Fresnel Lens

Introduction

Fresnel lenses are a special type of optic devices that are extremely thin — as thin as the order of the wavelength of a light — and are designed to have focusing, defocusing, and even aberration correction capabilities. A Fresnel lens is designed from an original lens by zoning and folding as depicted in Figure 1. First, the original lens is sectioned along its height in layers of thickness $m\lambda/(n-1)$, where m, λ, and n are the integer Fresnel lens order, the wavelength, and the refractive index, respectively. This corresponds to a $2m\pi$ radian phase of the light in the lens material. Then the residual parts on the top are folded to the bottom. The concept of this design method is based on the assumption that the incident light is a plane wave with a flat phase front and the material parts of height $m\lambda/(n-1)$ that are removed do nothing to the phase of the light. Under this assumption, the Fresnel lens is expected to exhibit approximately the same behavior as the original unfolded lens. When a continuous Fresnel lens is designed, it is typically digitized to a digital Fresnel lens for manufacturing reasons.

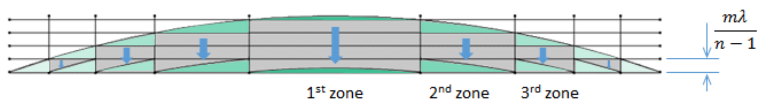


Figure 1: Design of a Fresnel lens.

Model Definition

In this model, a plano-convex spherical lens of 50 μm diameter and 150 μm focal length for 0.5 μm wavelength is converted to a 16-level digitized Fresnel lens of the first order. A plane wave is incident on the bottom boundary, passes through the Fresnel lens, and exits at the exit boundary above the Fresnel lens. The transmitted wave is then propagated to the focal plane.

A comparison is made between the analytical solution using the Fresnel approximation and the solution obtained using both the Electromagnetic Waves, Frequency Domain and the Electromagnetic Waves, Beam Envelopes interfaces. For this model you get a large speed advantage using the Electromagnetic Waves, Beam Envelopes interface compared to the Electromagnetic Waves, Frequency Domain interface.

In addition to the comparison between the analytical solution and the solutions from the two physics interfaces, the model also demonstrates how to add and run a model method. In this case, the model method helps rebuild the geometry when the geometry parameters have been changed.

Results and Discussion

Figure 2 shows the electric field amplitude around the Fresnel lens and Figure 3 shows the corresponding electric field norm. In the amplitude plot, it is clearly seen that the wavelet exiting from each zone generates a curved wavefront in the air domain and that all wavefronts are in phase and are making one converging wave. In the norm plot, the nonuniformity of the field norm in the lens is a result of Fresnel reflection because the lens surface is not covered by any antireflective coating. You can also see some diffraction at the corner of each digitized step.

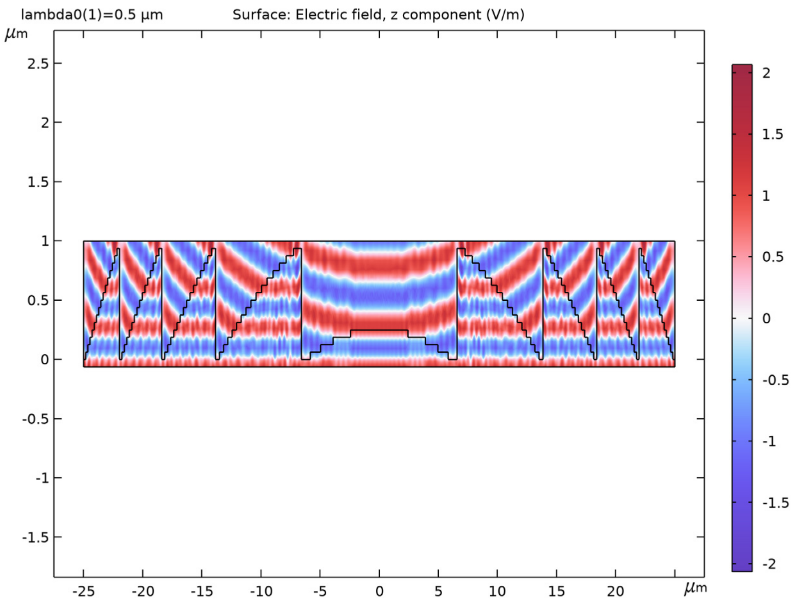


Figure 2: Electric field amplitude around the Fresnel lens. The plot is scaled as x:y = 1:10.

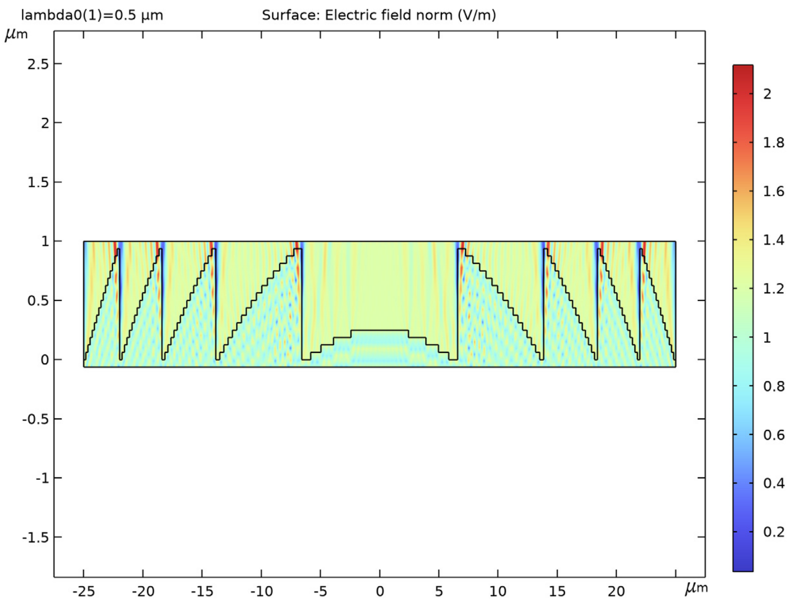


Figure 3: Electric field norm around the Fresnel lens. The plot is scaled as x:y = 1:10.

Figure 4 shows the phase function of the wave at the exit boundary, just above the Fresnel lens, at $y = 1\ \mu m$. This quadratic phase function is approximately the same as in the case of the original unfolded lens.

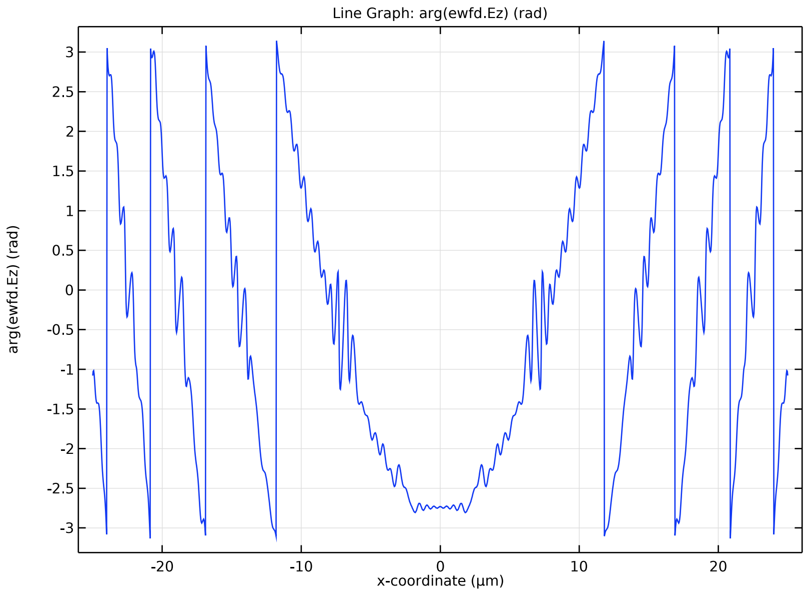


Figure 4: Phase of the electric field at the exit boundary, $y = 1\ \mu m$.

Figure 5 is a plot of the electric field norm in the entire air domain showing that the incident plane wave is focused at the focal plane.

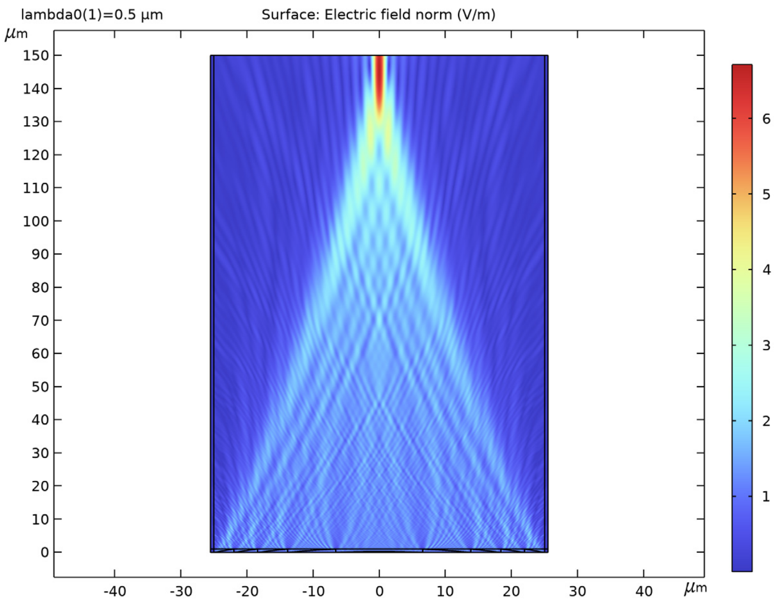


Figure 5: Electric field norm in the air domain. Note that the plot is scaled as x:y = 2:1.

Finally, Figure 6 is the electric field amplitude at the focal plane. This is a typical image of digitized Fresnel lenses at the focus. There is the main peak at the center, as a result of the first order diffraction and there are some background caused by the zeroth order and the diffraction orders that were not considered in the design. The magnitude of this background varies depending on the digitization level and the straightness of the surface discontinuities. The result of the full-wave simulation based on the Helmholtz equation is overlaid with the result of the Fresnel approximation, which is computed by the Fresnel diffraction formula

$$E(u,f) = \frac{i}{\sqrt{\lambda f}}\exp(-i(2\pi f/\lambda + \pi u^2/\lambda f))$$
$$\int_{-\infty}^{\infty} E(x, 1[\mu m])\exp(-i\pi x(x-2u)/(\lambda f))dx$$

applied to the electric field at the exit boundary at $y = 1$ µm. The results of the full-wave simulations and the Fresnel approximation are in very good agreement.

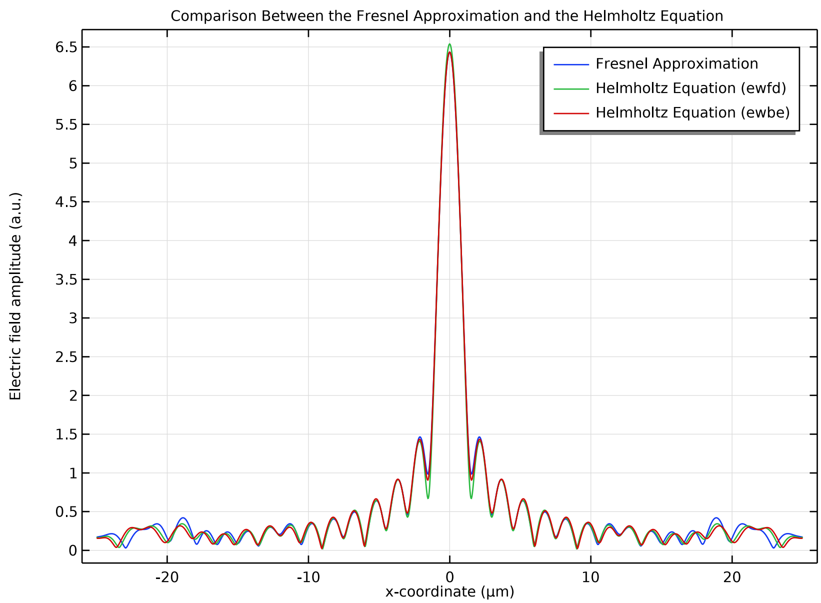


Figure 6: Comparison between the Fresnel approximation and the full-wave simulation, using both the Electromagnetic Waves, Frequency Domain and the Electromagnetic Waves, Beam Envelopes interfaces.

Reference

1. Donald. C. O'Shea, Thomas J. Suleski, Alan D. Kathman, and Dennis W. Prather, *Diffractive Optics: Design, Fabrication, and Test*, SPIE Press, 2003.

Application Library path: `Wave_Optics_Module/Verification_Examples/fresnel_lens`

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**.

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

3 Click **Add**.

4 In the **Select Physics** tree, select **Optics>Wave Optics>Electromagnetic Waves, Beam Envelopes (ewbe)**.

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GLOBAL DEFINITIONS

First add some parameters that will be used for defining the geometry, material, and the study.

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 Click **Load from File**.

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

Now, add a geometry part that defines one zone of the Fresnel lens.

FRESNEL LENS ZONE

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts> 2D Part**.

2 In the **Settings** window for **Part**, type `Fresnel Lens Zone` in the **Label** text field.

Add the input parameter for this part.

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

Name	Default expression	Value	Description
zone	1	1	Zone index

4 Locate the **Units** section. From the **Length unit** list, choose **µm**, to make all lengths in plots appear in units of µm.

Now, add the internal parameters that will be used when defining the part.

Local Parameters

1 In the **Geometry** toolbar, click **Programming** and choose **Local Parameters**.

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

3 Click **Load from File**.

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

The parameter `Mi` defines the number of discretization levels for the zone. For all zones except the first one, this parameter will be equal to the global parameter `M`. However, for the first zone, the parameter `Mi` is normally smaller than `M`.

The remaining parameters define the x- and y-coordinates for the Fresnel lens zone.

Polygon 1 (pol1)

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

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

3 From the **Data source** list, choose **Vectors**.

4 In the **x** text field, type `x0 x0 x1 x1 x2 x2 x3 x3 x4 x4 x5 x5 x6 x6 x7 x7 x8 x8 x9 x9 x10 x10 x11 x11 x12 x12 x13 x13 x14 x14 x15 x15 x16 x16`.

5 In the **y** text field, type `y0 y1 y1 y2 y2 y3 y3 y4 y4 y5 y5 y6 y6 y7 y7 y8 y8 y9 y9 y10 y10 y11 y11 y12 y12 y13 y13 y14 y14 y15 y15 y16 y16 y0`.

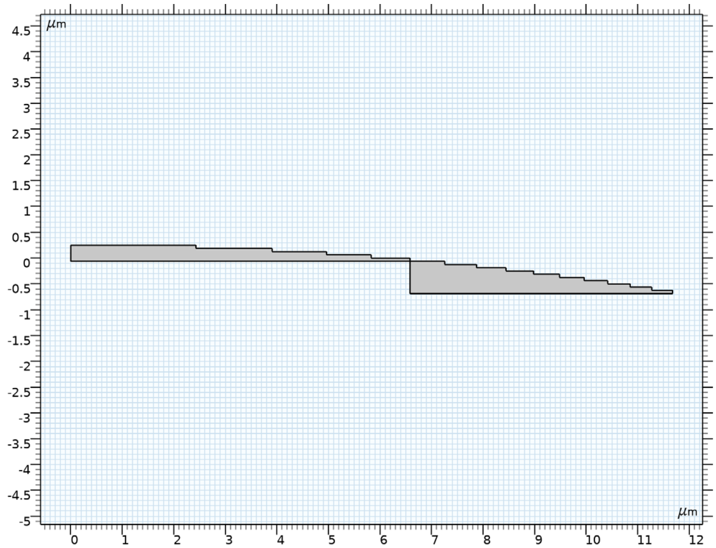
6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog box, type `Fresnel Lens Zone` in the **Name** text field.

8 Click **OK**.

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

Notice that the shape for the first Fresnel lens zone has two parts, where the rightmost part is located below the base line. This part will be removed later when building the Fresnel lens geometry.



GEOMETRY 1

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 **µm**, to make all lengths in plots appear in units of µm.

The geometry is created by adding instances of the **Fresnel Lens Zone** part that we just have created. For each zone, the zone index input parameter is changed, to generate the correct coordinates for the zone.

Fresnel Lens Zone 1 (pi1)

1 In the **Geometry** toolbar, click **Parts** and choose **Fresnel Lens Zone**.

2 In the **Settings** window for **Part Instance**, click to expand the **Domain Selections** section.

3 Click **New Cumulative Selection**.

4 In the **New Cumulative Selection** dialog box, type `Fresnel Lens` in the **Name** text field.

5 Click **OK**.

6 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

7 In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Fresnel Lens Zone		√	Fresnel Lens

Fresnel Lens Zone 2 (pi2)

1 Right-click **Fresnel Lens Zone 1 (pi1)** and choose **Duplicate**.

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
zone	2	2	Zone index

Fresnel Lens Zone 3 (pi3)

1 Right-click **Fresnel Lens Zone 2 (pi2)** and choose **Duplicate**.

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
zone	3	3	Zone index

Fresnel Lens Zone 4 (pi4)

1 Right-click **Fresnel Lens Zone 3 (pi3)** and choose **Duplicate**.

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
zone	4	4	Zone index

Fresnel Lens Zone 5 (pi5)

1 Right-click **Fresnel Lens Zone 4 (pi4)** and choose **Duplicate**.

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

3 In the table, enter the following settings:

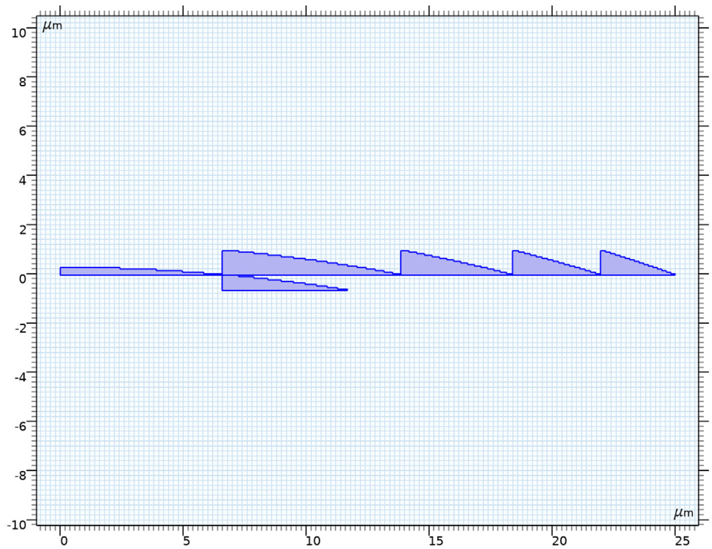
Name	Expression	Value	Description
zone	5	5	Zone index

Union 1 (uni1)

Before proceeding further, make the created half Fresnel lens a single domain, using a Union operation.

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

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



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

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

5 Click **Build Selected**.

Mirror 1 (mir1)

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

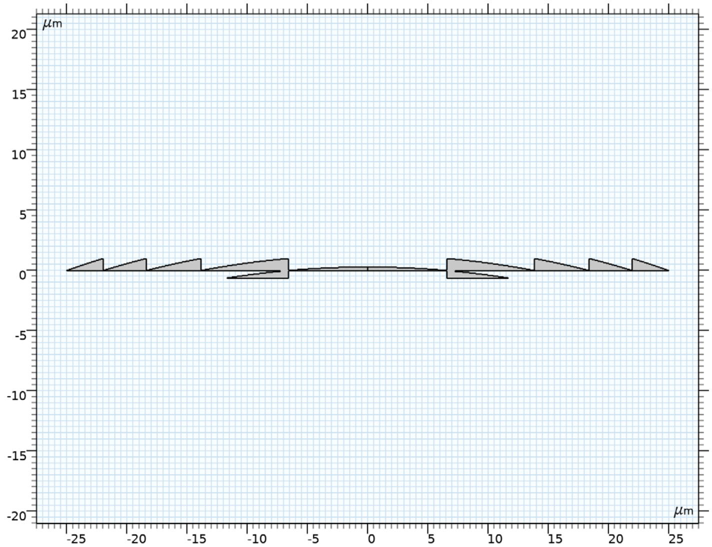
2 Select the object **uni1** only.

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

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

5 Click **Build Selected**.

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



Union 2 (uni2)

Make a single domain out of the two lens halves.

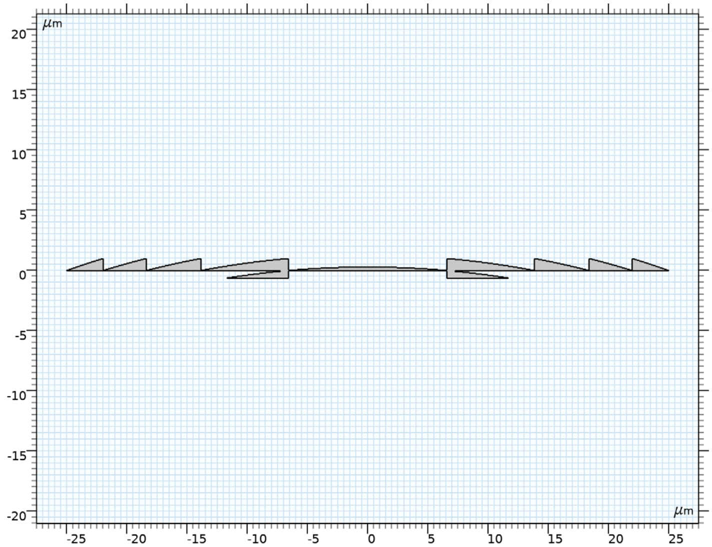
1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

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

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

5 Click **Build Selected**.



Rectangle 1 (r1)

The first zone has a domain that is below the base of the Fresnel lens. That part is removed by making an intersection with a rectangle having the size of the expected bounding box for the Fresnel lens.

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`.

4 In the **Height** text field, type `d+dm`.

5 Locate the **Position** section. In the **x** text field, type `-D/2`.

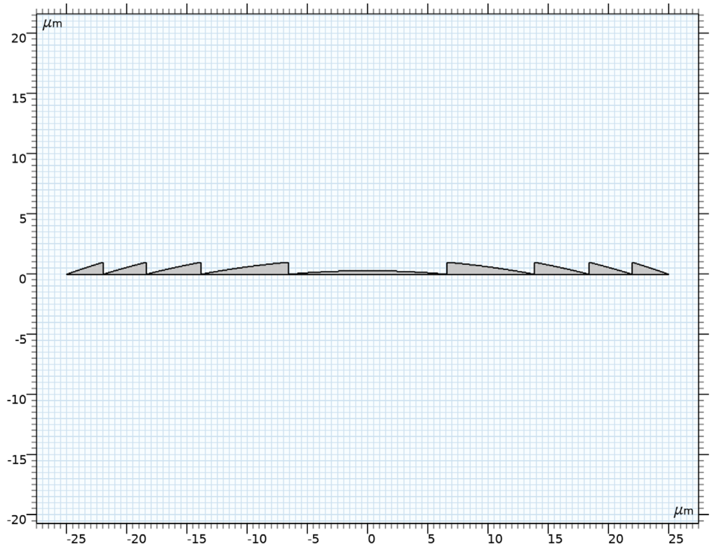
6 In the **y** text field, type `-dm`.

Intersection 1 (int1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Intersection**, click **Build All Objects**.



Rectangle 2 (r2)

Now define the propagation domain from the lens to the focal plane.

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`.

4 In the **Height** text field, type `f+dm`.

5 Locate the **Position** section. In the **x** text field, type `-D/2`.

6 In the **y** text field, type `-dm`.

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

Layer name	Thickness (µm)
Layer 1	d+dm

Rectangle 3 (r3)

Now, add two rectangular domains that will be used for perfectly matched layers (PMLs).

1 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

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

3 In the **Width** text field, type `lda0`.

4 Locate the **Position** section. In the **x** text field, type `-D/2-lda0`.

Rectangle 4 (r4)

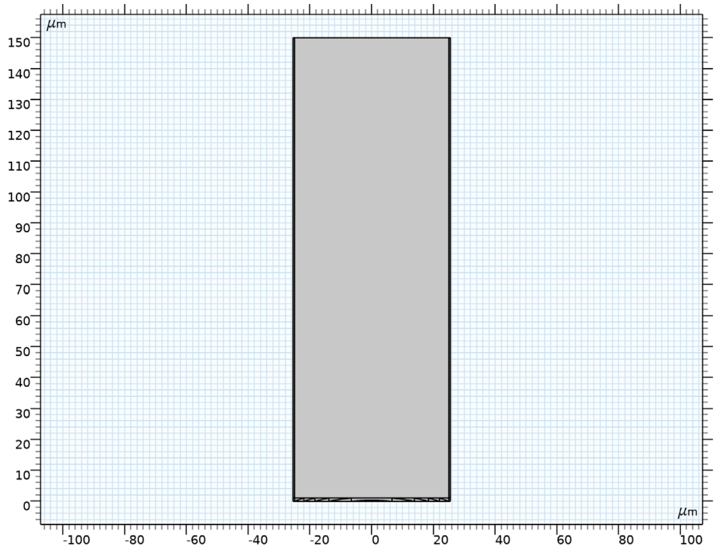
1 Right-click **Rectangle 3 (r3)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **x** text field, type `D/2`.

4 Click **Build All Objects**.

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



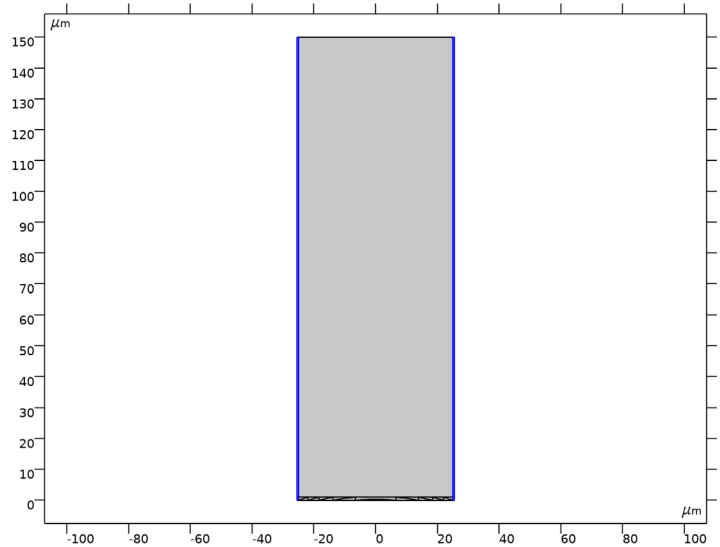
DEFINITIONS

Add the perfectly matched layer.

Perfectly Matched Layer 1 (pml1)

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

2 Select Domains 1, 2, 6, and 7 only.



MATERIALS

Now add the materials. In COMSOL the first material is applied to all domains, so we start by adding the air to all domains and then add the glass to the Fresnel lens.

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>Air**.

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

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

MATERIALS

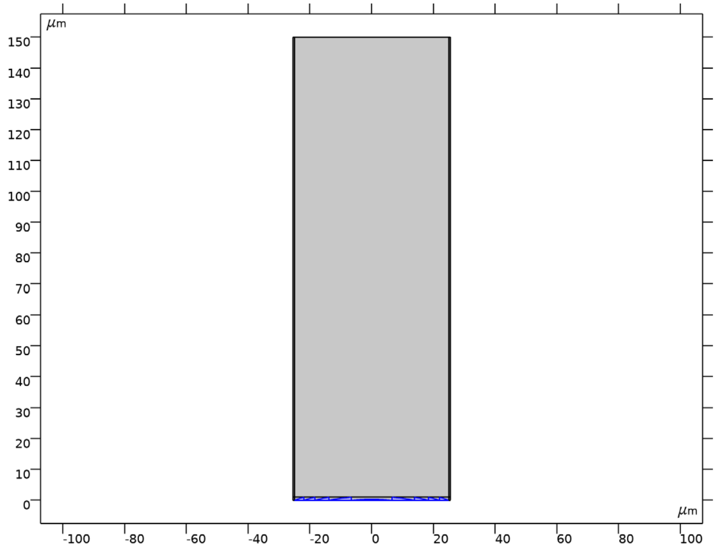
Glass

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Fresnel Lens**.

4 Click **Zoom to Selection**.



5 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	1	Refractive index
Refractive index, imaginary part	ki_iso ; kiii = ki_iso, kiij = 0	0	1	Refractive index

DEFINITIONS

Now define some selections that will be used later.

Exterior Boundaries

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

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

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

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

Input Plane

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

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

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

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Under **Selections**, click **Add**.

6 In the **Add** dialog box, select **Exterior Boundaries** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Box**, locate the **Box Limits** section.

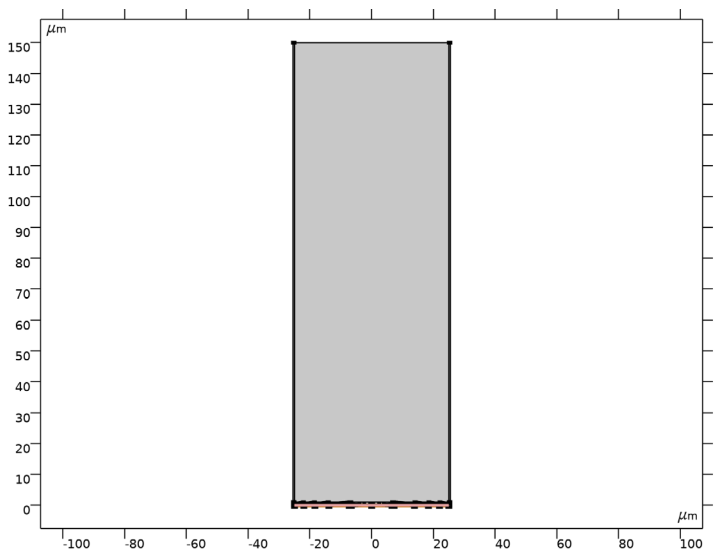
9 In the **x minimum** text field, type `-D/2`.

10 In the **x maximum** text field, type `D/2`.

11 In the **y minimum** text field, type `-dm`.

12 In the **y maximum** text field, type `-dm/2`.

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



Scattering Boundaries

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

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

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

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

5 In the **Add** dialog box, select **Exterior Boundaries** in the **Selections to add** list.

6 Click **OK**.

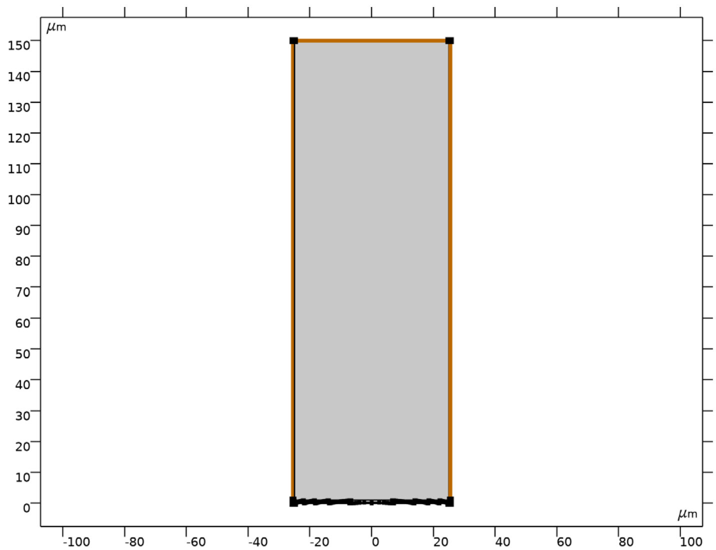
7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

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

9 In the **Add** dialog box, select **Input Plane** in the **Selections to subtract** list.

10 Click **OK**.

11 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Exit Plane

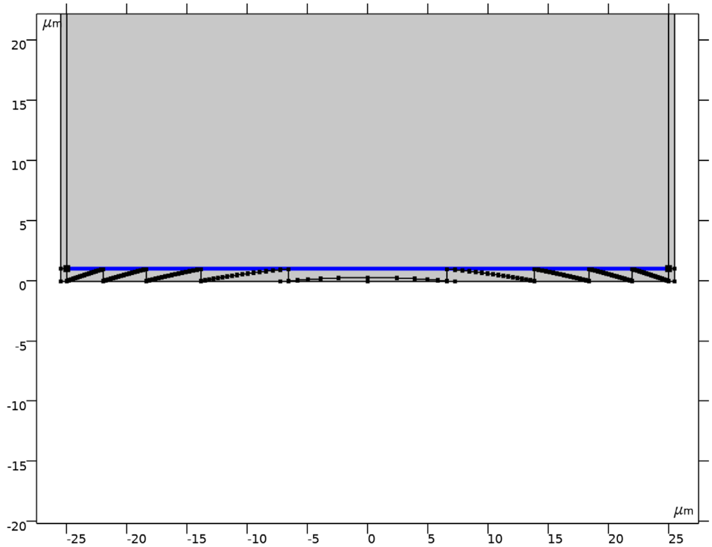
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundary 11 only. This corresponds to the horizontal boundary just above the Fresnel lens.

5 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Integration 1 (intop1)

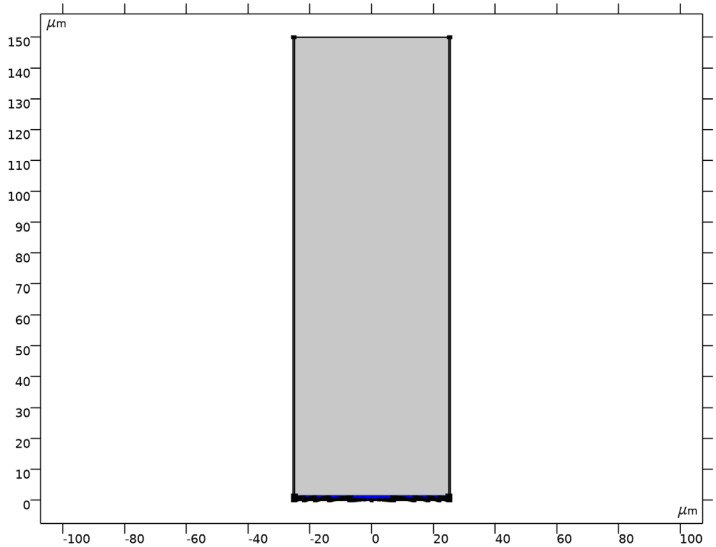
Before setting up the physics, a nonlocal integration coupling will be defined for the **Exit Plane** selection. This operator will be used for calculating the analytic Fresnel diffraction integral.

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

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

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

4 From the **Selection** list, choose **Exit Plane**.



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 **Components** section.

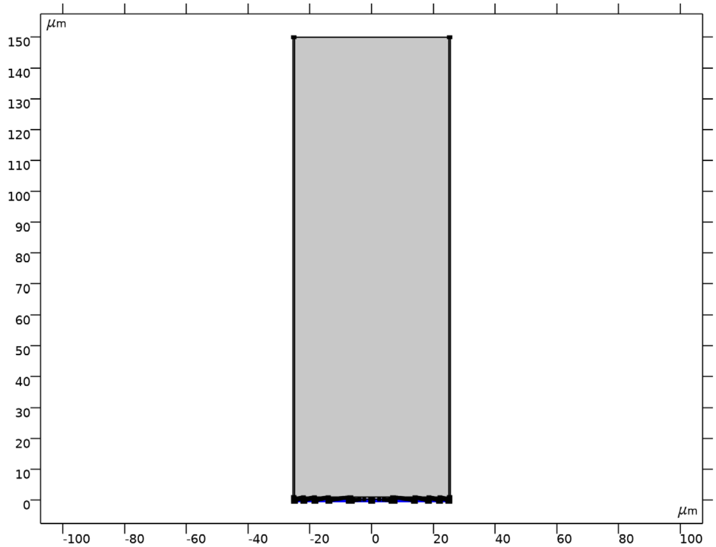
3 From the **Electric field components solved for** list, choose **Out-of-plane vector**.

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 **Input Plane**.



4 Locate the **Scattering Boundary Condition** section. From the **Incident field** list, choose **Wave given by E field**.

5 Specify the $\mathbf{E}_0$ vector as

0	x
0	y
1	z

6 From the **Order** list, choose **Second order**, to reduce the reflections from radiation incident with a nonnormal direction to this boundary.

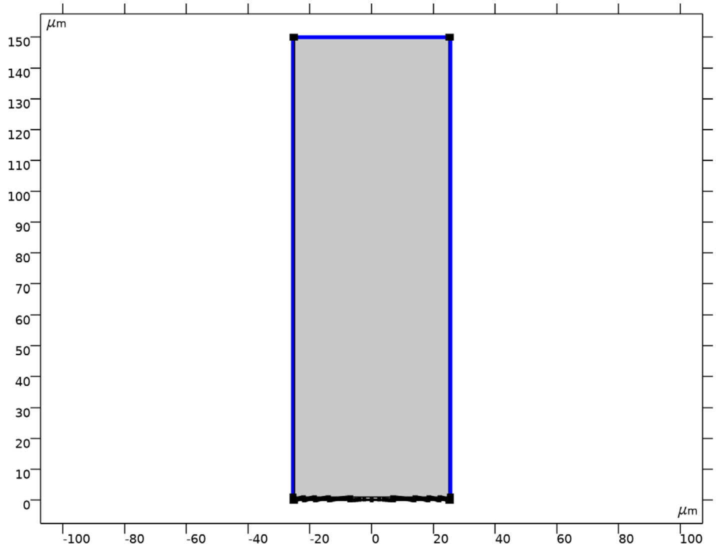
Scattering Boundary Condition 2

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 **Scattering Boundaries**.

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



5 Locate the **Scattering Boundary Condition** section. From the **Order** list, choose **Second order**, to reduce the reflections from radiation incident with a nonnormal direction to this surrounding boundary.

FINE MESH

Add a name for the mesh, as we later will add a second mesh for the Electromagnetic Waves, Beam Envelopes interface.

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

2 In the **Settings** window for **Mesh**, type `Fine Mesh` in the **Label** text field.

Free Triangular 1

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

Size

1 In the **Model Builder** window, click **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. In the **Maximum element size** text field, type `lda0/Nmesh`.

Free Triangular 1

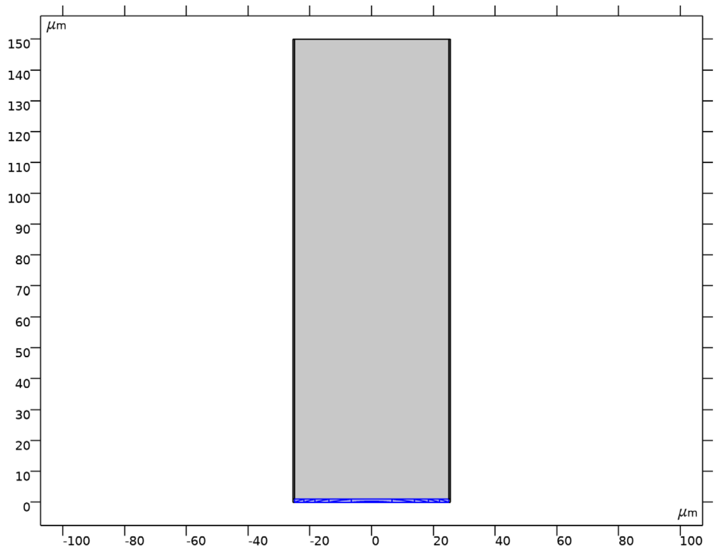
1 In the **Model Builder** window, click **Free Triangular 1**.

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

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

4 Select Domains 3 and 4 only.

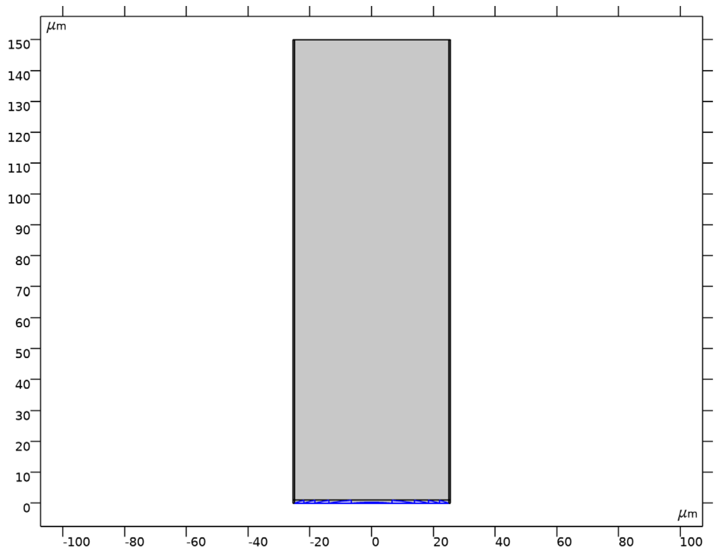
5 Click **Zoom to Selection**.



Size 1

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

2 Select Domain 3 only. The simplest way to do this is to select entity 4 in the **Selection box** and then click the **Remove from Selection** (the minus sign) button in the **Selection** toolbar.



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

4 Click the **Custom** button.

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

6 In the associated text field, type `lda0/n/Nmesh`.

Mapped 1

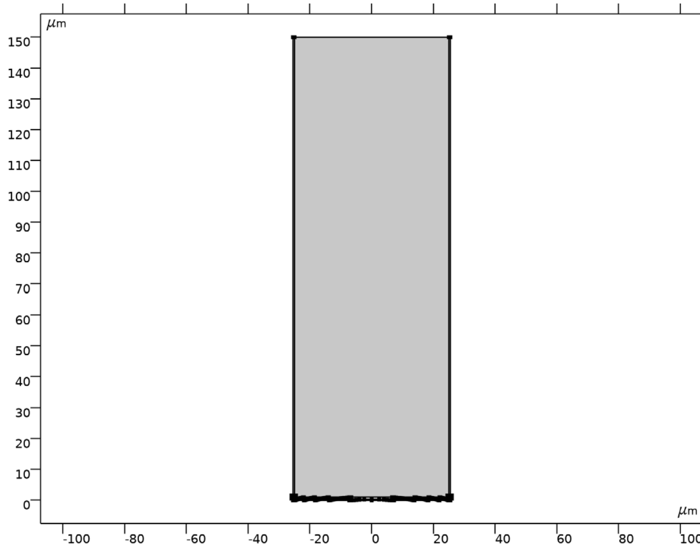
In the **Mesh** toolbar, click **Mapped**.

Distribution 1

A Distribution node defines the number of mesh elements in the PMLs.

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 4 and 301 only.



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

4 In the **Number of elements** text field, type `8`.

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 In the **Wavelengths** text field, type `lda0`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Electromagnetic Waves, Beam Envelopes (ewbe)**.

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

RESULTS

Electric Field Norm (ewfd)

1 In the **Settings** window for **2D Plot Group**, type `Electric Field Norm (ewfd)` in the **Label** text field.

Change the view now to make the plot wider to improve the aspect ratio.

2 Locate the **Plot Settings** section. From the **View** list, choose **View 1**.

3 Click **Go to Source**.

DEFINITIONS

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

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

3 From the **View scale** list, choose **Manual**.

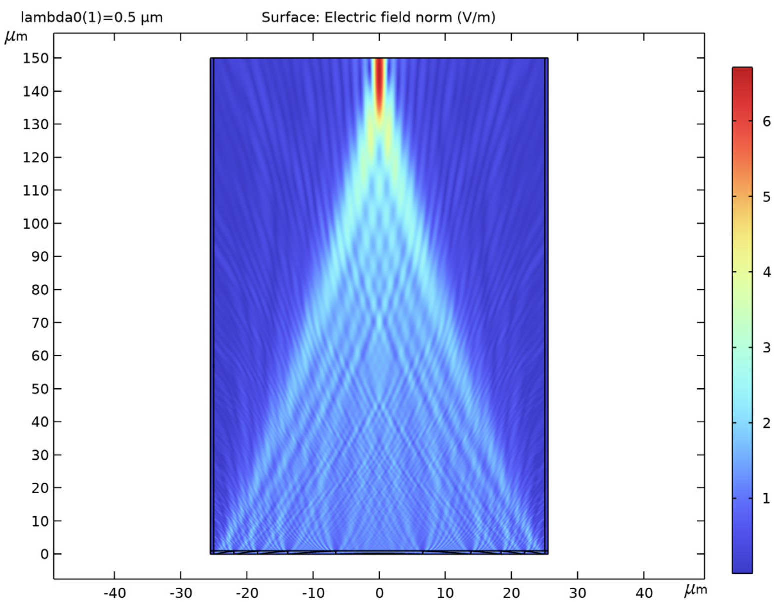
4 In the **x scale** text field, type `2`.

5 Click **Update**.

RESULTS

Electric Field Norm (ewfd)

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



Electric Field Amplitude (ewfd)

1 In the **Model Builder** window, right-click **Electric Field Norm (ewfd)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Field Amplitude (ewfd)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Electric Field Amplitude (ewfd)** node, then click **Surface 1**.

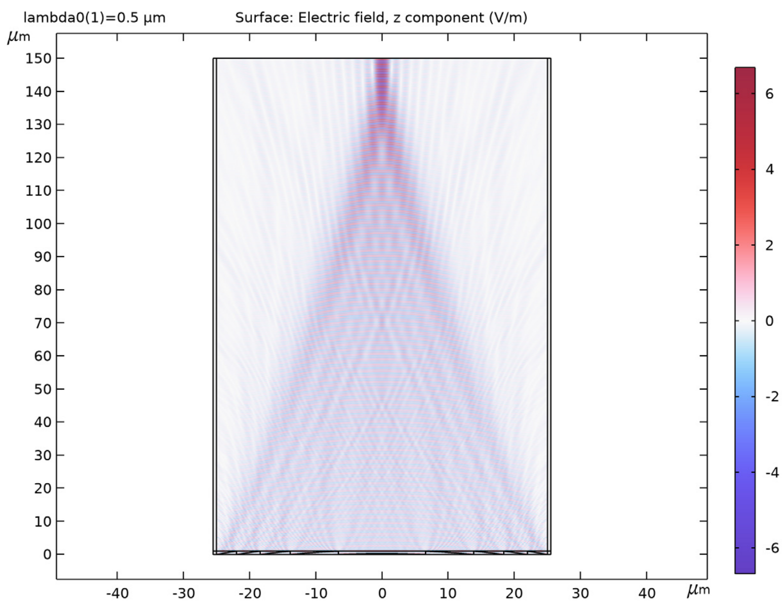
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `ewfd.Ez`.

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

5 From the **Scale** list, choose **Linear symmetric**.

6 In the **Electric Field Amplitude (ewfd)** toolbar, click **Plot**.



Electric Field Norm Near Lens (ewfd)

1 In the **Model Builder** window, right-click **Electric Field Norm (ewfd)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Field Norm Near Lens (ewfd)` in the **Label** text field.

DEFINITIONS

Add a new view that will be used to show the near field close to the lens.

View 3

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

Axis

1 In the **Model Builder** window, expand the **View 3** node, then click **Axis**.

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

3 In the **x scale** text field, type 1.

4 In the **y scale** text field, type 10.

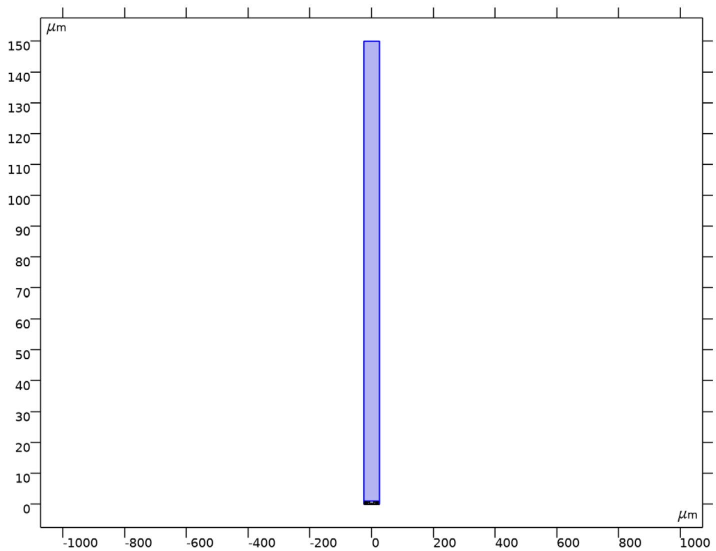
5 Click **Update**.

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 3** and choose **Hide for Physics**.

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

3 Select Domains 1, 2, and 5–7 only.



RESULTS

Electric Field Norm Near Lens (ewfd)

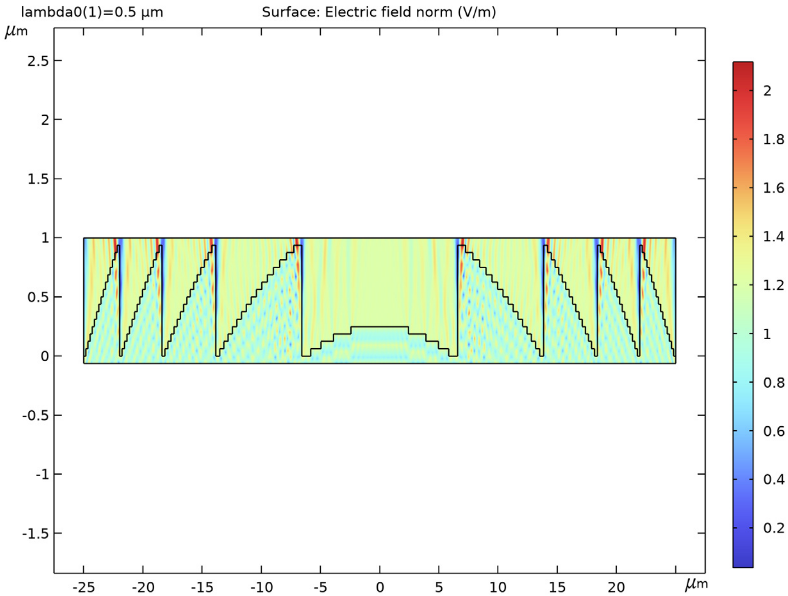
1 In the **Model Builder** window, under **Results** click **Electric Field Norm Near Lens (ewfd)**.

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

3 From the **View** list, choose **View 3**.

4 In the **Electric Field Norm Near Lens (ewfd)** toolbar, click **Plot**.

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



Electric Field Amplitude Near Lens (ewfd)

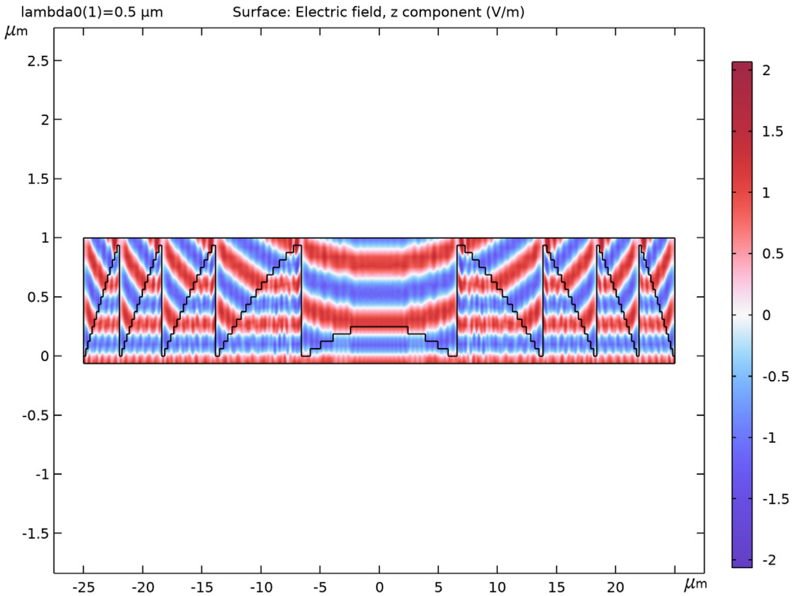
1 In the **Model Builder** window, right-click **Electric Field Amplitude (ewfd)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Field Amplitude Near Lens (ewfd)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **View 3**.

4 In the **Electric Field Amplitude Near Lens (ewfd)** toolbar, click  **Plot**.

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



Electric Field Amplitude at Exit Plane (ewfd)

Now create two line plots displaying the electric field amplitude and phase, respectively.

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

2 In the **Settings** window for **1D Plot Group**, type `Electric Field Amplitude at Exit Plane (ewfd)` in the **Label** text field.

Line Graph 1

1 Right-click **Electric Field Amplitude at Exit Plane (ewfd)** and choose **Line Graph**.

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

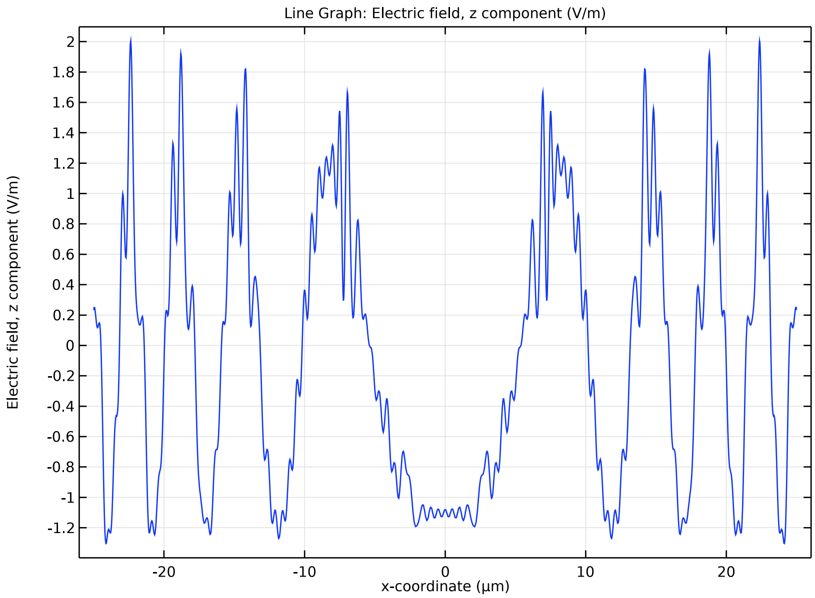
3 From the **Selection** list, choose **Exit Plane**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `ewfd.Ez`.

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

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

7 In the **Electric Field Amplitude at Exit Plane (ewfd)** toolbar, click **Plot**.



Electric Field Phase at Exit Plane (ewfd)

1 In the **Model Builder** window, right-click **Electric Field Amplitude at Exit Plane (ewfd)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Electric Field Phase at Exit Plane (ewfd)` in the **Label** text field.

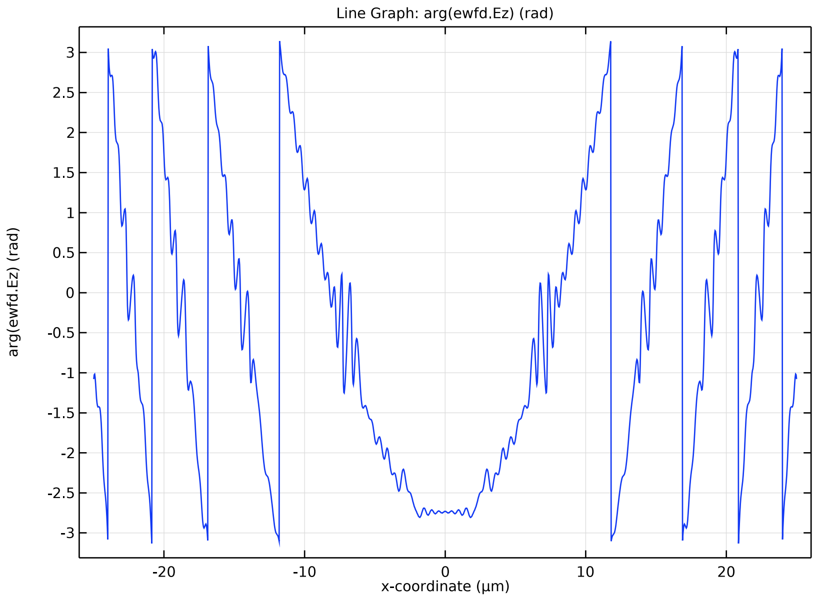
Line Graph 1

1 In the **Model Builder** window, expand the **Electric Field Phase at Exit Plane (ewfd)** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `arg(ewfd.Ez)`.

4 In the **Electric Field Phase at Exit Plane (ewfd)** toolbar, click **Plot**.



Grid 1D 1

Now, create a line plot that compares the fields calculated by the Fresnel diffraction integral and when propagating the field from the **Exit Plane** using COMSOL (denoted Helmholtz Equation (ewfd) in the following discussion).

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 1D**. This dataset defines the destination points for which the Fresnel diffraction integral will be evaluated.

2 In the **Settings** window for **Grid 1D**, locate the **Parameter Bounds** section.

3 In the **Name** text field, type `u`.

4 In the **Minimum** text field, type `-D/2`.

5 In the **Maximum** text field, type `D/2`.

Fresnel Versus Helmholtz Comparison

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

2 In the **Settings** window for **1D Plot Group**, type `Fresnel Versus Helmholtz Comparison` in the **Label** text field.

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

Fresnel Approximation

1 Right-click **Fresnel Versus Helmholtz Comparison** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Fresnel Approximation` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the **Expression** text field, type `1/sqrt(lda0*f)*abs(intop1(ewfd.Ez*exp(-i*k0*x^2/(2*f))*exp(i*2*pi*dest(u)*x/(lda0*f))))`.

4 Select the **Description** check box.

5 In the associated text field, type `Diffraction integral`.

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

7 In the **Expression** text field, type `u`, to express `u` in µm.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Fresnel Approximation`

Helmholtz Equation (ewfd)

1 In the **Model Builder** window, right-click **Fresnel Versus Helmholtz Comparison** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Helmholtz Equation (ewfd)` in the **Label** text field.

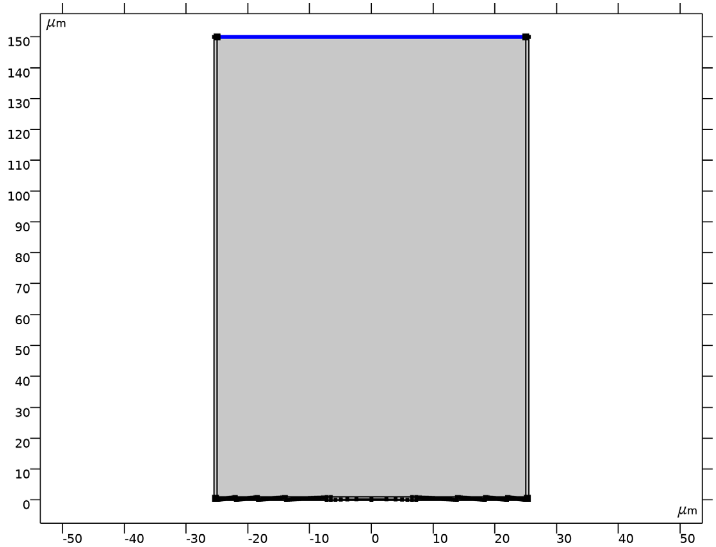
3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

5 In the **Graphics** window toolbar, click ▾ next to **Go to Default View**, then choose **Go to View 1**.

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

7 Select Boundary 12 only. This is the top boundary.



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

9 In the **Expression** text field, type `x`.

10 Locate the **Legends** section. Select the **Show legends** check box.

11 From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
`Helmholtz Equation (ewfd)`

Fresnel Versus Helmholtz Comparison

1 In the **Model Builder** window, click **Fresnel Versus Helmholtz Comparison**.

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

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

4 In the associated text field, type `x-coordinate (µm)`.

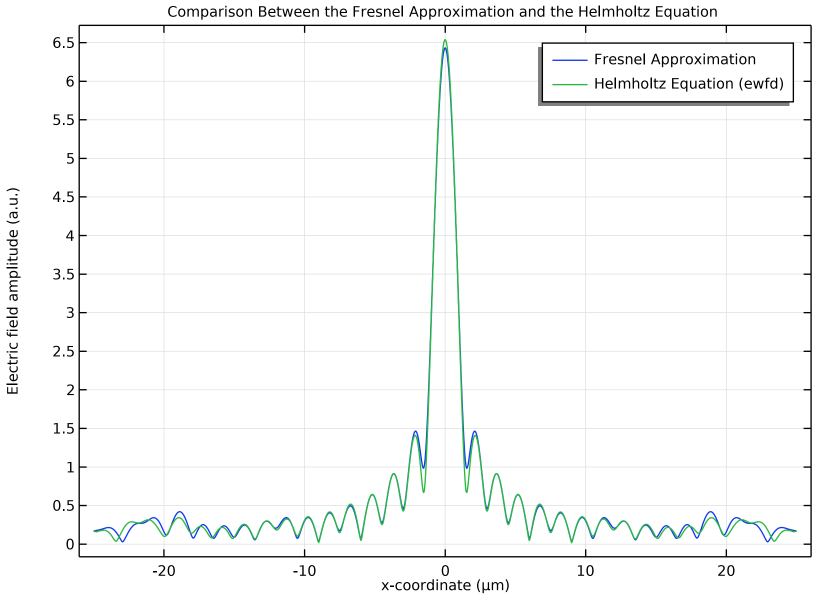
5 Select the **y-axis label** check box.

6 In the associated text field, type `Electric field amplitude (a.u.)`.

7 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

8 In the **Title** text area, type `Comparison Between the Fresnel Approximation and the Helmholtz Equation`.

9 In the **Fresnel Versus Helmholtz Comparison** toolbar, click **Plot**.



ELECTROMAGNETIC WAVES, BEAM ENVELOPES (EWBE)

Now redo the simulation for the Electromagnetic Waves, Beam Envelopes interface.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Beam Envelopes (ewbe)**.

2 In the **Settings** window for **Electromagnetic Waves, Beam Envelopes**, locate the **Components** section.

3 From the **Electric field components solved for** list, choose **Out-of-plane vector**.

4 Locate the **Wave Vectors** section. From the **Number of directions** list, choose **Unidirectional**.

5 Specify the $\mathbf{k}_1$ vector as

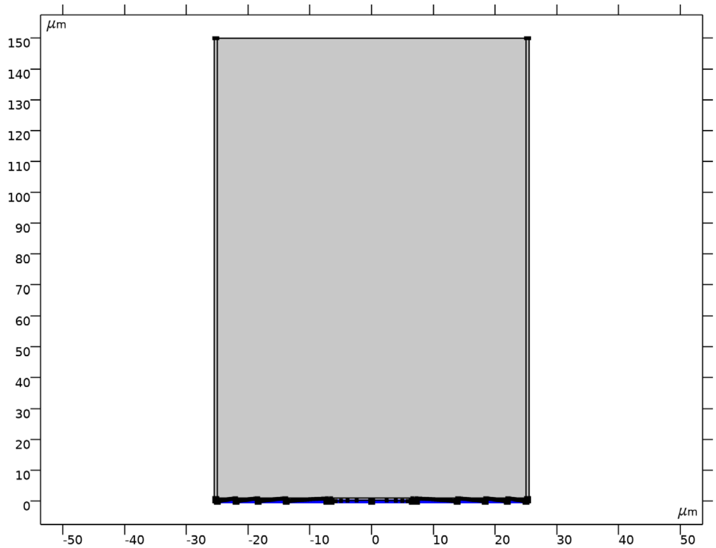
0	x
ewbe.k0	y

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 **Input Plane**.



4 Locate the **Scattering Boundary Condition** section. Specify the $\mathbf{E}_0$ vector as

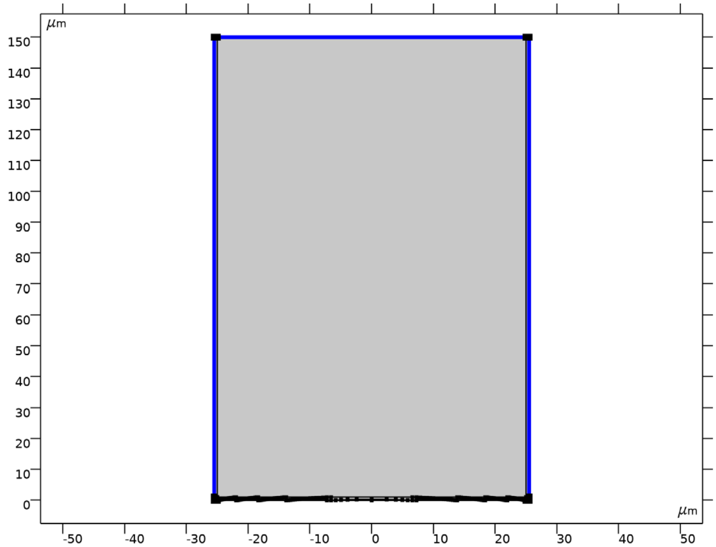
0	x
0	y
1	z

Scattering Boundary Condition 2

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 **Scattering Boundaries**.



FINE MESH

Add a second, coarser, mesh that will be used for the simulation with the Electromagnetic Waves, Beam Envelopes interface.

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

COARSE MESH

In the **Settings** window for **Mesh**, type `Coarse Mesh` in the **Label** text field.

Size 1

1 In the **Model Builder** window, expand the **Coarse Mesh** node.

2 Right-click **Mapped 1** and choose **Size**.

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

4 Click the **Custom** button.

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

6 In the associated text field, type `2*lda0`.

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 Selected Physics Interfaces>Wavelength Domain**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electromagnetic Waves, Frequency Domain (ewfd)**.

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

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

STUDY 2

Step 1: Wavelength Domain

, to provide some more space for the **Graphics** window.

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

2 In the **Wavelengths** text field, type `lda0`.

3 In the **Home** toolbar, click **Compute**.

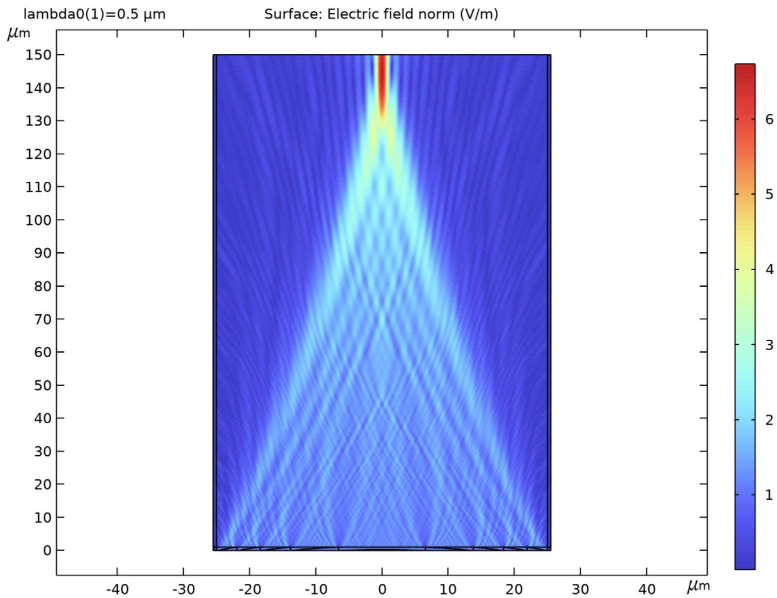
RESULTS

Electric Field (ewbe)

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

2 From the **View** list, choose **View 1**.

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



Helmholtz Equation (ewbe)

1 In the **Model Builder** window, right-click **Helmholtz Equation (ewfd)** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, type `Helmholtz Equation (ewbe)` in the **Label** text field.

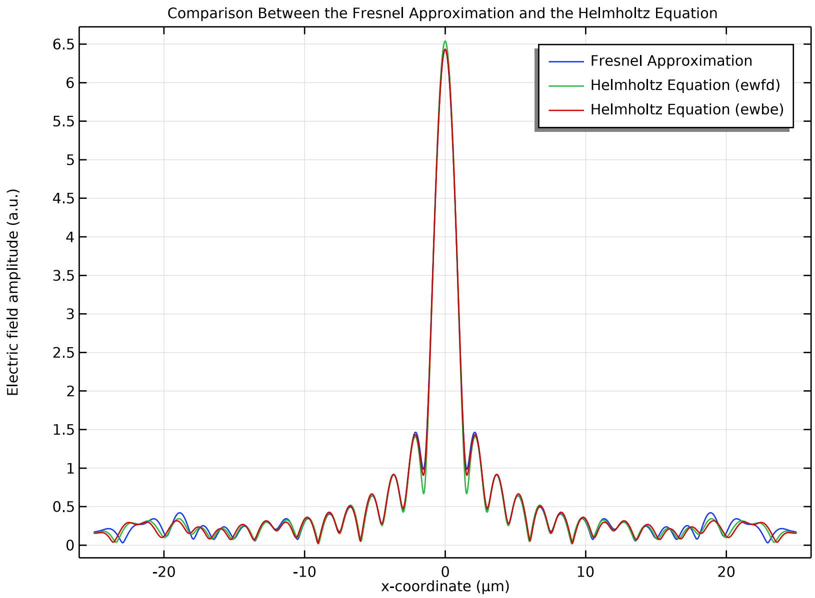
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `ewbe.normE`.

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

Legends
`Helmholtz Equation (ewbe)`

6 In the **Fresnel Versus Helmholtz Comparison** toolbar, click **Plot**.



The comparison shows the comparison is excellent also with the Electromagnetic Waves, Beam Envelopes interface. However, the simulation time is much faster, as the Electromagnetic Waves, Beam Envelopes interface can be performed with a much coarser mesh.

Creating and Running a Model Method

NEW METHOD

The last part of this instruction shows how to implement a Model Method. If you change the parameter `M`, the number of discretization levels, you need to change the number of points in the polygon in the **Fresnel Lens Zone** geometry part. However, once you have created the Model Method, you can call the Model Method from the GUI to automatically rebuild the geometry part and the geometry. You can only create and edit the Model Method using Windows, but you can run it from both Linux and Mac.

1 In the **Developer** toolbar, click **New Method**.

2 In the **New Method** dialog box, type `rebuildGeometry` in the **Name** text field.

3 Click **OK**.

rebuildGeometry

1 In the **Application Builder** window, under **Methods** click **rebuildGeometry**.

2 Copy the following code into the **rebuildGeometry** window:

```
// This method clears the coordinates in the Fresnel Lens Zone geometry parts
// and then calculates new coordinates

GeomSequence gs = model.geom("part1");
ModelParam modelParam = gs.localParam();

// Remove all local parameters, except the Digitization count for zone parameter,
Mi

for (String name : modelParam.varnames())
 if (!"Mi".equals(name))
 modelParam.remove(name);

// Read the digitization count parameter and generate local x-coordinate
parameters
int digitizationLevelCount = (int) model.param().evaluate("M");
for (int i = 0; i < digitizationLevelCount+1; i++) {
 String param = "x"+i;
 if (i == 0)
 modelParam.set(param, "if(zone > 1,sqrt(R^2-(R-(sag-(N+1-zone)*d))^2),0)",
"Left end of zone");
 else if (i == digitizationLevelCount)
 modelParam.set(param, "sqrt(R^2-(R-(sag-(N-zone)*d))^2)", "Right end of
zone");
 else {
 String paramOrder;
 if (i == 1)
 paramOrder = "1st";
 else if (i == 2)
 paramOrder = "2nd";
 else if (i == 3)
 paramOrder = "3rd";
 else
 paramOrder = ""+i+"th";
 modelParam.set(param, "sqrt(R^2-(z0+(N-zone+1)*d-(M-Mi+"+i+")*dm)^2)",
paramOrder+" step right end");
 }
}

// Generate local y-coordinate parameters
for (int i = 0; i < digitizationLevelCount+1; i++) {
 String param = "y"+i;
 if (i == 0)
 modelParam.set(param, "-dm", "Base level");
 else {
 String paramOrder;
 if (i == 1)
 paramOrder = "1st";
```

```
 else if (i == 2)
 paramOrder = "2nd";
 else if (i == 3)
 paramOrder = "3rd";
 else
 paramOrder = ""+i+"th";
 modelParam.set(param, "(Mi-"+i+")*dm", paramOrder+" level");
 }
}

// Set the new coordinate vectors for the polygon
String xCoordinates = "";
String yCoordinates = "";
for (int i = 0; i < digitizationLevelCount+1; i++) {
 xCoordinates += "x"+i+" "+"x"+i+(i < digitizationLevelCount ? " " : "");
 yCoordinates += "y"+i+" "+"y"+(i < digitizationLevelCount ? (i+1)+" " : "0");
}

GeomFeature polygon = gs.feature("pol1");
polygon.set("x", xCoordinates);
polygon.set("y", yCoordinates);

// Recreate the geometry
model.component("comp1").geom("geom1").run();
```

Now, go back to the **Model Builder**, change the digitization count parameter `M`, and then run the new model method.

METHODS

In the **Home** toolbar, click **Model Builder**.

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
M	8	8	Digitization level

4 In the **Home** toolbar, click **Run Method** and choose **rebuildGeometry**.

DEFINITIONS

Verify that the structure now is using eight steps per zone.

Hide for Physics 1

In the **Model Builder** window, under **Component 1 (comp1)>Definitions>View 3** right-click **Hide for Physics 1** and choose **Enable**.

Exit Plane

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Exit Plane**.
2 In the **Settings** window for **Explicit**, in the **Graphics** window toolbar, click ▾ next to **Go to Default View**, then choose **Go to View 3**.
3 Click the **Zoom Extents** button in the **Graphics** toolbar.