ECE 451 Automated Microwave Measurements

Homework #5 Due Wednesday, March 29, 2017

This homework consists of two (02) parts. They are to teach you about RF modeling using CAD tools. You will be introduced to Keysight ADS and Ansys HFSS and will be walked through simple exercises to learn about the tools.

PART I. MODELING MICROSTRIP LINE USING ADS

Introduction

While direct measurement of a device is the most accurate way to determine its behavior, it is often also the most expensive and time-consuming. With the advent of the digital computing age, we now have access to powerful simulation tools that allow for computer-aided engineering (CAE). These tools allow an engineer to simulate a design with extremely high-fidelity so that he or she can optimize a design before fabrication.

In this homework assignment, you will be using Keysight Advanced Design System (ADS) to model a pre-existing microstrip line whose S parameters you measured in Experiment 5. You will use the existing microstrip line models in ADS along with some lumped elements to generate S parameters which will be compared against measurement.

Background

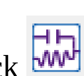
You will be modeling a microstrip line that looks like this (which you measured in Experiment 5):

Figure 1: Microstrip Line of Interest for this Experiment

It is on an FR4 substrate, which is an extremely common substrate due to its affordability. FR4 is a composite material composed of woven fiberglass cloth with a flame-resistant epoxy material that binds it together. A thin layer of copper foil is laminated to both sides of it. This foil is either milled or etched away to form the desired microstrip pattern and then connectors are soldered on.

The connectors for this device are SMA connectors. They are coaxial RF connectors that offer excellent electrical performance from DC to 17 GHz. Typically these connectors are tightened to 8 lb·in with a torque wrench to ensure a good impedance match and minimize reflections.

Procedure

1) Open ADS and create a new workspace. In the main workspace window, click  to create a new schematic and name it “microstrip.” Close the Schematic Wizard Window that pops up.

2) First, you are going to create the microstrip substrate. This item defines the parameters of the substrate that the microstrip line is on (permittivity, loss tangent, thickness, etc.). It can be found in the "TLines-Microstrip" library as shown in Figure 2 and it is 

, which can be clicked and placed on the schematic. Set the following parameters:
   a. H = 1.56 mm
   b. Er = 4.8
   c. T = 0.02 mm

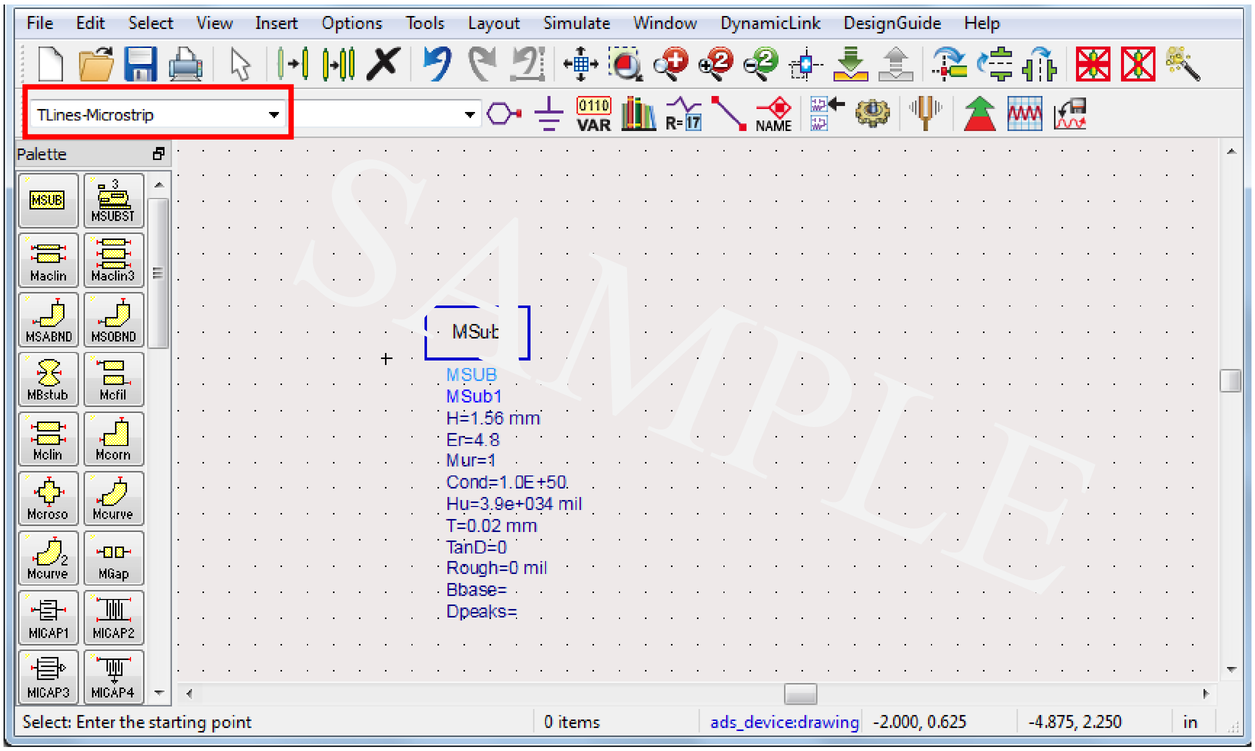
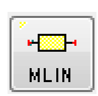


Figure 2: Creating the microstrip substrate. Library selection is boxed in red.

3) Next, find the microstrip line in the same library: 

. Place five of them down on the schematic as shown in Figure 3. Give them the same widths and lengths as shown in the figure. Each one of these represents a segment of the microstrip line that we are modeling. We measured the width and length of these microstrip lines with a caliper for your convenience.

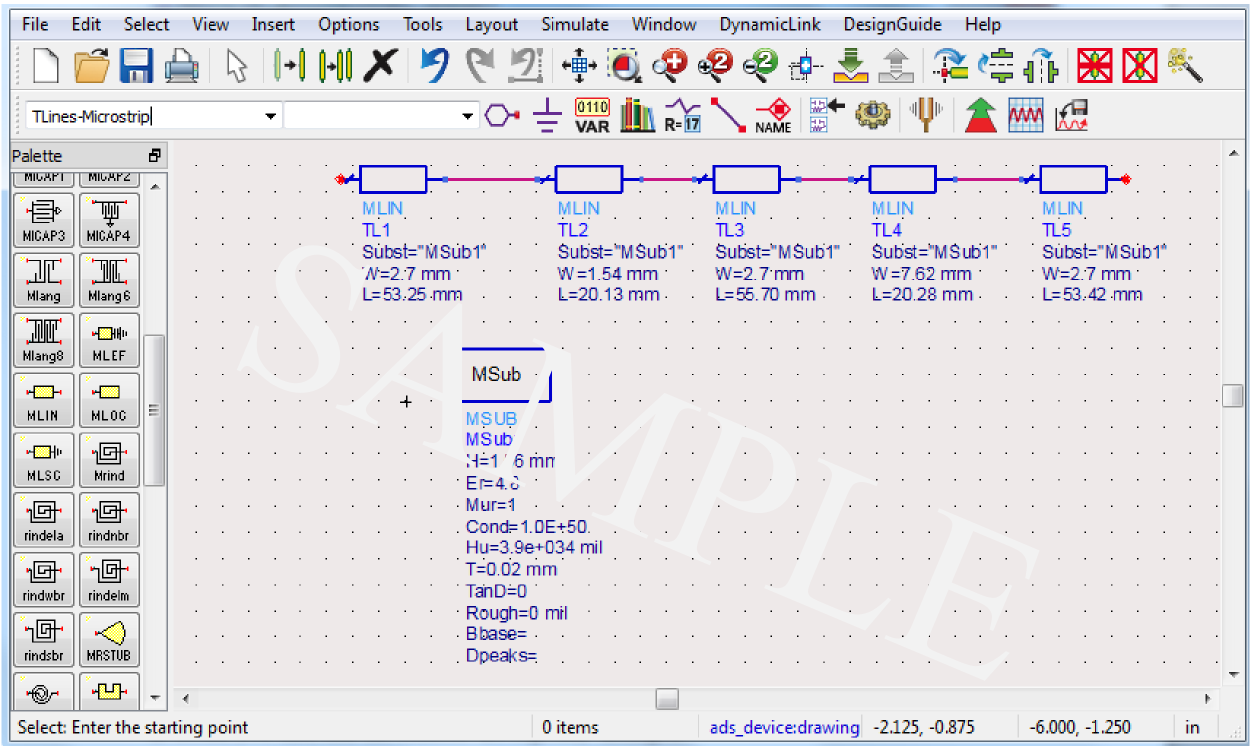
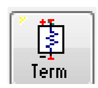
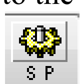


Figure 3: Placing the microstrip line segments.

4) Now, switch to the "Simulation-S_Param" library. Click on 

and place two of them then click on 

and place one of them as shown in Figure 4. The terminations are like the measurement ports on a network analyzer and the "SP" block is the simulation controller.

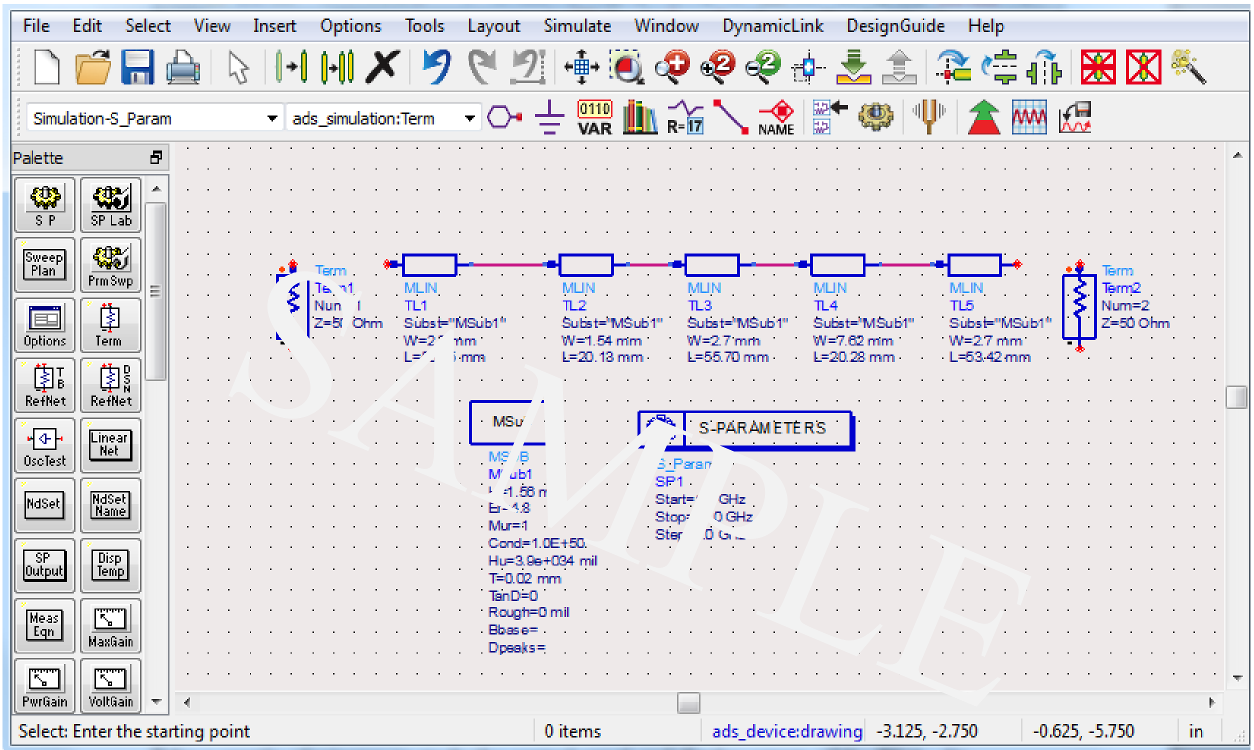
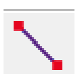


Figure 4: Adding S-parameter Terminations and Simulation Controller.

5) Double click on the "S-Parameters" Block and set the parameters as shown in Figure 5. Then use  and  to complete the circuit as shown in Figure 6 to complete the circuit. Then press  to simulate the circuit. Doing this generates the S parameters for this circuit. **Print this schematic out and include it with your homework.**

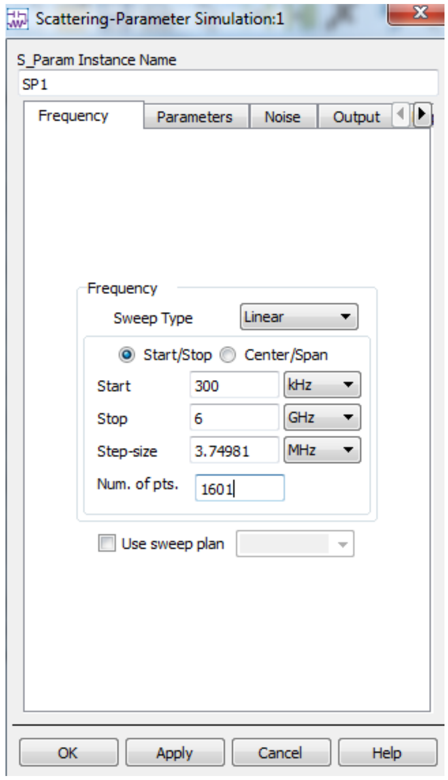


Figure 5: Setting the S-parameter simulation block parameters.

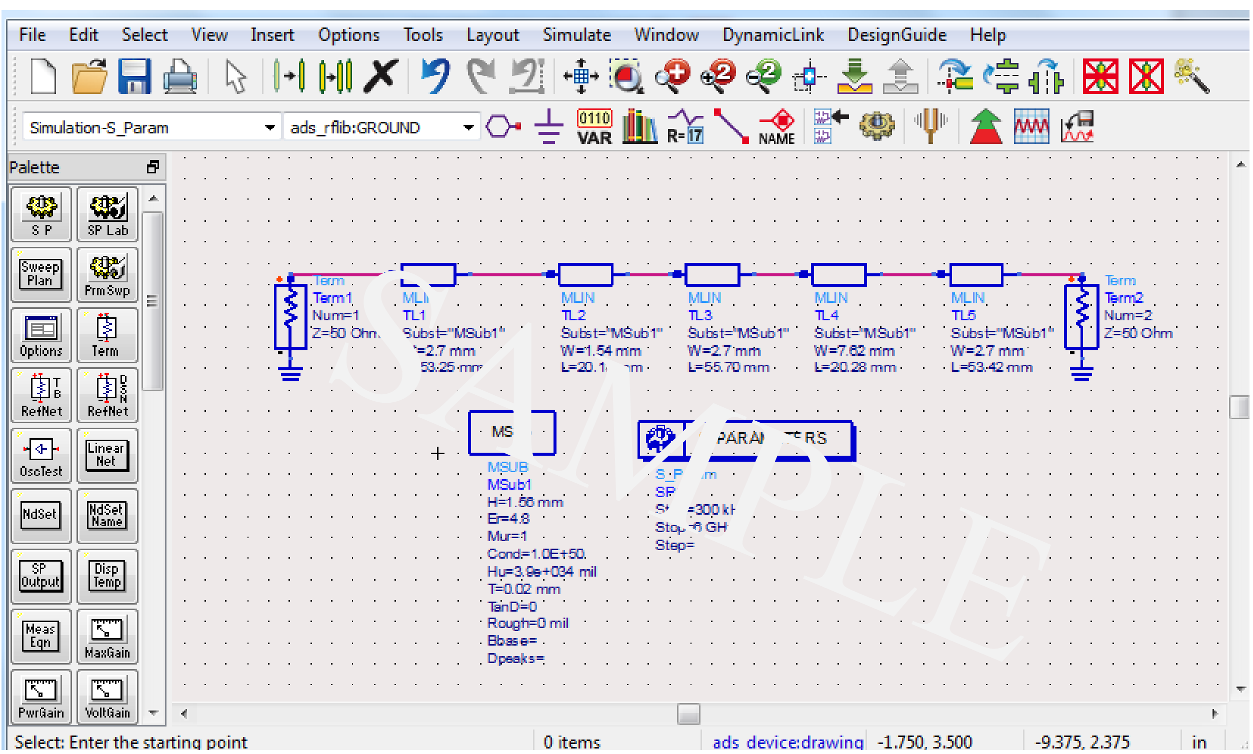


Figure 6: The circuit ready for simulation.

6) When the data display window pops up, use the datafile tool to import the "Touchstone" S-paramter you measured in Part 2, Experiment 5. Plot S_{11}, S_{21}, S_{12}, and S_{22} in dB for both datasets as shown in Figure 7.
7) Notice that we have excellent match for S_{11} and S_{22}, but a poor match for S_{21} and S_{12}.

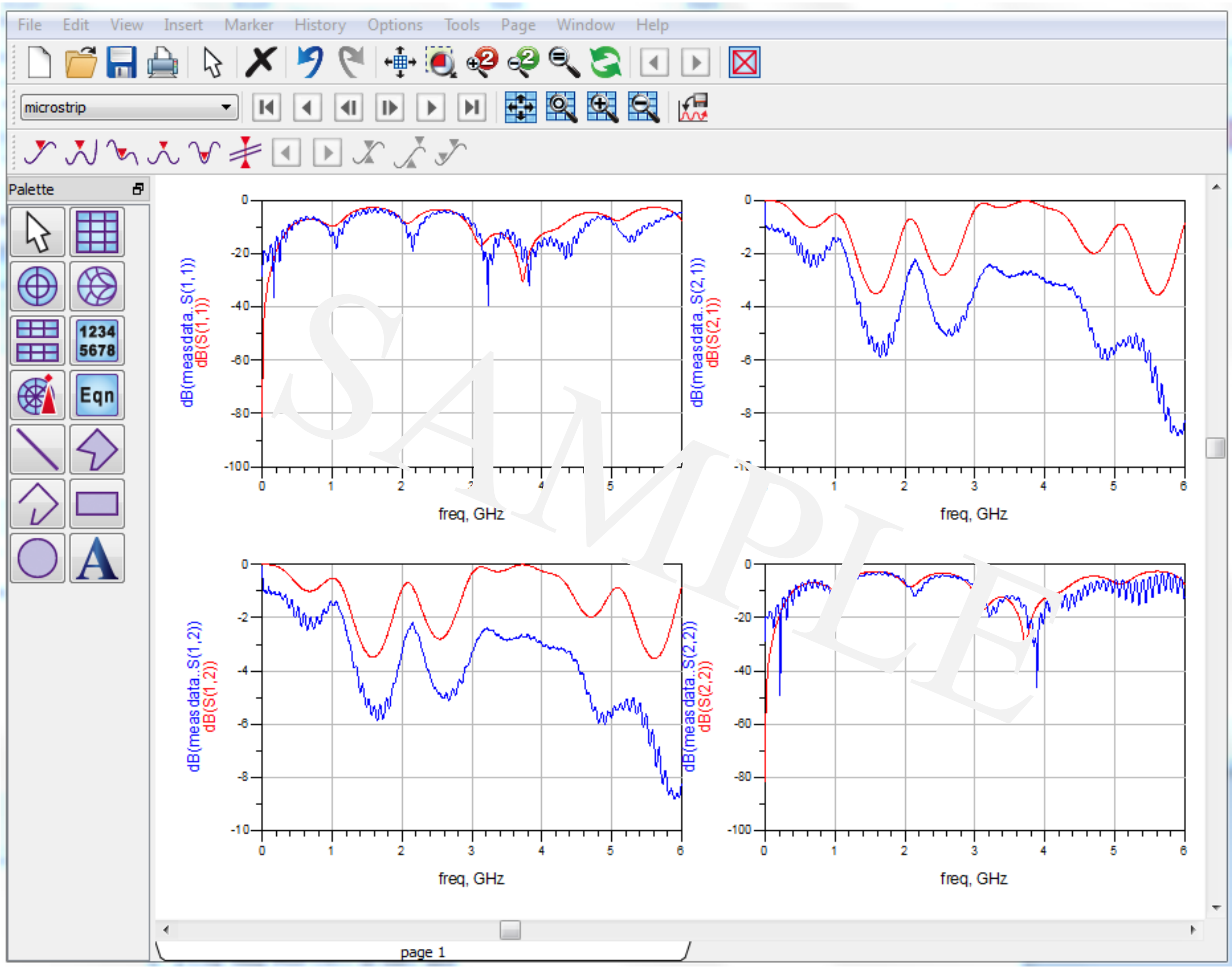


Figure 7: Data display comparing simulated data with measured data. The S_{11} and S_{22} match reasonably well, but the S_{21} and S_{12} do not match.

Improving the Model

8) We now attempt to improve the model by providing a simple lumped element model for the SMA connector. Our simple model for an SMA connector is a series resistor and inductor and a shunt capacitor, as shown in Figure 8. Lumped elements can be found in the "Lumped-Components" library. The inductor lumped element model also contains a series resistance parameter that can be used instead of having an actual series resistor, as the figure denotes. An SMA Connector Model should be made for both the input and the output, as shown in Figure 9. Good values for the inductor, resistor, and capacitor are 6.4 ohms, 2 nH, and 0.5 pF, respectively. Use the tuning feature in ADS to tune your R, L, C values in order to

optimize your lumped model. The link below is to a video that describes how to tune your parameters to optimize the match of your model to your measurements in ADS.

Tuning instructional video:
https://www.youtube.com/watch?v=OcHV9qFGy1U

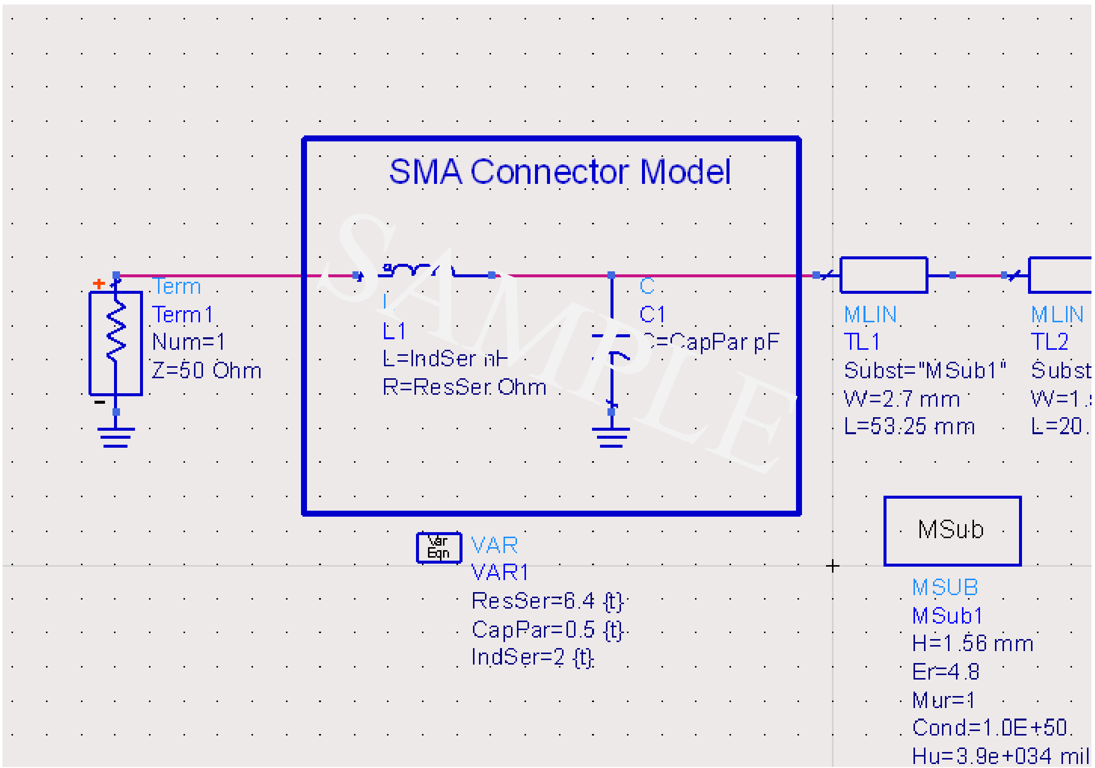


Figure 8: Microstrip line model with inserted SMA Connector Model. The series resistance is being modeled with the inductor model. The box and label around the model were added for clarity purposes and do not affect the simulation.

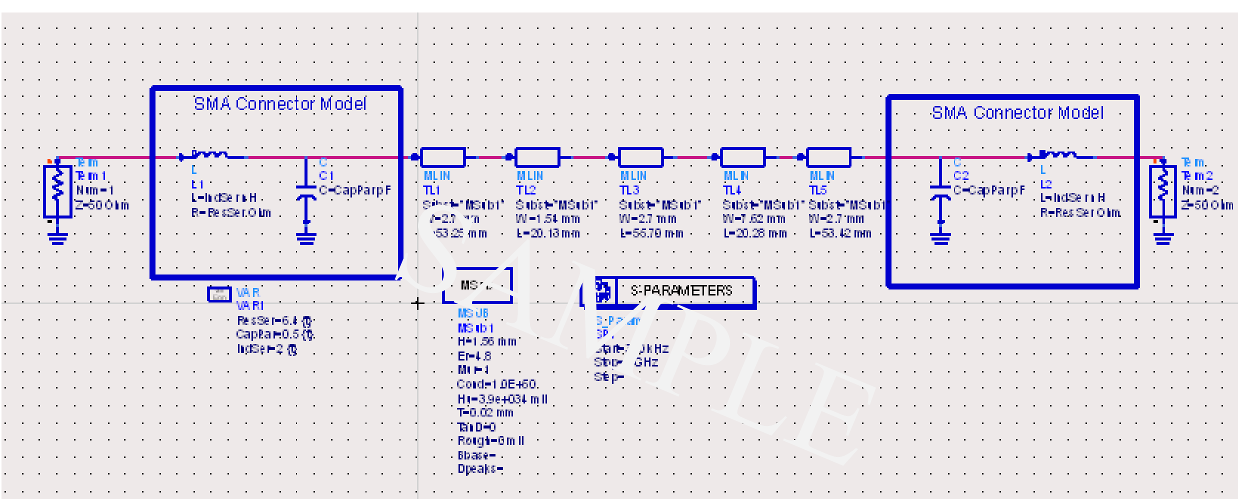


Figure 9: The completed microstrip line model included the SMA connector models at both the input and the output.

Discussion

9) These values should improve your match, particularly for lower frequencies, but they will not be perfect since there are higher-order effects at work that cannot be accurately described with such a simplistic model. There is always a tradeoff between the sophistication of a model and how accurate it is. The benefit of generating an accurate model is that it can have some useful predictive power that might save design time in the future. Be cautious in your use of tuning and optimization algorithms. It is very easy to depend on the simulation power of your software while disregarding the fundamentals that you have learned, leading to much wasted time. It can also be tempting to over-optimize your model, wasting time perfecting something that only needs to work roughly. Gaining work experience and seeking the advice of those who have this experience will give you insight on how to find the best balance for these concerns.

   For further study, here is a link to a paper describing a very sophisticated model of the SMA connector. Feel free to explore it if you are interested:
   http://www.cst.com/Content/Documents/Articles/article434/CST_ITEM_2008.pdf

Conclusion Questions:

1. How does an electromagnetic wave propagate through a microstrip line?
2. Briefly explain how a microstrip line differs from a coplanar waveguide?
3. How did adding the lumped element model of the SMA connectors affect our match to the measured data? After tuning, what R, L, C values gave you the optimized match of your model to measurements?
4. What might account for the differences between simulation and measurement? What aspects of the real device were not modelled?
5. Over which frequency range is our model accurate? Why do you think this is true?

PART II. INTRODUCTION TO HFSS

Go through the HFSS tutorial available from the ECE451 course website. Attach screenshots of your design, as well as your plots with comments/observations. Also, export the profile of your simulation (HFSS→Results→Solution Data → Profile). You should be able to see the adaptive passes, the matrix size, the elapsed time, etc in the profile. Attach the profile as an appendix at the end with your homework.

Theory and Conclusion Questions:

Theory:

1. What is HFSS? Describe the benefits, features, applications of HFSS and why it is used so much in industry today?
2. What are the differences between stripline and microstrip line? Where do they commonly exist and what are the advantages (and disadvantages) over the other.
3. What are the non-idealities of the interconnect at high frequencies, and how can they be modeled accurately?

Conclusion:

1. Look at your simulated S-Parameter matrices from the HFSS tutorial
   a. Is this two port reciprocal? Why or why not?
   b. Is this two port lossless? Why or why not?
2. Look at your simulated S-Parameter plots from the HFSS tutorial, do you notice any resonances in your plots? If so, at frequencies do they occur? Why do you think this is the case?

   Hint: $\varepsilon_e = (\varepsilon_r+1)/2 + ((\varepsilon_r-1)/2)\frac{1}{\sqrt{1+12*d/w}}$

   $V_p = \frac{c}{\sqrt{\varepsilon_e}}$ **where d/w=substrate_H/trace_W**