HERIOT-WATT UNIVERSITY

B39HF | B31HD – High Frequency Circuit Design

Laboratory 3: Wireless Local Area Network (LAN) Matching Circuit Design and Test

Please read this document prior to entering the laboratory. As part of the pre-lab you should be familiar with the theory and equations outlined in this lab guide as well as Keysight ADS.

Background:

The first goal of this experiment is to become more familiar with ADS. The second objective is to simulate and design a high frequency circuit which uses a quarter-wave matching section and then measure the performance of your fabricated design using a USB-based Vector Network Analyzer (VNA). See http://pocketvna.com/ for further information on the lab VNAs.

Introduction:

We learned in the first and second labs that ADS is a design tool for RF/Microwave Engineers. It has a library of typical components (lumped elements, transmission lines, power splitters, hybrids, diodes, transistors, sources, etc.), which are then used to build and simulate a circuit or system. The most basic results of the simulation are outputs such as S11 and S21 which refer to the logarithmic values in decibels of the reflection coefficient and the transmission coefficient for a high frequency circuit or device. In the last experiment you measured S11 (or the reflection coefficient) in magnitude and phase. Recall that S11 is a measure of the reflection at port 1 for a high frequency device, while S21 is a measure of loss (or gain) from the input to the output of a device being measured.

In class we also discussed quarter-wave transformer sections of transmission lines using empirical formulas. In this experiment you will use this theory as a starting point for the simulation and optimization of some high frequency circuits that require some type of matching, to improve the power delivered to the load, and to minimize reflections. Basically we are trying to match a load Z_L to a transmission line which has a characteristic impedance where $Z_o \neq Z_L$. To do this we can use a matching technique called a quarter-wave transformer which uses a transmission line of a fixed $\lambda/4$ length (see Fig. 1). The impedance $Z_{\lambda/4}$ of the transformer is given by Eq. (1). We will also simulate our circuit using values typically used for dielectric substrates and microstrip lines, and just like in the second lab where you determined the dielectric constant of the substrate.

$$Z_{\lambda/4} = \sqrt{Z_0 Z_L} \qquad (1)$$

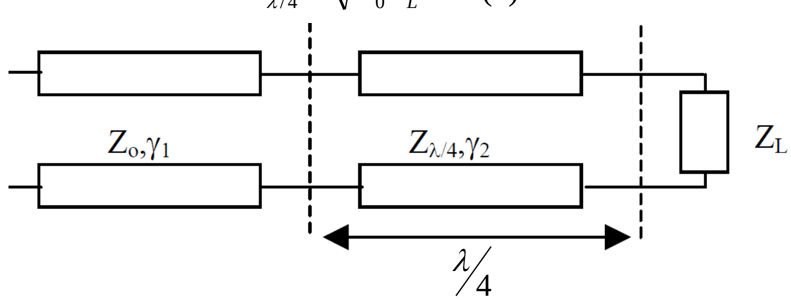

Figure 1 *Basic schematic of a quarter-wave transformer.*

<u>Procedure</u>:

1. Open Keysight *ADS*.

2. Create a new project, and then open a schematic window (Window — New Schematic). Make sure to specify millimetres and NOT mil in the new Workspace Wizard. Select the option: “Standard ADS Layers, 0.0001 millimetre layout resolution”.

3. In the schematic window (see Fig. 2), the element bar on the left side of the screen contains various lumped elements which may be deposited on the schematic. You can try different items if you want (try a resistor by pressing the resistor button, then drop the resistor on the schematic using the left mouse button. Hit escape to stop placing resistors). Double click on the resistor to change its value to 100 Ohm.

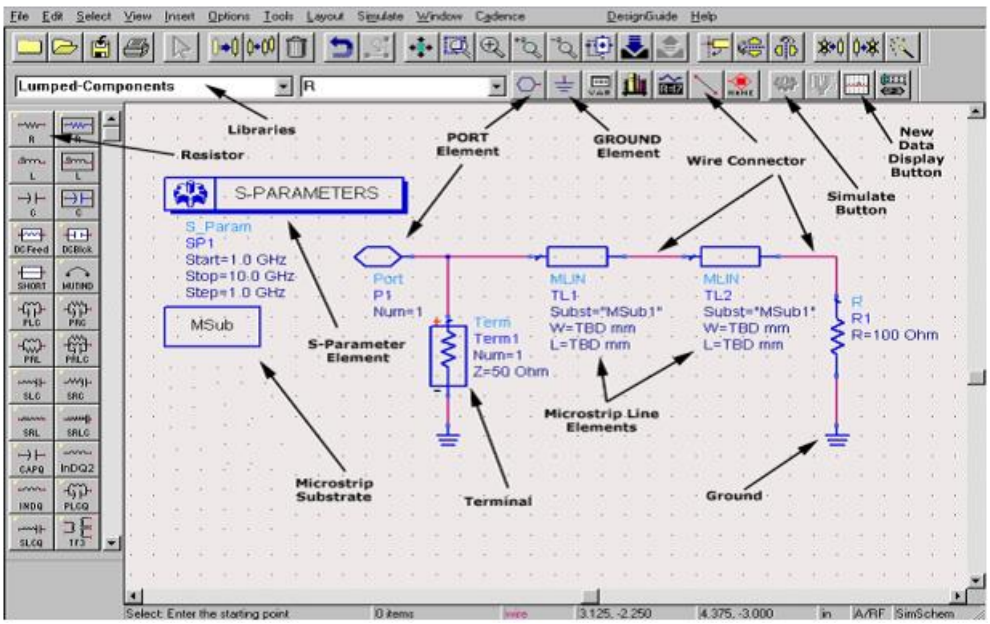

Figure 2 *Schematic Window and Libraries / Elements.*

4. As you get more and more comfortable with the user interface, the buttons on the element bar may be changed to other libraries using the selection block right above the element menu (it should, as a default, read “**Lumped – Components**” library). Select **TLines-Microstrip** library to get microstrip devices. Deposit two microstrip lines (**MLIN**) and a microstrip substrate (**MSUB**). Double click on the microstrip substrate to set the substrate parameters. ***Set the substrate to FR4 (Er = 4.3, tan(d) = 0.019, thickness = 1.6 mm).*** Keep the other values as default.

5. Change to the **Simulation-S-param** library, and then deposit a port, a terminal (**TERM**), a ground connection (see Fig. 2 to see where to get the port and ground), and an S-parameter simulation block (**SP**). Set the start, stop, and step frequencies of the S-parameter simulation as you think appropriate. Set ‘Term1’ as 50 ohms, which is telling the software that the reference impedance of the system will be 50 ohms.

6. Use wires to connect the simulation items together using (**Component—Wire** library), or by pushing the wire button.

7. Using equations given in class and this lab guide, find the required impedance for a quarter-wave transformer to match a 50 Ω transmission line to a 100 Ω load at 1.5 GHz.

8. Open the LineCalc tool **Tools – LineCalc – Start LineCalc**. LineCalc (see Fig. 3) is used to determine the width and length of transmission lines based on the substrate parameters and frequency of operation.

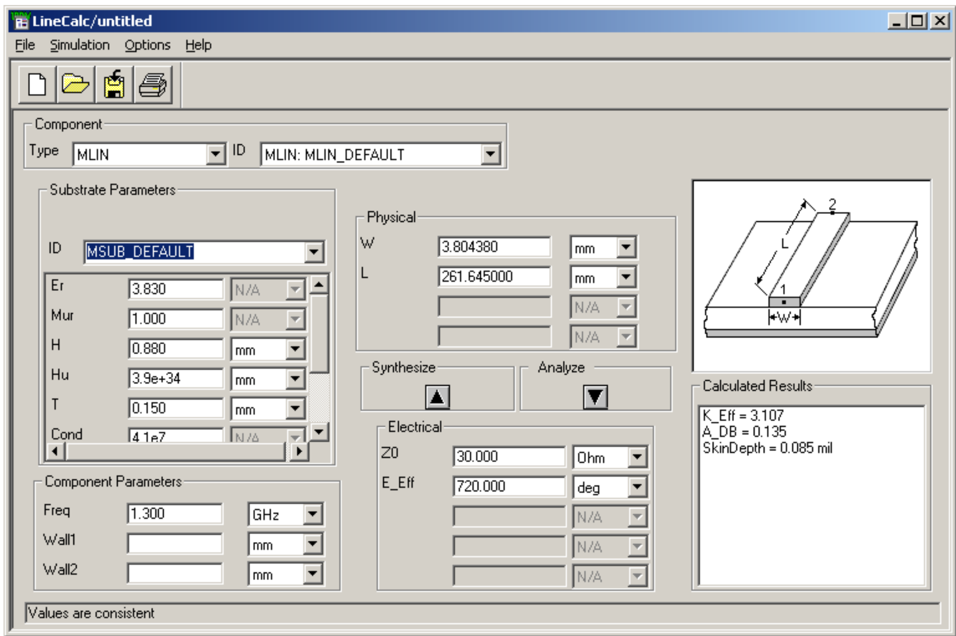

Figure 3 *LineCalc window showing microstrip calculations.*

a. Ensure that Type is set to MLIN, and Substrate Parameters are set to ID MSUB_DEFAULT.
b. Set the dielectric constant to FR4 (Er = 4.3, tan(d) = 0.019, thickness = 1.6 mm).
c. Set the frequency to 1.5 GHz.
d. Ensure all other parameters match the MSUB1 parameters in your schematic window.
e. Put Z_O = 50 (impedance in ohms) and E_Eff = 90 (electric length in degrees) under the 'Electrical' Section.
f. The button 'Synthesize' is used to determine the line width and length required for a 50 Ω transmission line which measures 90 degrees long when built on the substrate defined from points a to d above list.
g. Click the Synthesize button, the analyser will run, and the width (W) and length (L) will appear in the "Physical" section.
h. Go back to the schematic window, and put the L and W values you have just computed in the MLIN1 element (the 50 ohm line).

9. Use the same method in part 8, to determine the dimensions of the matching section which has the impedance calculated in part 7. You may put “90” into the E_Eff box to determine the length of the quarter-wave section. This corresponds to a length of 90˚ long or $\lambda/4$.

10. **Save** your design. Simulate the system by selecting **Simulate**, or pressing F7, or clicking the Simulate icon on the toolbox (it looks like a spinning gear).

11. Plot the result by opening a new data display (**Simulate — New Data Display**, or pressing the graph icon on the toolbox). In the new window that pops up (Fig. 4), select **Insert — Plot**, then insert the square box in the blank display by clicking. Select S11, click “Add”, then chose “dB”. Select “OK”, and the plot of S11 will be displayed. Note the frequencies at which the circuit is matched. Is this what you expected? Explain.

12. Tweak the parameters of the matching network to achieve a match at 1.5 GHz if necessary. Plot the result. Account for the differences in the predicted and previously observed results.

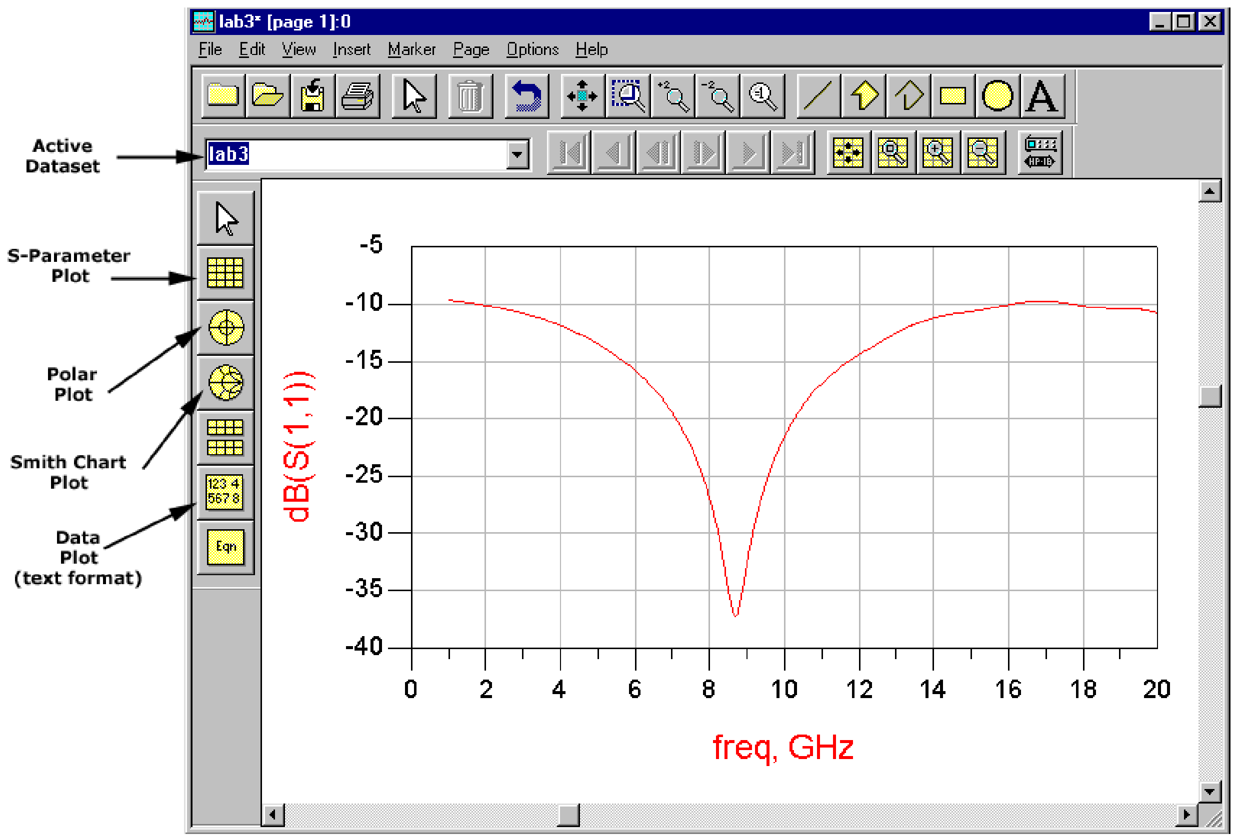

Figure 4 *Schematic Window and Library / Elements.*

.

13. In this part of the lab you are to design a wireless local area network (WLAN) communication system (see Fig. 5) which use two antennas in parallel. You can model these antennas using equivalent circuits defined as Z_{L1} = 82 Ω and Z_{L2} = 36 Ω. These loads need to be matched to a transmitter with an internal impedance $Z_g = Z_o$ = 50 Ω.
    - In ADS design a matching network consisting of **2 quarter-wave transformers MLIN**, to achieve Z_{in}= 50 Ω at the input impedance for the transmitter (represented as Port 1).
    - The substrate is to be FR4 as defined above in this lab guide.
    - Use the fact that Z_{in1} and Z_{in2} are in parallel.
    - Design frequency; if you are group #X, your design frequency is 1.X GHz. For example, for group #3, the design frequency for your circuit is 1.3 GHz.
    - Make sure that you have a minimum length of about 3 cm for TL3. This ensures space for connector soldering at each of the four ports.

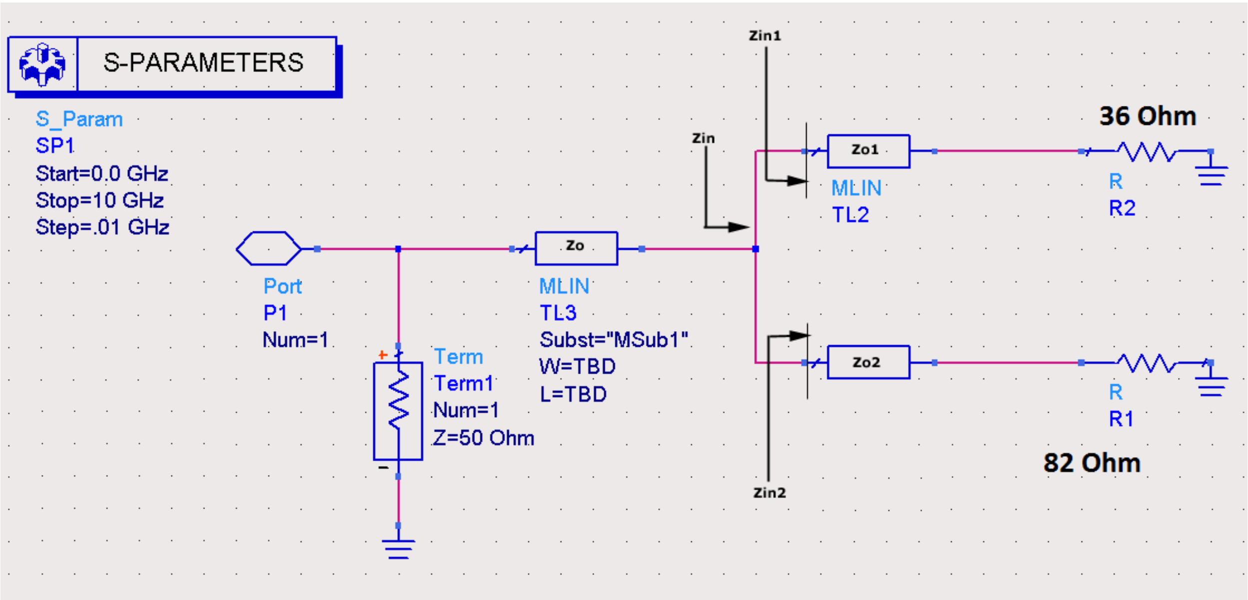

Figure 5 *Representative Wireless Local Area Network (WLAN) Communication System.*

14. Implement the WLAN circuit and simulate its frequency response in terms of S11.

15. The next step in the design process will be to generate the *layout* of your circuit. This consists of the coordinates of the metallization. Basically the physical realization of the matching circuit. The layout will then be sent to another ADS module called Momentum.

During a full-wave simulation, Momentum will solve Maxwell's equations to determine the physical currents flowing on the circuit. This specialized piece of software uses a numerical technique known as the *method of moments* to solve Maxwell's equations, which is well beyond the scope of this course. In fact, Momentum will not even be explored extensively in this lab. All you need to know is that Momentum will discretize the metal into a finite number of points, known as a *mesh* and then solve Maxwell's equations during a full-wave simulation.

16. The first step is to generate a Momentum layout of your schematic by clicking on Layout → Generate/Update Layout. Accept any dialogue boxes that may appear. A new Momentum window will be generated that shows the artwork associated with your design, as illustrated in Fig. 6.

17. You might notice that the generated layout structure in Fig. 6 looks a little misplaced. This is because improper *physical* connections were included between TL1, TL2, and TL3. This can be corrected by including microstrip T-junction (**MTEE**) and two transmission line corner bends (**MCORN**) as shown in Fig. 6 for the circuit schematic.

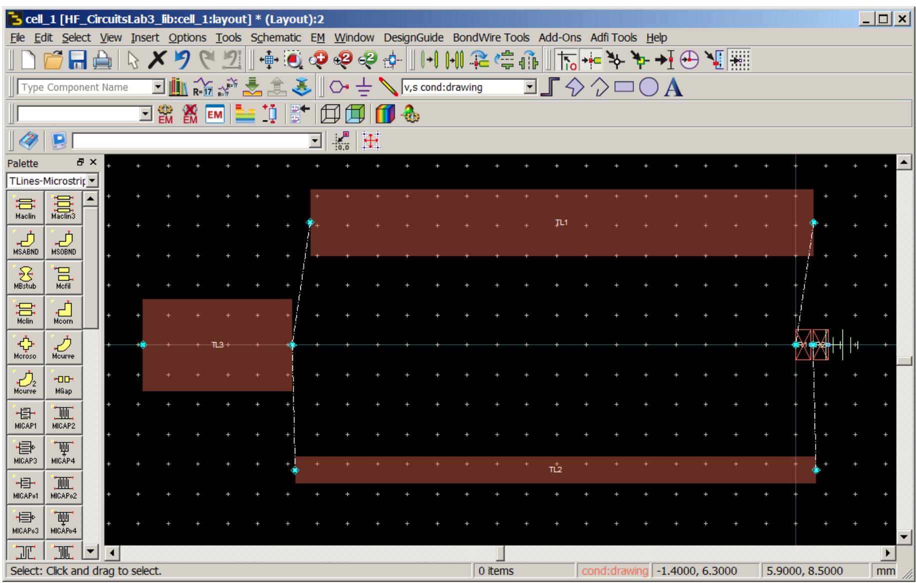

Figure 6 *Momentum layout of the matching circuit.*

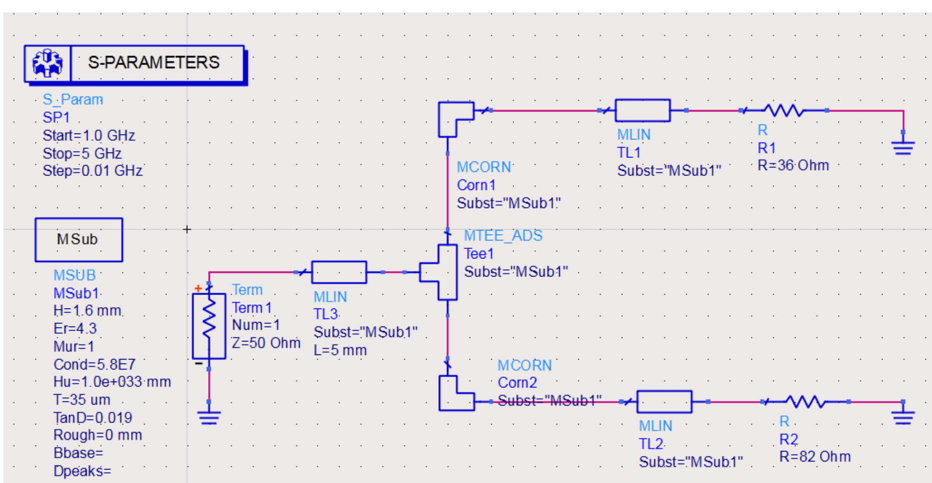

Figure 7 *WLAN circuit with bends and a microstrip T-Section.*

18. Re-simulate your design in the schematic window and record your results. You should notice that the S11 minimum might be shifted because of the added transmission line lengths of **MTEE** and **MCORN**. Given this, you need to optimize your structure further such that your required design frequency is still achieved. Thus you need to tweak your circuit in the schematic window. Continue until you are satisfied with the results. You are free to make use of the other transmission line bends, T-section elements, and alike.

19. Once you have completed the previous step, re-generate the momentum layout of your circuit. You will also need to delete and ports, resistors blocks, and ground connections at this stage.

20. Now generate a .dxf layout file. The laboratory technician will use this file format to fabricate your circuit (to be measured in the upcoming weeks). From the Momentum menu, click on File → Export. Under FILE TYPE, choose DXF (HIERARCHICAL), take note of the default file name (don't change it), then use the BROWSE button to save the .dxf file in a convenient location where you can easily access it afterwards (ADS buries the file deep in a directory location otherwise).

Once this is done, **send an email to the teaching assistant**, attach your .dxf file, and be sure to include your last names in the email message.

Measurement:

Using the provided lab kits, measure the matching circuit and compare with your simulation results. Prior to your measurements, ensure that your VNA has been properly calibrated.

Lab Report:

Lab reports are to be completed and submitted to EPSSO by days end. No excuses, No exceptions. Detailed lab dates and submissions will be posted on the website.

One typed lab report per is required for each two-person lab group team. The report should be typed and not more than 2 or 3 pages including figures and a short description where necessary. This excludes, an appendix (should it be needed) and a cover page, etc.

Please note that screen shots from the ADS simulator, measurement results, and photographs of your fabricated circuit are encouraged in your report.

The lab document should really just be a summary of you comparing your final simulations of the matching circuit with your measurements. You must also state your design frequency and comment appropriately.