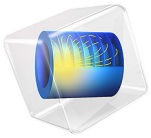


Model created in COMSOL Multiphysics 6.4

Topology Optimization of a Metalens

Introduction

A metalens is a flat lens that utilizes subwavelength features to focus electromagnetic waves. They are popular for applications where space constraints prevent the use of conventional lenses.

This model demonstrates how to design a metalens to focus a Gaussian beam to a point using topology optimization. A narrow band is considered, but the methodology can be extended to design for broadband performance instead. Furthermore, axisymmetry is assumed and a special manufacturing constraint is imposed, but this manufacturing constraint can be relaxed for applications involving in-plane propagation.

Model Definition

Figure 1 shows the initial geometry used for the topology optimization.

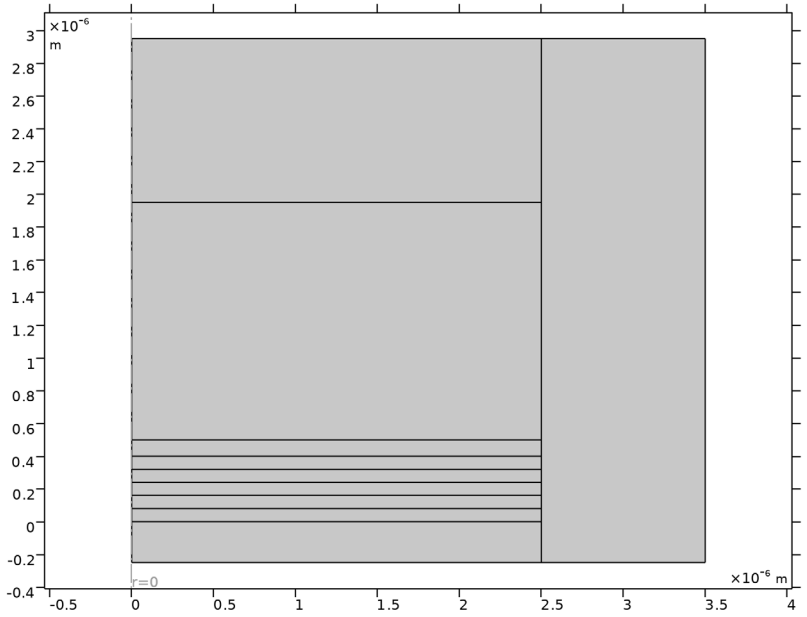


Figure 1: The initial geometry with the axisymmetry line to the left and the design domain as the second lowest rectangle.

The design is defined implicitly using a Control Variable Field feature by letting the refractive index depend on the design variable, θ:

$$\begin{aligned} \underline{n} &= n + ik \\ &= 1 + (n_{\text{lens}} - 1)\theta_p \\ &+ i\alpha(1 - \theta_p)\theta_p \end{aligned}$$

where α, is a damping coefficient that can cause loss of energy for intermediate variables. It can thus also be thought of as implicit boundary (*Tikhonov*) penalization. The manufacturing constraint is intended to prevent designs that are incompatible with planar processing. For this reason, planar freedom in a discrete number of layers is allowed, and the layers are coupled to prevent overhang. This is implemented through control variables at the bottom edge of each layer and the variables are coupled using a General Extrusion operator:

$$\begin{aligned} \theta_f &= L_{\text{min}}^2 \nabla^2 \theta_f + \theta_c \\ \theta_m^i &= \begin{cases} \theta_f & , i = 0 \\ 1 - (1 - \theta_f)(1 - \theta_m^{i-1}) & , i > 0 \end{cases} \end{aligned}$$

The model also uses projection in the way described in the model Topology Optimization of an MBB Beam.

Results and Discussion

The model uses performs three optimizations with increasing values of the projection slope and decreasing values of the attenuation damping. different initial values. A **Segregated** solver is used to improve robustness and GCMMA is used with an iteration limit of 50. The amount of intermediate design variables decreases as the optimization progresses, but it does not complete disappear as shown in Figure 2.

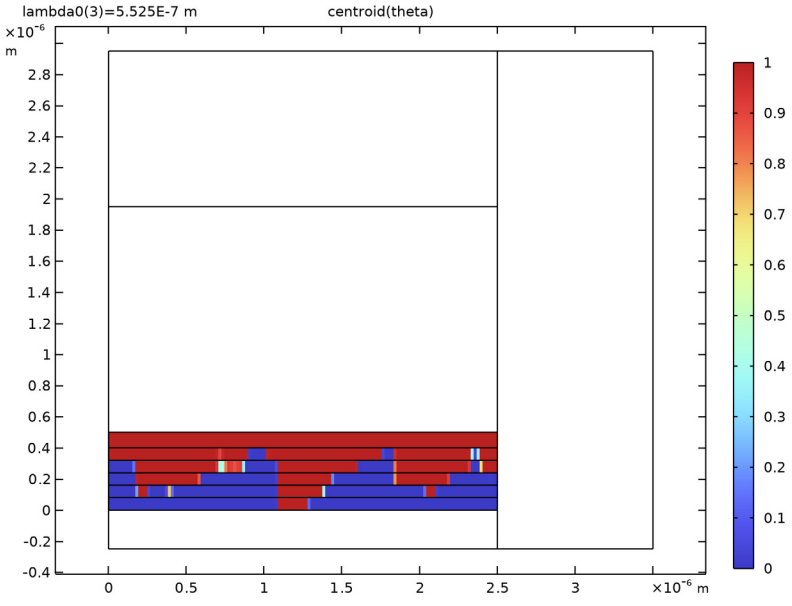


Figure 2: The final topology optimization result is not fully discrete.

Then the result is transferred to a body-fitted representation and shape optimization is performed using the **Transformation** feature so that the manufacturing constraint remains respected as illustrated in Figure 4.

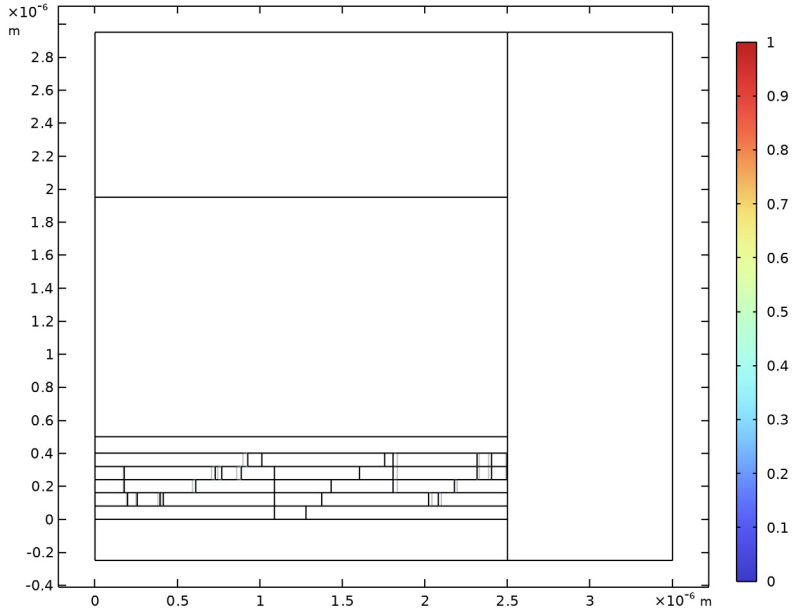


Figure 3: The shape optimization is only allowed to change the design marginally.

Afterward, a verification is performed, which reveals poor performance for wavelengths different from those considered in the optimization as shown in Figure 4.

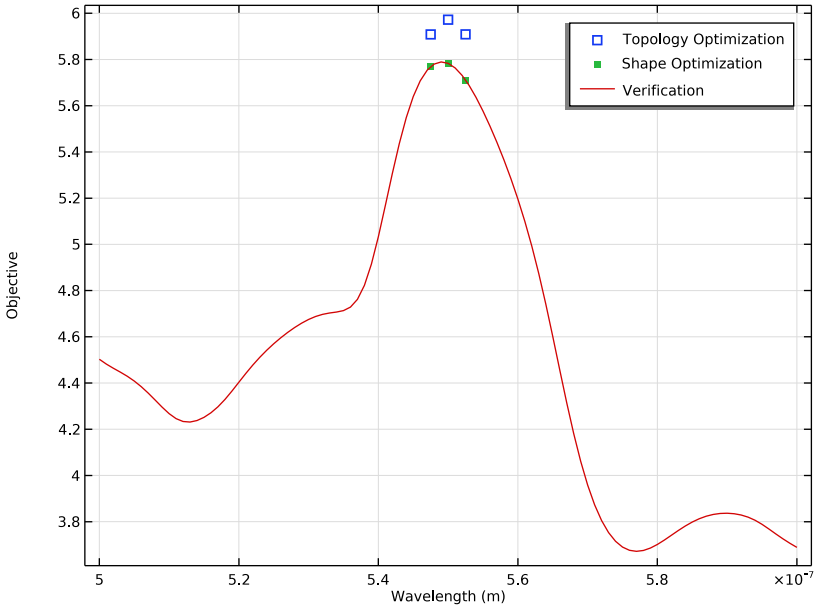


Figure 4: The objective function as a function of the wavelength for all three studies, showing good agreement between the implicit and explicit design representations.

Notes About the COMSOL Implementation

The model uses a **Parametric Sweep** to implement a continuation in the both the damping coefficient and, α, and projection slope, β. The purpose is to achieve a discrete design with good performance.

Application Library path: `Optimization_Module/Topology_Optimization/metalens_topology_optimization`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Mathematics** > **PDE Interfaces** > **Lower Dimensions** > **Coefficient Form Boundary PDE (cb)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Optics** > **Wave Optics** > **Electromagnetic Waves, Frequency Domain (ewfd)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces** > **Wavelength Domain**.

8 Click **Done**.

GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

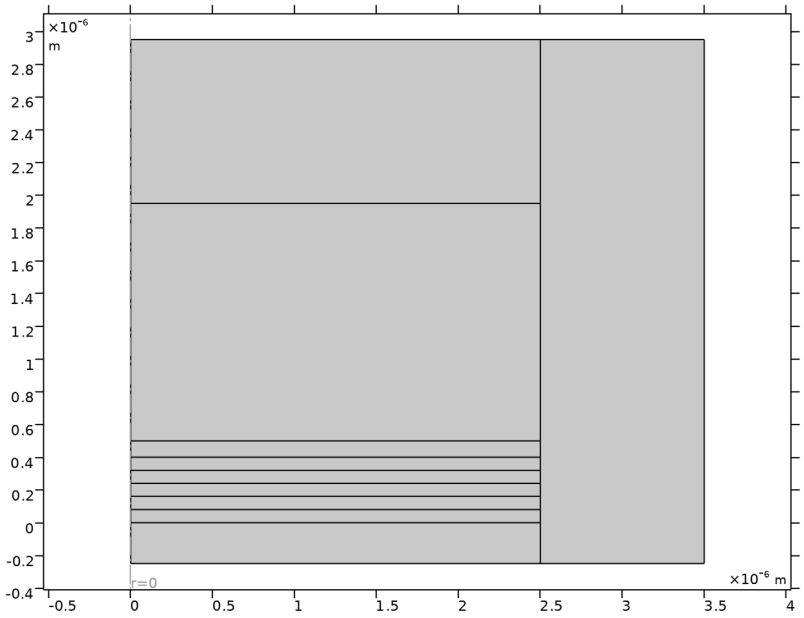
1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

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

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

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

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



The geometry should now look like that in Figure 1.

GLOBAL DEFINITIONS

Geometry Parameters

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

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

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

Name	Expression	Value	Description
`lens_l`	`5[um]`	5E-6 m	`Width of lens`

Parameters 2

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
dlambda	5[nm]	5E-9 m	Wavelength width
lambda2	550[nm]	5.5E-7 m	Center wavelength
lambda1	lambda2-0.5*dlambda	5.475E-7 m	First wavelength
lambda3	lambda2+0.5*dlambda	5.525E-7 m	Last wavelength
n_lens	2.4	2.4	Refractive index of lens material
meshsz	lambda2/n_lens/12	1.9097E-8 m	Mesh size
meshsz2	lambda2/12	4.5833E-8 m	Mesh size for air
beta	8	8	Projection parameter
damping	1	1	Attenuation penalization
w0	0.25*lens_l	1.25E-6 m	Beam width
theta0	0.25	0.25	Initial control

Add a **Control Variable Field** on the edges to avoid vertical variations within each layer.

DEFINITIONS

Control Variable Field 1 (p1)

1 In the **Definitions** toolbar, click **Control Variables** and choose **Control Variable Field**.

2 In the **Settings** window for **Control Variable Field**, type `theta_c` in the **Name** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Control layers**.

5 Locate the **Initial Value** section. In the **Initial value** text field, type `theta0`.

6 Locate the **Bounds** section. Select the **Use bounds** checkbox.

7 In the **Upper bound** text field, type `1`.

8 Locate the **Discretization** section. From the **Shape function type** list, choose **Discontinuous Lagrange**.

9 From the **Element order** list, choose **Constant**.

COEFFICIENT FORM BOUNDARY PDE (CB)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Coefficient Form Boundary PDE (cb)**.

2 In the **Settings** window for **Coefficient Form Boundary PDE**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Control layers**.

4 Locate the **Units** section. Click **Select Source Term Quantity**.

5 In the **Physical Quantity** dialog, select **General** > **Dimensionless (1)** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Coefficient Form Boundary PDE**, click to expand the **Discretization** section.

8 From the **Element order** list, choose **Linear**.

9 From the **Frame** list, choose **Material**.

10 Click to expand the **Dependent Variables** section. In the **Field name (1)** text field, type `theta_f`.

11 Click **Add Dependent Variable**.

12 In the **Dependent variables (1)** table, enter the following settings:

`theta_f`
`theta_m`

Coefficient Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Coefficient Form Boundary PDE (cb)** click **Coefficient Form PDE 1**.

2 In the **Settings** window for **Coefficient Form PDE**, locate the **Diffusion Coefficient** section.

3 In the c text-field array, type `(2*meshsz)^2` in the first column of the first row.

4 In the c text-field array, type `0` in the second column of the second row.

5 Locate the **Absorption Coefficient** section. In the a text-field array, type `1` in the first column of the first row.

6 In the a text-field array, type `1` in the second column of the second row.

7 Locate the **Source Term** section. In the f text-field array, type `theta_c` on the first row.

8 In the f text-field array, type `if(Z<eps,theta_f,1-(1-theta_f)*(1-genext1(theta_m)))` on the second row, so that the value of `theta_m` always increases going up the layers, because `genext1` refers to the layer below.

9 Locate the **Damping or Mass Coefficient** section. In the d_a text-field array, type `0` in the first column of the first row.

10 In the d_a text-field array, type `0` in the second column of the second row.

DEFINITIONS

General Extrusion 1 (genext1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **General Extrusion**.

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

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

4 From the **Selection** list, choose **Control layers**.

5 Locate the **Destination Map** section. In the **r-expression** text field, type `R`.

6 In the **z-expression** text field, type `Z-(lens_H-lens_minH)/layers`.

7 Locate the **Source** section. From the **Source frame** list, choose **Material (R, PHI, Z)**.

General Extrusion 2 (genext2)

1 Right-click **General Extrusion 1 (genext1)** and choose **Duplicate**.

2 In the **Settings** window for **General Extrusion**, locate the **Destination Map** section.

3 In the **Z-expression** text field, type `floor(Z/(lens_H-lens_minH)*layers)*(lens_H-lens_minH)/layers`.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

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

4 From the **Selection** list, choose **Rectangle 4**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
`theta`	`(tanh(beta*(genext2(theta_m)-0.5))+tanh(0.5*beta))/2/tanh(0.5*beta)`		`Material volume factor`
`theta_p`	`theta^3`		`Penalized material volume factor`

Variables 2

1 Right-click **Variables 1** and choose **Duplicate**.

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

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

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

Name	Expression	Unit	Description
theta	1		Material volume factor

Variables 3

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

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

4 Select Boundary 4 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
dummy	1		

Variables 4

1 Right-click **Variables 3** and choose **Duplicate**.

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

3 Click  **Clear Selection**.

4 Select Boundaries 6, 8, 10, and 12 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
dummy2	2		

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

3 In the **Material properties** tree, select **Electromagnetic Models** > **Refractive Index** > **Refractive index, imaginary part (ki)**.

4 Right-click and choose **Add This Property Group to Material**.

5 In the **Material properties** tree, select **Electromagnetic Models** > **Refractive Index** > **Refractive index, real part (n)**.

6 Right-click and choose **Add This Property Group to Material**.

7 Locate the **Material Contents** section. 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_lens	1	Refractive index

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Air**.

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

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

MATERIALS

Material Link 1 (matlnk1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials** > **Material Link**.

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

3 From the **Selection** list, choose **Rectangle 1**.

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

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

3 From the **Selection** list, choose **Rectangle 1**.

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

Property	Variable	Value	Unit	Property group
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	1+(n_lens-1)*theta_p	1	Refractive index
Refractive index, imaginary part	ki_iso ; kiii = ki_iso, kiij = 0	damping*theta*(1-theta)	1	Refractive index

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (ewfd)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Out-of-Plane Wave Number** section.

3 In the m text field, type `-1`.

4 Click to expand the **Discretization** section. From the **Electric field** list, choose **Linear**.

Scattering Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

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

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

4 Locate the **Scattering Boundary Condition** section. From the **Incident field** list, choose **Gaussian beam**.

5 In the w_0 text field, type `w0`.

6 Specify the $\mathbf{E}_{g0}$ vector as

1	r
j	phi

7 Specify the $\mathbf{k}_{dir}$ vector as

-ewfd.nr	r

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

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

4 Locate the **Geometry** section. From the **Type** list, choose **Cylindrical**.

5 Locate the **Scaling** section. In the **PML scaling factor** text field, type `2`.

Objective

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.

2 In the **Settings** window for **Point Probe**, type `Objective` in the **Label** text field.

3 In the **Variable name** text field, type `obj`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Point 1**.

5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Electric** > **ewfd.normE - Electric field norm - V/m**.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click  **Mapped**.

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

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

4 From the **Selection** list, choose **Rectangle 1**.

Size 1

1 Right-click **Mapped 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.

5 Select the **Maximum element size** checkbox. In the associated text field, type `meshsz`.

Size

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

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 In the **Maximum element size** text field, type `meshsz2`.

Free Triangular 1

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

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

STUDY 1

Step 1: Wavelength Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Wavelength Domain**.

2 In the **Settings** window for **Wavelength Domain**, locate the **Study Settings** section.

3 From the **Wavelength unit** list, choose **m**.

4 In the **Wavelengths** text field, type `lambda1 lambda2 lambda3`.

5 Click to expand the **Results While Solving** section. From the **Probes** list, choose **None**.

Topology Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Topology Optimization**.

2 In the **Settings** window for **Topology Optimization**, locate the **Optimization Solver** section.

3 In the **Maximum number of iterations** text field, type `50`.

4 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **comp1.obj - Objective - V/m**.

5 Locate the **Objective Function** section. From the **Type** list, choose **Maximization**.

6 From the **Solution** list, choose **Minimum of objectives**.

7 Find the **Objective settings** subsection. From the **Objective scaling** list, choose **Manual**.

8 In the **Scale** text field, type `10`.

9 Click to expand the **Output** section. From the **Probes** list, choose **None**.

10 In the **Study** toolbar, click **Get Initial Value** to initialize the solution and the default plots. Afterward, you can set up a segregated solver for improved robustness and speed. Moreover, you can show the default topology optimization plot while optimizing.

RESULTS

Line 1

1 In the **Model Builder** window, expand the **Coefficient Form Boundary PDE** node.

2 Right-click **Line 1** and choose **Disable**.

Topology Optimization

1 In the **Model Builder** window, under **Results** click **Coefficient Form Boundary PDE**.

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

3 Click to expand the **Quality** section. From the **Smoothing** list, choose **None**.

Surface 1

1 Right-click **Topology Optimization** and choose **Surface**.

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

3 In the **Expression** text field, type `centroid(theta)`.

4 Click to expand the **Range** section. Select the **Manual color range** checkbox.

5 In the **Maximum** text field, type `1`.

Set up a segregated solver for improved robustness and speed.

STUDY 1

Solver Configurations

In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** node.

Solution 1 (sol1)

1. In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Optimization Solver 1** node.
2. Right-click **Stationary Solver 1** and choose **Segregated**.
3. Right-click **Segregated 1** and choose **Segregated Step** twice.
4. In the **Settings** window for **Segregated Step**, locate the **General** section.
5. Under **Variables**, click **Add**.
6. In the **Add** dialog, in the **Variables** list, choose **Electric Field (comp1.E)** and **Electric Field out of Plane (comp1.ewfd.Eoop)**.
7. Click **OK**.
8. In the **Model Builder** window, click **Segregated Step 1**.
9. In the **Settings** window for **Segregated Step**, locate the **General** section.
10. Under **Variables**, click **Add**.
11. In the **Add** dialog, in the **Variables** list, choose **Control Variable Field (comp1.theta_c)** and **Dependent Variable Theta_m (comp1.theta_m)**.
12. Click **OK**.
13. In the **Model Builder** window, click **Segregated Step**.
14. In the **Settings** window for **Segregated Step**, locate the **General** section.
15. In the **Variables** list, choose **Electric Field (comp1.E)**, **Electric Field out of Plane (comp1.ewfd.Eoop)**, and **Dependent Variable Theta_m (comp1.theta_m)**.
16. Under **Variables**, click **Delete**.

Topology Optimization

1. In the **Model Builder** window, under **Study 1** click **Topology Optimization**.
2. In the **Settings** window for **Topology Optimization**, locate the **Output** section.
3. Select the **Plot** checkbox.
4. In the table, enter the following settings:

Plot group	Plot window
Topology Optimization	Graphics

5 In the **Model Builder** window, click **Study 1**.

6 In the **Settings** window for **Study**, type `Study 1: Topology Optimization` in the **Label** text field.

7 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox to avoid more plots.

One can find better designs by performing a continuation in the damping and projection slope parameters using a **Parametric Sweep**.

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** twice.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
beta (Projection parameter)	`8 16 32`	
damping (Attenuation penalization)	`0.8 0.4 0`	

5 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

6 Click to expand the **Advanced Settings** section. Select the **Reuse solution from previous step** checkbox.

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

RESULTS

Topology Optimization

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

2 In the **Topology Optimization** toolbar, click **Plot**.

Filter 1

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

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

3 In the **Expression** text field, type `centroid(genext2(theta_m))*(R<0.5*lens_l)`.

4 Locate the **Filter** section. In the **Lower bound** text field, type `0.5`.

5 Locate the **Evaluation** section. From the **Smoothing** list, choose **None**.

6 Clear the **Use derivatives** checkbox.

Filter 2

1 Right-click **Filter 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type `1`.

4 Click **Plot**.

5 Right-click **Filter 2** and choose **Create Mesh Part**.

MESH PART 1

Import 2

1 In the **Model Builder** window, under **Global Definitions** > **Mesh Parts** > **Mesh Part 1** right-click **Import 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Filter 1**.

4 Click **Import**.

5 Click **Build Selected**.

Merge each domain group by deleting the interior boundaries, as this allows better selection logic for the shape optimization.

Adjacent Selection 1

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

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

3 From the **Selection** list, choose **Rectangle 1(1)**.

4 Locate the **Output Entities** section. Select the **Interior boundaries** checkbox.

5 Clear the **Exterior boundaries** checkbox.

6 Click **Build Selected**.

Delete Entities 1

1 In the **Mesh** toolbar, click **Entities** and choose **Delete Entities**.

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

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

4 From the **Selection** list, choose **Adjacent Selection 1**.

5 Locate the **Adjacent Entities** section. Clear the **Delete adjacent lower-dimensional entities** checkbox to preserve the corners.

6 Click **Build All**.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **2D Axisymmetric**.

COMPONENT 2: VERIFICATION

In the **Settings** window for **Component**, type `Component 2: Verification` in the **Label** text field.

COMPONENT 1: OPTIMIZATION

1. In the **Model Builder** window, click **Component 1 (comp1)**.
2. In the **Settings** window for **Component**, type `Component 1: Optimization` in the **Label** text field.

GEOMETRY 2

In the **Model Builder** window, under **Component 2: Verification (comp2)** click **Geometry 2**.

Import 1 (imp1)

1. In the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Source** section.
3. From the **Source** list, choose **Mesh**.
4. From the **Mesh** list, choose **Mesh Part 1**.
5. Click **Build Selected**.

Adjacent Selection 1 (adjsel1)

1. In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.
2. In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Click **Add**.
5. In the **Add** dialog, select **z-axis(1) (Import 1)** in the **Input selections** list.
6. Click **OK**.
7. In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.
8. From the **Geometric entity level** list, choose **Adjacent domains**.

Form Union (fin)

1. In the **Model Builder** window, click **Form Union (fin)**.
2. In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Transformation

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

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

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

4 In the **Add** dialog, select **Rectangle 3(1) (Import 1)** in the **Selections to add** list.

5 Click **OK**.

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

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

8 In the **Add** dialog, select **Adjacent Selection 1** in the **Selections to subtract** list.

9 Click **OK**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

In the **Model Builder** window, under **Component 1: Optimization (comp1)** right-click **Electromagnetic Waves, Frequency Domain (ewfd)** and choose **Copy**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD2)

1 In the **Model Builder** window, right-click **Component 2: Verification (comp2)** and choose **Paste Electromagnetic Waves, Frequency Domain**.

2 In the **Messages from Paste** dialog, click **OK**.

Scattering Boundary Condition 1

1 In the **Model Builder** window, expand the **Electromagnetic Waves, Frequency Domain (ewfd2)** node, then click **Scattering Boundary Condition 1**.

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

3 From the **Selection** list, choose **Source (Import 1)**.

ARTIFICIAL DOMAINS, DEFINITIONS (COMP1)

Objective (obj), Perfectly Matched Layer 1 (pml1)

1 In the **Model Builder** window, under **Component 1: Optimization (comp1)** > **Definitions**, Ctrl-click to select **Objective (obj)** and **Artificial Domains** > **Perfectly Matched Layer 1 (pml1)**.

2 Right-click and choose **Copy**.

DEFINITIONS (COMP2)

In the **Model Builder** window, under **Component 2: Verification (comp2)** right-click **Definitions** and choose **Paste Multiple Items**.

Objective (point2)

1. In the **Settings** window for **Point Probe**, type `obj` in the **Variable name** text field.
2. Locate the **Source Selection** section. From the **Selection** list, choose **Point 1 (Import 1)**.
3. Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2: Verification (comp2)** > **Electromagnetic Waves, Frequency Domain** > **Electric** > **ewfd2.normE - Electric field norm - V/m**.

ARTIFICIAL DOMAINS

Perfectly Matched Layer 1 (pml2)

1. In the **Model Builder** window, under **Component 2: Verification (comp2)** > **Definitions** > **Artificial Domains** click **Perfectly Matched Layer 1 (pml2)**.
2. In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **PML (Import 1)**.

MATERIALS

Air (mat2), Material Link 1 (matlnk1)

1. In the **Model Builder** window, under **Component 1: Optimization (comp1)** > **Materials**, Ctrl-click to select **Air (mat2)** and **Material Link 1 (matlnk1)**.
2. Right-click and choose **Copy**.

In the **Model Builder** window, under **Component 2: Verification (comp2)** right-click **Materials** and choose **Paste Multiple Items**.

Material Link 1 (matlnk2)

1. In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.
2. From the **Selection** list, choose **Rectangle 1(1) (Import 1)**.

Material Link 3 (matlnk3)

1. Right-click **Component 2: Verification (comp2)** > **Materials** > **Material Link 1 (matlnk2)** and choose **Duplicate**.
2. In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Fixed Lens (Import 1)**.

MESH 2

Free Triangular 1

In the **Mesh** toolbar, click  **Free Triangular**.

Size 1

1. Right-click **Free Triangular 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 **Domain**.
4. From the **Selection** list, choose **Rectangle 1(1) (Import 1)**.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** checkbox. In the associated text field, type `meshsz`.
8. Select the **Minimum element size** checkbox. In the associated text field, type `meshsz/2`.

Size

1. In the **Model Builder** window, under **Component 2: Verification (comp2)** > **Mesh 2** click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size Parameters** section.
3. In the **Maximum element size** text field, type `meshsz2`.
4. In the **Minimum element size** text field, type `meshsz2/2`.

COMPONENT 2: VERIFICATION (COMP2)

Free Shape Domain 1

1. In the **Physics** toolbar, click  **Optimization** and choose **Shape Optimization**.
2. In the **Settings** window for **Free Shape Domain**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Rectangle 1 (Import 1)**.

Symmetry/Roller 1

1. In the **Shape Optimization** toolbar, click  **Symmetry/Roller**.
2. In the **Settings** window for **Symmetry/Roller**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Symmetry/Roller (Import 1)**.

Transformation 1

1. In the **Shape Optimization** toolbar, click  **Transformation**.
2. In the **Settings** window for **Transformation**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Rectangle 1(1) (Import 1)**.

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

	Lock	Lower bound (m)	Upper bound (m)
R		-1E-7	1E-7

5 Locate the **Scaling** section. From the **Scaling type** list, choose **Anisotropic**.

6 In the table, clear the **Lock** checkbox for **R**.

Transformation 2

1 Right-click **Transformation 1** and choose **Duplicate**.

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

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

Transformation 1

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

2 In the **Settings** window for **Transformation**, locate the **Translation** section.

3 In the table, select the **Lock** checkbox for **R**.

4 Click to expand the **Center of Scaling and Rotation** section. In the table, enter the following settings: to prevent movement of the points on the axisymmetry line.

Coordinates	Center type	Center of scaling and rotation (m)
Rg	User defined	0

Create a **P-norm** to prevent discourage badly shaped elements.

DEFINITIONS (COMP2)

P-Norm 1 (pnorm1)

1 In the **Definitions** toolbar, click **Physics Utilities** and choose **P-Norm**.

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

3 From the **Selection** list, choose **Rectangle 1 (Import 1)**.

4 Click **Section toolbar** in the upper-right corner of the **P-Norm** section. From the menu, choose **Component 2: Verification (comp2)** > **Frames** > **Transforms** > **comp2.material.detInvF - Volume factor from Material frame to Geometry frame - 1**.

5 Locate the **P-Norm** section. In the a text field, type `comp2.material.detInvF/2`.

6 Locate the **Quadrature Settings** section. From the **Integrate on frame** list, choose **Geometry**.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces** > **Wavelength Domain**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Coefficient Form Boundary PDE (cb)**.

5 Click the **Add Study** button in the window toolbar twice.

6 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Wavelength Domain

1 In the **Settings** window for **Wavelength Domain**, locate the **Study Settings** section.

2 From the **Wavelength unit** list, choose **mil**.

3 In the **Wavelengths** text field, type `lambda1 lambda2 lambda3`.

4 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, clear the checkbox for **Component 1: Optimization (comp1)**.

5 Locate the **Results While Solving** section. From the **Probes** list, choose **None**.

6 In the **Model Builder** window, click **Study 2**.

7 In the **Settings** window for **Study**, type `Study 2: Shape Optimization` in the **Label** text field.

Shape Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

3 In the **Maximum number of iterations** text field, type `25`.

4 In the **Move limits** text field, type `0.2`.

5 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 2: Verification (comp2)** > **Definitions** > **comp2.obj - Objective - V/m**.

6 Locate the **Objective Function** section. From the **Type** list, choose **Maximization**.

7 From the **Solution** list, choose **Minimum of objectives**.

8 Find the **Objective settings** subsection. From the **Objective scaling** list, choose **Manual**.

9 In the **Scale** text field, type `10`.

10 Locate the **Control Variables** section. In the table, clear the **Solve for** checkbox for **Control Variable Field 1 (theta_c)**.

11 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 2: Verification (comp2)** > **Definitions** > **comp2.pnorm1 - P-norm**.

12 Locate the **Constraints** section. In the table, enter the following settings:

Expression	Lower bound	Upper bound
comp2.pnorm1		1

13 Click to expand the **Output** section. From the **Probes** list, choose **None**.

14 In the **Study** toolbar, click **Get Initial Value**.

Solver Configurations

In the **Model Builder** window, expand the **Study 2: Shape Optimization** > **Solver Configurations** node.

Solution 6 (sol6)

1 In the **Model Builder** window, expand the **Study 2: Shape Optimization** > **Solver Configurations** > **Solution 6 (sol6)** > **Optimization Solver 1** node.

2 Right-click **Stationary Solver 1** and choose **Segregated**.

3 Right-click **Segregated 1** and choose **Segregated Step**.

4 In the **Settings** window for **Segregated Step**, locate the **General** section.

5 Under **Variables**, click **Add**.

6 In the **Add** dialog, in the **Variables** list, choose **Electric Field (Spatial and Material Frames) (comp2.E)** and **Electric Field out of Plane (Spatial and Material Frames) (comp2.ewfd2.Eoop)**.

7 Click **OK**.

8 In the **Model Builder** window, click **Segregated Step**.

9 In the **Settings** window for **Segregated Step**, locate the **General** section.

10 In the **Variables** list, choose **Electric Field (comp1.E)**, **Electric Field out of Plane (comp1.ewfd.Eoop)**, **Control Variable Field (comp1.theta_c)**, **Dependent Variable Theta_f (comp1.theta_f)**, **Dependent Variable Theta_m (comp1.theta_m)**, **Electric Field (Spatial and Material Frames) (comp2.E)**, and **Electric Field out of Plane (Spatial and Material Frames) (comp2.ewfd2.Eoop)**.

11 Under **Variables**, click **Delete**.

Shape Optimization

1 In the **Model Builder** window, under **Study 2: Shape Optimization** click **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Output** section.

3 Select the **Plot** checkbox.

4 In the table, enter the following settings:

Plot group	Plot window
Shape Optimization	Graphics

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

RESULTS

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

2 In the **Model Builder** window, under **Results** click **Shape Optimization**.

3 In the **Shape Optimization** toolbar, click **Plot**.

STUDY 3

Step 1: Wavelength Domain

1 In the **Model Builder** window, under **Study 3** click **Step 1: Wavelength Domain**.

2 In the **Settings** window for **Wavelength Domain**, locate the **Study Settings** section.

3 In the **Wavelengths** text field, type `range(500,1,600)`.

4 From the **Wavelength unit** list, choose **nm**.

5 Locate the **Results While Solving** section. From the **Probes** list, choose **None**.

6 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, clear the checkbox for **Component 1: Optimization (comp1)**.

7 In the **Solve for** column of the table, under **Component 2: Verification (comp2)**, clear the checkbox for **Deformed Geometry**.

8 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

9 From the **Method** list, choose **Solution**.

10 From the **Study** list, choose **Study 2: Shape Optimization, Wavelength Domain**.

11 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

12 From the **Method** list, choose **Solution**.

13 From the **Study** list, choose **Study 2: Shape Optimization, Wavelength Domain**.

Saving the solution on a selection can reduce the file size dramatically.

14 Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	Output	Selection
Electromagnetic Waves, Frequency Domain (ewfd2)	Selection	

15 In the table, click to select the cell at row number 3 and column number 3.

16 Under **Selections**, click + **Add**.

17 In the **Add** dialog, select **Point 1 (Import 1)** in the **Selections** list.

18 Click **OK**.

19 In the **Model Builder** window, click **Study 3**.

20 In the **Settings** window for **Study**, locate the **Study Settings** section.

21 Clear the **Generate default plots** checkbox.

22 In the **Label** text field, type `Study 3: Verification`.

23 In the **Study** toolbar, click = **Compute**.

RESULTS

Verification

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

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

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Wavelength (m)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Objective`.

Global 1

1 Right-click **Verification** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **obj - Objective - V/m**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
obj	V/m	Topology Optimization

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2: Shape Optimization/Solution 6 (5) (sol6)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
obj	V/m	Shape Optimization

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

6 In the **Expression** text field, type `lambda0`.

7 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

Global 3

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

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

3 From the **Dataset** list, choose **Study 3: Verification/Solution 7 (7) (sol7)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
obj	V/m	Verification

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **None**.

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

8 In the **Verification** toolbar, click  **Plot**.

Revolution 2D 2

1 In the **Model Builder** window, under **Results** > **Datasets** click **Revolution 2D 2**.

2 In the **Settings** window for **Revolution 2D**, click to collapse the **Axis Data** section.

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

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `250`.

5 In the **Revolution angle** text field, type `270`.

Revolution 2D 1

1 In the **Model Builder** window, click **Revolution 2D 1**.

2 In the **Settings** window for **Revolution 2D**, locate the **Revolution Layers** section.

3 In the **Start angle** text field, type `250`.

4 In the **Revolution angle** text field, type `270`.

Selection

1 Right-click **Revolution 2D 1** and choose **Selection**.

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

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

4 Select Domains 1–8 only.

Thumbnail

1 In the **Results** toolbar, click  **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Thumbnail` in the **Label** text field.

Volume 1

1 Right-click **Thumbnail** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Coloring and Style** section.

3 Clear the **Color legend** checkbox.

Filter 1

1 Right-click **Volume 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `0.5<centroid(genext2(theta_m))`.

Transparency 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Transparency**.

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

3 Find the **Transparency** subsection. In the **Transparency** text field, type `1`.

Isosurface 1

1 In the **Model Builder** window, right-click **Thumbnail** and choose **Isosurface**.

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

3 In the **Expression** text field, type `if(0.5*lens_H<Z || 0<rev1phi,ewfd.normE, nan)`.

4 Locate the **Levels** section. In the **Total levels** text field, type `8`.

Color Expression 1

1 Right-click **Isosurface 1** and choose **Color Expression**.

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

3 From the **Color table** list, choose **Thermal**.

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

Visual Effects 1

1 In the **Model Builder** window, right-click **Isosurface 1** and choose **Visual Effects**.

2 In the **Settings** window for **Visual Effects**, locate the **Visual Effects** section.

3 Find the **Direct shadows** subsection. From the **Mode** list, choose **Manual**.

4 Clear the **Casts shadows** checkbox.

Transformation 1

1 Right-click **Isosurface 1** and choose **Transformation**.

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

3 Select the **Rotate** checkbox.

4 In the **Angle** text field, type `1`.

Surface 1

1 In the **Model Builder** window, right-click **Thumbnail** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Blue**.

6 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

7 From the **Smoothing** list, choose **None**.

Filter 1

1 Right-click **Surface 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(0.5<centroid(theta)) && rev1phi<-220/360*pi`.

4 In the **Thumbnail** toolbar, click **Plot**.

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

Disable the interfaces of the second component in the first study.

STUDY 1: TOPOLOGY OPTIMIZATION

Topology Optimization

1 In the **Model Builder** window, under **Study 1: Topology Optimization** click **Topology Optimization**.

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

3 In the table, clear the **Solve for** checkbox for **Transformation 1**.

Step 1: Wavelength Domain

1 In the **Model Builder** window, click **Step 1: Wavelength Domain**.

2 In the **Settings** window for **Wavelength Domain**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, clear the checkbox for **Component 2: Verification (comp2)**.

Geometry Modeling Instructions

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

NEW

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

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
lens_l	5[um]	5E-6 m	Width of lens
lens_H	0.1*lens_l	5E-7 m	Height of lens
lens_F	0.24*lens_l	1.2E-6 m	Focal length
d_PML	0.2*lens_l	1E-6 m	Width of PML
cell_H	lens_F + 2*lens_H + 2*d_PML	4.2E-6 m	Total geometry height
d_src_lens	0.05*lens_l	2.5E-7 m	Separation b/w src and lens
lens_minH	0.2*lens_H	1E-7 m	Minimum lens thickness
layers	5	5	Design layers

ADD COMPONENT

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

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.5*lens_l`.

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

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

Layer name	Thickness (m)
Layer 1	lens_H-lens_minH

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

Point 1 (pt1)

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

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

3 In the **z** text field, type `lens_F+lens_H/2+d_src_lens`.

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

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `d_PML`.

4 In the **Height** text field, type `cell_H-d_PML`.

5 Locate the **Position** section. In the **r** text field, type `lens_l/2`.

6 In the **z** text field, type `-d_src_lens`.

Rectangle 3 (r3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Rectangle 1 (r1)** and choose **Duplicate**.

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

3 In the **Height** text field, type `cell_H-d_PML`.

4 Locate the **Position** section. In the **z** text field, type `-d_src_lens`.

5 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`d_src_lens`
`Layer 2`	`lens_H`
`Layer 3`	`cell_H-2*d_PML-lens_H-d_src_lens`

Rectangle 4 (r4)

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

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

3 In the **Width** text field, type `0.5*lens_l`.

4 In the **Height** text field, type `(lens_H-lens_minH)/layers`.

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

Control layers

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

2 In the **Settings** window for **Box Selection**, type `Control layers` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **z minimum** text field, type `-eps`.

5 In the **z maximum** text field, type `eps`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Array 1 (arr1)

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

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Rectangle 4**.

4 Locate the **Size** section. In the **z size** text field, type `layers`.

5 Locate the **Displacement** section. In the **z** text field, type `(lens_H-lens_minH)/layers`.

Form Union (fin)

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

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

Symmetry/Roller

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

2 In the **Settings** window for **Box Selection**, type `Symmetry/Roller` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **r minimum** text field, type `0.1*lens_l`.

5 In the **r maximum** text field, type `0.4*lens_l`.

6 In the **z minimum** text field, type `-0.001*lens_H`.

7 In the **z maximum** text field, type `1.001*lens_H`.

Source

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

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

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

4 Locate the **Box Limits** section. In the **z maximum** text field, type `-eps`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

PML

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

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

3 Locate the **Box Limits** section. In the **r minimum** text field, type `lens_l*0.49`.

4 In the **z minimum** text field, type `cell_H-lens_H/2-3*d_PML/2`.

Fixed Lens

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Source (boxsel3)** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Fixed Lens` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **z minimum** text field, type `lens_H-1.01*lens_minH`.

5 In the **z maximum** text field, type `1.01*lens_H`.

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

The model geometry is now complete.