

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 surfaceto-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

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🔗 Model Wizard.

MODEL WIZARD

- I 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)

- I In the Physics toolbar, click \equiv 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 I

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:

- I 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 **Come Extents** button in the **Graphics** toolbar.

You should now see the geometry shown in Figure 1.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.

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)]	I.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

3 In the table, enter the following settings:

ADD MATERIAL

- I 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 Right-click and choose Add to Component I (compl).

MATERIALS

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

I In the Model Builder window, under Component I (compl)>Materials click Tungsten [solid,Ho et. al.] (matl). 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

- I 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.] I (mat2)

- I 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

- I 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	Ср	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.

- I In the Model Builder window, under Component I (compl) 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 I

- I In the Model Builder window, under Component I (compl)> Heat Transfer in Solids and Fluids (ht) click Fluid I.
- 2 In the Settings window for Fluid, locate the Domain Selection section.
- **3** From the **Selection** list, choose **Argon**.

Initial Values 1

- I In the Model Builder window, click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** From the *T* list, choose **Ambient temperature (amprl)**.

Heat Source 1

- I 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

- I 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 (amprl).

SURFACE-TO-SURFACE RADIATION (RAD)

- I In the Model Builder window, under Component I (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

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

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

- I 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)

- I 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	I	Basic

ADD MATERIAL

- I 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)

- I 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	ht.pA*Mw_a/ (R_const*T)	kg/m³	Basic

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 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 I

Step 1: Time Dependent

- I In the Model Builder window, under Study I click Step I: 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.

- I Click the \longleftrightarrow 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

- I 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.

- I 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)

- I 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

- I 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

- I In the **Results** toolbar, click \sim **ID** Plot Group.
- 2 In the Settings window for ID Plot Group, type Temperature vs. Time in the Label text field.

Point Graph 1

- I 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 I.
- 4 Click Replace Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (compl)>Heat Transfer in Solids and Fluids>Temperature>T Temperature K.
- **5** In the **Temperature vs. Time** toolbar, click **O 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

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID 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 I

- I 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 I (compl)>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

- I 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 I (compl)>Surface-to-Surface Radiation>Radiative heat flux> rad.rflux Radiative heat flux W/m².
- 6 Click **= Evaluate**.

TABLE

I Go to the Table window.

The result should be close to 45 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 I

- I 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, so you can start by zooming out a few steps.

Composite Curve I (ccI)

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



- I In the **Geometry** 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 two control points 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 I** 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 I (ccI)

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

Polygon I (poll)

- In the Model Builder window, expand the Component I (compl)>Geometry I>
 Composite Curve I (ccl) node, then click Polygon I (poll).
- 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 I (cb1)

- I In the Model Builder window, click Cubic Bézier I (cbl).
- 2 In the Settings window for Cubic Bézier, locate the Control Points section.
- **3** In row **2**, set **r** to **18**.
- 4 In row 2, set z to 41.
- **5** In row **4**, set **r** to 13*sqrt(2).
- 6 In row 4, set z to 13*sqrt(2)+1.

Circular Arc 1 (cal)

- I In the Model Builder window, click Circular Arc I (cal).
- 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)

- I In the Model Builder window, click Composite Curve I (ccl).
- **2** In the Settings window for Composite Curve, locate the Selections of Resulting Entities section.
- **3** Select the **Resulting objects selection** check box.
- **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.





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 are 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.



Use the drawing tools in the following order:

- I In the Model Builder window, click Composite Curve 2 (cc2).
- **2** Start with a **Polygon** to draw an edge perpendicular to the rotation axis. Its first vertex is located inwards from the start vertex of the outer shape.
- 3 Continue with a Cubic Bézier polygon. Try to follow the outer shape.
- 4 Add a Circular Arc that ends on the centerline.
- 5 Draw a Polygon up along the centerline to about halfway up the geometry.
- 6 Continue with a Circular Arc that curves away from the centerline.
- 7 Use the Polygon tool to draw an edge that tilts towards the centerline.
- 8 Draw another **Circular Arc** that curves away from then back towards the centerline. The start and end vertices can be aligned vertically.
- 9 Switch to an Interpolation Curve to create a curved segment that first curves towards 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.
- **IO** Close the shape with a vertical edge, using the **Polygon** tool.

Polygon I (poll)

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

- I In the Model Builder window, expand the Composite Curve 2 (cc2) node, then click Polygon I (poll).
- 2 In the Settings window for Polygon, locate the Coordinates section.
- **3** In the table, enter the following settings:

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

Cubic Bézier I (cb1)

I In the Model Builder window, click Cubic Bézier I (cbl).

- 2 In the Settings window for Cubic Bézier, locate the Control Points section.
- **3** In row **2**, set **r** to **18**.
- **4** In row **2**, set **z** to 40.
- 5 In row 3, set r to 9.
- 6 In row 3, set z to 29.
- 7 In row 4, set r to 12.5*sqrt(2).
- 8 In row 4, set z to 12.5*sqrt(2)+1.
- 9 Locate the Weights section. In the 2 text field, type 3/4.

Polygon 2 (pol2)

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

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

3 In the table, enter the following settings:

r (mm)	z (mm)
0	11

Circular Arc 2 (ca2)

- I 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 **z** text field, type 13.
- 4 Locate the Radius section. In the Radius text field, type 2.

Polygon 3 (pol3)

- I In the Model Builder window, click Polygon 3 (pol3).
- 2 In the Settings window for Polygon, locate the Coordinates section.
- **3** In the table, enter the following settings:

r (mm) z (mm) 1 24

Circular Arc 3 (ca3)

- I 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)

- I In the Model Builder window, click Interpolation Curve I (icl).
- 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)
3	29
4	31

- 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)

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



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

- 3 In the Model Builder window, click Composite Curve 2 (cc2).
- **4** Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.
- 5 From the Show in physics list, choose Off.

Tungsten

- I 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** check box.
- 6 From the Show in physics list, choose All levels.



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

- 8 In the Geometry toolbar, click *sketch*.
- 9 In the Home toolbar, click 📑 Windows and choose Selection List.

SELECTION LIST

I 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.

Let's take a look at the three objects that comprise the geometry.

2 In the Graphics window toolbar, click ▼ next to Select Objects, then choose Select Objects.



3 In the tree, select Composite Curve I.





The domains for the glass, and the argon gas, and the tungsten filament result after 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 I

Form Union (fin)

- I In the Model Builder window, under Component I (compl)>Geometry I click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, click 틤 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 entries in the Domains list.

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

Glass

- I 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 Add right below the Selections to add list.
- 4 In the Add dialog box, select Composite Curve I in the Selections to add list.
- 5 Click OK.
- 6 Locate the Input Entities section. Click Add right below the Selections to subtract list.
- 7 In the Add dialog box, select Composite Curve 2 in the Selections to subtract list.
- 8 Click OK.



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

Glass Boundaries

- I 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.



Argon



- 2 In the Settings window for Difference Selection, type Argon in the Label text field.
- 3 Locate the Input Entities section. Click Add right below the Selections to add list.
- 4 In the Add dialog box, select Composite Curve 2 in the Selections to add list.
- 5 Click OK.
- 6 Locate the Input Entities section. Click Add right below the Selections to subtract list.
- 7 In the Add dialog box, select Tungsten in the Selections to subtract list.



Interior Radiation

I In the Geometry toolbar, click 🐚 Selections and choose Difference Selection.

Combine the previously defined **Resulting objects 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 Add right below the Selections to add list.
- 5 In the Add dialog box, in the Selections to add list, choose Tungsten and Glass Boundaries.
- 6 Click OK.
- 7 Locate the Input Entities section. Click Add right below the Selections to subtract list.
- 8 In the Add dialog box, select Composite Curve I in the Selections to subtract list.



Exterior Radiation

I 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

I 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.



7 In the Settings window for Union Selection, click 틤 Build Selected.