PCB Design

Introduction

Printed circuit boards (PCBs) make up the platforms upon which most modern circuits are built. In basic terms, they are formed by an insulating substrate that holds conductive traces connecting different components in a circuit. Using modern fabrication methods, highly complex and repeatable circuit boards can be made, containing sometimes many layers of conducting planes. Thanks to the miniaturization of components in modern manufacturing processes, parasitic effects can be reduced, enabling the practical implementation of high-frequency circuits in PCBs. When dealing with RF frequencies, PCBs must include features that render the circuit viable, such as conductive traces with a desired characteristic impedance, low parasitic capacitances or inductances, RF isolation among different parts of the circuit, and low RF emissions, to name a few. In this lab, you will make measurements of a test PCB using the FieldFox, and using these measurements, you will design a new PCB that will be manufactured. In a later lab, you will assemble and characterize the PCB you designed.

Overview of the test PCB

The test PCB was created to allow you to measure the characteristics of a PCB and a mismatched test load. Using these measurements, you will be able to determine the features for your own design. The test board circuit is shown in Figure 1. It was designed for operation at 1 GHz.

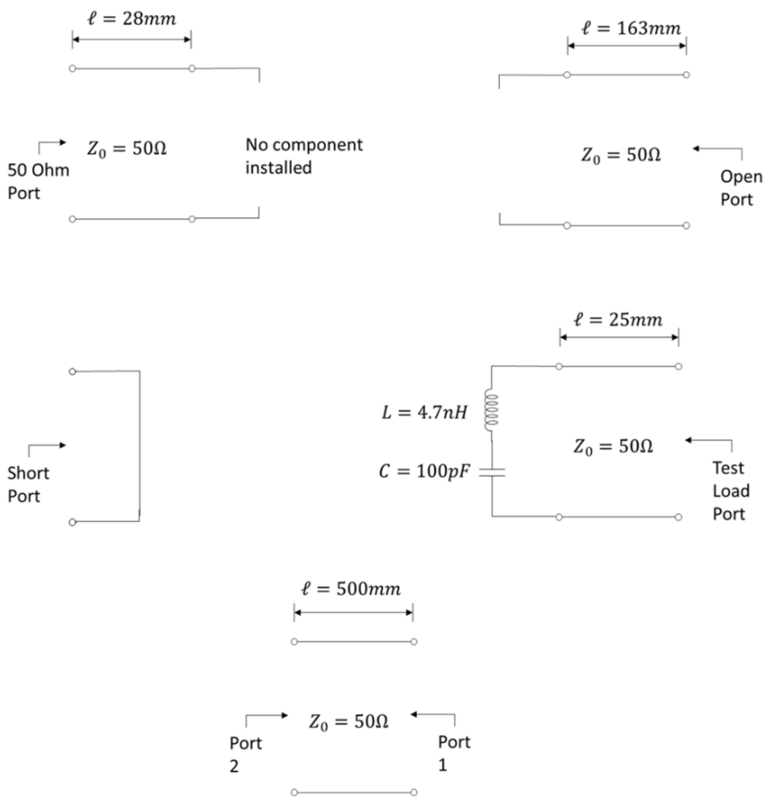

Figure 1 – Test board circuit

The test board has 5 independent sub-circuits. They are laid out similarly to the schematic, see picture of the board in Figure 2.

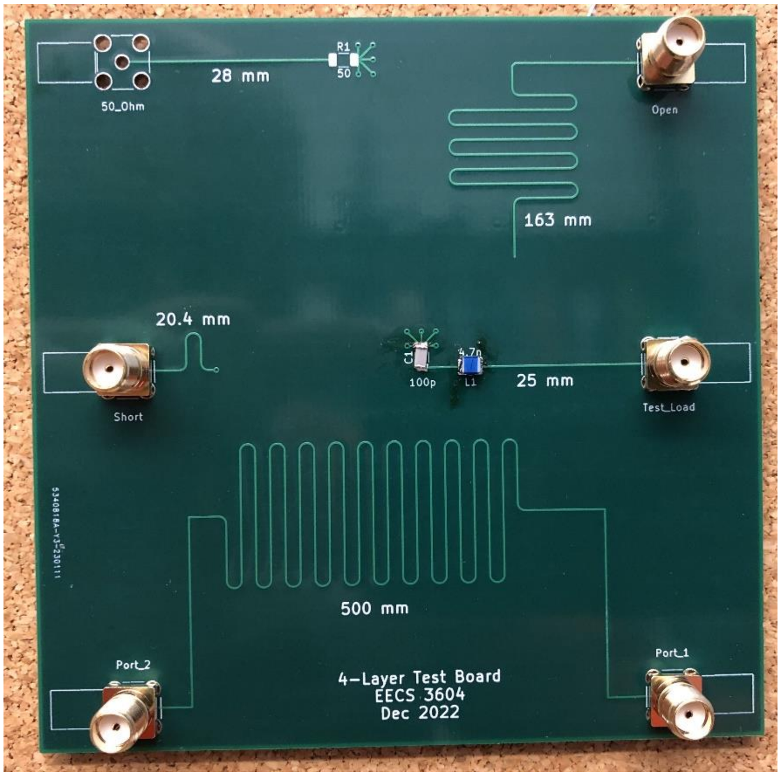

Figure 2 – Picture of the test PCB.

1. The sub-circuit with the 50-Ohm port has a 28 mm transmission line between the SMA connector and a pad. The pad was designed for a 50 Ohm surface-mount resistor, but that component was left unpopulated due to cost.
2. The sub-circuit with the Open port simply contains a 163 mm transmission line and it is left open-circuited.
3. The sub-circuit with the Short port has a 20.4 mm transmission line that is left short-circuited. NOTE: The board has been modified, and it has a short-circuit soldered on the back of the SMA connector, so ignore the 20.4 mm trace.
4. The sub-circuit with the Test Load port has a 25 mm transmission line and a 4.7 nH inductor in series with a 100 pF capacitor. Although components with much smaller form factor exist, these sizes permit hand-soldering with some care.
5. The last sub-circuit is a 2-port network consisting of a 500 mm transmission line.

Measurements on the test board

The FieldFox will now be used to make measurements on the test board. In preparation for these measurements, which are meant to determine the effective wavelength of the transmission lines as fabricated on the PCB, we want to make sure that the cables and connectors do not contribute to the measurement. A procedure called “port extension” allows the measurement plane of the FieldFox to be moved from the port of the instrument to the beginning of the PCB trace itself. Here is how this is accomplished.

1. Set up the FieldFox to operate on the external power supply. Start up the FieldFox and after it has loaded and launched the default CAT application, press the Mode button and then the NA soft key to load and launch the Network Analyzer application. If they are not already present, install a Type-N to SMA adapter on port 1 and port 2 of the FieldFox.
2. The NA application starts up showing the S_{11} parameter over the full frequency range of 30 kHz to 4 GHz and presents this in a log-magnitude format. The S_{11} parameter represents voltage measured at port 1 in relation to the voltage being supplied into port 1. With nothing attached to port 1, this should be a nearly flat line across the screen centered over 0 dB, indicating nearly 100% reflection from port 1.
3. Connect one end of a 24-inch long coaxial cable with SMA connectors to port 1. We will now remove the effect of the cable, so that we can measure the response of the PCB without the cable. This procedure is done by activating the “port extension” feature, which will be explained in more detail in the next lab.
4. Set the frequency range between 0.9 and 1.1 GHz by pressing Freq/Dist and entering the desired values under Start and Stop menus. Set the marker at 1 GHz by pressing Marker and typing 1 GHz.
5. Press Measure, Format, and select Smith. The Smith chart should show a trace revolving several times around the screen (see Figure 3).
6. Press Measure Setup, Port Extensions, then the Port Extensions menu to activate the port extension feature.
7. Select Port1 Extension, then type 3 ns. This is the propagation delay we are guessing the cable will have. Once entered, this parameter will reduce the trace length on the Smith chart. However, the load does not appear to be an open circuit, and we need to fine-tune the port extension.
8. Select Port1 Extension again, and rotate the selector knob until the trace reaches the open circuit location on the Smith Chart and the trace collapses to a small circle and the equivalent reactance of the load is minimized (the equivalent reactance is shown as an inductance or capacitance on the top-right of the screen) – see Figure 3. Finally, press the Back button.
9. Attach another 24-inch ling coaxial cable with SMA connectors to port 2, select Measure, S22.
10. Repeat steps 4-8 to make a port extension on port 2 The Fieldfox now measures the characteristics of the networks connected to the cables without including the effect of the cables.
11. <u>Make a note of the time delays used for port extensions on ports 1 and 2.</u>

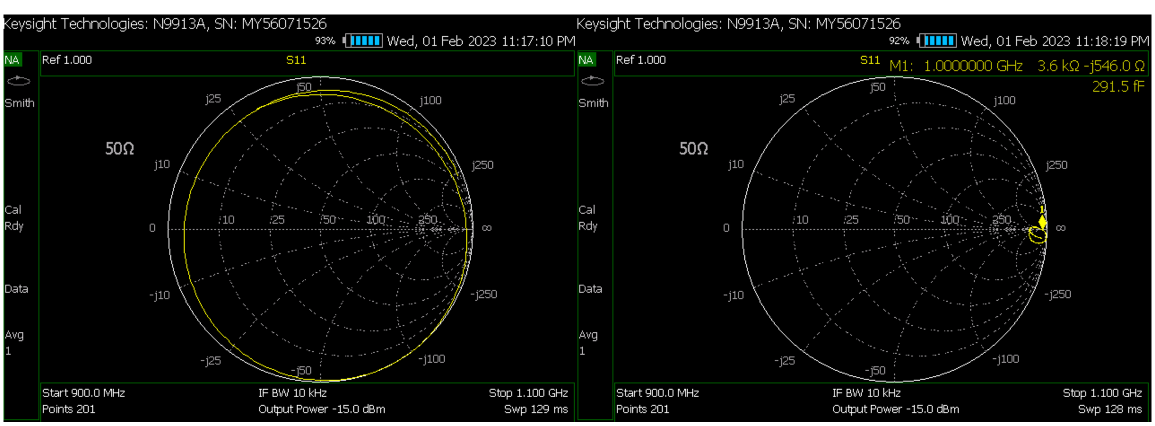

Figure 3 – Smith Chart of the S11 measurement between 0.9 and 1.1 GHz on the 24-inch cable before and after port extension.

The procedure above moved the measurement planes of the FieldFox ports 1 and 2 to the end of the cables you attached. This process needs to be repeated if you change cables, as each will have slightly different characteristics, and this matters at these frequencies. In fact, in order to measure the characteristic properties of the PCB traces, we shouldn't just remove the effect of the cables as we did so far, we should also remove the effect of the SMA connectors soldered on the PCB itself!

In order to also account for the SMA connectors, we need to repeat the process above and connecting the FieldFox to the test PCB on an appropriate test point. The SMA connector labeled as Short on the test PCB has been intentionally shorted right behind the connector on the PCB to provide this calibration.

1. Repeat the port extension procedure above for both ports 1 and 2 of the FieldFox using the Short test point on the PCB. Note that a properly calibrated test plane should look like a small circle near the short circuit point on the Smith chart, not the open circuit point as we did for the unterminated cable.
2. Make a note of the time delays used for port extensions on ports 1 and 2 that include the SMA connector.

You are now ready to make measurements of the test board. One parameter that will be of interest is the phase velocity of the transmission lines. This will be needed to relate the physical length of the traces to fractions of wavelength of the waves propagating in them.

1. Set the measurement to S11, format Phase. You should see a flat curve near zero degrees with the port extensions properly set.
2. Connect the cable from port 1 to the Open port on the PCB. Select the frequency of the sweep to start at 0 Hz, by pressing Freq/Dist and setting the start at 0 Hz. You should see a sawtooth trace. The amount of phase accumulated up to 1 GHz is related to the line length through the phase velocity. Keeping in mind the plot wraps around at 180 degrees (the phase value resets at +/- 180^{o} but keeps accumulating), determine the total phase shift at 1 GHz. This should be a number near -750 degrees. Divide this number by 2, since the signal on an S11 measurement travels two ways (to the load and back). Convert the number into radians.
3. Calculate the phase velocity of the wave at 1 GHz using the equation $u_p = \frac{2\pi f}{\beta}$, knowing that phase [in radians] $= \beta l$, where the line length is 163 mm. You should obtain a phase velocity which is about half of c. Make a note of this phase velocity.

You can compare the phase velocity obtained this way with another measurement on the PCB.

1. Set the measurement to S12, connect port 1 of the FieldFox to Port 1 on the PCB and port 2 of the FieldFox to Port 2 on the PCB.
2. Determine the total phase of the trace and repeat the calculation in step 3 above. Make a note of this phase velocity. It will likely be slightly different from the previous measurement due to some imperfections in the methodology. Pick a sensible value and use it for the design of your PCB.

Let's now verify whether the characteristic impedance of the lines in the test PCB is close to the desired value of 50 Ohm. We will do this in a few different ways.

1. We know from theory that a lossless transmission line terminated by an open circuit will have an impedance given by $Z = -jZ_0 \cot \beta l$. So, measuring an open-circuited trace can yield information on Z_0.

2. Switch the measurement to S11, Smith format. Connect port 1 of the FieldFox to the Open port of the PCB.
3. Set the Frequency span to start from 0.9 GHz as before.
4. With the cursor set to 1 GHz, read the complex impedance on the top-right corner of the FieldFox. We are interested in the reactance, which can be used with the equation above to determine Z_0, knowing that the line length is 163 mm. The value of β was already determined above. You should get a value close to 50 Ohms. Make a note of this value.
5. Connect port 1 of the FieldFox to Port 1 of the PCB, and use a 50 Ohm termination on Port 2 of the PCB.
6. Read from the top-right corner of the FieldFox the complex value of the impedance. It should be a value close to 50 Ohm. Make a note of this value.

Finally, we can measure the test load.

1. Make an S11 measurement in Smith format by connecting port 1 of the FieldFox to the Test Load port of the PCB.
2. Read the complex impedance on the top-right cornet of the FieldFox. Make a note of this value. We will refer to it as Z_L.

The goal of the remainder of this lab will be to:

- Using the value of the test load as seen at the Test Load port of the PCB, design two different impedance matching networks.

It is important to realize that the value for Z_L you just measured is the combined effect of the capacitor and inductor on the PCB, the specific layout they are in (trace lengths and spacings) as well as the 25 mm line in front of them.

PCB designs – preliminary calculations

You will design two different impedance matching networks that will be inserted as part of a new PCB design. The networks will be placed between the SMA connector and the test load sub-circuit of the test PCB. In other words, you will keep the value and location of the capacitor and inductor as well as the 25 mm transmission line, and insert other circuitry before the SMA connector, so that once the PCB is fabricated, the impedance looking into the SMA connector will be matched to 50 Ohms.

Stub network design

The first impedance matching design will consist of a stub network, as we have seen in class. You can decide to use either an open-circuit or a short-circuit stub network. Whichever type you will pick, there will be two design variables: 1) the length of the stub, and 2) the location of the stub from the load. Remember, in this case the load will be the total impedance of the circuit (line plus components) Z_L you measured in the previous section. Using a Smith chart, calculate these parameters in units of λ, and then convert these values to physical dimensions of mm.

Quarter-wave transformer design

The second impedance matching design will consist of a section of 50 Ohm line to be added to Z_L, followed by a quarter-wave transformer. Starting from Z_L and using a Smith chard, determine the distance in units of λ and mm that will cause the impedance looking into the load to be a real value (no reactance). Then, use the quarter-wave transformer formula to determine the impedance necessary in a quarter-wavelength line to have an impedance of 50 Ohm. Note the designed values of distance and transformer impedance.

PCB designs – PCB layout assignment

Now that you have characterized the test board and obtained the design parameters, it is time to design your PCB. This task will be accomplished over the course of the next two labs. Your group will be preparing a set of design files that will be used for PCB manufacture. Later on, you will assemble and test the PCBs you have designed. You will submit one design per group.

Grading scheme

This lab is out of 15 marks.

- 6 marks for answering questions from the Measurements on the test board section.
- 6 marks for completing the PCB design preliminary calculations.
- 3 marks for INDIVIDUALLY answering additional questions about this lab found on eClass.

All questions can also be found on eClass, where you will enter your answers. Note that all group members may have similar answers, EXCEPT for the questions indicated that are to be answered INDIVIDUALLY, where each group member is expected to have their own answer.