Model created in COMSOL Multiphysics 6.4

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.

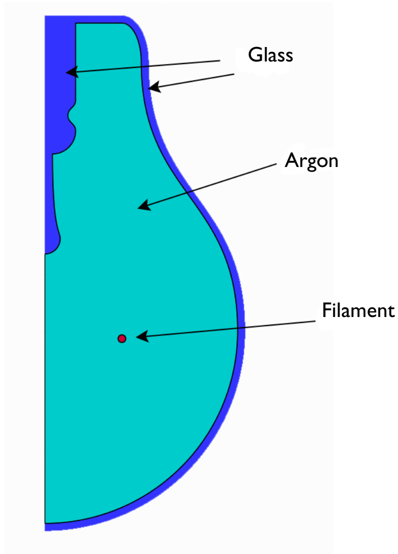


Figure 1: The model geometry.

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 *COMSOL Multiphysics Reference Manual*). 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.

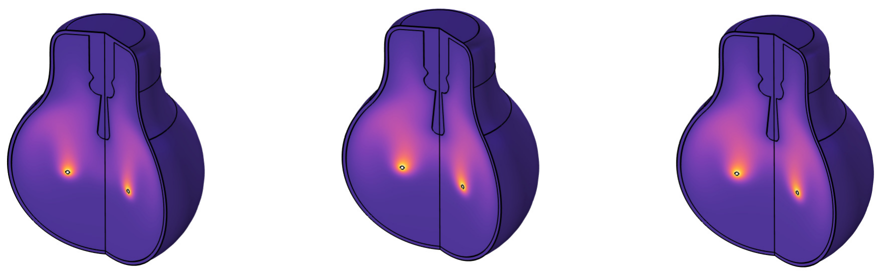


Figure 2: Temperature distribution at t = 2, 6, and 10 s. The color ranges differ between the plots.

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.

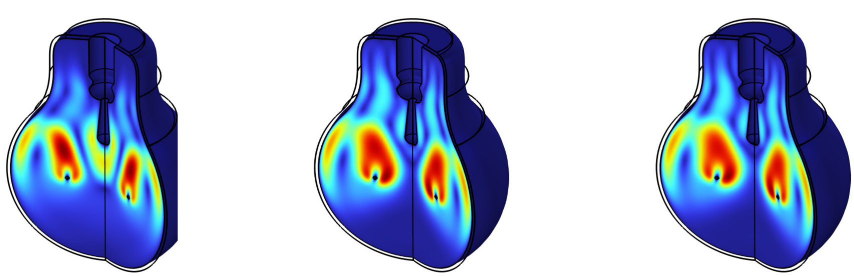


Figure 3: Velocity field after 2, 6, and 10 s. The color ranges differ between the plots.

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.

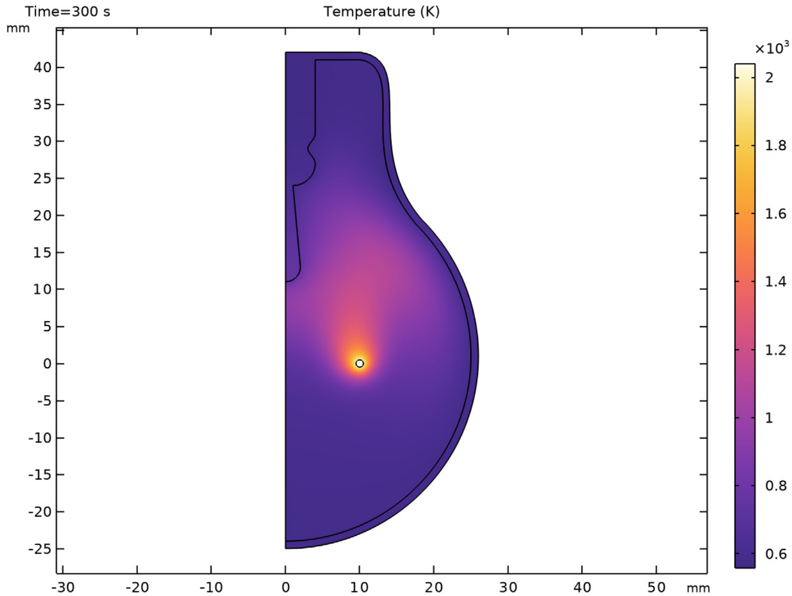


Figure 4: Temperature distribution after 5 minutes.

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.

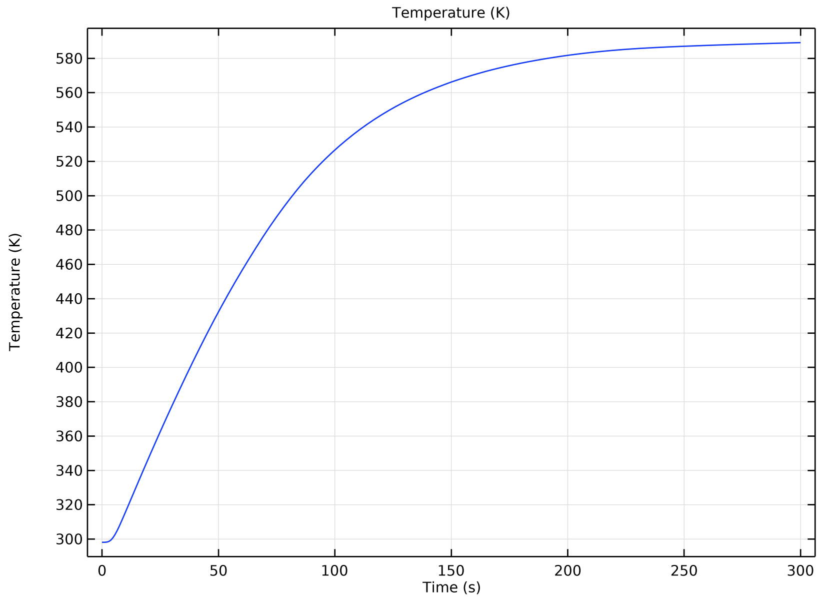


Figure 5: Temperature distribution at a point on the boundary of the bulb at the same vertical level as the filament.

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.

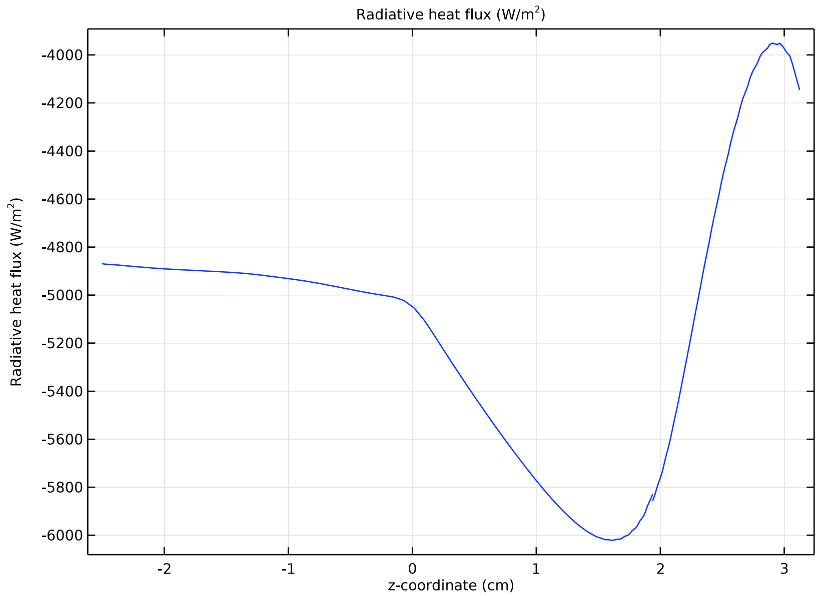


Figure 6: The net radiative heat flux leaving the bulb.

Notes About the COMSOL Implementation

To set up the model, use the Conjugate Heat Transfer predefined multiphysics coupling of the Heat Transfer Module. The model uses a 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 tutorial under `applications/COMSOL_Multiphysics/Geometry_Tutorials`. Otherwise, insert the geometry sequence as follows:

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

The imported sequence contains all required selections in addition to the actual geometry. Selections facilitate the work of assigning materials, setting boundary conditions, and plot the results.

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 **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** > **Tungsten** > **Tungsten [solid]** > **Tungsten [solid,Ho et al.]**.

4 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

Tungsten [solid,Ho et al.] (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Tungsten [solid,Ho et al.] (mat1)**.

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

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

4 In the **Materials** 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**.

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** checkbox.

   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. From the **Heat source** list, choose **Heat rate**.
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. From the **Flux type** list, choose **Convective heat flux**.
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 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 this setting using the **Opacity** feature in the **Surface-to-Surface Radiation** interface. 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** checkbox for **Laminar Flow (spf)**.

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

5 Click the **Add to Component** button 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]** > **Argon [gas, at 101 kPa (14.7 psi)]**.

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

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

MATERIALS

Argon [gas,at 101 kPa (14.7 psi)] (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
Density	rho	`spf.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 **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot shows the temperature on a 2D slice at the end of the simulation interval (Figure 4). You can add a predefined 3D version of the temperature plot and change the displayed time. Compare the resulting series of plots with those in Figure 2.

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

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Solids and Fluids** > **Temperature (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Temperature (ht) 1

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Time (s)** list, choose **2**.

3 In the **Temperature (ht) 1** toolbar, click **Plot**.

Compare with the left panel in Figure 2.

4 From the **Time (s)** list, choose **6**.

5 In the **Temperature (ht) 1** toolbar, click **Plot**.

Compare with the middle panel in Figure 2.

6 From the **Time (s)** list, choose **10**.

7 In the **Temperature (ht) 1** toolbar, click **Plot**.

Compare with the right panel in Figure 2.

Pressure (spf)

This default plot shows the pressure field in a surface plot. Change the unit to kPa and the color table type as follows.

Surface 1

1 In the **Model Builder** window, expand the **Pressure (spf)** node, then click **Surface 1**.

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

3 From the **Unit** list, choose **kPa**.

4 Locate the **Coloring and Style** section. From the **Color table type** list, choose **Continuous**.

5 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 **Results** toolbar, click **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 8.85 e-12 **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

1 Go to the **Table 1** window.

The result should be close to 44 W.

Geometry Modeling Instructions

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

NEW

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

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D Axisymmetric**.

GEOMETRY 1

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

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

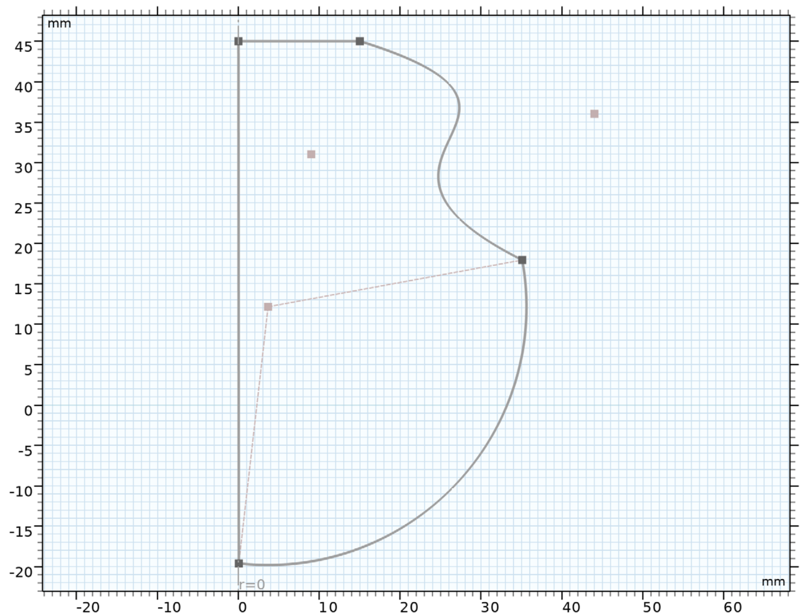
Begin by drawing a rough outline of the bulb. Do not worry about getting it exactly right as you will adjust it later.

The dimensions of the light bulb are larger than the default zoom level in the **Graphics** window. Adjusting the shape is easier if the original sketch is drawn closer to the final size.

3 Zoom out a few steps until the size of the canvas corresponds to the next image.

Composite Curve 1 (cc1)

Draw a shape similar to the figure below, starting from the top left corner and continuing clockwise.



Dark gray elements indicate geometrical objects, whereas light gray entities represent control points for higher-order polygons or center points of circular arcs.

1 In the **Sketch** toolbar, click **Polygon**, then in the **Graphics** window place the first vertex by clicking on the centerline close to the top of the canvas.

2 Move the pointer to the right, and at the end of the first horizontal segment click once to place a vertex.

3 To switch drawing a Cubic Bézier polygon, right-click and from the context menu choose **Cubic**.

4 Place the top right and then the lower left control point of the Bézier curve, followed by the vertex at the end by clicking once on the canvas for each point.

5 To switch drawing a circular arc, right-click in the **Graphics** window, and from the context menu choose **Circular Arc**, then choose **Start, Center, Angle**.

6 Place the center of the arc on the centerline, then move the pointer to draw the arc, and click to place the end vertex so that the arc finishes at the centerline.

7 Right-click, then from the context menu choose **Polygon**.

8 To close the shape, position the pointer on top of the first vertex, then click to place the last vertex. The shape will be closed automatically.

When done, the **Composite Curve 1** node is added to the geometry sequence. This node contains the polygon, cubic Bézier, and circular arc features that you have drawn. Note that the two adjacent straight segments are automatically combined into one feature.

Composite Curve 1 (cc1)

Next, adjust the features inside **Composite Curve 1** to obtain the outer shape of the light bulb.

Polygon 1 (pol1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Geometry 1** > **Composite Curve 1 (cc1)** node, then click **Polygon 1 (pol1)**.

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

3 In the table, enter the following settings:

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

When editing the coordinates of the features in a **Composite Curve**, the adjacent features are automatically updated to keep the start and end points of adjacent edges coincident.

Cubic Bézier 1 (cb1)

1 In the **Model Builder** window, click **Cubic Bézier 1 (cb1)**.

2 Since the coordinates of the first control point have already been adjusted by editing **pol1** change the remaining entries, only.

3 In the table, enter the following settings:

	r	z
2:	18	41
3:	10	29
4:	13*sqrt(2)	13*sqrt(2)+1

Circular Arc 1 (ca1)

1 In the **Model Builder** window, click **Circular Arc 1 (ca1)**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

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

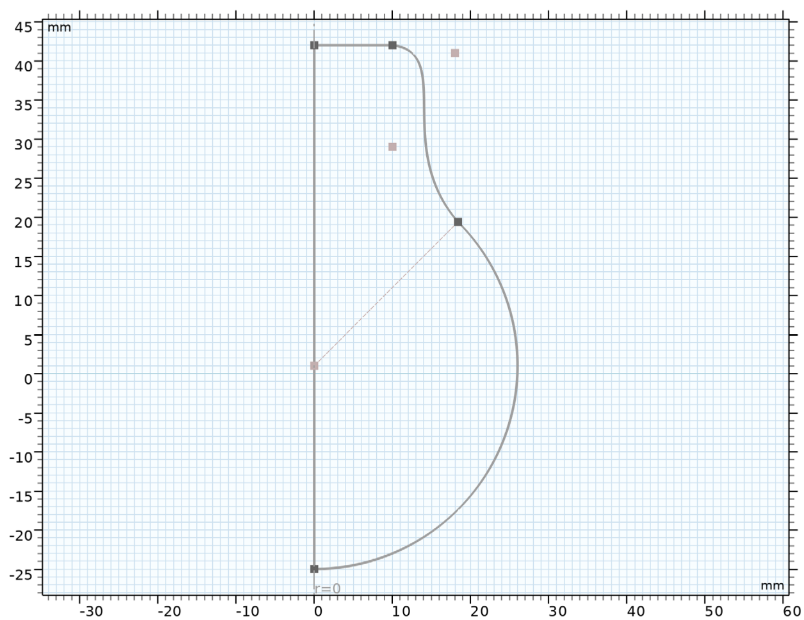
4 In the **z** text field, type 1.

5 Locate the **Radius** section. In the **Radius** text field, type 26.

6 Locate the **Angles** section. In the **Start angle** text field, type 45.

7 In the **End angle** text field, type -90.

8 Click **Build All Objects**.



Composite Curve 1 (cc1)

1 In the **Model Builder** window, click **Composite Curve 1 (cc1)**.

2 In the **Settings** window for **Composite Curve**, locate the **Selections of Resulting Entities** section.

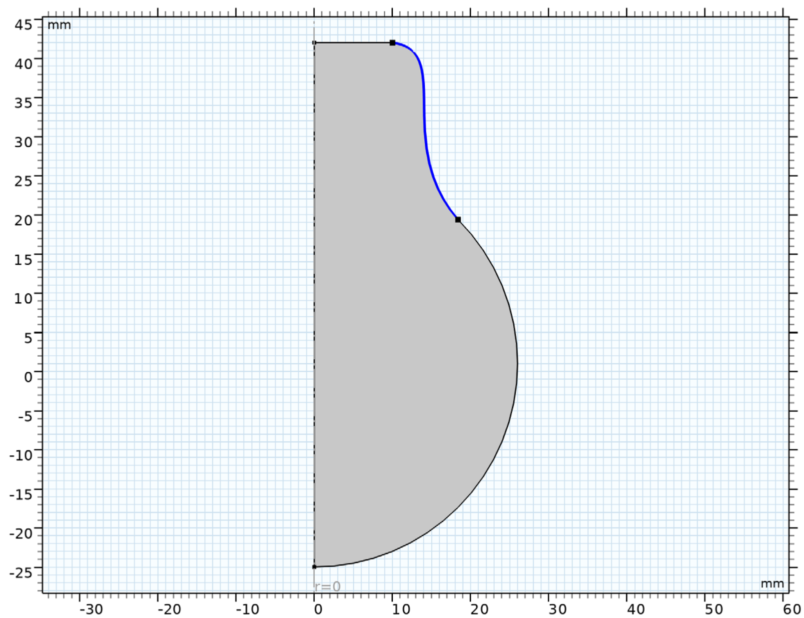
3 Select the **Resulting objects selection** checkbox.

4 From the **Show in physics** list, choose **Off**. With this setting the selection is available only as input for features in the geometry sequence. This way you can keep only the relevant selections in the list of selections when you are defining, for example, physics and mesh features.

Partition Edges 1 (pare1)

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

2 On the object **cc1**, select Boundary 3 only.



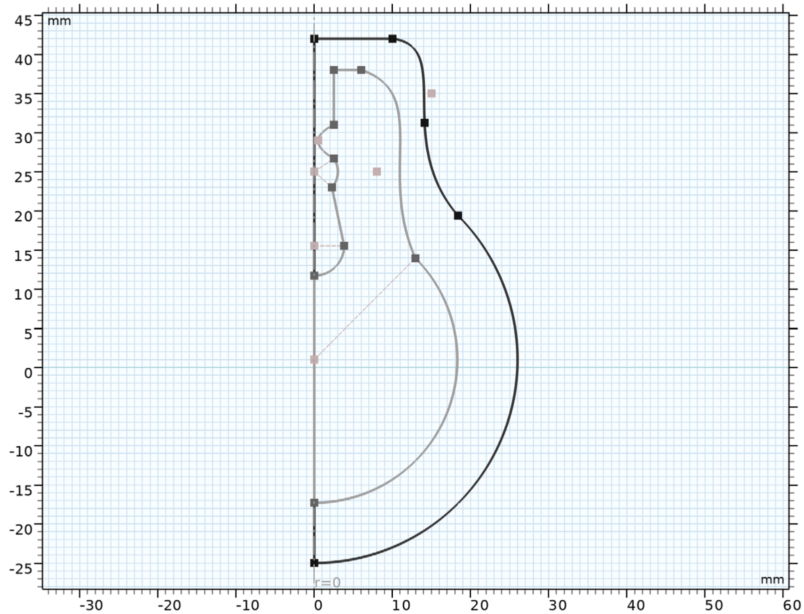
The partitioning operations can be useful in many cases. Here, we are partitioning the selected edge to create segments that reflect that a portion of the upper boundaries of the bulb is covered by a cap.

3 In the **Settings** window for **Partition Edges**, click **Build Selected**.

Composite Curve 2 (cc2)

Continue with creating the interior boundaries. Draw a rough outline by starting again from the top left corner, then continuing clockwise.

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Geometry 1** > **Composite Curve 2 (cc2)** node, then click **Polygon 1 (pol1)**.



2 In the **Sketch** toolbar, click **Sketch**.

Use the drawing tools in the following order:

3 Right-click in the **Graphics** window and select **Polygon**. Start to draw an edge perpendicular to the rotation axis. Its first vertex is located inward from the start vertex of the outer shape.

4 Continue with a **Cubic** Bézier polygon. Try to follow the outer shape.

5 Add a **Circular Arc** that ends on the centerline.

6 Draw a **Polygon** up along the centerline to about halfway up the geometry.

7 Continue with a **Circular Arc** that curves away from the centerline.

8 Use the **Polygon** tool to draw an edge that tilts toward the centerline.

9 Draw another **Circular Arc** that curves away from then back toward the centerline. The start and end vertices can be aligned vertically.

10 Switch to an **Interpolation Curve** to create a curved segment that first curves toward the centerline then away. Use the **Interpolation Points** option to define the curve, and add one interpolation point. Try to align the start and end vertices vertically.

11 Close the shape with a vertical edge, using the **Polygon** tool.

Continue with editing the features inside **Composite Curve 2**.

12 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

13 In the table, enter the following settings:

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

Cubic Bézier 1 (cb1)

1 In the **Model Builder** window, click **Cubic Bézier 1 (cb1)**.

2 Since the coordinates of the first control point have already been adjusted by editing **pol1** change the remaining entries, only.

3 In the table, enter the following settings:

	r	z
`2:`	`18`	`40`
`3:`	`9`	`29`
`4:`	`12.5*sqrt(2)`	`12.5*sqrt(2)+1`

4 In the **Settings** window for **Cubic Bézier**, locate the **Weights** section.

5 In the **2** text field, type `3/4`.

Circular Arc 1 (ca1)

1 In the **Model Builder** window, click **Circular Arc 1 (ca1)**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

3 In the **r** text field, type `0`.

4 Locate the **Radius** section. In the **Radius** text field, type `25`.

5 Locate the **Angles** section. In the **Start angle** text field, type `45`.

Polygon 2 (pol2)

1 In the **Model Builder** window, click **Polygon 2 (pol2)**.

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

3 Change the second coordinate only.

r (mm)	z (mm)
0	11

Circular Arc 2 (ca2)

1 In the **Model Builder** window, click **Circular Arc 2 (ca2)**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

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

4 In the **z** text field, type 13.

5 Locate the **Radius** section. In the **Radius** text field, type 2.

6 Locate the **Angles** section. In the **Start angle** text field, type 270.

Polygon 3 (pol3)

1 In the **Model Builder** window, click **Polygon 3 (pol3)**.

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

3 Change the second coordinate only.

r (mm)	z (mm)
1	24

Circular Arc 3 (ca3)

1 In the **Model Builder** window, click **Circular Arc 3 (ca3)**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

3 In the **r** text field, type 1.

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

5 Locate the **Radius** section. In the **Radius** text field, type 3.

6 Locate the **Angles** section. In the **Start angle** text field, type -90.

7 In the **End angle** text field, type 0.

Interpolation Curve 1 (ic1)

1 In the **Model Builder** window, click **Interpolation Curve 1 (ic1)**.

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

3 Change the radial part for the second and third coordinates.

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

The tangent of the curve at the starting point and endpoint follows the *z* direction.

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.

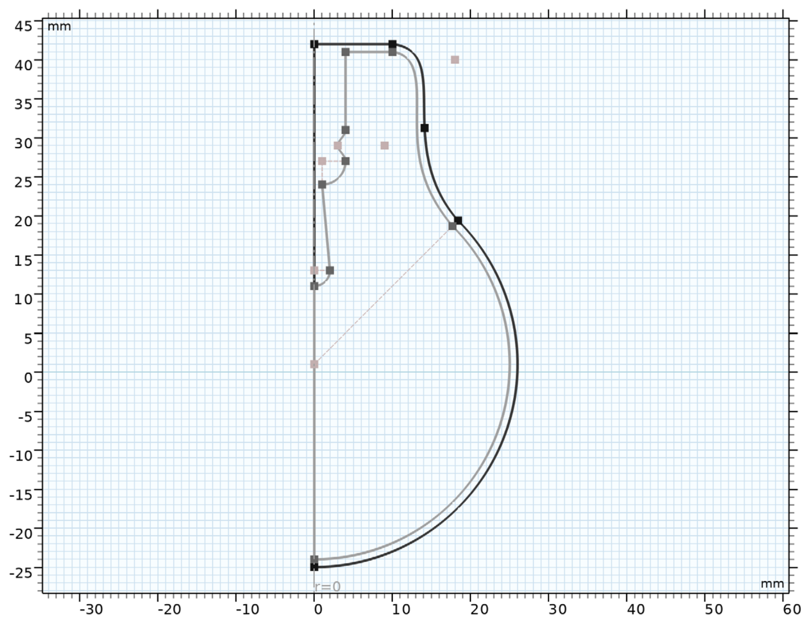
9 In the **z** text field, type 1.

Composite Curve 2 (cc2)

1 In the **Model Builder** window, click **Composite Curve 2 (cc2)**.

2 In the **Settings** window for **Composite Curve**, click **Build Selected**.

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



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

5 From the **Show in physics** list, choose **Off**.

Tungsten

Add the next feature from the **Geometry** toolbar. This allows you to enter the parameters for size and shape directly in the feature.

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

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

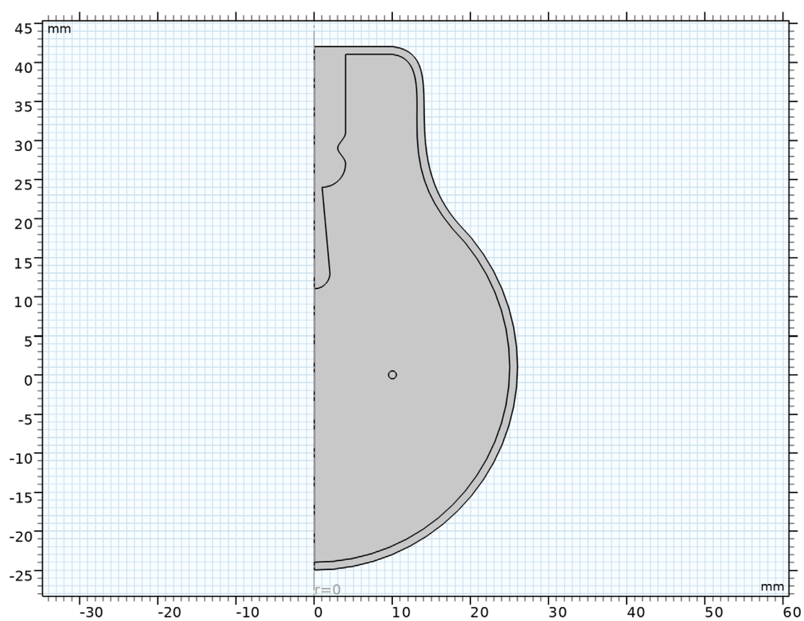
3 Locate the **Size and Shape** section. In the **Radius** text field, type `0.5`.

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

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

6 From the **Show in physics** list, choose **All levels**.

7 Click **Build Selected**.



The geometry is finished, but before continuing we can leave Sketch mode, and inspect the geometry using the **Selection List** window.

8 In the **Sketch** toolbar, click **Sketch**.

SELECTION LIST

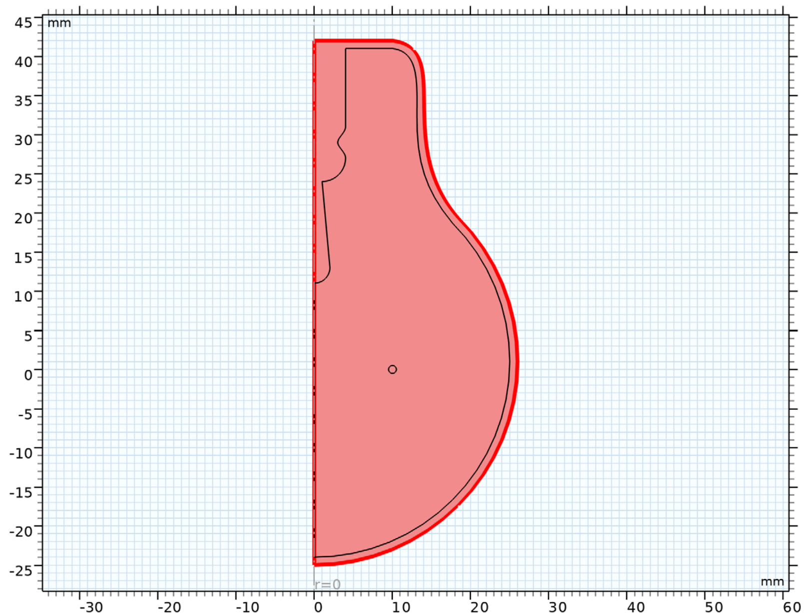
1 In the **Geometry** toolbar, click **Selection List** to open the **Selection List** window.

2 Go to the **Selection List** window.

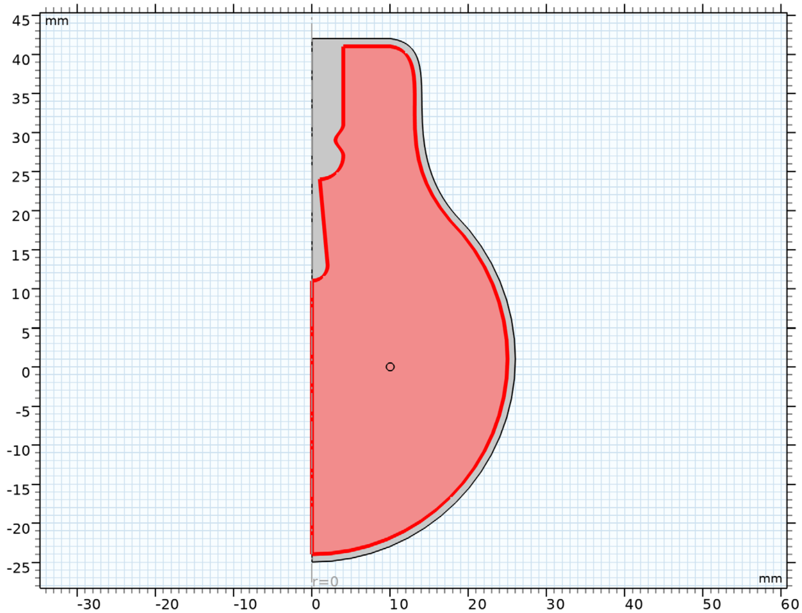
Here you can view a list of geometric objects and entities, and named selections, that exist in the geometry at the current build state for the selected entity level. The list on the top contains objects and entities, and the one at the bottom displays the named selections.

Continue with examining the three objects that comprise the geometry.

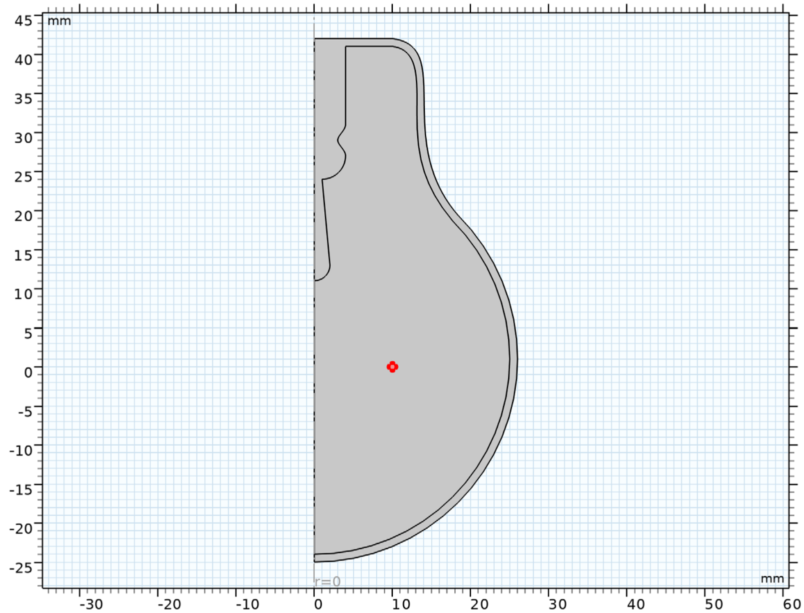
3 In the **Object selections** tree, select **Composite Curve 1**.



4 In the **Object selections** tree, select **Composite Curve 2**.



5 In the **Object selections** tree, select **Tungsten**.



The domains for the glass, and the argon gas, and the tungsten filament result after a geometric Boolean operations of these three objects. Namely, the domain for the glass is the difference of the Composite Curve 1 and Composite Curve 2 objects, and the domain for the argon gas is the difference of the Composite Curve 2 and tungsten objects.

Fortunately, COMSOL Multiphysics automatically computes these domains in the Form Union operation, which is at the end of the geometry sequence, and creates the union of all geometry objects that exist in the sequence while preserving interior boundaries to separate domains.

GEOMETRY 1

Form Union (fin)

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

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

After **Form Union**, only one object is displayed in the upper list of the **Selection List** window. This finalized geometry is divided into domains along the boundaries of the initial objects.

3 In the **Graphics** window toolbar, click ▾ next to **Select Objects**, then choose **Select Domains**.

4 Check the domains corresponding to the glass, argon gas, and tungsten filament by clicking the entities in the Domains list.

In the following sections we will set up named selections that you can use when defining the physics settings.

Glass

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

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

3 Locate the **Input Entities** section. Click the + **Add** button for **Selections to add**.

4 In the **Add** dialog, select **Composite Curve 1** in the **Selections to add** list.

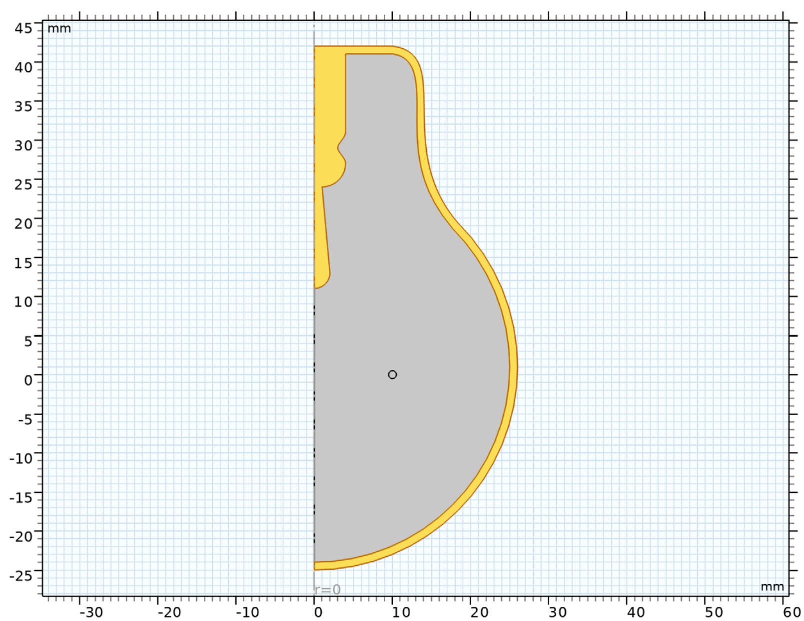
5 Click **OK**.

6 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

7 Click the + **Add** button for **Selections to subtract**.

8 In the **Add** dialog, select **Composite Curve 2** in the **Selections to subtract** list.

9 Click **OK**.



Now that you have a selection for the glass domain, use an **Adjacent Selection** feature to obtain its boundaries.

Glass Boundaries

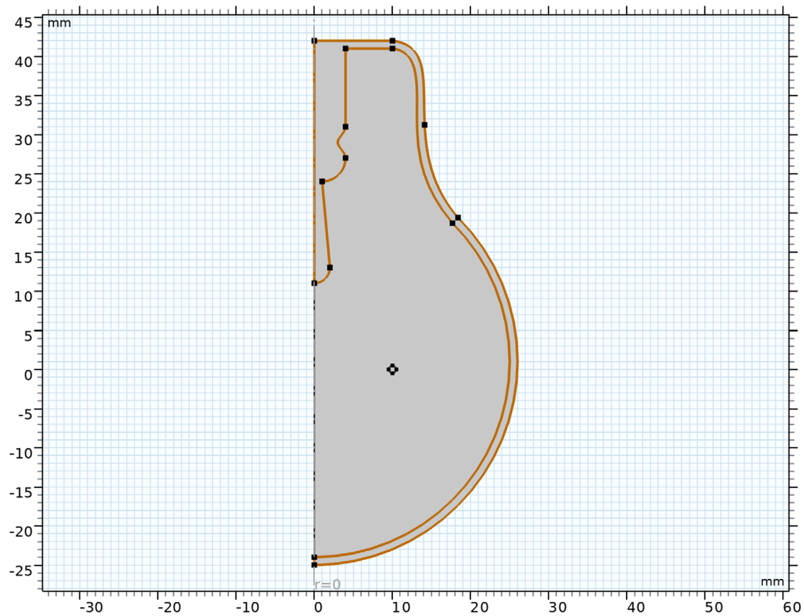
1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

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, select **Glass** in the **Input selections** list.

5 Click **OK**.



Argon

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

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

3 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

4 In the **Add** dialog, select **Composite Curve 2** in the **Selections to add** list.

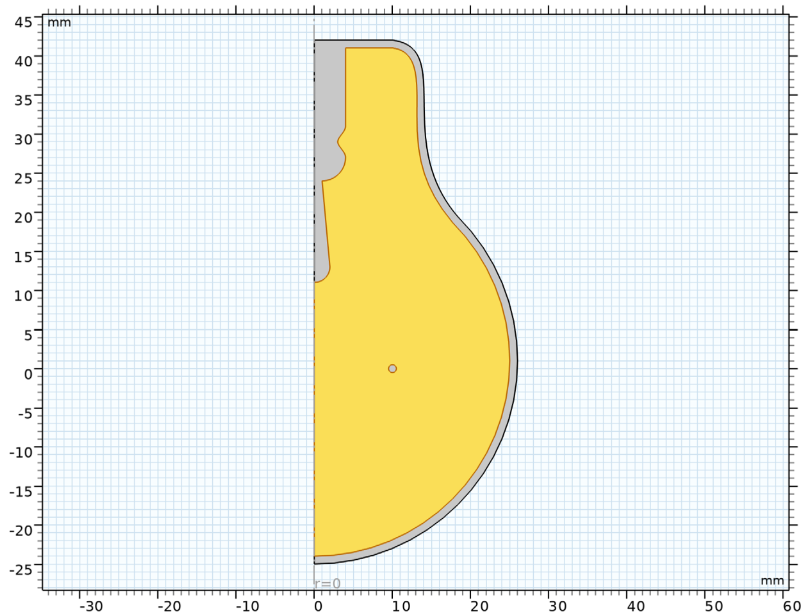
5 Click **OK**.

6 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

7 Click the **Add** button for **Selections to subtract**.

8 In the **Add** dialog, select **Tungsten** in the **Selections to subtract** list.

9 Click **OK**.



Interior Radiation

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

Combine previously defined selections to get the boundaries for the interior radiation.

2 In the **Settings** window for **Difference Selection**, type `Interior 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 the **Add** button for **Selections to add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Tungsten** and **Glass Boundaries**.

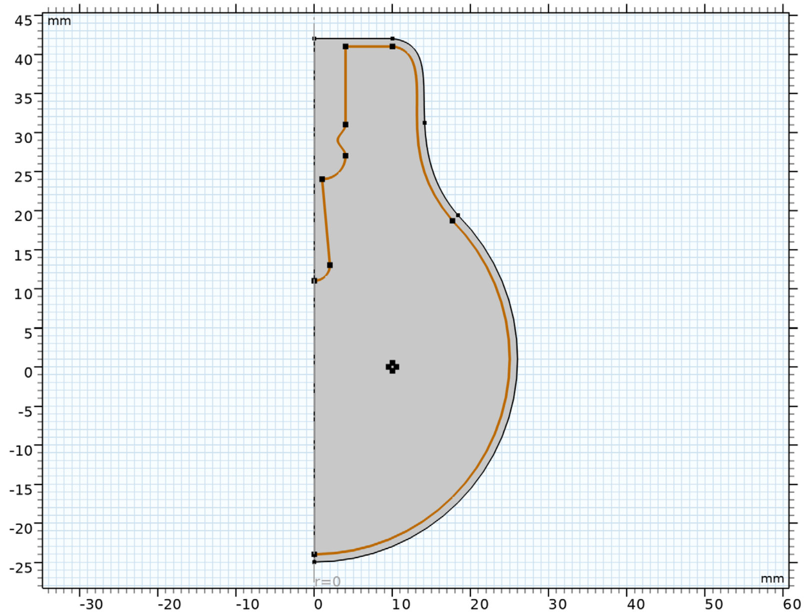
6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Composite Curve 1** in the **Selections to subtract** list.

10 Click **OK**.



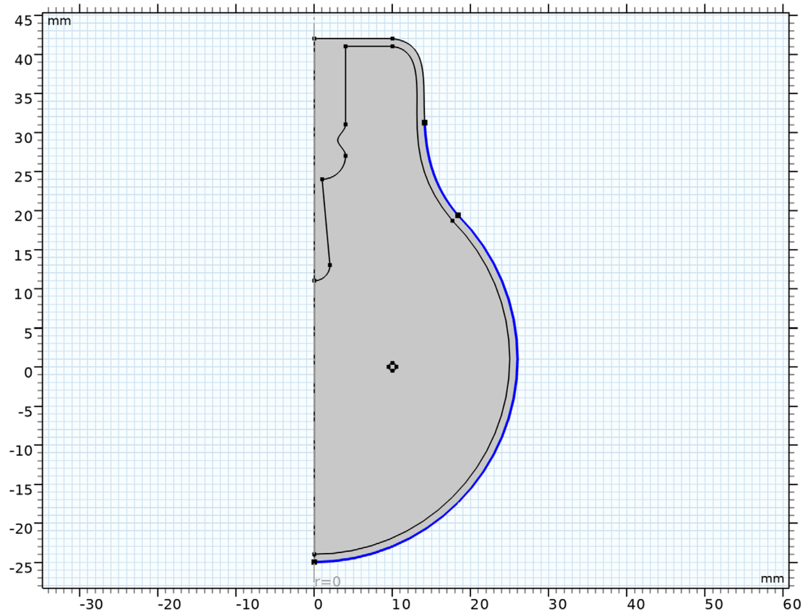
Exterior Radiation

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 14 and 15 only.



Radiation

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

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

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

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

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

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

7 Click **OK**.

8 In the **Settings** window for **Union Selection**, click **Build Selected**.