Created in COMSOL Multiphysics 6.2

# Coaxial to Waveguide Coupling

## Introduction

Feeding a waveguide from a coaxial cable is a straightforward way to get electromagnetic waves inside the waveguide. This example shows how to compute the reflection and transmission in a coupling from a coaxial cable to a rectangular waveguide.

## Model Definition

As seen in the results plot in Figure 1, the coaxial cable is attached to an opening in the bottom of the rectangular waveguide. The inner conductor of the cable continues into the waveguide in order to transmit the waves. The model is run at a frequency of 6 GHz.

The wave coming in through the coaxial cable is set up using a Port boundary condition, with a 1 W coaxial feed selected. At the passive output port, the fundamental rectangular TE10 mode is assumed. The interpretation of the port conditions is that the coaxial cable and the rectangular waveguide continue indefinitely outside the model. The model therefore isolates the properties of the coupling from the properties of any outside circuit connecting to the other end of the waveguide or the cable.

The port boundary conditions are perfectly transparent only to their own specified mode. These same modes are also used in the automatic S-parameter analysis. For these reasons, it is important that the modeled sections are long enough that any evanescent waves have almost completely died out before they reach the ports. This leaves only the propagating modes - at 6 GHz, the only supported propagating modes are the specified fundamental modes.

An impedance boundary condition with material properties for copper is used for the coaxial conductors and the metal surfaces of the waveguide. At the considered frequency, copper is such a good conductor that a perfect electric conductor condition would be a reasonable alternative. However, the impedance condition also helps to accommodate the port boundaries, by smoothing out any numerical inconsistencies at the edges where they meet.

## Results and Discussion

For cable feeds it is interesting to compare the reflected wave going back into the coaxial cable with the wave propagating in the rectangular waveguide. You can make such a comparison by evaluating the S-parameter on the coaxial port and compare it with the S-parameter on the rectangular port. The following table shows the result for a frequency of 6 GHz in dB scale:

| PARAMETER | VALUE | DESCRIPTION |
|---|---|---|
| S11 | -8.4 dB | Reflected wave into the coaxial cable |
| S21 | -0.7 dB | Wave exiting the rectangular waveguide |

The following plot shows the electric field distribution. The *z*-component is shown in the rectangular waveguide and the *y*-component on a vertical slice through the coaxial cable and into the waveguide. Except for just at the end of the inner coaxial conductor, the *x*- and *y*-components of the field are negligible in the waveguide. Likewise, the *z*-component is barely present in the coaxial cable.

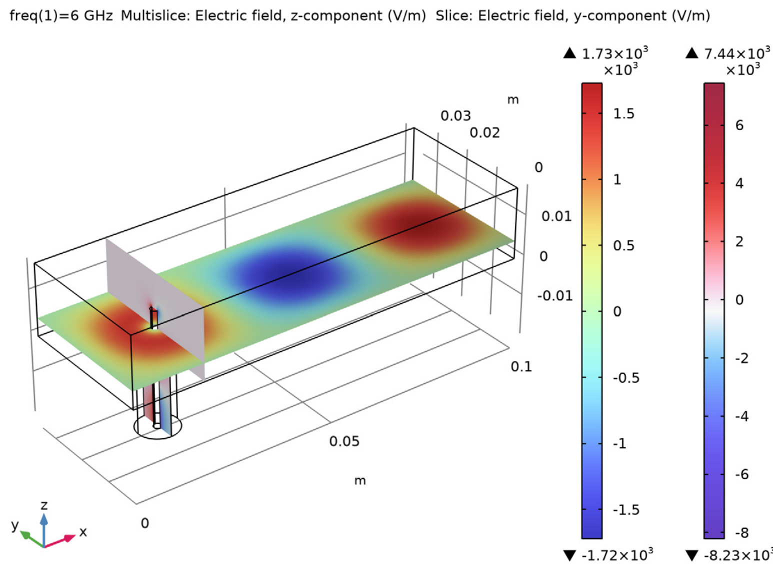



*Figure 1: Horizontal slice through the rectangular waveguide showing the z-component of the field, and vertical slice through the coaxial cable showing the y-component.*

**Application Library path:** `RF_Module/Transmission_Lines_and_Waveguides/coaxial_waveguide_coupling`

## *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 specified frequency value.

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 `6[GHz]`.

### GLOBAL DEFINITIONS

To help in creating the geometry, define some parameters defining its dimensions.

#### *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 |
|---|---|---|---|
| H | 17.5[mm] | 0.0175 m | Height, waveguide |
| L | 100[mm] | 0.1 m | Length, waveguide |
| l | 20[mm] | 0.02 m | Length, coaxial cable |
| R | 5[mm] | 0.005 m | Outer radius, coaxial cable |
| r | 1[mm] | 0.001 m | Inner radius, coaxial cable |

The width of the waveguide is taken to be twice the height.

## GEOMETRY 1

*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 `L`.

**4** In the **Depth** text field, type `2*H`.

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

*Cylinder 1 (cyl1)*

**1** In the **Geometry** toolbar, click **Cylinder**.

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

**3** In the **Radius** text field, type `R`.

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

**5** Locate the **Position** section. In the **x** text field, type `H`.

**6** In the **y** text field, type `H`.

**7** In the **z** text field, type `-l`.

**8** Click **Build All Objects**.

*Cylinder 2 (cyl2)*

**1** In the **Geometry** toolbar, click **Cylinder**.

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

**3** In the **Radius** text field, type `r`.

**4** In the **Height** text field, type `l+H/2`.

**5** Locate the **Position** section. In the **x** text field, type `H`.

6 In the **y** text field, type `H`.

7 In the **z** text field, type `-1`.

8 Click **Build All Objects**.

As you are going to represent all conductors as boundaries, you will now create a hole in the geometry replacing the inner conductor of the coaxial cable.

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

*Difference 1 (dif1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Click in the geometry to select and add the outer cylinder of the coaxial cable and the box representing the waveguide.

3 Select the objects **blk1** and **cyl1** only.

4 In the **Settings** window for **Difference**, locate the **Difference** section.

5 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

Select and add the inner cylinder of the coaxial cable.

6 Select the object **cyl2** only.

7 Click **Build All Objects**.

## DEFINITIONS

With the geometry set up, define a couple of boundary selections to allow for quick access to some commonly used boundaries.

*Metal Boundaries*

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Metal Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **All boundaries** check box.

5 Select Boundaries 1–7 and 9–20 only.

Remove Boundaries 8 and 21 from the selection. These are the boundaries where the wave will enter and leave your waveguide coupling.

*Input Port*

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Input Port` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

**4** Select Boundary 8 only.

Boundary 8 is the outer face of the coaxial cable.

*Output Port*

**1** In the **Definitions** toolbar, click **Explicit**.

**2** In the **Settings** window for **Explicit**, type `Output Port` in the **Label** text field.

**3** Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

**4** Select Boundary 21 only.

Boundary 21 is the boundary on the far end of the rectangular waveguide section.

## 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.

## MATERIALS

*Air (mat1)*

Select Domain 1 only.

*Dielectric*

**1** In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

**2** In the **Settings** window for **Material**, type `Dielectric` 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** |
|---|---|---|---|---|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | 2 | 1 | Basic |

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| 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 |

### ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in>Copper**.

3 Click **Add to Component** in the window toolbar.

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

### MATERIALS

*Copper (mat3)*

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

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

3 From the **Selection** list, choose **Metal Boundaries**.

### ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

*Impedance Boundary Condition 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose the boundary condition **Impedance Boundary Condition**.

2 In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Metal Boundaries**.

*Port 1*

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

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

3 From the **Selection** list, choose **Input Port**.

**4** Locate the **Port Properties** section. From the **Type of port** list, choose **Coaxial**.

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

*Port 2*

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

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

**3** From the **Selection** list, choose **Output Port**.

**4** Locate the **Port Properties** section. From the **Type of port** list, choose **Rectangular**.

## MESH 1

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

## STUDY 1

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

## RESULTS

*Multislice*

**1** In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.

**2** In the **Settings** window for **Multislice**, 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 **Multiplane Data** section. 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 `H/4`.

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

*Slice 1*

**1** In the **Model Builder** window, 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.Ey - Electric field, y-component**.

**3** Locate the **Plane Data** section. From the **Entry method** list, choose **Coordinates**.

**4** In the **X-coordinates** text field, type `H`.

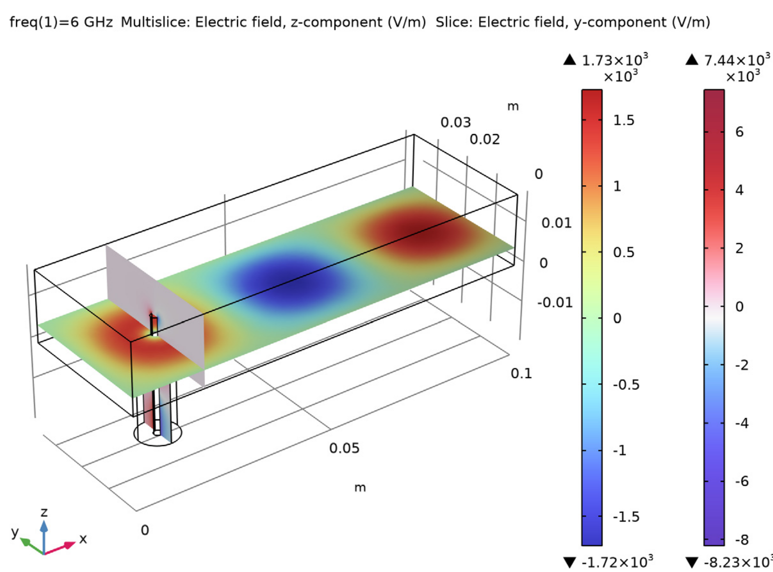
**5** Click to expand the **Inherit Style** section. Locate the **Coloring and Style** section. Click **Change Color Table**.

**6** In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.

**7** Click **OK**.

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

In the coaxial cable, the field is largely radial, meaning the $y$ component dominates. The other slice shows the field in the rectangular waveguide, where the $z$ component takes over.




The port conditions automatically compute and supply the S-parameters.