

## *Wireless Circuits and Systems Laboratory*

### Procedure #6 Circuit Design Using Microstrip Lines in ADS

This procedure relies on information contained in Procedures #1 and #5. The exercise outlines the approach for creating an ADS circuit schematic that uses microstrip transmission line elements as part of the design.

#### 1. Creating a Circuit Design with Microstrip

1. Open your project directory so that you have a fresh schematic window. Click on File>Save As.. and give the schematic a name such as msdemo
2. Click on the Component Palette list pull-down menu and select **Tlines – Microstrip**. At this point you should see a collection of buttons along the left-hand side of the window, each of which represent a different element from the microstrip library.
3. The first ‘element’ we will place in the schematic is a substrate definition. Click on the **MSUB** button at the top, left-hand side of the element list, and place a copy of the element into the schematic. Double-click on the element and enter the following settings: **H** (substrate thickness) = 1.57 mm, **Er** (dielectric constant) = 4.3, **Mur** (relative permeability) = 1, **Cond** (metal conductivity) = 5.8e7, **Hu** (upper shield height) = 1e33 mm, **T** (conductor thickness) = 0.043 mm, **TanD** (dielectric loss tangent) = 0.022, **Rough** (conductor surface roughness) = 0 mm. Click OK. [Note: from this point forward, you can associate microstrip line elements with this substrate, such that the parameters entered here will be applied to the microstrip line. Although it is possible to have multiple MSUBs in a given circuit (each MSUB has its own ID), you will generally use only one substrate in a circuit design.]
4. Now we can proceed to add a microstrip transmission line element. Scroll down the list of buttons on the left-hand side of the schematic window and select the MLIN (microstrip line element) and place one copy into your schematic. Double-click on the element and enter the following parameter values: **Subst** (substrate definition) = Msub1, **W** (line width) = 1.38 mm, **L** (line length) = 21.4 mm. The remaining parameters (Wall1, Wall2 and Temp) do not have to be set.
5. Now let’s add a discrete resistor to our design. Click on the Component Palette list pull-down menu and select Lumped-Components. Click on the **R** (resistor) button on the left-hand side of the window and place a copy of it into your schematic---to the right of the microstrip line. Double-click on the element and change the resistance to **112 Ohms**. You do not need to adjust the other parameters. Save your design.
6. As the last step, we will add an RF short circuit (i.e., a quarter-wavelength open circuit microstrip line) to the right-hand side of the resistor. Repeat the procedure in steps 2 and 4 above, but this time choose the **MLEF** button (Microstrip Line Open-End Effect) instead of MLIN. Set **W**=6.49 mm, **L**=19.93 mm and assign **Subst** = MSub1 as before. [NOTE: The open-end effect models the effects of the fringing fields that extend beyond the physical limit of the end of the microstrip line. The fringing fields increase the effective **electrical** length of the line.]
7. Now add a termination to this 1-port circuit so that we can simulate the design. Click on the Component Palette list pull-down menu and select **Simulation-S\_Param**. Click on the **Term** button on the left-hand side of the schematic window and place a copy of it into the schematic, to the left of the first microstrip line. The **Z** (reference impedance) parameter should be set to 50 Ohms.
8. To complete the schematic, attach a ground to the bottom of the Term element and use the ‘wire’ tool to connect all the elements in the schematic together.

- Finally, to enable an S-parameter simulation, click on the **SP** button (S\_Param: S-parameter Simulation) on the left-hand side of the window and place a copy into your schematic. Set the start frequency to **0.1 GHz**, the stop frequency to **5 GHz**, and the step size to **0.1 GHz**.

At this point, your circuit design should look as follows:

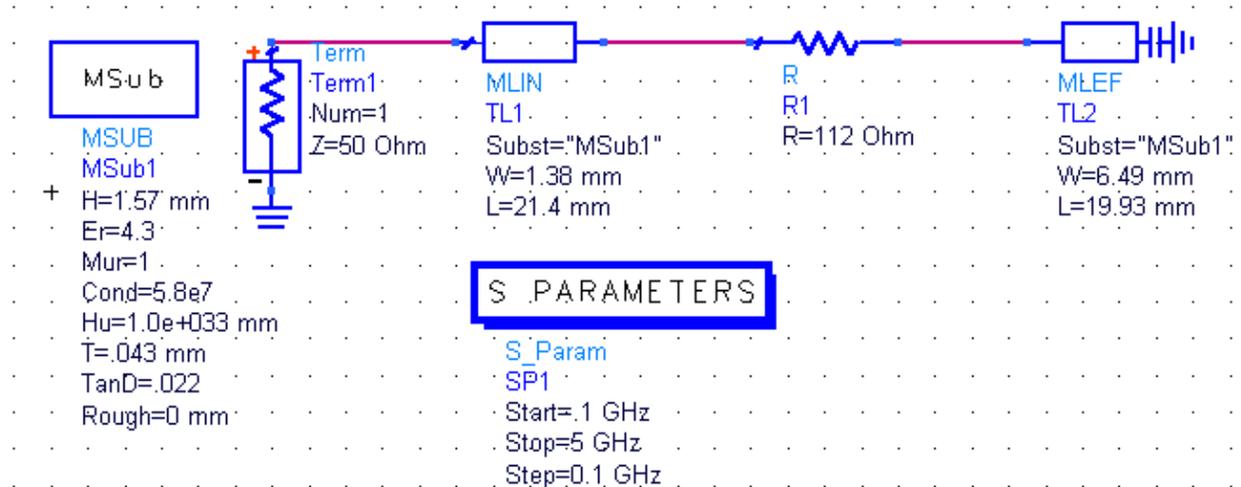


Figure 1. ADS circuit schematic using microstrip lines (and a discrete resistor).

## S-Parameter Simulation

The procedure for testing the 1-port microstrip circuit design is essentially the same as the procedure outlined in Procedure #1.

- Click on the **Simulate** button, which is located five buttons to the right of the Library (books) icon.
- Click on the New Data Display Window button, which is seven buttons to the right of the Library (books) icon. This will cause a graphics window to pop up.
- Choose the Rectangular Plot, which is represented by the first button below the large white arrow, on the left-hand side of the window. Click on this button and drag the rectangle into the window, and click.
- To generate a plot of S11 versus frequency, click on this entry under the Data Sets and Equations list, and click the Add button in the middle of the window. Select the dB format. Now click on OK to generate the graph.
- You can also double-click on the rectangular plot, and add S11 phase to the graph. The results are shown below:

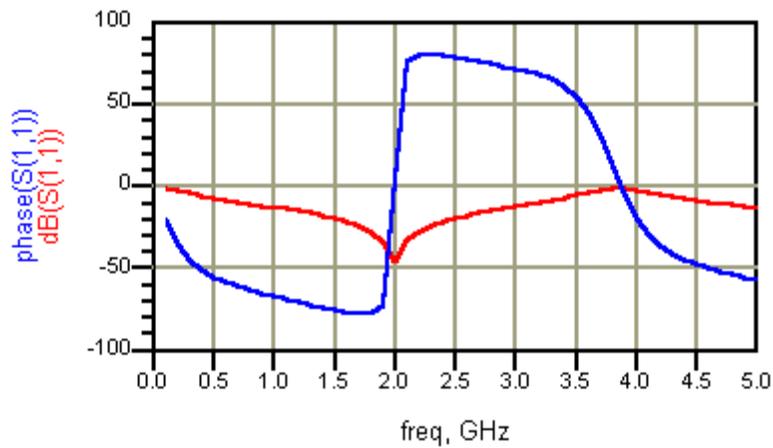


Figure 2. Simulated  $S_{11}$  (magnitude and phase) for the quarter-wave impedance matching circuit shown in Figure 1.

It should be evident at this time that the schematic in Figure 1 represents a quarter-wavelength impedance matching circuit that is designed to operate at 2 GHz. Notice that  $S_{11}$  reaches at minimum value of approximately  $-50$  dB at this frequency. Also, the phase of  $S_{11}$  passes through 0 degrees at 2 GHz, indicating that the input impedance is purely real (and equal to 50 Ohms).