Calibrating Network Analyzers
&
Tools and Tips for RF Electronics

January 22, 2013

1 Introduction

The purpose of this first lab is to learn the tools required to be successful in later labs. Network analyzers are expensive precision measurement instruments, but so are the cables, connectors, adapters and calibration standards that go along with them. If any of these are broken or damaged, and you don't know enough to realize that they're not working properly, you can waste time. Your measurements also won't make any sense.

In later labs, when you find things aren't working as expected, you'll want to use a general debugging strategy of "going back to something that works". This lab is designed to give you a few of those atomic units of "something basic that works right" that you can go back to. (Is my circuit design flawed, or is my cable bad? Is my code wrong, or am I not driving the CAD tool correctly? Is my network analyzer broken or did I push the wrong buttons?) In this lab, you'll *test your test equipment* and *test your CAD tools* to make sure that you have a foundation for successful measurements.

There are 3 topics in this lab:

- Network analyzer tender loving care and calibration

- Precision soldering

- RF CAD tools

You should work in groups in this lab, as you will in subsequent labs. In the later labs, your group can turn in a single lab report. However in this lab, each student needs to do the exercises and get GSI check-offs. When everyone in your group knows how to "see" a dirty connector under a microscope, or tell when a cal is bad, or spot a cold solder joint, then you'll have good eyeballs on future experiments, helping to spot problems and prevent wasted time.

1.1 Lab Etiquette

- Make sure every network analyzer has a keyboard and mouse hooked up. When working in groups, use the mouse rather than the touchscreen, so that your lab members can see which buttons you're pushing.

- Always inspect SMA cables under the stereo microscope, and clean them, before attaching them to a network analyzer port.

- Brush all bits of wire and solder off your bench, leaving it clean for the next student. Make sure every station has both a 7 mm open-end wrench and an 8 mm open-end wrench (the torque wrenches should stay with the cal kits).

- Tin the soldering irons at your station before you leave the lab, so they don't oxidize and not work well for the next student.

- Hang SMA cables back up on the rack. Be gentle with the cables as they're easily kinked, which causes signal reflections.

- If you're the last one out of the lab, check the calibration standard boxes and make sure all standards are back in their kits. Check the benches and vises to make sure cal standards haven't been left out.

- If you break something, or find something broken, don't put it back. Leave a note, attach a label, tell a GSI or see if you can figure out how to fix it.

Hopefully, all these things will make later labs go more smoothly, so you can spend more time learning how to make measurements and models, and less time fighting tools and equipment.

2   Calibration and Connector Care

Skim though the handout for Lab 1 on measuring high-frequency passives, so you get an idea of where this is going. At high frequencies, we use network analyzers which make S-parameter (scattering parameter) measurements. Instead of impedance measurements where a port would need to be either open or shorted (hard to achieve at high frequency), an S-parameter measurement system uses a 50 ohm load at each port. Network analyzers let you save these measurements to a file format, .s1p or .s2p, which RF CAD tools can read. You'll save .s2p files of your measurements to a thumb drive and then import them into a CAD tool. The various CAD tools let you convert from S-parameter format to Y-parameter or Z-parameter formats so that you can build circuit models. You'll need to model device parasitics so that you can account for them in Labs 2 and 3. At high frequencies, if you don't account for these parasitics, then your circuits won't perform as designed.

Carefully read through the documentation in the ppt file on the course resources web site on network analyzer error-correction. With network analyzers, the term "calibration" is used to refer to the procedure you perform at the beginning of every session. If your setup requires cables, you set out their geometry first - connect them to the network analyzer and tape them down, etc. You then attach calibration standards to the ends of the cables, and let the anaylzer read these known devices. A software cal kit loaded into the analyzer tells the analyzer what the model is for those standards. The analyzer then compares what it measures to what it knows and comes up with a set of error-correction coefficients which it then applies on subsequent measurements.

Note that you need to redo the error-correction calibration every time you change cables or re-arrange your measurement setup. Essentially, the calibration procedure moves the measurement reference plane (the plane for zero loss, zero phase, zero mismatch) to the plane where the cables mated with the calibration standards. In subsequent measurements, you'll take the standards off and connect your circuit. All measurements reported by the network analyzer are with respect to that reference plane where the cables now mate to your circuit's connectors. The assumption buried in all of this is that the connectors on your circuit are very similar (in terms of mismatch, loss, etc.) to the connectors on the calibration standards. Always try to think about what assumptions are buried in your measurement procedures. Most of the assumptions end up revolving around connectors being good or "ideal" at some level.

Read the "Connector Care" writeup from Agilent, which is posted on the lab resources website. It will give you a feel for what's involved in keeping the connector's in good enough shape so that they can perform as closely as possible to a uniform perfectly matched transmission line.

Also, watch Joel Dunsmore's "Introduction to Lab" video.

3 Precision Soldering of 0603 Lead-Free Components

Read the ppt file on the course resources web site on precision soldering. The key is to always use flux.

4 Learning the CAD Tools

You can use whichever RF CAD tools you prefer. This class has primarily used Agilent's ADS in the past, but AWR's Microwave Office (MWO) came out more recently and is also very good. Mathworks now has an RF Toolbox for Matlab which is convenient for some things (has built-in functions for converting between S-params and Y or Z params, etc.) One component of Cadence's suite, SpectreRF, has a few features which you'll use towards the end of the course.

Here we give some examples of working between MWO and Matlab. The example creates two very basic circuits in MWO, plots their S-parameters and then writes out an .s2p file for each circuit. Then with an example Matlab script, we read in the first .s2p file, convert its S-parameters to Y-parameters and extract a pi model circuit. Then we read in the second .s2p file, convert its S-parameters to Z-parameters and extract a tee model circuit.

Read through the cheatsheet on MWO on the course resources website to get started with Microwave Office. There's a similar pointer for getting started with ADS, and you can duplicate this exercise in ADS if you like. The goal is to get started with learning whatever CAD tool you plan to use for later labs and homework.

4.1 Y-parameters and Pi Models

You will learn more about two-port networks and S, Y, Z and ABCD parameters later in this course, but here is a brief introduction for the purpose of this example and for doing Lab 1. See the handout on two-ports in the syllabus and/or the lecture notes for Lecture 7 for more detail.

A two-port network is a circuit abstraction convenient for representing certain classes of circuits in a compact way. First, a port is defined as a terminal pair where the current entering one terminal is equal and opposite to the current exiting the second terminal (i.e. so that we can attach a source or a component across the port as shown in Fig. 1). By convention, we typically draw port 1 on the left and port 2 on the right.

All the complexity of any two-port is captured by four complex numbers (which are, in general, frequency dependent). The parameters represent linear relationships between the port 1 and port 2 voltages and currents. If we take linear combinations of current and voltage, we can derive any other parameter set.

$$\begin{pmatrix} i_1 \\ i_2 \end{pmatrix} = \begin{pmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{pmatrix} \begin{pmatrix} v_1 \\ v_2 \end{pmatrix}$$
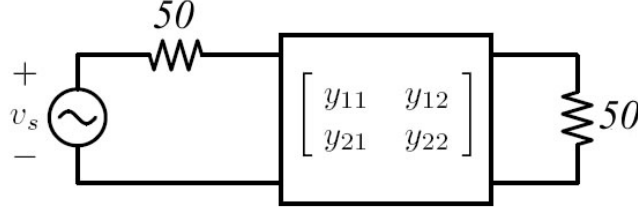

Figure 1: A two-port network modeled by its Y-parameters.

Fig. 1 shows a voltage source and 50 ohm source impedance attached to port 1 and a 50 ohm load attached to port 2. This is typically how a network analyzer is configured. You can think of port 1 of your network analyzer as providing a voltage source and a 50 ohm impedance to your circuit under test, and the network analyzer also presents a 50 ohm load on its port 2. The network analyzer then delivers the measured S-parameters which represent the ratios of incident-to-reflected and incident-to-transmitted voltage waves. (Internally to the network analyzer, there are switches so that after it applies the signal to port 1 and the load to port 2, it changes things around and then applies the source at port 2 and the load at port 1.) The network analyzer measures S-parameters, but it can be convenient to work with other parameters. Y-parameters are defined as in the matrix equation at the top of Fig. 1:

$$y_{11} = \frac{i_1}{v_1}\Big|_{v_2=0}$$

$$y_{12} = \frac{i_1}{v_2}\Big|_{v_1=0}$$

$$y_{21} = \frac{i_2}{v_1}\Big|_{v_2=0}$$

$$y_{22} = \frac{i_2}{v_2}\Big|_{v_1=0}$$

Y-parameters are convenient if we want to model our circuit under test with elements in a pi topology (one component across, and two in shunt). For instance, a short piece of transmission line can be modeled by an L across and an equal C in each shunt leg. The admittances of these components are easily found from Y-parameters. For instance, as Fig. 2 illustrates, shorting port 2 ($v_2 = 0$) lets us calculate y_{11} and y_{21} (since Y_2 is shorted, we're left with Y_1 in parallel with Y_3).

Similarly, from Fig. 3, we see that shorting port 1 ($v_1 = 0$) lets us calculate y_{12} and y_{22} (since Y_1 is shorted, we're left with Y_3 in parallel with Y_2).
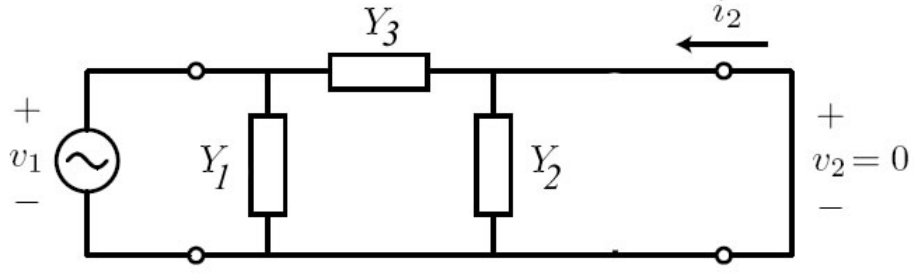
With these four equations:

5

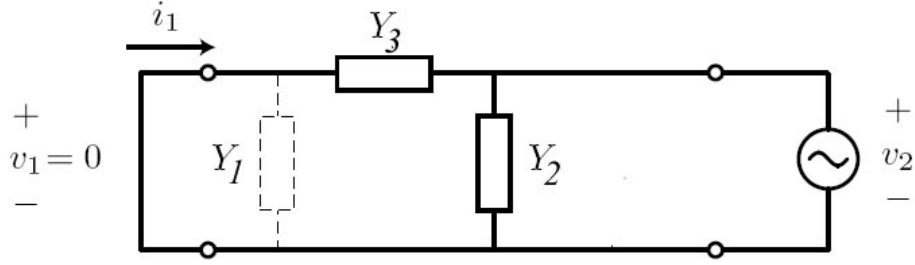Figure 2: Circuit for finding $\frac{i_1}{v_1}\big|_{v_2=0}$ and $\frac{i_2}{v_1}\big|_{v_2=0}$.

Figure 3: Circuit for finding $\frac{i_1}{v_2}\big|_{v_1=0}$ and $\frac{i_2}{v_2}\big|_{v_1=0}$.

$$y_{11} = Y_1 + Y_3$$

$$y_{21} = -Y_3$$

$$y_{12} = -Y_3$$

$$y_{22} = Y_2 + Y_3$$

...we can find Y_1, Y_2 and Y_3:

$$Y_1 = y_{11} + y_{12} = y_{11} + y_{21}$$

$$Y_2 = y_{22} + y_{12} = y_{22} + y_{21}$$

$$Y_3 = -y_{12} = -y_{21}$$

For the example of the C-L-C model of a short piece of transmission line, we would find C from $Y_1 = j\omega C$, or $C = Im\{Y_1\}/\omega$. Similarly, we would find the value of L from $Y_3 = -j/(\omega L)$ or $L = -1/(\omega Im\{Y_3\})$.

4.2 Z-parameters and Tee Models

Z-parameters are convenient when we want to model the circuit under test with a tee topology (two components in series with a shunt element between them). Fig. 4 illustrates a two-port

$$\begin{pmatrix} v_1 \\ v_2 \end{pmatrix} = \begin{pmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{pmatrix} \begin{pmatrix} i_1 \\ i_2 \end{pmatrix}$$
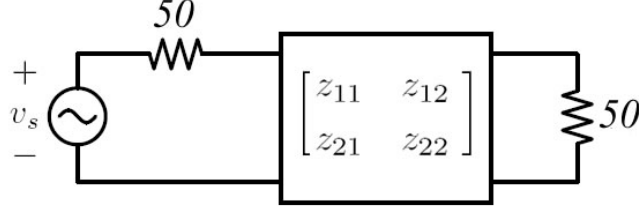

Figure 4: A two-port network modeled by its Z-parameters.

circuit again connected to a network analyzer. The network analyzer delivers S-parameter measurements, and then we can convert to Z-parameters in order to find circuit elements to fit the tee model topology of Fig. 5. For instance, measuring the S-parameters of a capacitor soldered to ground between two traces could be modeled with Z_3 as a capacitor and Z_1 and Z_2 as inductors.

Z-parameters are defined this way:

$$z_{11} = \frac{v_1}{i_1}\Big|_{i_2=0}$$

$$z_{12} = \frac{v_1}{i_2}\Big|_{i_1=0}$$

$$z_{21} = \frac{v_2}{i_1}\Big|_{i_2=0}$$

$$z_{22} = \frac{v_2}{i_2}\Big|_{i_1=0}$$

Fig. 5 and Fig. 6 illustrate what leads to 4 equations:

$$z_{11} = Z_1 + Z_3$$

$$z_{21} = Z_3$$

$$z_{12} = Z_3$$

$$z_{22} = Z_2 + Z_3$$

We can now find Z_1, Z_2 and Z_3:

$$Z_1 = z_{11} - z_{12} = z_{11} - z_{21}$$

$$Z_2 = z_{22} - z_{12} = z_{22} - z_{21}$$
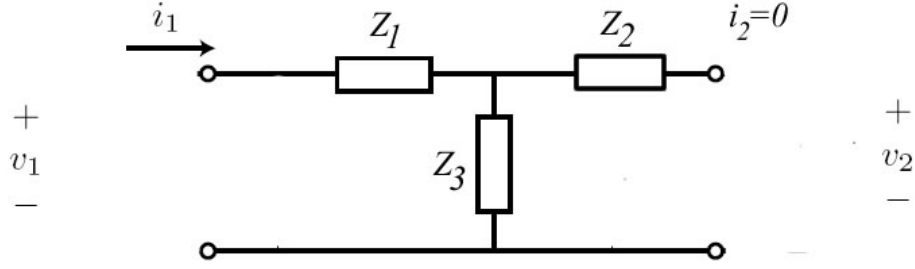
$$Z_3 = z_{12} = z_{21}$$

7

Figure 5: Circuit for finding $\frac{v_1}{i_1}\big|_{i_2=0}$ and $\frac{v_2}{i_1}\big|_{i_2=0}$.
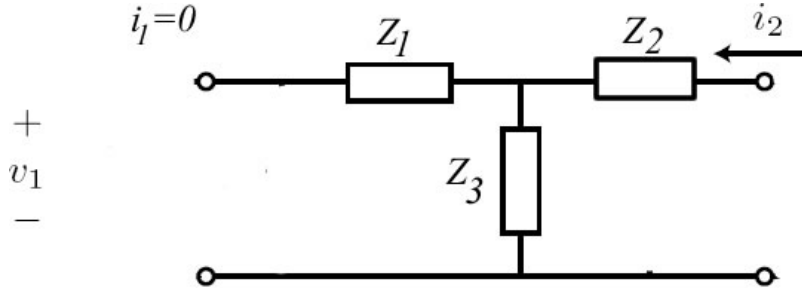

Figure 6: Circuit for finding $\frac{v_1}{i_2}\big|_{i_1=0}$ and $\frac{v_2}{i_2}\big|_{i_1=0}$.

For the example of the L-C-L model of a capacitor shunted to ground, we would find the values of L from $Z_1 = Z_2 = j\omega L$ or $L = Im\{Z_1\}/\omega$, and we would find C from $Z_3 = -j/\omega C$ or $C = -1/(\omega Im\{Z_3\})$.

5 Prelab

1. Read through the ppt file on the course website on calibrating the network analyzer.

2. Watch Joel Dunsmore's "Introduction to Lab" video from 2007. He explains how not to break the network analyzers.

3. Read the Agilent FAQ note on Connector Care, posted on the course website.

4. Also read through the ppt file on precision soldering.

5. Look at the cheatsheet on AWR's Microwave Office (or go through the ADS tutorials if you prefer to learn ADS).

6. Download and install the MWO software as explained in the cheatsheet. Download the example MWO project and the Matlab script for this lab.

6 Experimental Work

6.1 Connectors, Cables & Calibration

1. Using the stereo microscope, inspect your SMA cables' connectors before connecting them to anything.

2. Use the pin-depth gages to measure both the pin depth and the dielectric depth of all cable connectors, network analyzer test port adapters or calibration standards you'll be using. Don't use any of them if they're out of spec.

3. Always use two wrenches, an open-end wrench and a torque-wrench, when making connections. Put the wrenches on so that they're less than 90 degrees apart. Hold the torque wrench at the end beyond the grooves and stop as soon as it breaks (don't over-torque the connection).

4. Test your test equipment. On the network analyzer, press the Preset button and hit OK to bring the analyzer to the factory default state. Perform a calibration using the 85033E mechanical cal standard kit. Then verify with other standards that weren't used in the calibration (e.g. from the 85052D kit). Each student in the group can do a slightly different cal. The first person can do a full 1-port cal without any cables. Then verify that cal. The next student can do a 1-port cal to the end of a cable. Again, verify. The third can do a full 2-port cal. Verify to make sure it's good. You'll learn something by watching each process.

5. Perform a calibration using the Ecal. Verify using mechanical standards (e.g. from the 85033E or 85052D kits).

6. Always do a pre-test before performing a calibration procedure. That is, execute a preset and if you are using cables, tape them down - and with their ends either open or shorted, ensure that the traces are stable when you gently touch/jiggle the cables. Keep the cables in primarily the same position when you calibrate or make subsequent measurements. Don't touch the cables during a cal or during a measurement (as touching them will change their electromagnetic environment).

7. Get checked off by the GSI that you successfully calibrated and verified your cal. Demonstrate that you know how to torque a connection using two wrenches that start less than 90 degrees apart. Make sure you know the answers to these questions:

   Why do we use a torque wrench when making connections? Why should you never rotate the cal standard when you connect it? How can an SMA cable connector damage a cal standard? What's the difference between a 3.5 mm connector and an SMA connector? Where is their reference plane? Why is it recommended to remove the rubber gaskets from N and SMA connectors? Why is it a bad idea to use a cal standard as a load for your amplifier? Why should you turn the power down on the network analyzer when you hook up an amplifier? What hard key do you press to find the menu to change the power level?

8. Save an .s2p file of something. Perhaps add a second thru adapter between your Port 1 and Port 2 cables. You can read that file into your CAD tool later.

6.2 Soldering

1. For Lab 1, you'll need to use 4 boards: an Open board, a ShuntShort board, a Thru board and a ShuntDUT board. Each student in the group can solder SMA connectors onto one of these boards. Save these soldered boards for Lab 1. Make sure the backside of each board gets soldered appropriately (as pictured in the ppt and as illustrated in Chapter 9 of Dunsmore's book). Use flux whenever you solder, and inspect the board under the stereo microscope.

2. Solder an 0603 inductor onto either an Open/SeriesDUT board or a ShuntDUT board. Inspect it under a microscope.

3. Unsolder the inductor that you just soldered. Use flux and two soldering irons. After you remove the inductor, inspect the bottom side of the inductor under the microscope. It's common to rip the leads off the inductors because their plating is very thin and the wires are very fragile. If you rip the leads off, you'll run into problems when you attempt to solder it onto a subsequent board (in Lab 1, you'll need to solder the same inductor onto two different boards).

4. Solder the inductor back on. Inspect under the microscope. You can ohm it out with a voltmeter to make sure it's soldered correctly. Show the board your GSI and get a check-off.

6.3 CAD Tools

1. Save a few .s2p files from network analyzer measurements to play around with in your CAD tool.

7 Post Laboratory

1. Open the MWO project (an .emp project file and a .vin window-layout file) and the Matlab script (.m file). Make sure you set the paths correctly so that you can write files from MWO and then read them into Matlab. Fig. 7 and Fig. 8 illustrate the example for the pi model circuit (C-L-C). The schematic contains two 0.05 pF capacitors and a single 0.04 nH inductor. When you hit the lightning bolt icon in MWO, all the graphs update and an .s2p file gets written which contains the S-parameters of this circuit.

2. In Matlab, run the script. The script reads the .s2p file, converts the S-parameters to Y-parameters and then extracts the pi model's component values. Fig. 8 shows that Matlab deduces from the Y-parameter data that the capacitors are 0.05 pF and the inductor is 0.04 nH.
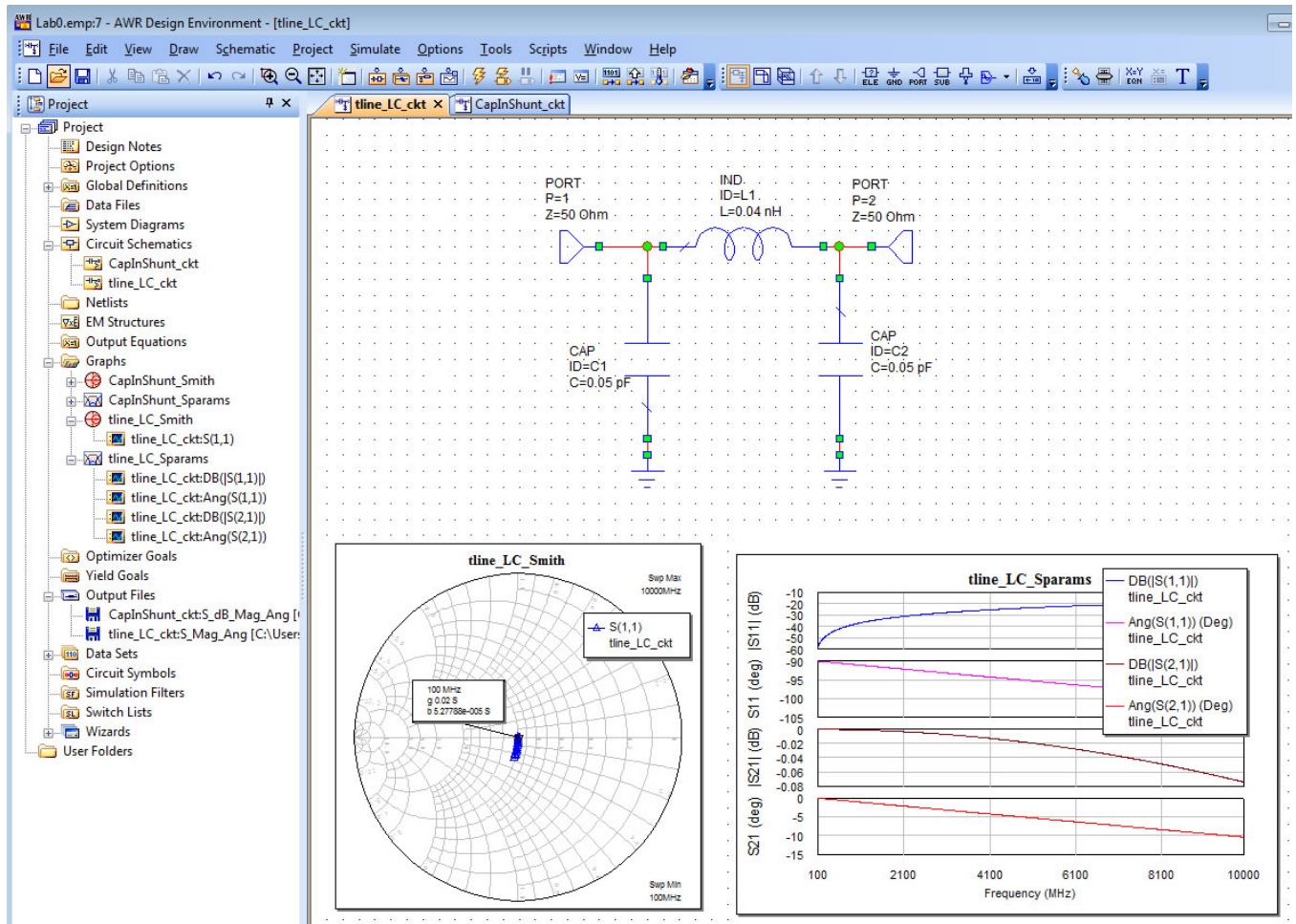
Figure 7: Screenshot from Microwave Office's window-in-window layout. The S11 Smith Chart's grid is in admittance mode and the marker is set to display the real and imaginary parts of the admittance, de-normalized to 50 ohms.
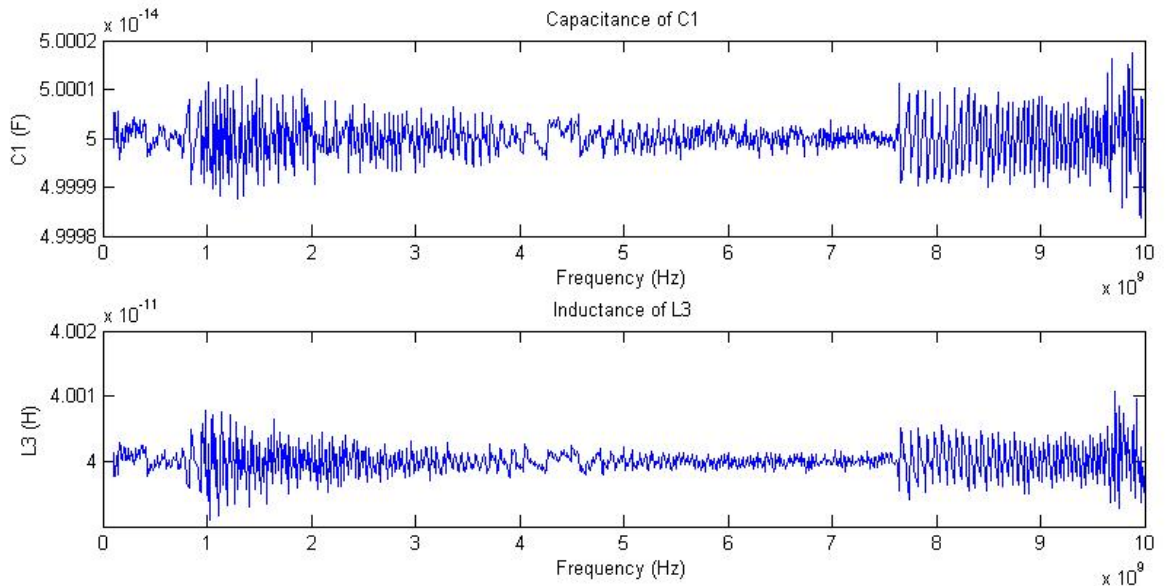
Figure 8: Matlab graphs of pi model component values.

3. Play around with these example files to learn the tools. There is a second example circuit (Fig. 9) which extracts a tee model circuit.

4. Learn how to use the Tuning feature.

5. MWO (and ADS) also has an Optimization feature. Optimization takes advantage of the tuning facility, but in Optimization you set up constraints and the computer (essentially) pushes on the sliders until the constraints are met. There are many example projects that ship with the MWO installation, and the cheatsheet discusses one of them. Play around with that Optimization example.

6. Take the .s2p file you saved from your network analyzer measurement, read it into MWO or ADS and plot its S-params in the graphing tools.

There is no report to turn in for this lab. The GSI will keep track of who he/she signed off on the calibration and soldering steps. In Lab 1, you'll be using all of the things you learned in this lab.

8   References

Joel Dunsmore's book, *Handbook of Microwave Component Measurements* is the final word on network analyzer measurements. We have a copy in the lab and it's also available online at the UCB library site.
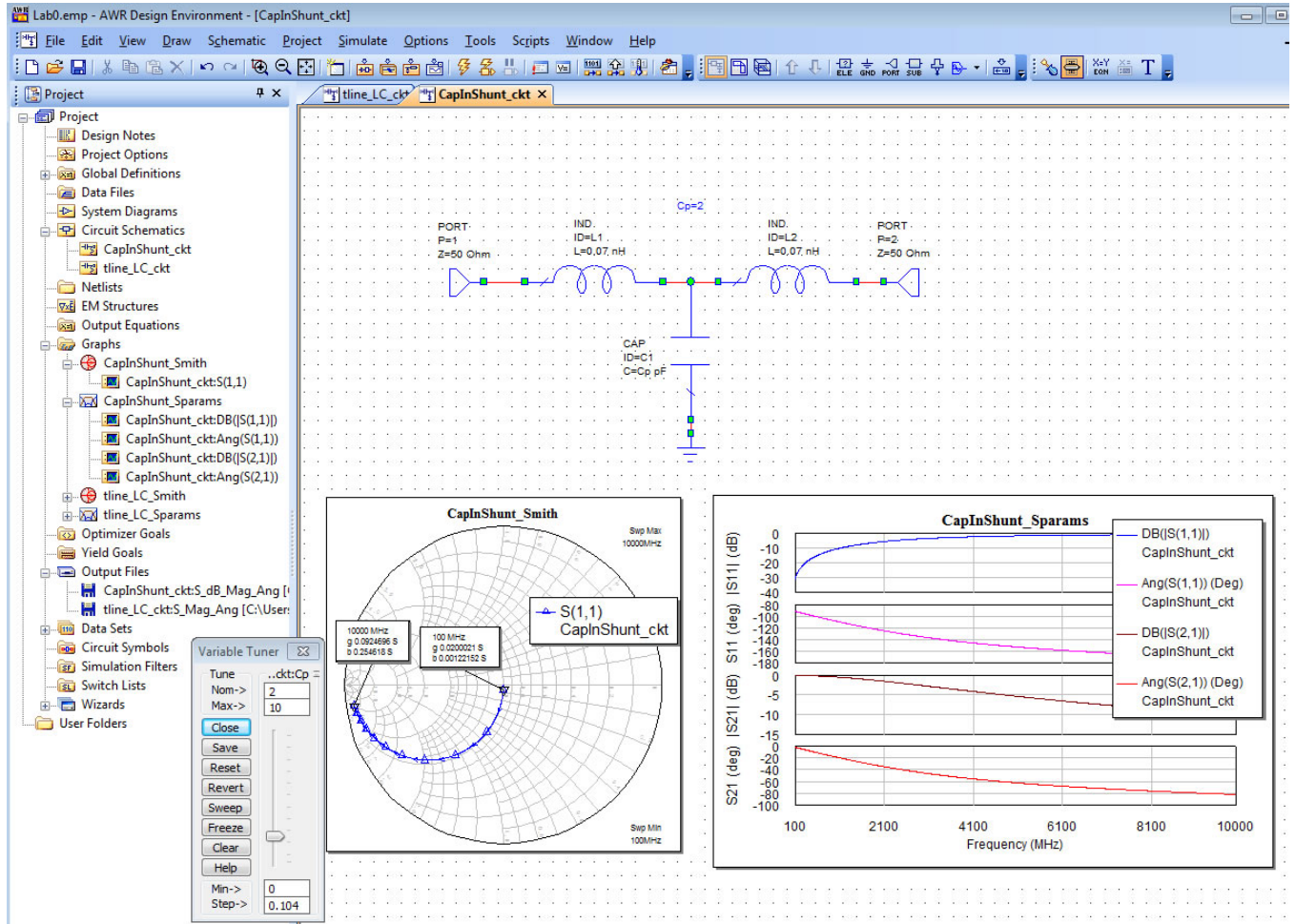
Figure 9: Capacitor in shunt to ground with small inductors in the tee arms. The tuning tool is open. Moving the sliders changes the capacitor's value and all the graphs update as the slider is adjusted.