Created in COMSOL Multiphysics 6.0

Thermally Induced Focal Shift in High-Power Laser Focusing Systems

Introduction

Modern high-power industrial fiber laser systems can deliver up to 3 kW of single-mode laser radiation onto surfaces to be cut, drilled, welded, or marked (Ref. 1). Even when the optical components used to focus the beam are almost completely transparent, the amount of heat absorbed by these optical components can degrade the ability of the system to correctly focus the beam.

The heat generated in a lens can change the paths of rays through several different mechanisms, including the following:

- Temperature dependence of the refractive index
- Stress-optical effects resulting from thermal stress
- Thermal expansion of the lenses

In this example of a high-power laser focusing system, rays are traced through an imaging relay in which the rays can deposit energy. The lenses are heated as a result. The temperature dependence of the refractive index and the thermal expansion of the lenses is considered. However, any stress-induced changes in the refractive index are neglected. The resulting thermally induced focal shift is then computed.

Model Definition

The high-power laser focusing system used in this tutorial consists of two identical silica glass plano-convex lenses. The first lens collimates the output of an optical fiber (numerical aperture of 0.1) and the second lens focuses the collimated beam at a target surface.

The model geometry consists of two 50 mm diameter fused silica glass lenses with an effective focal length of approximately 150 mm. The lenses are used to focus a laser beam with free-space wavelength λ_0 = 1064 nm. The position of each lens is assumed to be fixed at three locations. The effects of changes in the lens temperature on the ray paths are modeled for two different values of the source power, 1 W and 3 kW. The thermal effects are negligible when the 1 W source is used. When a 3 kW beam is released, the change in temperature in the lenses causes a noticeable change in the position of the focal plane.

The model uses the **Geometrical Optics** interface to trace the paths of rays through the lens system. The **Heat Transfer in Solids** and **Solid Mechanics** interfaces are used to model the thermal expansion of the lenses.

ATTENUATION OF RAYS IN AN ABSORBING MEDIUM

The intensity and power of a plane wave in an absorbing medium decay exponentially as the wave propagates, assuming the absorption coefficient remains constant,

$$I = I_0 \exp\left(-\frac{2k_0 \kappa L}{n}\right)$$
$$P = P_0 \exp\left(-\frac{2k_0 \kappa L}{n}\right)$$

where k_0 is the free-space wave number,

$$k_0 = \frac{2\pi}{\lambda_0}$$

λ_0 is the free-space wavelength, L is the optical path length in the medium,

$$L = ct$$

c is the speed of light in a vacuum, and t is the current time. The complex-valued refractive index is expressed as $n - \kappa i$, where n and κ are dimensionless real numbers. Positive values of κ correspond to attenuating media whereas negative values indicate gain media.

In the **Geometrical Optics** interface it is possible to assign separate degrees of freedom for ray intensity and power. You can solve for either, both, or neither of these quantities. The ray power only changes due to absorption or gain by the surrounding media and is unaffected by the focusing or defocusing of rays. The intensity increases where a thin pencil of rays would be focused, and decreases where a thin pencil of rays would diverge. Whatever power is lost by the rays due to absorption becomes a heat source of equal magnitude on the underlying domain, through the **Ray Heat Source** multiphysics coupling feature.

COUPLING RAY OPTICS AND HEAT TRANSFER

The ray trajectories and temperature distribution affect each other through a bidirectional coupling. In other words, the ray trajectories affect the temperature field, which in turn perturbs the ray trajectories, both directly and through the resulting structural deformation. To solve for the ray trajectories and temperature in a self-consistent manner, the dedicated Ray Heating interface and **Bidirectionally Coupled Ray Tracing** study step are used. The **Bidirectionally Coupled Ray Tracing** study step sets up a solver loop in which the ray trajectories and temperature are computed in alternating steps for a number of iterations. The results of each iteration are used to assign the **Values of variables not solved for** in the iteration that immediately follows it. This iterative loop can also be set up manually by adding **For** and **End For** nodes to the solver sequence, but the **Bidirectionally Coupled Ray Tracing** study step adds these nodes to the solver sequence automatically.

For more details on the physics implementation and theory for the Geometrical Optics interface, see the *Ray Optics Module User's Guide*.

Results and Discussion

The trajectories or rays in the 3 kW beam are shown in Figure 1. The grayscale coloring along the ray paths indicates the amount of power that the rays transfer. It is constant in free space, decreases due to absorption by the lenses. The surface plot shows the temperature distribution, which is nearly identical in the two lenses. The maximum temperature in the lenses is approximately 504 K.

The von Mises stress and deformation resulting from absorption of the high power laser light in the lenses are shown in Figure 2. A fixed color range has been used to more clearly show the stress distribution throughout the lenses. The maximum displacement is approximately 3.3 µm.

The power deposited in the lenses and at the target surface is shown in Figure 3 and Figure 4, respectively. The maximum heat source in the lenses is about 1.5 mW/mm^3. It shows some asymmetry because of discretization error; the deposited power is piecewise discontinuous across different mesh elements within the lenses. The boundary heat source at the target surface has a maximum value of about 450 kW/mm^2.

The change in the temperature of the lenses causes a change in their refractive indices, which is plotted in Figure 5 for the focusing lens. The figure displays the difference between the real part of the calculated refractive index (n_r) and the refractive index at room temperature (n_0). The change in the refractive index reaches a maximum at the center, where it is approximately 50 % greater than the change at the edges.

Figure 6 shows the RMS spot size as a function of focal plane position. From this plot it is evident that the location of the minimum spot size is shifted by just over 2 mm as the lenses are heated. In order to generate this figure, the **Ray Tracing** study uses smaller intervals when the rays are in the vicinity of the focal plane; this is not strictly necessary for an accurate ray trace but does make the results somewhat easier to visualize in this example.

Figure 7 shows the spot diagrams with the low and high powered beams at the best focus plane. As seen in Figure 6, the best focus planes are at 206.46 mm and 204.22 mm for the low and high powered beams respectively. The RMS spot size on these planes are similar.

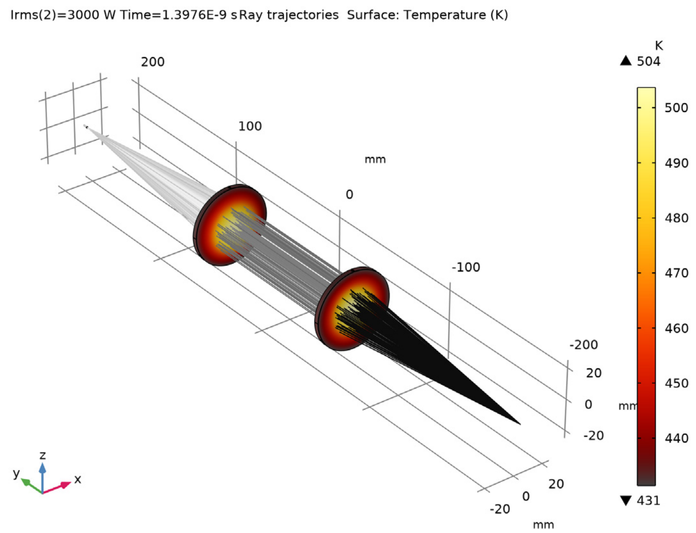

Figure 1: Ray trajectories and surface temperature for the 3 kW source case.

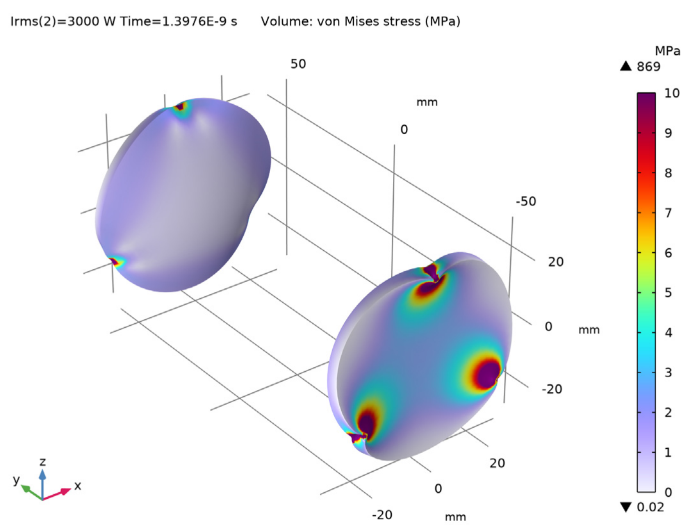

Figure 2: Von Mises stress and deformation of the lenses when illuminated by the 3 kW source.

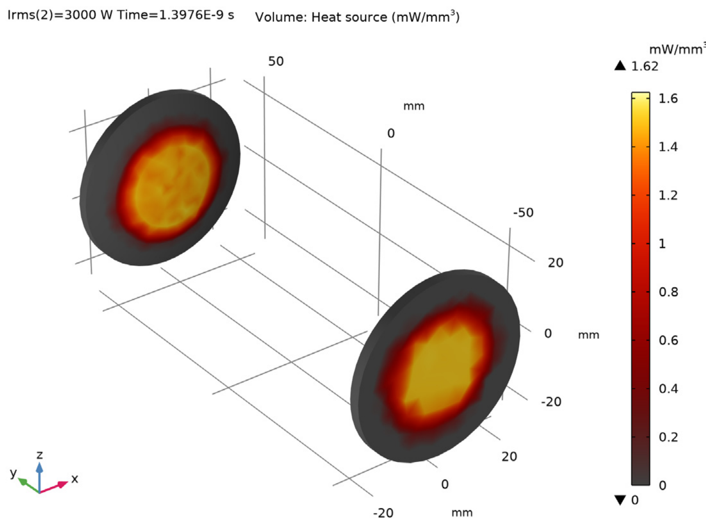

Figure 3: Volumetric heat source in the lenses due to attenuation of the 3 kW beam.

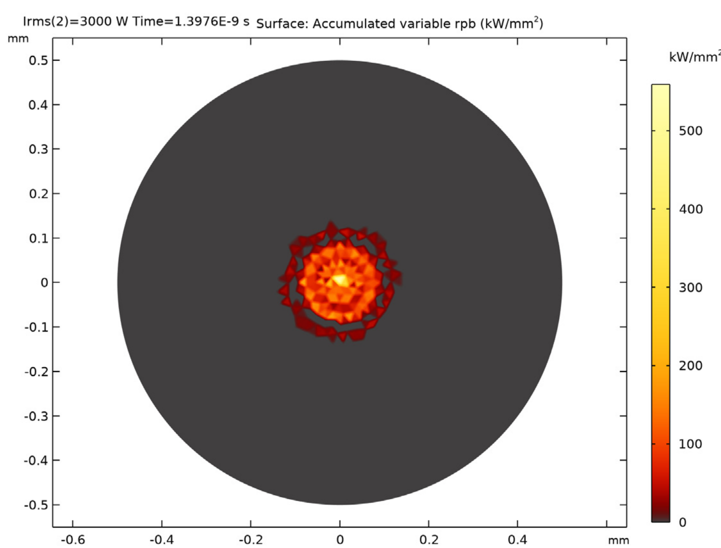

Figure 4: Boundary heat source generated in the focal plane by the 3 kW beam.

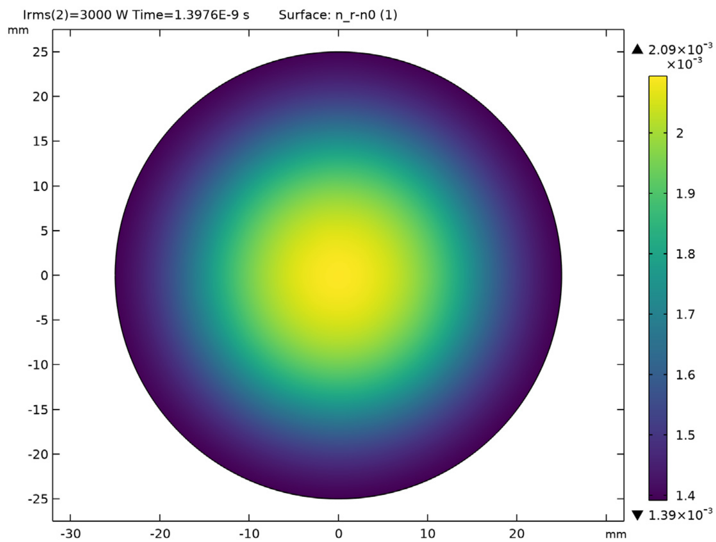

Figure 5: Change in the refractive index of the lens due to the 3 kW beam.

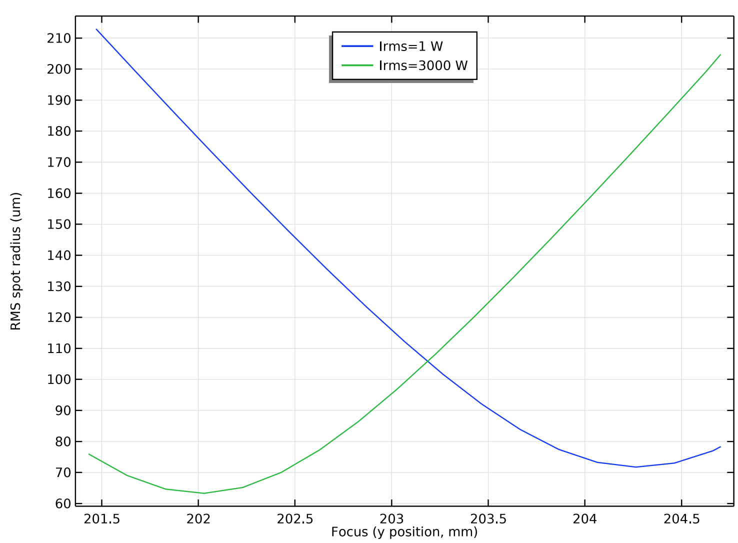

Figure 6: RMS spot radius as a function of focal plane position.

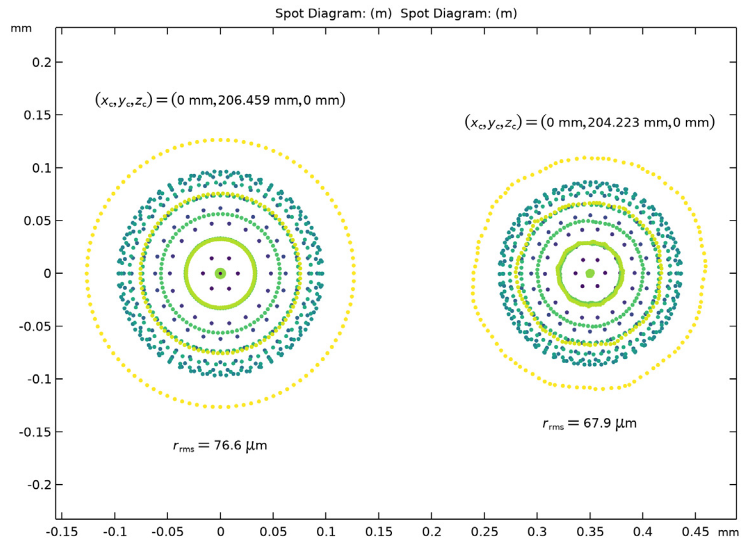

Figure 7: Spot diagram for the 1W (left) and 3 kW (right) sources.

Reference

1. O. Maerten, R. Kramer, H. Schwede, S. Wolf, and V. Brandl, "The Characterization of Focusing Systems for High-Power Lasers with High Beam Quality," *Laser+ Photonics*, pp. 60–64, 2009.

Application Library path: `Ray_Optics_Module/Structural_Thermal_Optical_Performance_Analysis/thermally_induced_focal_shift`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Optics>Ray Optics>Ray Heating**.
3 Click **Add**. This will add interfaces for **Geometrical Optics** and **Heat Transfer in Solids** and a **Ray Heat Source** multiphysics coupling.
4 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Geometrical Optics>Bidirectionally Coupled Ray Tracing**.
8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Load the parameters for the geometry and physics setup from a file.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `thermally_induced_focal_shift_parameters.txt`.

COMPONENT 1 (COMP1)

1 In the **Model Builder** window, click **Component 1 (comp1)**.
2 In the **Settings** window for **Component**, locate the **Curved Mesh Elements** section.
3 From the **Geometry shape function** list, choose **Cubic Lagrange**. The ray tracing algorithm used by the Geometrical Optics interface computes the refracted ray direction based on a discretized geometry via the underlying finite element mesh. A cubic geometry shape

order usually introduces less discretization error compared to the default, which uses linear and quadratic polynomials.

GEOMETRY 1

Select a more appropriate length unit for the geometry.

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

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

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

PART LIBRARIES

Load the Spherical Lens 3D part from the built-in Part Library for the Ray Optics Module.

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **Ray Optics Module>3D>Spherical Lenses> spherical_lens_3d** in the tree.

3 Click **Add to Geometry**.

4 In the **Select Part Variant** dialog box, select **Specify clear aperture diameter** in the **Select part variant** list.

5 Click **OK**.

GEOMETRY 1

Spherical Lens 3D 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Spherical Lens 3D 1 (pi1)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
R1	R	68.8 mm	Radius of curvature, surface 1 (+ convex/-concave)
R2	0	0 m	Radius of curvature, surface 2 (- convex/+concave)
Tc	Tc	7.7029 mm	Center thickness
d0	d	50 mm	Lens full diameter
d1	0	0 m	Diameter, surface 1
d2	0	0 m	Diameter, surface 2

Name	Expression	Value	Description
d1_clear	0	0 m	Clear aperture diameter, surface 1
d2_clear	0	0 m	Clear aperture diameter, surface 2
nix	0	0	Local optical axis, x-component
niy	-1	-1	Local optical axis, y-component
niz	0	0	Local optical axis, z-component

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **yw** text field, type `-dis/2`.

5 Click to expand the **Boundary Selections** section. Click to select row number 2 in the table.

6 Click **New Cumulative Selection**.

7 In the **New Cumulative Selection** dialog box, type `Clear Apertures` in the **Name** text field.

8 Click **OK**.

Add cylinders to the geometry to create three circular boundaries along the perimeter of each lens. These surfaces will be used to apply fixed constraints when modeling the thermal expansion.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `0.75`.

4 In the **Height** text field, type `20`.

5 Locate the **Position** section. In the **y** text field, type `-58`.

6 In the **z** text field, type `10`.

Rotate 1 (rot1)

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

2 Select the object **cyl1** only.

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

4 In the **Angle** text field, type `120`.

5 From the **Axis type** list, choose **y-axis**.

6 Locate the **Input** section. Select the **Keep input objects** check box.

Rotate 2 (rot2)

1 Right-click **Rotate 1 (rot1)** and choose **Duplicate**.

2 Select the object **cyl1** only.

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

4 In the **Angle** text field, type `-120`.

Use the **Partition Objects** node to create surfaces where the cylinders intersect the lens.

Partition Objects 1 (par1)

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

2 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

3 Find the **Objects to partition** subsection. Click to select the **Activate Selection** toggle button.

4 Select the object **pi1** only.

5 Find the **Tool objects** subsection. Click to select the **Activate Selection** toggle button.

6 Select the objects **cyl1**, **rot1**, and **rot2** only.

Use the **Union** operation to remove some interior boundaries that are no longer needed.

Union 1 (uni1)

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

2 Select the object **par1** only.

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

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

Now, create selections to be used when defining physics features.

Exposed Lens Surfaces

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

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

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

4 On the object **uni1**, select Boundaries 1–4, 7, and 8 only.

Fixed Lens Surfaces

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

2 In the **Settings** window for **Complement Selection**, type `Fixed Lens Surfaces` 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, select **Exposed Lens Surfaces** in the **Selections to invert** list.

6 Click **OK**.

Create the focusing lens, which is a mirror image of the collimating lens.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Plane of Reflection** section. In the **x** text field, type `1`.

6 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type `-1`.

7 In the **z** text field, type `0`.

PART LIBRARIES

Create a small surface near the nominal (low power) focal plane. The Circular Planar Annulus from the built-in Part Library for the Ray Optics Module can be used. This surface will be finely meshed to resolve the deposited power.

1 In the **Geometry** toolbar, click **Parts** and choose **Part Libraries**.

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

3 In the **Part Libraries** window, select **Ray Optics Module>3D>Apertures and Obstructions>circular_planar_annulus** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Target

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Circular Planar Annulus 1 (pi2)**.

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

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

Name	Expression	Value	Description
d0	1.0[mm]	1 mm	Diameter, outer
d1	0	0 m	Diameter, inner
nix	0	0	Local optical axis, x-component
niy	1	1	Local optical axis, y-component
niz	0	0	Local optical axis, z-component

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **yw** text field, type `dis/2+Tc+bfl`.

5 Locate the **Boundary Selections** section. In the table, select the **Keep** check box for **All**.

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

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

DEFINITIONS

Load the variable definitions from a text file. These variables define the temperature dependence of the refractive index and are used during postprocessing of the results.

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `thermally_induced_focal_shift_variables.txt`.

The nonzero value of `dndT` makes the refractive index temperature dependent; this is by far the largest contributor to the focal shift in this model. By setting `dndT` to zero it is possible to isolate the effect of thermal deformation on the focal position.

GEOMETRICAL OPTICS (GOP)

Now, set up the ray releases and optical material properties.

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

2 In the **Settings** window for **Geometrical Optics**, locate the **Ray Release and Propagation** section.

3 In the **Maximum number of secondary rays** text field, type `0`. Since an anti-reflective coating will be applied to the lens surfaces, it is not necessary to allocate secondary rays to model the reflection of stray light by the lens system.

Material Discontinuity 1

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

2 In the **Settings** window for **Material Discontinuity**, locate the **Coatings** section.

3 From the **Thin dielectric films on boundary** list, choose **Anti-reflective coating**.

This idealized antireflective coating sets the reflectance to zero for all incident rays.

Ray Properties 1

1 In the **Model Builder** window, click **Ray Properties 1**.

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

3 In the λ_0 text field, type `lam`.

Release from Grid 1

1 In the **Physics** toolbar, click **Global** and choose **Release from Grid**.

2 In the **Settings** window for **Release from Grid**, locate the **Initial Coordinates** section.

3 In the $q_{y,0}$ text field, type `fiber_pos`.

4 Locate the **Ray Direction Vector** section. From the **Ray direction vector** list, choose **Conical**.

5 From the **Conical distribution** list, choose **Hexapolar**.

6 In the N_θ text field, type `18`. A conical hexapolar distribution with 18 rings will release 1027 rays.

7 Specify the $\mathbf{r}$ vector as

0	x
1	y
0	z

8 In the α text field, type `theta`.

9 Locate the **Total Source Power** section. In the P_{src} text field, type `Irms`.

Create a **Wall** boundary condition to stop rays as they reach the focal plane and compute the deposited ray power.

Wall 1

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

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

3 From the **Selection** list, choose **All (Target)**.

4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Pass through**. Allow the rays to pass through the target so that the location of the best focus planes can be computed.

Accumulator 1

1 In the **Physics** toolbar, click **Attributes** and choose **Accumulator**. The built-in accumulator for **Deposited Ray Power** cannot be used because the rays pass through the target.

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

3 In the R text field, type `gop.Q`.

4 Locate the **Units** section. Click **Custom Unit**.

5 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	`W/m^2`

HEAT TRANSFER IN SOLIDS (HT)

Next, set up boundary conditions for the temperature computation. Apply natural convection to the exposed surfaces of each lens.

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 **Cubic Lagrange**. A cubic shape order usually introduces less discretization error compared to the default.

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 **Exposed Lens Surfaces**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `10`.

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

SOLID MECHANICS (SOLID)

Now set up the boundary conditions for the Solid Mechanics interface. Each lens is fixed in place at three locations and is subjected to thermal expansion.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
2. In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.
3. From the **Displacement field** list, choose **Cubic Lagrange**. The ray tracing will now be done on a deformed mesh. In order to reduce the discretization error a cubic shape order should be used.

Fixed Constraint 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2. In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Fixed Lens Surfaces**.

MULTIPHYSICS

Finally, add the multiphysics coupling for thermal expansion between the **Heat Transfer in Solids** and **Solid Mechanics** interfaces.

Thermal Expansion 1 (te1)

1. In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Thermal Expansion**.
2. Click in the **Graphics** window and then press Ctrl+A to select both domains.
3. In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.
4. Click **Go to Source** for **Volume reference temperature**.

GLOBAL DEFINITIONS

Default Model Inputs

1. In the **Model Builder** window, under **Global Definitions** click **Default Model Inputs**.
2. In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.
3. Find the **Expression for remaining selection** subsection. In the **Volume reference temperature** text field, type `T0`.

   Note that, after adding the **Thermal Expansion** node, the ray trajectories are still computed in the undeformed geometry. To make the rays interact with the deformed surfaces of the lenses, it is important to select the **Include geometric nonlinearity** check box, described in the instructions for setting up the **Study 1** node.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Silica glass**.

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

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

MATERIALS

Silica glass (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	n	1	Refractive index

MESH 1

Revise the default mesh to improve the resolution on the target focal plane and on the lens surfaces.

Size 1

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

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

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

4 From the **Selection** list, choose **All (Target)**.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `0.025`.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

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

3 From the **Selection** list, choose **Fixed Lens Surfaces**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `0.5`.

Free Triangular 2

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

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

3 From the **Selection** list, choose **Clear Apertures**.

Size 1

1 Right-click **Free Triangular 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `3.0`.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

STUDY 1

Step 1: Bidirectionally Coupled Ray Tracing

1 In the **Model Builder** window, under **Study 1** click **Step 1: Bidirectionally Coupled Ray Tracing**.

2 In the **Settings** window for **Bidirectionally Coupled Ray Tracing**, locate the **Study Settings** section.

3 From the **Time-step specification** list, choose **Specify maximum path length**.

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

5 In the **Lengths** text field, type `0 400 range(414,0.2,419)`. By using smaller optical path length intervals in the vicinity of the focal plane it will be easier to observe where the mean radial displacement of the rays reaches a minimum.

6 Select the **Include geometric nonlinearity** check box. When this check box is selected, rays are traced through the deformed geometry in which thermal expansion has been taken into account. If this check box is cleared, the temperature dependence of the refractive index still affects the ray trajectories, but the thermal expansion has no effect.

7 Locate the **Iterations** section. In the **Number of iterations** text field, type 3.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Irms (Root mean square power of the source)	1 3000	W

The first parameter value results in a very small change in temperature and a negligibly small focal shift. The larger value shows a substantial focal shift.

Set manual scaling for the displacement field components to improve convergence during the first iteration.

Solution 1 (sol1)

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

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

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 2** node, then click **Displacement field (comp1.u)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

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

RESULTS

Study 1/Parametric Solutions 1 (sol2)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/ Parametric Solutions 1 (sol2)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Clear Apertures**. This will be used to limit the results of some plots to the convex surfaces of the two lenses.

Ray Trajectories (gop)

1 In the **Model Builder** window, under **Results** click **Ray Trajectories (gop)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Select the **Show maximum and minimum values** check box.

4 Select the **Show units** check box.

5 In the **Model Builder** window, expand the **Ray Trajectories (gop)** node.

Color Expression 1

1 In the **Model Builder** window, expand the **Results>Ray Trajectories (gop)>Ray Trajectories 1** node, then click **Color Expression 1**.

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

3 In the **Expression** text field, type `gop.Q`.

4 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

5 Click to expand the **Range** section. Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayPrint**.

6 From the **Color table transformation** list, choose **Reverse**.

Filter 1

. Use the **Filter** node to plot only a fraction of the rays, making them easier to see.

1 In the **Model Builder** window, click **Filter 1**.

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

3 From the **Rays to render** list, choose **Fraction**.

4 In the **Fraction of rays** text field, type `0.1`.

Add a **Surface** plot to view the temperature along with the ray trajectories.

Surface 1

1 In the **Model Builder** window, right-click **Ray Trajectories (gop)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Temperature>T - Temperature - K**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayBody**.

4 In the **Ray Trajectories (gop)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar. The plot should now look like Figure 1.

Stress (solid)

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

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

3 Clear the **Plot dataset edges** check box.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

5 Select the **Show units** check box.

Volume 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.

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

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

4 Click to expand the **Range** section. Specify a manual color range to make the von Mises stress easier to see.

5 Select the **Manual color range** check box.

6 In the **Minimum** text field, type `0`.

7 In the **Maximum** text field, type `10`.

8 In the **Stress (solid)** toolbar, click **Plot**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar. The plot should now look like Figure 2.

Create a plot of the deposited power in the lenses.

Deposited Ray Power (lenses)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Deposited Ray Power (lenses)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

6 Select the **Show units** check box.

Volume 1

1 Right-click **Deposited Ray Power (lenses)** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Heating and losses>rhs1.Qsrc - Heat source - W/m³**.

3 Locate the **Expression** section. In the **Unit** field, type `mW/mm^3`.

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

5 From the **Color table transformation** list, choose **Nonlinear**.

6 Set the **Color calibration parameter** value to **1.5**.

7 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

8 In the **Deposited Ray Power (lenses)** toolbar, click **Plot**.

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

Next, plot the deposited ray power in the focal plane.

Surface 1

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Selection** section. From the **Selection** list, choose **All (Target)**.

Deposited Ray Power (target)

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Deposited Ray Power (target)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Surface 1**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

1 Right-click **Deposited Ray Power (target)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Geometrical Optics>Accumulated variables>**

Accumulated variable comp1.gop.wall1.bacc1.rpb>gop.wall1.bacc1.rpb - Accumulated variable rpb - W/m².

3 Locate the **Expression** section. In the **Unit** field, type `kW/mm^2`.

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

5 From the **Color table transformation** list, choose **Nonlinear**.

6 Set the **Color calibration parameter** value to **-1**.

7 In the **Deposited Ray Power (target)** toolbar, click **Plot**. The plot should now look like Figure 4.

Create another **Surface** dataset to plot the change in the refractive index over one of the lens surfaces.

Surface 2

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Select Boundary 7 only.

Refractive Index

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Refractive Index` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Surface 2**.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 Right-click **Refractive Index** and choose **Surface**.

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

3 In the **Expression** text field, type `n_r-n0`.

4 Click to expand the **Title** section. Locate the **Coloring and Style** section. From the **Color table** list, choose **Viridis**.

5 In the **Refractive Index** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar. The plot should now look like Figure 5.

Next, plot the RMS spot size as a function of time around the focal plane.

Spot Size

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Ray 1**.

4 From the **Time selection** list, choose **Manual**.

5 Click **Range**.

6 In the **Integer Range** dialog box, type `3` in the **Start** text field.

7 In the **Stop** text field, type `28`.

8 Click **Replace**.

Global 1

1 Right-click **Spot Size** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`gop.rrms`	`um`	`RMS ray radial position, relative to average over rays`

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

6 In the **Expression** text field, type `gop.qavey`. This is the average position along the optical axis at each time step.

Spot Size

1 In the **Model Builder** window, click **Spot Size**.

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

3 Select the **x-axis label** check box.

4 Select the **y-axis label** check box.

5 In the **x-axis label** text field, type `Focus (y position, mm)`.

6 In the **y-axis label** text field, type `RMS spot radius (um)`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

8 In the **Spot Size** toolbar, click **Plot**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar. The plot should now look like Figure 6.

Ray 1

In the following steps a spot diagrams at each of the power settings will be created. First, create ray datasets for the solutions to the parameter sweep. This will make it possible to use the **Spot Diagram** to automatically calculate the plane of best focus.

Ray 2

1 In the **Model Builder** window, under **Results>Datasets** right-click **Ray 1** and choose **Duplicate**.

2 In the **Settings** window for **Ray**, locate the **Ray Solution** section.

3 From the **Solution** list, choose **Irms=1 (sol3)**.

Ray 3

1 Right-click **Ray 2** and choose **Duplicate**.

2 In the **Settings** window for **Ray**, locate the **Ray Solution** section.

3 From the **Solution** list, choose **Irms=3000 (sol4)**.

Spot Diagrams

Now, create the spot diagrams.

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Spot Diagrams` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 Locate the **Color Legend** section. Clear the **Show legends** check box.

Spot Diagram 1

1 In the **Spot Diagrams** toolbar, click **More Plots** and choose **Spot Diagram**.

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

3 From the **Image surface** list, choose **Ray 2**.

4 Click to expand the **Focal Plane Orientation** section. Click **Create Focal Plane Dataset**. This will create an **Intersection Point 3D** dataset on the plane that minimizes the RMS spot size.

5 Click to expand the **Annotations** section. Select the **Show spot coordinates** check box.

6 From the **Coordinate system** list, choose **Global**.

7 In the **Display precision** text field, type `6`.

Color Expression 1

1 Right-click **Spot Diagram 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `at(0,gop.phic)`.

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

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Viridis**.

Spot Diagram 2

1 In the **Model Builder** window, under **Results>Spot Diagrams** right-click **Spot Diagram 1** and choose **Duplicate**.

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

3 From the **Image surface** list, choose **Ray 3**.

4 Locate the **Focal Plane Orientation** section. Click **Create Focal Plane Dataset**. As before, this action will create another **Intersection Point 3D** dataset.

5 Click to expand the **Position** section. In the **x** text field, type `0.35`.

6 In the **Spot Diagrams** toolbar, click **Plot**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar. The plot should now look like Figure 7.