Xidian University & Heriot-Watt University

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

(B39HF | B31HD)

High Frequency Circuit Design

Team Members:

Name	XDU ID	HWU ID
Wenchuan Zhang	21012100092	H00392711
Yirui Fu	21012100086	H00392656
Fengliang Mi	21012100040	H00392671
Yihong Yang	21012100009	H00392698
Yiqian Zhang	21012100052	H00393159
Zhuoyu Wu	21012100038	H00392691
Xunchi Ma	21012100015	H00392669
Yang Yu	21012100011	H00392718

1. Introduction

ADS software provides a wealth of capabilities and tools for circuit simulation, electromagnetic simulation, mixed-signal simulation, system-level simulation, and more to help engineers design, optimize, and validate a wide range of electronic devices and systems. It supports a variety of model libraries, including various device models and transmission line models, while also providing powerful layout and routing tools.

The primary goal of this experiment is to increase understanding and familiarity by using ADS software. Secondly, the experiment aims to design and simulate a high-frequency circuit, including a quarter wave matching segment, and evaluate the performance of the designed circuit using a vector network analyzer (VNA) based on a USB interface.

2. Theory

The working principle of the quarter-wavelength transformer is based on the principle of impedance matching, which realizes the transmission and conversion of electrical energy by matching the impedance of the input and output ports.

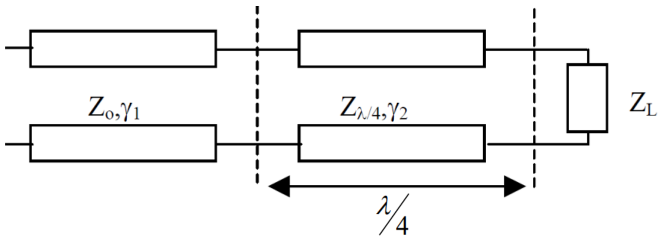

Figure 1 *Basic schematic of a quarter-wave transformer.*

According to the input impedance, the corresponding Γ_{in} is

$$\Gamma_{in} = \frac{Z_{in} - Z_0}{Z_{in} + Z_0}$$

To match the impedance, $\Gamma_{in} = 0,\ Z_{in} = Z_0$,

$$Z_{in} = \frac{Z_{\lambda/4}^2}{Z_L} = Z_0;\ Z_{\lambda/4} = \sqrt{Z_0 Z_L}$$

A Microstrip is a commonly used high-frequency transmission line whose structure consists of a flat conductor line and a large ground parallel to it. Microstrip line is a kind of transmission line that is easy to manufacture and design and can be widely used in microwave circuits.

The width and length of microstrip wire are two important parameters that affect its electrical properties. The width determines the impedance of the microstrip line, and the length determines the transmission characteristics of the microstrip line, such as transmission loss and group delay.

For a microstrip line with a common dielectric constant, its width can be calculated by the following formula:

$$W = \frac{2h}{\sqrt{\varepsilon_r + 1}} \times \left[\frac{\varepsilon_r + 1}{2\varepsilon_r} + \frac{1}{2}\left(\frac{\varepsilon_r - 1}{\varepsilon_r + 1}\right) \times \left(1 + \frac{12h}{W}\right)^{-\frac{1}{2}} \right]$$

Where W is the width of the microstrip line, h is the thickness of the microstrip line, and ε_r is the relative dielectric constant of the medium.

For a given operating frequency, after determining the impedance of the microstrip line, the length of the microstrip line can be calculated using the following formula:

$$L = \frac{v_{Ph}}{2f} \times \left(\frac{l}{\lambda} + \frac{\lambda}{4h}\right)$$

Where L is the length of the microstrip line, v_{Ph} is the phase speed, f is the operating frequency, l is the electrical length when the microstrip line is open at both ends, λ is the propagation wavelength, and h is the thickness of the microstrip line.

Phase Velocity is the speed at which the wave crest or trough moves as the electromagnetic wave propagates through the medium. The phase velocity can be expressed by the following formula:

$$v_{ph} = \frac{\omega}{k} = \frac{c}{\sqrt{\varepsilon_r}}$$

Where v_{ph} is the phase velocity, ω is the angular frequency, k is the wave vector, c is the speed of light, and ε_r is the relative permittivity of the medium.

In microwave circuit design, it is usually necessary to select the appropriate width and length of the microstrip line according to specific design requirements, such as impedance matching, transmission loss and group delay.

Vector network analyzer (VNA) is a test instrument for measuring electrical network parameters. They are essential for the analysis of radio frequency (RF) and microwave components in a variety of passive and active devices, including filters, antennas, and power amplifiers. Network analyzers are ideal for transmission, reflection and impedance measurements as well as S-parameter measurements during design and production.

S-parameters have always occupied the most important position in microwave theory and technology, and they include measurement items that have long been familiar to engineers, such as S11 (input matching), S22 (output matching), S21 (gain/loss), S12 (isolation), etc., and the test results of these measurement items can be easily imported into electronic simulation tools.

In the case of impedance matching, $s11 = |\Gamma|$

$$S11(dB) = 20\, lg(|S11|) = -RL$$

Specifically, if the impedance matching is well realized, then the reflection loss RL should be small, that is, the power loss of the reflected signal is small; The scattering parameter S11 should also be small, that is, the intensity of the reflected signal is small. Under normal circumstances, RL should be greater than 10 dB, S11 should be less than -10 dB, to ensure that the impedance matching achieves better results.

3. Procedure or Experimental Method

1. Open Keysight *ADS*.

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

3. In the schematic window, 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.

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.

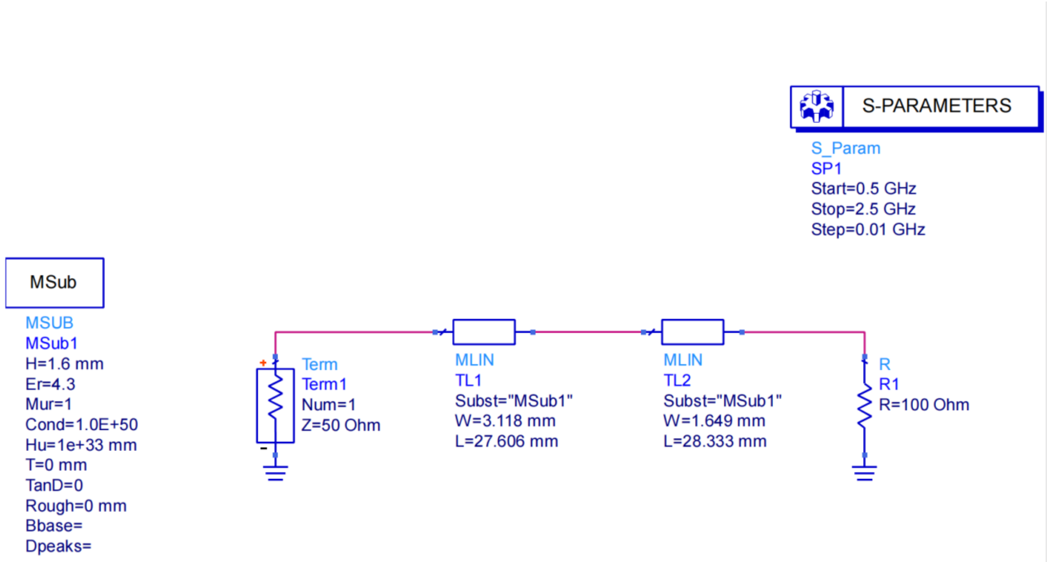

Figure 2 *Schematic Window and Libraries / Elements*

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.

$$\text{the required impedance } Z_{\lambda/4} = \sqrt{Z_0 Z_L} = \sqrt{50 \times 100} = 70.711\ \Omega$$

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.

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 ZO = 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).

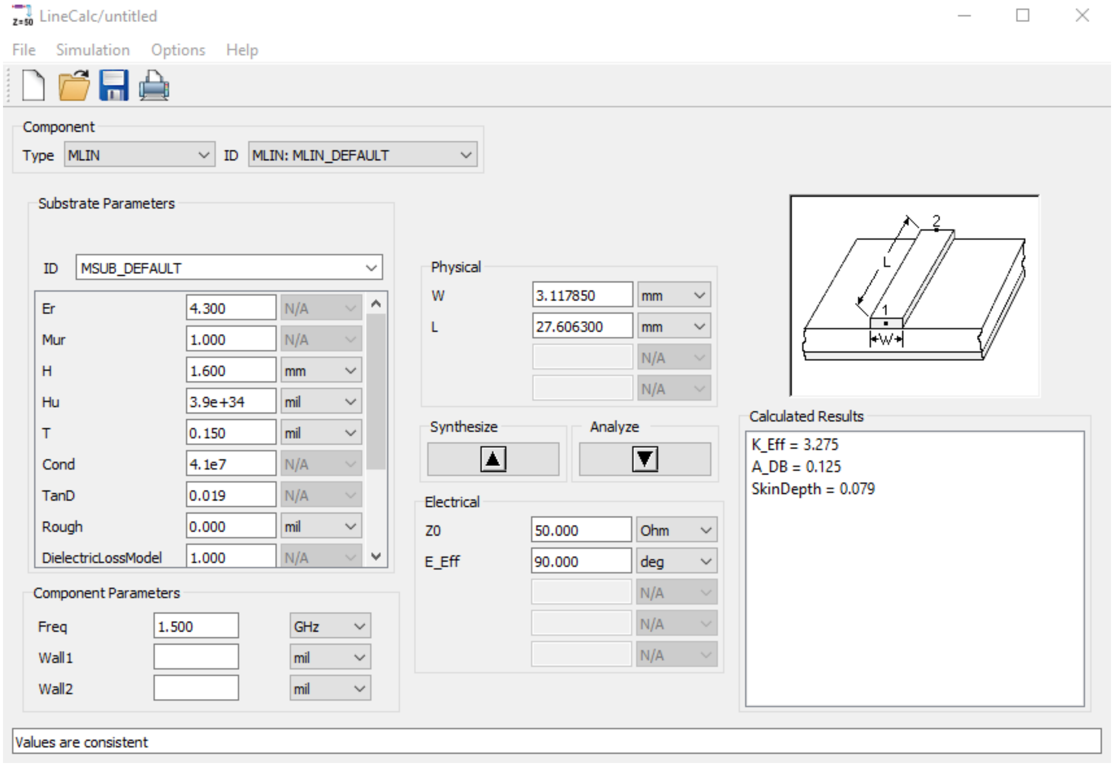

Figure 3 *LineCalc window showing microstrip calculations*

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 λ/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.

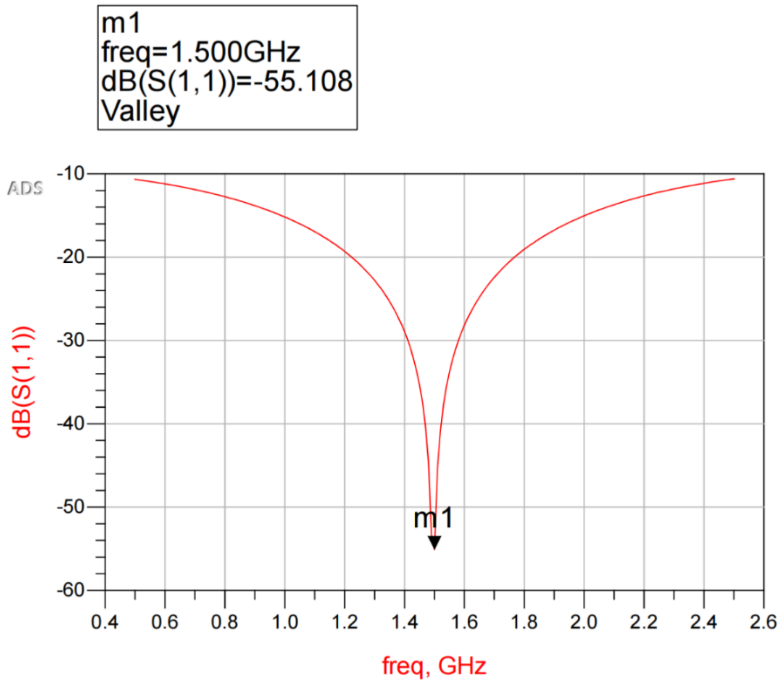

Figure 4 *S-parameter plot*

Yes, the result we was expecting was that the minimum value of the S11 image appeared at the 1.5 GHz frequency. In the previous steps 8 and 9, we chose 1.5 Ghz as one of the parameters to generate the corresponding length and width of the microstrip line. The final circuit should also achieve impedance matching at 1.5 GHz, resulting in a minimum S11. Therefore, we can conclude that 1.5GHz is the best impedance-matching frequency for this circuit.

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.

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 $Z_{L1} = 82\ \Omega$ and $Z_{L2} = 36\ \Omega$. These loads need to be matched to a transmitter with an internal impedance $Z_g = Z_o = 50\ \Omega$.

- 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 group #3, the design frequency for our 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.

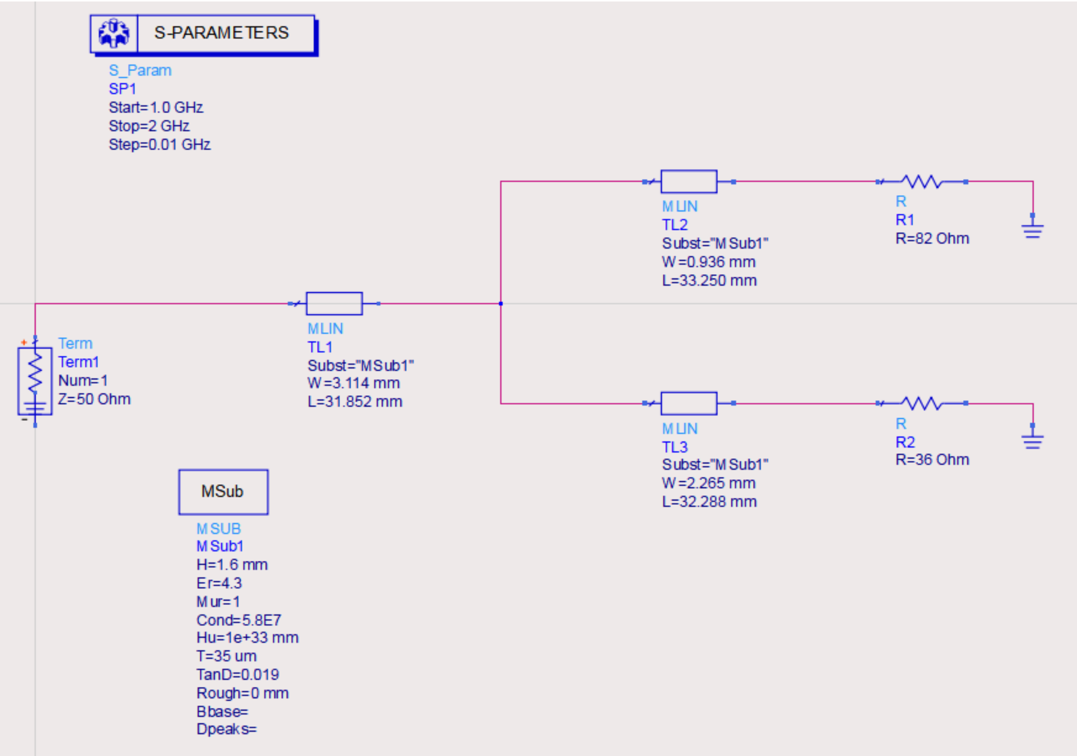

Figure 5 *Representative Wireless Local Area Network (WLAN) Communication System*

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

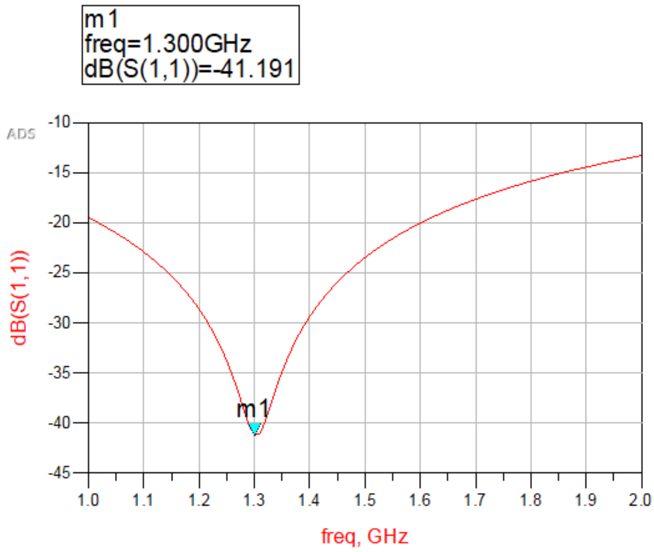

Figure 6 *the frequency response in terms of S11*

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. 7.

17. You might notice that the generated layout structure in Fig. 7 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. 7 for the circuit schematic.

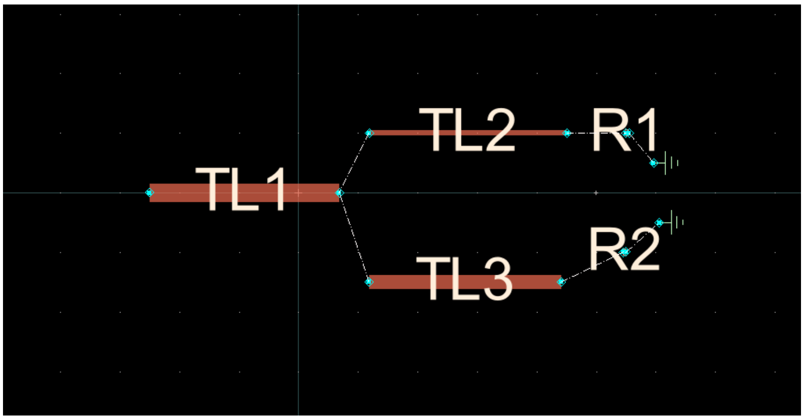

Figure 7 *Momentum layout of the matching circuit*

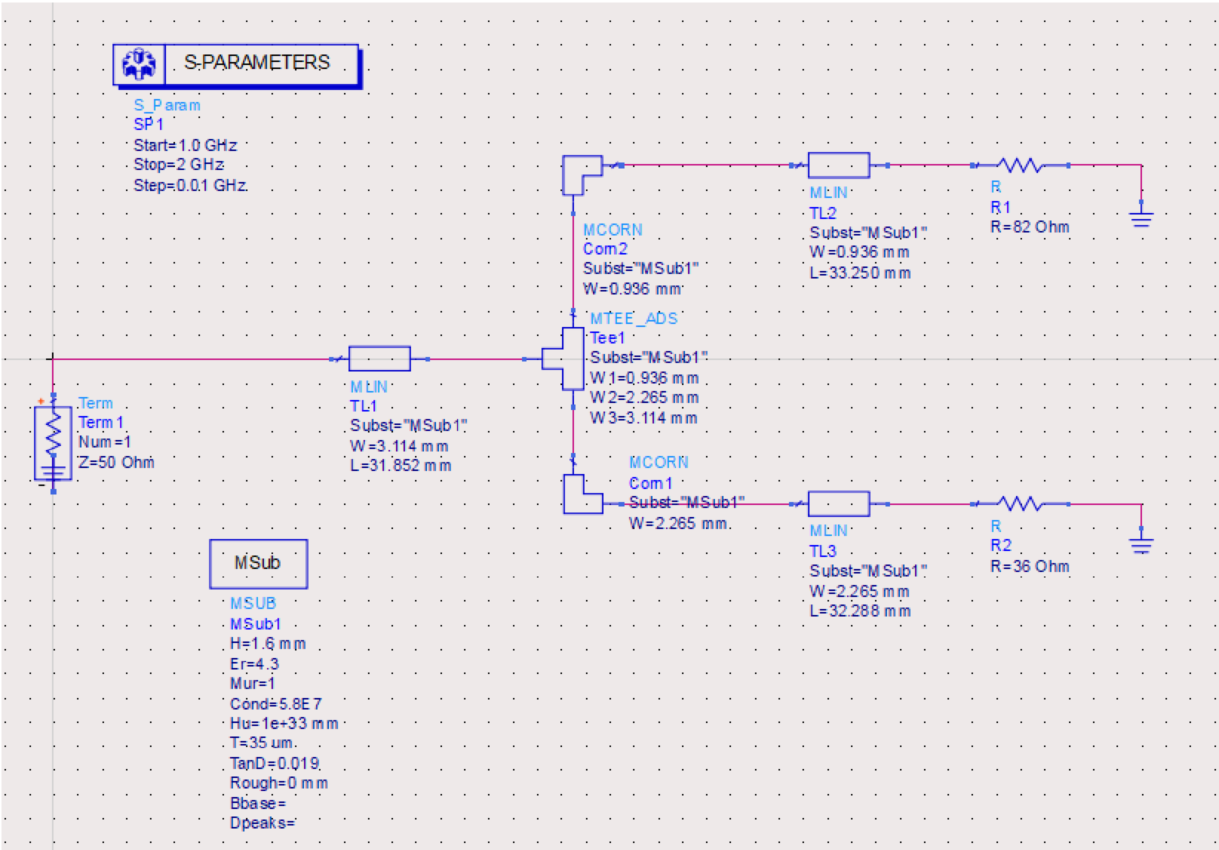

Figure 8 *WLAN circuit with bends and a microstrip T-Section*

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.

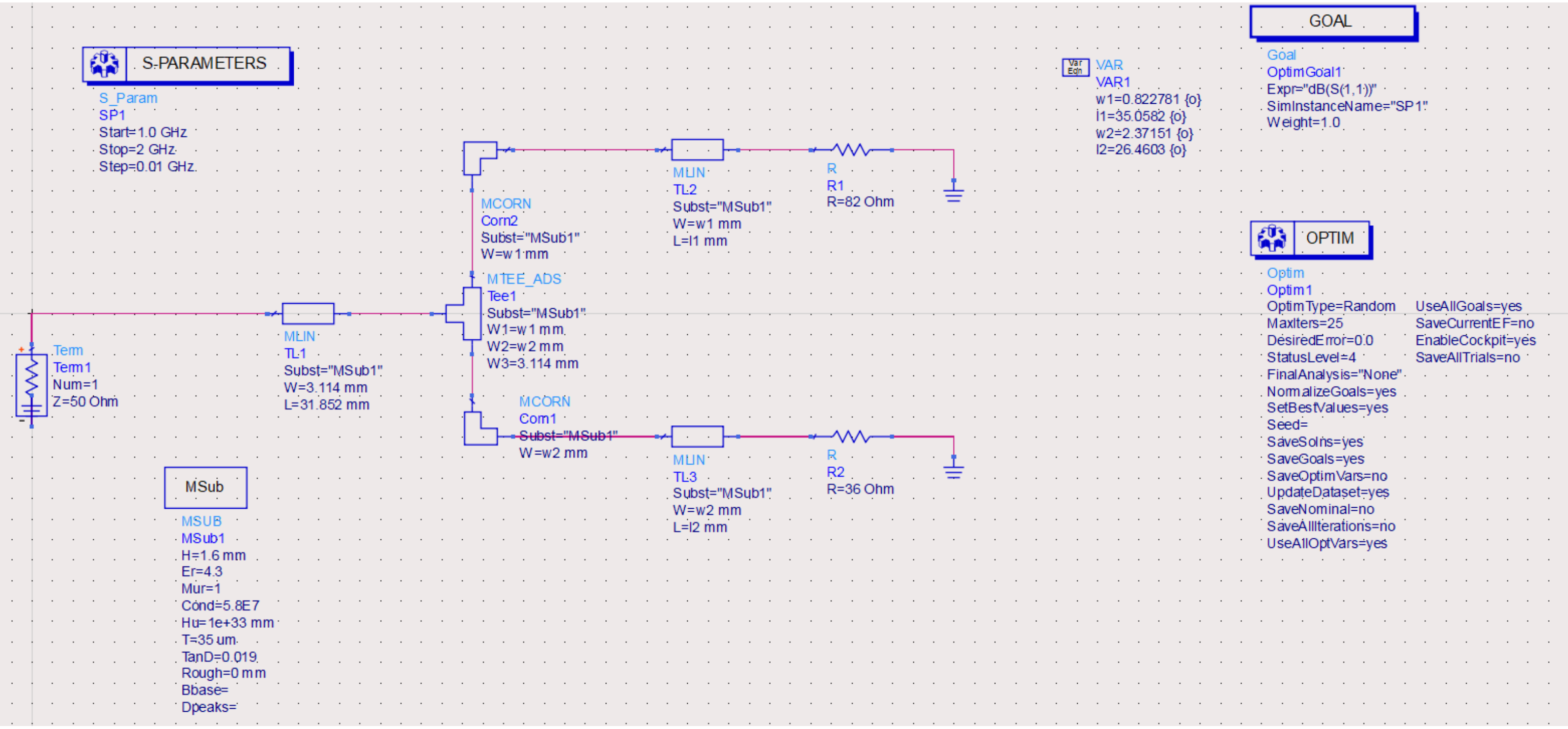

Figure 9 *Developed WLAN circuit*

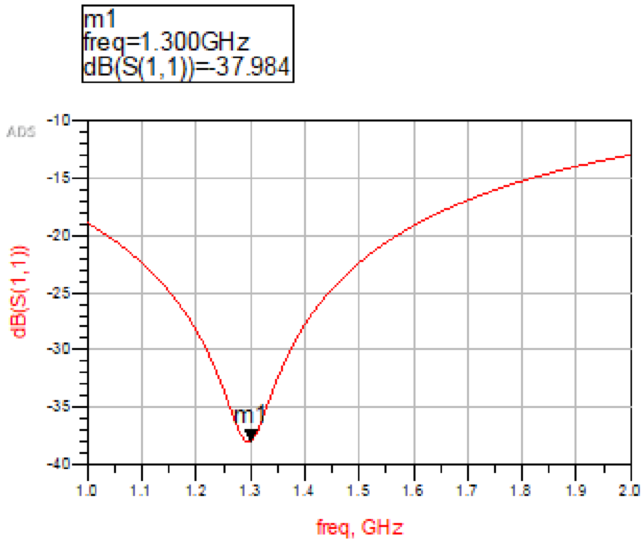

Figure 10 *S-parameter plot after adding the MTEE and MCORN*

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.

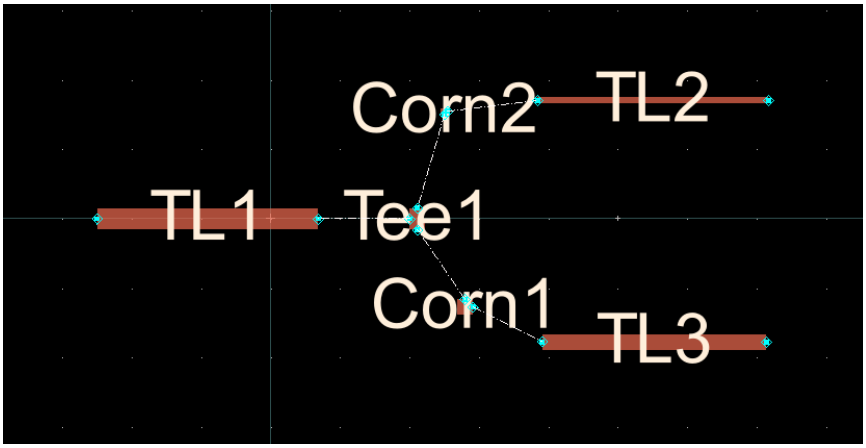

Figure 11 *Momentum layout of the matching circuit.*

4. 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.

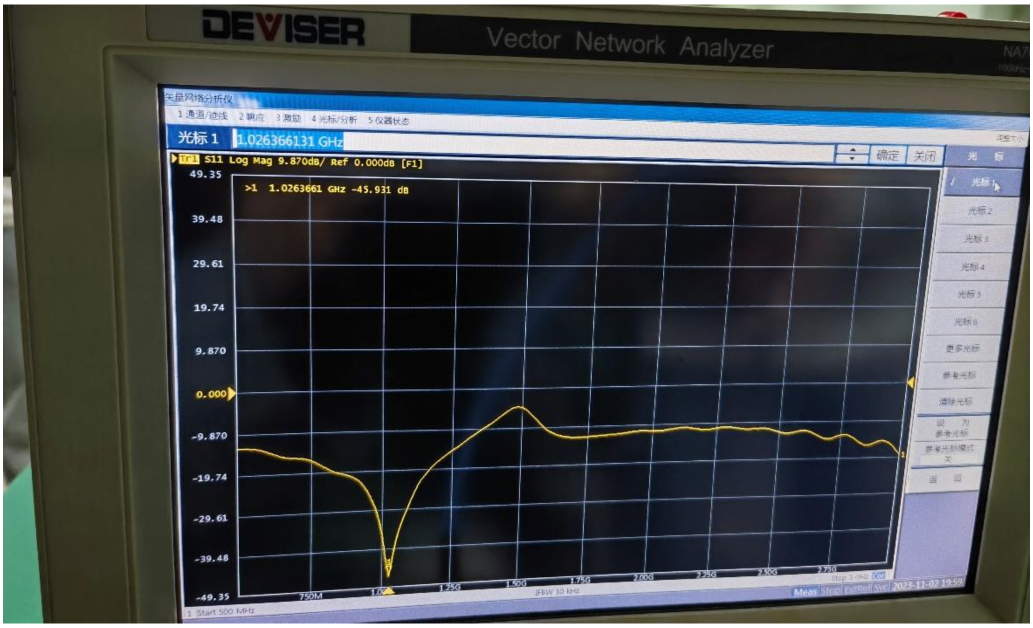

Figure 12 *Observation result*

5. Analysis and Discussion of Results

The simulation result using ADS is that S11 reaches the minimum at 1.3Ghz to achieve impedance matching, but the actual result using the vector network analyzer reaches the minimum at 1.026Ghz. The reasons for this difference may include:

1) Inaccurate model: The component model or parameters used by the ADS may differ from the actual device, causing the simulation results to be inconsistent with the actual measurement results. In this case, you can try to use a more accurate model or adjust the component parameters to get a simulation that is closer to the actual situation.

2) Environmental effects: In the actual test, environmental factors (such as wiring, grounding, interference, etc.) may have an impact on signal transmission that are not considered in the simulation. Therefore, in the simulation, we should try to simulate the actual test environment, and consider the impact of environmental factors.

3) Measurement errors: The vector network analyzer used in the actual measurement may also have measurement errors, such as calibration errors, wire losses, noise, etc. These errors may lead to differences between the actual measurement results and the simulated results. When evaluating actual results, the impact of measurement errors needs to be taken into account and calibrated and corrected accordingly.

Improvements can include the following:

1) Verify model accuracy: Verify the component model and parameters used to ensure that they match the actual device. The accuracy of the model can be verified by comparing it with other reliable data sources or by conducting real-world tests.

2) Consider environmental factors: try to simulate the actual test environment in the simulation, and consider the impact of possible environmental factors on signal transmission. For example, details such as wiring, grounding, etc. are added to more accurately simulate the actual situation.

3) Calibration and correction of measurement results: If there is a large error in the actual measurement results, calibration and correction can be performed. Through accurate calibration of vector network analyzer, reducing wire loss and considering noise, the measurement accuracy is improved.

4) Optimize design: Optimize design based on actual measurement results combined with simulation data. Different matching network topologies, parameter adjustments, etc. can be tried to improve impedance matching performance.

6. Conclusions

This experiment aims to give students an understanding of the principles and methods of matching network design in high-frequency circuits through circuit design and simulation using ADS software, and practical measurement using VNA. The steps include creating the project, placing the components, setting the microstrip line media parameters, adding simulation components and parameters, determining the width and length of the microstrip line using the LineCalc tool, implementing matching circuits, and simulating their frequency response, generating circuit layouts and conducting full-wave simulations using the Momentum module. Finally, the matching circuit is measured by the experimental suite and compared with the simulation results. Through this experiment, students will become familiar with the ADS software, master the matching network design method in high frequency circuits, and understand how to use VNA for practical measurement verification.

7. References/Bibliography

B39HF B31HD Labratory 3 - Matching Circuit Design.pdf

http://pocketvna.com/