Model created in COMSOL Multiphysics 6.4

H-Bend Waveguide 3D

Introduction

This example shows how to model a rectangular waveguide for microwaves. A single hollow waveguide can conduct two kinds of electromagnetic waves: transversal magnetic (TM) or transversal electric (TE) waves. This example examines a TE wave, one that has no electric field component in the direction of propagation. More specifically, for this example you select the frequency and waveguide dimension so that TE_{10} is the single propagating mode. In that mode the electric field has only one nonzero component — a sinusoidal with two nodes, one at each of the walls of the waveguide. This makes it possible to set up and solve the model in 2D, which is done in a separate version; see H-Bend Waveguide 2D.

One important design aspect is how to shape a waveguide to go around a corner without incurring unnecessary losses in signal power. Unlike in wires, these losses usually do not result from ohmic resistance but instead arise from unwanted reflections. You can minimize these reflections by keeping the bend smooth with a large enough radius. In the range of operation the transmission characteristics (the ability of the waveguide to transmit the signal) must be reasonably uniform for avoiding signal distortions.

With air as the inside medium of the waveguide, the transmission is nearly perfect throughout the range of operation. In this example, to make the simulation and the results more interesting, the bend is filled with silica glass, a dielectric medium.

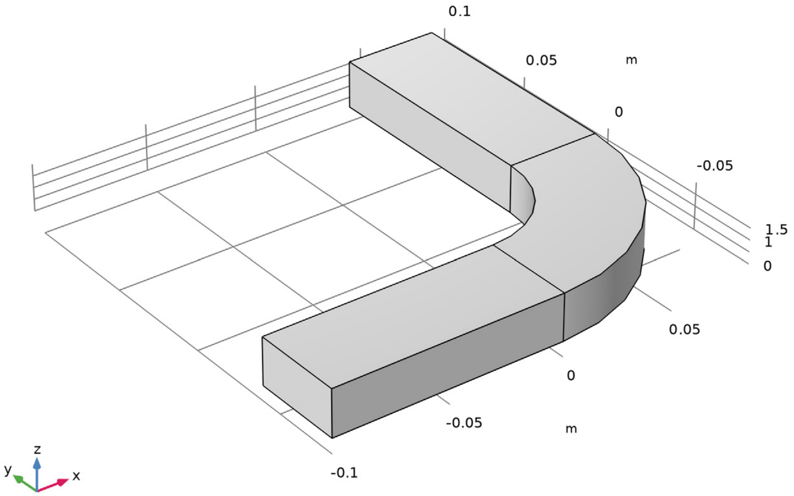
The model also shows how to systematically compute and export all S-parameters to a Touchstone file.

Model Definition

This example illustrates how to create a model that computes the electromagnetic fields and transmission characteristics of a 90° bend for a given radius. This type of waveguide bends changes the direction of the **H** field components and leaves the direction of the **E** field unchanged. The waveguide is therefore called an *H-bend*. The H-bend design used in this example is well-proven in real-world applications and you can buy similar waveguide bends online from a number of manufacturers. This particular bend performs optimally in the ideal case of perfectly conducting walls.

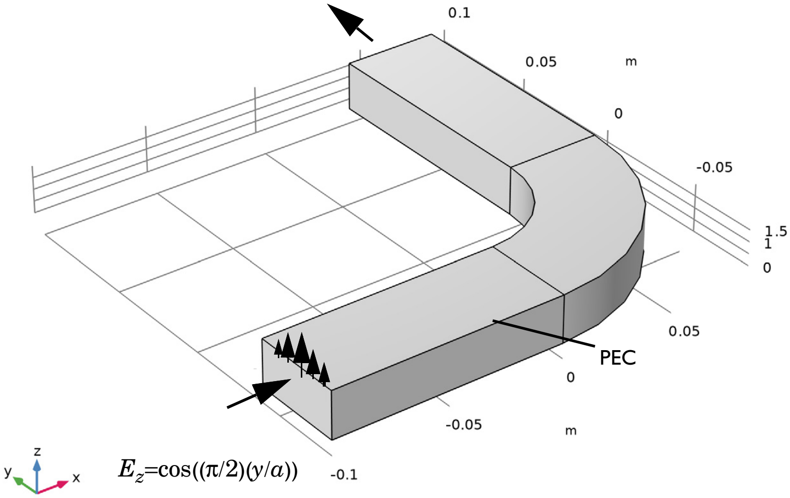
The waveguide walls are typically plated with a very good conductor, such as silver. In this example the walls are considered to be made of a perfect conductor, which means that the tangential component of the electric field is zero, or that $\mathbf{n} \times \mathbf{E} = \mathbf{0}$ on the boundaries. This boundary condition is referred to as a *perfect electric conductor* (PEC) boundary condition.

The geometry is as follows:



The waveguide is considered to continue indefinitely before and after the bend. This means that the input wave needs to have the form of a wave that has been traveling through a straight waveguide. The shape of such a wave is determined by the boundary conditions of Maxwell's equations on the sides of the metallic boundaries, that is, the PEC boundary condition. If polarized according to a TE_{10} mode, the shape is known analytically to be $\mathbf{E} = (0, 0, \sin(\pi(a-y)/(2a)))\cos(\omega t)$ given that the entrance boundary is centered around the $y = 0$ axis, and that the width of the waveguide, in the y direction, is $2a$.

The model is set up using the time-harmonic Electromagnetic Waves interface. This means that only the phasor component of the field is modeled. The incident field then has the form $\mathbf{E} = (0, 0, E_{0z}) = (0, 0, \sin(\pi(a-y)/(2a)))$, and is considered as part of the expression $\mathbf{E} = \mathrm{Re}\{(0, 0, \sin(\pi(a - y)/(2a))e^{j\omega t})\} = \mathrm{Re}\{\mathbf{E}e^{j\omega t}\}$, where complex-valued arithmetic has been used (also referred to as the $j\omega$ method).



The width of the waveguide is chosen so that it has a cutoff frequency of 3.7 GHz. This makes the waveguide operational up to 7.5 GHz. At higher frequencies other modes than the TE_{10} appear, causing a “dirty” signal. The input wave then splits into several modes that are hard to control without having large power losses. Below the cutoff frequency, no waves can propagate through the waveguide. This is an intrinsic property of microwave waveguides.

The cutoff frequency of different modes in a straight waveguide is given by the relation

$$(\nu_c)_{mn} = \frac{c}{2}\sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where m and n are the mode numbers ($m = 1, n = 0$ for the TE_{10} mode), a and b are the lengths of the sides of the waveguide cross-section, and c is the speed of light.

For this waveguide, $a = 2b$ and $b = 2$ cm.

The first few cutoff frequencies are $(\nu_c)_{10} = 3.7$ GHz, $(\nu_c)_{01} = 7.5$ GHz, $(\nu_c)_{11} = 8.4$ GHz. The frequencies used in this example are from 4.0 GHz to 5.2 GHz, and hence entirely within the single-mode range.

On the input boundary, the Port boundary condition lets you choose which mode to send in. Any reflected waves having the same shape are transmitted back through this same boundary. The output boundary also uses a Port condition, but without field excitation, to specify the shape of the wave that it lets pass through. Using port boundary conditions means that you automatically gain access to postprocessing variables for the S-parameters.

Results and Discussion

The wave is found to propagate through the bend with a varying amount of reflection depending on the frequency.

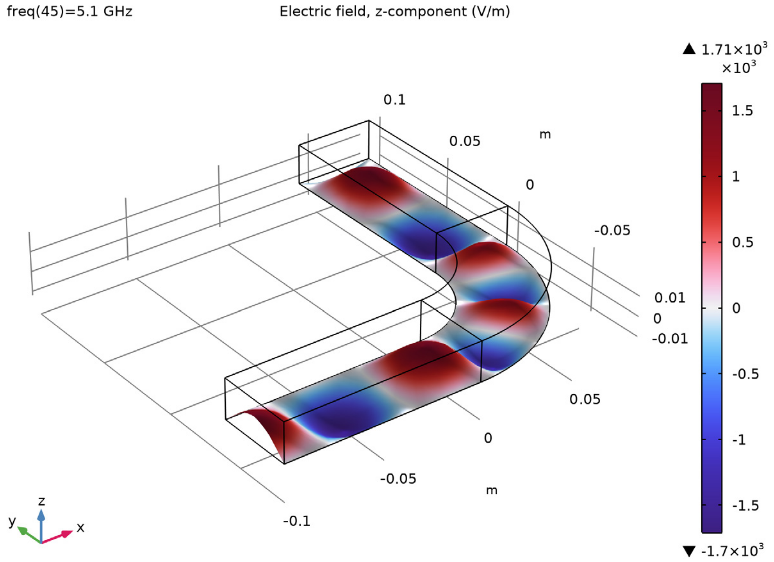


Figure 1: The z-component of the electric field for a frequency of 5.1 GHz.

The S-parameters are shown as functions of the frequency in Figure 2.

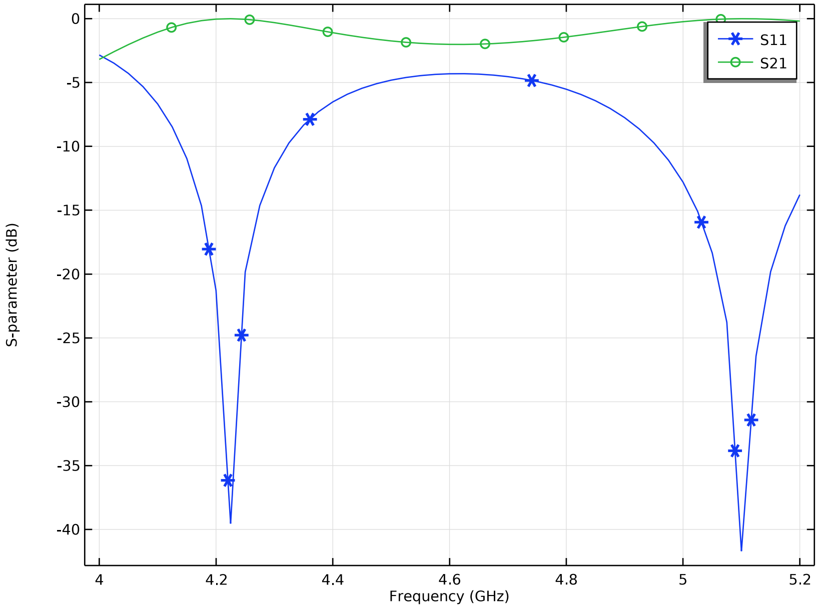


Figure 2: The S-parameters, on a dB scale, as a function of the frequency.

The two dips in S_{21} closely correspond to cavity resonances of the dielectric region in the bend. At these frequencies, the transmission is almost perfect. (Without the dielectric, the transmission would be nearly as good throughout the frequency range.)

Application Library path: `RF_Module/Transmission_Lines_and_Waveguides/h_bend_waveguide_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**.

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 **General Studies** > **Frequency Domain**.
6. Click **Done**.

STUDY 1

Step 1: Frequency Domain

Define the study frequency ahead of performing any frequency-dependent operation such as building mesh. The physics-controlled mesh uses the highest frequency value in the specified range.

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 `range(4[GHz],25[MHz],5.2[GHz])`.

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
`l_wg`	`10[cm]`	0.1 m	`Waveguide length`
`w_wg`	`4[cm]`	0.04 m	`Waveguide width`
`h_wg`	`2[cm]`	0.02 m	`Waveguide height`

GEOMETRY 1

Work Plane 1 (wp1)

1. In the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, click **Go to Plane Geometry**.

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 `w_wg`.

Work Plane 1 (wp1) > Circle 2 (c2)

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 `2*w_wg`.
4. In the **Sector angle** text field, type `90`.
5. Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

Work Plane 1 (wp1) > Difference 1 (dif1)

1. In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.
2. Select the object **c2** 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 object **c1** only.

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 `w_wg`.
4. In the **Height** text field, type `l_wg`.
5. Locate the **Position** section. In the **xw** text field, type `w_wg`.

Work Plane 1 (wp1) > Rectangle 2 (r2)

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 `l_wg`.
4. In the **Height** text field, type `w_wg`.
5. Locate the **Position** section. In the **xw** text field, type `-l_wg`.
6. In the **yw** text field, type `-2*w_wg`.
7. In the **Work Plane** toolbar, click **Build All**.

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

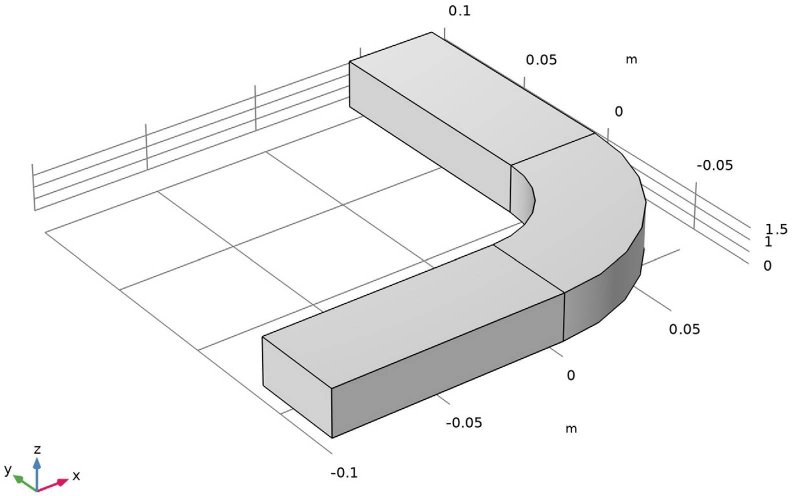
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
h_wg

4 Click **Build All Objects**.

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



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Wave Equation, Electric 1

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

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Refractive index**.

Port 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Port**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Port**, locate the **Port Properties** section.
4. From the **Type of port** list, choose **Rectangular**.

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

Port 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Port**.
2. Select Boundary 15 only.
3. In the **Settings** window for **Port**, locate the **Port Properties** section.
4. From the **Type of port** list, choose **Rectangular**.

   The default boundary condition is perfect electric conductor, which is fine for all exterior boundaries except the ports. The software automatically imposes continuity on interior boundaries.

MATERIALS

Air

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Air` in the **Label** text field.
3. Select Domains 1 and 3 only.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	1	1	Refractive index

Silica Glass

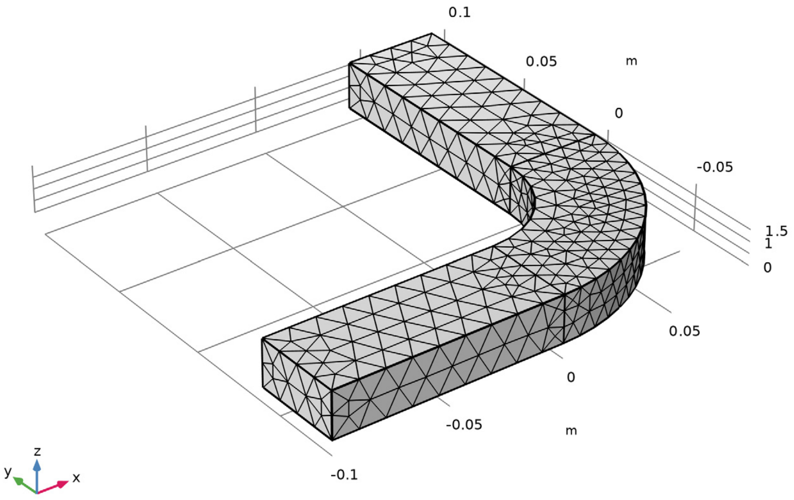
1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Silica Glass` in the **Label** text field.
3. Select Domain 2 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	1.44	1	Refractive index

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.



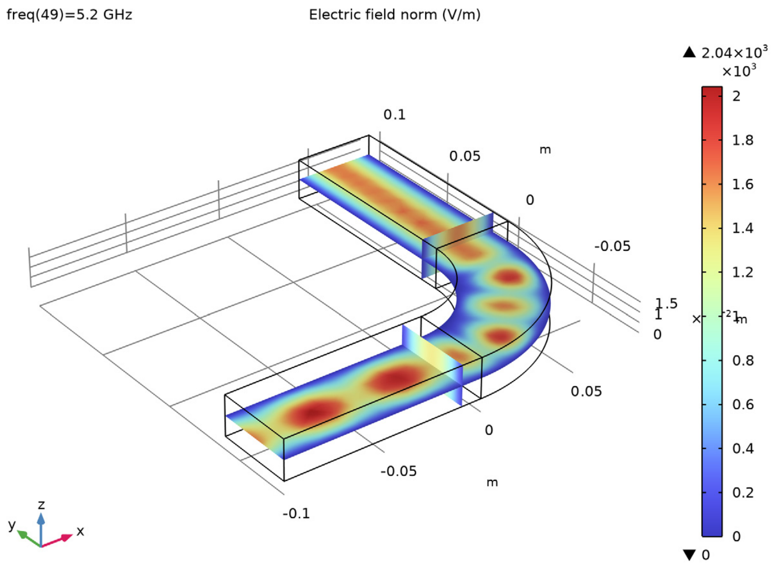
If you look closely at the mesh, you can see that it is indeed a bit finer in the bend than elsewhere.

STUDY 1

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

RESULTS

Electric Field (emw)



The default plot shows the distribution of the electric field norm on slices of the waveguide, for the highest frequency in the sweep. Note the wave pattern in the bend and the rectangular input section. This indicates standing waves caused by reflections in the bend. In contrast, the pattern beyond the bend is independent of the *y*-coordinate, showing that the output port does a good job of transmitting the wave.

An S-parameter plot gives you a quantitative measure of how much of the wave is transmitted and reflected at different frequencies.

Global 1

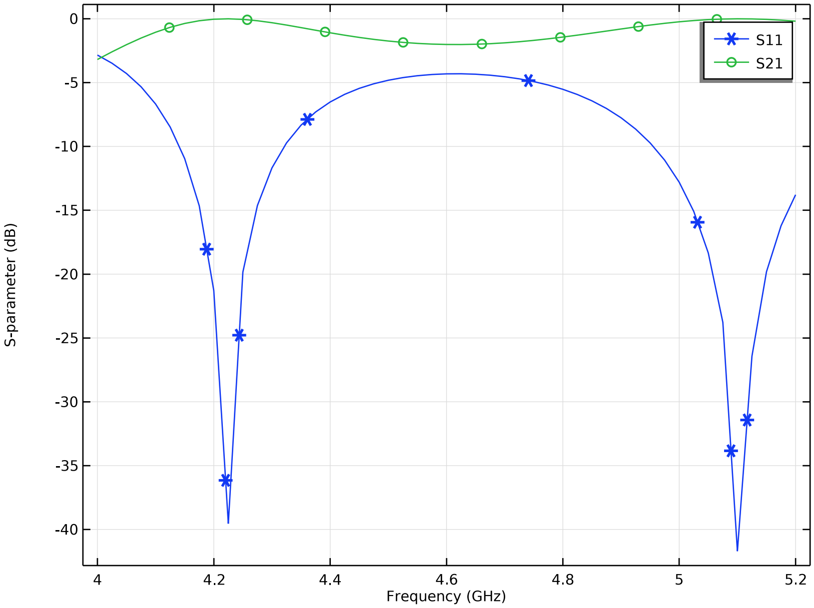
1 In the **Model Builder** window, expand the **Results** > **S-Parameter (emw)** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click to expand the **Coloring and Style** section.

3 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

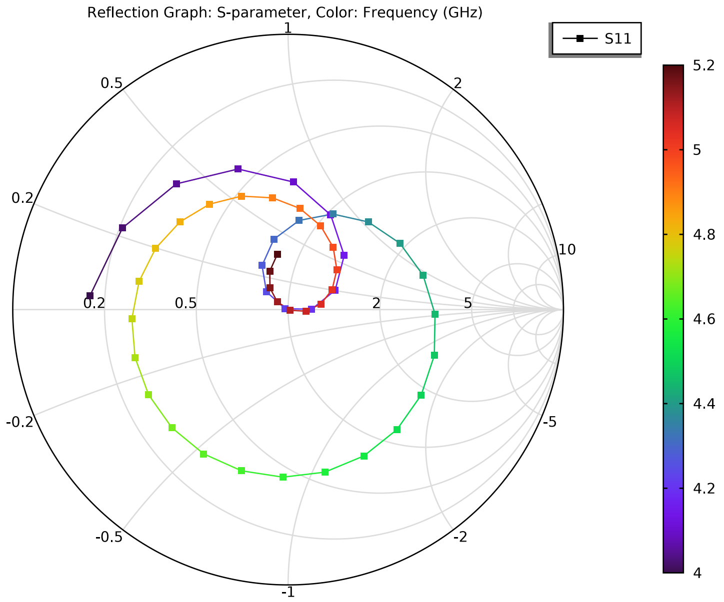
5 In the **S-Parameter (emw)** toolbar, click **Plot**.



The result, which should look like Figure 2, shows that the transmission varies throughout the frequency range. Note in particular that S_{21} has two deep dips, corresponding to almost perfect transmission. This is the result of resonances in the bend. To confirm this, try looking at the field distribution for the frequency where the upper peak is located, 5.1 GHz.

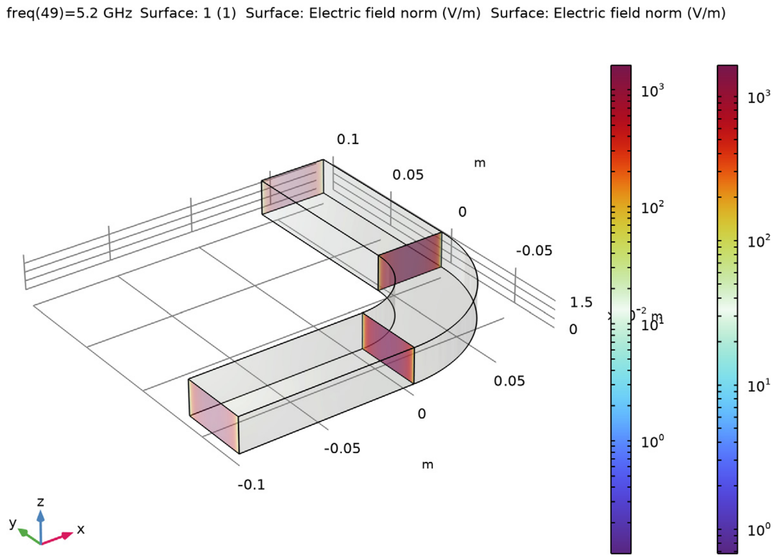
Smith Plot (emw)

In the **Model Builder** window, under **Results** click **Smith Plot (emw)**.



Electric Field, Logarithmic (emw)

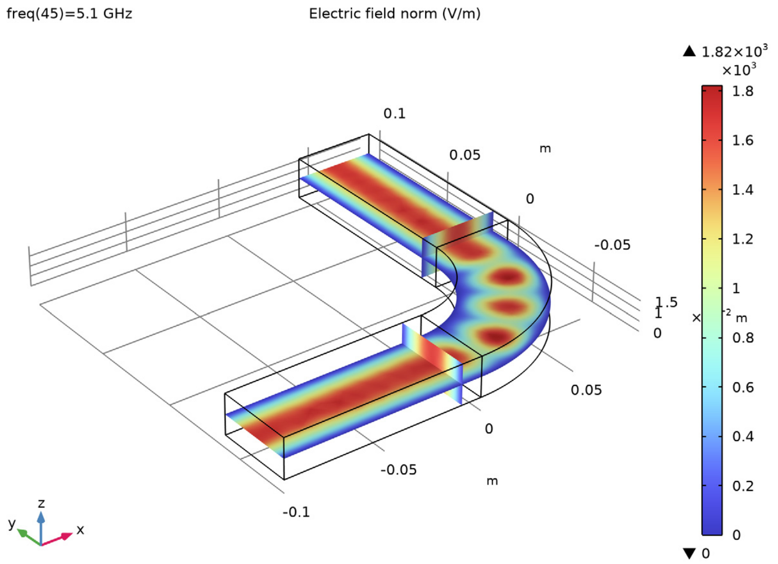
In the **Model Builder** window, click **Electric Field, Logarithmic (emw)**.



Electric Field (emw)

1. In the **Model Builder** window, click **Electric Field (emw)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Parameter value (freq (GHz))** list, choose **5.1**.

4. In the **Electric Field (emw)** toolbar, click **Plot**.



The standing wave pattern still remains in the bend, but at this frequency it is almost completely gone in the input section.

For an alternative view, you can plot the instantaneous value of the electric field inside the waveguide. Only the z component will be substantially nonzero. For a better view, add also deformation. Replace the Multislice with a single horizontal slice plot.

Multislice 1

1. In the **Model Builder** window, expand the **Electric Field (emw)** node.
2. Right-click **Multislice 1** and choose **Delete**.

Slice 1

1. Right-click **Electric Field (emw)** and choose **Slice**.
2. In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Electric** > **Electric field - V/m** > **emw.Ez - Electric field, z-component**.
3. Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.
4. From the **Entry method** list, choose **Coordinates**.
5. Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

6 In the **Electric Field (emw)** toolbar, click **Plot**.

The Wave color table looks its best using a symmetric range. You can also play with a deformed shape plot to make the waves appear more clearly.

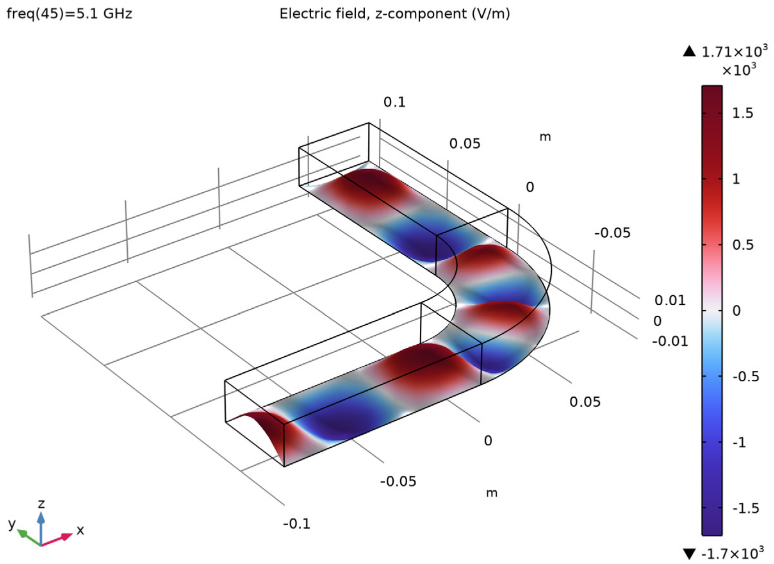
7 Click to expand the **Range** section. Locate the **Coloring and Style** section. From the **Scale** list, choose **Linear symmetric**.

Deformation 1

1 Right-click **Slice 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Electric** > **emw.Ex,emw.Ey,emw.Ez - Electric field**.

3 In the **Electric Field (emw)** toolbar, click **Plot**.



The remaining instructions show you how to systematically solve with one port active at a time, and save the results in the Touchstone format.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

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

2. In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Port Sweep Settings** section.
3. Select the **Use manual port sweep** checkbox.
4. Click **Configure Sweep Settings**. By clicking the **Configure Sweep Settings** button, all necessary port sweep settings such as sweep parameter and parametric study step will be automatically added. It is necessary to run the parametric sweep with port names to get a full S-parameter matrix.
5. Select the **Export Touchstone file** checkbox.
6. Click the **Browse** button and select a file to which you want to export the results in the Touchstone format. If the file does not exist, it will be created.

STUDY 1

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

RESULTS

The Touchstone file should now contain the complete output from the model. The new solution dataset contains two frequency sweeps, one for each port.

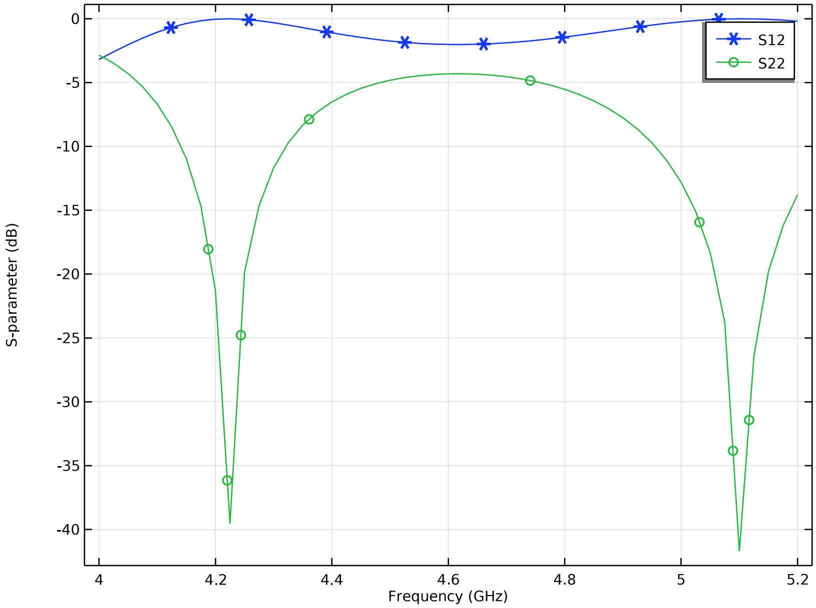
Global 1

As you can see, after performing the parametric sweep over the ports, the S-parameter plot you created previously is empty. To restore the plot, you need to change the dataset and specify the inner parameter - that is, the frequency - as the quantity to display along the horizontal axis.

1. In the **Model Builder** window, under **Results** > **S-Parameter (emw)** click **Global 1**.
2. In the **Settings** window for **Global**, locate the **Data** section.
3. From the **Dataset** list, choose **Probe Solution 2 (sol1)**.

   To verify the reciprocity of the waveguide, you can add the S-parameters S12dB and S22dB to the Expressions table and change the parameter selection for PortName:
4. Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Ports** > **S-parameter, dB - dB** > **emw.S12dB - S12**.
5. Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Ports** > **S-parameter, dB - dB** > **emw.S22dB - S22**.

6 In the **S-Parameter (emw)** toolbar, click **Plot**.



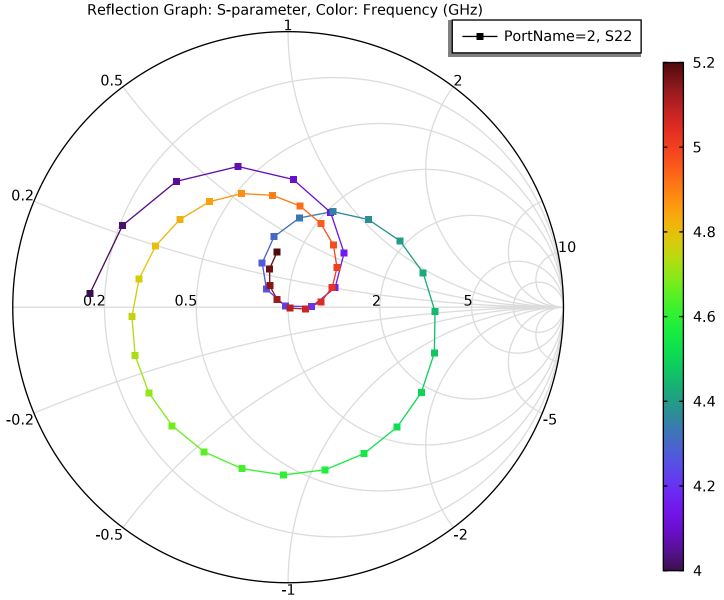
Smith Plot (emw)

1. In the **Model Builder** window, under **Results** click **Smith Plot (emw)**.
2. In the **Settings** window for **Smith Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4. From the **Parameter selection (PortName)** list, choose **Last**.

Reflection Graph 1

1. In the **Model Builder** window, expand the **Smith Plot (emw)** node, then click **Reflection Graph 1**.
2. In the **Settings** window for **Reflection Graph**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Ports** > **S-parameter** > **emw.S22 - S22**.

3 In the **Smith Plot (emw)** toolbar, click **Plot**.



Reflection Graph 1

1. In the **Model Builder** window, expand the **Smith Plot (emw) 1** node, then click **Reflection Graph 1**.
2. In the **Settings** window for **Reflection Graph**, click to expand the **Coloring and Style** section.
3. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
4. From the **Positioning** list, choose **Interpolated**.

Finish by verifying the reciprocity of the waveguide on the Smith plot.