Created in COMSOL Multiphysics 6.2

Ray Release from a Dipole Antenna Source (3D)

Introduction

In this tutorial model, the far-field radiation pattern of a dipole antenna is computed in a 3D model component. Then, in a separate 3D model component, a ray is released using the far-field radiation pattern to initialize the ray's intensity, polarization, and phase.

Model Definition

This model offers some guidelines for multiscale modeling of electromagnetic wave propagation. The main idea is to use the Electromagnetic Waves, Frequency Domain interface to model wave propagation over a region that is similar in size to the wavelength, and then to use the Geometrical Optics interface to model propagation over much longer distances.

MOTIVATION

The Electromagnetic Waves, Frequency Domain interface can be used to obtain an accurate full-wave solution to Maxwell's Equations using the finite element method (FEM). However, the finite element mesh must be fine enough to resolve the individual oscillations of the electric field. Following the *Nyquist criterion*, the mesh should have at least 10 linear or 5 second-order mesh elements per wavelength. For example, if the wavelength is 500 nm and the domain is a cube that is 10 μm on each side, then the simulation domain is about 20 wavelengths in each direction. For second-order shape functions, a swept mesh of this domain would include one million elements.This 10 μm cube would be a very difficult problem to solve on a desktop computer; a room several meters in width is simply infeasible.

In contrast, the Geometrical Optics interface can be used to model electromagnetic wave propagation over very large distances because it does not require the mesh to be fine enough to resolve individual wavelengths. However, a ray tracing approach treats each ray as a wavefront that is locally plane, and therefore effects like diffraction are not considered.

One compromise is to solve Maxwell's Equations using the Electromagnetic Waves, Frequency Domain interface in the immediate vicinity of any object similar in size to the wavelength, and then trace rays over longer distances that lack such fine geometric details.

This example introduces a type of near-field to far-field coupling. First, the Electromagnetic Waves, Frequency Domain interface is used to solve for the radiation pattern of a dipole antenna. Then, the Geometrical Optics interface is used to release rays with intensity, polarization, and phase based on the far-field radiation pattern. In principle, the rays could then be traced over an arbitrarily large distance.

FAR-FIELD CALCULATION THEORY

The **Far-Field Domain** node and **Far-Field Calculation** subnode for the Electromagnetic Waves, Frequency Domain interface define a set of functions that describe the asymptotic behavior of electromagnetic radiation as it propagates outward from a source. The far field is calculated from the near field (FEM solution) using the Stratton–Chu formula. In 3D, the far field in the direction of a point p, denoted $\mathbf{E}_p$ (SI unit: V/m) is

$$\mathbf{E}_p = \frac{jk}{4\pi}\mathbf{r}_0 \times \int [\mathbf{n} \times \mathbf{E} - \eta \mathbf{r}_0 \times (\mathbf{n} \times \mathbf{H})] \exp(jk\mathbf{r} \cdot \mathbf{r}_0) dS$$

while in 2D the formula is

$$\mathbf{E}_p = \sqrt{\lambda}\frac{jk}{4\pi}\mathbf{r}_0 \times \int [\mathbf{n} \times \mathbf{E} - \eta \mathbf{r}_0 \times (\mathbf{n} \times \mathbf{H})] \exp(jk\mathbf{r} \cdot \mathbf{r}_0) dS$$

where

- $\mathbf{E}$ (SI unit: V/m) and $\mathbf{H}$ (SI unit: A/m) are the electric and magnetic field on the set of boundaries enclosing the radiation source,
- $\mathbf{r}_0$ (dimensionless) is the unit vector pointing from the origin to the point p,
- $\mathbf{n}$ (dimensionless) is the unit normal to the surface S,
- η (SI unit: Ω) is the impedance,

$$\eta = \sqrt{\mu/\varepsilon}$$

- k (SI unit: rad/m) is the wave number,
- λ (SI unit: m) is the wavelength, and
- $\mathbf{r}$ (SI unit: m) is the radius vector of the surface S.

The integration is over a surface enclosing the radiation source, which might be the set of boundaries where a **Scattering Boundary Condition** is defined, or the inside surface of a **Perfectly Matched Layer**.

In the COMSOL implementation, the far-field function gives the field at a distance of 1 m from the radiation source. The asymptotic behavior of the electric field, then, is given by

$$\mathbf{E}_{\mathrm{far}} = \mathbf{E}_p \frac{\exp(-jkr) \times (1\ \mathrm{m})}{r}$$

where $\mathbf{E}_{\mathrm{far}}$ is the electric field at a distance r from the radiation source along a line that passes through point p. For this asymptotic limit to be valid, r should be larger than the wavelength and any geometric details of the radiation source.

MODEL VALIDATION

In a 3D model the number of degrees of freedom needed to solve for the electric field using FEM grows quite rapidly with increasing geometry size. Thus, the simulation domain is only a few wavelengths wide. Three symmetry planes are used to reduce the number of degrees of freedom further.To confirm that the Geometrical Optics interface correctly initializes the ray intensity, polarization, and phase information, this verification model uses three physics interfaces:

- Electromagnetic Waves, Frequency Domain (emw): defined in a small domain in the first 3D model component to perform the far field calculation.
- Electromagnetic Waves, Frequency Domain 2 (emw2): defined in a larger domain in the first 3D model component to directly compute the electric field over several wavelengths, for validation.
- Geometrical Optics: (gop): defined in a second 3D model component. A ray is released using the dedicated **Release from Far-Field Radiation Pattern** feature. In the Geometrical Optics interface, the electric field along individual rays can be obtained because each ray stores its own information about Stokes parameters and instantaneous phase.

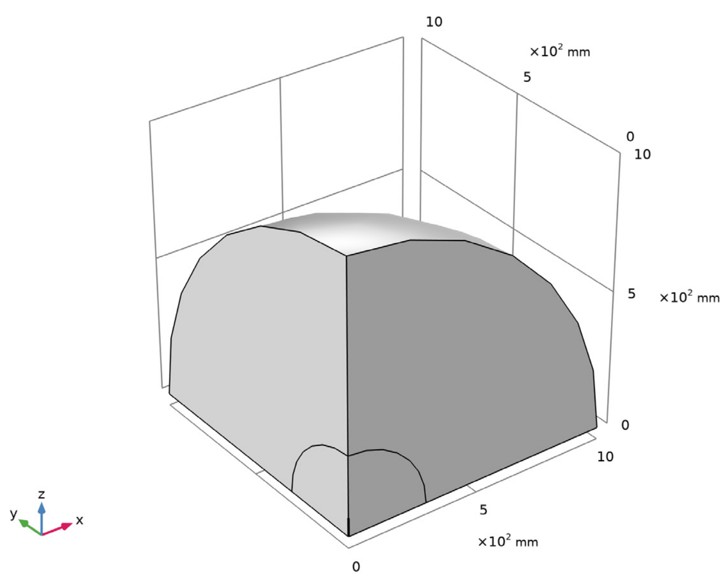

Figure 1: The simulation domain uses a small inner sphere to compute the far-field radiation pattern, and a larger circle to directly compute the electric field for validation purposes. Three symmetry planes are used to reduce the number of degrees of freedom in the FEM calculation.

Results and Discussion

In this example the radiation source is a dipole antenna. The surfaces of the antenna are treated as a **Perfect Electric Conductor**. The antenna is excited using the **Lumped Port** boundary condition.

In Figure 2 the z-component of the electric field is plotted over several wavelengths. This component of the electric field is strongest in the horizontal (x and y) directions and weakest in the vertical (z) direction.

In the Geometrical Optics interface, a ray is released in the $(1,0,1)$ direction. To verify that the amplitude and phase of the electric field along this ray are both correct, a **Cut Line** dataset is inserted into the first 3D geometry and the electric field is plotted along this line. The two plots are directly compared in Figure 3. It is reasonable for the two plots to differ significantly at the origin; the far field is an asymptotic solution and does not apply so close to the radiation source. The agreement between the two curves becomes better as the ray propagates away from the dipole antenna.

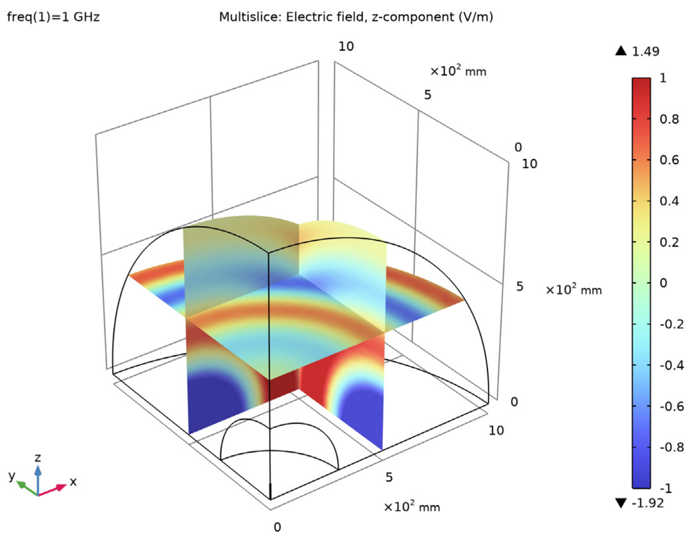

Figure 2: Electric field z-component, computed using FEM.

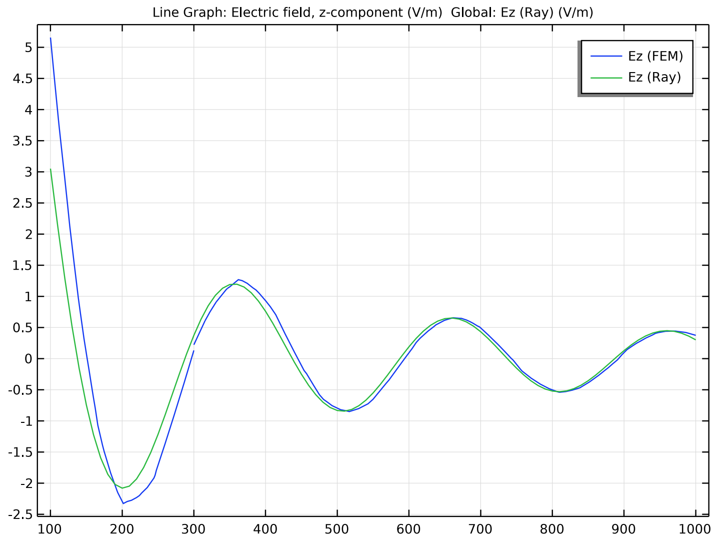

Figure 3: Comparison of the electric field z-component computed using FEM and ray optics.

Application Library path: `Ray_Optics_Module/Tutorials/ray_release_from_dipole_antenna_source_3d`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

Add two instances of the Electromagnetic Waves, Frequency Domain interface: one for far-field calculation only, and one for verification by extending the mesh out to several wavelengths.

2 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

3 Click **Add**.

4 Click **Add**.

5 Click **Study**.

6 In the **Select Study** tree, select **General Studies>Frequency Domain**.

7 Click **Done**.

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
f0	1[GHz]	1E9 Hz	Frequency
lam0	c_const/f0	0.29979 m	Vacuum wavelength
theta0	45[deg]	0.7854 rad	Polar angle
phi0	0[deg]	0 rad	Azimuthal angle
L0x	sin(theta0)*cos(phi0)	0.70711	Ray direction, x-component
L0y	sin(theta0)*sin(phi0)	0	Ray direction, y-component
L0z	cos(theta0)	0.70711	Ray direction, z-component

GEOMETRY 1

Construct the dipole antenna geometry. The antenna will be surrounded by two concentric spheres. The small sphere will be used to compute the far-field radiation pattern. The large sphere will be used to solve for the field directly, to validate the Ray Optics model results.

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

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

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

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `2.5`.

Keep the default height of `1 mm`.

Cylinder 2 (cyl2)

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

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

3 In the **Radius** text field, type 2.5.

4 In the **Height** text field, type 69.

5 Locate the **Position** section. In the **z** text field, type 1.

Sphere 1 (sph1)

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

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

3 In the **Radius** text field, type 300.

Sphere 2 (sph2)

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

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

3 In the **Radius** text field, type 1000.

4 Click **Build All Objects**.

Difference 1 (dif1)

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

2 Select the objects **sph1** and **sph2** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **cyl1** and **cyl2** only.

Block 1 (blk1)

Only solve for the field in one octant of the domain by exploiting plane symmetries.

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

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

3 In the **Width** text field, type 1000.

4 In the **Depth** text field, type 1000.

5 In the **Height** text field, type 1000.

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 Selected**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar. Compare the resulting plot to Figure 1.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

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

2 Select Domain 1 only. This interface will be used for the far-field calculation. It is only necessary to select the smaller domain surrounding the antenna.

Lumped Port 1

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

2 Select Boundary 8 only. This is the small curved surface closest to the origin.

3 In the **Settings** window for **Lumped Port**, locate the **Lumped Port Properties** section.

4 From the **Type of lumped port** list, choose **User defined**.

5 In the h_{port} text field, type `2[mm]`.

6 In the w_{port} text field, type `2.5*2*pi[mm]`.

7 Specify the $\mathbf{a}_{\mathrm{h}}$ vector as

0	x
0	y
1	z

Scattering Boundary Condition 1

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

2 Select Boundary 6 only. This is the outer surface of the small sphere.

Far-Field Domain 1

In the **Physics** toolbar, click **Domains** and choose **Far-Field Domain**.

Far-Field Calculation 1

1 In the **Model Builder** window, expand the **Far-Field Domain 1** node, then click **Far-Field Calculation 1**.

2 Select Boundary 6 only. This is the outer surface of the small sphere.

Notice the **Far-field variable name** text field. The default name is `Efar`. This name must match the corresponding **Far-field variable name** in the **Release from Far-Field Radiation Pattern** node, which will be added to the Geometrical Optics interface later.

3 In the **Settings** window for **Far-Field Calculation**, locate the **Far-Field Calculation** section.

4 Select the **Symmetry in the x=0 plane** check box.

5 Select the **Symmetry in the y=0 plane** check box.

6 Select the **Symmetry in the z=0 plane** check box.

7 From the **Symmetry type** list, choose **Symmetry in H (PEC)**.

Perfect Magnetic Conductor 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Magnetic Conductor**.

2 Select Boundaries 1 and 2 only. These are the flat surfaces parallel to the *xz* and *yz* planes. The default **Perfect Electric Conductor** boundary condition is applied to the *xy* plane.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EMW2)

The second interface will be used to compute the electric field over several wavelengths using FEM, in order to validate the Ray Optics model.

In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain 2 (emw2)**.

Lumped Port 1

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

2 Select Boundary 8 only. This is the small curved surface closest to the origin.

3 In the **Settings** window for **Lumped Port**, locate the **Lumped Port Properties** section.

4 From the **Type of lumped port** list, choose **User defined**.

5 In the h_{port} text field, type `2[mm]`.

6 In the w_{port} text field, type `2.5*2*pi[mm]`.

7 Specify the $\mathbf{a}_{\mathrm{h}}$ vector as

0	x
0	y
1	z

Scattering Boundary Condition 1

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

2 Select Boundary 7 only. This is the outer surface of the large sphere.

Perfect Magnetic Conductor 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Magnetic Conductor**.

2 Select Boundaries 1, 2, 4, and 5 only. These are the flat surfaces parallel to the *xz* and *yz* planes. The default **Perfect Electric Conductor** boundary condition is applied to the *xy* plane.

MATERIALS

Assign air as the material for all domains.

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.

STUDY 1

Step 1: Frequency Domain

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

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

3 In the **Frequencies** text field, type `f0`.

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

RESULTS

Electric Field (emw2)

The default plots include **Multislice** plots of the electric field norm for each interface. Modify one of these plots to instead show the real part of the *z*-component.

Multislice

1 In the **Model Builder** window, expand the **Results>Electric Field (emw2)** node, then click **Multislice**.

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

3 In the **Expression** text field, type `emw2.Ez`.

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

5 In the **Minimum** text field, type `-1`.

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

7 In the **Electric Field (emw2)** toolbar, click **Plot**. Compare the resulting plot to Figure 2.

Now set up a second model component for the Ray Optics simulation.

ADD COMPONENT

Right-click **Results>Electric Field (emw2)>Multislice** and choose **3D**.

GEOMETRY 2

In this example, the ray will just propagate in a straight line, but the Geometrical Optics interface requires a boundary condition to be applied on at least one surface. The plane used here might represent the flat ground.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `-3[m]`.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `3[m]`.

ADD PHYSICS

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

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

3 In the tree, select **Optics>Ray Optics>Geometrical Optics (gop)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component 2** in the window toolbar.

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

GEOMETRICAL OPTICS (GOP)

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

2 In the **Maximum number of secondary rays** text field, type `0`.

3 Locate the **Intensity Computation** section. From the **Intensity computation** list, choose **Compute intensity and power**.

4 Select the **Compute phase** check box.

5 Locate the **Additional Variables** section. Select the **Compute optical path length** check box.

Release from Far-Field Radiation Pattern 1

1 Right-click **Component 2 (comp2)>Geometrical Optics (gop)** and choose **Release from Far-Field Radiation Pattern**.

2 In the **Settings** window for **Release from Far-Field Radiation Pattern**, locate the **Ray Direction Vector** section.

3 From the **Ray direction vector** list, choose **Conical**.

4 Specify the **r** vector as

L0x	x
L0y	y
L0z	z

5 In the N_w text field, type `1`.

6 In the α text field, type `1[deg]`.

When the conical release uses only a single ray, that ray will always be released along the cone axis. However, it is still necessary to define a cone angle, because the solid angle subtended by each ray is used to initialize its power.

Wall 1

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

2 Select Boundary 1 only.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Electromagnetic Waves, Frequency Domain (emw)** and **Electromagnetic Waves, Frequency Domain 2 (emw2)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Ray Tracing**.

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: Ray Tracing

1 In the **Settings** window for **Ray Tracing**, locate the **Study Settings** section.

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

Use the default maximum optical path length, `1 m`.

3 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

5 From the **Study** list, choose **Study 1, Frequency Domain**.

6 Click **Compute**.

RESULTS

In the **Model Builder** window, under **Results** click **Datasets**.

Cut Line 3D 1

1 In the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **X** to `100*L0x`.

4 In row **Point 1**, set **Y** to `100*L0y`.

5 In row **Point 1**, set **Z** to `100*L0z`.

6 In row **Point 2**, set **X** to `1000*L0x`.

7 In row **Point 2**, set **Y** to `1000*L0y`.

8 In row **Point 2**, set **Z** to `1000*L0z`.

9 Click **Plot**. The cut line should extend radially outward from the center of the dipole antenna.

1D Plot Group 6

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

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

3 From the **Dataset** list, choose **Cut Line 3D 1**.

Line Graph 1

1 Right-click **1D Plot Group 6** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain 2>Electric>Electric field - V/m>emw2.Ez - Electric field, z-component**.

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

4 In the **Expression** text field, type `sqrt(x^2+y^2+z^2)`.

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

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

7 In the table, enter the following settings:

Legends
`Ez (FEM)`

8 In the **1D Plot Group 6** toolbar, click **Plot**.

1D Plot Group 6

Now add a **Global** plot to show the electric field amplitude along the ray as a function of optical path length.

Global 1

1 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Global**.

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

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

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

5 Click **Range**.

6 In the **Integer Range** dialog box, type `11` in the **Start** text field.

7 In the **Stop** text field, type `101`.

8 Click **Replace**.

The manual indices exclude the ray release point, where the ray intensity becomes infinite under the ray optics approximation.

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

10 In the table, enter the following settings:

Expression	Unit	Description
`gop.sum(gop.Ez)`	`V/m`	`Ez (Ray)`

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

12 In the **Expression** text field, type `gop.sum(gop.L)`.

13 From the **Unit** list, choose **mm**.

14 In the **1D Plot Group 6** toolbar, click **Plot**. Compare the resulting plot to Figure 3.