INTRODUCTION TO MICROSTRIP TRANSMISSION LINES AND COMPUTER-AIDED DESIGN

Lab #6

This lab, which will lay the groundwork for the next several labs on design and construction of a microstrip-based microwave circuits, will serve as an introduction to microstrip transmission lines and the concept and effects of transmission line discontinuities. Over the course of the next several lab sessions, you will design, build, and test an amplifier, mixer, or oscillator for signals above 1 GHz. The basic structure of the next several lab sessions includes a study of the microstrip transmission lines to be used and the active device to supply the amplification (Agilent AT-41485, a microwave bipolar junction transistor), the design of appropriate bias and matching networks, and finally full characterization of the circuit you have designed, including its gain, frequency response, input and output reflection coefficients or impedances, noise performance, and large-signal (intermodulation) performance. In this lab session, we'll focus on the measurement and assessment of the performance of microstrip line for your circuit.

Microstrip Line Characterization

In this subsection of the lab, measured data from a test microstrip line will be taken using the network analyzer. This data will be used in subsequent sections to obtain the behavioral parameters of the line (e.g. characteristic impedance, loss) and from this the parameters of the substrate material (e.g. permittivity, conductivity, loss tangent, etc.) can be determined. These substrate parameters will be used in your circuit designs in later labs, as the same substrate material will be used.

1.) To automate the transfer of measurement data from the analyzer to the CAD software, we'll use the IC-CAP software package. IC-CAP can be started by typing "iccap" from the command line (you'll need to run "source ee40458.csh" (for csh users) or "source ee40458.sh" (for bash or sh users) first as in previous labs to set up your path and environment variables). Since this is (likely) the first time you've run IC-CAP, you need to set up the connection between the software and the measurement hardware. Click on the "GPIB" icon in the toolbar (looks like an oscilloscope with an arrow). In the window that appears, click "Add Interface" and type "lan[cush116L488.ee.nd.edu]:gpib0" in the box. Then click "Rebuild" and the interface bus will be scanned for active instruments. When this completes, close the hardware setup window. To read the s-parameters from the analyzer, a pre-configured measurement setup is available on the course web page (lab6.mdl). Load this into IC-CAP (under the "File" menu), and click on the "microstrip" device (icon looks like a FET). Select "microstrip_s" in the tree in the left-hand window pane.

2.) To set up the network analyzer to measure the s-parameters of the test microstrip line from 300 MHz to 6 GHz with 201 data points, you enter the start and stop frequencies and number of points in the "freq" block on the Measure tab. Set the IF bandwidth (under the "Instrument Options" tab) to 100 Hz. On the Measure tab, click "Calibrate." This will download the measurement setup into the analyzer and prepare it for calibration (but will not automatically perform the calibration). Perform a <u>full 2-port</u> calibration of the network analyzer, using the precision 3.5 mm calibration kit and male connectors on both network analyzer ports. Note: you will need to push the "local" button on the analyzer to enable the front panel. Be sure to select the "SMASOLT" user kit under the "calibration" menu of the network analyzer; if selecting "user kit" does not result in the SMASOLT kit being displayed, contact the instructor for help. After calibration, save (or re-save) the calibration, and *make a note of the register used*. **Enter this register number in IC-CAP under the Instrument Options tab** (in the "Cal Set No [1..33]" field).

3.) Measure the s-parameters of the microstrip test line (this board consists of a single microstrip line with 3.5 mm (sma) connectors on both ends; this is not the microstrip stub board we've used in previous labs). To measure and collect the data, connect the board to the network analzyer, and click "Measure" on the Measure tab. When complete, save the file (to a .mdl file) for importing into ADS for analysis and modeling, being sure to note the filename used. To check that the measurement was successful and that the data is reasonable, click the display plots icon in the toolbar (looks like 3 IV plots). This will bring up plots of the measured S_{11}, S_{12}, S_{21}, and S_{22} (magnitude and phase) for inspection. If satisfied with the data, close IC-CAP.

4.) Measure and record the length and width of the microstrip as accurately as possible (use the dial calipers for this measurement); these dimensions are needed for computer simulations of the microstrip line. When making this measurement, be careful not to scratch the surface of the microstrip line or to have the calipers gouge the edge of the line.

Microstrip Line Modeling

In this portion of the lab, a model of the microstrip test line will be built and simulated to compare with the actual measured results. The real-world effects of discontinuities will be examined, and a simple approach for modeling these effects will be implemented.

1.) First, import the measured data into the ADS CAD software. ADS can be started by typing "ads" at the command line. Make a new project, and click on the data file tool icon (looks a little like a floppy disk with an arrow and squiggle) on the toolbar. Choose the input file format to be ICCAP, and browse for the .mdl file you saved the measurement data in. Enter a dataset name to hold this data within ADS in the lower part of the window, and click "Read File." If all goes well, a status box will indicate successful read-in of the data. Close the data file tool window.

2.) To start modeling the microstrip line, examine the expected s-parameter results for a microstrip board. To do this, build a circuit in ADS similar to the one shown in Figure 1. The detailed steps for this process are outlined below:

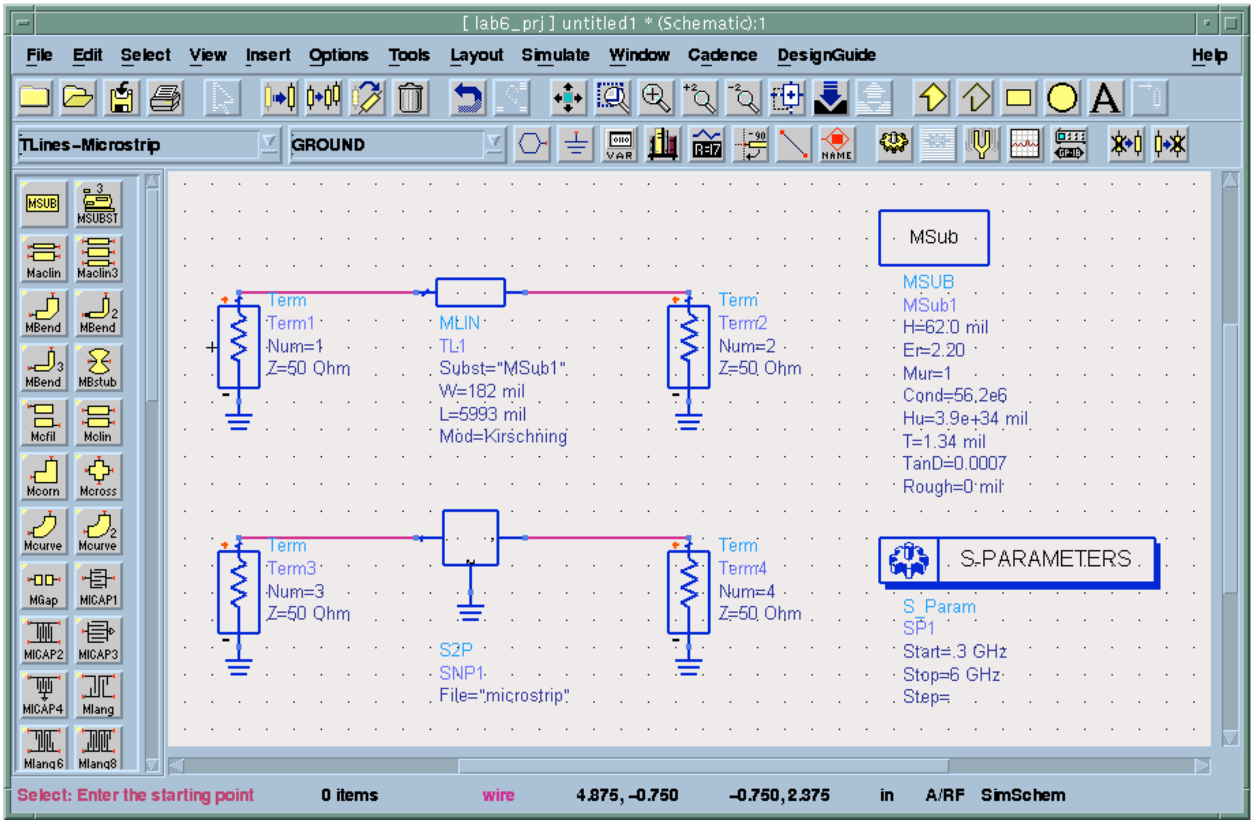

Figure 1. Schematic for transmission line model and comparison with measurement.

a. At the "component palette list" pull down menu, select "TLines – Microstrip." This will change the palette on the left-hand side of the schematic window to include a number of microstrip components.
b. Since the microstrip test line consists of a single section of microstrip line, select the MLIN element from the palette.
c. Place one of these elements in the schematic window by clicking; the component will be placed where the outline is shown in the schematic window. Change the L and W parameters to match the measured values for the width and length of the physical board tested. Remember that 1 mil = 0.001 inch.
d. To provide the simulator with information about the microstrip substrate, add an "MSUB" element to the schematic page.
e. To set up the physical parameters of the microstrip substrate, double-click on the MSUB element to bring up a parameter editing box. Change the parameters to the nominal values from the manufacturer's data sheet shown below:

H = 62 mil	thickness of substrate
Er = 2.20	relative permittivity
Mur = 1	relative permeability
Cond = 56.2e6	Conductivity
Hu = 3.9e34 mil	cover height – use large value to neglect cover effects
T = 1.34 mil	conductor thickness
TanD = 0.0007	dielectric loss tangent

Note: 1 mil = 0.001 inch.

f. Since we're interested in the s-parameters of this circuit, we need to add the appropriate simulation elements to the circuit. In the "component palette list" menu, select "Simulation – S_Param." Add two "term" elements to the circuit; these are equivalent to the test ports on a network analyzer.

g. To control the simulation, add a "SP" element to the schematic page as well; this sets the frequency range and other simulation options.

h. All of the required elements are now added to the schematic; what remains is to wire them together and set the physical dimensions and parameters for the circuit. To wire the circuit, click on the "wire" icon (or press control-w), and click on successive pairs of nodes to connect them.

i. Add grounds to the - terminals on both of the "term" elements by clicking on the ground icon below the menu bar and placing one on each term.

j. Finally, to set up the simulation, double-click on the "S parameters" item. Set the start frequency to 300 MHz, the stop frequency to 6 GHz, and the number of points to 201 (to match the measurements that were taken). This completes the design entry for a microstrip line for s-parameter simulation.

k. To facilitate comparison of the measured data and the simulation results, it is helpful to "import" the measured data into the simulation. This is done indirectly by adding an additional circuit to the schematic page. Add two more "term" elements to the schematic page, and place an S2P element from the "data items" component palette between them, wiring them together. Be sure that term 3 is connected to port 1 of the S2P, and that term 4 is connected to port 2 of the S2P. Ground the "ref" terminal of the S2P element. To select the data that the S2P element should import, double-click on the S2P element and enter the filename and file type (this will need to be changed from "Touchstone" to "dataset" in this case) for the measured microstrip test line data. In simulation, the s-parameters measured between ports 3 and 4 will be the measured data obtained previously from the test line measurements.

2.) To run the simulation, click on the "gear" icon (or press F7); a simulation status window should appear and indicate when the simulation has completed. A blank graph window will also pop up when the simulation is complete.

3.) Plot the results of this simulation. Note that the schematic page has four test ports, so the simulator calculated a 16-element matrix of s-parameters at each simulation frequency. Of these, the only physically meaningful elements are S_{11}, S_{12}, S_{21}, S_{22}, S_{33}, S_{34}, S_{43}, and S_{44}. The others are all cross-terms between the two separate circuits, and will all be zero. Note that S_{33} is the measured data for the line that is simulated as S_{11}, that S_{43} is the measured data for the simulation of S_{21}, and so on. How well does the simulation and the measurement match? Are there any clear similarities or clear discrepancies? Can you detect any patterns in the results?

Except under unusual circumstances, the measured data and the simulation data generally won't match well on the first try. This can be due to several physical effects: i) not including all of the circuit elements in the simulation, ii) not modeling, or modeling incorrectly, the effects of discontinuities, and iii) inaccurate substrate parameter values. Each of these potential problems can be fixed by increasing the sophistication (and complexity!) of the model.

4.) In the simple model used so far, the connectors on the board and the discontinuity between the coaxial line in the cable and connector and the microstrip on the board have not been accounted for. Also, the nominal dimensions and substrate parameters have been assumed to be exactly correct, even though variation there is measurement uncertainty and variation in the parameters.

a. To include the effects of the finite length of the connectors, insert a length of ideal transmission line (TLIN) between the test ports and the microstrip line. This line can be found in the “TLines – Ideal” component palette. Set the reference frequency to 1 GHz; any frequency could be chosen, but this is a convenient choice.
b. To account for the discontinuity between the coax connector and the microstrip board, insert an inductor (found in the "Lumped – Components" palette) between each of the TLIN elements and the microstrip line. At modest frequencies, this is a reasonably accurate model to account for the transition of the electric and magnetic fields from the cylindrical field distribution in the cable and connector to the quasi-planar distribution in the microstrip line.
c. Since it is difficult to determine with sufficient accuracy the electrical length of the connector or the inductance of the discontinuity from first principles, the optimization capabilities of ADS can be used to find these values. To permit this, insert a VAR element (by choosing the VAR icon next to the ground icon on the toolbar) in the schematic. Insert two variables in the VAR element—one for the electrical length of the connector and one for the inductance of the discontinuity. Choose descriptive names for these variables; generic names like “L” are used internally by the simulator and will generate confusing error messages. To start with, select “reasonable” estimates of the electrical length and inductance; 20 degrees of electrical length (at 1 GHz) and 0.5 nH of inductance is a decent starting point. Insert the variable names you chose for the electrical length of the connectors and inductance of the discontinuities in the schematic elements. Be careful about the units; if your VAR element contains units then the schematic element should not, and visa-versa.
d. To account for possible measurement uncertainty in the line dimensions and parameter variation in the substrate, add additional variables to the VAR elements. Entries for the permittivity (Er) of the line, the conductivity of the line (Cond), microstrip line length, and microstrip width should be added. Set these initial values to those used in the table previously, and modify the schematic elements to use the VAR element variables instead of constants. As with the electrical length and discontinuity inductance, be sure to choose descriptive names to avoid name conflicts.
e. To set up the optimization, limits on the variation of all of the parameters must be established. To set the limits for the variation, open the VAR element, and select “Optimization/Statistics Setup.” In this window, set the “Optimization Status” to “enabled,” and enter appropriate values for the minimum and maximum values. Be sure that the nominal value lies between these two limits, and be consistent with the units. Reasonable bounds on the electrical length is something like 12 to 35 degrees, for the discontinuity inductance is from 0.1 to 0.7 nH, for the permittivity is 2.1 to 2.3, for the conductivity is from 1e5 to 56e6, for the length is approximately ±0.1 inch (100 mils) and for the width is approximately ±4 mils.
f. The final step in setting up the optimization process is to give the simulator the criteria for a good solution – it needs to be told what the best solution would look like so that it can determine if a variation in the parameters has improve the situation or made it worse. To do this, add two “GOAL” elements from the “Optim/Stat/Yield” palette. In each goal

element, a mathematical expression that indicates the “quality” of the solution must be provided; the simulator will attempt to make this expression evaluate to a value within the “acceptable” range. Since the goal of this optimization is to make the simulation model match the measured s-parameters, two expressions that could be used are mag(S11-S33) and mag(S21-S43). In both cases, a perfect match between the model and the measurement would be signaled by these expressions evaluating to zero; any imperfections in the match will cause these expressions to evaluate to a positive value. Enter one of these expressions in each of the goal elements, and select a minimum value of 0 and a maximum value of 0.001 for the range of acceptable solutions. Set the SimInstanceName to match the s-parameter simulation schematic entry (probably SP1).

g. Add an OPTIM element to the schematic to control the optimization process. Add the two goals to the OptGoal list, set the optimization type to gradient, and set the number of iterations to 100. Figure 2 depicts a schematic ready for optimization. To perform the optimization, run the simulation (using the “gear” icon). The simulation window will monitor the progress of the optimization (it may take a few minutes). The “CurrentEF” variable gives a relative measure of how well the goal expressions are being achieved; if all goes well this number should drop as the optimization proceeds. The final values that the optimization process found are printed in the simulation window at the end of the optimization. Check these values over to make sure they appear to be reasonable values. If they are, under the Simulate menu select “Update Optimization Values.” This will put the values found in the optimization into the schematic automatically. To verify that the optimization worked, disable the OPTIM element (select it and then click on the “X-out” icon), and run the simulation again.
h. Using the same data display window as previously, how does the simulation and measurement data compare now? Do they match better? Are there any discrepancies or points of significant disagreement between the two? If so, it may be necessary to optimize again, either with different starting values or with different bounds in order to find a better solution.
i. Once acceptable agreement between the measurements and model have been achieved, examine the parameters and dimensions obtained by the optimizer. Do these seem reasonable? Make note of the parameters and dimensions obtained; these will be used for the design of the project design. Print out the comparison of the measured and modeled microstrip test line and include it in your lab notebook.

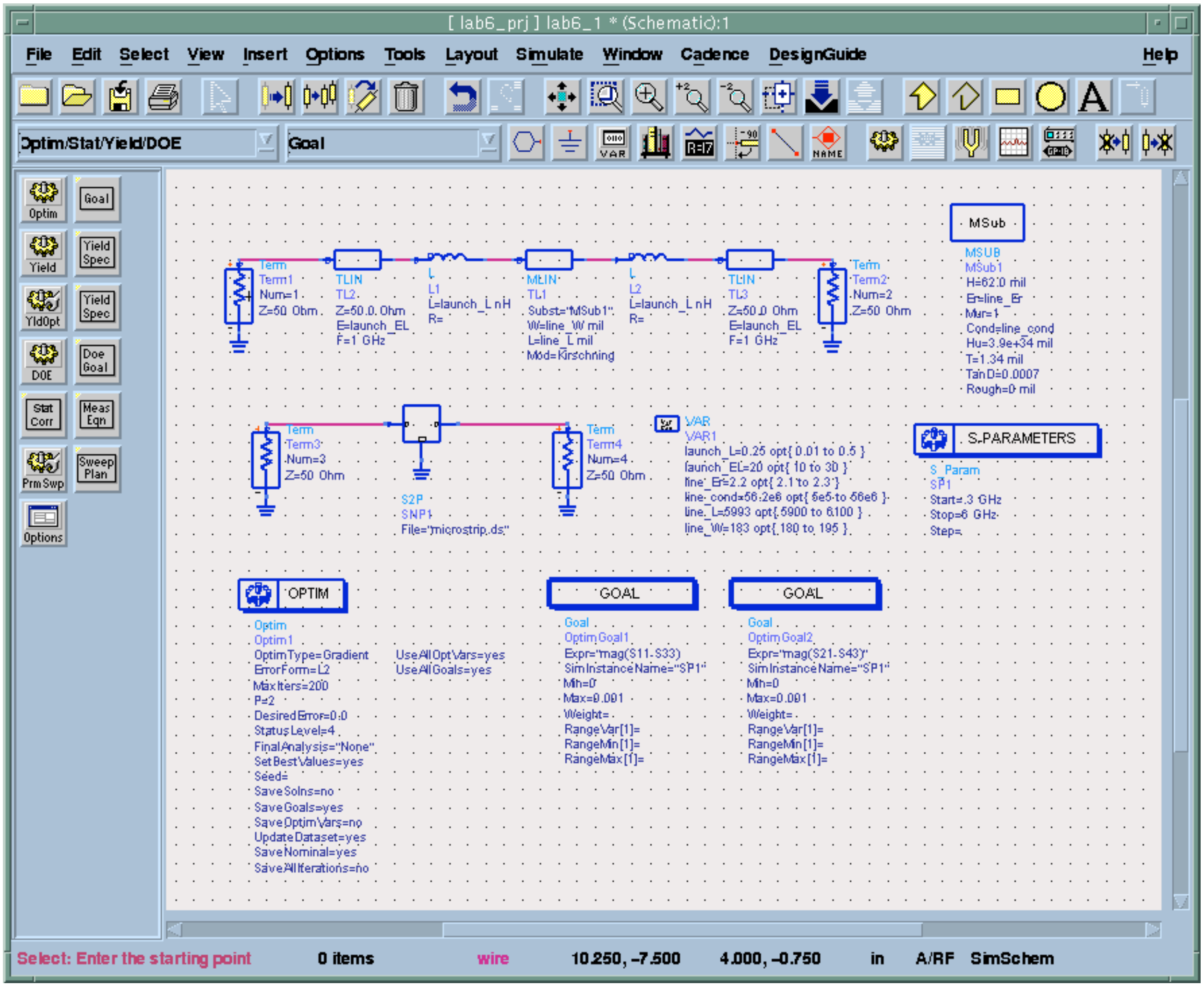

Figure 2. Schematic with optimization for microstrip line, including discontinuities and connector electrical length.

Microstrip Stub Filter Circuit Simulation

In this part of the lab, the behavior of the microstrip stub filter circuit that was measured in labs 2 through 5 will be simulated, using the substrate parameters obtained in the last section.

1.) In the main ADS window, select "File/New Project" to create a new project. Name the new project something like "stub_circuit," and click OK. A blank schematic window for the new project should open automatically.

2.) Construct a CAD model of the microstrip stub filter circuit in the schematic window. This can be done using the following steps:

a. Since the microstrip stub filter circuit consists of three sections of microstrip line, select the MLIN element from the "TLines – Microstrip" palette. Place three of these in the schematic window.

b. In the actual circuit, the fat stub is oriented at 90° with respect to the main line; to make the schematic reflect this, select the element you'll use for the stub portion of the circuit by clicking on the element (this will outline the selected element in a black box) and click on the "rotation" icon (below the menu bar, next to the trash can). Move the mouse around in the main window to see how the outline of the element re-orients itself as the mouse is moved. When satisfied with the orientation, click to fix the position, and press escape to exit rotation mode.

c. Copy the MSUB and VAR elements from the previous design to the schematic page. This is easiest by first opening the previous design, copying these elements to the buffer, then pasting them into the current design.
d. Add s-parameter terminations, an s-parameter simulation control, grounds, and wire the circuit together. Set the simulation to run from 30 kHz to 3 GHz with 201 points.
e. To set the dimensions of the three transmission line segments, double click on each and set the width and length as appropriate. For the filter circuit, these values are:
port 1: W = 190.6 mil, L = 2137.5 mil
stub: W = 637.4 mil, L = 2744.4 mil
port 2: W = 190.6 mil, L = 3862.5 mil
Note that these length values are "center-line" values; they are the lengths from the center to center of any intersections; equivalently, they are the lengths that would be recorded if all of the lines had zero width.

3.) Run the simulation, and graph the results. To read numeric values from the plot, use the marker function. Select Marker/New, and click on the appropriate plot trace to add a marker. Add a marker to the s_{11} trace, and record the simulation's prediction of $|s_{11}|$ for the frequencies measured in previous labs (100 MHz, 500 MHz, 1 GHz, 1.5 GHz, 2 GHz, 2.5 GHz, and 3 GHz). Save the plot under the file menu. How do these values compare to those obtained in Labs 2-5? Discuss any discrepancies and their possible origins.