

**Operation: Learn Microwave Office**  
**Assignment 3**

Prepared by Dr. Aaron Scher

**Watch video tutorials**

AWR Design Environment (Microwave Office) tutorials are found at the following link:

[http://www.aaronscher.com/Microwave\\_office\\_video\\_tutorials/Microwave\\_office\\_video\\_tutorials.html](http://www.aaronscher.com/Microwave_office_video_tutorials/Microwave_office_video_tutorials.html)

This assignment is based on the following tutorials (found by clicking the link above)

*Video 5: How to use Microwave Office's TXLine to design a 50 Ohm microstrip line*

*Video 6: How to use Microwave Office's EM Extract to setup and run EM simulation from the schematic*

**Simulate a microstrip patch antenna from the schematic (based partly on Video 5)**

In this example we are going to simulate the microstrip patch antenna that designed in Example 3.4 in Ref [1] using AWR Microwave Office. Figure 1 shows the layout of a microstrip patch antenna. The operating frequency is 3 GHz. The substrate is Duroid 5880 ( $\epsilon_r = 2.2$ ) with a thickness of 0.030 in (assume zero loss tangent and 1 oz copper board). The antenna is fed by a 50 Ohm line, and a quarter wavelength transformer is used for impedance matching. Example 3.4 in Ref [1] yields the following the dimensions:

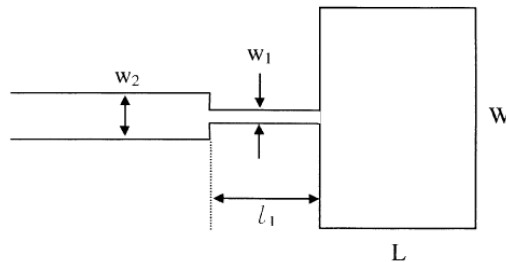
$$w_2 = .228 \text{ cm}$$

$$w_1 = .0442 \text{ cm}$$

$$l_1 = 1.90 \text{ cm}$$

$$W = 3.95 \text{ cm}$$

$$L = 3.34 \text{ cm}$$



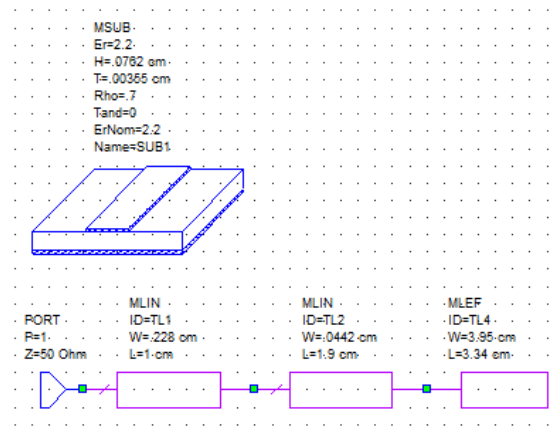
**Figure 1.** Microstrip patch antenna

**Part I. Simulate the circuit schematic**

Please follow these steps to complete part I.

1. Create a new schematic called *Schematic\_Simulation*.
2. Choose **Options > Project Options**. Under the **Frequency** tab set the frequency range to be 2.5 to 3.5 GHz with a step size of 0.001 GHz. Under the **Global Units** tab, set the project units for frequency to be GHz and length to be in cm.
3. In the schematic window, construct a model of the microstrip patch antenna using three segments of transmission lines like that shown in Figure 2. Note that the 50 Ohm line (connecting port 1 to the antenna) is arbitrarily set to be 1 cm long.
4. Choose **Project > Add Graph**. Create a rectangular graph. Right click on the graph and select **Add New Measurement**. Set the measurement to be  $S_{11}$  in dB.

- Once the measurement is setup, choose **Simulate > Analyze** to analyze the circuit. You should see a dip in  $S_{11}$  around 3GHz. Take a screenshot of  $S_{11}$  and turn this in with your assignment.



**Figure 2.** AWR Microwave office schematic of the microstrip patch antenna using MLIN model and MLEF models

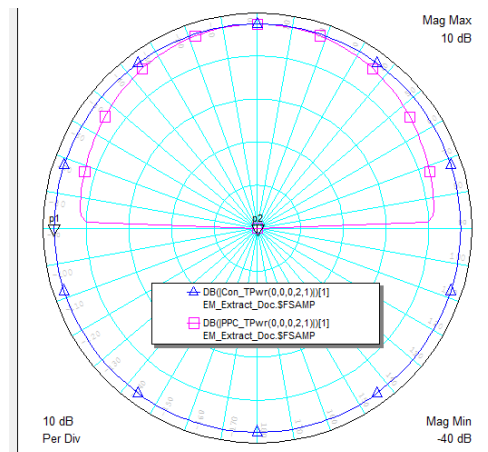
## Part II. Run an EM simulation from the schematic.

- To duplicate your schematic, right click **Circuit Schematics** in the Project Browser (located by default on the left column of the window), choose **Duplicate Schematic**. The default name will be *Schematic\_Simulation\_1*. Change the name by right clicking on **Schematic\_Simulation\_1** in the Project Browser, choose **Rename Schematic** and rename the schematic to *EM\_Simulation*. This will be the schematic that we will work on for Part 2.
- Following the basic procedure from *Video 6*, use the EM extract feature to setup an EM simulation from the schematic. Keep the *X\_cell\_size* and *X\_cell\_size* as their default values (1 cm). For your reference, here is the set of steps to accomplish this:
  - With the schematic *EM\_Simulation* active, choose **Scripts > EM > Create Stackup** and press OK. An EXTRACT block and a STACKUP block should appear in your schematic.
  - Double click on the STACKUP block in the schematic. Click on the **EM Layer Mapping** tab. You will see a list of drawing layers. The third row down is named "Copper" (it coincides with EM layer 2). Change the material from <blank> to "CU" (i.e. copper). You can do this by clicking on the blank material box in this row. Now click on the **Line Type** tab. Insert a new line by clicking on the **Insert** button. Change the material name to "CU" and make sure the EM layer is set to be 2. Delete all other lines in this window so you only have the one line you created. This procedure is presented in *Video 6*. Click OK to return to the schematic window.
  - Now right click any one of your transmission line segments in the schematic and choose **Properties**. Click on the **Model Options** tab. Check the Enable box to enable EM extraction for this element and click OK to return to the schematic window. Follow this same procedure for the other two transmission line segments to enable EM extraction for them as well.
  - Right click on the EXTRACT block in the schematic and select **Add Extraction**. A new EM structure will be created. By default, you should be now viewing the

patch antenna in the EM simulator (you will have a top down view of the antenna). If the EM simulator is not open, then double click on **EM\_Extract\_Doc** under **EM Structures** in the Project Browser to open it.

3. In the EM simulator, choose **View > View EM Layout** for a 3D view of the patch antenna. Take screenshot of a 3D view of the patch antenna and turn this in with your assignment. To go back to the regular top-down view, choose **View > View EM layout**.
4. In the EM simulator, click on the **Show 2D Mesh** in the toolbar (the tool bar is a row of buttons located just below the menu bar that provides shortcuts to frequently used commands). Take a screenshot of the mesh and turn this in with your assignment.
6. We want to compare  $S_{11}$  obtained using the schematic simulator with that obtained using the AXIEM EM simulator. To do this, choose **Project > Add Graph**. Create a rectangular graph. Right click on the graph and select **Add New Measurement**. Select *Schematic\_Simulation* as the Data Source Name and set the measurement to be  $S_{11}$  in dB and click the Apply button. Now select *EM\_Simulation* as the Data Source Name and set the measurement to be  $S_{11}$  in dB and click Apply then OK.
7. Once the measurement is setup, choose **Simulate > Analyze** to analyze the circuit. EM simulations are generally slow and this one may take a few minutes depending on your computer speed and memory. When the simulation is finished, you should now be looking at two plots of  $S_{11}$  in dB on the same graph. Take a screenshot of this graph and turn this in with your assignment. Both simulators plots should display a resonant dip around 3 GHz. However, the dip in  $S_{11}$  for the EM simulation should be much greater than the dip in  $S_{11}$  for the schematic simulation. Please comment on why this is and turn in your comments with your assignment. (hint: it has to do with radiation). For the EM simulation measurement, note that the resonant frequency is about 2.954 GHz.
8. Now you are going to plot the radiation patterns of the patch antenna at the resonant frequency 2.954 GHz. Below are the steps you can take to accomplish this:
  - a. Choose **Options > Project Options**. Under the **Frequency** tab set the frequency to be a single frequency by checking the box labeled "Single point". Set the frequency to be 2.954 GHz and click OK.
  - b. Go back to the schematic *EM\_Simulation*. Right click on the EXTRACT block and click **Properties**. In the **Frequencies** tab, check the box that says "Use project defaults" and click OK. Now right click on the EXTRACT block and choose **Add Extraction**.
  - c. Once the simulation is done, choose **Project > Add Graph**. Choose Antenna Plot (instead of Rectangular like we did above). Right click on the graph and select **Add New Measurement**. Make sure Data Source Name is set as *EM\_Extract\_Doc*. In the box labeled "Measurement Type", choose Electromagnetic → Antenna → Sweep Phi. In the box labeled "Measurement" choose "Con-TPwr" to measure the total power radiation pattern. Click Apply. Now change the "Measurement Type" to Electromagnetic → Antenna → Sweep Theta. In the box labeled "Measurement" choose "PPC\_TPwr". Click Apply. In the box labeled "Sweep Freq" click on the little box with a forward bracket ">". Choose the option "FSAMP: {2.954} GHz (n=1)." We need to do this because we have the AFS (Automatic Frequency Sweep) option on in our simulation. Click apply and then click OK.
  - d. In the Project Browser right click on **EM\_Extract\_Doc** under **EM Structures** and select **Options**. In the General tab, click on the check box labeled "Currents" to save the simulated currents to the document and click OK. If a warning dialog box appears, click OK.

- e. In the Project Browser right click on **EM\_Extract\_Doc** under **EM Structures**. Select **Force Re-Simulation**. If a warning dialog box appears, click OK.
  - f. Now simulate again by choosing **Simulate > Analyze**. The simulation should be pretty fast since you are only simulating one frequency point. The plot of your radiation pattern should look like that shown in Figure 3.
9. Turn in a plot of your radiation patterns with your assignment. What is the half power (3dB) beamwidth in degrees (for theta)?
10. Now you will be plotting the current distribution on the antenna at 2.954 GHz. To do this:
- a. Right click on **EM\_Extract\_Doc** under **EM Structures** in the Project Browser. Choose **Add Annotation**. Make sure the measurement type is selected as Planar EM from the options in the box labeled "Measurement Type". Now select EM\_CURRENT from the options in the box labeled "Measurement". . In the box labeled "Sweep Freq" in the bottom right, click on the little box with a forward bracket ">". Choose the option "FSAMP: {2.954} GHz (n=1)." Press OK.
  - b. Right click again on **EM\_Extract\_Doc** under **EM Structures** in the Project Browser. Choose **Options**. In the tab labeled **General**, check the box labeled "Currents" and click OK. A warning dialog box may show, click OK again.
  - c. Right click again on **EM\_Extract\_Doc** under **EM Structures** in the Project Browser. Choose **Force Re-simulation**.
  - d. Now simulate again by choosing **Simulate > Analyze**.
  - e. After the simulation is complete, right click again on **EM\_Extract\_Doc** under **EM Structures** in the Project Browser. Choose **View EN 3D Layout**. In the upper right of the screen in the toolbar you should see a forward arrow (the animate play button). Press this button to animate the current. You should see the standing wave pattern on the patch antenna. Pause the animation when the standing wave is most prominent, take a screenshot, and turn in this screenshot with your assignment.



**Figure 3.** Radiation pattern for microstrip patch antenna

## References

- [1] K. Chang, RF and Microwave Wireless Systems, Wiley, NY, 2000.