Created in COMSOL Multiphysics 5.6

Transient Modeling of a Coaxial Cable

Introduction

Time-domain simulations of Maxwell's equations are useful for

- observing transient phenomena,
- finding the time it takes for a signal to propagate, or
- modeling materials that are nonlinear with respect to the electric or magnetic field strength.

This example considers a pulse propagating down a coaxial transmission line for three different termination types: short, open, and matched. The signal propagation time is deduced from the reflected waves detected at the input port.

Model Definition

The model setup, schematically shown in Figure 1, is a short section of an air-filled coaxial transmission line. The symmetry of the structure allows for a 2D axisymmetric model geometry.

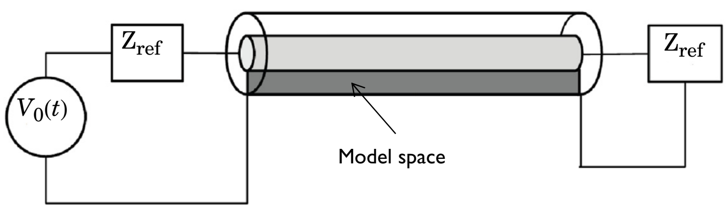


Figure 1: Schematic of a section of a coaxial transmission line connected to a transient voltage source and a load.

At one end of the coaxial cable, or coax for short, a *lumped port* boundary condition excites the structure; specify a transient excitation pulse, $V_0(t)$, by using a Gaussian pulse-windowed sine function. Apply the excitation as a current of magnitude $I(t) = V_0(t) / Z_{ref}$ flowing tangentially to the excitation boundary. Here Z_{ref} refers to the specified characteristic impedance between the voltage generator and the model.

At the other end of the coax, consider, in turn, three different boundary conditions:

1. *perfect electric conductor* (PEC) — to simulate the short condition;
2. *perfect magnetic conductor* (PMC) — to simulate an open condition; and
3. *lumped port* — to simulate a matched load.

On the walls of the coax, apply a PEC boundary condition; this condition is appropriate when both skin depth and losses in the conductors are very small.

Use a triangular mesh with the maximum element size chosen such that there are at least two elements in the radial direction and at least eight elements per wavelength.

The only changes required to the default solver settings are to tighten the relative tolerance from the default value, and to adjust the timespan and output time steps. The internal time steps taken by the solver are auto-selected based on the specified relative tolerance.

Results and Discussion

Figure 2 shows the results of the transient simulation for the three different termination types. The figure plots the radial component of the electric field at the input port as a function of time for the three different termination conditions. The short (PEC) and open (PMC) terminations reflect waves that are 180° out of phase, and the matched load produces almost no reflections. From the reflected waves in the plot, you can read off an approximate signal propagation time through the air-filled transmission line of (0.37 – 0.10) / 2 ns = 0.135 ns. This matches the expected value of L_{coax} / c, where L_{coax} = 40 mm is the length of the line and c is the speed of light in air.

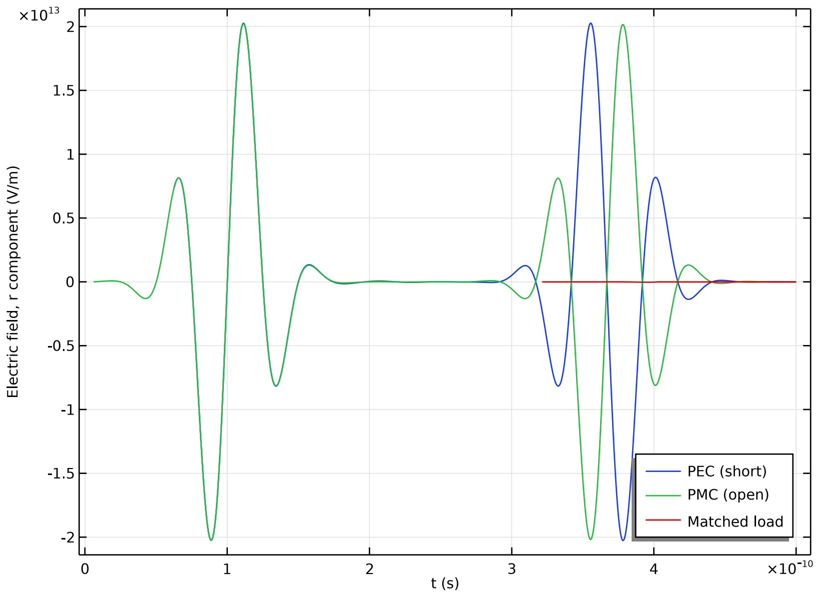


Figure 2: Radial component of electric field at the input port versus time for three different termination conditions: short (blue), open (green), and matched load (red).

Application Library path: `RF_Module/Verification_Examples/coaxial_cable_transient`

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 **Radio Frequency>Electromagnetic Waves, Transient (temw)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 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
r_coax	1[mm]	0.001 m	Coax inner radius
R_coax	2[mm]	0.002 m	Coax outer radius
L_coax	40[mm]	0.04 m	Length of coax core into cavity
f	20[GHz]	2E10 Hz	Pulse frequency
L	c_const/f	0.01499 m	Wavelength, free space
T	1/f	5E-11 s	Period
h_max	min(L/8,(R_coax-r_coax)/2)	5E-4 m	Maximum element size

Next, define the excitation, V0(t), in terms of a Gaussian pulse and a sine function.

Define a Gaussian pulse.

Gaussian Pulse 1 (gp1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Gaussian Pulse**.

2 In the **Settings** window for **Gaussian Pulse**, type gauss_pulse in the **Function name** text field.

3 Locate the **Parameters** section. In the **Location** text field, type 2*T.

4 In the **Standard deviation** text field, type T/2.

Now use this pulse in an analytic function for V0(t):

Analytic 1 (an1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `V0` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `gauss_pulse(t)*sin(2*pi*f*t)`.

4 In the **Arguments** text field, type `t`.

5 Locate the **Units** section. In the **Arguments** text field, type `s`.

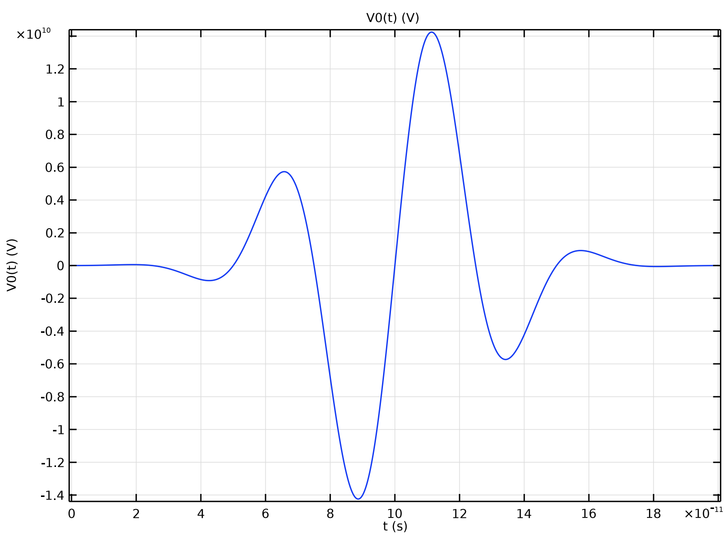
6 In the **Function** text field, type `V`.

To plot the function, you need to specify a suitable time interval.

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

Argument	Lower limit	Upper limit
t	0	0.2[ns]

8 Click to collapse the **Plot Parameters** section. Click **Plot**.



GEOMETRY 1

An elongated rectangle offset from the symmetry axis represents the straight coaxial cable.

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 `R_coax-r_coax`.

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

5 Locate the **Position** section. In the **r** text field, type `r_coax`.

6 Click **Build All Objects**.

DEFINITIONS

Set up a point probe for plotting the electric field component Er while solving. You will also use this plot to reproduce Figure 2.

Domain Point Probe 1

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

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

3 In row **Coordinates**, set **r** to `r_coax`.

4 Select the **Snap to closest boundary** check box.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Transient>Electric>Electric field - V/m>temw.Er - Electric field, r component**.

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Now set up the physics. Begin by defining the Lumped port input condition.

Lumped Port 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Transient (temw)** and choose **Lumped Port**.

2 Select Boundary 2 only, (the bottom boundary).

For the first port, wave excitation is **on** by default.

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

4 In the V_0 text field, type `V0(t)`.

5 Locate the **Settings** section. In the Z_{ref} text field, type `(Z0_const/2/pi)*log(R_coax/r_coax)`.

The open case uses a PMC condition at the termination.

Perfect Magnetic Conductor 1

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

2 Select Boundary 3 only, (the top boundary).

Finally, define a lumped port condition to use for the matched load case.

Lumped Port 2

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

2 Select Boundary 3 only.

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

4 In the Z_{ref} text field, type `(Z0_const/2/pi)*log(R_coax/r_coax)`.

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.

MESH 1

Free Triangular 1

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

Size

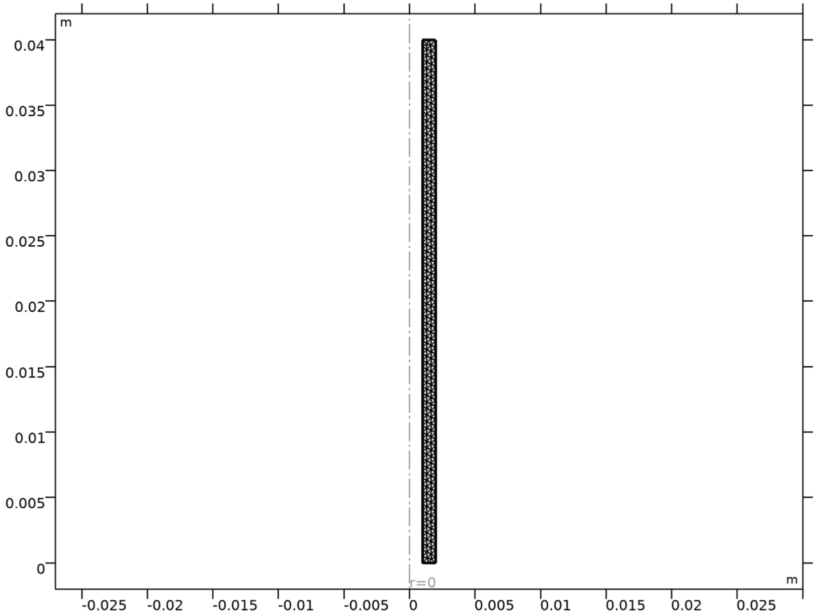
1 In the **Model Builder** window, click **Size**.

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 `h_max`.

5 Click **Build All**.



STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,T/24,10*T)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `0.0001`.

To study the short termination case first, disable the PMC and lumped port conditions so that the default PEC condition is activated on the termination boundary.

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box, disable **Perfect Magnetic Conductor 1** and **Lumped Port 2**.

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

RESULTS

2D Plot Group 1

Click on the **Probe Plot 1** tab to place it in focus.

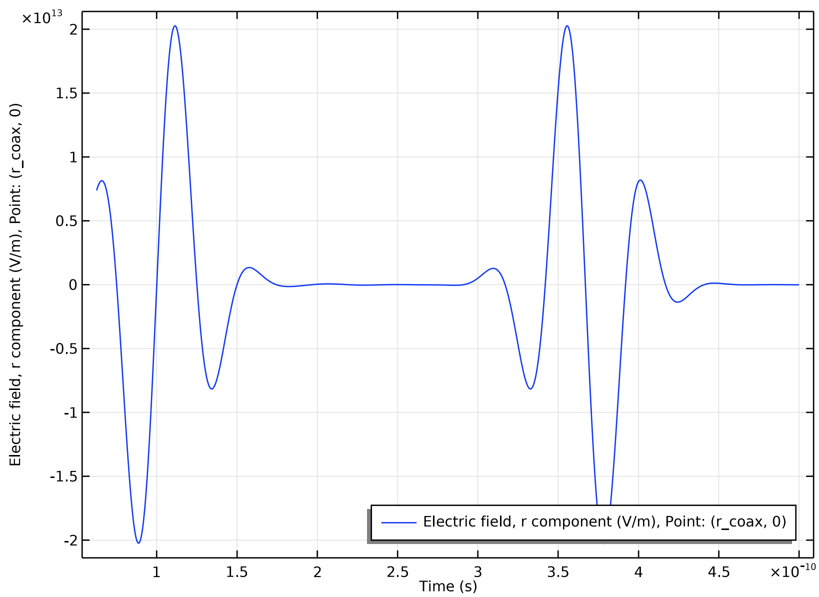
Probe Plot Group 2

1 In the **Model Builder** window, click **Probe Plot Group 2**.

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

3 From the **Position** list, choose **Lower right**.

When the solver finishes the plot should look like that in the figure below.



2D Plot Group 1

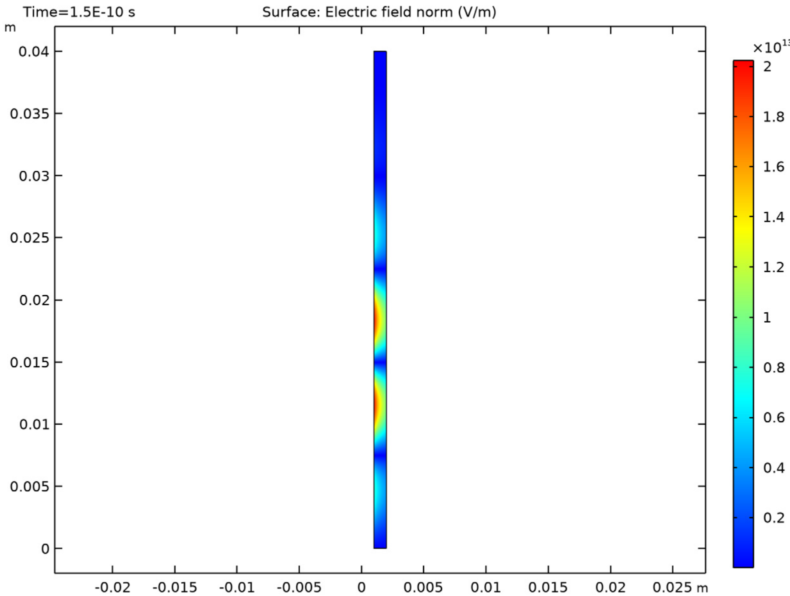
The default surface plot shows the electric field in the coax at the end of the simulation interval. Because the transient has died out, the solution you see is only noise. Modify the time to get a more interesting plot.

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

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

3 From the **Time (s)** list, choose **1.5E-10**.

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



Now turn to the open termination case.

DEFINITIONS

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions> Domain Point Probe 1** click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, click to expand the **Table and Window Settings** section.

3 From the **Output table** list, choose **New table**.

With these settings you get a plot for the short and open termination cases in the same plot window.

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Perfect Magnetic Conductor 1

In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Transient (temw)** right-click **Perfect Magnetic Conductor 1** and choose **Enable**.

STUDY 1

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

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

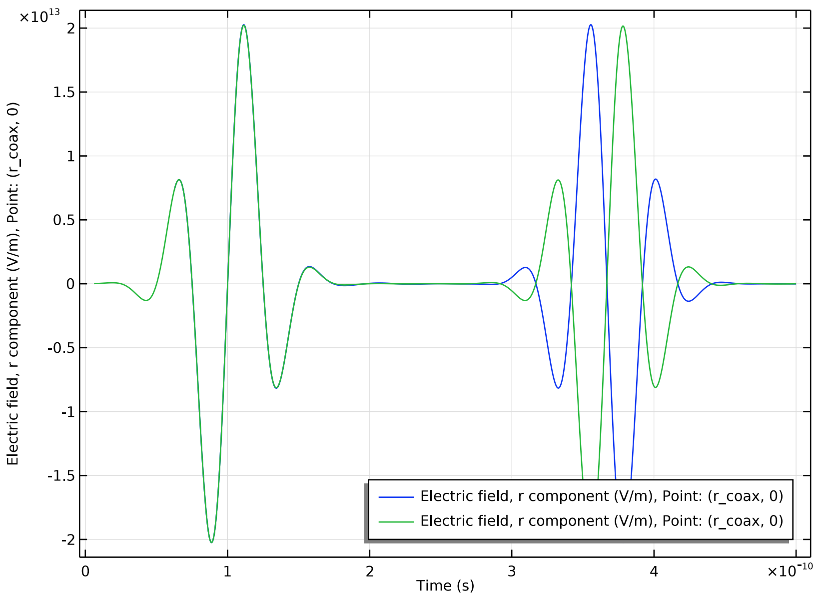
3 Clear the **Generate default plots** check box.

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

RESULTS

Probe Plot Group 2

The reflected waves for the short and open terminations are 180 degrees out of phase.



Finally, activate the matched load case.

DEFINITIONS

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions> Domain Point Probe 1** click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, locate the **Table and Window Settings** section.

3 From the **Output table** list, choose **New table**.

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Lumped Port 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Transient (temw)** right-click **Lumped Port 2** and choose **Enable**.

Note that you do not need to disable the PMC condition because it is overridden by the lumped port.

STUDY 1

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

RESULTS

Probe Plot Group 2

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.
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 `t (s)`.
5 Select the **y-axis label** check box.
6 In the associated text field, type `Electric field, r component (V/m)`.
7 In the **Probe Plot Group 2** toolbar, click **Plot**.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Probe Plot Group 2** node, then click **Probe Table Graph 1**.
2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.
3 From the **Legends** list, choose **Manual**.
4 In the table, enter the following settings:

Legends
PEC (short)

Probe Table Graph 2

1 In the **Model Builder** window, click **Probe Table Graph 2**.
2 In the **Settings** window for **Table Graph**, locate the **Legends** section.
3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`PMC (open)`

Probe Table Graph 3

1 In the **Model Builder** window, click **Probe Table Graph 3**.

2 In the **Settings** window for **Table Graph**, locate the **Legends** section.

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

4 In the table, enter the following settings:

Legends
`Matched load`

The plot should now look like that in Figure 2, with the red graph corresponding to the matched case.