Created in COMSOL Multiphysics 6.1

Modeling of a Differential Microstrip Line

Introduction

A differential line is composed of two transmission lines excited by two out-of-phase signals. This configuration is known to be useful to enhance the signal-to-noise ratio. The model example shows how to set up differential and single-ended microstrip lines using TEM-type ports. The impact on computed S-parameters due to a fictitious noise source is demonstrated for each microstrip line configuration, respectively.

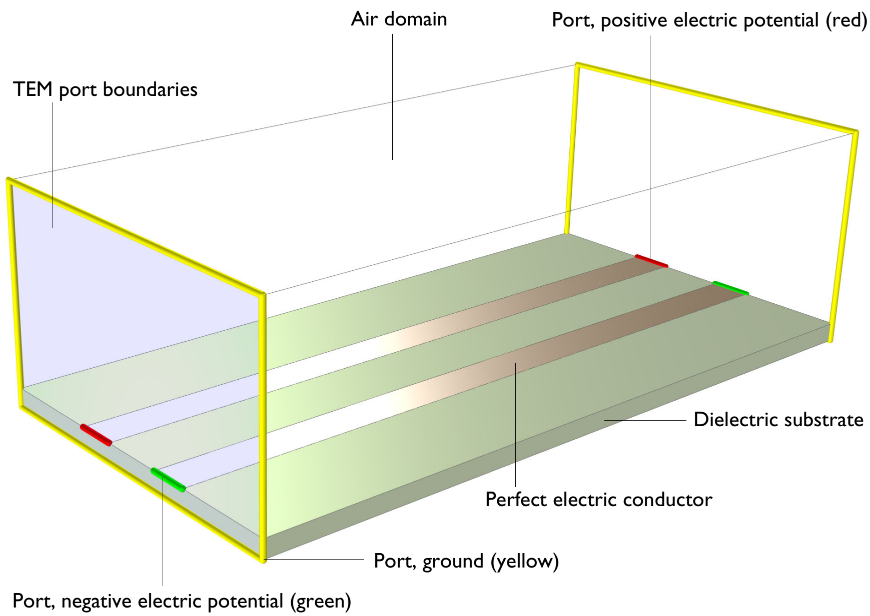


Figure 1: A differential line composed of two microstrip lines: the TEM type of ports is used to set up differential line properties.

Model Definition

The differential line model consists of two single microstrip lines printed on a substrate with a permittivity value of 3.38. The differential line is excited and terminated by port features. The port types are set to transverse electromagnetic (TEM) that is configured with electric potential and ground subfeatures. To generate the differential mode, the positive electric potential is assigned on one of the single-ended microstrip line edges while the negative electric potential is added on the end of the adjacent microstrip line. If the coupling between two lines is marginal, the characteristic port impedance of a differential line can be approximated as a series connection of two 50 Ω lumped devices, that is 100 Ω when the characteristic impedance of a single-ended microstrip line is 50 Ω. Note that two positive potential subfeatures are required to model a common mode and the characteristic impedance of the port with negligible coupling effects can be approximated as a parallel connection of two 50 Ω lumped devices, that is 25 Ω. Perfect electric conductor (PEC) is used to address all metallic parts. By default, all exterior boundaries are set to PEC. So, the model can be seen as enclosed by a metallic box.

The physics settings include all possible modeling cases such as single-ended line, differential line, and a fictitious noise. In each study step, these settings can selectively be enabled or disabled before running computation.

Results and Discussion

Figure 2 describes the electric field norm on the circuit board for a single-ended microstrip line. The field distribution on the top surface shows that the excited signal propagates to the other side and is terminated by the same type of port. Strong fields are confined around the edges of the PEC boundaries.

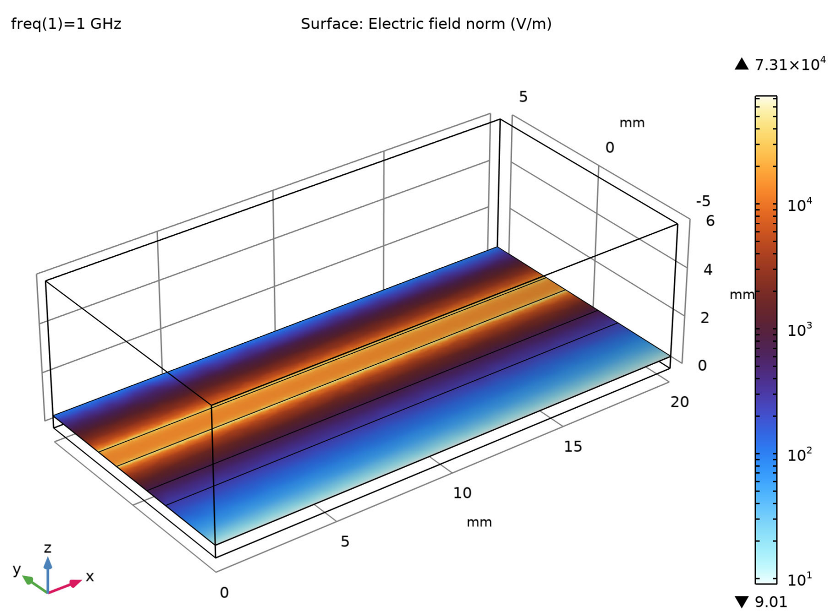


Figure 2: The electric field norm on the top surface of the circuit board when a single-ended microstrip line is computed.

The performance of the single-ended line is checked in terms of the S-parameters through the default global evaluation. The input impedance matching (S33dB) is below –30 dB and insertion loss (S43dB) is better than 0.05 dB. For the single-ended microstrip line, the port mode impedance (Zmode_3), calculated characteristic impedance, is about 49.5 Ω.

When the single-ended microstrip line is under the influence of undesirable noise, the signal integrity can be degraded. The impact on the circuit board due to a fictitious noise made by a point source is visualized in Figure 3. By reviewing the computed S-parameters, it is more obvious quantitatively. With the noise source, the input impedance matching (S33dB) is higher than –15 dB and insertion loss (S43dB) is worse than 2 dB.

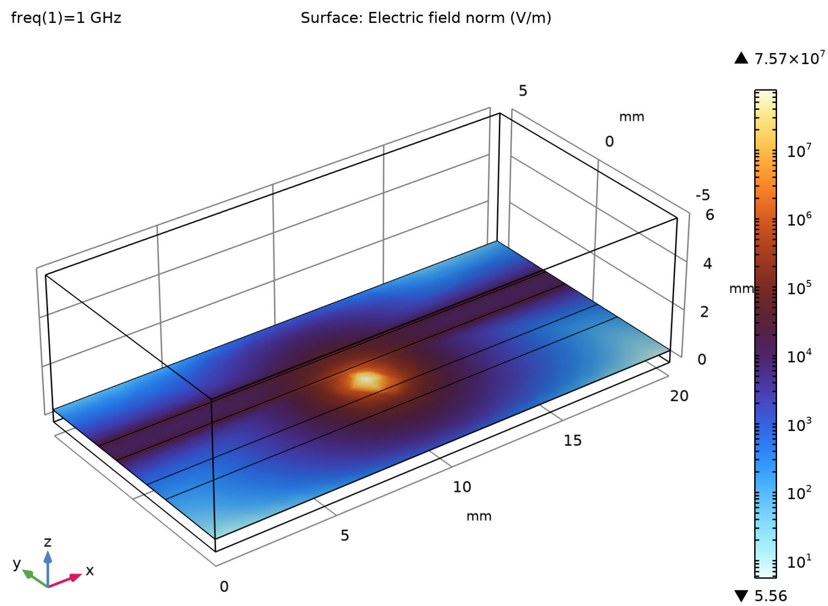


Figure 3: The electric field norm on the top surface of the circuit board when a fictitious noise is added next to a single-ended microstrip line.

The electric field norm of a differential line composed of two microstrip lines is plotted in Figure 4. When visualizing the z-component of the electric field, the polarity of differential signals can be displayed with a couple of colors (Figure 5). Since there is coupling between two single-ended lines, the computed port mode impedance (Zmode_1) is around 97 Ω deviated from the value of 100 Ω for the case of no coupling. The input impedance matching (S11dB) is below –30 dB and insertion loss (S21dB) is better than 0.05 dB.

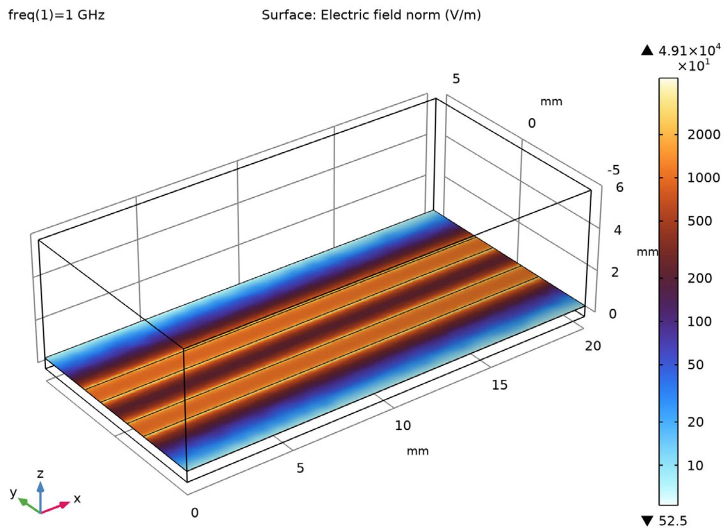


Figure 4: The electric field norm on the top surface of the circuit board when a differential microstrip line is computed.

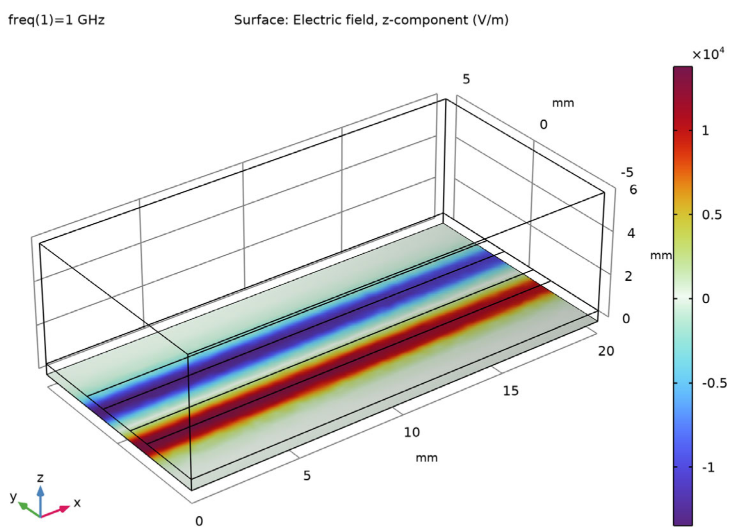


Figure 5: The z-component of the electric field on the bottom surface of the circuit board when a differential microstrip line is computed.

After reactivating the noise source in the differential line, its effect on the circuit board is presented in Figure 6. The input impedance matching (`S11dB`) is below –30 dB and insertion loss (`S21dB`) is better than 0.05 dB. So, those characteristics are not degraded as shown in the single-ended line case when the noise equally affects each line component of the differential pair.

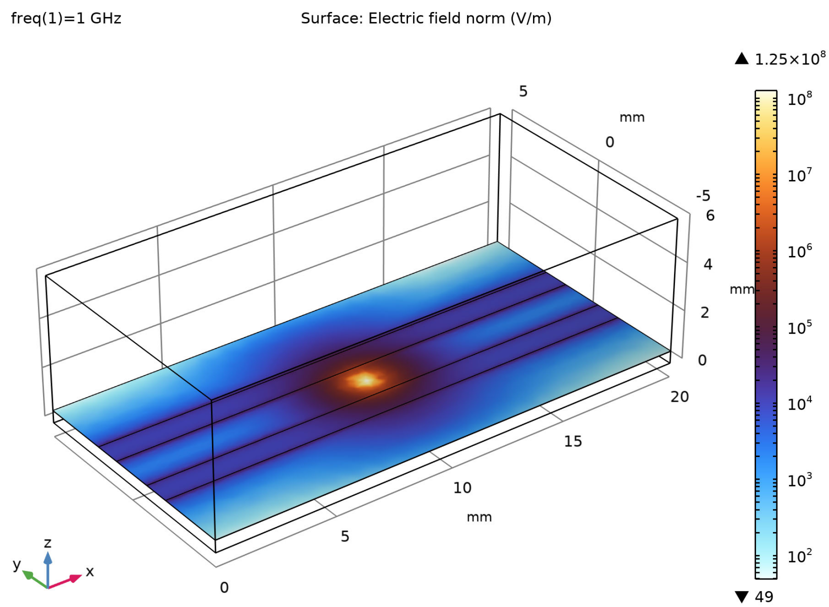


Figure 6: The electric field norm on the top surface of the circuit board when a fictitious noise is added.

Application Library path: `RF_Module/Transmission_Lines_and_Waveguides/microstrip_line_tem_differential`

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, Frequency Domain (emw)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> TEM Boundary Mode Analysis**.
6 Click **Done**.

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 **mm**.

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 `20`.
4 In the **Depth** text field, type `10`.
5 In the **Height** text field, type `6`.
6 Locate the **Position** section. In the **y** text field, type `-5`.
7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`20[mil]`

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 `20[mil]`.

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `20`.

4 In the **Height** text field, type `1.13`.

5 Locate the **Position** section. In the **yw** text field, type `-0.565+1.5`.

Work Plane 1 (wp1)>Mirror 1 (mir1)

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

2 Select the object **r1** 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 Line of Reflection** section. In the **xw** text field, type `0`.

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

Point 1 (pt1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **More Primitives>Point**.

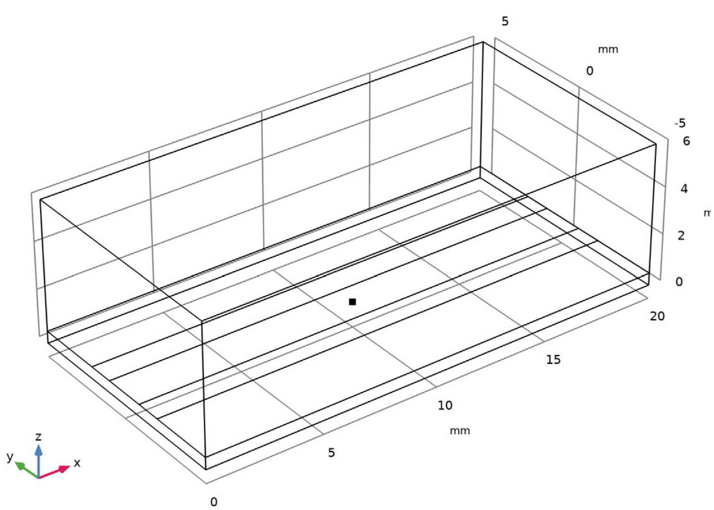
2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `10`.

4 In the **z** text field, type `20[mil]`.

5 Click **Build All Objects**.

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



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.

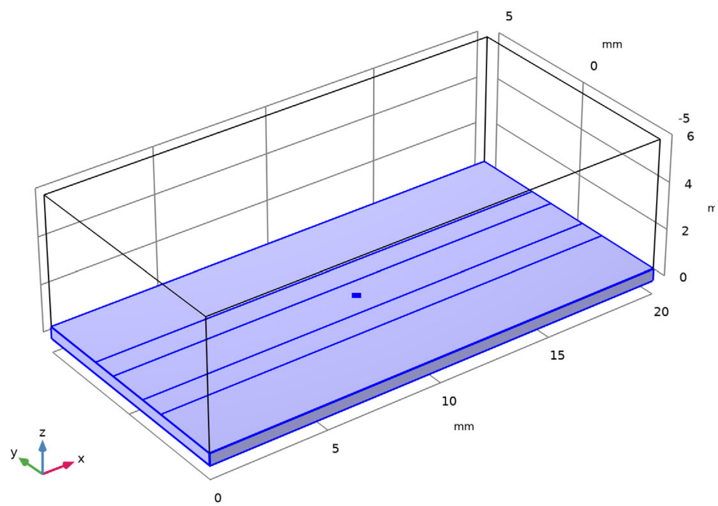
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material 2 (mat2)

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

2 Select Domain 1 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	3.38	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, FREQUENCY DOMAIN (EMW)

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, select **Physics>Advanced Physics Options** in the tree.

3 In the tree, select the check box for the node **Physics>Advanced Physics Options**.

4 Click **OK**.

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

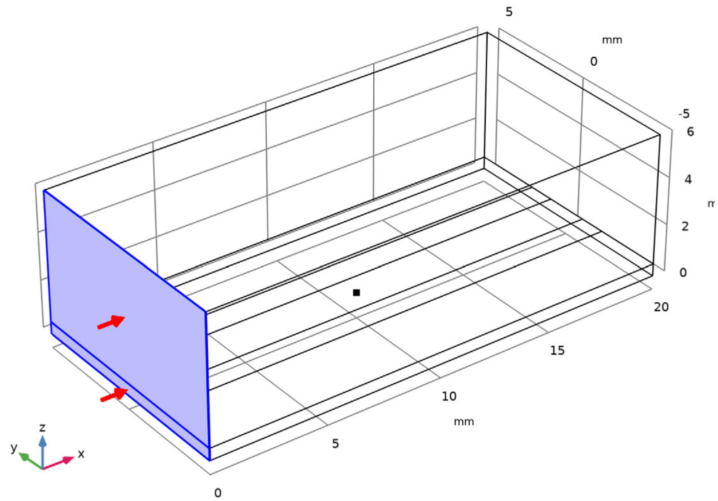
6 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, click to expand the **Port Options** section.

7 From the **Port formulation** list, choose **Constraint-based**.

Port 1

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

2 Select Boundaries 1 and 4 only.



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

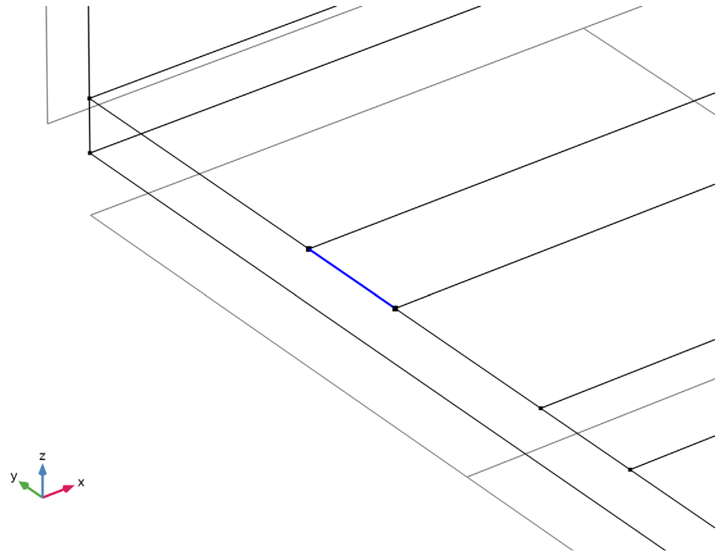
4 From the **Type of port** list, choose **Transverse electromagnetic (TEM)**.

5 In the Z_{ref} text field, type `100[ohm]`.

Electric Potential 1

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Potential**.

2 Select Edge 13 only.



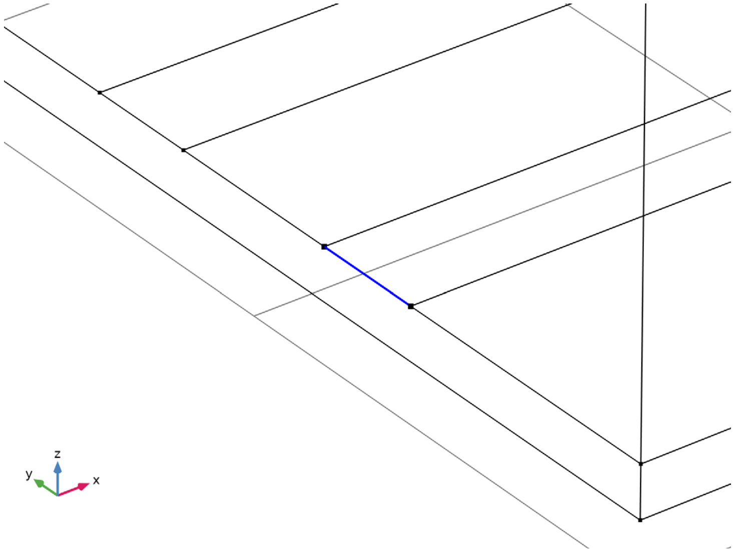
Port 1

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

Electric Potential 2

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Potential**.

2 Select Edge 9 only.



3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

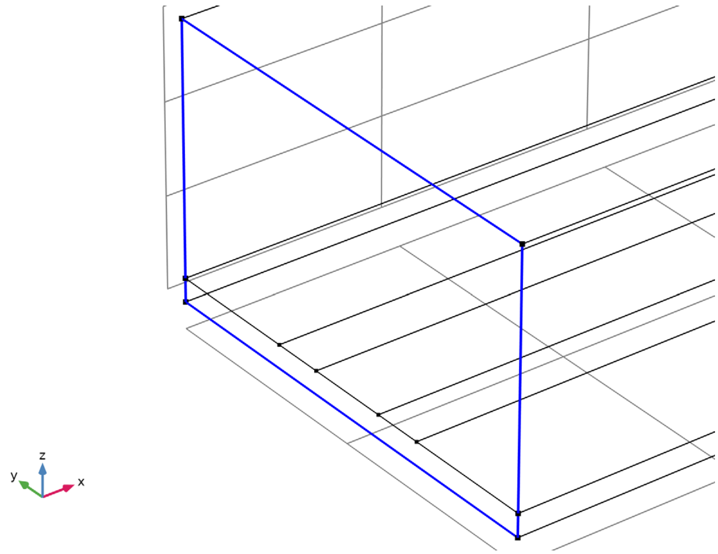
4 From the V_0 list, choose **Negative electric potential**.

Port 1

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

Ground 1

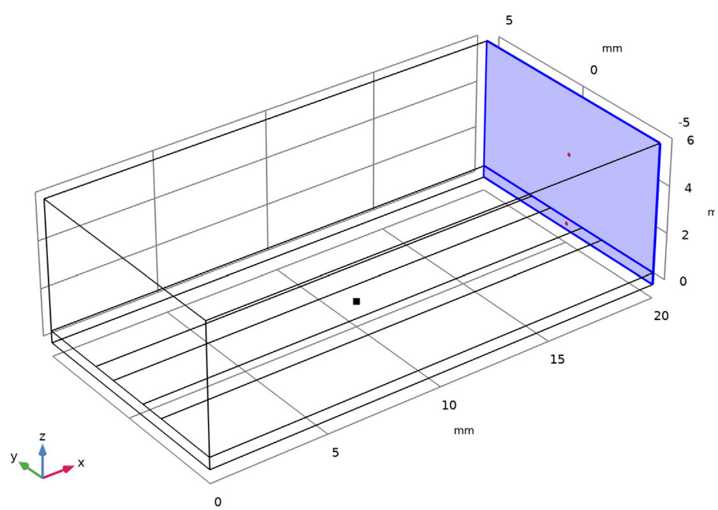
In the **Physics** toolbar, click **Attributes** and choose **Ground**.



Port 2

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

2 Select Boundaries 14 and 15 only.



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

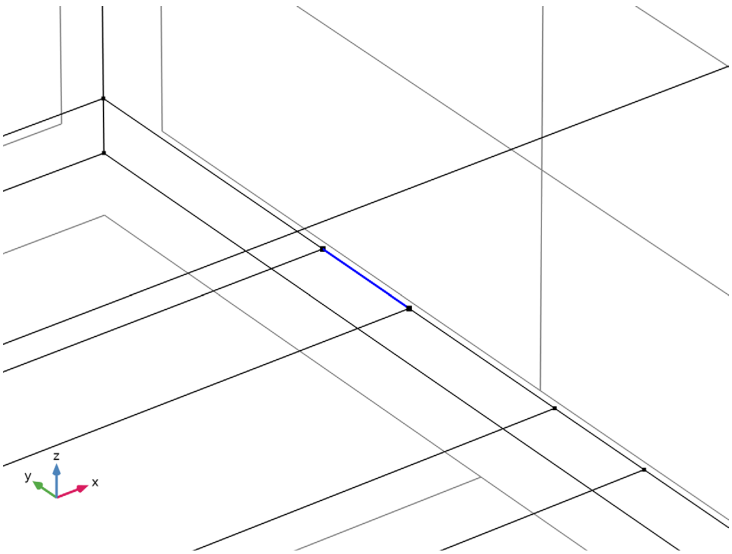
4 From the **Type of port** list, choose **Transverse electromagnetic (TEM)**.

5 In the Z_{ref} text field, type `100[ohm]`.

Electric Potential 1

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Potential**.

2 Select Edge 29 only.



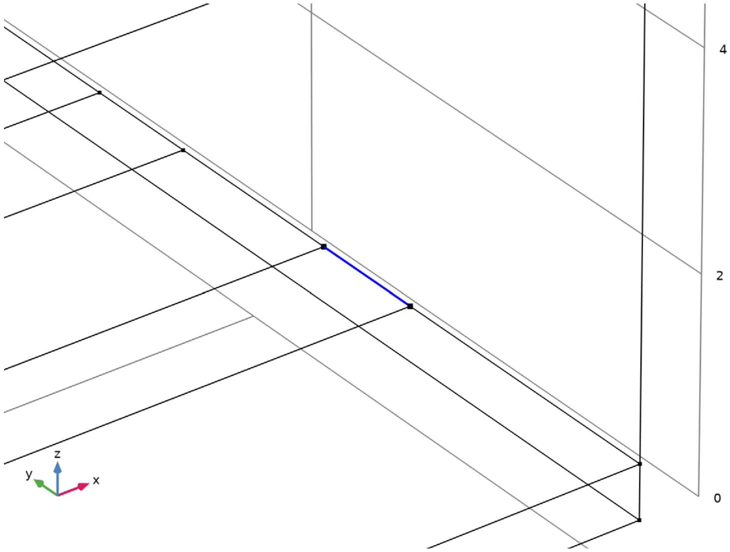
Port 2

In the **Model Builder** window, click **Port 2**.

Electric Potential 2

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Potential**.

2 Select Edge 27 only.



3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

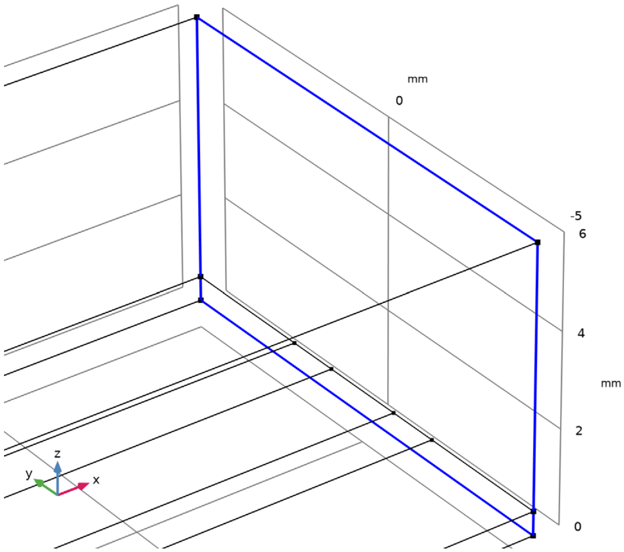
4 From the V_0 list, choose **Negative electric potential**.

Port 2

In the **Model Builder** window, click **Port 2**.

Ground 1

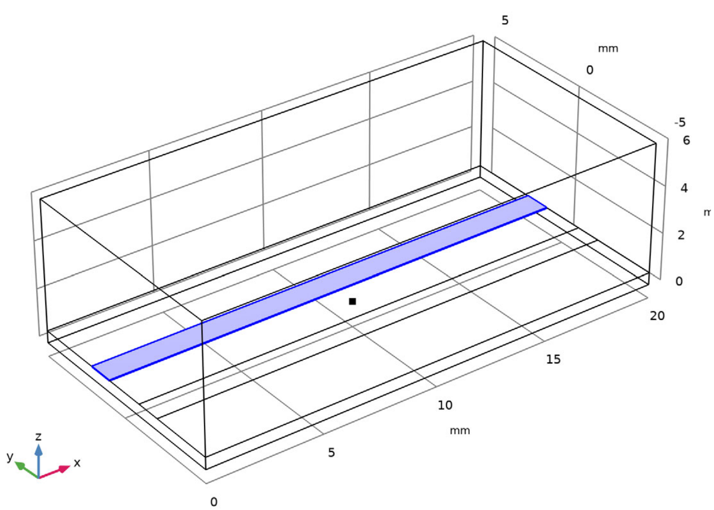
In the **Physics** toolbar, click **Attributes** and choose **Ground**.



Perfect Electric Conductor 2

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

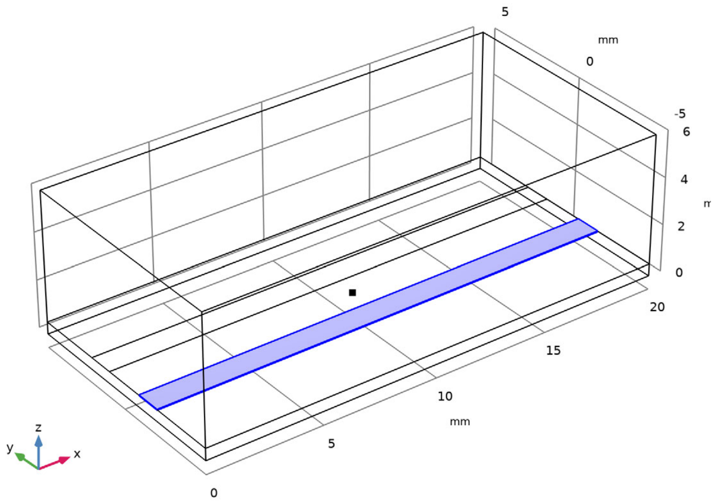
2 Select Boundary 10 only.



Perfect Electric Conductor 3

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

2 Select Boundary 8 only.



Electric Point Dipole 1

1 In the **Physics** toolbar, click **Points** and choose **Electric Point Dipole**.

2 Select Point 11 only.

3 In the **Settings** window for **Electric Point Dipole**, locate the **Dipole Parameters** section.

4 Specify the $\mathbf{n}_p$ vector as

0	x
0	y
-1	z

5 In the p text field, type `5e-4`.

Port 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** right-click **Port 1** and choose **Duplicate**.

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

3 In the Z_{ref} text field, type `50[ohm]`.

Electric Potential 2

1 In the **Model Builder** window, expand the **Port 3** node.

2 Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Port 3>Electric Potential 2** and choose **Delete**.

Port 4

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** right-click **Port 2** and choose **Duplicate**.

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

3 In the Z_{ref} text field, type `50[ohm]`.

Electric Potential 2

1 In the **Model Builder** window, expand the **Port 4** node.

2 Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Port 4>Electric Potential 2** and choose **Delete**.

Port 1, Port 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)**, Ctrl-click to select **Port 1** and **Port 2**.

2 Right-click and choose **Group**.

Differential Line Port Group

In the **Settings** window for **Group**, type `Differential Line Port Group` in the **Label** text field.

Port 3, Port 4

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)**, Ctrl-click to select **Port 3** and **Port 4**.

2 Right-click and choose **Group**.

Single-Ended Line Port Group

In the **Settings** window for **Group**, type `Single-Ended Line Port Group` in the **Label** text field.

DEFINITIONS

Hide for Physics 1

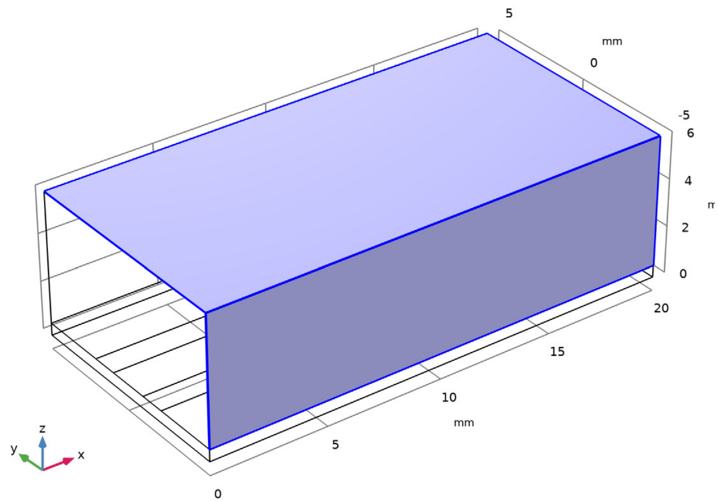
1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **View 1** and choose **Hide for Physics**.

3 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

5 Select Boundaries 5 and 7 only.



MESH 1

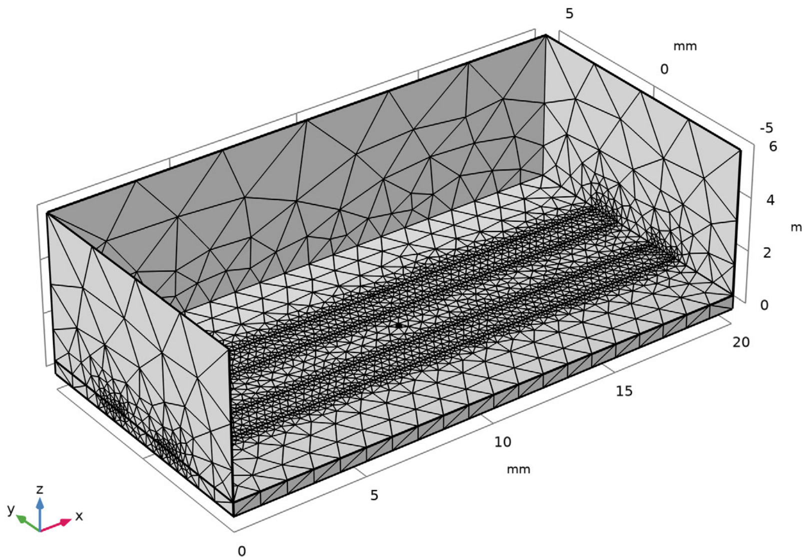
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Electromagnetic Waves, Frequency Domain (emw)** section.

3 Select the **Refine conductive edges** check box.

4 From the **Size type** list, choose **User defined**.

5 Click **Build All**.



STUDY 1

Step 1: TEM Boundary Mode Analysis

1 In the **Model Builder** window, under **Study 1** click **Step 1: TEM Boundary Mode Analysis**.

2 In the **Settings** window for **TEM Boundary Mode Analysis**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Differential Line Port Group**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Perfect Electric Conductor 3**.

7 Click **Disable**.

8 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Electric Point Dipole 1**.

9 Click **Disable**.

Step 2: Frequency Domain

1 In the **Model Builder** window, click **Step 2: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Differential Line Port Group**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Perfect Electric Conductor 3**.

7 Click **Disable**.

8 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Electric Point Dipole 1**.

9 Click **Disable**.

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

11 In the **Settings** window for **Study**, type `Study 1, Single-Ended Line` in the **Label** text field.

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

RESULTS

S-parameter, Single-Ended Line

1 In the **Model Builder** window, expand the **Results>Derived Values** node, then click **S-parameter (emw)**.

2 In the **Settings** window for **Global Evaluation**, type `S-parameter, Single-Ended Line` in the **Label** text field.

Port Mode Impedance, Single-Ended Line

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Port Mode Impedance, Single-Ended Line` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain> TEM boundary mode analysis>emw.Zmode_3 - TEM mode port characteristic impedance - Ω**.

4 Click **Evaluate**.

Electric Field Norm, Single-Ended Line

1. In the **Model Builder** window, under **Results** click **Electric Field (emw)**.
2. In the **Settings** window for **3D Plot Group**, type `Electric Field Norm, Single-Ended Line` in the **Label** text field.

Multislice

1. In the **Model Builder** window, expand the **Electric Field Norm, Single-Ended Line** node.
2. Right-click **Results>Electric Field Norm, Single-Ended Line>Multislice** and choose **Delete**.

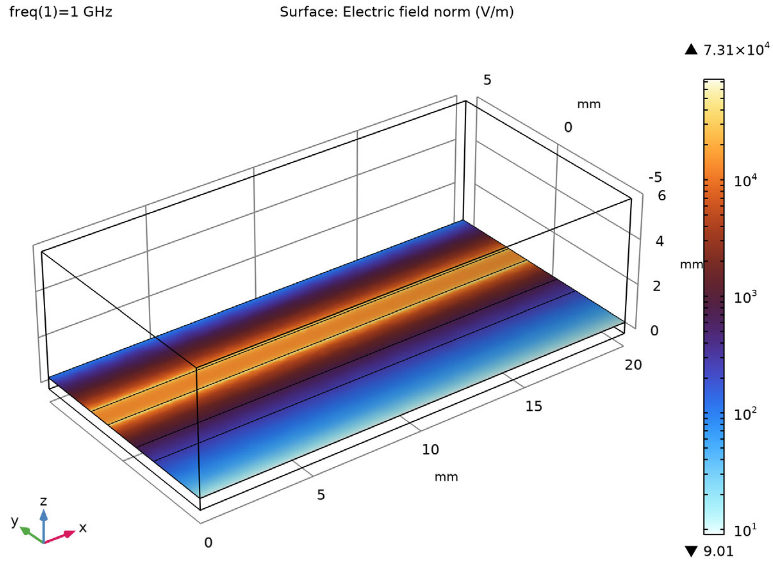
Surface 1

1. In the **Model Builder** window, right-click **Electric Field Norm, Single-Ended Line** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3. Click **Change Color Table**.
4. In the **Color Table** dialog box, select **Thermal>ThermalWave** in the tree.
5. Click **OK**.
6. In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
7. From the **Scale** list, choose **Logarithmic**.

Selection 1

1. Right-click **Surface 1** and choose **Selection**.
2. Select Boundaries 6 and 8–11 only.

3 In the **Electric Field Norm, Single-Ended Line** toolbar, click **Plot**.



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>TEM Boundary Mode Analysis**.

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

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

STUDY 2

Step 1: TEM Boundary Mode Analysis

1 In the **Settings** window for **TEM Boundary Mode Analysis**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Differential Line Port Group**.

4 Right-click and choose **Disable**.

5 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Perfect Electric Conductor 3**.

6 Click **Disable**.

Step 2: Frequency Domain

1 In the **Model Builder** window, click **Step 2: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Differential Line Port Group**.

5 Right-click and choose **Disable**.

6 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Perfect Electric Conductor 3**.

7 Click **Disable**.

8 In the **Model Builder** window, click **Study 2**.

9 In the **Settings** window for **Study**, type `Study 2, Single-Ended Line with Noise` in the **Label** text field.

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

RESULTS

S-parameter, Single-Ended Line with Noise

1 In the **Model Builder** window, under **Results>Derived Values** click **S-parameter (emw)**.

2 In the **Settings** window for **Global Evaluation**, type `S-parameter, Single-Ended Line with Noise` in the **Label** text field.

Electric Field Norm, Single-Ended Line with Noise

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Field Norm, Single-Ended Line with Noise` in the **Label** text field.

Multislice

1 In the **Model Builder** window, expand the **Electric Field Norm, Single-Ended Line with Noise** node.

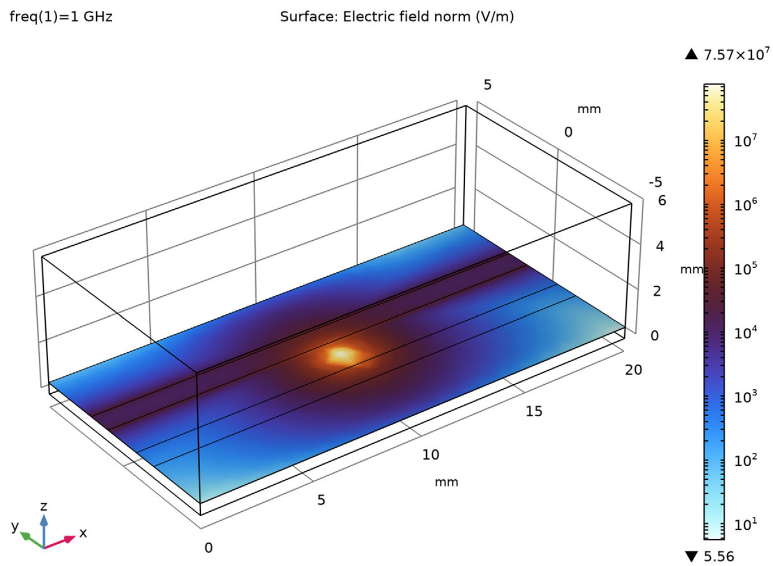
2 Right-click **Results>Electric Field Norm, Single-Ended Line with Noise>Multislice** and choose **Delete**.

Surface 1

In the **Model Builder** window, under **Results>Electric Field Norm, Single-Ended Line** right-click **Surface 1** and choose **Copy**.

Surface 1

In the **Model Builder** window, right-click **Electric Field Norm, Single-Ended Line with Noise** and choose **Paste Surface**.



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>TEM Boundary Mode Analysis**.

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

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

STUDY 3

Step 1: TEM Boundary Mode Analysis

1 In the **Settings** window for **TEM Boundary Mode Analysis**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Electric Point Dipole 1**.

4 Right-click and choose **Disable**.

5 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Single-Ended Line Port Group**.

6 Right-click and choose **Disable**.

Step 2: Frequency Domain

1 In the **Model Builder** window, click **Step 2: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Electric Point Dipole 1**.

5 Right-click and choose **Disable**.

6 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Single-Ended Line Port Group**.

7 Right-click and choose **Disable**.

8 In the **Model Builder** window, click **Study 3**.

9 In the **Settings** window for **Study**, type `Study 3, Differential Line` in the **Label** text field.

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

RESULTS

S-parameter, Differential Line

1 In the **Model Builder** window, under **Results>Derived Values** click **S-parameter (emw)**.

2 In the **Settings** window for **Global Evaluation**, type `S-parameter, Differential Line` in the **Label** text field.

Port Mode Impedance, Differential Line

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Port Mode Impedance, Differential Line` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3, Differential Line/ Solution 5 (sol5)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain> TEM boundary mode analysis>emw.Zmode_1 - TEM mode port characteristic impedance -** Ω.

5 Click **Evaluate**.

Electric Field Norm, Differential Line

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Field Norm, Differential Line` in the **Label** text field.

Multislice

1 In the **Model Builder** window, expand the **Electric Field Norm, Differential Line** node.

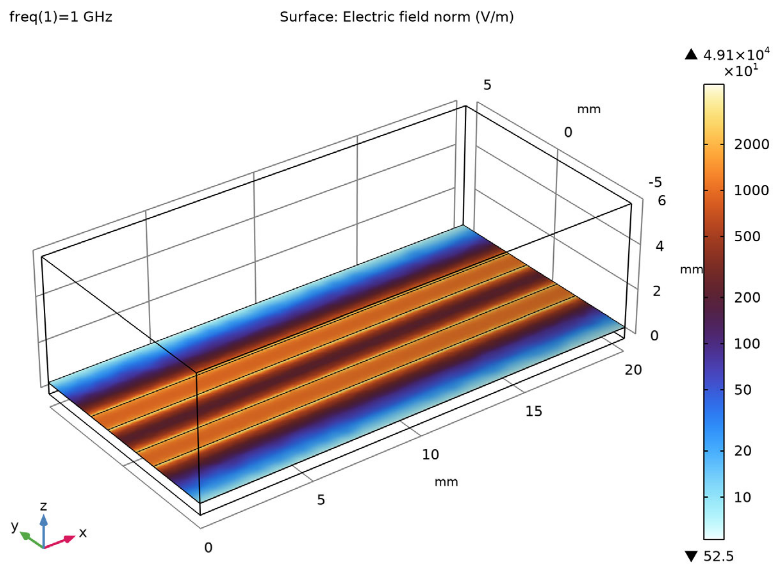
2 Right-click **Results>Electric Field Norm, Differential Line>Multislice** and choose **Delete**.

Surface 1

In the **Model Builder** window, under **Results>Electric Field Norm, Single-Ended Line** right-click **Surface 1** and choose **Copy**.

Surface 1

In the **Model Builder** window, right-click **Electric Field Norm, Differential Line** and choose **Paste Surface**.



Electric Field Ez, Differential Line

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

2 In the **Settings** window for **3D Plot Group**, type `Electric Field Ez, Differential Line` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3, Differential Line/ Solution 5 (sol5)**.

Surface 1

1 Right-click **Electric Field Ez, Differential Line** and choose **Surface**.

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

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

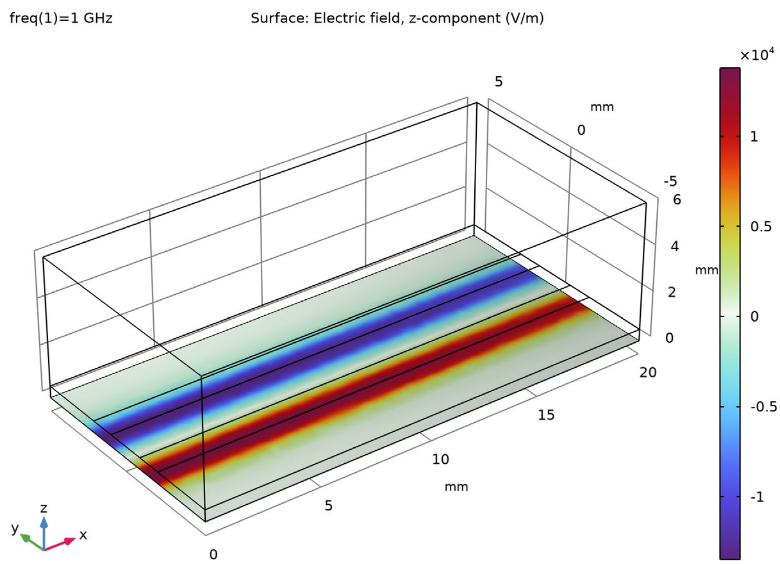
4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>Dipole** in the tree.

6 Click **OK**.

Selection 1

1. Right-click **Surface 1** and choose **Selection**.
2. Select Boundary 3 only.
3. In the **Electric Field Ez, Differential Line** toolbar, click **Plot**.



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>TEM Boundary Mode Analysis**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

Step 1: TEM Boundary Mode Analysis

1. In the **Settings** window for **TEM Boundary Mode Analysis**, locate the **Physics and Variables Selection** section.
2. Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Single-Ended Line Port Group**.

4 Right-click and choose **Disable**.

Step 2: Frequency Domain

1 In the **Model Builder** window, click **Step 2: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Single-Ended Line Port Group**.

5 Right-click and choose **Disable**.

6 In the **Model Builder** window, click **Study 4**.

7 In the **Settings** window for **Study**, type `Study 4, Differential Line with Noise` in the **Label** text field.

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

RESULTS

S-parameter, Differential Line with Noise

1 In the **Model Builder** window, under **Results>Derived Values** click **S-parameter (emw)**.

2 In the **Settings** window for **Global Evaluation**, type `S-parameter, Differential Line with Noise` in the **Label** text field.

Electric Field Norm, Differential Line with Noise

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Field Norm, Differential Line with Noise` in the **Label** text field.

Multislice

1 In the **Model Builder** window, expand the **Electric Field Norm, Differential Line with Noise** node.

2 Right-click **Results>Electric Field Norm, Differential Line with Noise>Multislice** and choose **Delete**.

Surface 1

In the **Model Builder** window, under **Results>Electric Field Norm, Single-Ended Line** right-click **Surface 1** and choose **Copy**.

Surface 1

In the **Model Builder** window, right-click **Electric Field Norm, Differential Line with Noise** and choose **Paste Surface**.