# ADS2008 Tutorial 3

**Problem:** Create a Quadrature (90°) hybrid at 2.5 GHz using ADS2008. Use Optimization in your analysis.

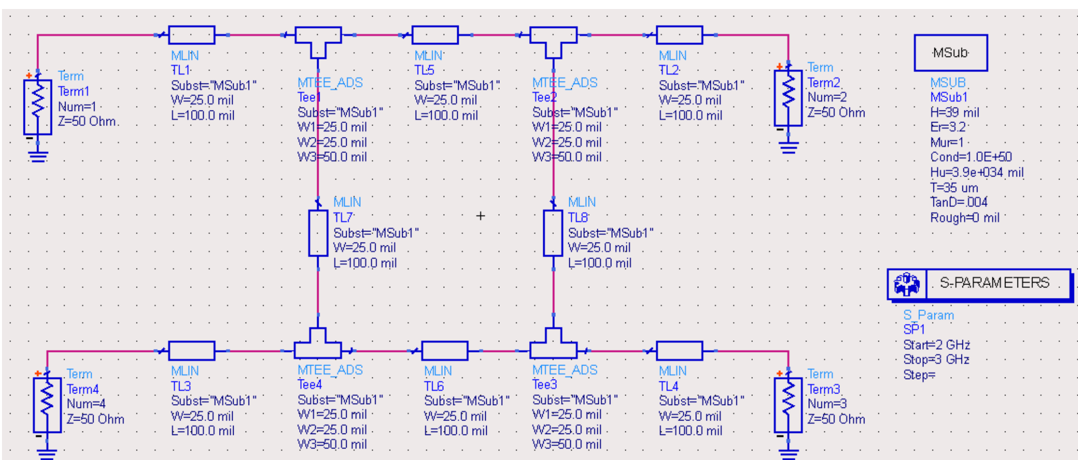
**Objective:** First, design your circuit using microstrip lines for Arlon 3.20 substrate. Use LineCalc to calculate the initial dimensions of all lines, and then optimize them to fit our design goals.

**Assumptions:** This tutorial assumes that your have completed Tutorial1 and that you are proficient in LineCalc and have a basic understanding of microwave circuits. It is also assumed that you know what a quadrature hybrid is. If not it can be looked up in Pozar, 3rd Ed., Section 7.2, pp. 333.

**Goals:** Your design must be matched to 50 Ω at all ports. At 2.5 GHz S(1,1) should be between -50 dB and -65 dB. Also phase difference between S(1,2) and S(1,3) should be 90°

**Procedure:**

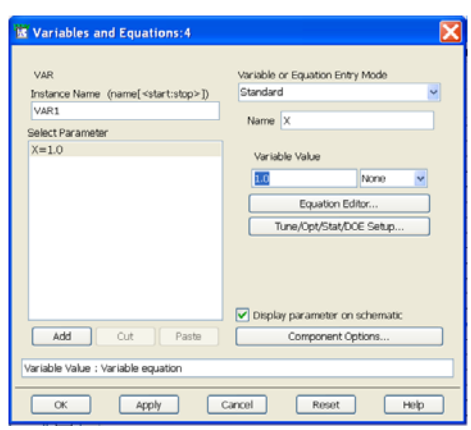
1) Start a new project. In your schematic window, define your substrate, place your simulation controller, terms, and grounds. Then place your microstrip lines, but do not edit their parameters yet. Just place an MLIN for each t-line required by the design. Then place a TEE anywhere that the line splits or combines. You should have eight lines and four tees. Connect them appropriately. Your design should look like this:




2) Notice that the design is symmetric. For example, TL1, TL2, TL3, TL4 have the same properties, which means the same length and width. But once you calculate their physical dimensions, you will manually have to input the same values into four different lines. Also, observe that the TEEs connect to three lines each. Depending on line width, each TEE will have to be set to match them. What happens when you change the line width, to fit a design goal? As a result, you will have to make the exact same change in the TEE as well. This results a lot of tedious work which can easily be avoided.

3) Go to the palette menu and pick Data Items. Select  and place it in the schematic.





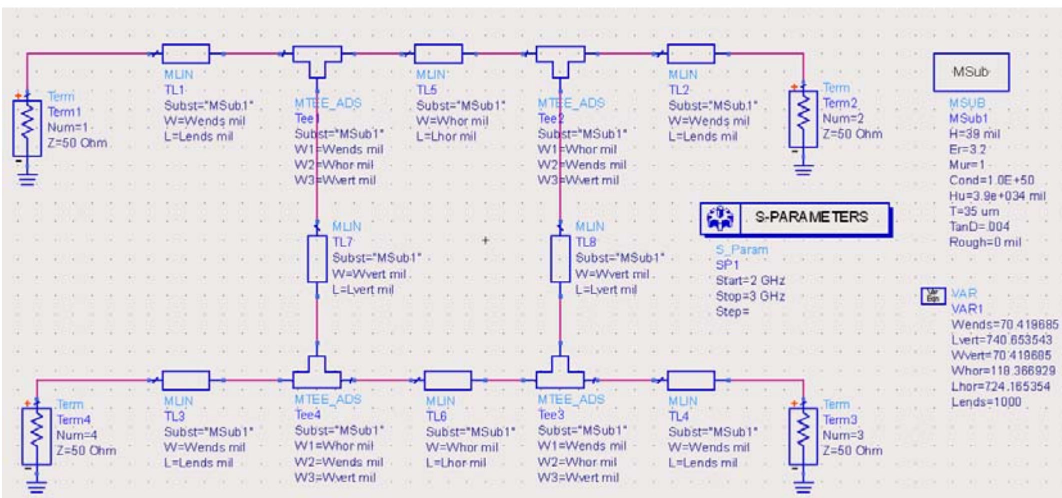

Double click on it. You should see this menu. →

- VAR1 is the SET of variables. You only need one for this project
- X is the variable name.
- 1.0 is the current value of X. Notice that it is unitless.

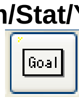
4) You will need to have one variable for each different line length or width. So looking back at the circuit, you will notice that due to symmetry, TL1, TL2, TL3, TL4 will be identical. Use LineCalc to determine the width. Don't forget to set your substrate before you do this. Let the length be 1000 mils. Now change X to Wends, and its value to whatever LineCalc produces. Click Apply. Now in 'NAME' type in Lends, set the value to 1000, and click ADD. You now have 2 variables, Wends, and Lends. Each contains a unitless value.

5) This takes care of the outer four t-lines. Now you have two lines with Zo = 50 Ω, and two with Zo = 50/sqrt(2) Ω. Their lengths are all λ/4, but we cannot assume that exact value due to non idealities. We will treat them in pairs, since they must be symmetric. Thus we need four more variables. Look at the center lines on p. 333. Two are horizontal and two are vertical. Create four more variables. Wvert, Lvert, Whor, Lhor. Use LineCalc to assign the correct initial values.

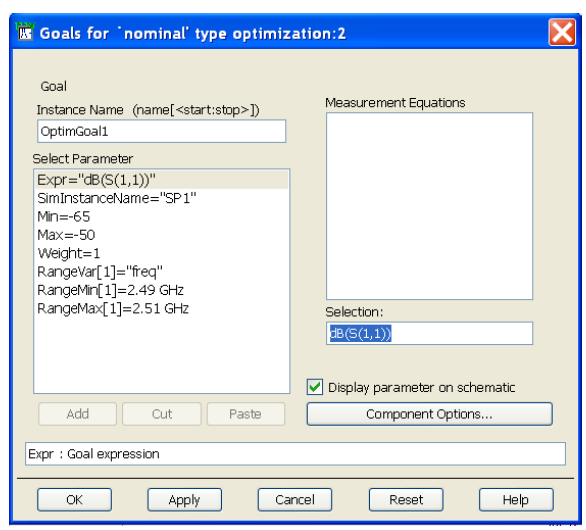
6) Now, go back to the schematic and put in the variable names into each parameter that you would otherwise use numbers for. Make sure you specify 'mils' each time. That's how your unitless variables take on physical lengths. Now, if you need to make any changes, all you need to do is change the desired variable, instead having to make the same exact modification to multiple objects. Don't forget about the TEEs. Your circuit should be like this:




Run simulation from 2 GHz to 3 GHz. Notice that S(1,1) has a 'dip' at 2.35 GHz instead of 2.5 GHz. We will now use optimization techniques to shift it.

7) Select **Optim/Stat/Yield/DOE** from the palette menu. Place 

in your schematic. Then double click on it, and make look as follows: →




*Here is what each parameter does:*

**Expr:**

This is what you are optimizing. Must be typed exactly as shown. Later you will modify this field to fit different design goals. Make no spelling errors here!!

**SimInstanceName:**

Simulation parameter used. You have only one.

**Min\Max:**

Your range of desired values.

**Weight:**

The importance of your goal. Used for multiple goals. 1 = default. Feel free to raise that number for higher importance

**RangeVar**

Your sweep variable. Use ‘freq’here.

**RangeMin/Max**

Specifies range across which your Min/Max expression from above must hold true.

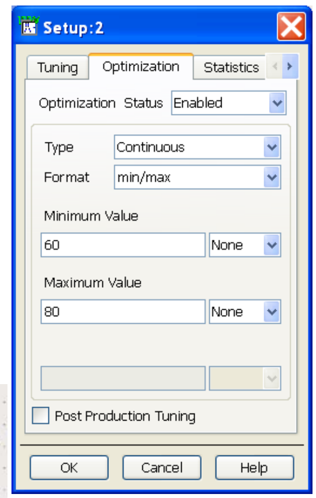
8) Place 

in your schematic. Set optimization type to Gradient, and number of iterations to 100.

9) Then go to your variables. For each variable, select 

.

Then select the optimization tab, and enable optimization. Then select a range of values that your variable may take on in order to make your circuit work. For instance my Wends was 70.4 mils, so I let my variable vary from 60 mils to 80 mils. Here is how it looks like →




10) Follow this procedure for all remaining variables except for Lends. Leave that at 1000 mils. When you are done, you will notice an {o} mark in your schematic for each variable that you wish optimized. You can disable optimization here, and you will see a {-o} mark for that variable. You can do this to save time, because each optimizable variable increases simulation time.


11) Hit F7. Plot phase difference as well as S(1,1). You will most likely notice that the phase difference is not exactly 90. Now add another goal, and specity the Expr to be ‘phase(S(2,1))-phase(S(3,1))’. Make the min max values between 89.99 and 90.01. Observe any changes to your variables.

12) You may specify as many goals as your design requires. It is recommended that you set the minimum number of goals, in order to ensure that all of them are met.

13) Once the simulation is complete, look at the graphs to see if the optimizer has indeed reached the goals. Sometimes, multiple goals can conflict with one another, and the only option is to get rid of the ones of lesser importance. For example, a phase shift of 89.8 degrees can still be considered as success, granted that you have traded off that 0.2 degrees in order to lower S(1,1) by additional 10 dB.

14) Sometimes the simulation might be terminated, for instance, due to zero gradient. In this case, switching to a different optimization method will often solve the problem. Once you have achieved the desired performance out of your circuit. Click simulation/update opt values. This will start future simulations using these values, thus saving simulation time. At this point, you can turn off optimizations for the parts that you know have desired value. You are now ready to perform analysis in RF momentum.

Written by Michael Pawelczyk
June 2009
Microwave Circuit Design