Created in COMSOL Multiphysics 5.6

Free Convection in a Light Bulb

Introduction

This application simulates the nonisothermal flow of argon gas inside a light bulb. The purpose of the model is to show the coupling between energy transport — through conduction, radiation, and convection — and momentum transport induced by density variations in the argon gas.

All three forms of heat transfer are taken into account. First, you have conduction, when a 60 W filament is heated thus transferring heat from the heat source to the light bulb. Then there is convection, which drives a flow inside the bulb transferring the heat from the filament throughout the bulb via the movement of fluids (in this case, argon gas). Finally, there is the radiation portion of the problem, and in this case that includes surface-to-surface and surface-to-ambient radiation. The Heat Transfer Module includes both of these types of radiation, so that you can account for shading and reflections between radiating surfaces, as well as ambient radiation that can be fixed or given by an arbitrary function. The light bulb physics involves both heat transfer and gas flow, which makes this a multiphysics problem and not "just" a heat transfer example.

Note: This application requires the Heat Transfer Module and the Material Library.

Model Definition

A light bulb contains a tungsten filament that is resistively heated when a current is conducted through it. At temperatures around 2000 K the filament starts to emit visible light. To prevent the tungsten wire from burning up, the bulb is filled with a gas, usually argon. The heat generated in the filament is transported to the surroundings through radiation, convection, and conduction. As the gas heats up, density and pressure changes induce a flow inside the bulb.

Figure 1 shows a cross section of the axially symmetric model geometry.

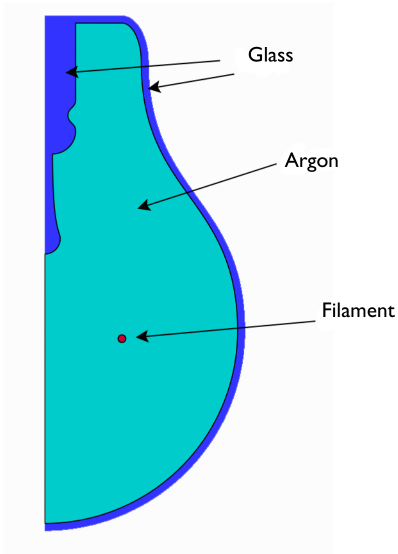

Figure 1: The model geometry.

The filament is approximated with a solid torus, an approximation that implies neglecting any internal effects inside the filament wire.

The equations governing the nonisothermal flow are the Navier-Stokes equations with the gravity forces (see *Gravity* in the *CFD Module User's Guide*). The density is given by the ideal gas law

$$\rho = \frac{Mp}{RT}$$

where M denotes the molar weight (kg/mol), R the universal gas constant (J/(mol·K)), and T the temperature (K).

The convective and conductive heat transfer are modeled using the heat transfer interface and account for the total light bulb power equal to 60 W.

BOUNDARY CONDITIONS

At the bulb's inner surfaces, radiation is described by surface-to-surface radiation. This means that the mutual irradiation from the surfaces that can be seen from a particular surface and radiation to the surroundings are accounted for. At the outer surfaces of the bulb, radiation is described by surface-to-ambient radiation, which means that there is no reflected radiation from the surroundings (blackbody radiation).

The top part of the bulb, where the bulb is mounted on the cap, is insulated:

$$-\mathbf{n} \cdot (-k\nabla T) = 0$$

Results

The heating inside the bulb has a long and a short time scale from $t = 0$, when the light is turned on. The shorter scale captures the heating of the filament and the gas close to it. The following series of pictures shows the temperature distribution inside the bulb at $t = 2$, 6, and 10 s.

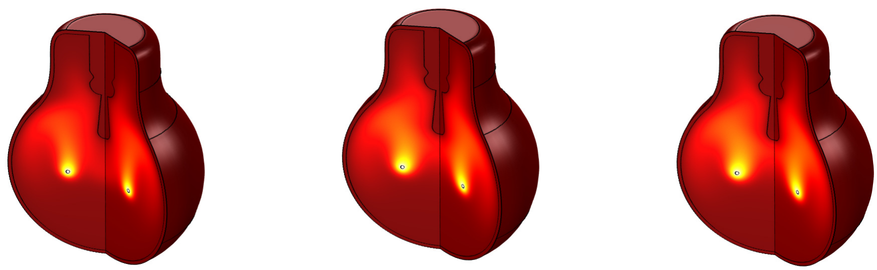

Figure 2: Temperature distribution at t = 2, 6, and 10 s. The color ranges differ between the plots.

When the temperature changes, the density of the gas changes, inducing a gas flow inside the bulb. The following series of pictures shows the velocity field inside the bulb after 2, 6, and 10 s.

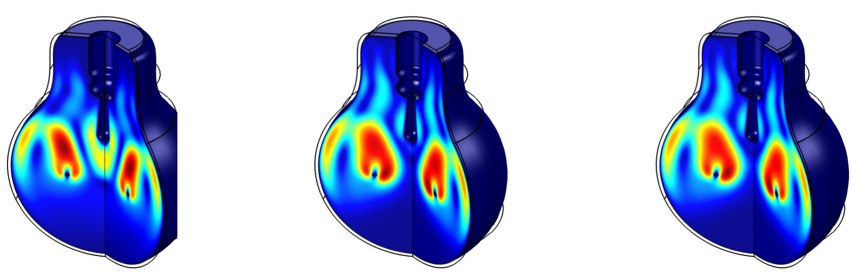

Figure 3: Velocity field after 2, 6, and 10 s. The color ranges differ between the plots.

On the longer time scale, the glass on the bulb's outer side heats up. The following plot shows the temperature distribution in the bulb after 5 minutes.

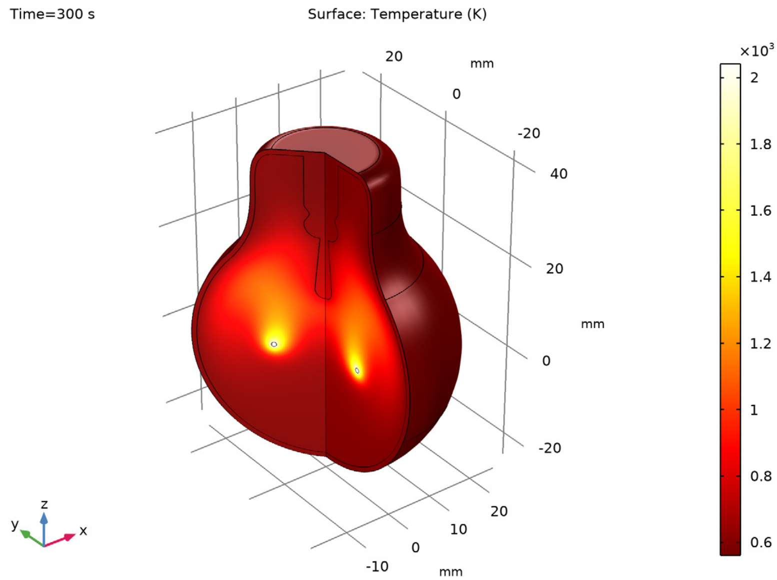

Figure 4: Temperature distribution after 5 minutes.

Figure 5 shows the temperature distribution at a point on the boundary of the bulb at the same vertical level as the filament. This plot shows the slow heating of the bulb. After 5 minutes, the bulb has reached a steady-state temperature of approximately 589 K.

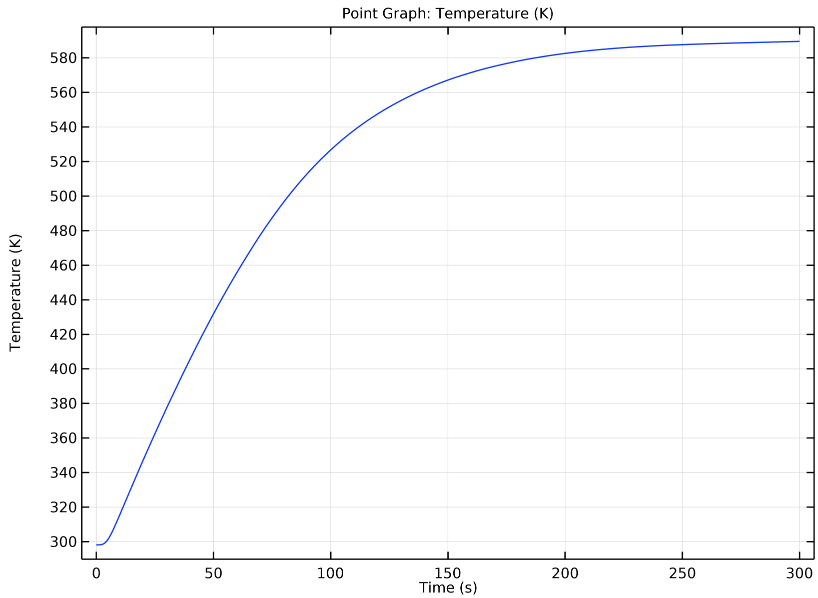

Figure 5: Temperature distribution at a point on the boundary of the bulb at the same vertical level as the filament.

Heat is transported from the boundary of the bulb through both convective heat flux and radiation. The net radiative heat flux leaving the bulb at $t = 300$ s is plotted in Figure 6, as function of the z-coordinate. The top boundaries of the bulb where the bulb is mounted on the cap are excluded from this plot. The distinct bump in the curve occurs around $z = 1.5$ cm, above the filament.

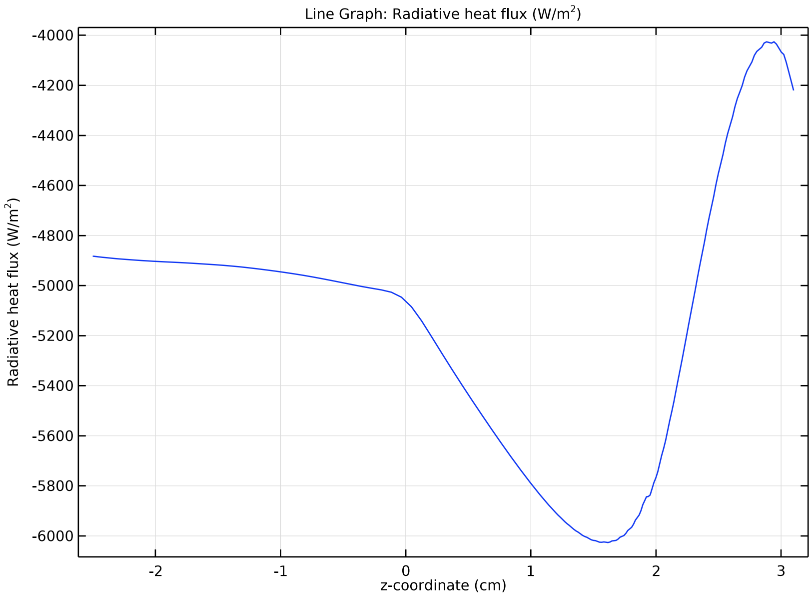

Figure 6: The net radiative heat flux leaving the bulb.

Notes About the COMSOL Implementation

To set up the model, use the Heat Transfer Module's Conjugate Heat Transfer predefined multiphysics coupling. The model uses material from the Material Library to accurately account for temperature-dependent properties over a wide range. The model setup is straightforward and also shows how to create your own material to treat argon as an ideal gas. When working with surface-to-surface radiation in COMSOL, fluid domains are considered as transparent and solid domains as opaque by default, which are the expected properties for this model. The assumption that the glass on the bulb is opaque might seem odd, but it is valid because glass is almost opaque to heat radiation but transparent to radiation in the visible spectrum.

Application Library path: `Heat_Transfer_Module/Thermal_Radiation/light_bulb`

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

NEW

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

MODEL WIZARD

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

2 In the **Select Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer>Radiation>Surface-to-Surface Radiation (rad)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 Click **Done**.

DEFINITIONS

Ambient Properties 1 (ampr1)

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

2 In the **Settings** window for **Ambient Properties**, locate the **Ambient Conditions** section.

3 In the T_{amb} text field, type `25[degC]`.

GEOMETRY 1

The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `light_bulb_geom_sequence.mph`.

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

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

You should now see the geometry shown in Figure 1.

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
h0	5[W/(m^2*K)]	5 W/(m²·K)	Heat transfer coefficient
Qf	60[W]	60 W	Heat source in filament
p0	50[kPa]	50000 Pa	Initial pressure
rho_glass	2595[kg/m^3]	2595 kg/m³	Density, glass
k_glass	1.09[W/(m*K)]	1.09 W/(m·K)	Thermal conductivity, glass
Cp_glass	750[J/(kg*K)]	750 J/(kg·K)	Heat capacity, glass
eps_glass	0.8	0.8	Surface emissivity, glass
Mw_a	39.94[g/mol]	0.03994 kg/mol	Molar mass, argon

ADD MATERIAL

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

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

3 In the tree, select **Material Library>Elements>Tungsten>Tungsten [solid]>Tungsten [solid, Ho et al]**.

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

MATERIALS

Tungsten [solid,Ho et al] (mat1)

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

2 From the **Selection** list, choose **Tungsten**.

To apply the surface emissivity for tungsten as a material property, you also need to define tungsten as the material for the filament surface.

ADD MATERIAL

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

2 In the tree, select **Material Library>Elements>Tungsten>Tungsten [solid]>Tungsten [solid, Ho et al]**.

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

4 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Tungsten [solid,Ho et al] 1 (mat2)

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

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

3 From the **Selection** list, choose **Tungsten Boundary**.

Glass

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

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

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

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	`k_glass`	W/(m·K)	Basic
Density	rho	`rho_glass`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`Cp_glass`	J/(kg·K)	Basic

Now, set up the physics to let COMSOL Multiphysics flag what properties you need to specify manually.

LAMINAR FLOW (SPF)

As the flow is driven by buoyancy, gravity matters.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 Select the **Include gravity** check box.

Since the density variation is not small, the flow cannot be regarded as incompressible. Therefore set the flow to be compressible.

4 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

5 Locate the **Domain Selection** section. From the **Selection** list, choose **Argon**.

Define the pressure reference level in the interface properties.

6 Locate the **Physical Model** section. In the p_{ref} text field, type `p0`.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>** **Heat Transfer in Solids and Fluids (ht)** click **Fluid 1**.

2 In the **Settings** window for **Fluid**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Argon**.

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 From the T list, choose **Ambient temperature (ampr1)**.

Heat Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Tungsten**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `Qf`.

Heat Flux 1

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

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

3 From the **Selection** list, choose **Exterior Radiation**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `h0`.

6 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

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 **Boundary Selection** section.

3 From the **Selection** list, choose **Radiation**.

Diffuse Surface 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Surface-to-Surface Radiation (rad)** click **Diffuse Surface 1**.
2 In the **Settings** window for **Diffuse Surface**, locate the **Ambient** section.
3 Find the **Ambient temperature** subsection. From the T_{amb} list, choose **Ambient temperature (ampr1)**.

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. For this model, the default settings apply.

ADD MULTIPHYSICS

1 In the **Physics** toolbar, click **Add Multiphysics** to open the **Add Multiphysics** window.
2 Go to the **Add Multiphysics** window.
3 Find the **Select the physics interfaces you want to couple** subsection. In the table, clear the **Couple** check box for **Laminar Flow (spf)**.
4 In the tree, select **Heat Transfer>Radiation>Heat Transfer with Surface-to-Surface Radiation**.
5 Click **Add to Component** in the window toolbar.
6 In the **Physics** toolbar, click **Add Multiphysics** to close the **Add Multiphysics** window.

MATERIALS

Glass (Boundaries)

1 In the **Materials** toolbar, click **Blank Material**.
2 In the **Settings** window for **Material**, type `Glass (Boundaries)` 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 **Glass Boundaries**.
5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Surface emissivity	epsilon_rad	eps_glass	1	Basic

ADD MATERIAL

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

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

3 In the tree, select **Material Library>Elements>Argon>Argon [gas]**.

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

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Argon [gas] (mat5)

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

2 From the **Selection** list, choose **Argon**.

As you can see, COMSOL Multiphysics warns about required properties that have not been defined yet. Define these as follows.

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

Property	Variable	Value	Unit	Property group
Ratio of specific heats	gamma	`1.6`	1	Basic
Density	rho	`ht.pA*Mw_a/(R_const*T)`	kg/m³	Basic

MESH 1

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

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

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

4 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.1,1) range(1.5,0.5,20) range(21,3,300)`.

4 In the **Home** toolbar, click **Compute**.

RESULTS

Temperature, 3D (ht)

The first default 3D plot shows the temperature at the end of the simulation interval (Figure 4). Look at the temperature field at different times and compare the resulting series of plots with those in Figure 2.

1 Click the Zoom Extents button in the **Graphics** toolbar.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Time (s)** list, choose **2**.
4 In the **Temperature, 3D (ht)** toolbar, click **Plot**.

   Compare with the left panel in Figure 2.
5 From the **Time (s)** list, choose **6**.
6 In the **Temperature, 3D (ht)** toolbar, click **Plot**.

   Compare with the middle panel in Figure 2.
7 From the **Time (s)** list, choose **10**.
8 In the **Temperature, 3D (ht)** toolbar, click **Plot**.

   Compare with the right panel in Figure 2.

Pressure (spf)

This default plot shows the pressure field in a 2D contour plot. Change the unit to kPa as follows.

Contour

1 In the **Model Builder** window, expand the **Pressure (spf)** node, then click **Contour**.
2 In the **Settings** window for **Contour**, locate the **Expression** section.
3 From the **Unit** list, choose **kPa**.
4 In the **Pressure (spf)** toolbar, click **Plot**.

Velocity, 3D (spf)

This default plot shows the velocity magnitude in a 3D plot, obtained by a revolution of the 2D axisymmetric dataset, at the end of the simulation interval. Now proceed to reproduce the velocity field plots in Figure 3.

Surface

Because the velocity magnitude is a quadratic expression in the basic velocity variables it looks less smooth than the temperature plot. You can easily remedy the situation by adjusting the Quality settings.

1 In the **Model Builder** window, expand the **Velocity, 3D (spf)** node, then click **Surface**.
2 In the **Settings** window for **Surface**, click to expand the **Quality** section.
3 From the **Resolution** list, choose **Fine**.
4 In the **Velocity, 3D (spf)** toolbar, click **Plot**. This ensures that the resolution is sufficient.

Velocity, 3D (spf)

1 In the **Model Builder** window, click **Velocity, 3D (spf)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Time (s)** list, choose **2**.
4 In the **Velocity, 3D (spf)** toolbar, click **Plot**.

Compare with the left panel in Figure 3.

5 From the **Time (s)** list, choose **6**.
6 In the **Velocity, 3D (spf)** toolbar, click **Plot**.

Compare with the middle panel in Figure 3.

7 From the **Time (s)** list, choose **10**.
8 In the **Velocity, 3D (spf)** toolbar, click **Plot**.

Compare with the right panel in Figure 3.

To visualize the heating of the bulb surface with time by plotting the temperature at a point at the same vertical level as the filament, follow the steps below.

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.
2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.
3 In the **r** text field, type `26`.
4 In the **z** text field, type `1`.

Temperature vs. Time

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

Point Graph 1

1 Right-click **Temperature vs. Time** and choose **Point Graph**.
2 In the **Settings** window for **Point Graph**, locate the **Data** section.
3 From the **Dataset** list, choose **Cut Point 2D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids and Fluids>Temperature>T - Temperature - K**.

5 In the **Temperature vs. Time** toolbar, click **Plot**.

Finally, study the radiative heat flux from the bulb. First plot the radiative heat flux versus the vertical coordinate, z.

Radiative Heat Flux along z-Coordinate

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

2 In the **Settings** window for **1D Plot Group**, type `Radiative Heat Flux along z-Coordinate` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **Last**.

Line Graph 1

1 In the **Radiative Heat Flux along z-Coordinate** toolbar, click **Line Graph**.

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

3 From the **Selection** list, choose **Exterior Radiation**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Surface-to-Surface Radiation>Radiative heat flux> rad.rflux - Radiative heat flux - W/m²**.

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Geometry>Coordinate>z - z-coordinate**.

6 Locate the **x-Axis Data** section. From the **Unit** list, choose **cm**.

7 In the **Radiative Heat Flux along z-Coordinate** toolbar, click **Plot**.

You can readily compute the total radiative heat flux from the bulb at steady state as follows.

Line Integration 1

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

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

3 From the **Time selection** list, choose **Last**.

4 Locate the **Selection** section. From the **Selection** list, choose **Exterior Radiation**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Surface-to-Surface Radiation>Radiative heat flux> rad.rflux - Radiative heat flux - W/m²**.

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The result should be close to 45 W.

Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

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**.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

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

r (mm)	z (mm)
0	-25
0	42
10	42

Polygon 2 (pol2)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

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

r (mm)	z (mm)
4	31
4	41
10	41

Interpolation Curve 1 (ic1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Interpolation Curve**.

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

3 In the table, enter the following settings:

r (mm)	z (mm)
4	31
3	29
4	27

4 Locate the **End Conditions** section. From the **Condition at starting point** list, choose **Tangent direction**.

5 In the **r** text field, type 0.

6 In the **z** text field, type -1.

7 From the **Condition at endpoint** list, choose **Tangent direction**.

8 In the **r** text field, type 0.

9 In the **z** text field, type -1.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type 3.

5 In the **Sector angle** text field, type 90.

6 Locate the **Position** section. In the **r** text field, type 1.

7 In the **z** text field, type 27.

8 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.

Circle 2 (c2)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type 2.

5 In the **Sector angle** text field, type 90.

6 Locate the **Position** section. In the **z** text field, type 13.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **c1**, select Point 1 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Find the **End vertex** subsection. Select the **Activate Selection** toggle button.

5 On the object **c2**, select Point 2 only.

Circle 3 (c3)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `26`.

5 In the **Sector angle** text field, type `135`.

6 Locate the **Position** section. In the **z** text field, type `1`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

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

Layer name	Thickness (mm)
Layer 1	1

Cubic Bézier 1 (cb1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Cubic Bézier**.

2 In the **Settings** window for **Cubic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `10` .

4 In row **2**, set **r** to `18`.

5 In row **3**, set **r** to `10`.

6 In row **4**, set **r** to `13*sqrt(2)`.

7 In row **1**, set **z** to `42`.

8 In row **2**, set **z** to `41`.

9 In row **3**, set **z** to `29`.

10 In row **4**, set **z** to `13*sqrt(2)+1`.

Cubic Bézier 2 (cb2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Cubic Bézier**.

2 In the **Settings** window for **Cubic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `10` .

4 In row **2**, set **r** to `18`.

5 In row **3**, set **r** to `9`.

6 In row **4**, set **r** to `12.5*sqrt(2)`.

7 In row **1**, set **z** to `41`.

8 In row **2**, set **z** to `40`.

9 In row **3**, set **z** to `29`.

10 In row **4**, set **z** to `12.5*sqrt(2)+1`.

11 Locate the **Weights** section. In the **2** text field, type `3/4`.

Partition Edges 1 (pare1)

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

2 On the object **cb1**, select Boundary 1 only.

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

4 In the table, enter the following settings:

Relative arc length parameters
0.509

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

Delete Entities 1 (del1)

1 In the **Geometry** toolbar, click **Delete**.

2 On the object **c1**, select Boundaries 2 and 3 only.

3 On the object **c2**, select Boundaries 2 and 3 only.

4 On the object **c3**, select Boundaries 1–4 only.

Circle 4 (c4)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `0.5`.

5 Locate the **Position** section. In the **r** text field, type `10`.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

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

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Argon

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 On the object **fin**, select Domain 2 only.

Glass

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 On the object **fin**, select Domain 1 only.

Tungsten

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 On the object **fin**, select Domain 3 only.

Interior Radiation

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Interior Radiation` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** check box.

5 On the object **fin**, select Boundaries 5–7, 9–16, 18, and 21 only.

Exterior Radiation

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Exterior Radiation` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 8, 17, and 20 only.

Radiation

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `Radiation` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Interior Radiation** and **Exterior Radiation**.

6 Click **OK**.

Glass Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

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

3 Locate the **Input Entities** section. Click **Add**.

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

5 Click **OK**.

Tungsten Boundary

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Tungsten Boundary` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

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

5 Click **OK**.