



# H-Bend Waveguide 2D

## Introduction

This example is a 2D version of H-Bend Waveguide 3D, which shows how to model a bending rectangular waveguide for microwaves. For a general introduction, see the model [H-Bend Waveguide 3D](#).

The dimensions of the waveguide and the frequency range used in this example are such that  $TE_{10}$  is the single propagating mode. In this mode, if the bend is in the  $xy$ -plane, the electric field only has a  $z$ -component, which furthermore is independent of the  $z$ -coordinate. This makes it possible to set up and solve the model in a 2D geometry.

## Model Definition

The considered geometry is an  $xy$ -plane cross section of the 3D geometry, as seen in [Figure 1](#). This figure also sums up the material and boundary settings, which are the same as in the 3D model. The main advantage with setting up the model in 2D is that it solves much faster and uses less memory. As a consequence, this version of the example does not stress the need to adapt the mesh to the wavelength, but simply lets you apply a mesh that is more than fine enough.

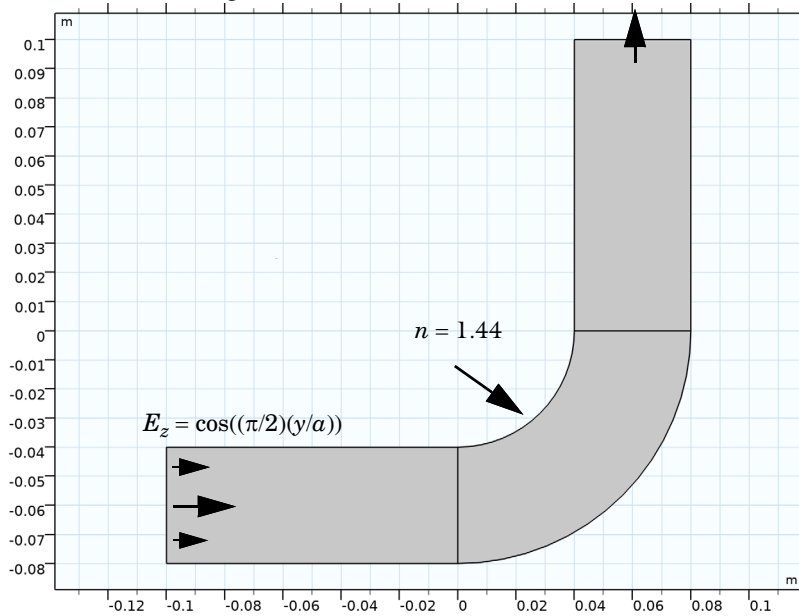


Figure 1: The geometry of the example.

The examples also shows the usage of frequency domain source sweep study that solves for fast port sweep. This study step helps solve a frequency domain study that is sweeping among ports and lumped ports calculating a full S-parameter matrix.

## Results and Discussion

Figure 2 shows the norm of the electric field at one of the frequencies where the bend has a resonance. The absence of a wave pattern in the input section indicates that the transmission is nearly perfect.

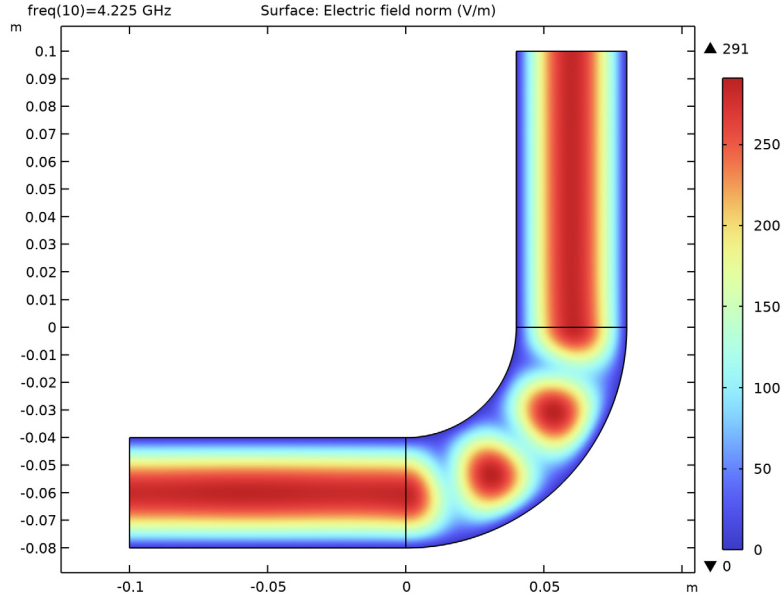


Figure 2: The electric field norm at a frequency of 4.225 GHz.

Figure 3 shows the S-parameters in a dB scale. The result agrees very well with that of the 3D model. The h-band waveguide is passive and reciprocal and the full S-parameter matrix computed at 5.1 GHz is symmetric.

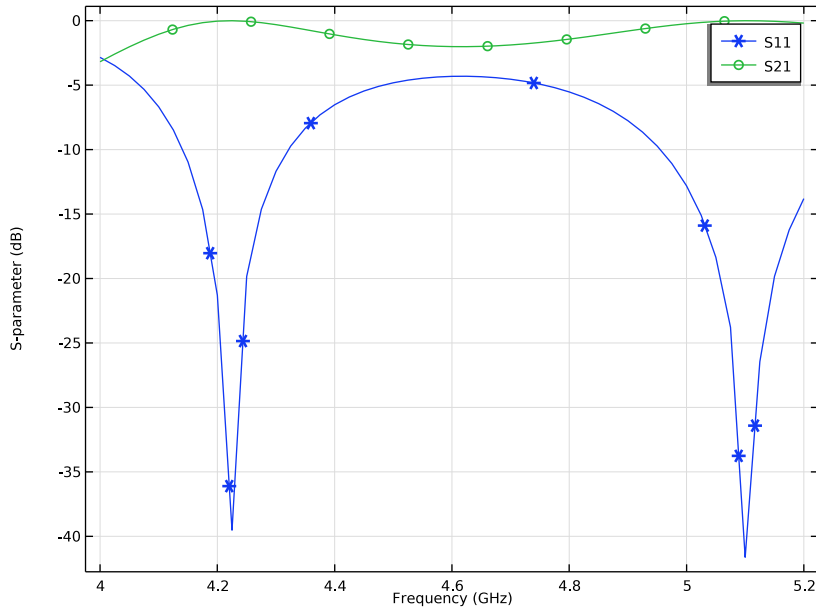


Figure 3: The S-parameters, in a dB scale, as functions of the frequency.

---

**Application Library path:** RF\_Module/Transmission\_Lines\_and\_Waveguides/h\_bend\_waveguide\_2d


---

### 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**.
- 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] ).

## GEOMETRY 1


### *Circle 1 (c1)*

- 1 In the **Geometry** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 0.08.


### *Circle 2 (c2)*

- 1 In the **Geometry** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 0.04.


### *Square 1 (sq1)*

- 1 In the **Geometry** toolbar, click  **Square**.
- 2 In the **Settings** window for **Square**, locate the **Size** section.
- 3 In the **Side length** text field, type 0.08.
- 4 Locate the **Position** section. In the **y** text field, type -0.08.




### *Compose 1 (co1)*

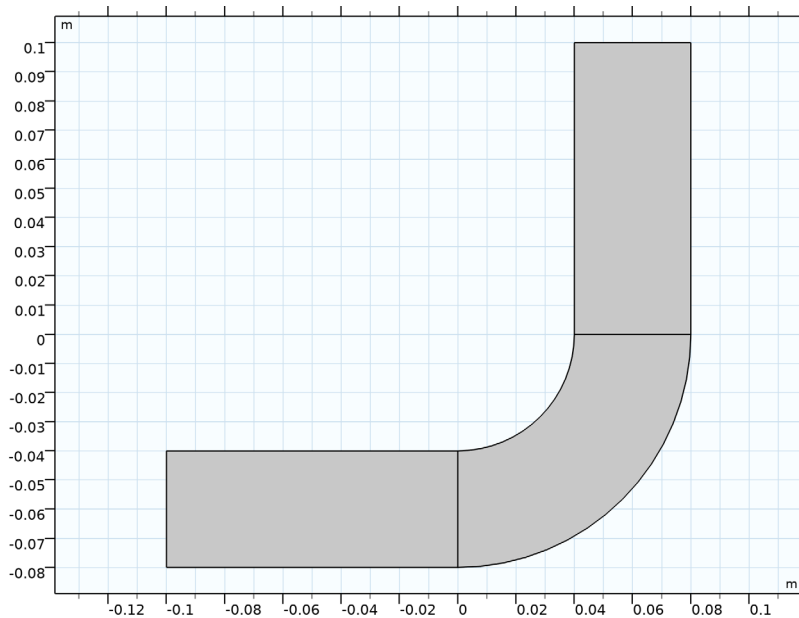
- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Compose**.
- 2 Click in the **Graphics** window and then press Ctrl+A to select all objects.
- 3 In the **Settings** window for **Compose**, locate the **Compose** section.
- 4 In the **Set formula** text field, type  $sq1 * (c1 - c2)$ .

*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 0.04.
- 4 In the **Height** text field, type 0.1.
- 5 Locate the **Position** section. In the **x** text field, type 0.04.

*Rectangle 2 (r2)*

- 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 0.1.
- 4 In the **Height** text field, type 0.04.
- 5 Locate the **Position** section. In the **x** text field, type -0.1.
- 6 In the **y** text field, type -0.08.
- 7 Click  **Build All Objects**.
- 8 Click the  **Zoom Extents** button in the **Graphics** toolbar.



## 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 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index, real part (n)**.
- 5 Click **+ Add to Material**.
- 6 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Refractive index, real part	$n_{iso}$ ; $n_{ii} = n_{iso}$ , $n_{ij} = 0$	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 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index, real part (n)**.
- 5 Click **+ Add to Material**.
- 6 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Refractive index, real part	$n_{iso}$ ; $n_{ii} = n_{iso}$ , $n_{ij} = 0$	1.44	1	Refractive index

## ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

With TE waves, only the  $z$  component of the electric field needs to be solved for.


- 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 **Components** section.
- 3 From the **Electric field components solved for** list, choose **Out-of-plane vector**.

#### *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**.

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.

#### *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.

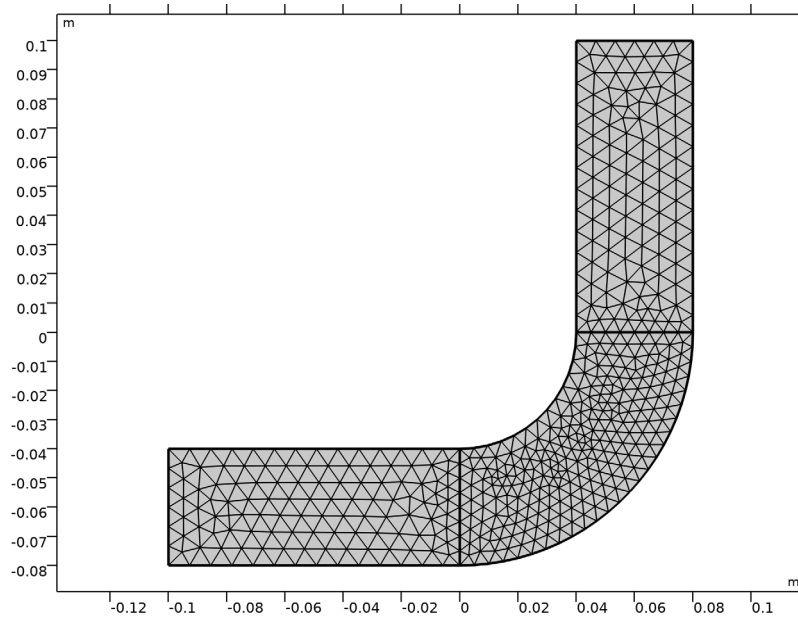
#### *Port 2*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Port**.
- 2 Select Boundary 7 only.
- 3 In the **Settings** window for **Port**, locate the **Port Properties** section.
- 4 From the **Type of port** list, choose **Rectangular**.




## MESH I

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

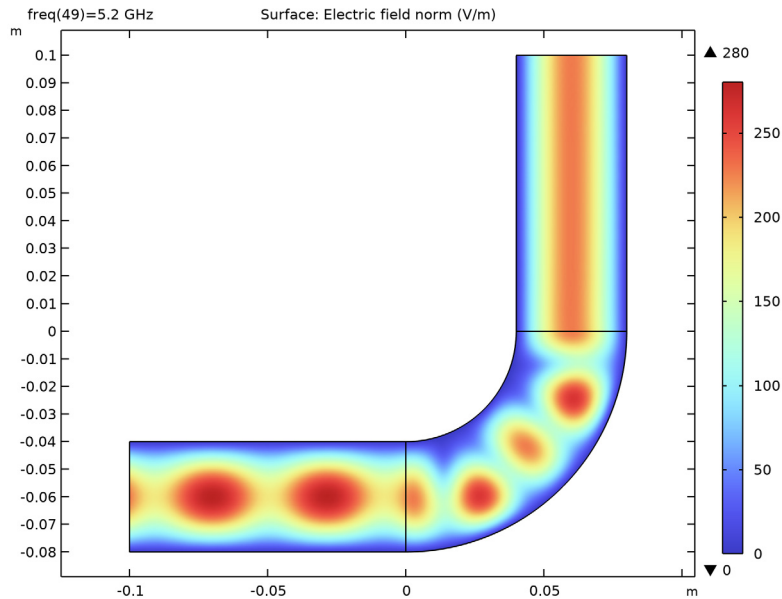


## STUDY I

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

## RESULTS

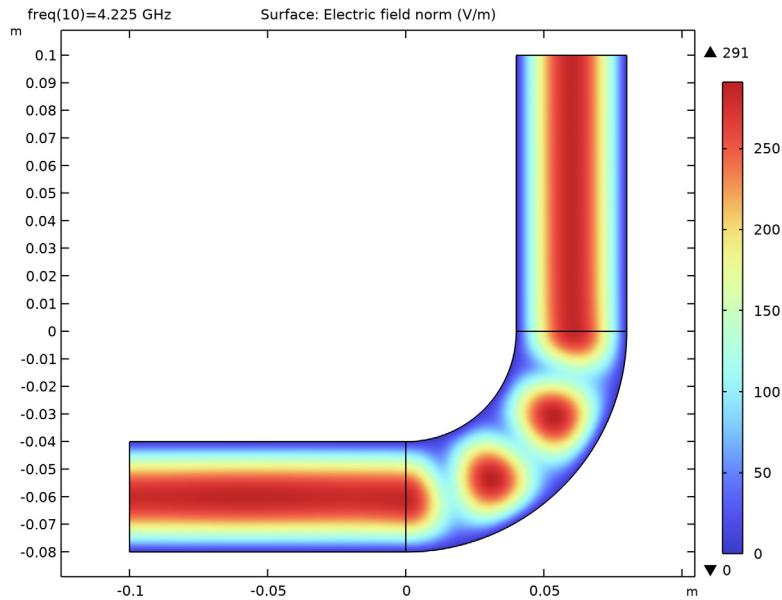
*Electric Field (emw)*



The default plot shows the norm of the electric field for the highest frequency, 5.2 GHz. To verify that the solution resembles the 3D version, try plotting a frequency where you expect a transmission peak.

- 1 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 2 From the **Parameter value (freq (GHz))** list, choose **4.225**.


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

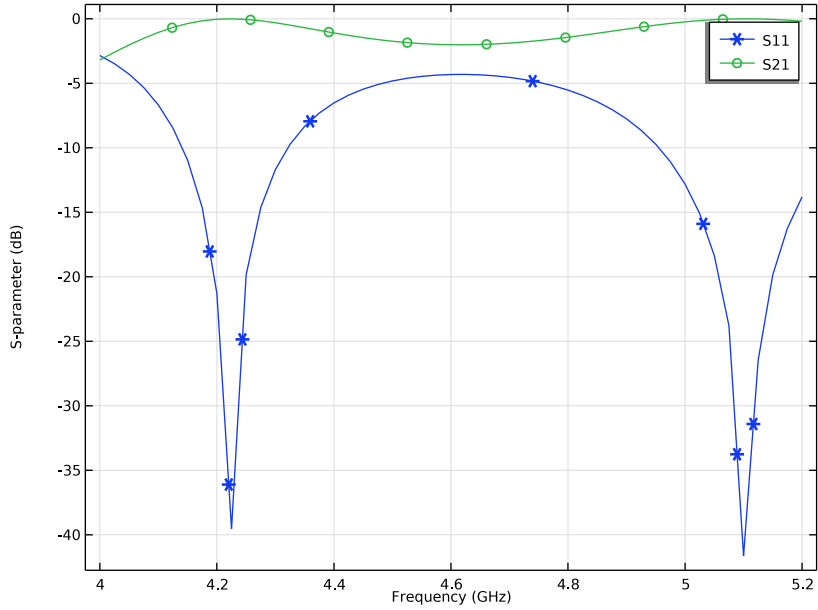


Finally, plot the S-parameters.

*Global 1*

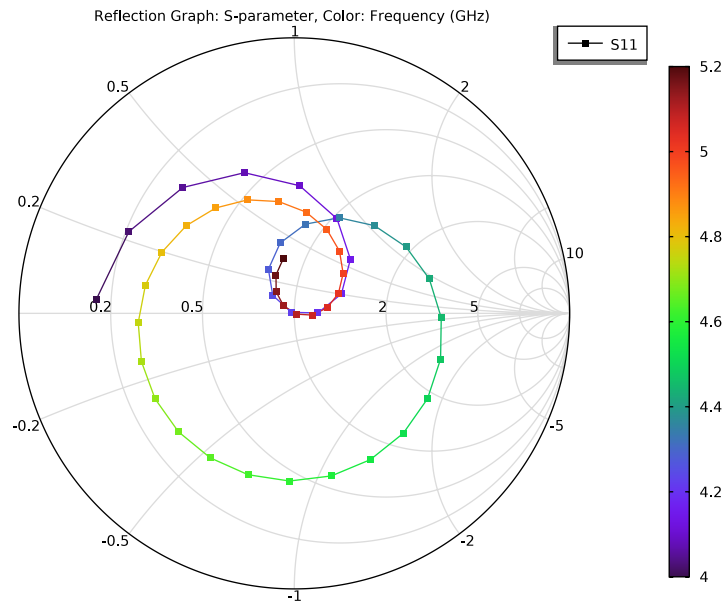
- 1 In the **Model Builder** window, expand the **Results>S-parameter (emw)** node, then click **Global 1**.
- 2 In the **Settings** window for **Global**, click to expand the **Coloring and Style** section.
- 3 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 4 From the **Positioning** list, choose **Interpolated**.

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





### Smith Plot (emw)

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



Next, add a study step to solve a frequency domain study sweeping over ports to extract a full S-parameter matrix.


### 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>Frequency Domain Source Sweep**.
- 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: Frequency Domain Source Sweep

- 1 In the **Settings** window for **Frequency Domain Source Sweep**, locate the **Study Settings** section.
- 2 In the **Frequencies** text field, type 5.1 [GHz].

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

## **RESULTS**

*S-parameter (emw)*

Review the evaluated full S-parameter matrix. Since the device is passive and reciprocal, the computed matrix is also symmetric.