ECE 145A/218A, Lab Project #1a: passive Component Test.

Oct. 13, 2022

Overview

Goals:
(a) Introduction to the vector network analyzer and measurement of S-parameters.

1. Learn to calibrate the network analyzer. This will be described in the lab demo. (Note. Please treat the cal kit with respect. Don't leave the standards sitting on the bench. Return them to the box.

2. Understand the use of port extensions.

3. Understand use of Smith Chart and rectangular displays
(b) Observation of transmission line behavior in PCB layout.
(c) Understanding non-ideal behavior of RF components.

Precautions to avoid instrument damage
(a) Observe static precautions when working with the network analyzer. Wear the wrist strap.

(b) Never connect a network analyzer directly to a circuit carrying dc. Make sure your circuit is dc blocked, or you will destroy the NA.

Safety precautions
(a) Safety note: please be very careful with the X-acto razor-blade knives. Use the same level of care you would use with a very sharp kitchen knife ! We want no injuries.

(b) Common solder is tin-lead and is toxic. If ingested, lead accumulates in your body and slowly and progressively causes brain damage as well as damage to other organs. The shop sells non-lead-containing solder. ***Do not bring lead-containing solder into the lab.*** That means, do not purchase solder at Radio-shack or other vendors unless you very carefully check its metal composition. The new non-lead solders have a higher melting point, which makes soldering harder, particularly soldering to ground planes. Use a higher-power (hotter !) iron for soldering to the ground plane.

(c) In case some student fails to follow the solder rules above, ***do not eat or bring food into the lab, wash your hands immediately after leaving the lab***, and sweep up and dispose of any solder debris. ***Do not use solder-suckers for desoldering , as these spray a fine powder of solder all over the room***. Horribly toxic if some fool uses them with lead solder ! Use desoldering wick (braid) instead; it is better anyway.

(d) Basic rules for electrical safety apply. Circuit voltages are low (~5-15 Volts) but avoid bringing in high-voltage DC supplies, use common caution in plugging in 120 V connections, do not stand in or work around water, and do not work with electrical equipment with bare feet.

Reading:
Refer to HP 8751A Network Analyzer User's Guide (RBR and in lab - do not remove!)

Network analyzer calibration

If you have change the frequency range (START, STOP), then the instrument must be calibrated again for the new range.

1. Set the required frequency range with the START, STOP Keys (Stimulus menu).
2. CAL menu. >MORE. Turn off PORT EXTENSIONS.
3. CAL menu. >Cal Kit > USER KIT [*kit_name_goes_here] > RETURN.
4. Select Full 2 Port. Follow the sequence as prompted by the buttons, attaching the short, open and load standards sequentially.
5. CAUTION: DO NOT ROTATE THE BODY OF THE STANDARDS WHILE TIGHTENING THE CONNECTOR! THIS WEARS OUT THE CENTER PIN AND WRECKS THE STANDARD. THE KIT COSTS ABOUT $5000! AVOID OVERTIGHTENING - USE THE TORQUE WRENCH.
6. Skip the Isolation step.

DON'T hit the PRESET green button. This erases the calibration!

7: After calibrating, you must VERIFY:

a) Turn ON the port extensions. Set these to the correct values for the open and short standards of the kit. As I (Rodwell) write this, I do not remember the values: check with your TA.

b) Measure the shorts, opens and load on the 2 ports, Make sure that the S-paremeters are the expected values.

c) TURN OFF The port extensions.

d) Measure a through connection between ports 1 and 2. Make sure that the S-paremeters are the expected values.

e) Make SURE that the port extensions are turned OFF before making measurenents (unless instructed below to turn them off). Make SURE that the port extensions are turned OFF when doing calibrations.

*Re "kit_name_goes_here": In some of our instruments, the calibration kit settings are hard-coded into the instrument. In other cases, the TA must load this information into the instrument for you. Please check with the TA regarding the name of the kit. For those instruments where the TA has loaded the information, beware ! Other members of the class might have erased or changed the kit coefficients. Please don't yourself erase the coefficients by messing around!

Components and Transmission-line characteristics.

<u>mechanics</u>

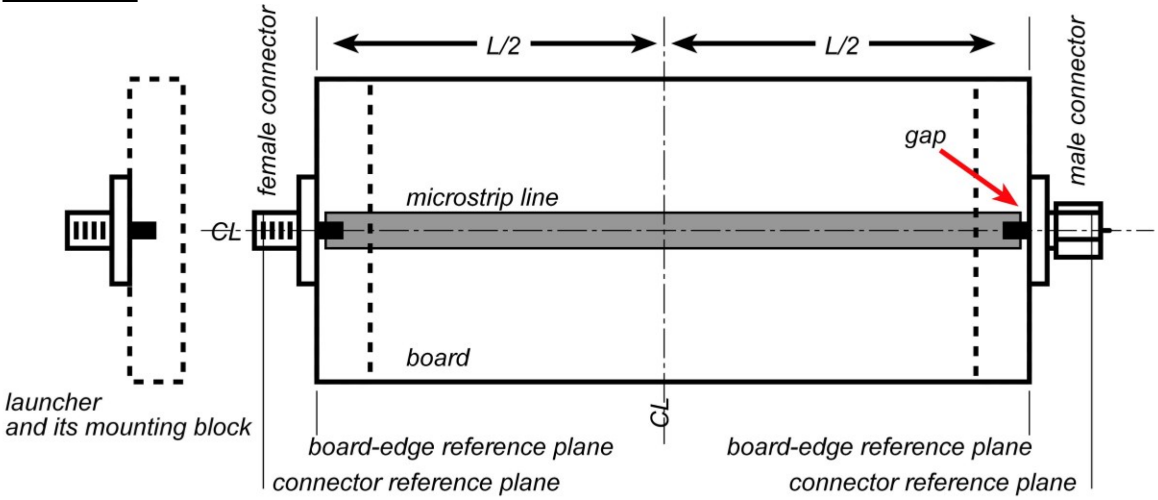


Figure 1: Launcher, block, board and microstrip line

You will be building a set of passive structures to test on the network analyzer. Figure 1 shows the basic structure. There is a Duriod circuit board with a ground plane on the back side, a microstrip line (a piece of sticky copper tape), and launchers and launcher blocks. Note the use of a female connector in the left (input) port and a male connector in the right (output) port. The network analyzer cables will be set up to work with this configuration, and so you must be sure to construct your board in this manner.

Different widths of microstrip line will give you different characteristic impedance and propagation velocity. You can use the LINECALC tool in ADS for these calculations, but be warned that LINECALC does not account for the effect of the glue's thickness, and hence will be somewhat in error. Figure 7 and Figure 6(in the appendix) show screen-shots of how to start linecalc and how to run it.

Although any circuit-board material is suitable for microstrip lines, we are using Rogers Duriod, which is an expensive Teflon laminate used in high-performance microwave systems. Its advantages include low dielectric attenuation and very reproducible dielectric constant. The key side-benefit for class work is that the dielectric constant is very low, hence the wave propagation velocity is high. This means that lines of some needed phase-shift will be relatively long, and therefore easier to cut accurately by hand.

The Duriod is 0.062” (1.57048mm) thick. It has a dielectric constant of approximately 2.3.

The dimensions have to be very tightly controlled. Make the gap (Figure 1) between the microstrip line and the launcher less than 1 mm (0.5mm would be better) to minimize the parasitic inductance of the launch. Throughout the lab, please be very careful and precise in construction. This will have great impact upon the accuracy of your measurements.

Characteristics of through-line

We will be measuring the characteristics of through-lines. The first problem we face is the location of the reference plane (Figure 1). After we calibrate the NWA, the measurement reference plane will be at the coaxial surface of the connectors, not at the edge of the board. We want the reference plane at the edges of the boards.

determining the reference plane offset

To measure this difference, first construct a board like so (Figure 2). Wrap the sticky copper tape around the edges of the board, cut it, and then solder it, at a point just beyond where the launcher blocks will touch, so that the solder does not mechanically interfere with the launchers. Now mount the blocks and the launchers----you have short-circuited the two launchers.

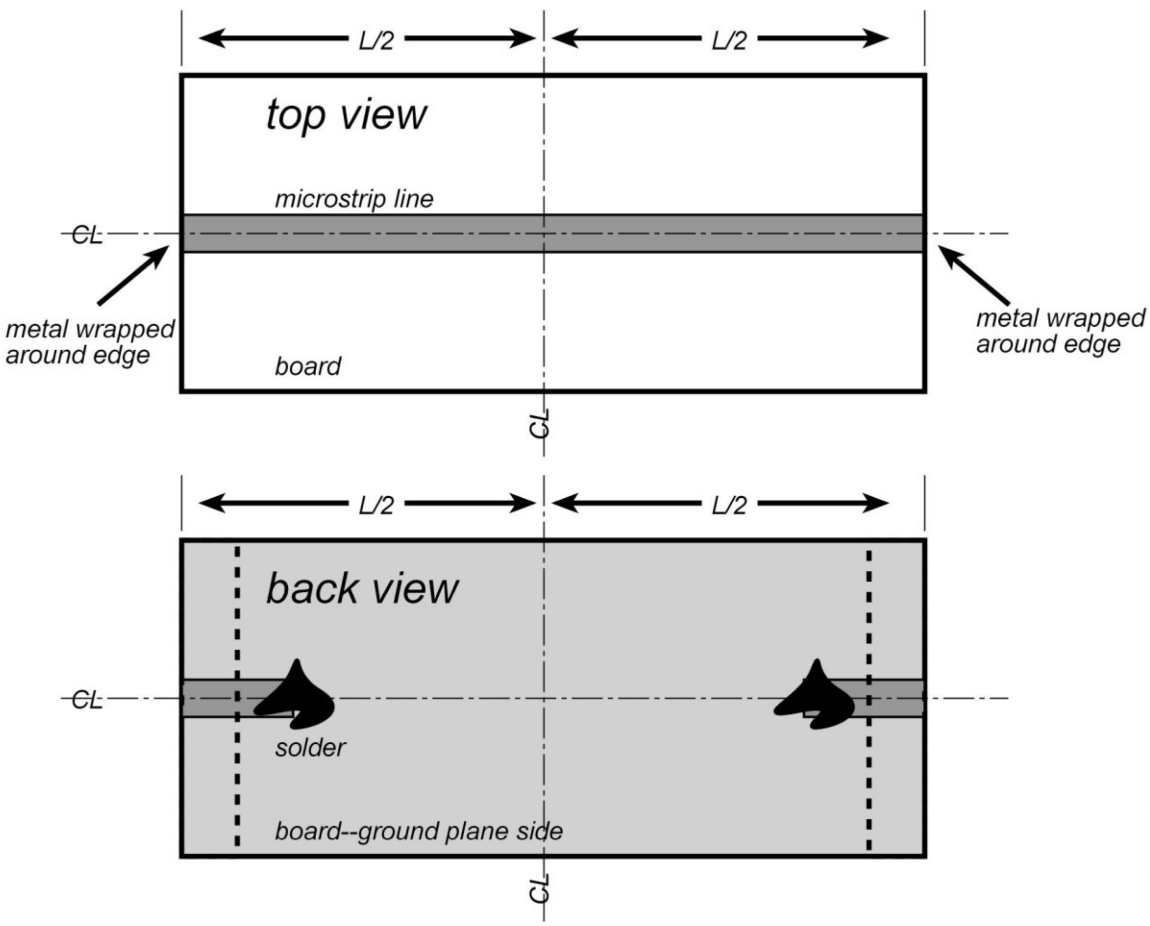


Figure 2: board with short-circuits at edges.

Calibrate the network analyzer using the coaxial cal kit. Then connect the NWA to the board. Display S11 and S22. You should see short-circuits on the Smith chart, but with some phase rotation associated with the shift in the reference planes. Now, find the "offset reference plane" settings in the NWA, and adjust these until S11 and S22 are as close as is possible to a perfect short-circuit. Write down the resulting numbers, and then set the reference plane offsets back to zero.

Line measurements

Now construct a through-line as in Figure 1. ***First do this with the 5mm wide line, and then later with the 10mm and 3mm wide lines***. Again, precise construction here is very important. After constructing the line, calibrate the NWA using the coaxial calibration kit, and then dial in the port extensions you had found earlier. Then measure all four S-parameters. Note that the 10mm line should be approximately 50 Ohms characteristic impedance.

Record all S-parameters, and transfer into the PC and open up the data in ADS. The TA will show you how to do this.

Now please determine the characteristic impedance and phase velocity from these data. Do this as follows. When the frequency varies, the line will vary from being odd numbers of a quarter-wavelength and even numbers of a quarter-wavelength. As we will learn later, the input impedance (hence S11) of the line will be, in this case, (NOTE CORRECTED FORMULA-->) $Z_{in} \cdot (50\Omega) = (Z_{o,line})^2$. From this, you can determine the line characteristic impedance.

Having imported your measurement data, determine the line characteristic impedance and propagation velocity by simulating a through-line in ADS and comparing S11 and S22 between theory and measurement. Adjust the velocity to fit the measured and simulated phase of S21. Adjust the line characteristic impedance (as noted above) to best fit the measured and simulated magnitude of S11. If this is done well, the Smith chart (magnitude and phase) of S11 and S22 will also fit well. The line has some attenuation, and this will also affect the magnitude of S21 and S12.

Passive component tests: making and characterizing the fixture.

First, construct a board as in Figure 3. The microstrip line should be 50 Ohms, or as close as is feasible. Take a copper strip of the correct width, wrap it around the back side, cut it off at the back side ***no more than 2-3 mm away from the edge***, and then solder the edge to the ground plane. Calibrate the network analyzer, measure S11, and then adjust the port one reference plane offset until you see a short circuit for S11. Record this reference plane offset.

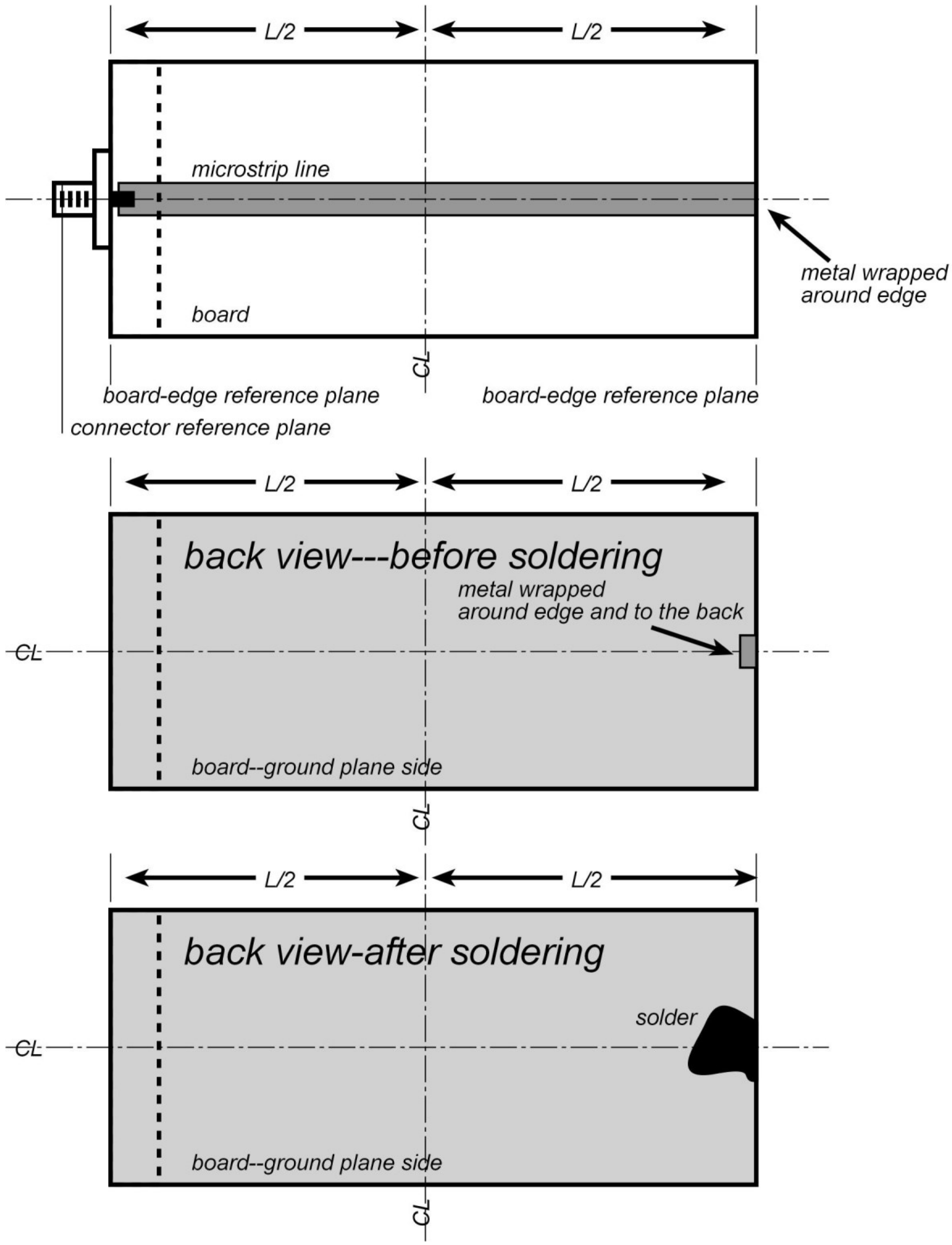


Figure 3: Passive component test structure part 1: to measure reference plane offset.

Taking the fixture from Figure 3 and now cut (Figure 4) the metal conductor close to the end of the board. ***The cut should make a gap less than 1 mm wide. The gap should be less than 2 mm from the board edge***.

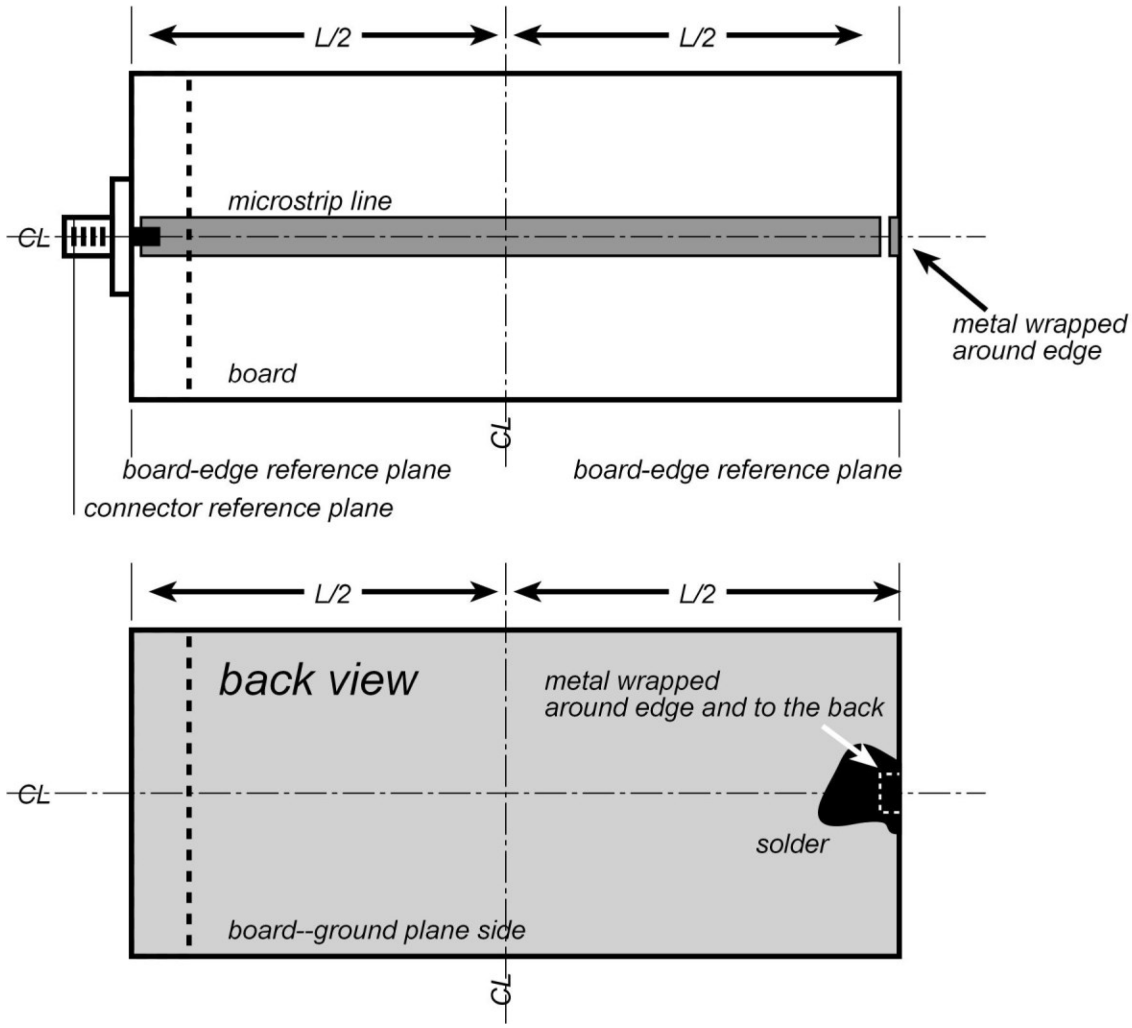


Figure 4: Passive component test structure part 2, cut to mount passive component. .

Passive component measurements.

You will now mount and characterize several passive components. For each, you will mount the component, connect the board to the calibrated NWA, dial in the correct reference plane offset corresponding to the procedure explained in Figure 3, and then measure and store the resulting S11.

First mount and measure (Figure 5) a normal leaded 50 Ohm resistor using ~1/4 inch lead lengths (a). In the resulting S-parameter measurement, you should see a great deal of parasitic inductance. Then re-mount the resistor with as short (b) lead lengths as you can manage (bending wires with needle-nose pliers will help greatly) and re-measure. Then mount and measure a (correction) 51 Ohm chip resistor (c); for this, you should see very little parasitic series inductance. Repeat the procedure with a 100 pF leaded capacitor (d), with a 100 pF chip capacitor (c) , and with a ~30 pH hand-made inductor (d). Please see the appendix for how you make this inductor. For the capacitors, look in s11 for the signature of an LC series resonance. For the inductor, compare the measured to the actual inductance, and look for the signature of an LC parallel resonance.

In your report you should analyze carefully the S-parameters and impedance of these components, analyzing and explaining as best you can what you have observed.

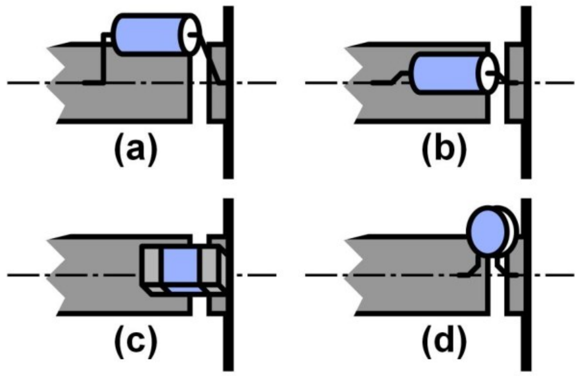


Figure 5: Component mounting details (solder not shown). Mounting of 50 Ohm resistor with long lead lengths (a) and with very short lead lengths (b), mounting of chip resistor or chip capacitor (c), mounting of leaded capacitor (d). The inductor should be mounted as in (b).

Appendices

<u>Running Linecalc</u>

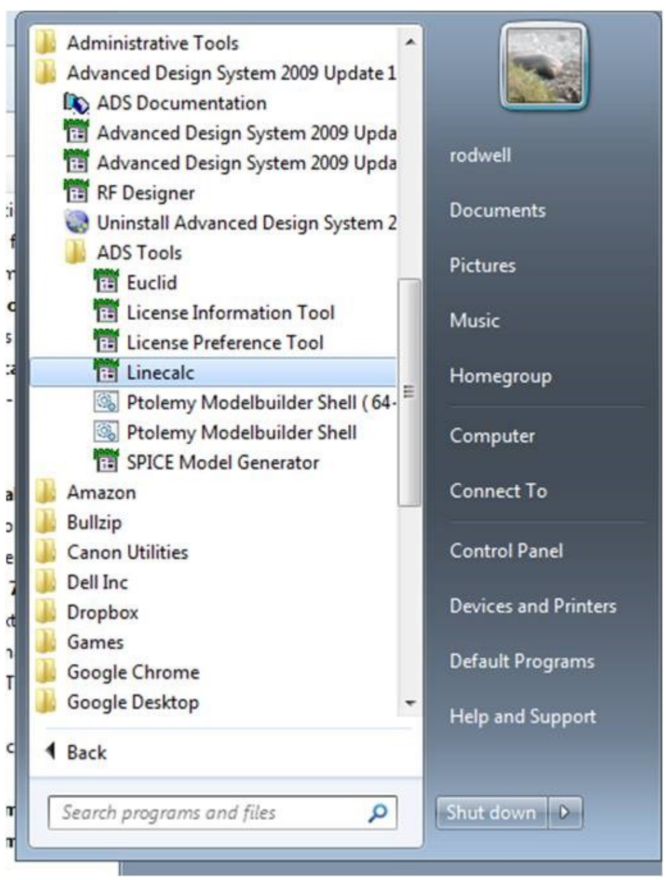


Figure 6: Starting linecalc. It can also be started within ADS

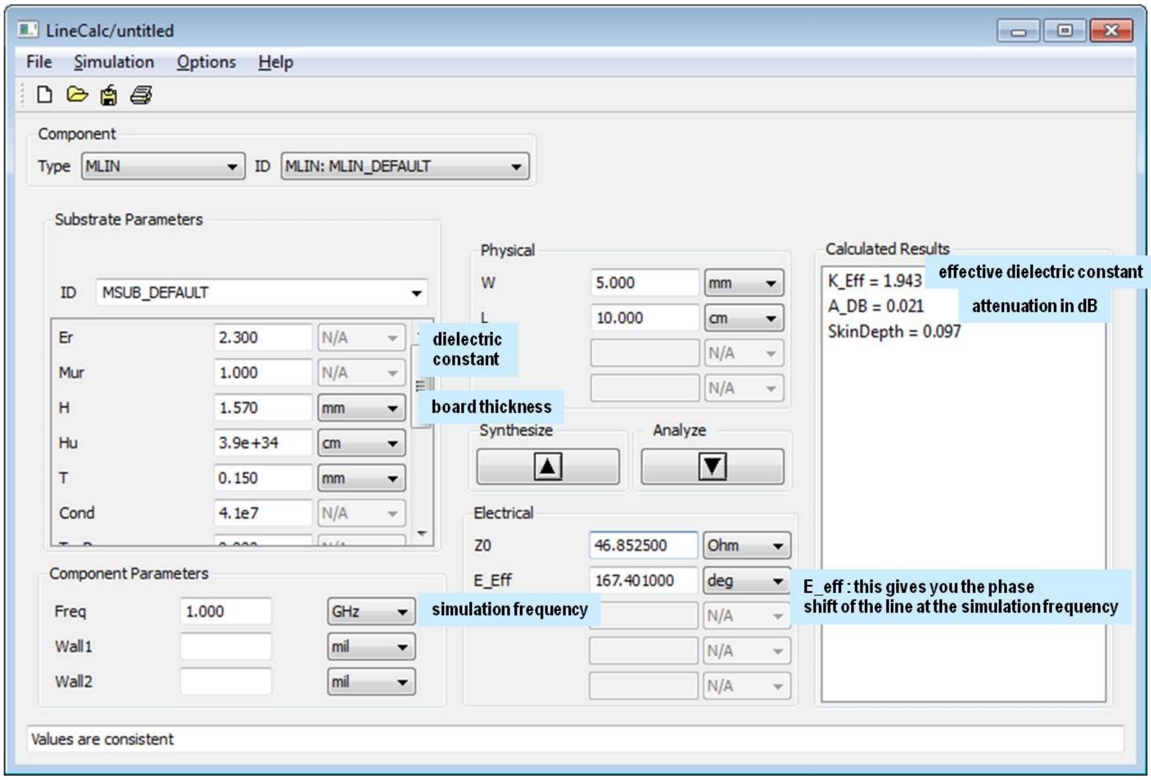
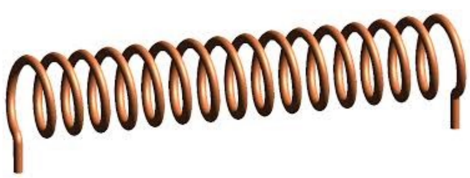


Figure 7: Linecalc simulation window

Making a small wirewound inductor

Inductors can be in the form of a solenoid or a planar spiral (Figure 8). Approximate formulas for inductors of several shapes can be found in https://en.wikipedia.org/wiki/Inductor . Some web sites impement these formulas; you can search for "inductor calculator" on the web, or use

https://m0ukd.com/calculators/air-cored-inductor-calculator/
http://kaizerpowerelectronics.dk/calculators/spiral-coil-calculator/
https://www.diyaudioandvideo.com/Calculator/AirCoreInductorDesigner/
https://www.diyaudioandvideo.com/Calculator/AirCoreInductorDesigner/
http://www.66pacific.com/calculators/coil-inductance-calculator.aspx
Since all these web pages use formulas with varying approximations, it is wisest to use the relationships from the Wikipedia web page above: you know which formulas are appropriate for your geometry.



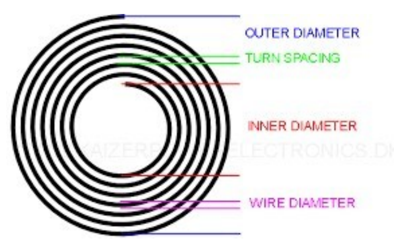
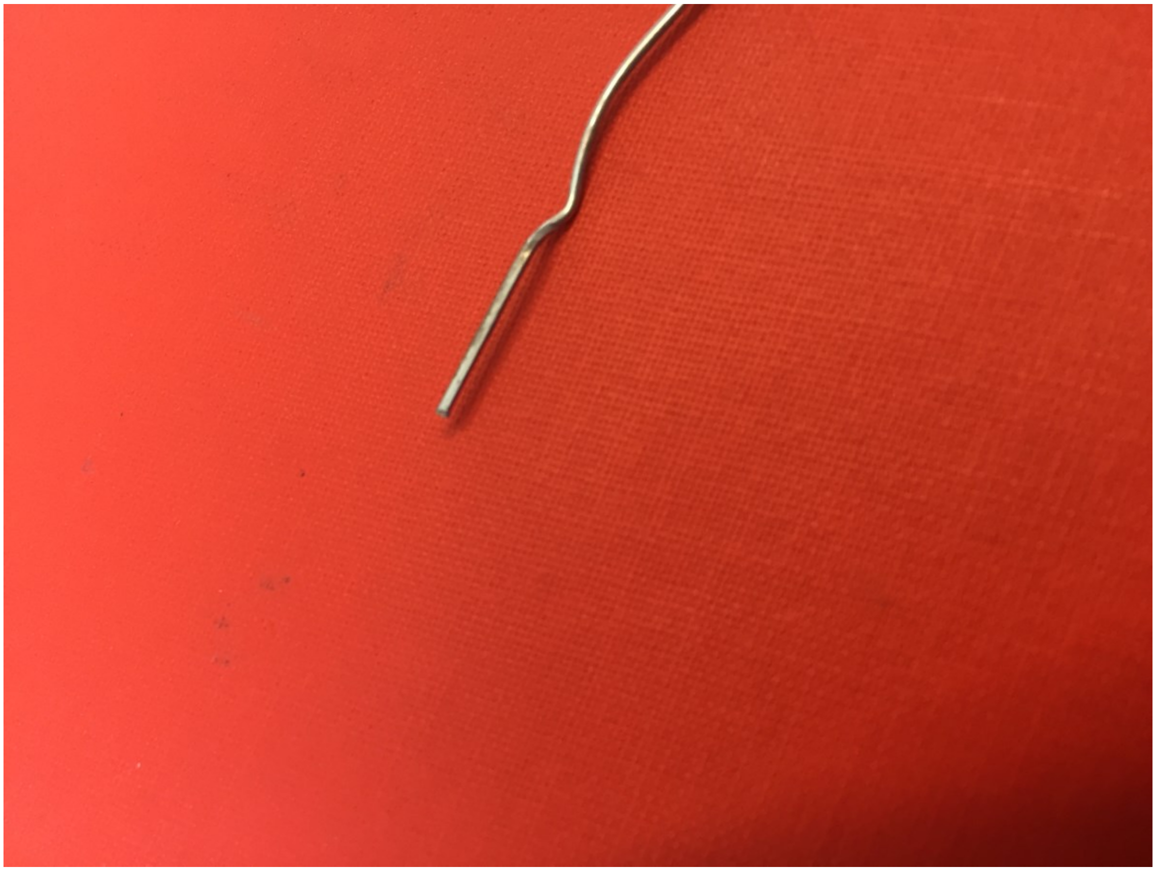


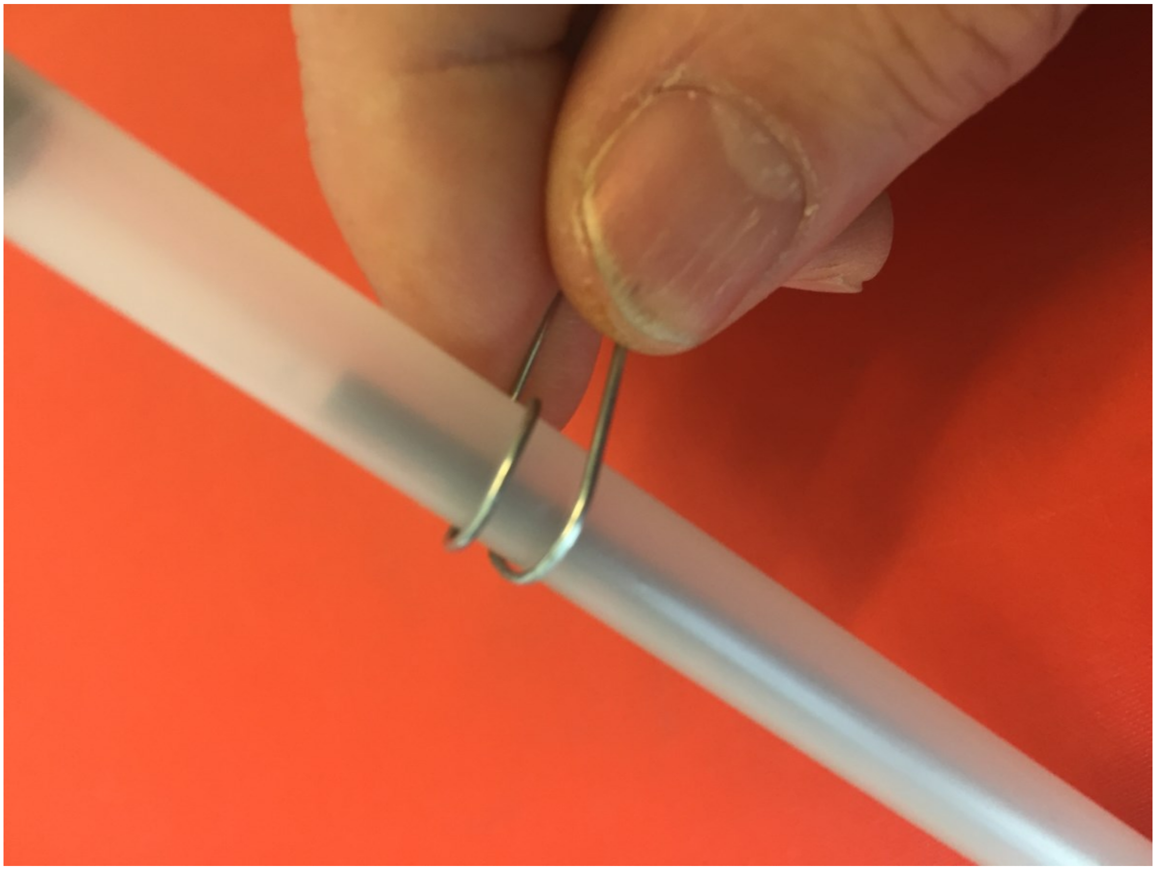
Figure 8: Solenoid (left) and planar spiral (right) inductor. From https://en.wikipedia.org/wiki/Solenoid and http://kaizerpowerelectronics.dk/calculators/spiral-coil-calculator/

From http://kaizerpowerelectronics.dk/calculators/spiral-coil-calculator/ , we find that an 8mm diameter coil of 1 mm diameter wire, with 1.5 turns should give us about 30 pH inductance. Ignore the parasitic capacitance value on the web page: it is wrong.

Start with a resistor lead:



Wrap it round a pen of 8mm diameter. That is a typical cheap plastic ball point pen.



Here is another view

Cut the wire, and bend the leads. Note that we have 1.5 turns and about 8mm diameter