Settings** window for **For**, locate the **General** section.

4 In the **Number of iterations** text field, type `3`.

5 Move the **For 1** node on top of the study sequence. This will include the sequence in the loop.

Generate the initial radiative heat flux for the **Heat Transfer in Solids** interface computation.

6 In the **Model Builder** window, right-click **Solution 2 (sol2)** and choose **Compile Equations**.

7 In the **Settings** window for **Compile Equations**, type `Compile Equations: Time Dependent 0` in the **Label** text field.

8 Locate the **Study and Step** section. From the **Use study step** list, choose **Step 2: Time Dependent**.

9 Move the **Compile Equations: Time Dependent 0** node on top of the sequence above the **For 1** node.

10 Right-click **Solution 2 (sol2)** and choose **Dependent Variables**.

11 In the **Settings** window for **Dependent Variables**, type `Dependent Variables 0` in the **Label** text field.

12 Move the **Dependent Variables 0** node in between the **Compile Equations: Time Dependent 0** node and the **For 1** node.

13 In the **Model Builder** window, click **Dependent Variables 0**.

14 In the **Settings** window for **Dependent Variables**, locate the **General** section.

15 From the **Defined by study step** list, choose **User defined**.

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.

16 In the **Model Builder** window, click **Dependent Variables 1**.

17 In the **Settings** window for **Dependent Variables**, locate the **General** section.

18 From the **Defined by study step** list, choose **User defined**.

19 Locate the **Values of Variables Not Solved For** section. From the **Method** list, choose **Solution**.

20 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.

21 In the **Model Builder** window, click **Dependent Variables 2**.

22 In the **Settings** window for **Dependent Variables**, locate the **General** section.

23 From the **Defined by study step** list, choose **User defined**.

24 Locate the **Initial Values of Variables Solved For** section. From the **Method** list, choose **Initial expression**.

25 From the **Solution** list, choose **Zero**.

26 In the **Model Builder** window, click **Time-Dependent Solver 1**.

27 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.

28 From the **Maximum step constraint** list, choose **Constant**.

29 In the **Maximum step** text field, type `0.01`.

30 Click **Compute**.

Generate a plot of the surface temperatures.

RESULTS

3D Plot Group 2

1 In 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 **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 1

1. In the **3D Plot Group 2** toolbar, 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 **Component 1>Heat Transfer in Solids> Temperature>T - Temperature - K**.
3. In 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

1. In the **Results** toolbar, click **More Datasets** 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. Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
5. Select Boundaries 13 and 21 only.

2D Plot Group 3

In the **Results** toolbar, click **2D Plot Group**.

Surface 1

1. In the **2D Plot Group 3** toolbar, 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 **Component 1>Heat Transfer in Solids> Temperature>T - Temperature - K**.
3. In the **2D Plot Group 3** toolbar, click **Plot**.
4. Click the **Zoom Extents** button in the **Graphics** toolbar.

   Generate a figure displaying heat flux across the width of the plates.

1D Plot Group 4

1. In 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 **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
4. From the **Time selection** list, choose **Last**.
5. Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

1. In the **1D Plot Group 4** toolbar, click **Line Graph**.
2. Select Edge 28 only.

3. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `q`.
5. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `x`.
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
`Lower plate`

10. Locate the **Quality** section. From the **Resolution** list, choose **No refinement**.

Line Graph 2

1. Right-click **Line Graph 1** and choose **Duplicate**.
2. In the **Settings** window for **Line Graph**, locate the **Legends** section.
3. In the table, enter the following settings:

Legends
`Upper plate`

4. Locate the **Selection** section. Select the **Active** toggle button.
5. Click **Clear Selection**.
6. Select Edge 31 only.
7. In the **1D Plot Group 4** toolbar, click **Plot**.