ECE 451

Automated Microwave Measurements Laboratory

ADS Tutorial On TRL Design

February 14, 2017

Before going through this tutorial, please refer to the following link to learn the basics for using ADS:
https://www.youtube.com/watch?v=04BfajzXnmM

- Follow the instruction in the video to creat a new workspace. Select "New Schematic Window" and draw the components for you TRL standard as shown in Figure 1. Note the numbers presented here are **NOT** for this design problem.

  For this design, we suggested you to creat a custom MSUB (microstrip-substrate) for MLIN (microstrip-line). Since MLIN could only resemable the top copper trace you desinged for the microstrip, we need to add in MSUB as a metal-backed substrate plane. In other words, MSUB is created to take in account the effects of the FR4 substrate. Please check the data sheet for your selected substrate and fill in the proper MSUB values (e.g. thickness, height, etc).

- As we know that the characteristic impedance (Z0) of a transmission line depends on its geometry factor, we need to obtain the correct geometry parameters of our trace line. In this case, we need to match Z0 to 50 Ohm as the measurmenet system in the lab takes 50 Ohm as the reference. To do so, we perform LineCalc by selecting "Tools –> LineCalc –> Start LineCalc" and Figure 2 should pop up. LineCalc is an integrated tool in ADS that help us to calcualte the geometry of the line we desired. More info about how the geometry parameters effect Z0 can be found in this webstie: https://www.microwaves101.com/encyclopedias/microstrip.

  Input each entry under "Substrate Parameters" as you did for MSUB. Also, remeber to change the frequency to a value between 1 to 6 GHz based on your choice. Then fill in Z0 and E_Eff (electrical length) for the TRL standard. Click on Synthesize and make a note for the physical lengths
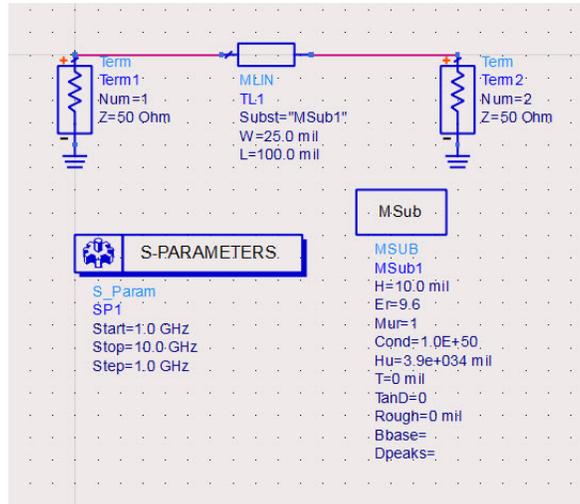
Figure 1: ADS Schematic for the THRU standard

ADS calcualted. Go back to your schematic and change the dimensions for your line with the new length and width.

- For sanity check, we recommand you to simulate and observe the s-parameters of your microstrip in data display window. The detailed steps can be found in the video link.

- After finishing the design for a TRL standard set, your final schematic should be similar to Figure 3. Again, the values in the example are **NOT** for your reference.
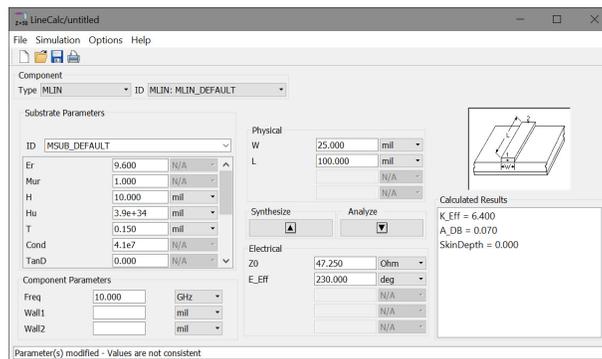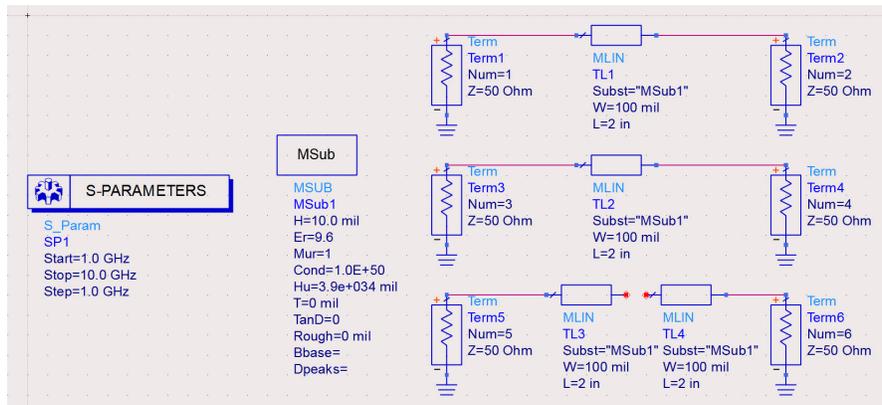

Figure 2: Window for LineCalc
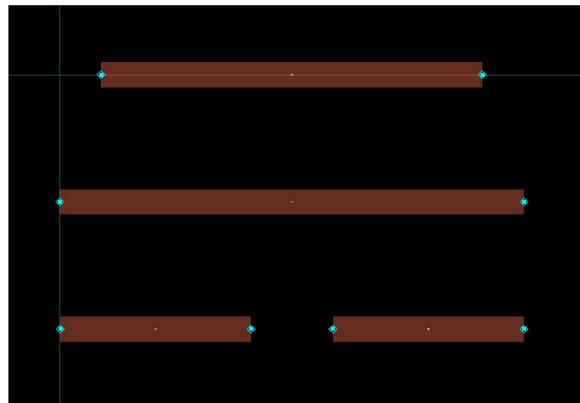
Figure 3: Schematic for the TRL design

Figure 4: Board Layout for TRL

- Generate the board layout by clicking "Layout –> Generate/Update Layout...". Now your layout should look like Figure 4.

- Save your file by clicking "File –> Export". Rembmber to change the file type to "Gerber/Drill". This should automatically give you a file named "cond.gbr" . Rename it and send a copy to your section TA before the due date.