Matthew Spencer – Spring 2018

E190AK Lab 3: VNA and S-Parameters

In this lab you will use a VNA to measure the S-parameters of a few specially PCBs. You will use the S-parameters to extract the values of common parasitic elements in high frequency boards.

Read References about the VNA

Read pages 5 and 6 of the RF basics guide and the Impedance matching guide on the RF lab web page.

Calibrate the VNA

Before calibrating the VNA attach an open calibration standard to port 1. Observe the S11 magnitude, the S11 phase, and the S11 Smith Chart. Explain what you see in these plots.

Adjust the VNA to look at the range from 100MHz to 600MHz. Calibrate your setup with the calibration standard. Perform a few sanity checks to be sure that your calibration is good: look at the Smith Charts for the short and open calibration standards and perform a through test. Explain what you see.

Attach the calibrated setup to an open PCB. Do the S-parameters look the same as the open calibration standard? Why not? Repeat for the short PCB. Can you think of a way to move your reference plane to fix this issue?

Be sure to back your assertions and observations throughout this section with analytical calculations.

Compare 40 Ohm and 50 Ohm Transmission Lines

There are two through PCBs available for this lab, a 40 ohm through and a 50 ohm through. What type of transmission line was built on these PCBs? Use equations to demonstrate that the dimensions of the traces and stack up of the PCB give rise to the listed impedances. Also, explain why the SMA connectors are mounted on the edge of the PCB rather than being mounted vertically into the board (you may need to look this up).

Measure the S-parameters of both transmission lines and compare your measured results to theory. Comment on the importance of impedance control when laying out a high speed PCB.

Extract Pad Capacitance and Via Inductance

Parasitic elements on your boards can change the performance of your designs. There are parasitics inside of your components, they should be modeled with a series inductor and a pair of shunt caps at the input and output, and there are parasitics inherent in the board, vias cause inductance and solder pads have capacitance.

Solder a capacitor in shunt on a blank filter PCB, use the VNA to measure the resonant frequency of the connection, and extract the inductance of the via from that measurement.

After that, replace the shunt capacitor with a zero ohm resistor. The via inductance will still be present in the board, but it will be in parallel with a shunt capacitance from the solder pad. Use the VNA to measure the resonant frequency of this structure and to predict the pad capacitance.