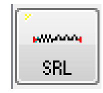
ADS2008 Tutorial 2

Problem: A load impedance ZL = 30 + j40 is to be matched to a 50 Ω line using a single shunt stub tuner. Find two solutions using open circuited stubs.

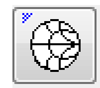
Objective: Using ADS2008, find S11. Then, using microstip lines, design the circuit. Then repeat the procedure using two stubs, placed 1/8λ apart. Use frequency of 2.4 GHz.

Note: This tutorial assumes that you have mastered all the techniques presented in the previous one. This one intends to teach you the smith chart utility in order to graphically illustrate matching procedures

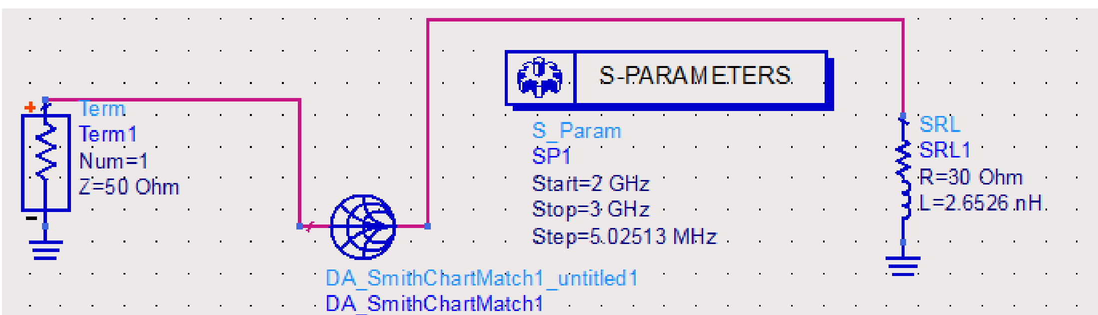
1) Launch ADS, and create a new project.



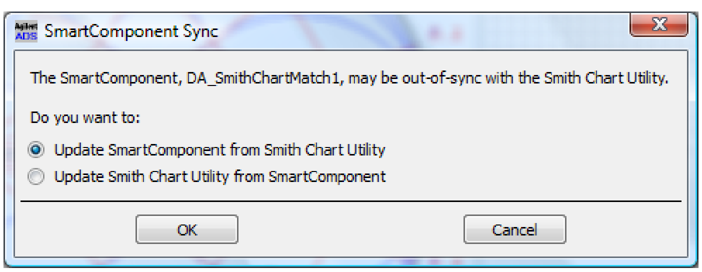
2) Place a term (S-Par palette) and series RL load (Lumped Components palette) in the schematic. (L = 2.6526 nH, R = 30 Ω). Then add S-Prameters simulation.



3) From the palette menu, select Smith Chart Matching, then click Smith Chart symbol and place it on the schematic. A message will appear saying that you must first use a certain utility for it to work. Next, wire the term to the smith chart, and smith chart to load. Don't forget appropriate grounds....

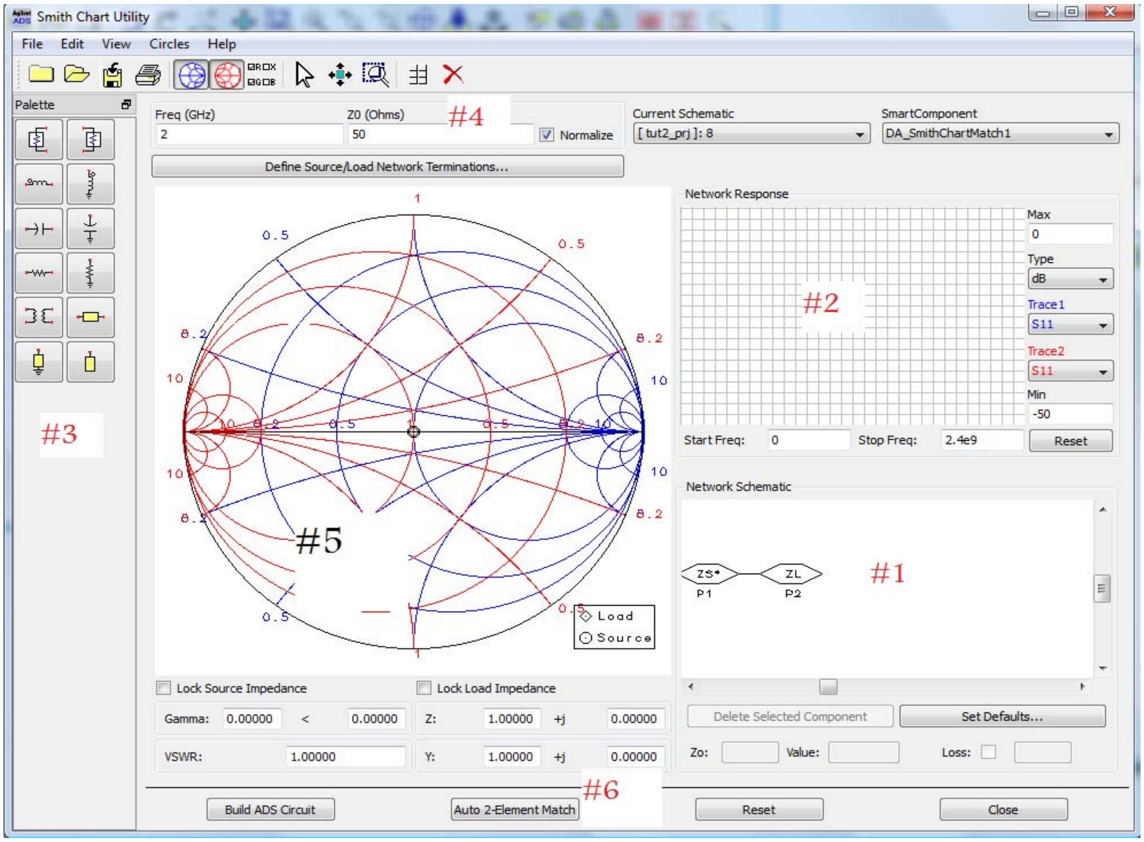


4) Select the Smith Chart, and go to Tools/Smith Chart...
The following window will pop up



5) Select the first option, and hit ok.

- The smith chart utility replaces the paper method you might be familiar with. For those of you who have made mistakes by turning the smith chart in the wrong direction, or who have read the incorrect scale, your troubles are soon to be over. Steps are relatively simple. First you define normalized source, then normalized load impedance. Finally you want to add stubs or T-lines to make the load 'travel' to the source (if both are purely real). We accomplish this with the following steps:



6) First familiarize yourself with this utility. I have outlined and labeled a few key components here.

#1) *This is your circuit*. Right now it has only a source and load, represented by hexagons. As you add more components, more items will appear.

#2) *This is the plot window*. Here is where you will complete one of your objectives, finding S11, as a function of frequency.

#3) *These are the components* you may add. Among these are series inductor, capacitor, stubs, etc. We will only use open circuited stubs and t-lines.

#4) *Main toolbar*. Here you specify frequency, set impedances. You may also choose to turn off admittance chart, or switch colors around. I leave it up to you to play with this.

#5) The Smith Chart. A graphic illustration of your impedances. Right now, there is a load and a source, plotted on top of one another at the origin. In the future, you will see how different elements affect what happens here.

#6) Here, you *set your source and load impedances*. You may also lock them, to prevent from accidentally moving them.

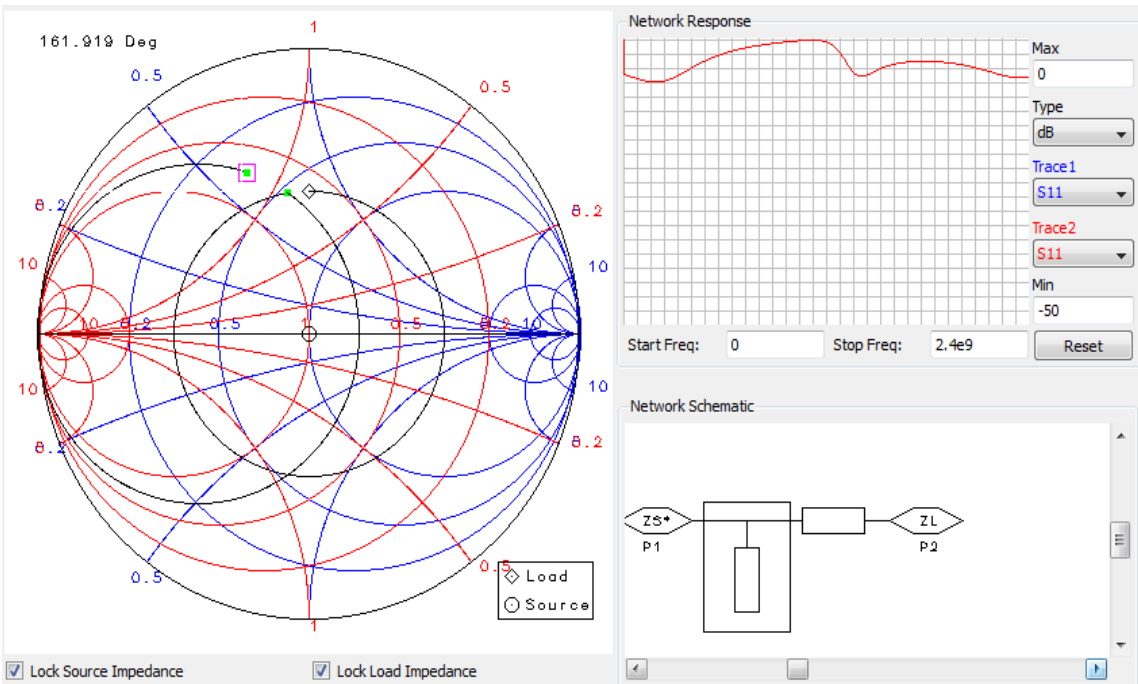
7) So, now that you are ready to design your stub tuner. First, select Z_L from window#1 and set the impedance to Z = 0.6 - j0.8. Lock the impedance. Similarly, do the same for source, setting it to 1+j0.

8) Now from window#3, add line length. This is a standard transmission line. Notice on the smith chart that your cursor draws the clockwise (toward the generator). If you try to go counterclockwise, you will see that this utility prevents you from making that mistake.

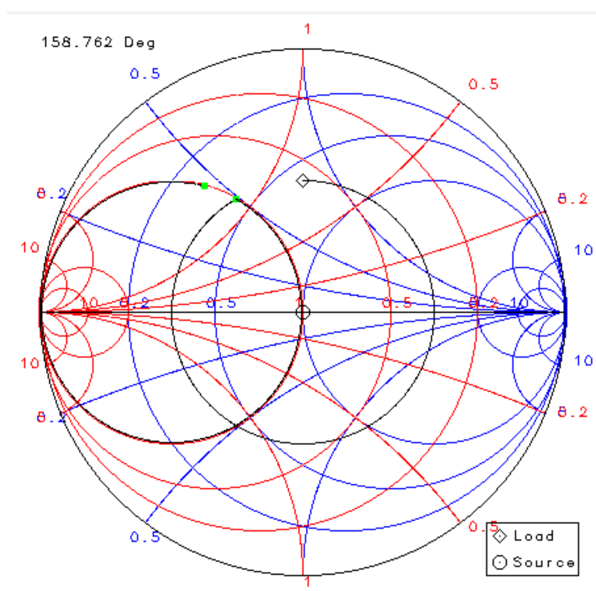
- **Note**: I will assume that you DO NOT know how to design a single stub tuner. Therefore i will illustrate a long approach to this design, which will make the double stub tuner (part2) much easier. Let's begin!

9) After selecting line length, make it almost half wavelength long. The cursor should draw almost a complete circle, and it should end up 'almost' back at the load.

10) Now, select an open circuit stub, and again, make it long enough to draw out another, almost complete circle. Notice the path that cursor takes. Also, observe that our design (window#1) consists of load, line, stub, and source. Right now, your impedance is the pink square, but you want it to be at the center, to match the source

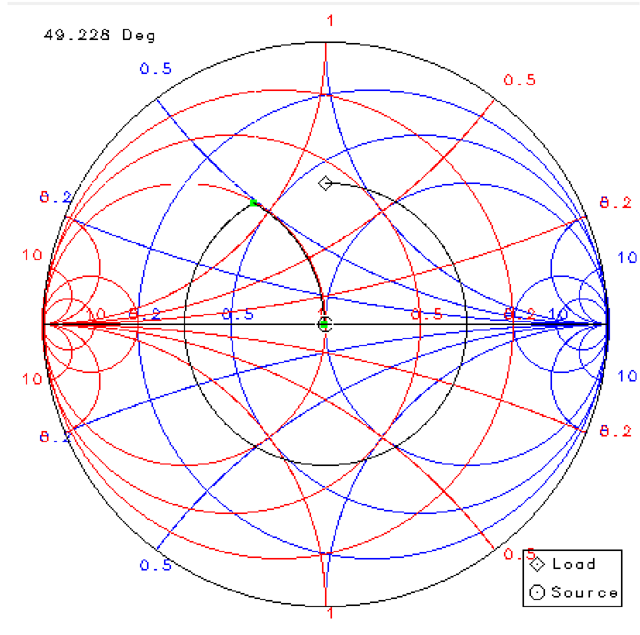


11) In window #1, select the t-line. Now you can manipulate its length, either numerically below window #1, or graphically on the chart. As you change length, the circle drawn out by the stub moves along with the termination of the t-line. Reduce the **t-line** length until the 'circle' passes thru the origin. Like so:



12)   Notice the length of the t-line to be around 164 degrees. Now, select the **stub** and reduce it's length until you end up at the origin. TA DA! You are now matched. By selecting your components in window#1, you can read their respective values.

Mine were 164.7 and 49.2 degrees. A half wavelength stub or line corresponds to 180 degrees

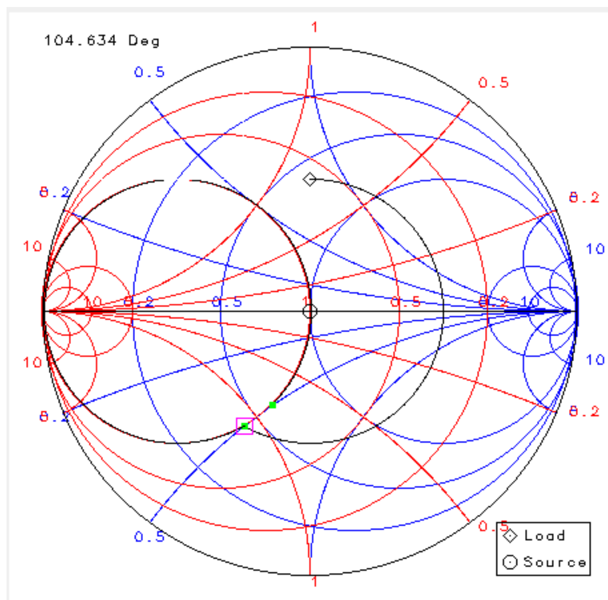


13)   The question is 'Do we really need the t-line to be that long'? No!

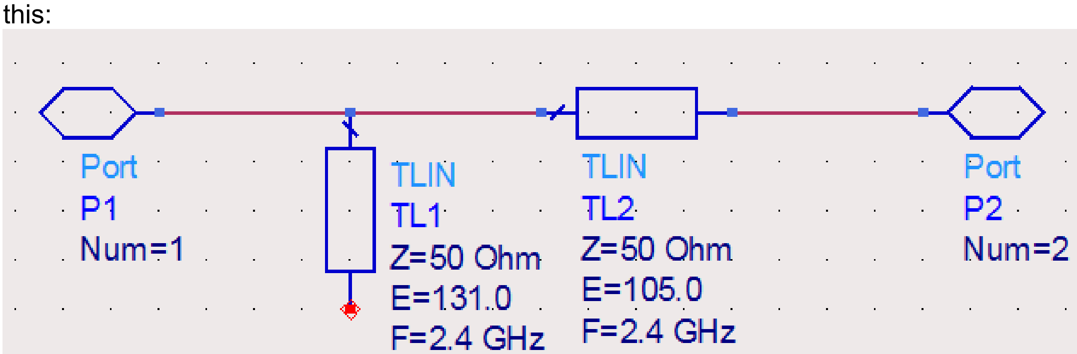
First make the **stub** long again, so cursor traces almost a complete circle. Reduce the **line** again, until origin is crossed by the stub line. Now, shorten the **stub** again and voila!!! You have created another solution. The reduction of line was accomplished however at the expense of a longer stub.

My values were 104.973, and 131.0 degrees for line and stub respectively.

Now look at the plot (window#2) and see the reflection coefficient. Make sure that frequency is set to 2.4 GHz. My result is -55dB

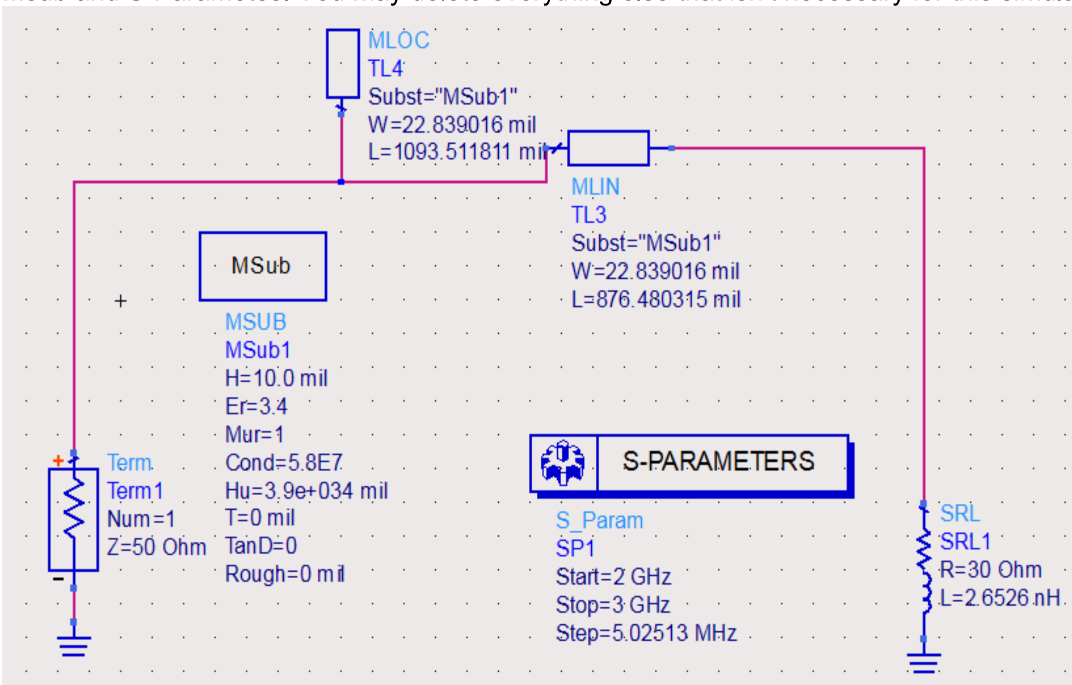


14)   Now you want to design the actual circuit. To do this, click **Build ADS Circuit**. You will notice that the smith chart in your schematic is now updated. Also another schematic is created. It looks similar to this:

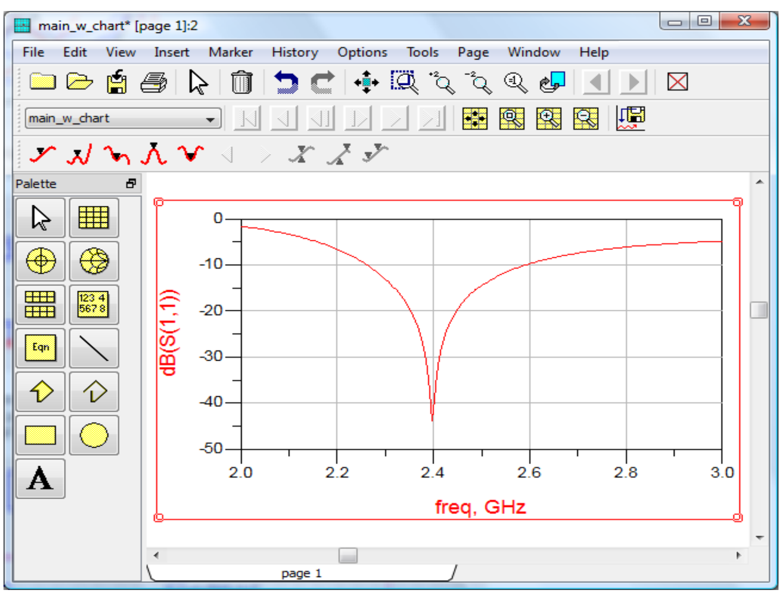


Next Goal: Modify this circuit to fit the design objective.

Notice that it consists of t-lines instead of microstrips that we desire. But given E_eff, Zo and frequency, we can create an equivalent circuit. Using techniques presented in previous tutorial, use LineCalc to determine the dimensions of the microstrips. Assume 10mil substrate with er of 3.4. Set conductivity to 5.8E7 S/m, and TanD to zero. Make sure to change port1 to term, and port2 to our load values. Place Msub and S-Parametes. You may delete everything else that isn't necessary for this simulation.



And the S11 plot should be as follows. Notice that the reflection coefficient is slightly greater than the one found on smith chart. Those are the non-idealities introduced by finite conductivity of the metal as well as other parasitics that you can further modify. Also, your frequency sweep is performed at discrete intervals. Sometimes your frequency 'misses' the minimum, and outputs a higher value. Generally S11 lower than -30 dB is quite decent. -40dB or less is great.



Remember that you now have 2 schematics. One contains Term, Smith Chart and Load.

The other consisted of t-lines which you have hopefully replaced with correct microstrip lines.

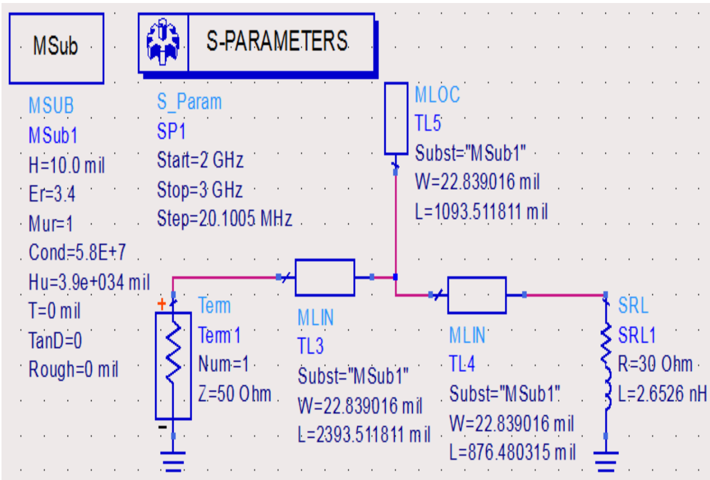
You may run simulations on either one, but

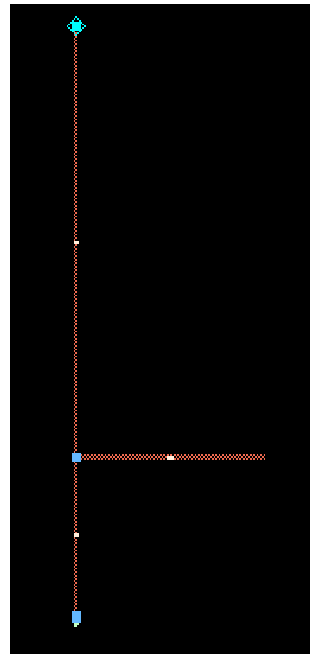
The smith chart can be easily modified if you need to switch to a different load

The microstrip circuit can more easily account for non idealities, and can eventually be transformed into a physical circuit.

Keep those differences in mind as you are constructing more advanced circuits in the future.

15) To generate layout, add another microstrip line of the same width and arbitrary length between the term and the stub. Otherwise, the stub will be placed 'in line' with the first strip, and we will not be able to determine where the line ends and the stub begins. Your final circuit and layout should look like this (not to scale):





Part2

1) Repeat steps 1-7 of Part1

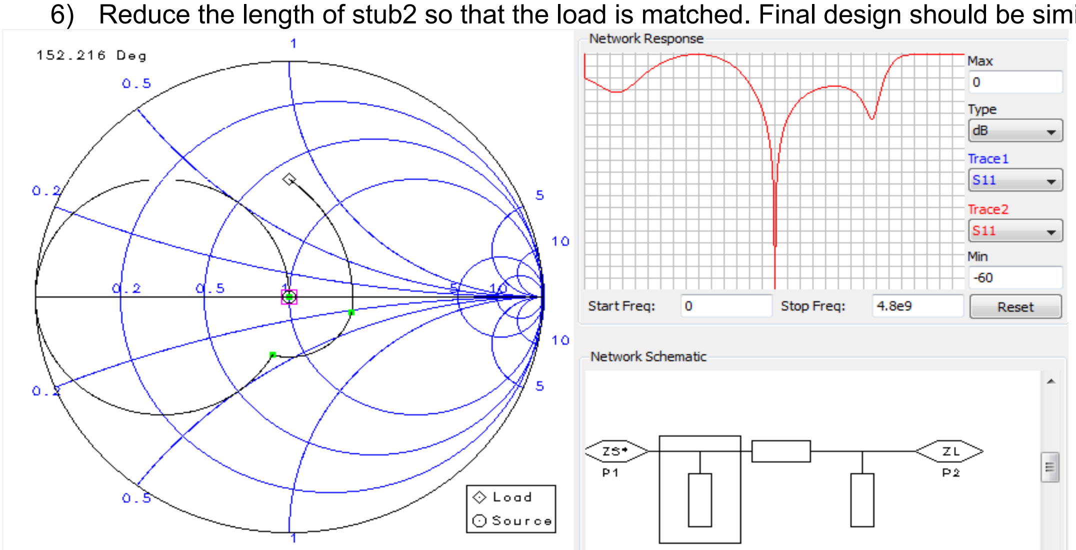
2) Add a OC stub, make it almost half wavelength long.

3) Place a 1/8 lambda line. (45 degrees)

4) Place another OC stub. Make that almost half wavelength long.

5) Select first stub (one closest to load). Make it as short as you can, so that the circle drawn by stub2 crosses the origin.

6) Reduce the length of stub2 so that the load is matched. Final design should be similar to:



7) Finally build use previously developed skills to turn this into a microstrip circuit. Run simulations. Compare to the original design. Which is better?

Written by Michael Pawelczyk, May 2009, Microwave Circuit Design