This tutorial will serve as an introduction to the Ansoft Designer Microwave CAD package by stepping through a simple design problem. Please note that there is a link to the Ansoft Designer Student Version download on the ece584 home page.

The problem is to match a 25 ohm resistor to a 50 ohm transmission line using a ¼ wave transformer at 6.0 GHz.

Transformer Impedance: 35.35 Ohm

1: Start Ansoft Designer SV. You should get the following window:
2. Click on Project and select “Insert Circuit Design” as shown above. The following pop up window should be displayed:
3. Select MS-RT Duroid 0.020 inch, 0.5 oz copper and click on **Open**. The screen should now have a project window, as seen below.
4. An interface port and a ground must be added to the schematic. Both of these parts are available in the Draw menu. Select the part from the Draw menu and then click inside the schematic window to add the part. After placing the component, press the Esc (escape) key to end the placement process for this component.

5. After adding the interface port and ground, the transmission line and resistor must be added. Both of these parts are available under the “Components” tab of the left hand three pane window. Select “Resistor” under the “Lumped – Resistors” portion of the component tree. Either double click or drag “Resistor” to place it in the schematic area. Press Esc when done with placement.
6. To add the quarter-wave transformer, select the “Microstrip” section of the Components tree, then select “Transmission Lines” and “Microstrip Transmission Line, Electrical Length” and drag the icon into the schematic window.
7. Connect the various parts by placing a wire between the interface port and the transmission line, the transmission line and the resistor and the resistor and the ground. The “Wire” is in the “Draw” menu. Place the wire using the mouse by clicking on the components to be connected. Alternately, components may be connected by dragging one of the components with the mouse so that the components ends overlap.

8. The parameters of the transmission line and resistor must be modified. Right click on the resistor and select properties. A new window will open. In the “Value” column of the “R” row type your resistance (in this case 25). Select OK.

9. Right click on the transmission line and select properties to modify the transmission line. In the row labeled “TRL”, press the gray bar labeled “TRL” (in the Value column) to bring up a new window labeled “Microstrip single”. In the “Electrical” box, enter the Z value, where Z is line impedance. For a quarter wave transformer, this value should be the geometric mean of the input impedance (50 ohms) and the load impedance (25 ohms) or Z=35.36 ohms. The E value is the electrical length of the line in degrees, which for a quarter wave line is 90 degrees. The Frequency value is the operating frequency or 6 (GHz is the default unit for Frequency). Press the Synthesis button to design a microstrip
line with the proper length and width to meet the design criteria you just entered. Press OK to exit this window and OK to exit the properties window.
This completes the drawing of the schematic. The circuit should appear as in the panel below.

10: Next the analysis of the circuit must be completed. Select “Add Analysis Setup” from the “Circuit” menu. A new window will open. We don’t need to change any of the options so select next.
In the sweep variables, highlight F and select the “Edit Button”.

11: Use 3 GHz, 8 GHz, and 10 MHz, for the Start, Stop, and Step frequencies. (Note this should be a linear sweep). Click Add and then OK.
12. The Sweep / Value entry in the Linear Network Analysis, Frequency Domain should change to LIN 3GHz 8 GHz 10 MHz. Select Finish in the Linear Network Analysis, Frequency Domain window. You should end up back in the main program window.

11: Select “Analyze” from the “Circuit” menu to analyze the circuit. Note that the program does not give any feed back that the analysis was completed properly other than a “Project 1 (C:\Program …” entry in the bottom left hand sub window.

13: Select “Create Report” from the “Circuit” menu to plot the response. In the “Create Report” dialog select OK

14. In the Traces dialog, select Return Loss as the Category, RTL1 as the Quantity, and dB as the Function. Click “Add Trace” to add this plot to the graph. Then “Done” to plot the graph (See graphics next page).
If you wish to make changes to the circuit, go to the Circuit drop down menu, and click on “Schematic Editor” to bring up the circuit diagram.