ADS2008 Tutorial 1

Objective: Match a 100 Ω load to a 50 Ω microstrip line using a Quarter Wave Transformer at 2.4 GHz. Measure S11 as a function of frequency, using ADS. Assume the microstrip lines are printed on a 10 mil TMM substrate with relative permittivity of 3.4.

Steps:

1) Start Advanced Design System Update 2. It should be under start menu. If not, look in C:\ADS2008U2\bin\ads.exe

2) In the main window, select File/New Project. Name it by entering a name in after the default directory (or the directory of your choosing which can be obtained from "Browse"), and click OK. At this point you can specify a standard unit. We will leave this one in mils. You may have trouble creating a project due to write protection in default folder. To fix that, specify a different path, and hit OK.

3) At this point a wizard will pop up, which will assist you in setting up your circuit or simulation. There is no need for it here. Click cancel. This will place you in the main window.

- Observe a pull down menu in the upper left hand corner. It will most likely say 'lumped components'. For our purposes, we'll use 'TLines-Microstrip' and 'S-Parameters'

4) Start with 'TLines-Microstrip'. Your first objective is to define you substrate. Click the MSUB button which is the first one you see in the palette (top left). Aim the MSUB somewhere on the main grid, and left click. Once it is placed, hit 'Esc'

- Esc is used when you do not wish to place any more of the selected items on the grid. It is equivalent to selecting the arrow key from the toolbar menu

- Notice that you can use the wheel of your mouse to zoom in and out of the circuit. If you zoom out too far, you can 'lose' the circuit. To view the entire schematic, click 'view all' icon in the toolbar. This places your entire design in the viewable area.

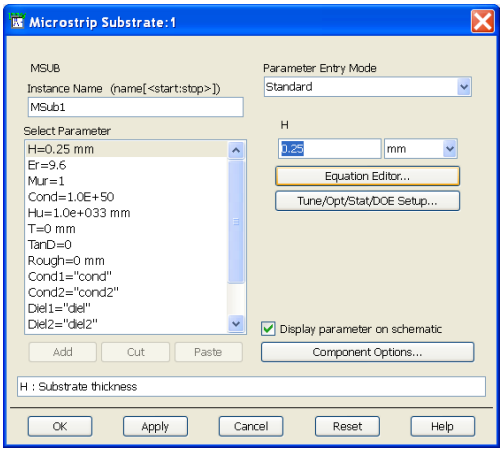
- Right now you are working in '**schematic**'. This is where you lay out all your circuit components, specify different parameters and set up simulations. Naturally, your physical circuit will look quite differently, but more on that later.... Just remember that you are in the schematic mode of your design.

5) Next, you want to edit your substrate. To do so, you can either double click on the rectangle which encloses the 'MSUB' in schematic window, or single click on the 'paremeters' right under it. Double click MSUB and a window will pop up.

- This is where you set the parameters of your substrate. You may select different measurement units as needed.
- 'Select Parameter' window displays the parameters of your substrate. Click on each individual one to see what it corresponds to. For instance, H is the substrate thickness. Er is the relative permittivity, and so on.

6) Set the following parameters

Loss tangent = .002
Conductivity = 5.8e7 S/m
Substrate thickness = 10 mils
Er = 3.4
The rest can remain at default value.

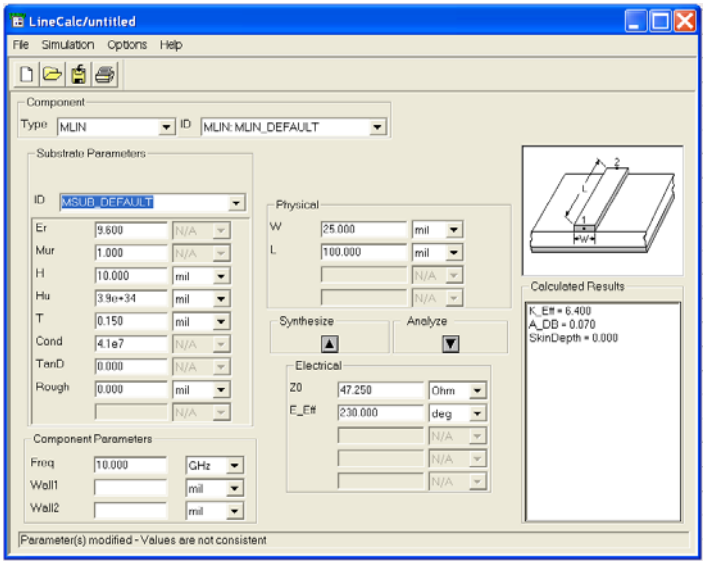


Your circuit will consist of the following:

- *Term* → This is a 50 Ω source impedance. This is where you will measure your reflection coefficient.
- *50 Ω transmission line* of arbitrary length → Remember that if a load is matched, the length of the line will not affect S11. This will connect to your Term.
- *QW transmission line* → This will have Zo of sqrt(100*50), and length of quarter wave at 2.4GHz. Remember that free space wavelength isn't the same the one here, due to the presence of dielectric material. We will rely on ADS to help us tackle this issue.
- *100 Ω load*→ This will connect to the QW Tline.
- *Step*→ Since our two transmission lines will be of different widths, a step must be placed in between the two in order to account for that. Step function adds a scattering matrix that represents the parasitics due to line discontinuities.

7) Let's build our circuit. Start with the 50 Ω line. Select MLIN from the palette and place it in the circuit

8) In the main window, there are tools provided to analyze lengths and widths of Tlines. Select Tools/Linecalc/Start Line Calc. Here you can specify constraints on your line and extract physical parameters. Here are your criteria for the first Tline:



Substrate Parameters:
Er = 3.4
H = 10 mil → height
T = leave default → thickness of Cu strip
Cond = 5.8e7 S/m

Component Parameters
Freq = 2.4 GHz

Electrical
Zo = 50 Ω
E_Eff = whatever, we will see why soon...

Physical
It's what you will read off and then place in your first tline, but not just yet.....

How this works: Given the Substrate and Component parameters, you can either:

- Set length and width, hit analyze, and see your Zo, and E_eff (phase) are….. OR
- Set Zo, and phase, hit synthesize, and output the length and width

9) Set all parameters as specified above for Substrate, Component and Electrical properties. Click Synthesize when done. Notice that when you change E_eff, only the length changes. Since our line is of arbitrary length, it this case E_eff doesn't matter. Set L and W for TL1 in your schematic. Use W that came from your synthesis, and an L of your choice.

10) In the schematic window place another tline in the schematic, to the right of the first one. To make things simple, you can click once, hit CTRL+C, CTRL+V and left click on the grid. Now, return to line calc

and set your parameters for the QW line. They will be the same, except for:
Zo = sqrt(100*50) = 70.71 Ω
E_Eff = 90 degrees, which corresponds to quarter wavelength

Note: E_eff is the phase accumulation of forward traveling wave. Numerically, E_eff = β*L. Thus if β = 2π/λ and L = .25λ, then E_eff = π/2 = 90°

11) Input L and W you have just found into TL2. Remember that line length DOES matter here. As an exercise, you can compare this length to one that would result in free space.

12) Insert Step 

in between the TL1 and TL2. Set the W1 and W2 to whatever the respective widths of the lines are. W1 is the width of Tline to the left of the step and W2 is the width of Tline to the right of the step.

13) Go to lumped components palette and select a resistor. 

Place it on far right. Set the value to 100 Ω.

Then go to 'Simulation-S_Param' palette, and select a term 

Place it on far left

14) Place ground the bottom of both, Term and R1. You can find ground in the main toolbar 

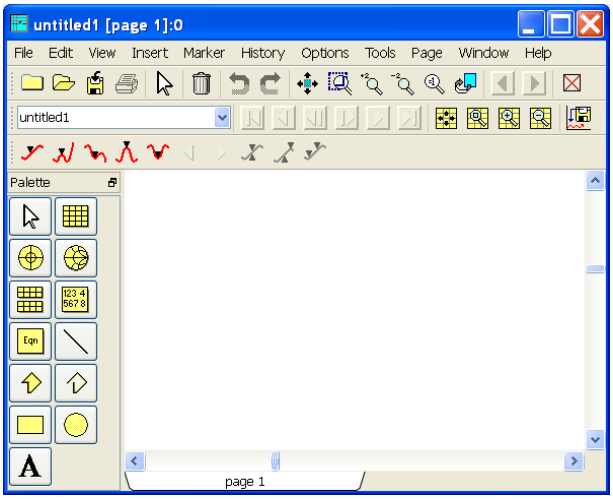
15) Using the wire command in main toolbar , wire all the components in series. 

16) In the 'Simulation-S_Param' palette, select SP simulation. 

This tells the program that you wish to perform an S parameter analysis on your system. The 'Term' in the schematic acts effectively as a matched 50 Ω input port. Since there is only one Term, you will only have S11 to plot. You must specify range of frequencies to analyze. For our purposes, we're interested in 2.4 GHz, therefore we can run sweep from 1-5 GHz. Make sure that the number of points is large enough to obtain desired resolution. I picked 200. When you are done setting up, hit F7 to run simulation

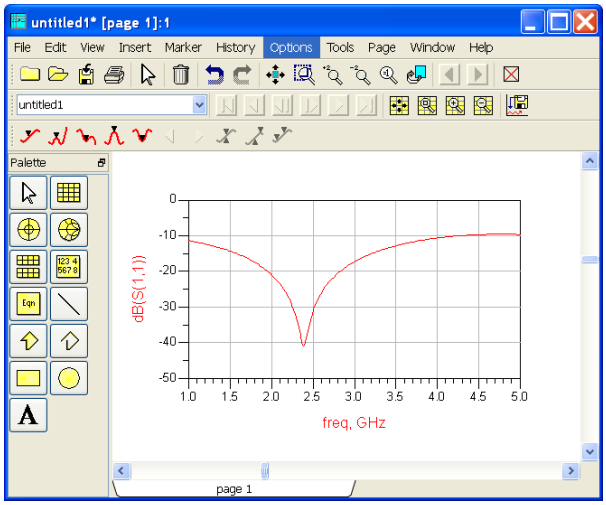
- Notice that at first a window will pop up. It will say that since the substrate metal thickness is zero, metal loss will not be included. Ignore that for now, but feel free to change that in the future.

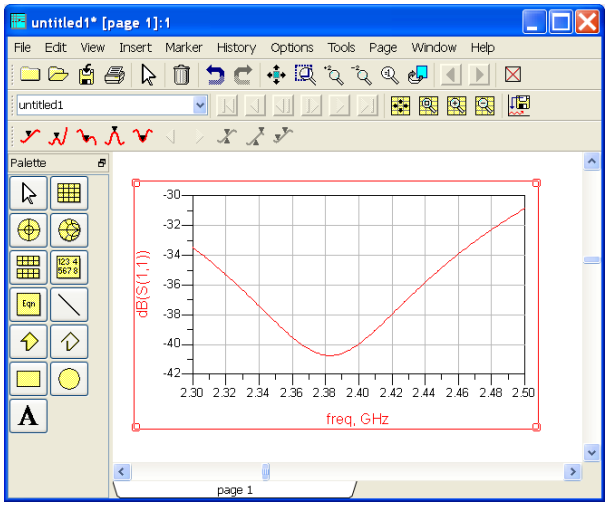
To see the results, wait for another window to pop up.
THIS ONE →



17) Now, click rectangular plot button 

Place it in the window. Then select S11, click add, pick dB and click ok. Now should see a 'dip' at which peaks at 2.4GHz, The 'deeper' the dip is, the better. Mine came out to be -40dB. If you have performed everything correctly, yours should peak at 2.4GHz as well. At this point change your frequency sweep to 2.3-2.5 GHZ in your S Parameters box in the schematic. Maintain a 200 point plot. Run the results again, and plot them. Observe that the sharp dip turns into a nice and smooth curve.





Your simulations should be similar to these

Now, notice that the new plot overwrites the previous one. Therefore in order to compare results for several simulations, go to Simulate/Simulation Setup in the main toolbar. There you can change the variable name (untitled1 is default). Change it to untitled2, click apply. Hit F7, and observe that you now have two windows. This allows you to make whatever changes you wish, and effectively compare how it affects your reflection coefficient.

18) In MSUB, change T, the conductor thickness to 35um. See how it affects the 'dip' in terms of frequency and magnitude.

19) Now you are ready to see what your physical design looks like. Select Layout/Generate/Update Layout. From here you can make any additional changes to the circuit. You can also print the mask. Make sure to deselect ‘Fit to page’. If it is selected, your circuit will not be to scale.