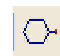
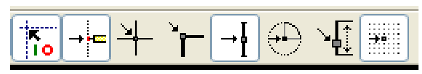
Tutorial Four – RF Momentum

While your design might look perfect in the schematic, the actual results will be quite different. Transmission line theory doesn't account for certain effects such as coupling, dispersion, radiation and higher order modes. Also, bizarre circuit geometries cannot be accounted for in schematic. Therefore, to obtain a better approximation, use RF momentum. This is done exclusively in the layout.

1) Generate your layout and verify that it looks similar to the one in your textbook. Now you have to attach ports to each end. Ports are just like terms. Select a port from and toolbar, then click ok. Place one port in the 'middle' of each leg, at the very edge. Ads pin snap utility should assist you in guiding the arrow cursor into the center of the line. It is located in the main toolbar. I will leave it up to you to master this technique

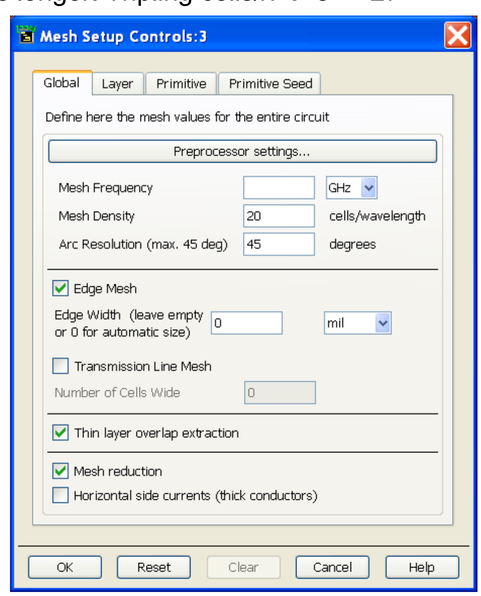




2) Click **Momentum\Enable RF Mode**. Next, define your substrate. To do this, you should use the substrate you have already defined in your schematic. Click **Momentum\Substrate\Update from Schematic**. Next, click **Momentum\Substrate\Create/Modify.** In *substrate layers*, select <u>ground</u> and make sure it has finite conductivity (roughly 4.7E7). In *layout layers*, set the strip conductor to the same conductivity. Then save the substrate.

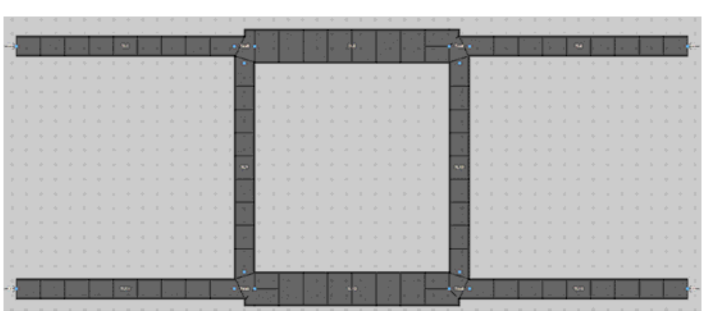
3) Next, you must set up a mesh. This divides your entire circuit into squares called cells, and then performs analysis, which solves for currents and fields throughout your device. The analysis is beyond the scope of this tutorial or even this course, but there are a few key facts that must be taken into account.

- The cell mesh is initially generated at a certain frequency. This does not mean that you have to use only that particular one. It is more of a general approximation of how many cells to create throughout the device. For our purposes, we will use 2.5 GHz.
- The number of cells per wavelength is self explanatory. Keep in mind that the more cells per wavelength you have, the more accurate your answer will be. The trade off however is simulation time. More cells → more time per iteration.
- Simulation time varies roughly in a cubic manner. That means, if you double the number of cells/λ, the resulting simulation time will be 2^3 = Eight times longer. Tripling cells/λ → 3^3 = 27 times longer simulation.
- As a general rule of thumb, 20 cells/λ is the bare minimum, 40 is quite decent, and 60 is a slight overkill and should only be used on fast machines.



4) Click **Momentum\Mesh\Setup.** Set *Mesh Frequency* to 2.5 GHz, *Mesh Density* to 30, *Arc Resolution* to 45, and *Edge Width* to 3 mils. Leave everything else as is. →

5) Click **Momentum\Mesh\Preview.** Here you will see what it actually does. Here is what is should look like



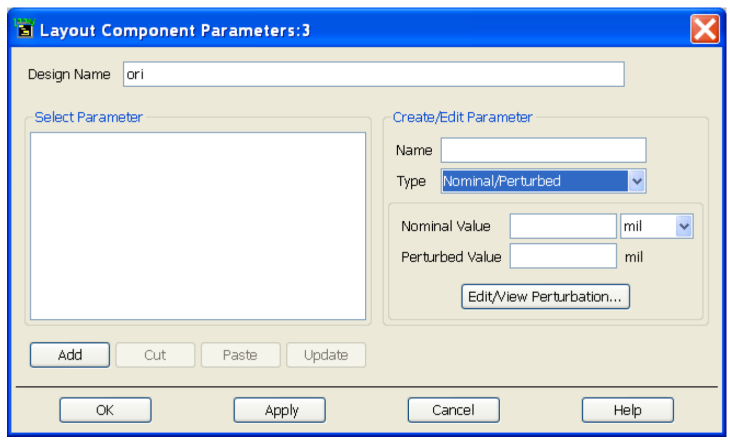
6) Although a bit hard to see, you can clearly see the cells in your layout. If you zoom in, you can observe that there are 'center' cells and 'edge' cells. Edge cells are small, 3 mils as specified. Later you will be able to diagnose problems by looking for Edge cells in the center of your conductor. This should not happen, since edge cells, like the name suggests should only be near the edges of conductors. More on that later.

7) Now set up the simulation. Go To **Momentum\Simulation\S-Parameters.** For now, select Linear Sweep, from 2.3 GHz to 2.7 GHz, with a 5 MHz step. Click **Update**. Every time you make changes to the frequency plan, make sure to click the update button before you simulate, otherwise the changes won't take effect.

8) Click **Simulate.** Notice how slow this process takes, compared to the schematic simulation. Also observe that the results obtained in Momentum are different from the ones in schematic!!

9) So what now? Has all our hard work so far been in vain? Not necessarily, but much more is needed. Now we will optimize the circuit in Momentum!

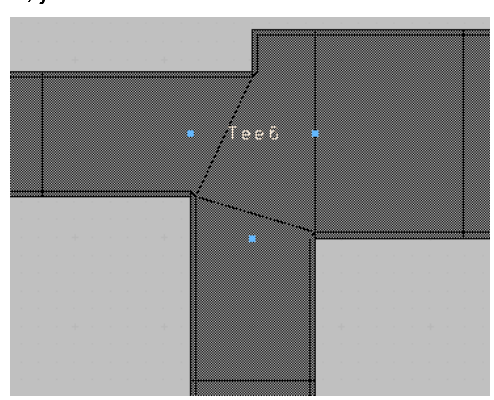
10) Double click a trace in your LAYOUT. Notice that the same variables from schematic are still listed as lengths and widths of all your lines and Ts. First you will need to redefine all the variables. Go to **Momentum\Component\Parameters.**



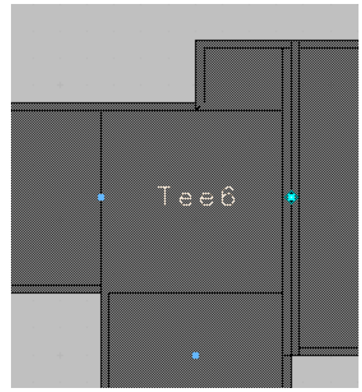
11) Change the type to '*Subnetwork*'. Add in the same values and variables as in your schematic, EXCEPT add lowercase 'm' in front of the variable name. Leave units at 'none'

12) The reason behind this is that for some reason the program doesn't let you reuse variable names from schematic. So you need to create those variables again, only with 'm' in front (or whatever). NOW, change all lines and Ts IN THE LAYOUT, by adding the letter 'm' to the already existing variables that define your lengths and widths. I know it's a bit tedious, but so far this is the quickest KNOWN method.

13) Once you have everything updated, run the simulation again, just to make sure the results are the same. If not, there are a few possible scenarios that may occur:

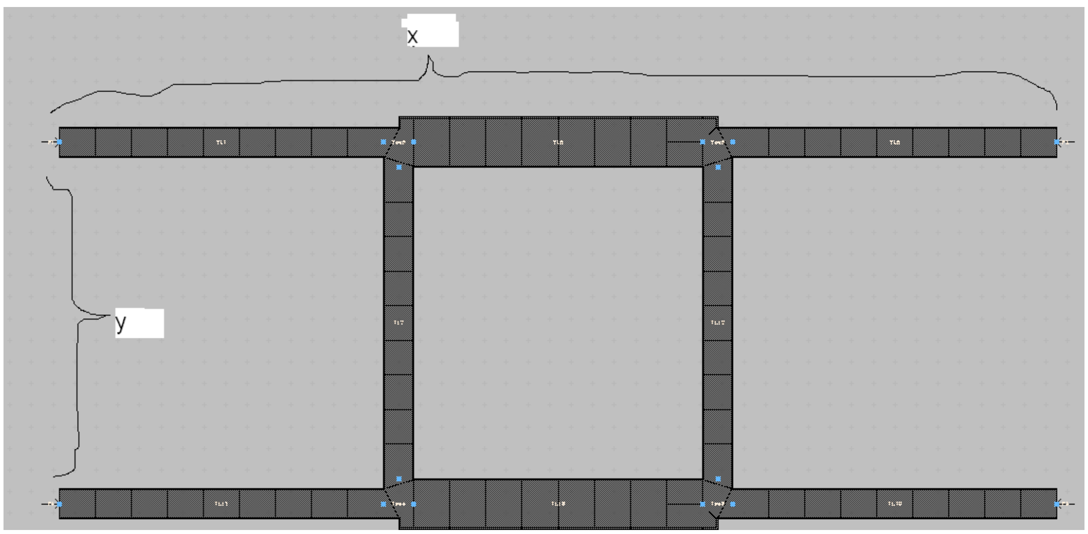


- First of all, if you haven't copied the dimensions EXACTLY, a slight rounding error might cause a minor shift in your plots. The general shape of the plots should be maintained, however. Proceed to step 14.
- If the plot is completely off, you may need to redo the mesh and look at areas where conductors touch. For example, here is a proper T connection. Notice that the thin edge cells are only on the outside. →
- If your conductors only appear to be touching, you will likely observe something like this one below:



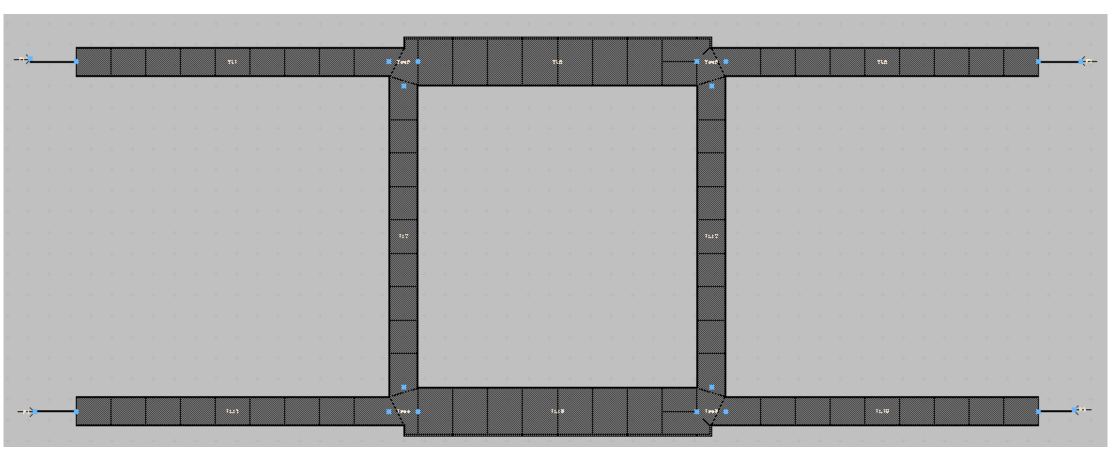
- Observe the edge effect. This sometimes happens when the snap mode is off. Two conductors side by side in reality have a gap in them that can only be observed under a close zoom.
- If this happens, enable the snap mode, as well as midpoint snap mode. Then try to get them to touch. Once in place, move them apart. If they are truly connected, there will be a wire drawn between them, once they are pulled apart again.
- This phenomenon can occur anywhere that conductors are supposed to be touching but aren't. The mesh preview function lets you pick out those spots, and show you the reasons why your simulations aren't producing the desired results

14) So now you have your circuit ready to be optimized. First consider what is going on, and what will be happening in the future. Right now you have your microstrip lines laid out, with a port at each 'leg'. Here is what it looks like:



- Right now your ports are separated by distances x and y.
- As you perform optimization, your trace dimensions will vary, but the ports sometimes have a nasty habit at remaining at their fixed positions. As the circuit shrinks, the ports lose contact with the conductors. As far as Momentum is concerned, you end up exciting free space, and your results are completely wrong.

15) An effective solution appears to be moving the ports out, so they are not sitting on the edge of the conductors. But when you move them, make sure that there is a thin wire being drawn that connects the two. This ensures that as circuit dimensions change, the ports are truly connected to the midpoints of the legs. Here is an example:



16) Now, create a component model. Select **Momentum\Component\Create\Update.** Use default settings here, but make sure that 'delete previous database' is checked. Also, verify that the simulation control will be performed in the SCHEMATIC!!!!!!

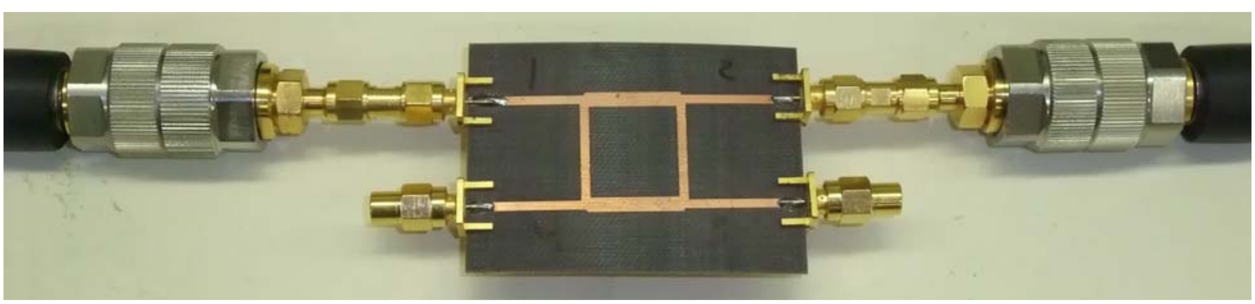
17) Create a new schematic. This is where you will insert the model that you can optimize. Select the bookshelf from toolbar menu. Then go Projects. Most likely you will see two components, but they are both the same. Right click on the name of your first design window, and click *Place Component*. Notice it looks like the one in layout. Now add four Terms, grounds and connect them appropriately. Terms here are what ports were in the layout. Add S parameters simulation. Notice that when you created a component previously, you have specified frequency range. Now you are limited to use that particular range. If it's too narrow, feel free to return to LAYOUT, and recreate the model again. Notice that you do not need to put down the substrate here. It is already included in the model database.

18) Simulate the circuit again, to make sure it is the same as the one in layout. Now double click the model, select parameters, and click optimization setup. Choose the same variables to optimize as you did previously. Then place an optimization controller as well as your goals on the schematic. Using previously acquired skills, optimize the circuit for a S(1,1) < -50 dB, and for the phase difference between port 2 and 3 to be 90 degrees. Most likely you will not be able to obtain exactly 90.00 degrees phase shift, with a minimum reflection at 2.5 GHz, but your circuit, once completed, should be accurate enough to be used for Quadrature Modulation.

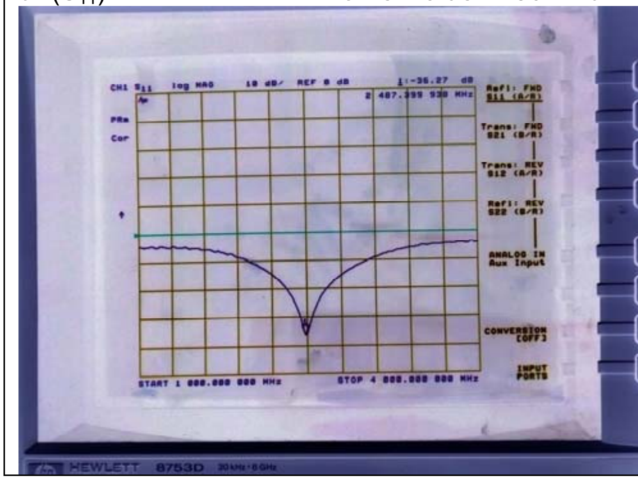
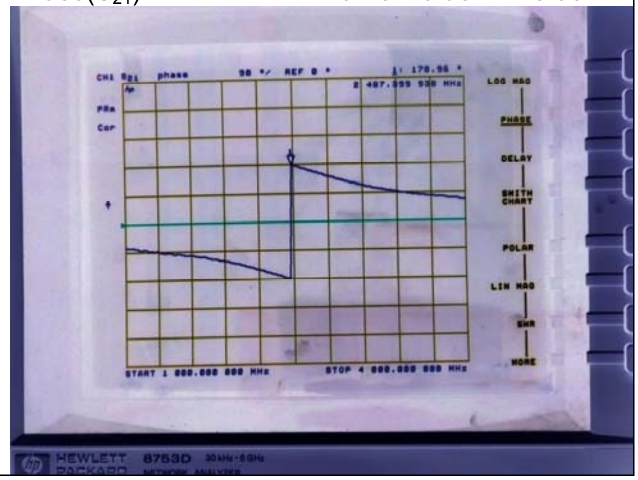
19) This is where things get frustrating. In order to print, you must take the UPDATED OPTIMIZED VALUES and return to your original microstrip line schematic. Since you have the variables put in, it will not be a long task. Create a NEW SCHEMATIC, copy and paste the original one, and change the values to ones given by the optimizer. Generate layout and print. You are done!

Testing and Importing Data

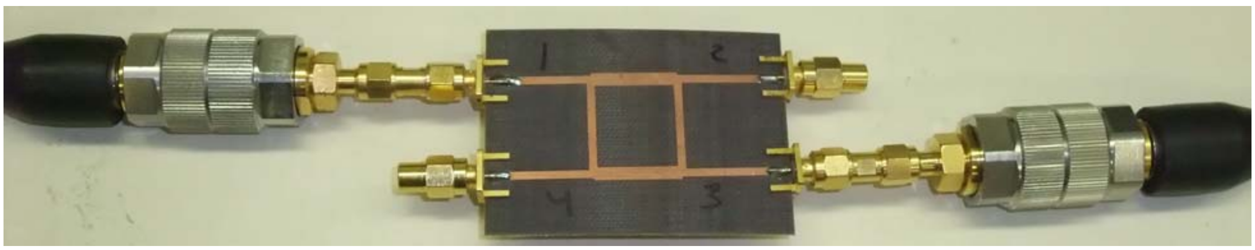
1) First, connect the VNA to port 1 and port 2. Cap the remaining ports with a 50 Ω load. Select **measure**, pick $\mathbf{S_{11}}$ and observe the results. Depending on the quality of your circuit, your results should be very similar to predicted values.

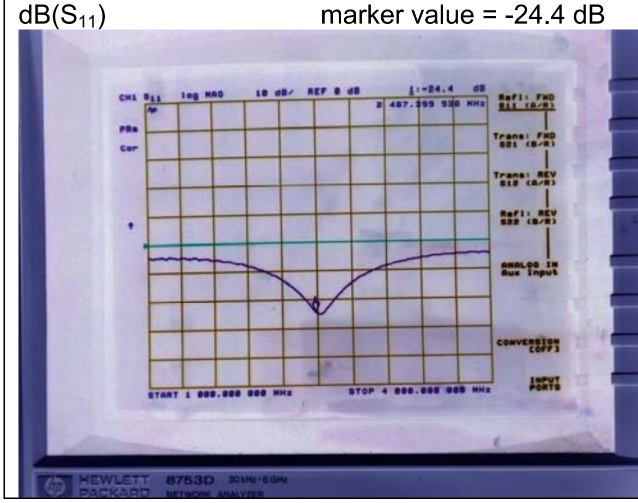
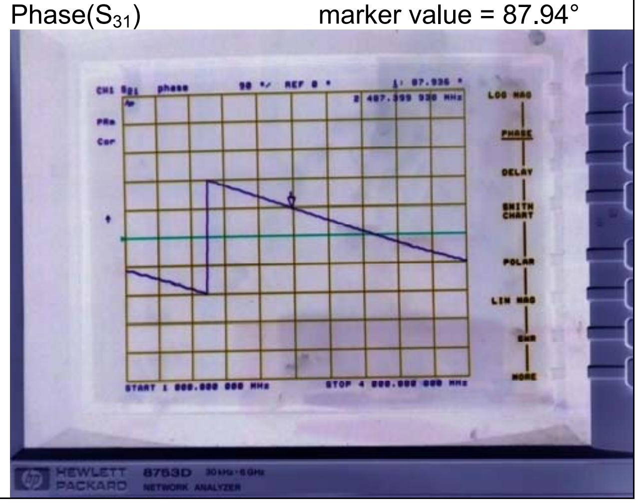


2) Despite your most sincere effort, it is unlikely that your circuit will be perfect. Certain defects, (usually asymmetrical) acquired during fabrication may degrade performance. Sometimes, those defects can be lessened by simply rotating the circuit 180 degrees (letting port 3 become port 1, etc). I have taken pictures of dB(S_{11}) and Phase (S_{21}).

dB(S_{11}) marker value = -36.27 dB	Phase(S_{21}) marker value = 178.96°
	

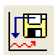
3) Since you do not have a third channel on your VNA, you must switch ports 2 and 3. Naturally, S_{31} used hereafter is still given to you as S_{21}

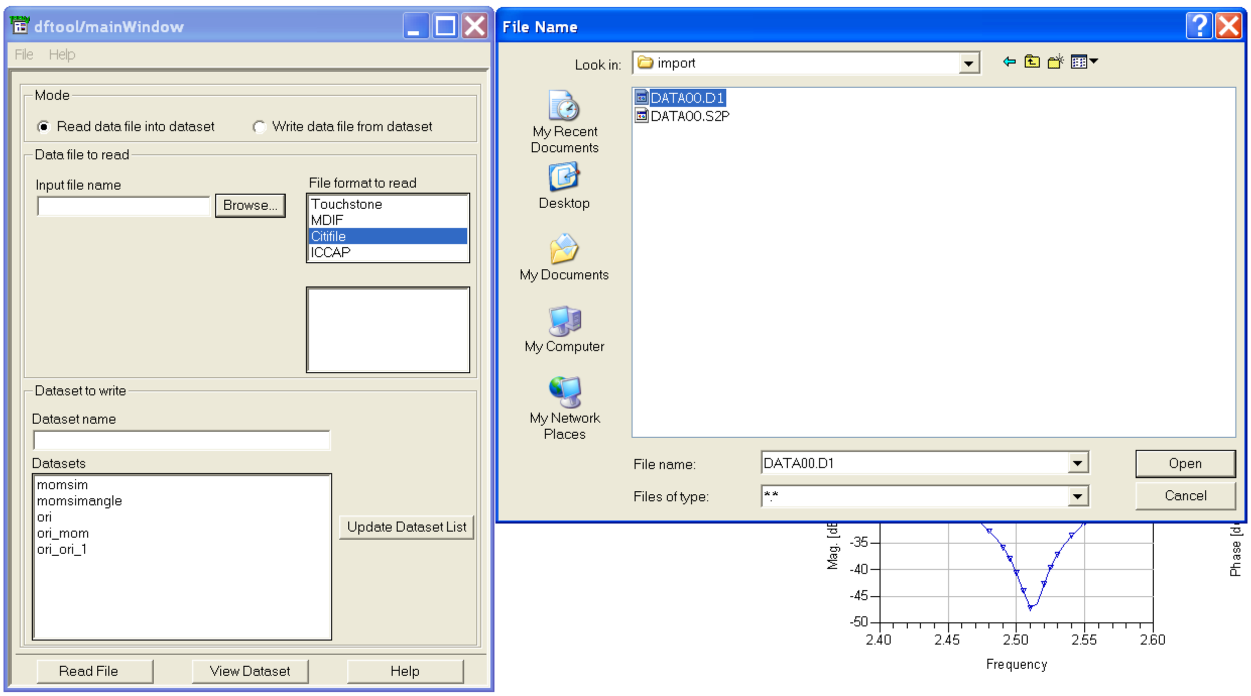


dB(S_{11}) marker value = -24.4 dB	Phase(S_{31}) marker value = 87.94°
	

4) Observe what happens to S_{11}. All you did was switch ports 2 and 3, and since they were both terminated by 50 Ω, they should have not affected S_{11} at all! Unfortunately the dummy loads aren't perfect, and as a result of the switch, you have a larger reflection loss, along with a slight frequency shift. Fortunately these results are still acceptable. You have input reflection loss of less than -20 dB and a 91.02° phase shift between the output ports. I will leave it to the reader to verify that the power has been split evenly between ports 2 and 3, and that virtually no power is delivered to port 4.

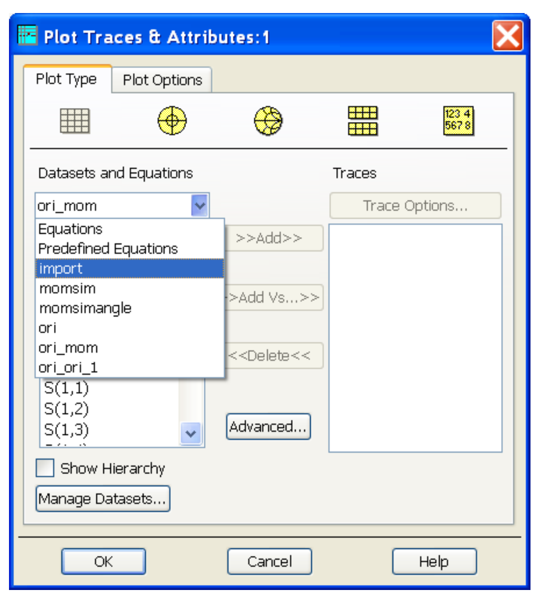
5) Taking pictures can get a bit annoying after a while, and it makes data extraction difficult. Therefore, let us capture the data, and plot it using ADS. To import VNA results, you must first save them on a disk. Insert a formatted disk into the VNA. Press **Save/Recall** then pick **Select Disk**. Choose **Internal Disk** as your destination. Return, and pick **Define Disk Save**. Here, set <u>Data Only</u> to **on,** and change format from binary to **ASCII**. Also set the top four options in that window to **off**. Again, return, and press **Save State**. This will initiate a long process of saving. If you are using the 6 GHz machine, it may lock up on you, or take at least 10 minutes to save. You may eject the disk after about two minutes, but this will cause the VNA will lock up on you, and you will have to redefine your standards and perform calibration all over again. Either way, your data will be saved.

6) Select  from the simulation data output window (the one that pops up each time you perform simulation). Then click **Browse**, and pick the file that has a .D1 extension. Make sure **citifile** is selected in format window. Then pick a dataset name (mine was 'import') and click **Read File.** It will most likely inform you that the data is written in an incorrect format. Ignore that and click **yes.**



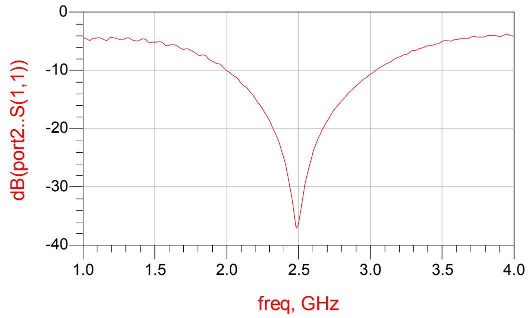
7) Create a new rectangular plot. Choose the dataset and equations to be 'import'. Example →

8) Now you have all the S parameters to choose from, just like any other plot. Plot the data and compare to the pictures taken. They should be identical. Don't forget that you must return to the VNA, switch ports around, save data and repeat this procedure once more. Figures below demonstrate that the plots are indeed the same as the snapshots. Once again, I will leave it to the reader to verify that we have an even power split, as well as a proper isolation between ports 1 and 4.

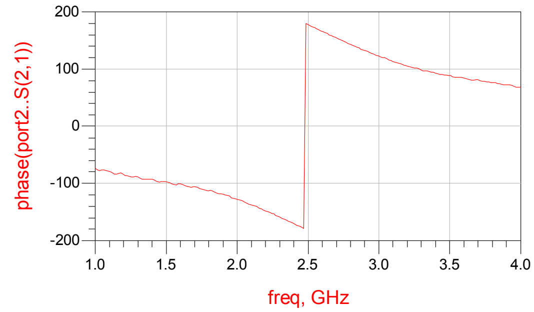


CPorts 1 and 2 are connected to VNA. Ports 3 and 4 are capped with dummy loads.

dB(S_{11})

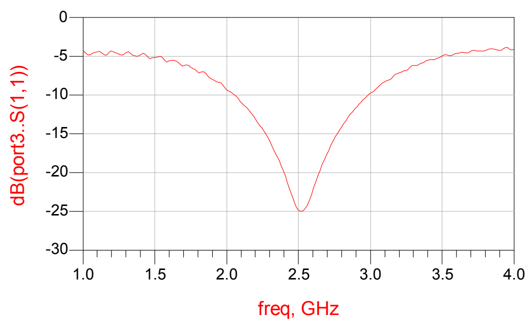


Phase(S_{21})

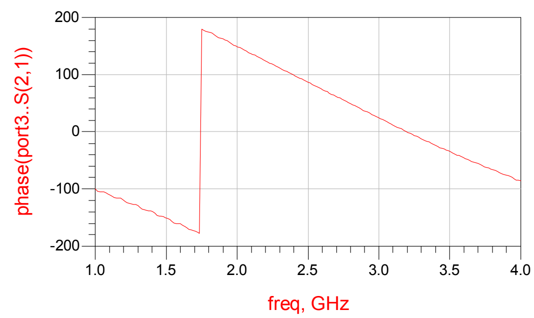


CPorts 1 and 3 are connected to VNA. Ports 2 and 4 are capped with dummy loads.

dB(S_{11})



Phase(S_{31})



Written by Michael Pawelczyk
June 2009
Microwave Circuit Design