Created in COMSOL Multiphysics 5.3

Thermo-Photo-Voltaic Cell

Introduction

The following example illustrates an application that maximizes surface-to-surface radiative fluxes and minimizes conductive heat fluxes.

A thermo-photo-voltaic (TPV) cell generates electricity from the combustion of fuel and through radiation. Figure 1 depicts the general operating principle. The fuel burns inside an emitting device that radiates intensely. Photo-voltaic (PV) cells—almost like solar cells—capture the radiation and convert it to electricity. The efficiency of a TPV device ranges from 1 % to 20 %. In some cases, TPVs are used in heat generators to co-generate electricity, and the efficiency is not so critical. In other cases TPVs are used as electric power sources, for example in automobiles (Ref. 1). In those cases efficiency is a major concern.

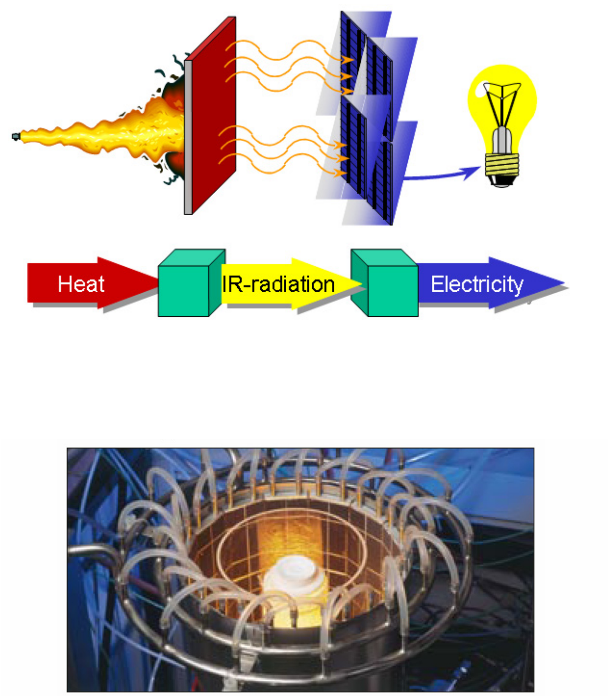


Figure 1: Operating principle of a TPV device (Ref. 2), and an image of a prototype system (Ref. 3).

TPV systems, unlike typical electronic systems, must maximize radiation heat transfer to improve efficiency. However, inherent radiation losses—radiation not converted to electric power—contributes to the PV cells' increased temperature. Further, heat transfer through conduction results in increased cell temperature. PV cells have a limited operating temperature range that depends on the type of material used. Solar cells are limited to temperatures below 80 °C, whereas high-efficiency semiconductor materials can withstand as much as 1000 °C. Photovoltaic efficiency is often a function of temperature with a maximum at some temperature above ambient.

To improve system efficiency, engineers prefer to use high-efficiency PV cells, which however can be quite expensive. To reduce system costs, engineers work with smaller-area PV cells and then use mirrors to focus the radiation on them. However, there is a limit for how much you can focus the beams; if the radiation intensity becomes too high, the cells can overheat. Thus engineers must optimize system geometry and operating conditions to achieve maximum performance at minimum material costs.

The following application, which uses the Heat Transfer with Surface-to-Surface Radiation interface, investigates the influence of operating conditions (flame temperature) on system efficiency and the temperature of components in a typical TPV system. The application can also assess the influence of geometry changes.

Model Definition

Figure 2 depicts the geometry and dimensions of the system under study. To reduce the temperature, the PV cells are water cooled on their back side (at the interface with the insulation).

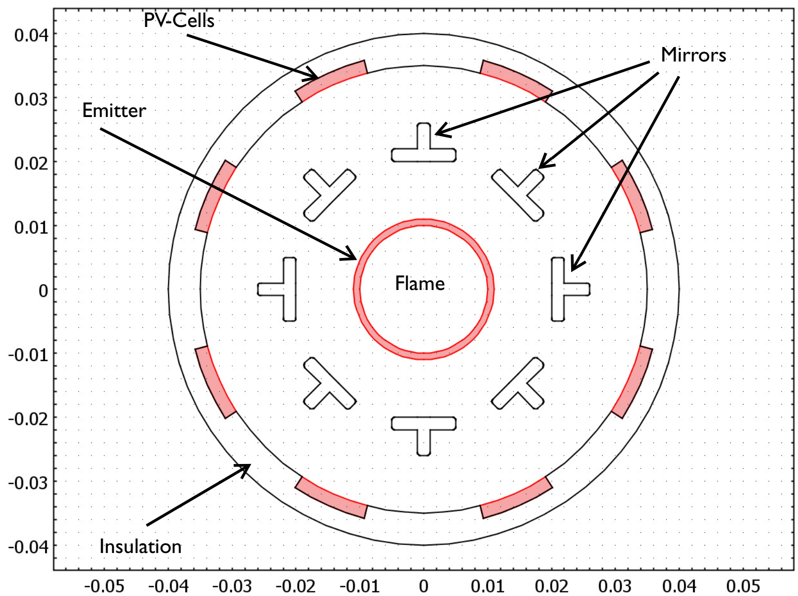


Figure 2: Geometry and dimensions of the modeled TPV system.

Conduction is always present on the different boundaries. The model simulates the emitter with a specific temperature, T_{heater}, on the inner boundary. At the outer emitter boundary, it takes radiation (surface-to-surface) into account in the boundary condition. It simulates the mirrors by taking radiation into account on all boundaries and applying a low emissivity. The inner boundaries of the PV cells and of the insulation also make use of radiation boundary conditions. However, the PV cells have a high emissivity and the insulation a low emissivity. Further, the PV cells convert a fraction of the irradiation to electricity instead of heat. Heat sinks on their inner boundaries simulate this effect by accounting for a boundary heat source, q, defined by

$$q = -G\eta_{pv}$$

where G is the irradiation flux (W/m^2) and η_{pv} is the PV cell's voltaic efficiency. The latter depends on the local temperature, with a maximum of 0.2 at 800 K:

$$\eta_{pv} = \begin{cases} 0.2\left[1-\left(\frac{T}{800\text{ K}}-1\right)^2\right] & T \le 1600\text{ K} \\ 0 & T > 1600\text{ K} \end{cases}$$

Figure 3 illustrates this expression for temperatures above 1000 K.

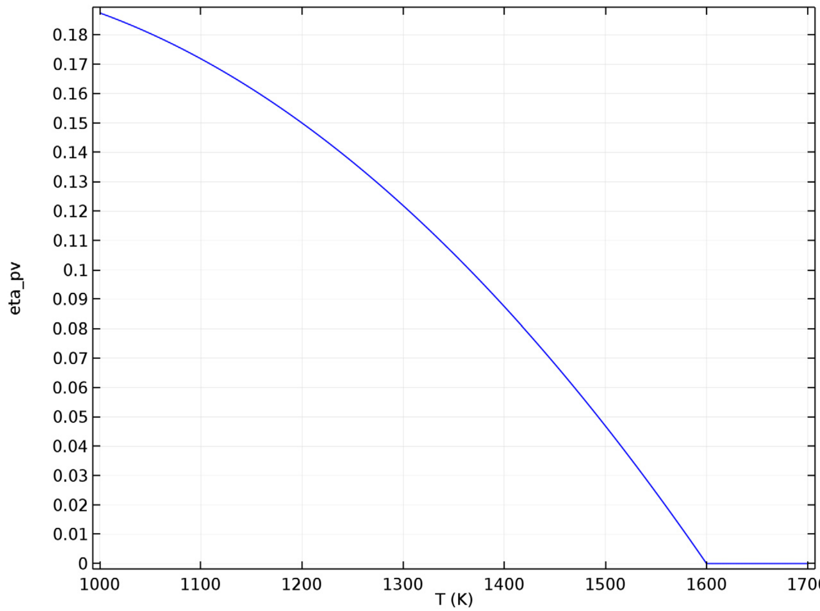


Figure 3: PV cell voltaic efficiency versus temperature.

At the outer boundary of the PV cells, the model applies convective water cooling by setting h to 50 W/(m^2·K), and T_{amb} to 273 K. Finally, at the outer boundary of the insulation it applies convective cooling with h set to 5 W/(m^2·K) and T_{amb} to 293 K.

Table 1 summarizes the material properties.

TABLE 1: MATERIAL PROPERTIES

COMPONENT	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]	ε
Emitter	10	2000	900	0.99
Mirror	10	5000	840	0.01
PV Cell	93	2000	840	0.99
Insulation	0.05	700	100	0.1

The model calculates the stationary solution for a range of emitter temperatures (1000 K to 2000 K) using the parametric solver.

Results and Discussion

The results shows that the device experiences a significant temperature distribution that varies with operating conditions. Figure 4 depicts the stationary distribution at operating conditions with an emitter temperature of 2000 K.

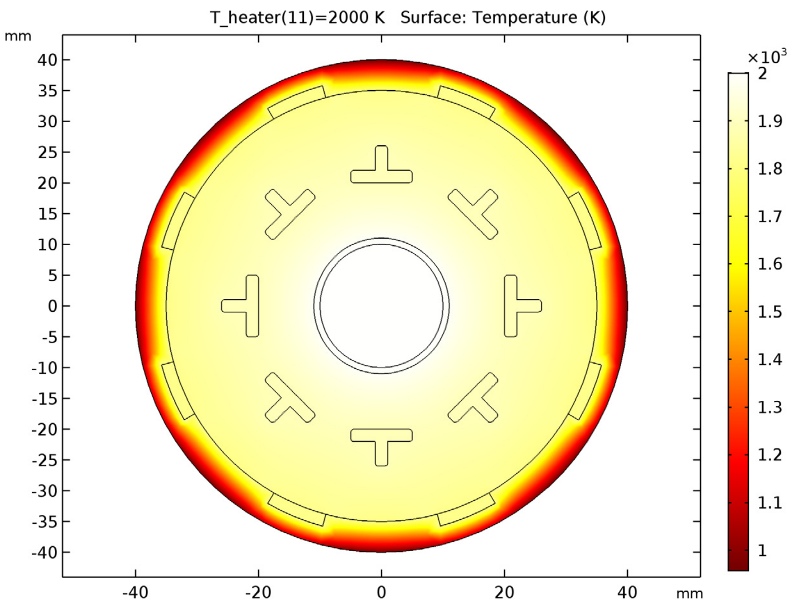
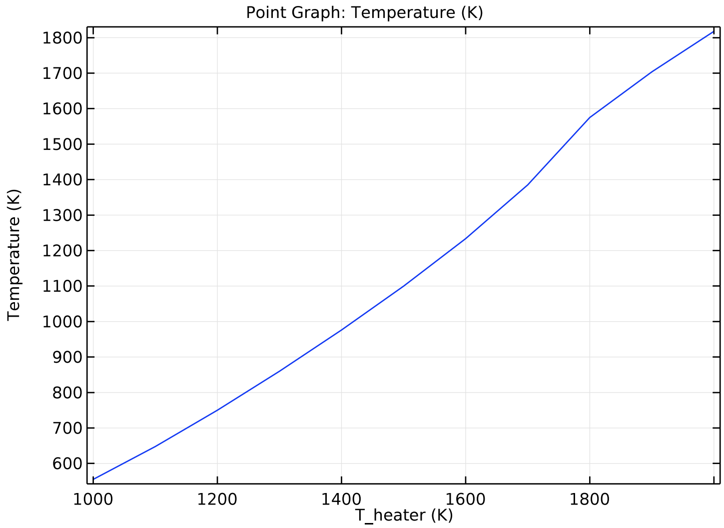


Figure 4: Temperature distribution in the TPV system when the emitter temperature is 2000 K.

As the upper plot in Figure 5 shows, the PV cells reach a temperature of approximately 1800 K. This is significantly higher than their maximum operating temperature of 1600 K, above which their photovoltaic efficiency is zero (see Figure 3).

It is interesting to investigate what the optimal operating temperature is. The lower plot in Figure 5 investigates at what temperature the system achieves the maximum electric power output. The optimal emitter temperature for this configuration seems to be between 1600 K and 1700 K, where the electric power (irradiation multiplied by voltaic efficiency) is maximum.



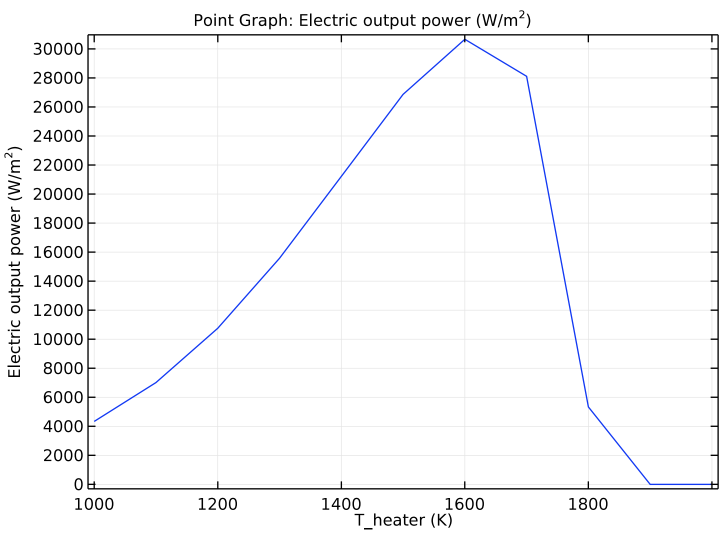


Figure 5: PV cell temperature (top) and electric output power (bottom) versus operating temperature.

The next step is to look at the temperature distribution at the optimal operating conditions (Figure 6).

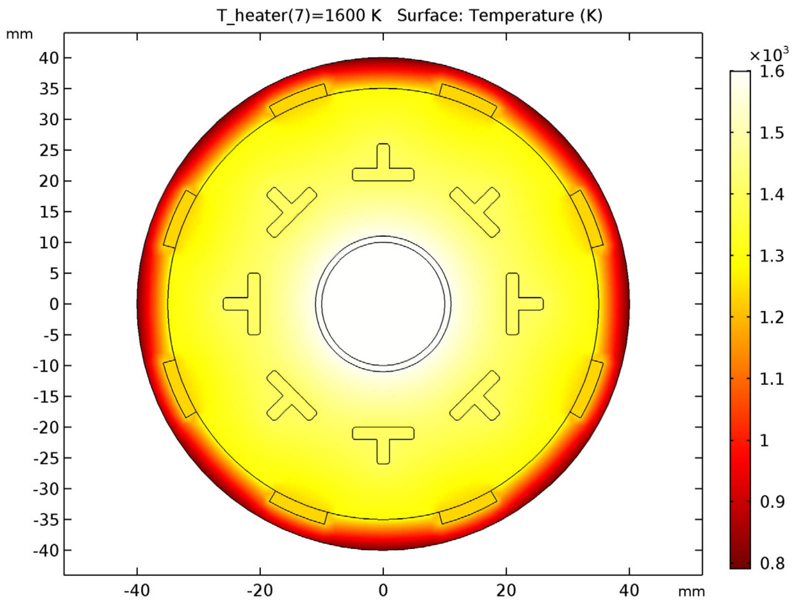


Figure 6: Temperature distribution and surface irradiation flux in the system at an operating emitter temperature of 1600 K.

When the emitter is at 1600 K, the PV cells reach a temperature of approximately 1200 K, which they can withstand without any problems. Note that the insulation reaches a temperature of approximately 800 K on the outside, suggesting that the system transfers a significant amount of heat to the surrounding air.

The plot also depicts the irradiative flux, which varies significantly along the circumference of the PV cell and insulation jacket. To further investigate this effect, Figure 7 plots the irradiative flux along a quarter of the circumference separately at this operating condition. Clearly the variation it shows is related to the positions of the mirrors and is an effect of shadowing.

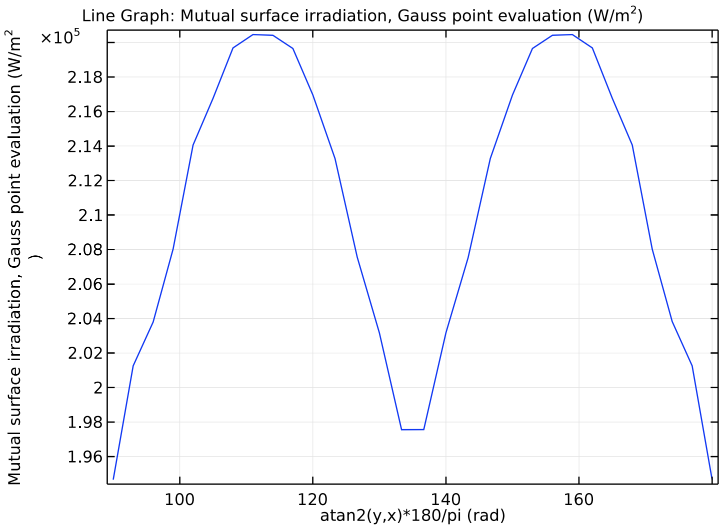


Figure 7: Irradiation flux along the TPV cell and insulation inner surface for one quarter of the device circumference.

This plot can help optimize the mirror geometry as well as help decide how large the PV cells should be and where they should be placed.

A general conclusion is that this type of modeling can shortcut the prototype development time and optimize the operating conditions for the finalized TPV device.

References

1. S. Christ and M. Seal, "*Viking 27—A Thermophotovoltaic Hybrid Vehicle Designed and Built at Western Washington University*", SAE Technical Paper 972650, 1997.

2. Courtesy of E. Fontes, Catella Generics AB, Sweden.

3. Courtesy of Dr. D. Wilhelm, Paul Sherrer Institute, Switzerland.

Application Library path: `Heat_Transfer_Module/Thermal_Radiation/tpv_cell`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Heat Transfer>Radiation>Heat Transfer with Surface-to-Surface Radiation (ht)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
T_heater	1000[K]	1000 K	Temperature, emitter inner boundary

GEOMETRY 1

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

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

3 From the **Length unit** list, choose **mm**.

Circle 1 (c1)

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

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

3 In the **Radius** text field, type 40.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	5
Layer 2	24
Layer 3	1

5 Click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **c1**, select Boundaries 1–12 only.

3 In the **Settings** window for **Delete Entities**, click **Build Selected**.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type 2.

4 In the **Height** text field, type 10.

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

6 In the **x** text field, type 21.

7 Click **Build Selected**.

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type 6.

4 In the **Height** text field, type 2.

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

6 In the **x** text field, type 23.

7 Click **Build Selected**.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **r1** and **r2** only.

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

4 Clear the **Keep interior boundaries** check box.

5 Click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **uni1**, select Points 1, 4, 5, and 8–10 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `0.5`.

5 Click **Build Selected**.

Circle 2 (c2)

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

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

3 In the **Radius** text field, type `37`.

4 In the **Sector angle** text field, type `360/24`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `360/24`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	2

7 Click **Build Selected**.

Delete Entities 2 (del2)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **c2**, select Domain 1 only.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **fil1** and **del2** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.

4 Click **Range**.

5 In the **Range** dialog box, type `0` in the **Start** text field.

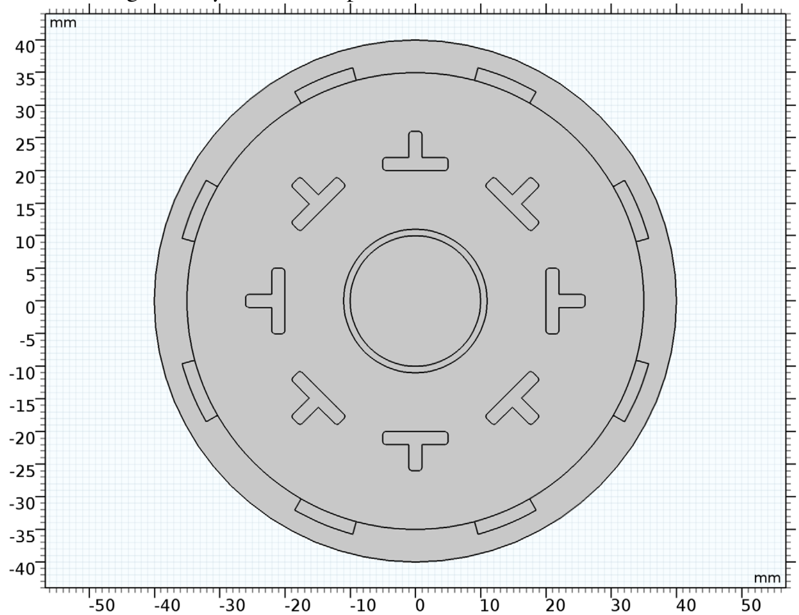
6 In the **Step** text field, type `360/8`.

7 In the **Stop** text field, type `360*7/8`.

8 Click **Replace**.

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

The model geometry is now complete.



DEFINITIONS

Next, create selections for the mirror domains and boundaries. These selections will be useful when you implement material settings and boundary conditions. You could create selections for other domain and boundary groups in the same manner but that is not assumed in the following instructions.

Explicit 1

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

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

3 Select Domains 5–7, 10, 11, and 14–16 only.

Adjacent 1

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

2 In the **Settings** window for **Adjacent**, type `Mirror Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Mirror Domains** in the **Input selections** list.

5 Click **OK**.

MATERIALS

Material 1 (mat1)

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

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

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

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	0.05	W/(m·K)	Basic
Density	rho	700	kg/m³	Basic
Heat capacity at constant pressure	Cp	100	J/(kg·K)	Basic

Material 2 (mat2)

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

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

3 Select Domains 2, 3, 8, 9, 12, 13, 17, and 18 only.

These are the PV-cell domains. Note that you can select these domains by copying the text in the modeling instructions and then clicking the **Paste Selection** button or pressing Ctrl+V. When you have selected the domains you can also create a named selection by clicking the **Create Selection** button.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	93	W/(m·K)	Basic
Density	rho	2000	kg/m³	Basic
Heat capacity at constant pressure	Cp	840	J/(kg·K)	Basic

Material 3 (mat3)

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Mirror Domains**.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	10	W/(m·K)	Basic
Density	rho	5000	kg/m³	Basic
Heat capacity at constant pressure	Cp	840	J/(kg·K)	Basic

Material 4 (mat4)

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

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

3 Select Domain 19 only.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	10	W/(m·K)	Basic
Density	rho	2000	kg/m³	Basic
Heat capacity at constant pressure	Cp	900	J/(kg·K)	Basic

ADD MATERIAL

1 On the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-In>Air**.

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

MATERIALS

Air (mat5)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat5)**.

2 Select Domains 4 and 20 only.

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

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

Fluid 1

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

2 Select Domain 4 only.

3 In the **Settings** window for **Fluid**, locate the **Thermodynamics, Fluid** section.

4 From the γ list, choose **User defined**.

Diffuse Surface 1

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

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

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

By default, the radiation direction is controlled by the opacity of the domains. The solid parts are automatically defined as opaque while the fluid parts are transparent. You can change these settings by modifying the **Opacity** subnode under the **Solid** and **Fluid** features.

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

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.01`.

Heat Flux 1

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

2 Select Boundaries 97, 98, 141, and 148 only.

These are the outer boundaries of the modeling domain.

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 `5`.

Diffuse Surface 2

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

2 Select Boundaries 97, 98, 141, and 148 only.

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

4 Clear the **Include surface-to-surface radiation** check box.

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.1`.

Diffuse Surface 3

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

2 Select Boundaries 101, 102, 105, 106, 133, 134, 142, 147, 167, 168, 183, and 184 only.

These are the arc-shaped boundaries connecting the PV cells.

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

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

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.1`.

Boundary Heat Source 1

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

2 Select Boundaries 99, 100, 117, 118, 157, 158, 181, and 182 only.

These are the outward-facing PV-cell boundaries.

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

4 In the Q_{b} text field, type `50[W/(m^2*K)]*(273.15[K]-T)`.

Diffuse Surface 4

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

2 Select Boundaries 103, 104, 123, 124, 155, 156, 179, and 180 only.

These are the inward-facing PV-cell boundaries.

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

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

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.99`.

Boundary Heat Source 2

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

2 Select Boundaries 103, 104, 123, 124, 155, 156, 179, and 180 only.

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

4 In the Q_{b} text field, type `-q_out`.

Diffuse Surface 5

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

2 Select Boundaries 127, 128, 143, and 146 only.

These are the outward-facing emitter boundaries.

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

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

5 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type 0.99.

Temperature 1

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

2 Select Boundaries 131, 132, 144, and 145 only.

These are the inward-facing emitter boundaries.

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

4 In the T_0 text field, type T_heater.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
eta_pv	if(T<1600[K],0.2*(1-(T/800[K]-1)^2),0)		Voltaic efficiency, PV cell
q_out	ht.Gm*eta_pv	W/m²	Electric output power

MESH 1

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

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

3 From the **Element size** list, choose **Coarser**.

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

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

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

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

4 From the **Selection** list, choose **Mirror Boundaries**.

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 Select the **Minimum element size** check box.

8 Select the **Maximum element growth rate** check box.

9 Select the **Curvature factor** check box.

10 In the **Maximum element size** text field, type 1.

Size 2

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

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

3 Click **Clear Selection**.

4 Select Boundaries 127, 128, 143, and 146 only.

Size 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

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

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

4 Select Boundaries 101–106, 123, 124, 133, 134, 142, 147, 155, 156, 167, 168, 179, 180, 183, and 184 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 Select the **Maximum element growth rate** check box.

8 In the **Maximum element size** text field, type 2.

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

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

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the parameter T_heater.

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

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

3 Click **Add**.

4 Click **Range**.

5 In the **Range** dialog box, type 1000 in the **Start** text field.

6 In the **Step** text field, type 100.

7 In the **Stop** text field, type 2000.

8 Click **Replace**.

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

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
T_heater	range(1000,100,2000)	K

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

Using constant prediction for the continuation sweep improves convergence when the solution is very nonlinear in the swept parameter.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

5 From the **Predictor** list, choose **Constant**.

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

RESULTS

Temperature (ht)

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

The first default surface plot shows the TPV-cell temperature for the last value in the sweep over operating temperatures.

Isothermal Contours (ht)

The second default plot shows isothermal contours.

Radiosity (ht)

The third default plot shows radiosity.

Reproduce the plots in Figure 5 with the following steps:

1D Plot Group 4

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `PV Cell Temperature` in the **Label** text field.
3. On the **PV Cell Temperature** toolbar, click **Point Graph**.

Point Graph 1

1. In the **Model Builder** window, under **Results>PV Cell Temperature** click **Point Graph 1**.
2. Select Point 6 only.
3. On the **PV Cell Temperature** toolbar, click **Plot**.

1D Plot Group 5

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Electric Output Power` in the **Label** text field.
3. On the **Electric Output Power** toolbar, click **Point Graph**.

Point Graph 1

1. In the **Model Builder** window, under **Results>Electric Output Power** click **Point Graph 1**.
2. Select Point 6 only.
3. In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Definitions>Variables>q_out - Electric output power**.
4. On the **Electric Output Power** toolbar, click **Plot**.

   As this last plot shows, the electric output power has a maximum near 1,600 K. To see the temperature distribution at this operating temperature, go back to the first plot group and change the parameter value.

Temperature (ht)

1. In the **Model Builder** window, under **Results** click **Temperature (ht)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Parameter value (T_heater (K))** list, choose **1600**.
4. On the **Temperature (ht)** toolbar, click **Plot**.

   Finally, reproduce the surface irradiation plot in Figure 7 as follows:

1D Plot Group 6

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

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

3 Locate the **Data** section. From the **Parameter selection (T_heater)** list, choose **From list**.

4 In the **Parameter values (T_heater (K))** list, select **1600**.

5 On the **Surface Irradiation** toolbar, click **Line Graph**.

Line Graph 1

1 In the **Model Builder** window, under **Results>Surface Irradiation** click **Line Graph 1**.

2 Select Boundaries 102, 104, 106, 124, and 134 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1> Heat Transfer with Surface-to-Surface Radiation>Radiation>Mutual surface irradiation> ht.Gm_gp - Mutual surface irradiation, Gauss point evaluation**.

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

5 In the **Expression** text field, type `atan2(y,x)*180/pi`.

6 Click to expand the **Quality** section. From the **Smoothing** list, choose **Everywhere**.

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

8 On the **Surface Irradiation** toolbar, click **Plot**.