Experiment No 10:	**Simulate a Rectangular waveguide in COMSOL Multiphysics® software**
Aim:	To design and simulate a rectangular waveguide operating in an X band of frequency.
Problem Statement:	Observe the traveling field pattern of the dominant mode inside an X-band waveguide. X band waveguides operate between frequency range 8 to 12 GHz. Their inner width is 2.286 cm and their inner height is 1.016 cm. Assume the waveguide walls are 8 mm thick.
Procedure:	1. Open COMSOL Multiphysics software. From the **File** menu, choose **New**. In the **New** window, click **Model Wizard**. **2. 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**. Click **Study**. 4. In the **Select study** tree, select **Preset Studies>Frequency Domain** 5. Click **Done**. **3. GLOBAL DEFINITION** 1. On the **Home** toolbar, click **Parameters**. 2. In the **Settings** window for Parameters, locate the **Parameters** section. 3. In the table, enter the following settings: **4. GEOMETRY**

Name	Expression	Value	Description
i_w	2.286[cm]	0.02286 m	Inner_width
i_h	1.016[cm]	0.01016 m	Inner_height
T	8[mm]	0.008 m	Thickness
F	8.3[GHZ]	8.3E9 Hz	Frequency

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry**
2. In the **Settings** window for Geometry, locate the **Units** section.
3. From the **Length unit** list, choose **cm.**

Block 1(blk1)

1. On the **Geometry** toolbar, click **Block**.
2. Locate the **Size and Shape** section. In the **Width** text field, type i_w.
3. In the **Depth** text field, type 8[cm].
4. In the **Height** text field, type i_h
5. Locate the **Position** section. From the **Base** list, choose **Center**.

Block 2(blk2)

1. On the **Geometry** toolbar, click **Block**.
2. Locate the **Size and Shape** section. In the **Width** text field, type **i_w+t**
3. **In** the **Depth** text field, type **8[cm].**
4. In the **Height** text field, type **i_h+t**
5. Locate the **Position** section. From the **Base** list, choose **Center**.
6. Click the **Build All Objects** button.

4. ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

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

Port 1

1. On 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. In **Type of port** select **Rectangular.**
5. From the **Wave excitation at this port** list, choose **On**.

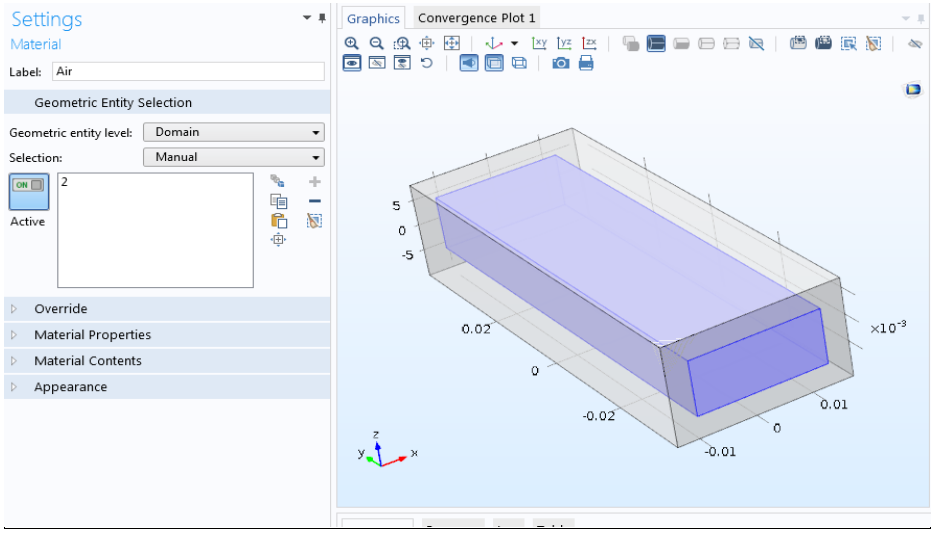
Port 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Port**.
2. Select Boundary 10 only.
3. In the **Settings** window for Port, locate the **Port Properties** section.
4. In **Type of port** select **Rectangular.**

5. MATERIALS: Assign material properties to the model. First, apply air to all domains.

Add Material

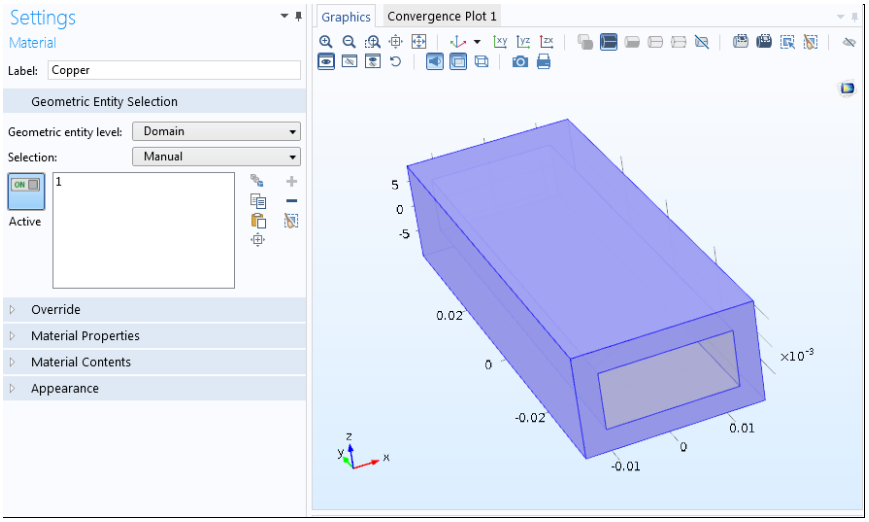
1. On the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-In>Air**.
4. Click **Add to Component** in the window toolbar.
5. Select **domain** in **geometric Entity Level.**
6. Select **domain 2. (as shown in fig.)**



Material : On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

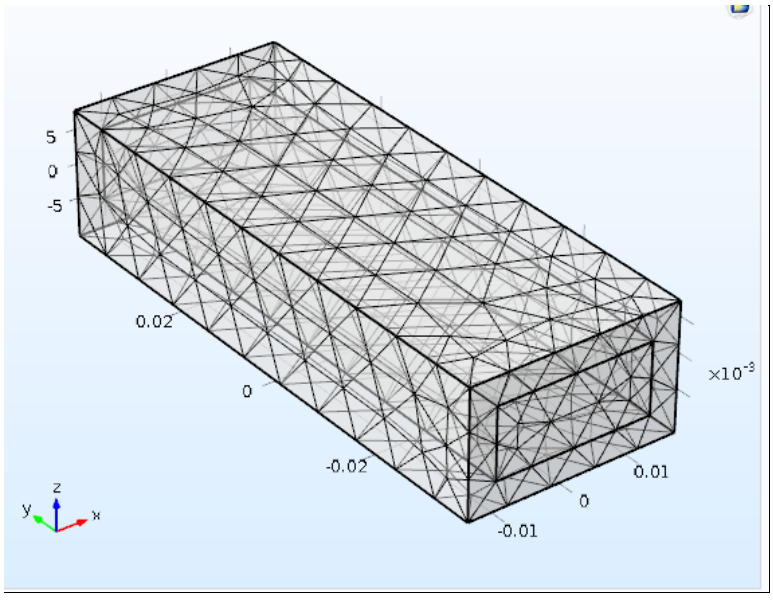
Material 2 (mat2)

1. On 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>copper**
4. Click **Add to Component** in the window toolbar.
5. Select **domain** in **geometric Entity Level.**
6. Select **domain 1 (as shown in fig.)**



6. MESH

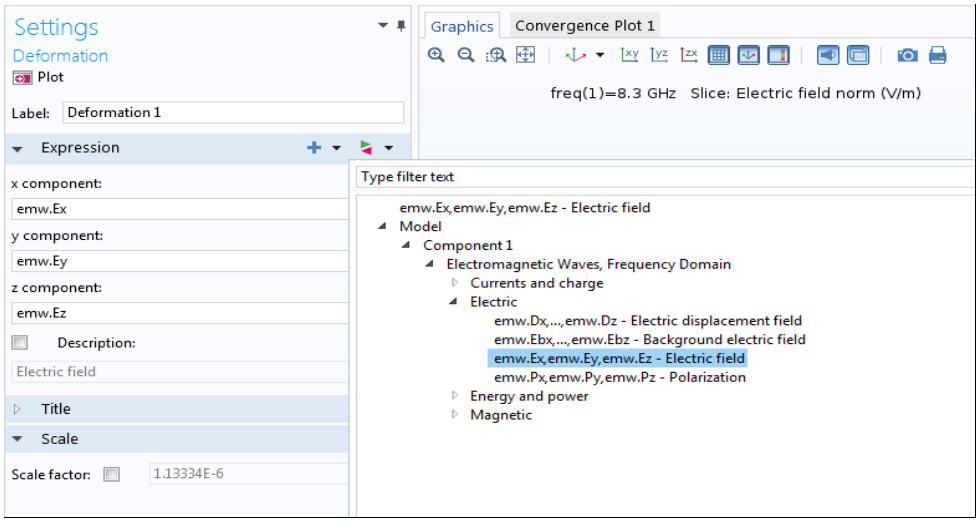
In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.



7. STUDY

Frequency Domain

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 **f.** 4. On the **Home** toolbar, click **Compute**. **8. RESULTS** *Electric Field (emw)* 2. In the **Model Builder** window, expand the **Electric Field** node, then click **Multislice 1**. 1. In the **Settings** window **Disable** Multislice. 2. Right click Electric Field (emw), click on Slice. 3. In Slice **setting** window under Plane **Data**, select xy plane and number of planes 1. 4. Right click on **Slice, select Deformation.** 5. In Deformation **Setting** window**,** click on **Replace Expression.** 6. Under **Electromagnetic Waves, Frequency Domain > Electric > emw.Ex,emw.Ey, emw.Ez-Electric Field.** 7. Click on **plot.** 
Observation:	Plot the electric and magnetic field patterns for TM_{11} mode and also check for TM_{10} mode and check if the mode exists.
Conclusion:	
Post Experiment Quiz:	1. Explain dominant, propagating, and evanescent modes in rectangular waveguides.