

## ADS Tutorial On PCB Design to Layout

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 create 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 create a custom **MSUB** (microstrip-substrate) for **MLIN** (microstrip-line). Since **MLIN** could only resemble the top copper trace you designed 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 ( $Z_0$ ) 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  $Z_0$  to 50 Ohm as the measurement 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 helps us to calculate the geometry of the line we desire. More info about how the geometry parameters effect  $Z_0$  can be found in this website: <https://www.microwaves101.com/encyclopedias/microstrip>.

Input each entry under “Substrate Parameters” as you did for **MSUB**. Also, remember to change the frequency to a value corresponding to  $E_{\text{Eff}}$  (electrical length) of your design. For instance, if you want the trace to be a quarter-wavelength at 1GHz, you should fill in “Freq” with 1 GHz and  $E_{\text{Eff}}$  with 90. Click on “Synthesize” and make a note for the physical dimensions that ADS calculated. Go back to your schematic and change the dimensions of your line with this length and width.

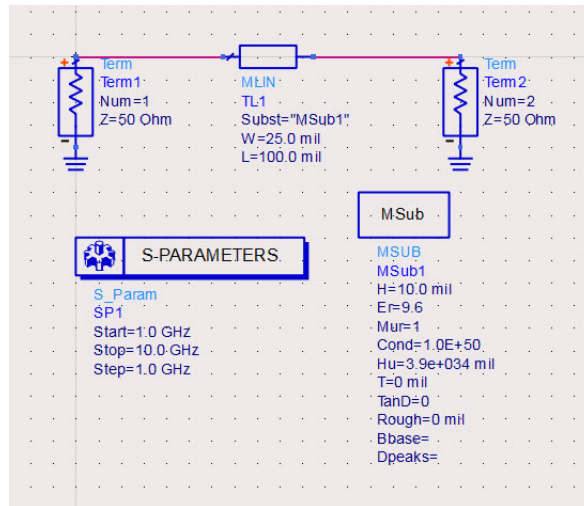


Figure 1: ADS Schematic for the THRU standard

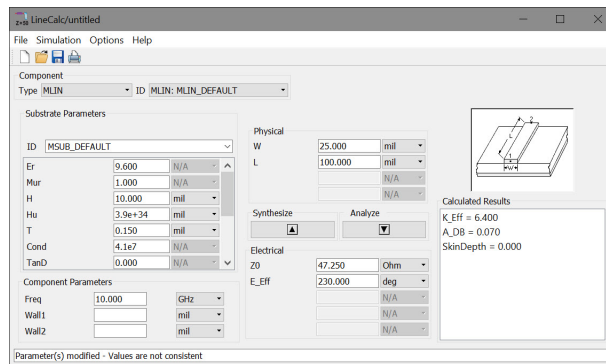


Figure 2: Window for LineCalc

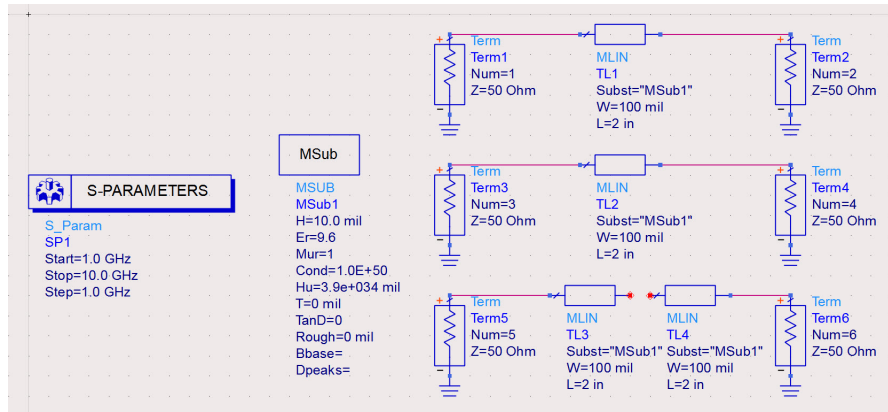


Figure 3: Schematic for the TRL design

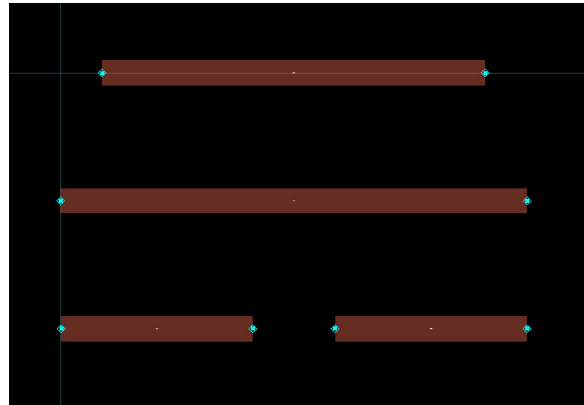


Figure 4: Board Layout for TRL

- For sanity check, we recommend 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 above.
- Below, we will show an example designing a TRL set. After finishing the design, your final schematic should be similar to Figure 3. Again, the values in the example are **NOT** for your reference.
- To send the design to E-shop for fabrication, you need to generate the board layout by clicking "*Layout -> Generate/Update Layout...*". Now your layout should look like Figure 4.
- Those cyan dots indicate you have put a port in your schematic, it would not matter during fabrication. Save your file by clicking "*File -> Export*".

Remember to change the file type to “Gerber/Drill”. This should automatically give you a file named “cond.gbr”. Rename it properly and send a copy to your section TA before the due date.