Propagation of a 3D Gaussian Beam Laser Pulse

Introduction

This is a 3D version of the model "Second Harmonic Generation of a Gaussian Beam" on page 292. The differences are that the nonlinear material parameters have been removed, the geometry is smaller, and the laser pulse is shorter. This is done to reduce the execution time and the size of the model file shipped with the product.

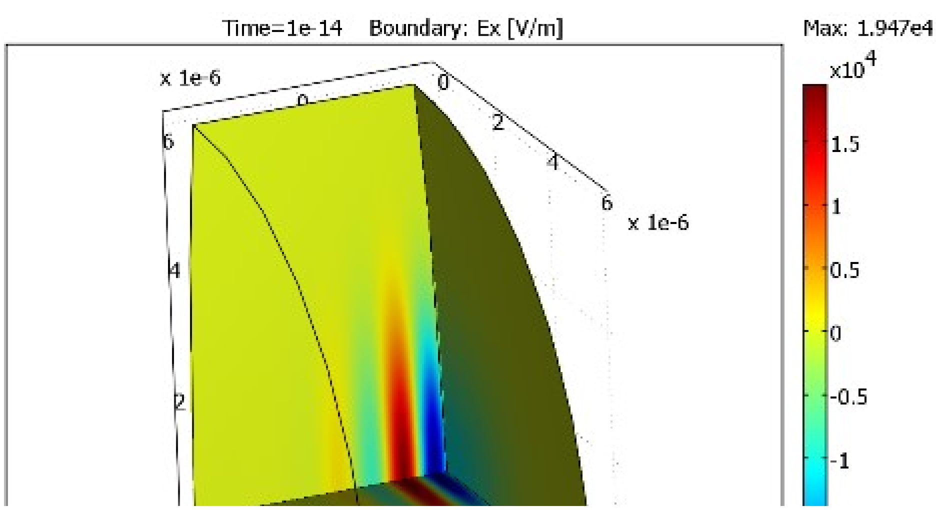
Model Definition

All the details about Gaussian laser beams are covered in the model "Second Harmonic Generation of a Gaussian Beam" on page 292. This model uses the same YAG laser pulse, but with a characteristic time of 4 fs. Because the Gaussian pulse has spherical wave fronts when it approaches and leaves the minimum waist, the input and output boundaries are made spherical.

The model also use the symmetry of the cylindrical shape so that it is only necessary to simulate one quarter of the full volume. The perfect electric and perfect magnetic boundary conditions are used as symmetry conditions one these boundaries. It is not straightforward to use axial symmetry, because the polarization of the beam is in the x direction.

Results and Discussion

Notice the spherical fronts in the following figures. These figures show the pulse that propagates at two different times: at the entry and just after passing the minimum waist.



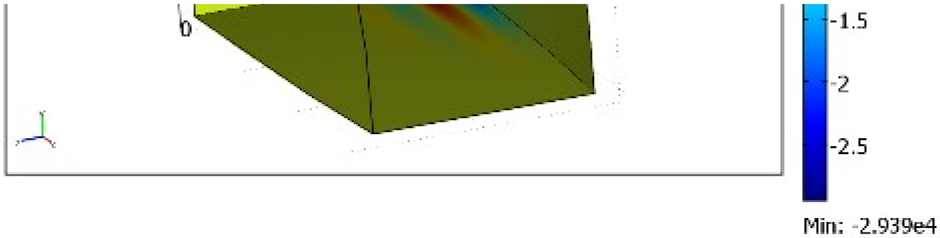


Figure 4-14: The pulse entering the domain at t = 10 fs. The spot size is shown on the input boundary to the left.

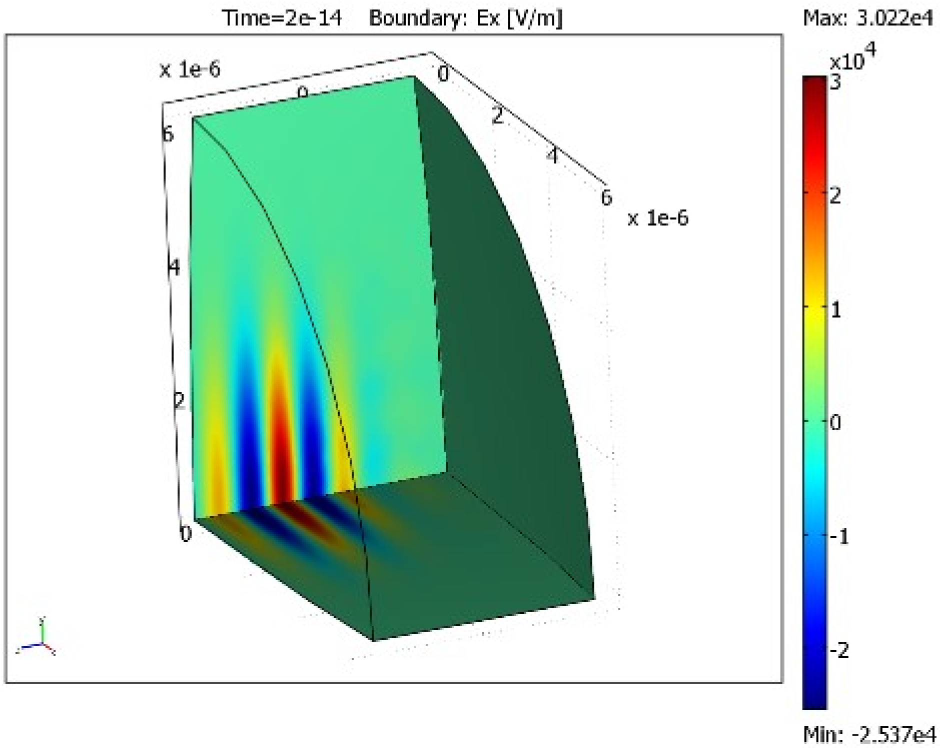


Figure 4-15: After 20 fs the pulse has passed the minimum waist and soon reaches the output boundary.

Model Library path: `RF_Module/Optics_and_Photonics/gaussian_beam_3d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **3D** from the **Space dimension** list.

2 Select the **RF Module>Electromagnetic Waves>Transient propagation** application mode. Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

2

In the **Constants** dialog box, define the following constants with names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
w0	2[um]	Minimum waist of laser beam
lambda0	1.06[um]	Wavelength of laser beam
E0	30[kV/m]	Peak electric field
z0	pi*w0^2/lambda0	Peak electric field z position
k0	2*pi/lambda0	Wave number
t0	10[fs]	Pulse time delay
dt	4[fs]	Pulse width

GEOMETRY MODELING

All the dialog boxes for specifying the primitive objects are available from the **Draw>Specify Objects** menu. The first column in the tables below contains the labels of the geometric objects. These are automatically generated by COMSOL Multiphysics, and you do not have to enter them. Just check that you get the correct label for the objects that you create.

Begin by drawing the cross section of the simulation domain followed by an extrusion to 3D. The input and output boundaries are then adjusted so they get a slight spherical shape, consistent with the spherical wave fronts of the Gaussian pulse.

1 From the **Draw** menu, select **Work-Plane Settings**.

2 In the **Work-Plane Settings** dialog box, select the **x-y** plane and enter -2e-6 in the **z** edit field. Click **OK**.

3 Draw a circle with the properties according to the table below.

NAME	RADIUS	BASE	(X, Y)
C1	6e-6	Center	(0, 0)

4 Draw rectangles with the properties given in the following table:

NAME	WIDTH	HEIGHT	BASE	CORNER
R1	6e-6	1.2e-5	Corner	(-6e-6, -6e-6)
R2	6e-6	6e-6	Corner	(0, -6e-6)

5 Click the **Zoom Extents** button on the Main toolbar.

6 Select both rectangles, R1 and R2, then click the **Union** toolbar button.

7 Click the **Delete Interior Boundaries** toolbar button.

8

Press Ctrl+A to select all objects, then click the **Create Composite Object** button on the Draw toolbar. In the dialog box that appears, enter `C1-CO1` in the **Set formula** edit field, then click **OK**.

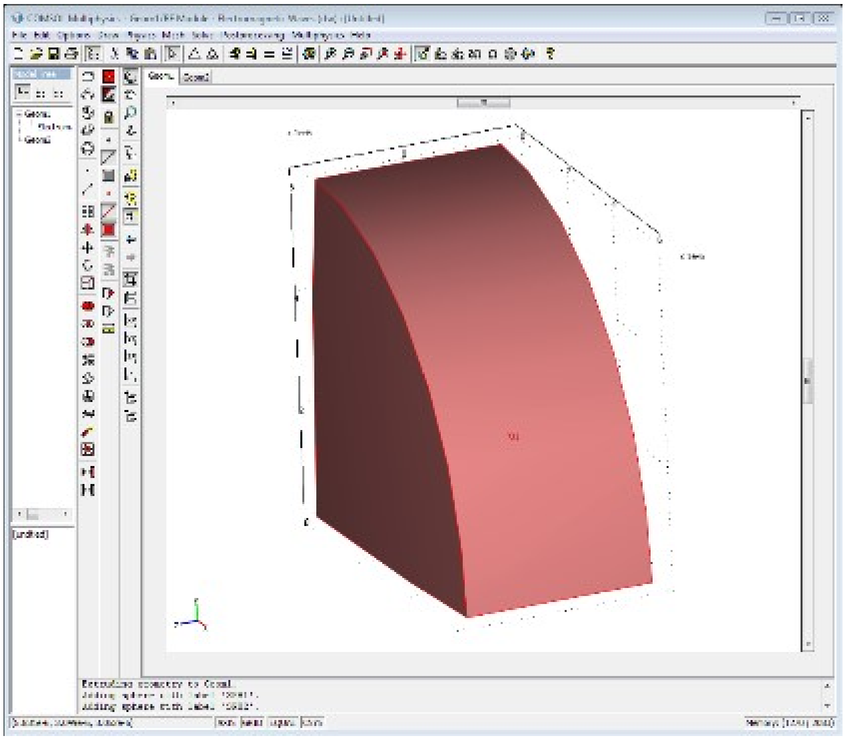
9 From the **Draw** menu, select **Extrude**. In the **Extrude** dialog box, specify the **Distance** as `4e-6`, then click **OK** to create the object EXT1.

10 It is desired to have the spherical wave fronts of the beam coincide with the input and output boundaries. To achieve this, for each of these two boundaries create a sphere with radius equal to the value of the expression `R`—the "analytical radius" along the *z*-axis that you will define shortly—evaluated at the boundary, place it on the *z*-axis so that it just touches the boundary, and then take the intersection with EXT1. Thus, create two spheres with the properties in the following table (the axis base point is the centerpoint):

NAME	RADIUS	AXIS BASE POINT
SPH1	7.22713e-5	(0, 0, 7.22713e-5-2e-6)
SPH2	7.22713e-5	(0, 0, -7.22713e-5+2e-6)

11 Press Ctrl+A to select all objects, then click the **Intersection** button.

12 Click the **Headlight** toolbar button. You should now see the geometry shown in the figure below. Note that the input and output boundary surfaces are spherical now.



PHYSICS SETTINGS

1 Go to geometry **Geom1** by clicking the **Geom1** tab.

2 From the **Options** menu, choose **Expressions>Scalar Expressions**.

3 In the **Scalar Expressions** dialog box, define the following variables with names, expressions, and descriptions (optional):

NAME	VALUE	DESCRIPTION
w	w0*sqrt(1+(z/z0)^2)	Analytical waist function along z
eta	atan(z/z0)	Analytical angle along z
R	z*(1+(z0/z)^2)	Analytical radius along z
r	sqrt(x^2+y^2)	Radial coordinate
c0	1/sqrt (epsilon0_rfw*mu0_rfw)	Speed of light
omega0	2*pi*c0/lambda0	Angular frequency

4 Click **OK**.

5 From the **Options** menu, choose **Expressions>Boundary Expressions**.

6 In the **Boundary Expressions** dialog box, define the following variables with names and expressions:

SETTING	BOUNDARY 3	ALL OTHER
E_bnd	w0/w*exp(-r^2/w_bnd^2)* cos(omega0*t-k0*z+eta_bnd-r^2*k0/ (2*R_bnd))	
E_pulse	exp(-(t-t0)^2/dt^2)	

Because of the use of an integration coupling variable (w_bnd), COMSOL Multiphysics cannot determine the unit for E_bnd and warns for an inconsistent unit here and in the specification of the scattering boundary condition. You can disregard these warnings.

7 Click **OK**.

8 From the **Options** menu, select **Integration Coupling Variable>Point Variables**.

9 In the **Point Integration Variables** dialog box, define the following variables with names and expressions. Use **Global destination** for all variables.

NAME	POINT 1	ALL OTHERS
w_bnd	w	
eta_bnd	eta	
R_bnd	R	

Boundary Conditions

1

From the **Physics** menu, open the **Boundary Settings** dialog box and enter the settings according the following two tables (leave all fields not specified at their default values):

SETTINGS	BOUNDARY 3	BOUNDARY 4
Boundary condition	Scattering boundary	Scattering boundary
$E_{0,x}$	`E0*E_pulse*E_bnd`	`0`
$E_{0,y}$	`0`	`0`
$E_{0,z}$	`0`	`0`

SETTING	BOUNDARY 2	BOUNDARIES 1, 5
Boundary condition	Perfect magnetic conductor	Perfect electric conductor

2 Click **OK**.

Subdomain Settings

Use the default values for the subdomain settings.

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.

2 Click the **Custom mesh size** button and enter `1e-7` in the **Maximum element size** edit field.

3 Click the **Advanced** tab.

4 Enter `0.12` in the **x-direction scale factor** and **y-direction scale factor** edit fields. This keeps the small mesh size in the z direction but stretches the elements in the x and y directions.

5 Click the **Remesh** button, then click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu, choose **Solver Parameters**.

2 In the **Times** edit field type `range(0,5e-15,2e-14)`.

3 Click the **Settings** button.

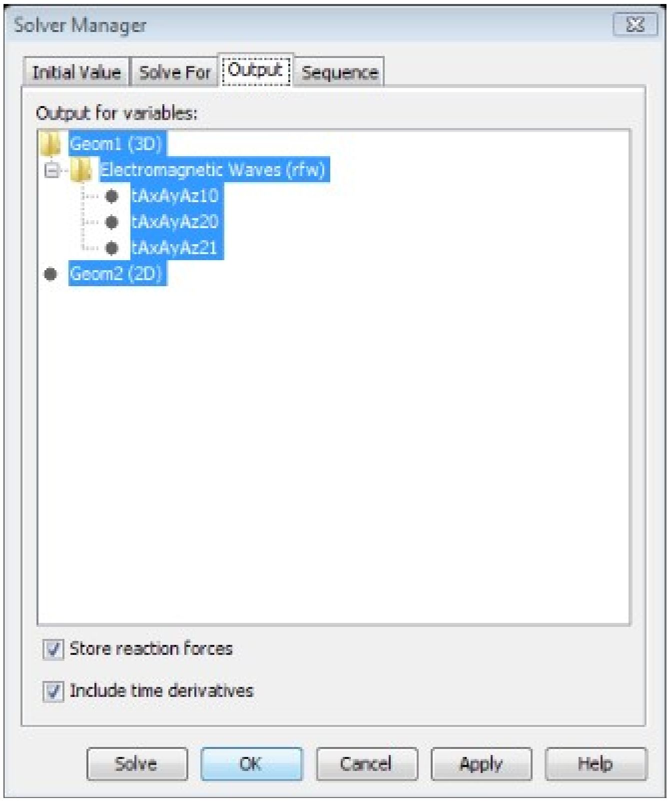
4 In the dialog box that appears, select the **Coarse solver** item in the tree view.

5 Choose **Off** from the **Check tolerances** list and click **OK**.

6 Click the **Time Stepping** tab and select the **Manual tuning of step size** check box. Enter `1e-16` in the **Initial time step** and **Maximum time step** edit fields.

7 Click **OK**.

8 From the **Solve** menu, choose **Solver Manager**. Click the **Output** tab and select the **Include time derivatives** check box. This ensures that the time derivatives of the $\mathbf{A}$ field, which is the $\mathbf{E}$ field, is calculated more accurately.



9 Click **OK**.

10 Click the **Solve** button on the Main toolbar.

The solving process takes some time because this is a large problem that takes about 300 time steps.

POSTPROCESSING AND VISUALIZATION

Plot the x-component of the $\mathbf{E}$-field on the boundaries. Some boundaries are suppressed (hidden) to make it possible to get a view into the simulation volume.

1 Select **Plot Parameters** from the **Postprocessing** menu.

2 Make sure that the **Boundary** and **Geometry edges** check boxes are selected under the **General** tab.

3 Click the **Boundary** tab and enter `Ex` in the **Expression** edit field.

4 Click **OK**.

5 From the **Options** menu, select **Suppress>Suppress Boundaries**. In the **Suppress Boundaries** dialog box, select boundaries 4 and 5. Click **OK**.

6 Compare the results at times `1e-14` and `2e-14` to see how the pulse propagates. These results are plotted in Figure 4-14 on page 306 and in Figure 4-15 on page 307.