Petzval Lens STOP Analysis with Surface-to-Surface Radiation

Introduction

Many optical systems are operated in extreme environments, where temperature changes are significant. This will invariably induce deformations in the optical geometry. In order to simulate the effects of structural and thermal deformation on the optical performance of a lens, a structural-thermal-optical-performance (STOP) analysis should be performed.

In the Petzval Lens STOP Analysis tutorial the model was assumed to have a uniform temperature throughout the geometry. However, this assumption will not always be valid. For example, during thermal vacuum chamber testing of a lens, the radiative flux from an external source (for example, a warm room) can create a thermal gradient within the lens assembly. In this model a simple thermal vacuum chamber geometry is added to the model. The **Heat Transfer with Surface-to-Surface Radiation** multiphysics interface is then used to model radiative heat transfer through an assembly that is being tested in a cold vacuum where the light source is external.

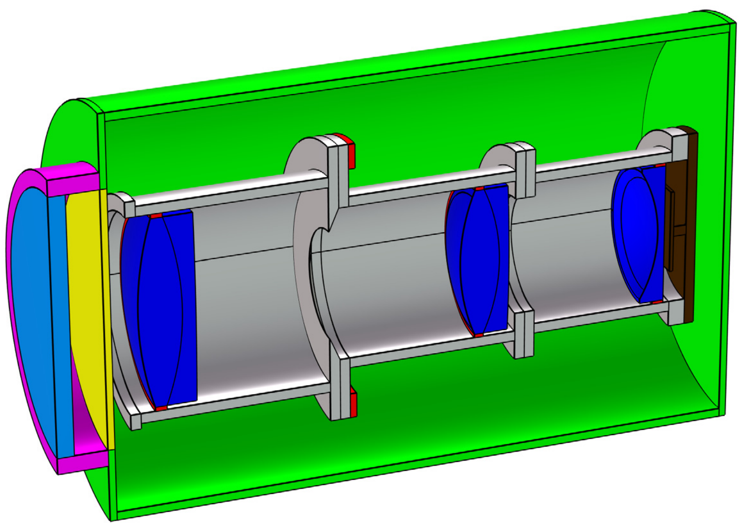


Figure 1: An overview of the Petzval Lens STOP Analysis with Surface-to-Surface Radiation geometry in cross-section. The Petzval lens elements are shown in dark blue, the lens supports are red, and the detector assembly is dark gray. The thermal shroud is green; the outside of the shroud is assumed to be actively cooled to a steady low temperature. The leftmost (light blue) element is the vacuum window and magenta element is vacuum chamber. The yellow element is the thermal shroud window.

Model Definition

Details of the lens simulated in this tutorial can be found in the Petzval Lens tutorial (see Ref. 1, p. 191) and a STOP analysis at a single isothermal temperature is demonstrated in Petzval Lens STOP Analysis. In that model a simple barrel geometry and detector assembly was been added (see Figure 2). Detailed instructions for creating the geometry can be found in the appendices of these two tutorials.

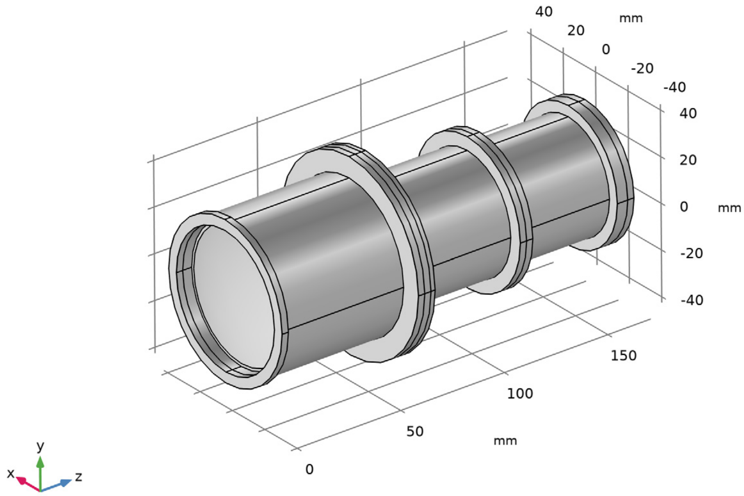


Figure 2: The Petzval Lens Stop Analysis geometry sequence. In this view, several boundaries have been hidden so that internal details can be seen.

In the current tutorial, a vacuum window, a vacuum chamber, and thermal shroud has been added to the geometry (Figure 3). The instructions for creating the geometry are included in the detailed step-by-step instructions for this model (see Modeling Instructions).

The exterior of the thermal shroud is held at a fixed temperature $T_0 = -25°C$, whereas the outside environment is at a nominal temperature of $T_1 = 22.5°C$.

The edges of the vacuum window are assumed to be in thermal contact with the vacuum chamber; that is, at a temperature $T_1 = 22.5°C$.

To reduce the heat load on the first element of the lens assembly, which is placed close to the vacuum window, a second “thermal window” is included as part of the thermal shroud. This window allows light (and heat) to pass from outside to inside the thermal enclosure.

The surfaces in the lens assembly are assumed to be transparent to visible light but opaque to infrared light. In this example the emissivity of all diffuse surfaces was set to a nominal value of $\varepsilon = 0.9$. These exterior surfaces and the thermal shroud interact via surface-to-surface radiation to reach an equilibrium temperature.

In this tutorial the surrounding void is assumed to be vacuum. Therefore, the refractive index of the exterior domains is set to $n = 1.0$ (that is, absolute vacuum)[1]. All other material properties remain unchanged from the basis model.

In this model, small adjustments are made to the mesh in order to reduce the effects of discretization on the ray tracing accuracy. The mesh is shown in Figure 4.

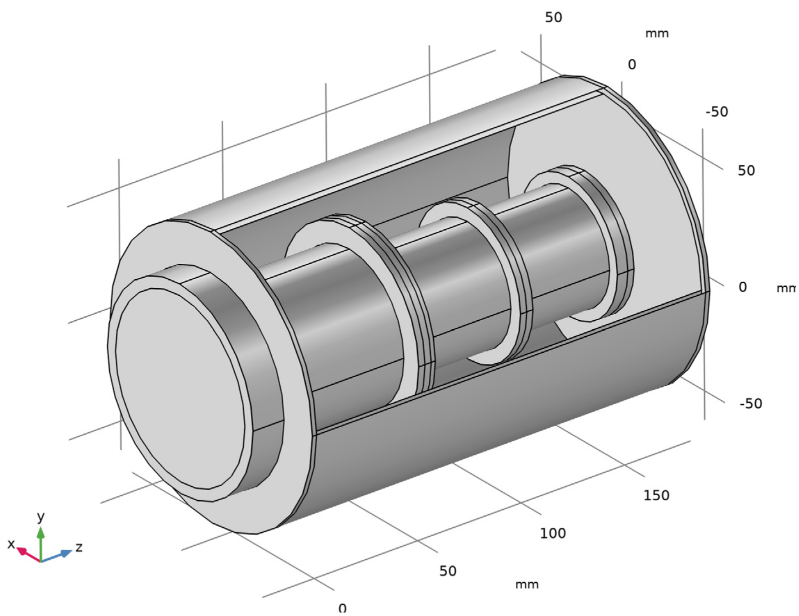


Figure 3: The complete geometry sequence including the thermal shroud, thermal window (which is attached to the shroud), vacuum chamber, and the vacuum window.

1. Note that in all other Petzval Lens models the surround domain is assumed to be air at the nominal temperature. Therefore, when making a comparison between the model with and without surface-to-surface radiation, make sure to use the same model for the refractive index of the exterior and unmeshed domains.

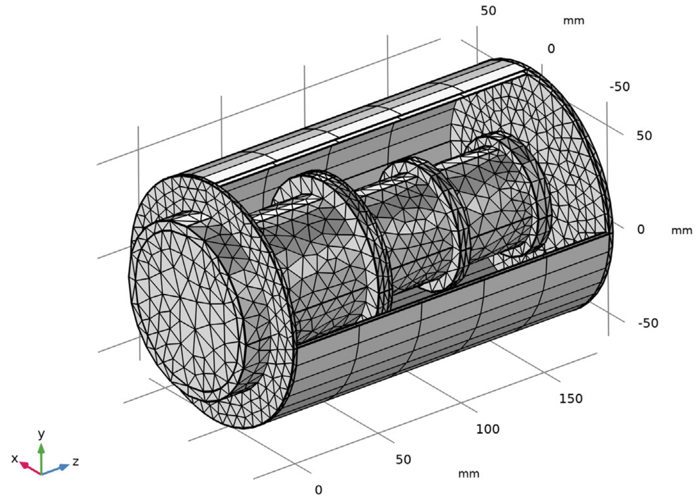


Figure 4: The Petzval Lens Stop Analysis mesh.

Results and Discussion

Following a stationary study where the deformation of the geometry is computed, a ray trace is performed using three wavelengths (475 nm, 550 nm, and 625 nm) over three field angles (0°, 3.5°, and 7.0°).

Figure 5 shows the resulting temperature field across the lens and barrel assembly. It is clear that the influence of the external warm room creates a substantial gradient across the first lens element. The remaining elements are close to isothermal, but slightly warmer than the nominal T_0 = –25°C. The temperature of the vacuum window and thermal window can also be seen in Figure 9. In Figure 6, the displacement field is shown together with the von Mises stress field. The overall stress remains very similar to the model without surface-to-surface radiation.

The image quality on the nominal and "best focus" image planes is shown in Figure 7 and Figure 8. It is an interesting coincidence to note that at T_0 = –25°C, with a vacuum exterior, there is very little difference between the nominal and best focus image planes. It should also be noted that the thermal gradient across the first lens element appears to have little impact on the image quality. That is, the variation in refractive index within this lens as a result of the thermal gradient, is small for a monochromatic set of rays that have been released from a single field.

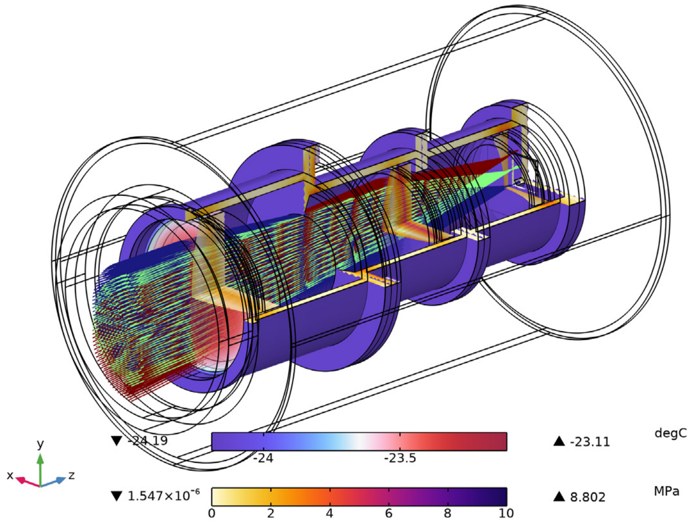


Figure 5: A ray trace shown together with a 3/4 section view of the Petzval lens assembly. The temperature gradient across the first lens element can be clearly seen.

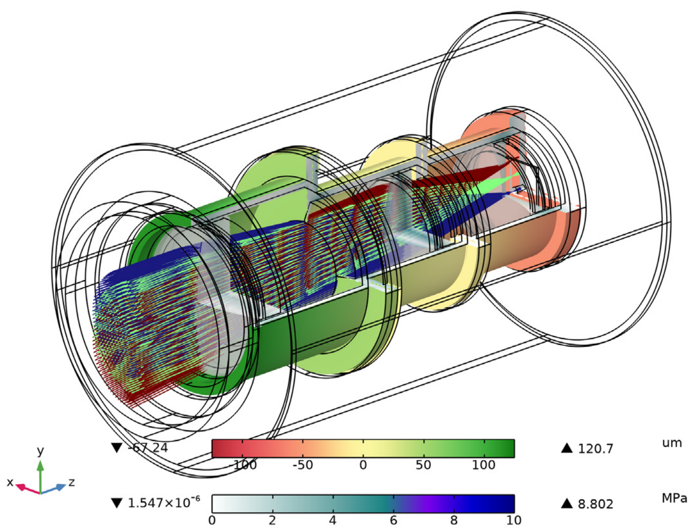


Figure 6: In this ray trace, the displacement field is shown together with the von Mises stress field.

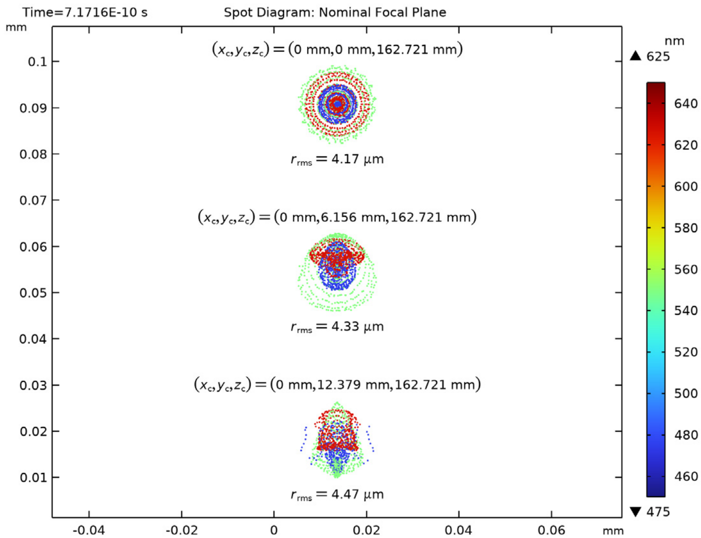


Figure 7: The image quality on the nominal image surface. The is the detector surface after being subject to thermal expansion.

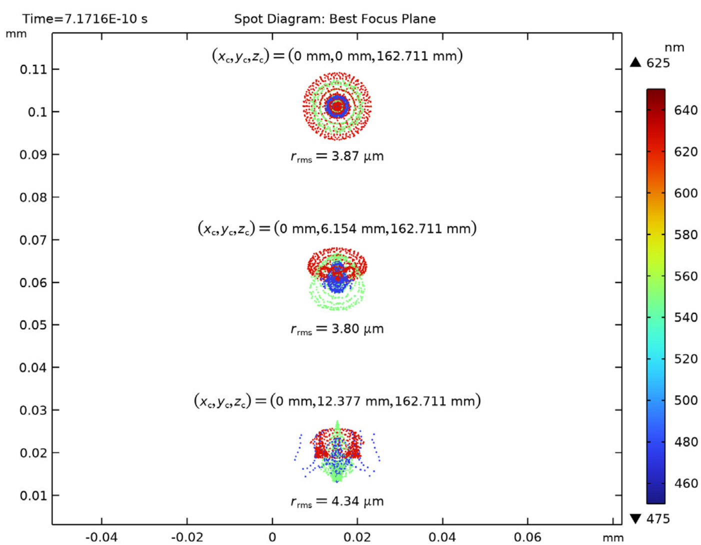


Figure 8: Image quality on the "best focus" plane. That is, this is the surface that gives the minimum RMS spot size on-axis.

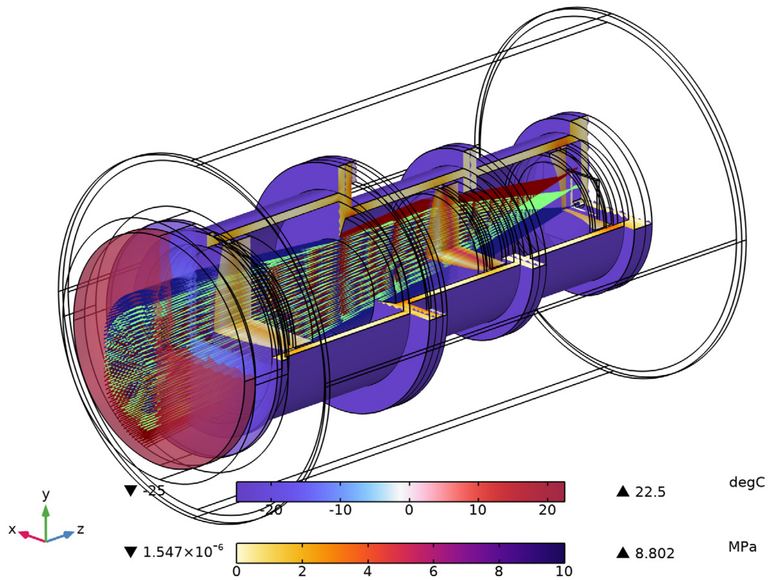


Figure 9: In this ray trace the temperature of the vacuum and thermal windows is also shown.

References

1. M.J. Kidger, *Fundamental Optical Design*, Bellingham WA, USA: SPIE Press, 2001.

2. https://www.schott.com/.

3. https://oharacorp.com/optical-glass/.

4. M.A. Salama, W.M. Rowe, and R.K. Yasui, "Thermoelastic Analysis of Solar Cell Arrays and their Material Properties," *Technical Memorandum 33-626*, NASA, 1973.

5. T.M. Mower, "Thermomechanical behavior of aerospace-grade RTV (silicone adhesive)," *Int. J. Adhes. Adhes.*, vol. 87, pp. 64–72, 2018.

6. P.R. Yoder, Jr., *Opto-Mechanical Systems Design*, Bellingham WA, USA: SPIE Press, 2006.

Application Library path: `Ray_Optics_Module/`
`Structural_Thermal_Optical_Performance_Analysis/`
`petzval_lens_stop_analysis_with_surface_to_surface_radiation`

Modeling Instructions

APPLICATION LIBRARIES

1. From the **File** menu, choose **Application Libraries**.
2. In the **Application Libraries** window, select **Ray Optics Module>** **Structural Thermal Optical Performance Analysis>petzval_lens_stop_analysis** in the tree.
3. Click **Open**.

ADD PHYSICS

The instructions for creating the geometry in the appendix of the Petzval Lens STOP analysis tutorial. Orient the view to match Figure 2.

Now, add the Heat Transfer with Surface-to-Surface Radiation multiphysics interface.

1. In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Heat Transfer>Radiation>Heat Transfer with Surface-to-Surface Radiation**.
4. Click **Add to Component 1** in the window toolbar.
5. In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

GLOBAL DEFINITIONS

General Properties

1. In the **Model Builder** window, under **Global Definitions** click **General Properties**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
T0	-25.0[degC]	248.15 K	Thermal shroud temperature

Temperatures

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 4**.

2 In the **Settings** window for **Parameters**, type `Temperatures` in the **Label** text field.

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

Name	Expression	Value	Description
T1	22.5[degC]	295.65 K	Ambient temperature

Add the thermal vacuum chamber geometry. Only the window of the vacuum chamber and the internal thermal shroud will be included in this simulation.

PETZVAL LENS STOP ANALYSIS GEOMETRY SEQUENCE

Vacuum Window

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

2 In the **Settings** window for **Cylinder**, type `Vacuum Window` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `40`.

4 In the **Height** text field, type `5`.

5 Locate the **Position** section. In the **z** text field, type `-25`.

6 Locate the **Coordinate System** section. From the **Take work plane from** list, choose **Lens 1 (pi1)**.

7 From the **Work plane** list, choose **Surface 1 vertex intersection (wp1)**.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 From the **Show in physics** list, choose **Boundary selection**.

10 Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Lens Material 1**.

Vacuum Window Front Annulus

1 In the **Geometry** toolbar, click **Part Instance** and choose **Circular Planar Annulus**.

2 In the **Settings** window for **Part Instance**, type `Vacuum Window Front Annulus` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
d0	90.0[mm]	90 mm	Diameter, outer
d1	80.0[mm]	80 mm	Diameter, inner

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Lens 1 (pi1)**.

5 From the **Work plane** list, choose **Surface 1 vertex intersection (wp1)**.

6 Find the **Displacement** subsection. In the **zw** text field, type `-25`.

Vacuum Window Front

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

2 In the **Settings** window for **Extrude**, type `Vacuum Window Front` in the **Label** text field.

3 On the object **pi15**, select Boundary 1 only.

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

Distances (mm)
15

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Vacuum Window 1` in the **Name** text field.

7 Click **OK**.

Thermal Window

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

2 In the **Settings** window for **Cylinder**, type `Thermal Window` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `40`.

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

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

6 Locate the **Coordinate System** section. From the **Take work plane from** list, choose **Lens 1 (pi1)**.

7 From the **Work plane** list, choose **Surface 1 vertex intersection (wp1)**.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 From the **Show in physics** list, choose **Boundary selection**.

10 Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Lens Material 1**.

Thermal Shroud Front Annulus

1 In the **Geometry** toolbar, click **Part Instance** and choose **Circular Planar Annulus**.

2 In the **Settings** window for **Part Instance**, type `Thermal Shroud Front Annulus` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
d0	`125.0[mm]`	125 mm	Diameter, outer
d1	`80.0[mm]`	80 mm	Diameter, inner

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Lens 1 (pi1)**.

5 From the **Work plane** list, choose **Surface 1 vertex intersection (wp1)**.

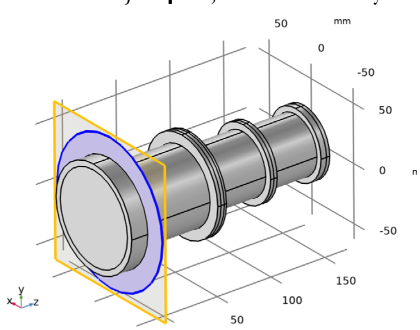
6 Find the **Displacement** subsection. In the **zw** text field, type `-10`.

Thermal Shroud Front

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

2 In the **Settings** window for **Extrude**, type `Thermal Shroud Front` in the **Label** text field.

3 On the object **pi16**, select Boundary 1 only. The selection is shown in the figure below:



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

Distances (mm)
2

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Thermal Shroud` in the **Name** text field.

7 Click **OK**.

Thermal Shroud End

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

2 In the **Settings** window for **Cylinder**, type `Thermal Shroud End` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `62.5`.

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

5 Locate the **Coordinate System** section. From the **Take work plane from** list, choose **Lens 1 (pi1)**.

6 From the **Work plane** list, choose **Surface 1 vertex intersection (wp1)**.

7 Locate the **Position** section. In the **zw** text field, type `180`.

8 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Thermal Shroud**.

Thermal Shroud Annulus

1 In the **Geometry** toolbar, click **Part Instance** and choose **Circular Planar Annulus**.

2 In the **Settings** window for **Part Instance**, type `Thermal Shroud Annulus` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
d0	`125.0[mm]`	125 mm	Diameter, outer
d1	`120.0[mm]`	120 mm	Diameter, inner
nix	`0`	0	Local optical axis, x-component
niy	`0`	0	Local optical axis, y-component
niz	`1`	1	Local optical axis, z-component

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Lens 1 (pi1)**.

5 From the **Work plane** list, choose **Surface 1 vertex intersection (wp1)**.

6 Find the **Displacement** subsection. In the **zw** text field, type `-8`.

7 Click to expand the **Boundary Selections** section. In the table, select the **Keep** check box for **All**.

Thermal Shroud Outer

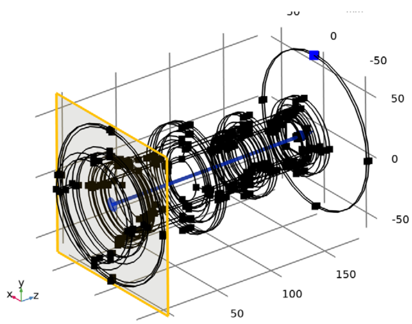
1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, type `Thermal Shroud Outer` in the **Label** text field.

3 On the object **pi17**, select Boundary 1 only.

4 Locate the **Distances** section. From the **Specify** list, choose **Vertices to extrude to**.

5 On the object **cyl5**, select Point 5 only. Click the **Wireframe Rendering** button in the **Graphics** toolbar to make it easier to see the selections. These selections are shown in the figure below:



6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Thermal Shroud**.

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

9 Click the **Zoom Extents** button in the **Graphics** toolbar. The resulting geometry should look like Figure 3.

Update and add selections to define the material and physics properties.

DEFINITIONS

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Lens Barrels, Detector and Thermal Shroud

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>Selections** node, then click **Lens Barrels and Detector**.

2 In the **Settings** window for **Union**, type `Lens Barrels, Detector and Thermal Shroud` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Vacuum Window 1** and **Thermal Shroud**.

5 Click **OK**.

All Lenses and Windows

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `All Lenses and Windows` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Lens Material 1**, **Lens Material 2**, **Lens Material 3**, and **Lens Material 4**.

5 Click **OK**.

Complete Lens Assembly

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Complete Lens Assembly` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Supports**, **All Lenses**, **All (Barrel 1)**, **All (Barrel 2)**, **All (Barrel 3)**, and **Detector Assembly**.

5 Click **OK**.

Lens and Window Exteriors

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 In the **Label** text field, type `Lens and Window Exteriors`.

5 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

6 In the **Add** dialog box, in the **Selections to add** list, choose **Lens Exteriors**, **Vacuum Window**, and **Thermal Window**.

7 Click **OK**.

All Domains

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

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

3 Locate the **Input Entities** section. Select the **All domains** check box.

All Exteriors

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

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

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

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

5 Click **OK**.

Thermal Shroud Exterior

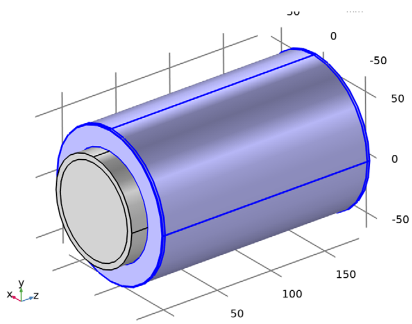
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 In the **Graphics** window toolbar, click ▾ next to **View Unhidden**, then choose **View All**.

5 Select Boundaries 1–5, 7, 8, 10, 18, 151–153, and 236–238 only. The selections should look like below:



6 In the **Graphics** window toolbar, click ▾ next to **View Unhidden**, then choose **View Unhidden**.

Vacuum Window Exterior

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

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

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

4 Select Boundaries 15–17, 155, and 234 only.

Vacuum Window and Thermal Window Entrance

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

2 In the **Settings** window for **Explicit**, type `Vacuum Window and Thermal Window Entrance` in the **Label** text field.

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

4 Select Boundaries 31–34, 37, 160, and 229 only.

Vacuum Window Edges

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

2 In the **Settings** window for **Explicit**, type `Vacuum Window Edges` in the **Label** text field.

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

4 Select Boundaries 29, 30, 159, and 228 only.

All Surface to Surface

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `All Surface to Surface` in the **Label** text field.

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

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **All Exteriors** in the **Selections to add** list.

6 Click **OK**.

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

8 Under **Selections to subtract**, click **Add**.

9 In the **Add** dialog box, in the **Selections to subtract** list, choose **Thermal Shroud Exterior**, **Vacuum Window Exterior**, and **Vacuum Window Edges**.

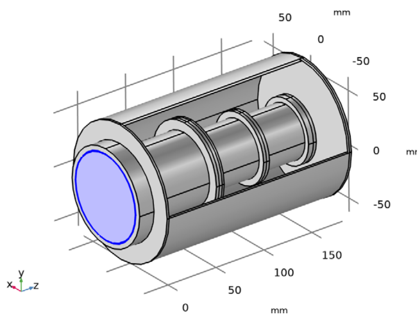
10 Click **OK**.

GEOMETRICAL OPTICS (GOP)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometrical Optics (gop)**.

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

3 From the **Selection** list, choose **All Lenses and Windows**. Use the predefined selection to add the vacuum and thermal windows to this interface. The selections should look like the figure below:



4 Locate the **Material Properties of Exterior and Unmeshed Domains** section. From the **Optical dispersion model** list, choose **Absolute vacuum**. In this model, the lens assembly is surrounded by vacuum. Only one exterior and unmeshed domain can be assigned this way.

Medium Properties 1

1 In the **Model Builder** window, expand the **Geometrical Optics (gop)** node, then click **Medium Properties 1**.

2 In the **Settings** window for **Medium Properties**, locate the **Model Inputs** section.

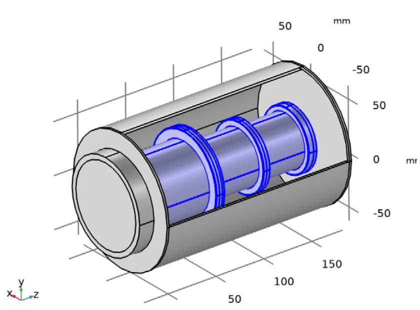
3 From the T list, choose **Temperature (ht)**.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

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

3 From the **Selection** list, choose **Complete Lens Assembly**. Only the lens assembly needs to be considered in the Solid Mechanics interface.



4 In the **Model Builder** window, expand the **Solid Mechanics (solid)** node.

Thermal Expansion 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)> Linear Elastic Material 1** node.

2 Right-click **Thermal Expansion 1** and choose **Disable**. A multiphysics coupling will be used instead. This is added next.

MULTIPHYSICS

Thermal Expansion 1 (te1)

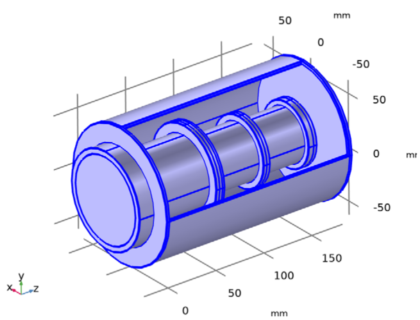
1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain> Thermal Expansion**.

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

3 From the **Selection** list, choose **Complete Lens Assembly**. Only the lens assembly needs to be included here.

HEAT TRANSFER IN SOLIDS (HT)

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



2 In the **Settings** window for **Heat Transfer in Solids**, click to expand the **Discretization** section.

3 From the **Temperature** list, choose **Linear**. A linear shape can be used based on the assumption that the temperature field is slowly varying across the geometry.

Temperature 1

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

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

3 From the **Selection** list, choose **Thermal Shroud Exterior**.

4 Locate the **Temperature** section. In the T_0 text field, type `T0`. This is the temperature of the thermal shroud.

Temperature 2

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

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

3 Click **Clear Selection**.

4 From the **Selection** list, choose **Vacuum Window Edges**.

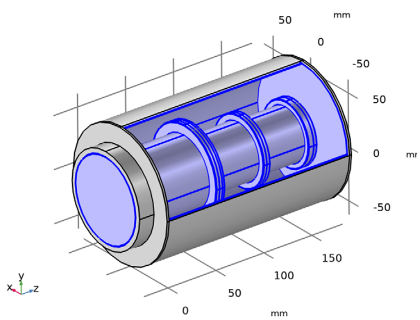
5 Locate the **Temperature** section. In the T_0 text field, type `T1`. The edges of the vacuum window edges are assumed to be in contact with the vacuum chamber.

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 **All Surface to Surface**. The surfaces to be considered in this interface have been predefined.



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 In the T_{amb} text field, type `T0`.

4 Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.9`.

Diffuse Surface 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.
2. In the **Settings** window for **Diffuse Surface**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Vacuum Window and Thermal Window Entrance**.
4. Locate the **Ambient** section. In the T_{amb} text field, type `T1`.
5. Locate the **Surface Emissivity** section. From the ε list, choose **User defined**. In the associated text field, type `0.9`.

MATERIALS

RTV (mat6)

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Materials** node, then click **RTV (mat6)**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	k_RTV	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	Cp_RTV	J/(kg·K)	Basic

MESH 1

Adjust the default mesh to simplify the meshing of the lens barrel and thermal chamber assemblies.

Free Triangular 2

1. In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.
2. Select Boundaries 6 and 42 only.

Swept 1

1. In the **Mesh** toolbar, click **Swept**.
2. In the **Settings** window for **Swept**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Thermal Shroud Outer**.

Free Tetrahedral 1

1. In the **Model Builder** window, click **Free Tetrahedral 1**.
2. Drag and drop below **Swept 1**.

Size

1. In the **Model Builder** window, click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Coarser**.
4. Click **Build All**. The updated mesh should look like Figure 4.

STUDY 1

Now, update the studies. The Freeze condition on the image surface remains disabled. The following steps are to remove the Heat Transfer in Solids and Surface-to-Surface Radiation interfaces from the Ray Tracing Study step.

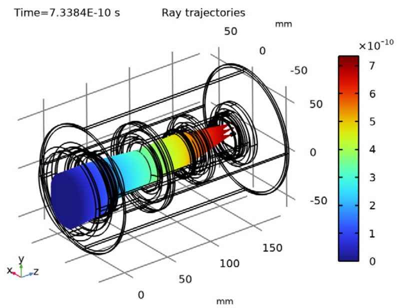
Step 2: Ray Tracing

1. In the **Model Builder** window, expand the **Study 1** node, then click **Step 2: Ray Tracing**.
2. In the **Settings** window for **Ray Tracing**, locate the **Study Settings** section.
3. In the **Lengths** text field, type `0 220`. Slightly increase the maximum optical path length.
4. Locate the **Physics and Variables Selection** section. In the tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)**, **Component 1 (comp1)>Surface-to-Surface Radiation (rad)**, **Component 1 (comp1)>Multiphysics>Heat Transfer with Surface-to-Surface Radiation 1 (htrad1)**, and **Component 1 (comp1)>Multiphysics> Thermal Expansion 1 (te1)**.
5. Click **Disable in Solvers**.
6. In the **Home** toolbar, click **Compute**.

RESULTS

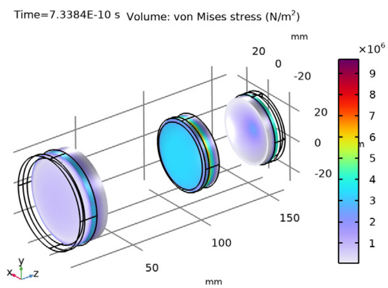
Ray Trajectories (gop)

The Ray Trajectories plot and the Stress plots are created by default with this combination of physics and study steps.



Stress (solid)

In the **Model Builder** window, click **Stress (solid)**.



Surface 1

1 In the **Model Builder** window, expand the **Temperature** node, then click **Surface 1**.

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

3 In the **Expression** text field, type T.

4 Click to expand the **Range** section. Clear the **Manual color range** check box.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 In the **Expression** text field, type T.

4 From the **Unit** list, choose **degC**.

5 Locate the **Range** section. Clear the **Manual color range** check box.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

7 In the **Temperature** toolbar, click **Plot**. The result should match Figure 5.

Displacement

1 In the **Model Builder** window, under **Results** click **Displacement**.

2 In the **Displacement** toolbar, click **Plot**. The result should look like Figure 6.

Spot Diagram, Nominal

Click the **Zoom Extents** button in the **Graphics** toolbar. The nominal image surface spot diagram should look like Figure 7.

Spot Diagram 1

1 In the **Model Builder** window, under **Results>Spot Diagram, Best Focus** click **Spot Diagram 1**.

2 In the **Settings** window for **Spot Diagram**, locate the **Filters** section.

3 Select the **Filter by release feature index** check box.

4 Click to expand the **Focal Plane Orientation** section. Click **Recompute Focal Plane Dataset**.

5 Locate the **Filters** section. Clear the **Filter by release feature index** check box.

6 In the **Spot Diagram, Best Focus** toolbar, click **Plot**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar. The best focus spot diagram should look like Figure 8.

Temperature (Thermal and Vacuum Windows)

1 In the **Model Builder** window, right-click **Temperature** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Temperature (Thermal and Vacuum Windows)` in the **Label** text field.

Surface 2

In the **Model Builder** window, expand the **Temperature (Thermal and Vacuum Windows)** node.

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Lens and Window Exteriors**.

4 In the **Temperature (Thermal and Vacuum Windows)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar. The figure should match Figure 9.