Created in COMSOL Multiphysics 6.1

Signal Integrity and TDR Analysis of Adjacent Microstrip Lines

Introduction

The signal integrity (SI) analysis gives an overview of the quality of an electrical signal transmitted through electrical circuits such as high-speed interconnects, cables, and printed circuit boards. The quality of the received signal can be distorted by noise from outside the circuit, and can be degraded by impedance mismatch, insertion loss, and crosstalk; in practice, EMC/EMI analyses are run to estimate the susceptibility of a device or a network to an undesired coupling. In this example model, we examine the crosstalk effect between two adjacent microstrip lines on a microwave substrate. The simulated results provide the time-domain reflectometry (TDR) response at the coupled ports and show increased distortion of a signal at higher data rates.

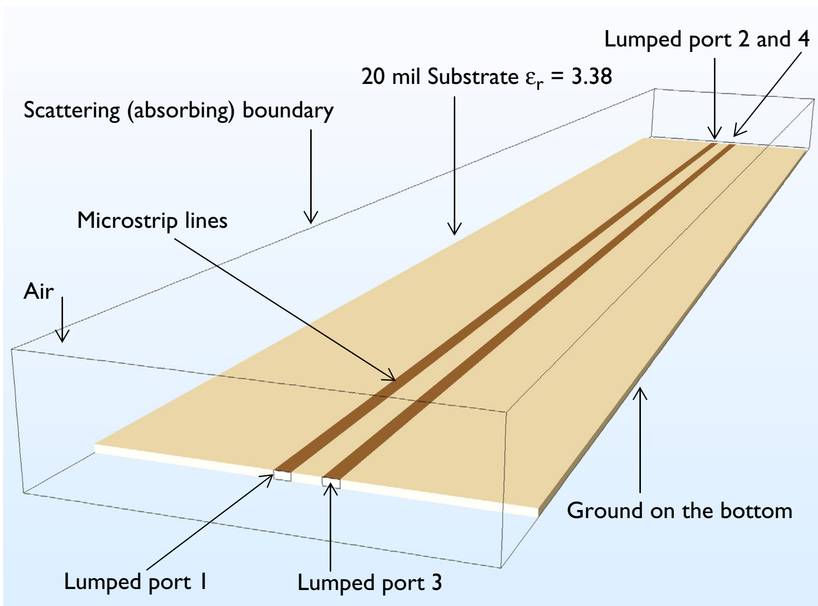


Figure 1: A microstrip line crosstalk model is composed of 20 mil microwave substrate with a ground plane and two adjacent microstrip lines 1.8 mm apart.

Model Definition

Two parallel 50 Ω microstrip lines are patterned on 20 mil substrate with a dielectric constant ε_r = 3.38. All metallic parts, including the patterned lines and bottom ground plane, are configured using perfect electric conductor (PEC) boundary conditions. The small rectangular surfaces, bridging between two parallel lines and the ground plane, are used to model lumped ports with which the microstrip lines are excited or terminated by

50 Ω. The air domain on top of the circuit board is defined using vacuum material properties. The exterior surfaces of the air are finished by a scattering boundary condition that is an absorbing boundary to describe an open space.

One bit of a single rectangular pulse is used to excite the circuit board. The widths of the two pulses are set to half of the 300 MHz and 600 MHz signals. The corresponding data rates for each frequency are 600 Mbit/s and 1.2 Gbit/s, respectively. A parametric sweep switches the frequency of the pulse during the simulation. It is necessary to apply smoothing to the transition zone of the pulse to remove undesirable high-frequency components from the signal.

The maximum simulation time is calculated using an approximated traveling time of a wave through a microstrip line based on the phase velocity. The effective dielectric constant for the phase velocity calculation is obtained using an equation in Ref. 1

$$\frac{\varepsilon_r + 1}{2} + \frac{\varepsilon_r - 1}{2\sqrt{1 + 12\frac{d}{W}}} \tag{1}$$

where d is the thickness of the substrate and W is the width of the line.

It is assumed that a frequency about ten times greater than the input pulse signal frequency is enough to describe the highest frequency component in the smoothed rectangular pulse. The maximum mesh element size is set to 0.2 wavelengths in the dielectric substrate.

It is also important to define a time step that resolves the wave well in time as the mesh does in space. Any longer time steps would not optimally utilize the fine mesh, and any shorter time steps would unnecessarily lead to a longer simulation time without gaining significant accurate results. While running a simulation, the time step is continuously adjusted to meet the specified tolerances by the time-dependent solver. If there is an exact time step the solver needs to take, it can be manually set. In the Settings window of the Time-Dependent Solver node, the time step can be specified manually. See the step by step instructions to learn how to access this setting.

Results and Discussion

Figure 2 shows the input pulse signal as well as the voltage at lumped port 1 with a data rate of 600 Mbit/s (300 MHz) and 1.2 Gbit/s (600 MHz), respectively. Since the input signal is flowing through a straight 50 Ω line terminated with a 50 Ω resistor without discontinuity on the line, no distortion is evident on the port voltage.

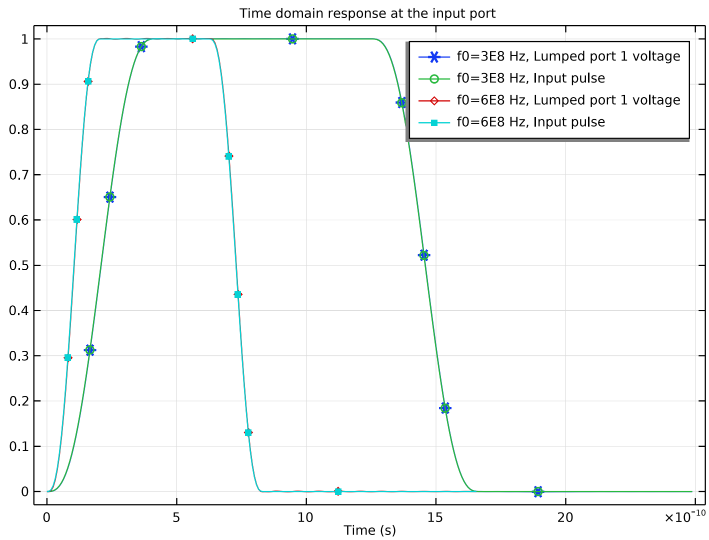


Figure 2: The input pulse and the voltage at lumped port 1 (the excitation port) with a data rate of 600 Mbit/s and 1.2 Gbit/s.

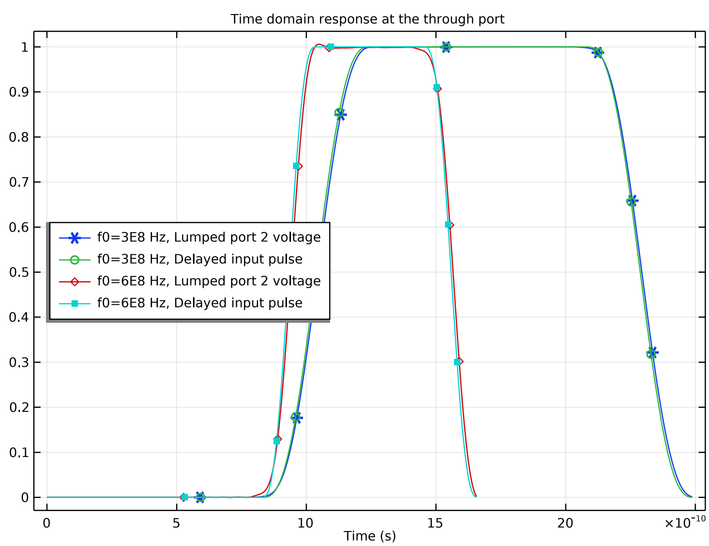


Figure 3: The delayed input pulse and the voltage at lumped port 2 (the through port) with a data rate of 600 Mbit/s and 1.2 Gbit/s.

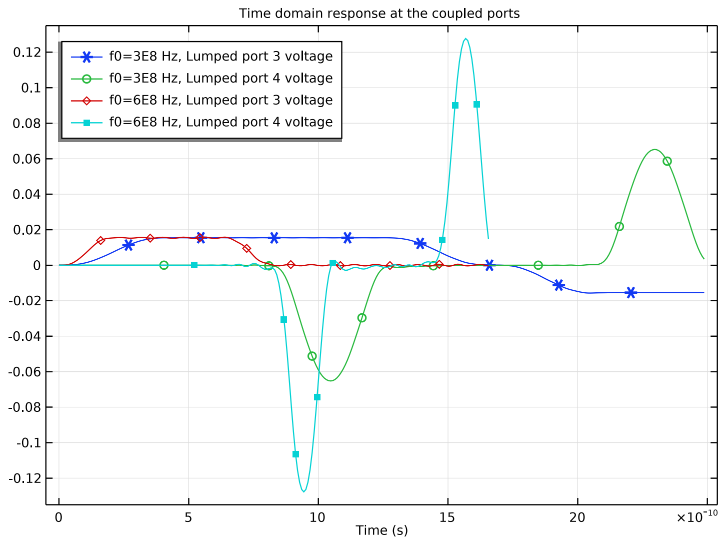


Figure 4: Voltage for the coupled signals at lumped ports 3 and 4. They are near-end crosstalk (NEXT) and far-end crosstalk (FEXT), respectively. The voltage of a coupled signal increases at a higher data rate.

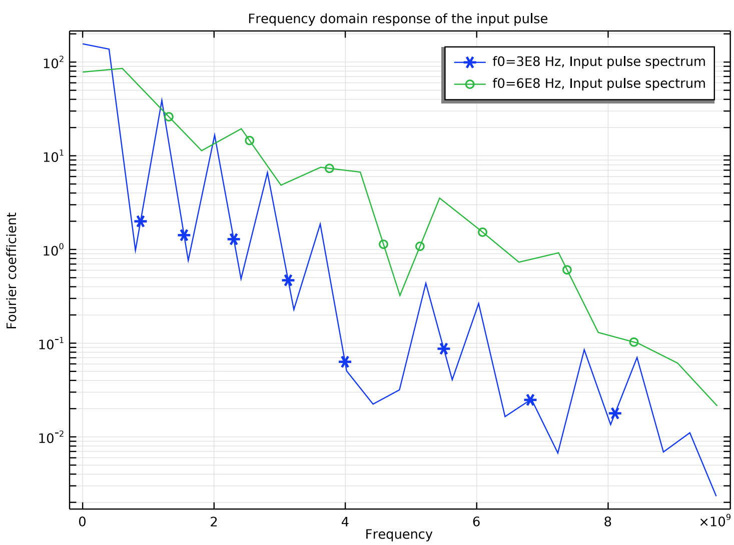


Figure 5: The spectrum of input pulses up to 10 GHz. The signal strength decreases as frequency increases.

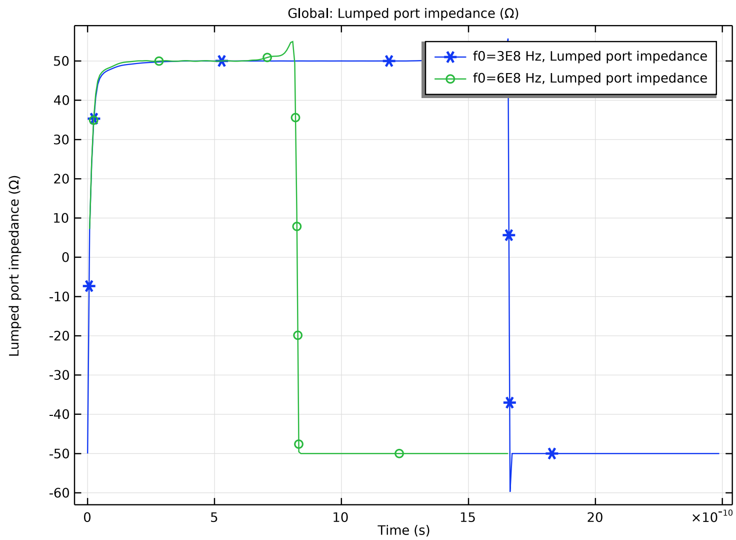


Figure 6: The impedance of lumped port 1 with data rates of 600 Mbit/s and 1.2 Gbit/s

Figure 3 shows the delayed input pulse and the received signals with two data rates at lumped port 2. The time domain response of the 1.2 Gbit/s signal is slightly distorted in the beginning when it reaches 1 V while that of the 600 Mbit/s signal seems to remain undistorted.

The crosstalk between two microstrip lines is observed in Figure 4. The coupled signal, near-end crosstalk (NEXT), level between two data rates is quite similar at lumped port 3, which is next to the excitation port. The time domain response at lumped port 4 next to the through port, far-end crosstalk (FEXT), shows that the higher data rate signal causes the stronger crosstalk on another signal path.

Figure 5 works as a reference to define the effective highest frequency component in the smoothed rectangular pulse since it provides the spectrum of results for 600 Mbit/s and 1.2 Gbit/s. A periodic rectangular pulse can be decomposed into a sum of sinusoidal functions. By estimating the level of a particular frequency, a proper frequency range can be defined for efficient simulations. The estimated highest frequency is used to choose the mesh size. With a finer mesh size, higher frequency components can be analyzed more accurately but it will increase the computation time. In this model, we set the maximum frequency component to 5 GHz that is two orders of magnitude smaller than the level of the DC component of each rectangular pulse.

In Figure 6, the TDR at lumped port 1 is presented in terms of impedance. The computed port impedance is around 50 Ω while the signal level is 1 V.

Notes About the COMSOL Implementation

Changing the number of output times in the **Step 1: Time Dependent** node configures the output times for the results analysis but has a minimal effect on the time steps taken by the solver.

Reference

1. D.M. Pozar, *Microwave Engineering*, John Wiley & Sons, 1998.

Application Library path: `RF_Module/EMI_EMC_Applications/microstrip_line_crosstalk`

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**.

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 Click **Load from File**.

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

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 **in**.

Block 1 (blk1)

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 `blength+0.5`.

4 In the **Depth** text field, type `bwidth`.

5 In the **Height** text field, type `tsub*15`.

6 Locate the **Position** section. In the **x** text field, type `-0.25`.

7 In the **y** text field, type `-bwidth/2`.

8 Click to expand the **Layers** section. Find the **Layer position** subsection. Select the **Left** check box.

9 Select the **Right** check box.

10 Clear the **Bottom** check box.

11 In the table, enter the following settings:

Layer name	Thickness (in)
Layer 1	0.25

12 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Block 2 (blk2)

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 `blength`.

4 In the **Depth** text field, type `bwidth`.

5 In the **Height** text field, type `tsub`.

6 Locate the **Position** section. In the **y** text field, type `-bwidth/2`.

Block 3 (blk3)

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 `blength`.

4 In the **Depth** text field, type `lwidth`.

5 In the **Height** text field, type `tsub`.

6 Locate the **Position** section. In the **y** text field, type `-spacing/2-lwidth`.

Mirror 1 (mir1)

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

2 Select the object **blk3** only.

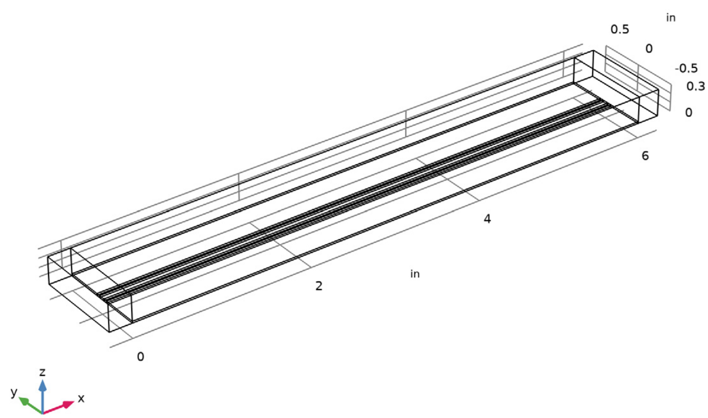
3 In the **Settings** window for **Mirror**, locate the **Input** section.

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

5 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type `1`.

6 In the **z** text field, type `0`.

7 Click **Build All Objects**.



GLOBAL DEFINITIONS

Rectangle 1 (rect1)

1 In the **Home** toolbar, click **Functions** and choose **Global>Rectangle**.

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

3 In the **Lower limit** text field, type `0`.

4 In the **Upper limit** text field, type `Tb-Tb/4`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `Tb/4`.

Analytic 1 (an1)

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

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

3 In the **Expression** text field, type `rect1((t-Tb/8)/1[s])`.

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

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

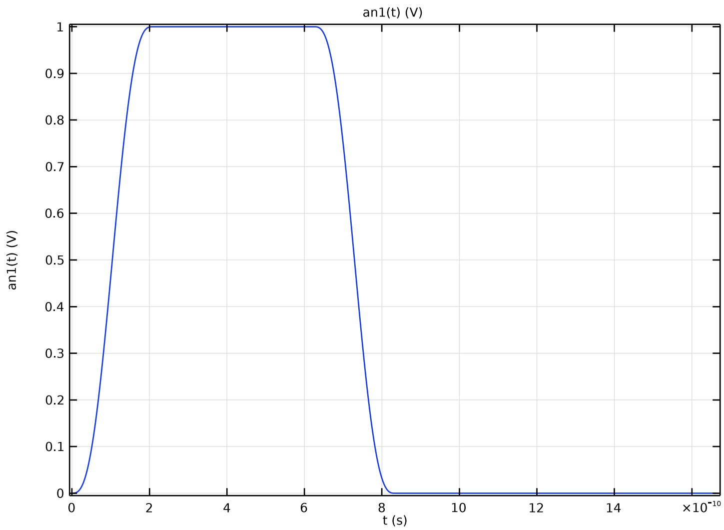
Argument	Unit
t	s

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

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

Argument	Lower limit	Upper limit	Unit
t	0	2*Tb	s

8 Click **Plot**.



MATERIALS

Material 1 (mat1)

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

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

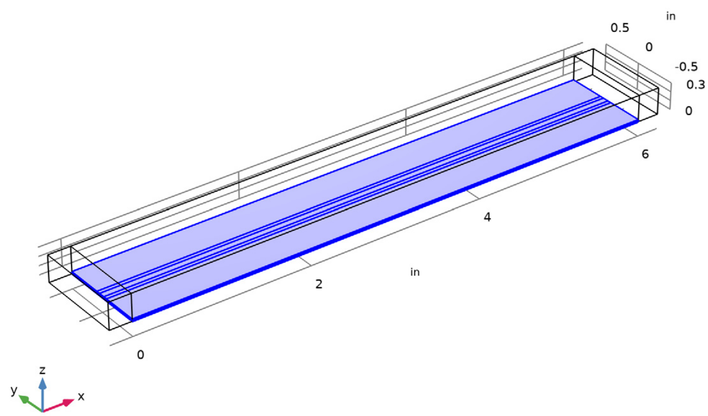
3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

Material 2 (mat2)

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

2 Select Domains 2 and 4–7 only.



3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

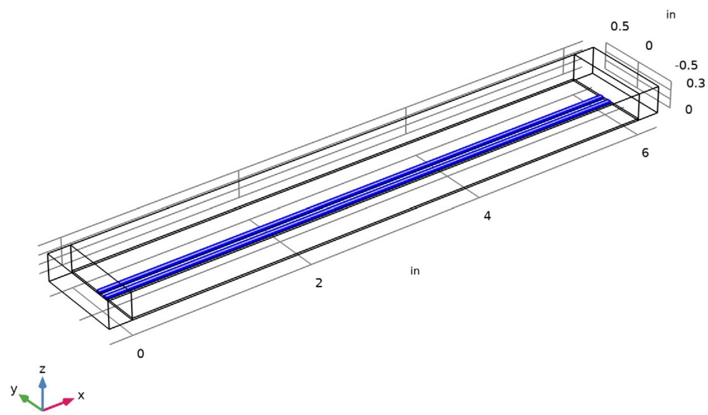
Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`er_sub`	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Perfect Electric Conductor 2

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

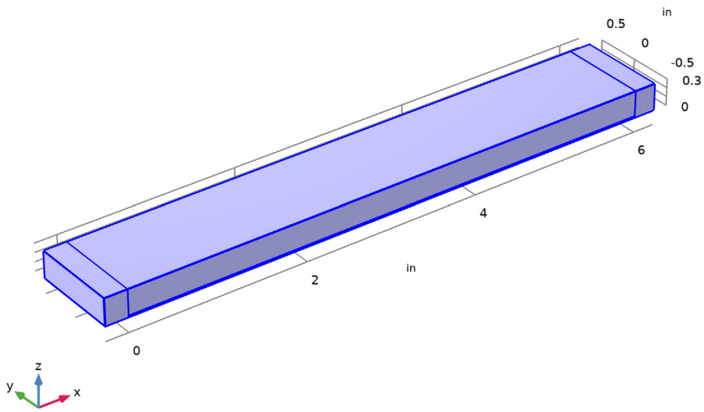
2 Select Boundaries 16 and 24 only.



Scattering Boundary Condition 1

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

2 Select Boundaries 1, 2, 4, 5, 7, 10, 12, 29, 30, 32, 35, 40, and 41 only.



Lumped Port 1

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

2 Select Boundary 21 only.

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

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

4 In the V_0 text field, type `an1(t)`.

Lumped Port 2

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

2 Select Boundary 38 only.

Lumped Port 3

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

2 Select Boundary 13 only.

Lumped Port 4

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

2 Select Boundary 36 only.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size

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

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

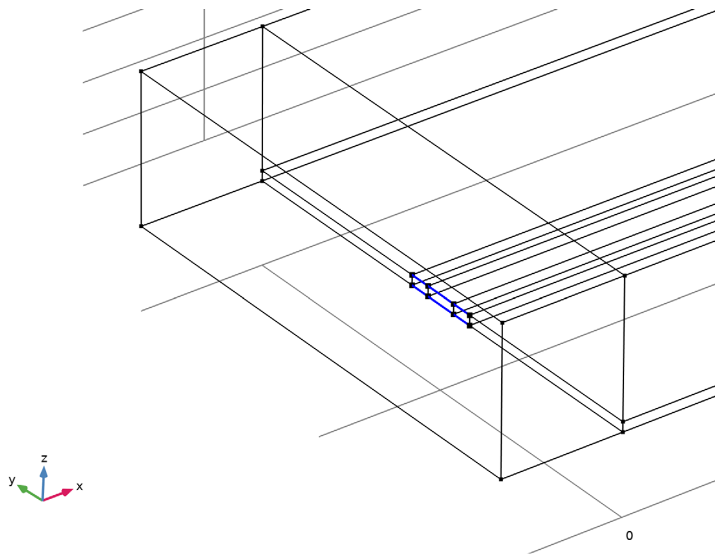
3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `hm`.

Edge 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

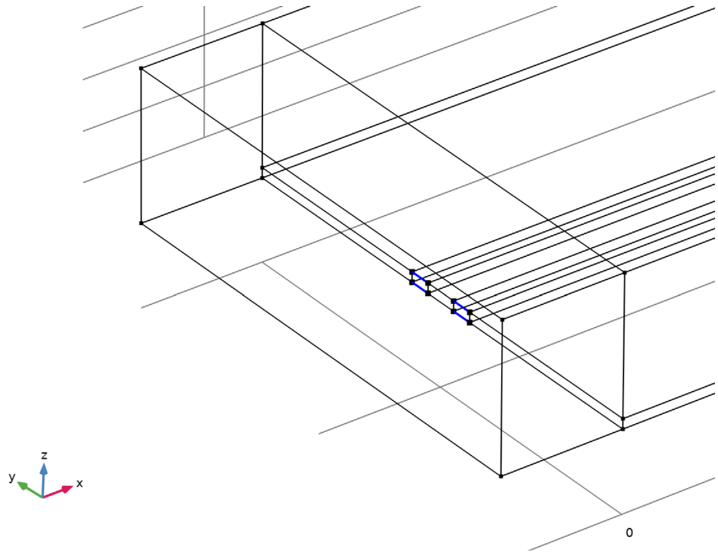
2 Select Edges 18, 20, 23, 25, 28, and 30 only.



Distribution 1

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

2 Select Edges 18, 20, 28, and 30 only.



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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 3.

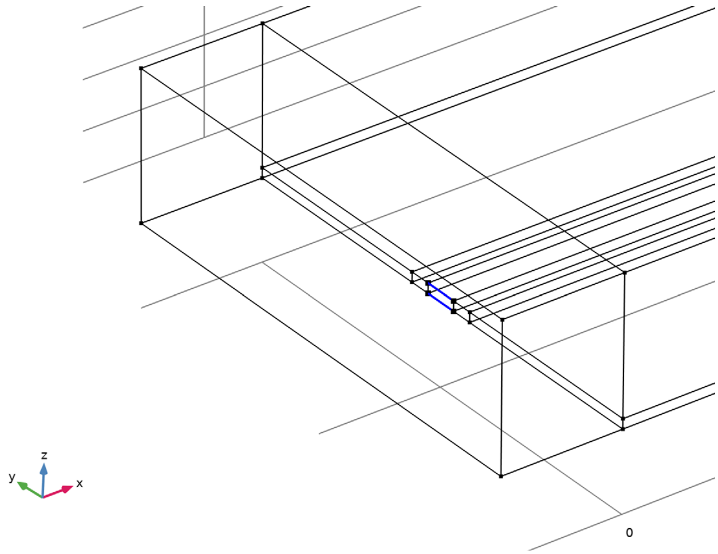
6 In the **Element ratio** text field, type 4.

7 Select the **Symmetric distribution** check box.

Distribution 2

1 In the **Model Builder** window, right-click **Edge 1** and choose **Distribution**.

2 Select Edges 23 and 25 only.



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

4 From the **Distribution type** list, choose **Predefined**.

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

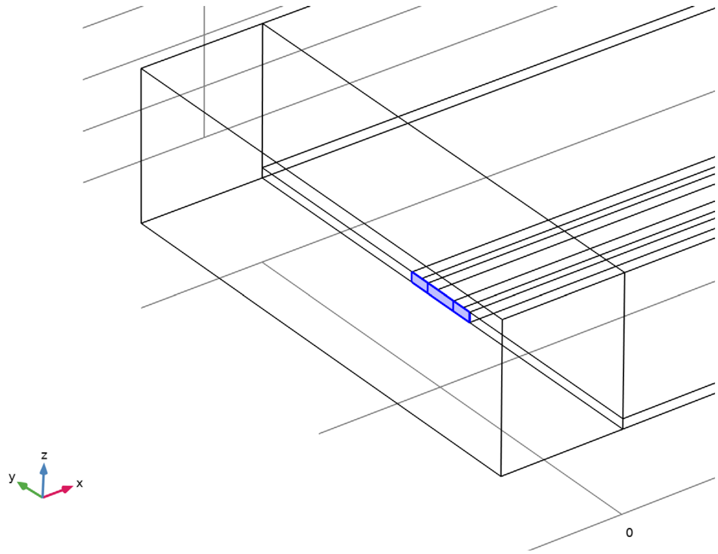
6 In the **Element ratio** text field, type 4.

7 Select the **Symmetric distribution** check box.

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

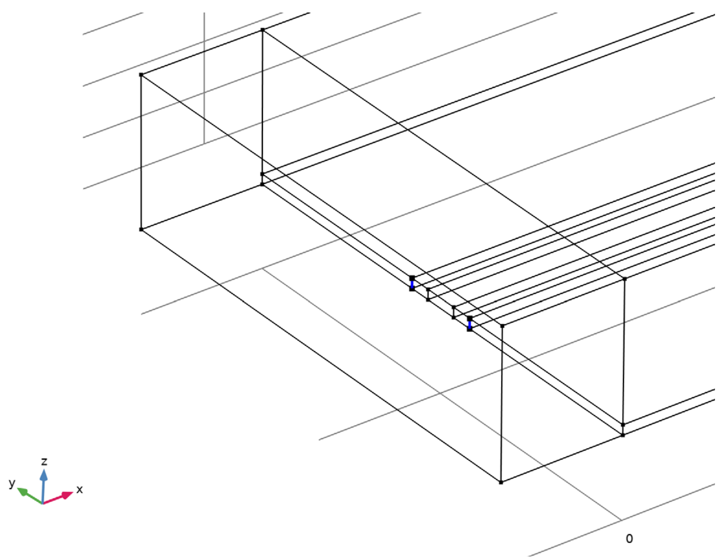
2 Select Boundaries 13, 17, and 21 only.



Distribution 1

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

2 Select Edges 17 and 32 only.



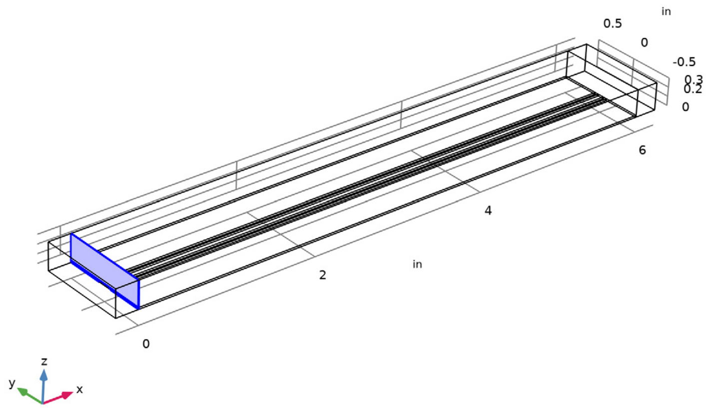
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 2.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundaries 6, 9, and 25 only.



Swept 1

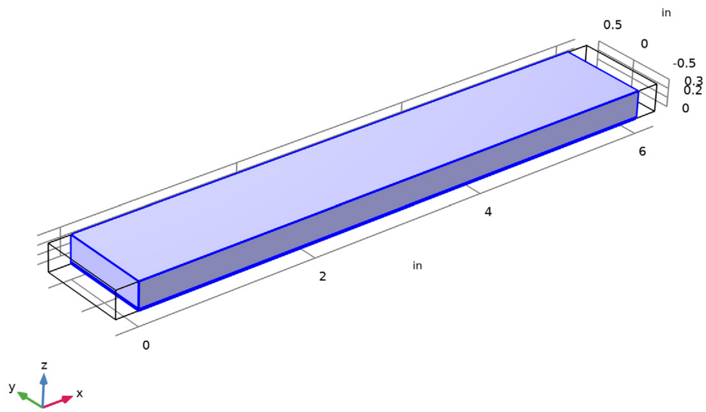
1 In the **Mesh** toolbar, click **Swept**.

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

3 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

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

5 Select Domains 2–7 only.



Swept 2

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

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Delete**.

2 Right-click **Mesh 1** and choose **Build All**.

3 Click the **Click and Hide** button in the **Graphics** toolbar.

4 In the **Graphics** window toolbar, click ▼ next to **Select Edges**, then choose **Select Boundaries**.

5 Select Boundary 12 only.

6 Select Boundary 10 only.

7 Select Boundary 4 only.

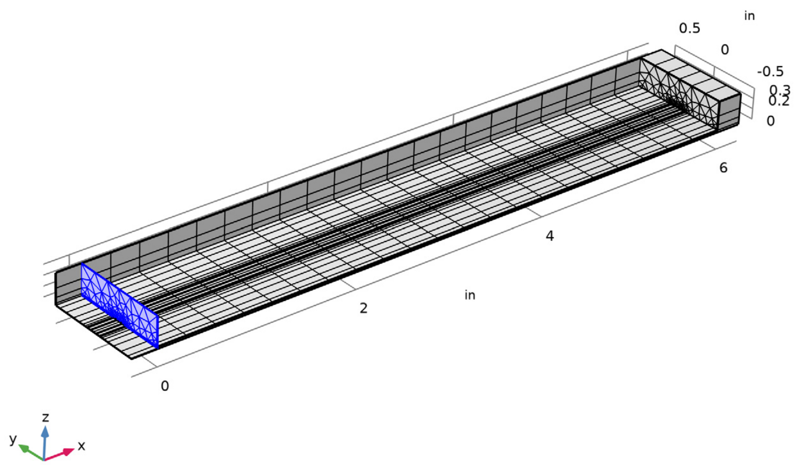
8 Select Boundary 1 only.

9 Select Boundary 2 only.

10 Click the **Click and Hide** button in the **Graphics** toolbar.

Free Triangular 1

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



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,sim_time_step,sim_time_max)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `sim_time_step`.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Direct**.

7 In the **Settings** window for **Direct**, locate the **General** section.

8 From the **Solver** list, choose **PARDISO**.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Frequency)	`300[MHz] 600[MHz]`	Hz

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

RESULTS

3D Plot Group 1

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Time (s)** list, choose **6E-10**.

Multislice 1

1 In the **Model Builder** window, expand the **3D Plot Group 1** node, then click **Multislice 1**.

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

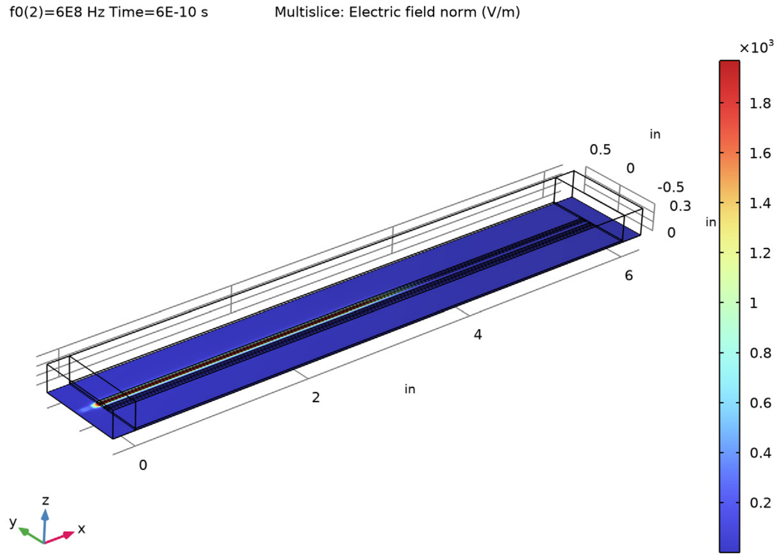
3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.

5 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `0`.

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



1D Plot Group 2

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

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

5 In the **Title** text area, type `Time domain response at the input port`.

Global 1

1 Right-click **1D Plot Group 2** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`temw.Vport_1`	V	`Lumped port 1 voltage`
`an1(t)`		`Input pulse`

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 From the **Positioning** list, choose **Interpolated**.

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

Figure 2 shows the input pulse and the voltage at lumped port 1.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 2** and choose **Duplicate**.

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

3 In the **Title** text area, type `Time domain response at the through port`.

4 Locate the **Legend** section. From the **Position** list, choose **Middle left**.

Global 1

1 In the **Model Builder** window, expand the **1D Plot Group 3** node, then click **Global 1**.

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

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`temw.Vport_2`	V	`Lumped port 2 voltage`
`an1(t-delay)`	V	`Delayed input pulse`

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

Figure 3 shows the delayed input pulse and the voltage at lumped port 2.

1D Plot Group 4

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Time domain response at the coupled ports`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **1D Plot Group 4** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
temw.Vport_3	V	Lumped port 3 voltage
temw.Vport_4	V	Lumped port 4 voltage

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

5 From the **Positioning** list, choose **Interpolated**.

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

The coupled signals at lumped port are shown in Figure 4.

1D Plot Group 5

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Frequency domain response of the input pulse`.

Global 1

1 Right-click **1D Plot Group 5** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
an1(t)	V	Input pulse spectrum

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Discrete Fourier transform**.

5 From the **Show** list, choose **Frequency spectrum**.

6 Select the **Frequency range** check box.

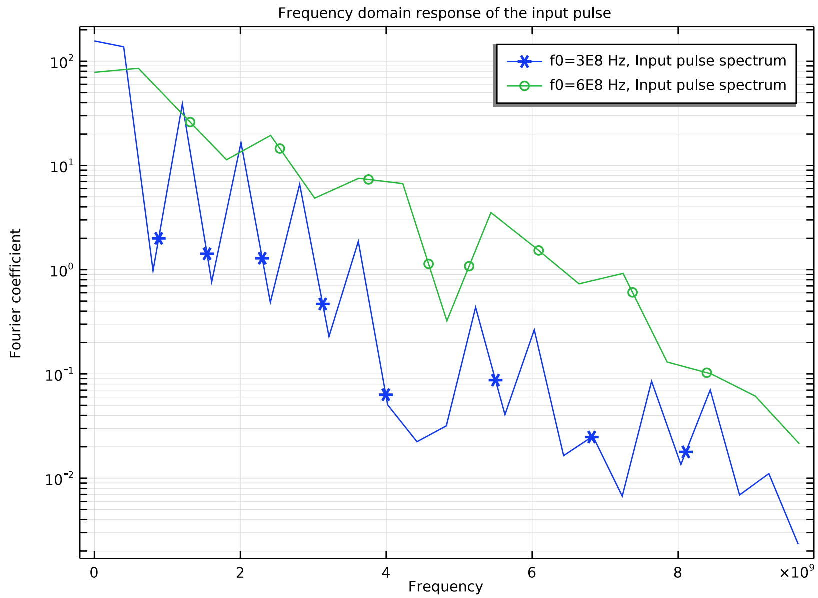
7 In the **Maximum** text field, type `10[GHz]`.

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

9 From the **Positioning** list, choose **Interpolated**.

10 In the **1D Plot Group 5** toolbar, click **Plot**.

11 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.



Compare to the spectra of input pulses in Figure 5.

1D Plot Group 6

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

Global 1

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

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Transient>Ports>temw.Zport_1 - Lumped port impedance - Ω**.

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

4 From the **Positioning** list, choose **Interpolated**.

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

Figure 6 describes the impedance of lumped port 1 with two data rates as a function of time.