

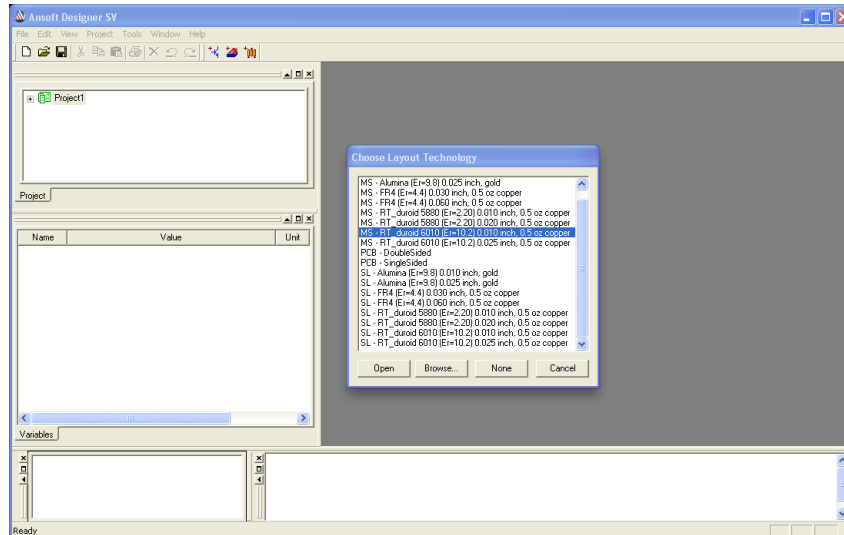
Instructions for using Ansoft Designer SV (2009)
Written by Paul Siqueira
University of Massachusetts, Amherst

1.) Download the software from the site:

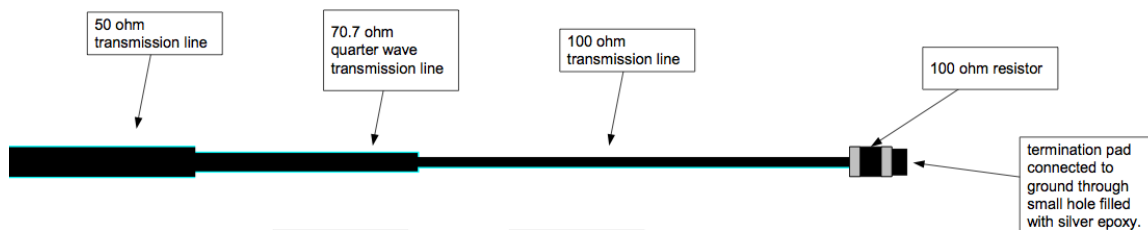
<http://www.ansoft.com/ansoftdesignersv/>

... or, find a computer that already has this loaded.

2.) Begin a project and choose under 'Project', 'Insert Circuit Design'. A window will appear, where it will ask you for the layout technology. Choose MS- RT_duroid 5880 (Er=2.2) 0.020 inch, 0.05 oz copper. This indicates 'Microstrip (MS), Rogers Duroid material 5880 (with a dielectric of 2.2), and 0.010 inches thick (2.54 mm).

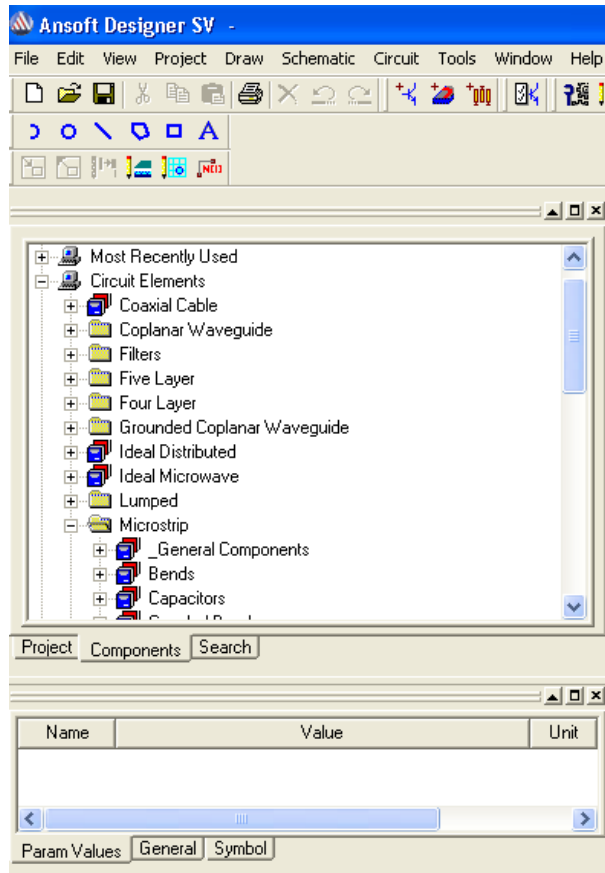


3.) Next, we want to layout $\frac{1}{4} \lambda$ lengths of transmission line, as on page 23 of the lab manual. Where the center frequency is half-way between 2 and 4 GHz or 3 GHz.

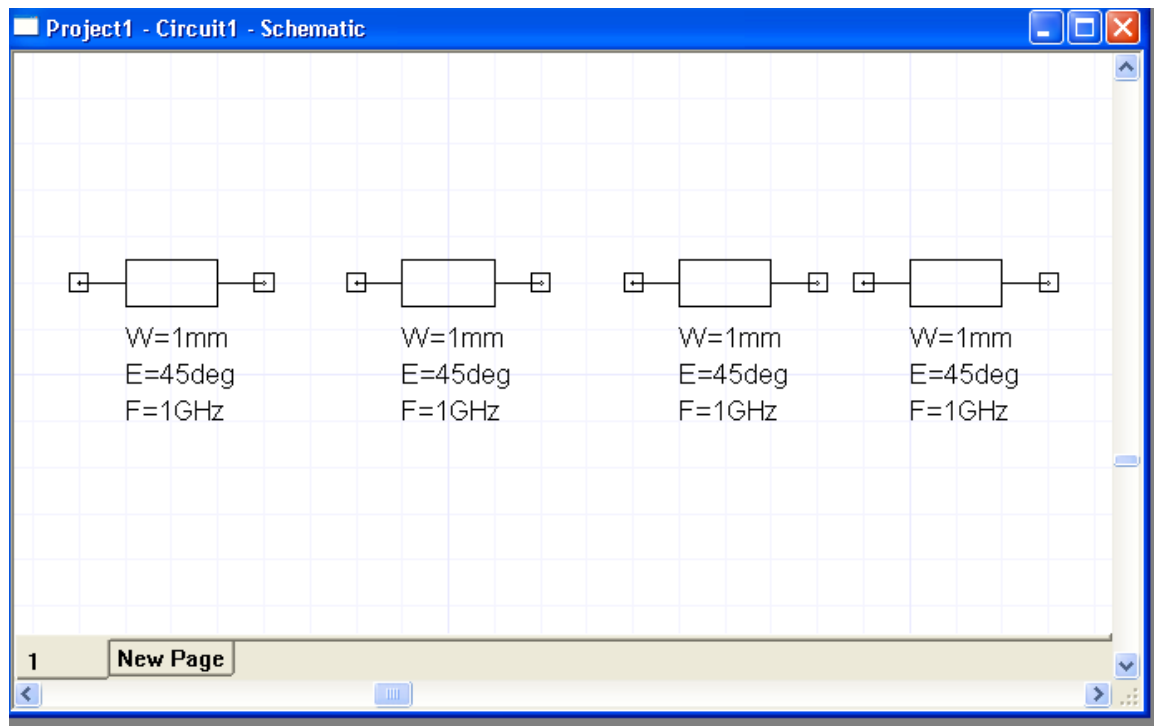


To get these transmission lines in the form of microstrip, we go to the components tab at the left window of the Designer workspace. Under the circuit elements folder, you will find another folder labeled Microstrip. At the bottom of the list of

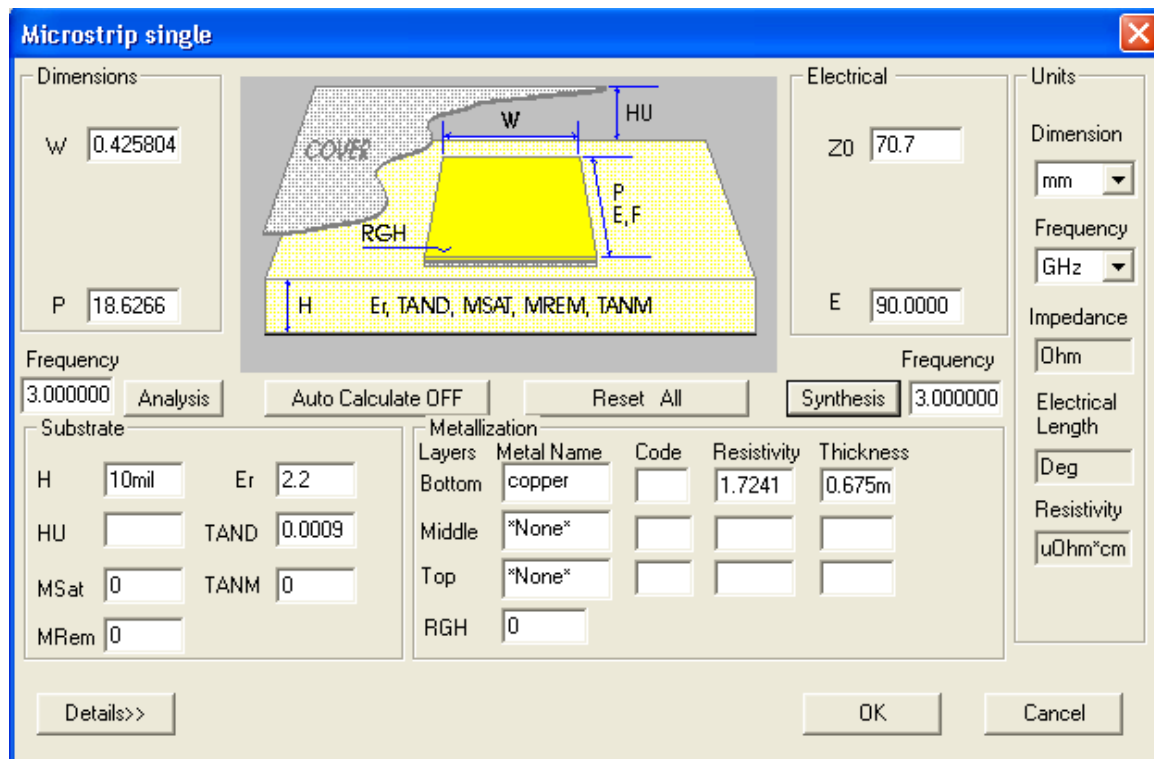
microstrip components, you will find an entry called 'Transmission Lines'. Within that entry, you will see something labeled 'MSTRLE: MS Trans. Line, Electrical Length', which specifies a microstrip line of a given electrical length.



Choose this with the mouse and drag it over to the schematic on the right. With every click, you can create a new version. Lay down four of these, and press the escape key when you are done. Displayed are their widths (W, in mm), and their electrical lengths (E, in deg). These can be joined together by moving so that their contact points overlap. Once combined, their values may be changed



To get a specific impedance, you need to use the TRL calculator, which can be found when you select a component, and look at the middle window on the left side of the screen. When you click on 'TRL', you will get the following window:



On the right hand side of that window, you will see the electrical characteristics of the line. You can enter the impedance under Z0, the electrical length (in degrees; 90 degrees is equal to 1/4) under E, and 3.0 under Frequency. When you click on the 'Synthesis' button, the physical dimensions of the component will be computed. When you click 'ok', this will be entered into the circuit diagram. Don't forget to click the 'Synthesis' button first!

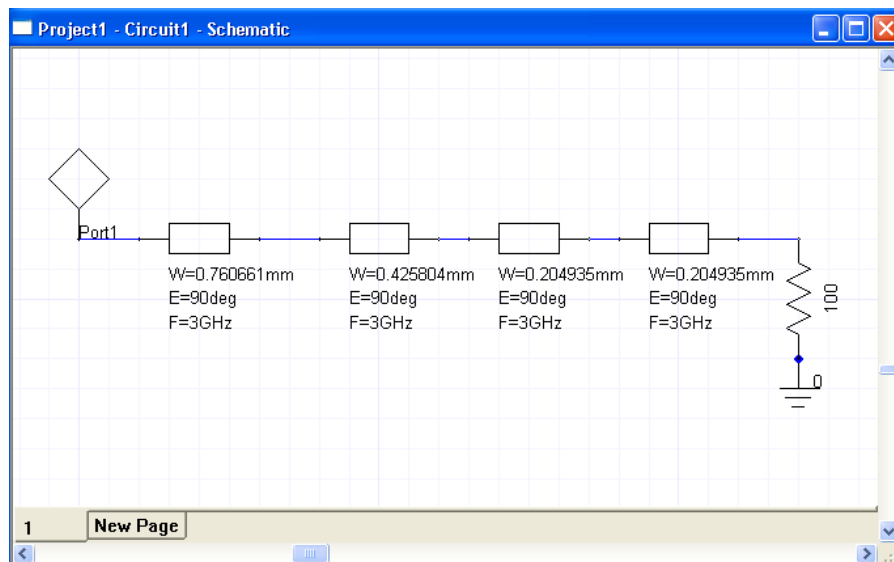
Do this for all transmission line components in the schematic. Note, for the simulation being done here, there are two 100 ohm components on the right hand side, stacked one after the other.

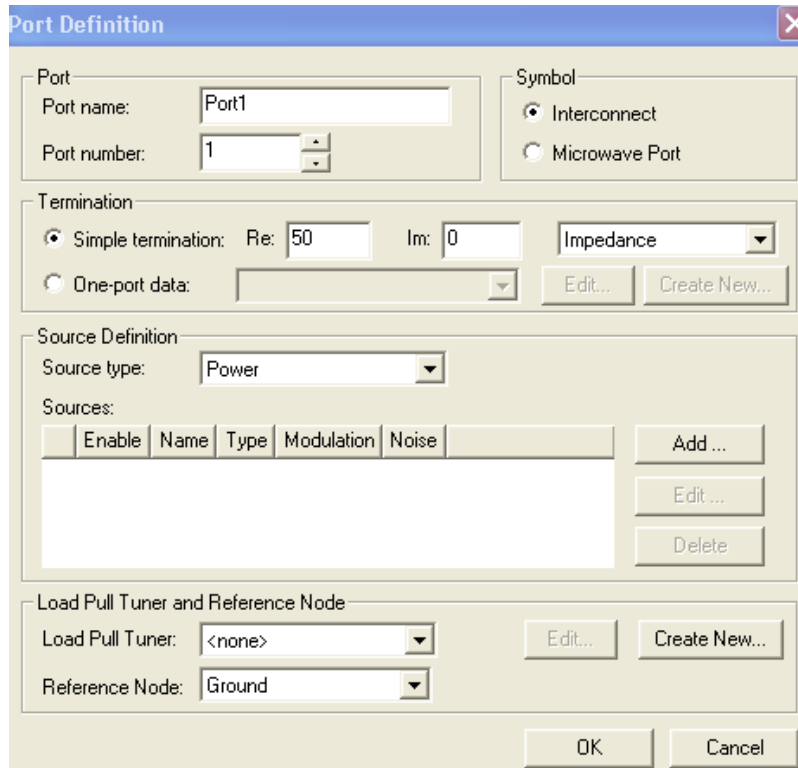
Now, you need to add a 100 ohm termination resistor to the back end. You do this by going back to the "Circuit Elements" menu under the "Components" tab, and choose 'Lumped'. You will see a number of choices underneath... and you will want the one that says 'RES', for Resistor. Click and drag this onto the right hand side of the circuit. Use the escape key to get control of the mouse back. You can rotate the resistor by selecting it and typing 'Control-R', or using the menu item under the 'Draw' menu.

Now, just add a port to the front part, and a termination to the back, and you will be ready for the next step. This is done as follows.

Click on the 'Draw' Menu, and choose a ground symbol. Drag this onto the schematic, and place it where desired. Use the escape key to get control of your mouse back. Go back to the 'Draw' menu, and choose 'Interface Port', and put this onto the left-most node of the circuit. If you 'Edit' the port, you will see that you can change its impedance... which should default to 50 ohms... no need to change it.

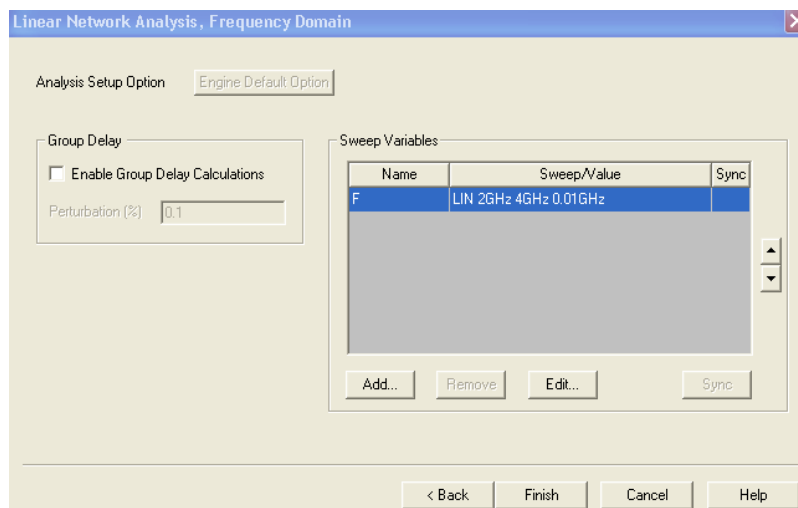
Once done, your schematic should look like:





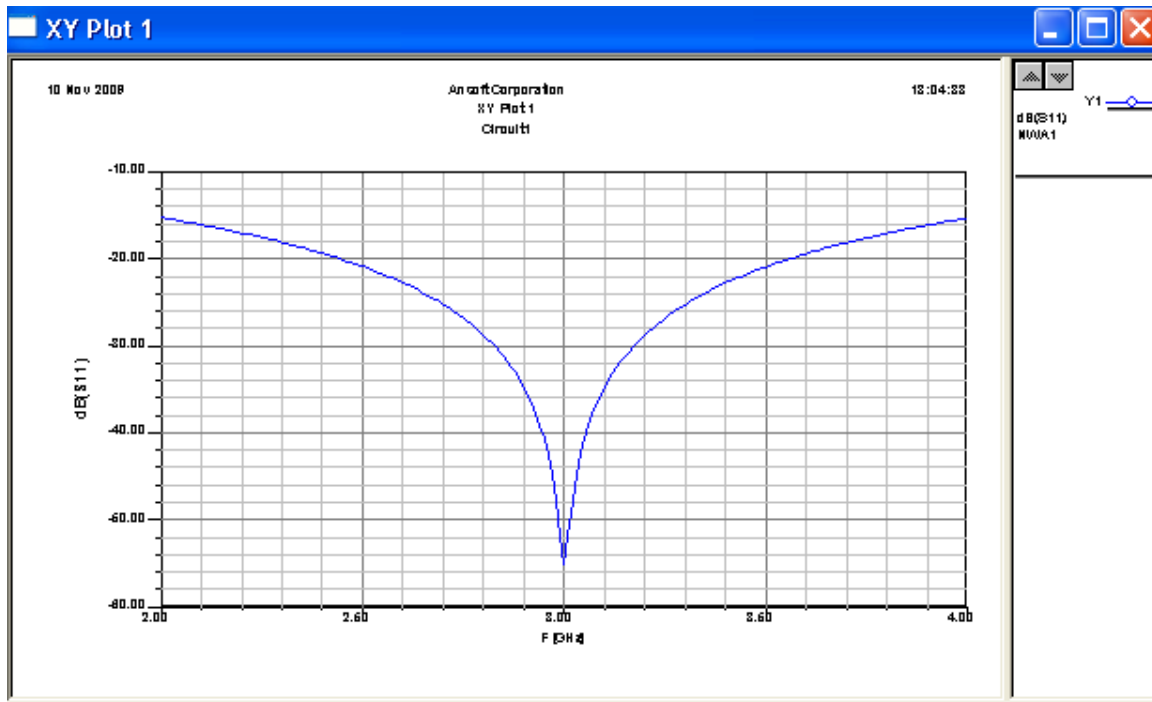
Now, you are ready to do a simulation.

Under the 'Circuit' menu, you will see something that says 'Add Simulation Setup'... choose this, and use the defaults on the first window. On the next window, use the 'Add' button to do a linear sweep on the frequency variable... beginning at 2 GHz, and going to 4 GHz, in steps of 0.01 GHz. Use the 'Add >>' button to enter these into the 'Sweep Values' window, and press 'Ok'. You should now see the sweep details in the Simulation Setup window for Linear Network Analysis... as follows:



You can now click on 'Finish', and under the 'Circuit' menu, click 'Analyze'.

Next, you want to display the results. To do this, click on the 'Create Report' option under the 'Circuit' menu, and after accepting the defaults on the first menu, choose, S11 in dB as the trace to display. Once you click on Done, you will see the desired result!



Some things to try: Add a second circuit using a different matching network... like that shown in the lab with four impedances instead of three. If you put this on the same diagram (using cut and paste), you can add a second port, and display the reflection parameters for both on the same plot.

Play with the different options so that you get some familiarity with the software. Enjoy!

