

This chapter shows how to make S-parameter simulations and how to determine matching network values.

Lab 4: S-parameter Simulations

OBJECTIVES

- Measure gain and impedance with S-parameters
- Use a sweep plans, parameter sweeps, and equation based impedances
- Plot and manipulate data in new ways

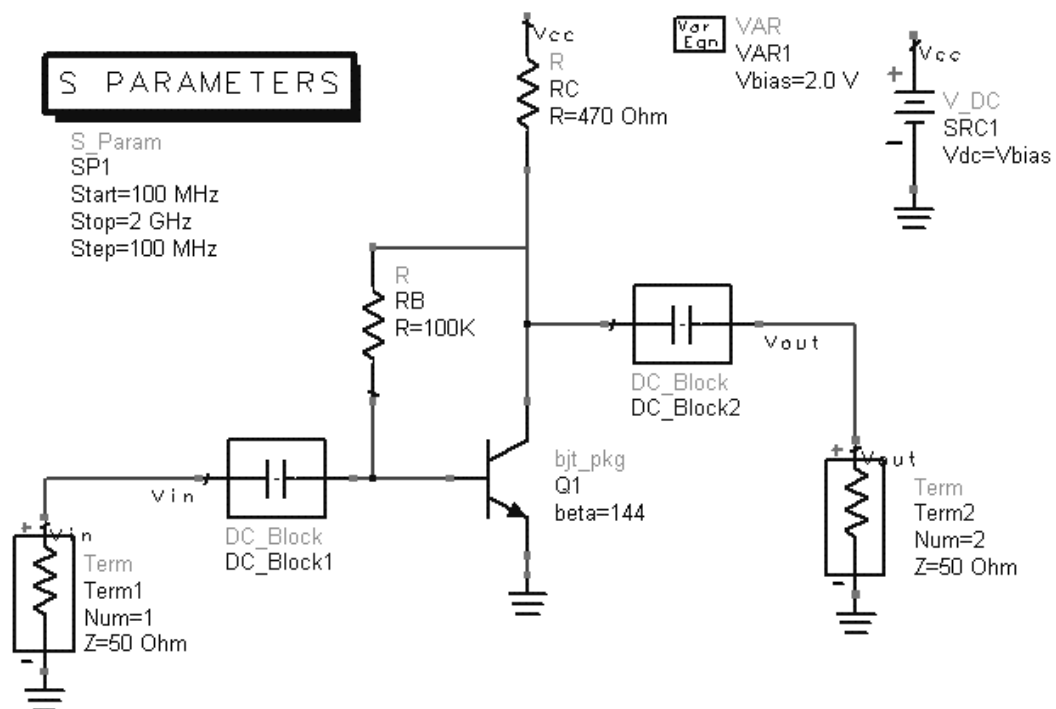
About this lab: This lab continues the mixer testing by making various S-parameter measurements to determine circuit performance: gain and impedance.

PROCEDURE

1. Copy the last lab and save it as a new design named: **s_params**.

After saving the schematic as **s_params**, continue modifying the design to match the schematic shown here, according to the following steps:

- Delete the AC sources and simulations. Also delete the measurement equations, parameters sweep, etc. – These are components you will not use for S-parameter simulations.
- Insert an S-Parameter simulation controller (Simulation-S_Param palette) and set: **Start=100 MHz**, **Stop=2 GHz**, and **Step=100 MHz**.
- Insert S-parameter port terminations (**Term**) instead of a source and load. The Node Names are the same as before: Vin and Vout.

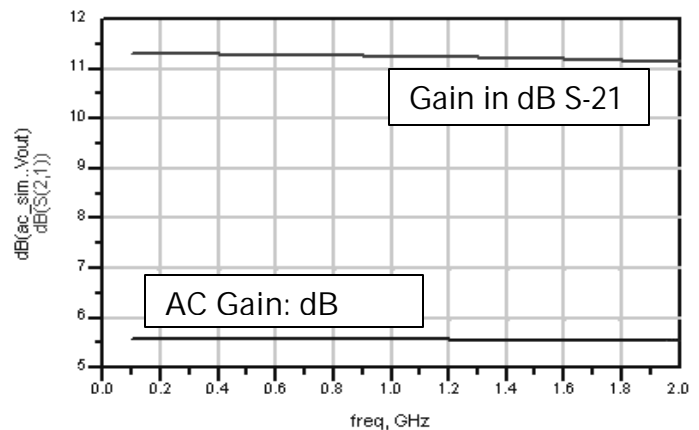


2. Simulate and display results: use the data display options

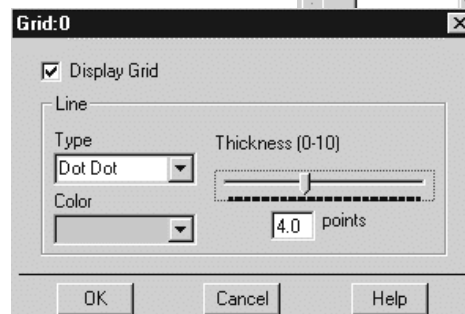
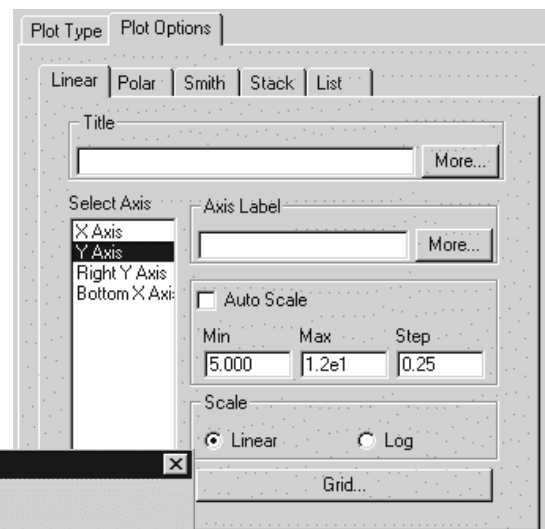
The following steps will compare the S21 measurement to the AC simulation data and show you more about plots, traces, markers and text.

- Be sure the name of the dataset is: **s_params** and then **simulate**. It is a good idea to always check dataset names before simulating.
- When the simulation is finished, open a **new Data Display** window and save it as **s_params**. Then insert a rectangular plot of S-21 (dB).
- On the same plot, insert the trace the db of **Vout** from the **ac_sim** dataset from the last lab.

NOTE on S-21 vs dB gain from AC simulation: The dB values of gain are the same because the AC simulation uses the standing wave voltage only. But the S-parameter simulation uses output power divided by incident power (V and I). In addition, the S-parameter source impedance is 50 ohms and the V_AC source 0 ohms.



- Edit the plot (double click). Go to **Plot Options** and try adjusting both axes by deactivating the auto-scale and setting: Min, Max and Step.
- Reset the axes to Auto Scale and try the plot-zooming icons. Reset to Auto Scale when finished.

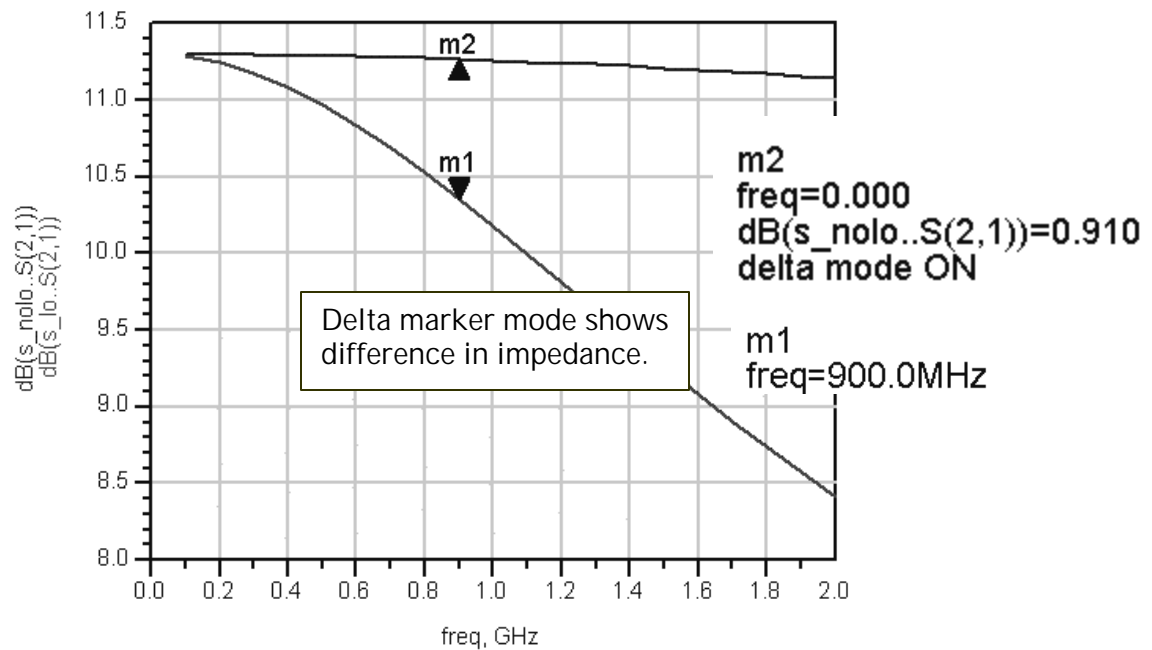
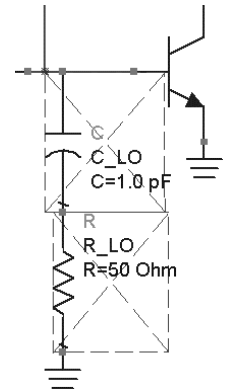
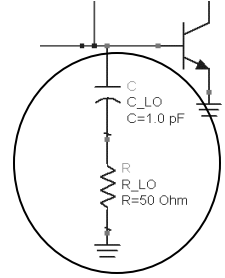


- Also, click on the Grid button and try changing the grid as desired.

You may want to use these features later.

3. Add the LO impedance: series R-C to ground

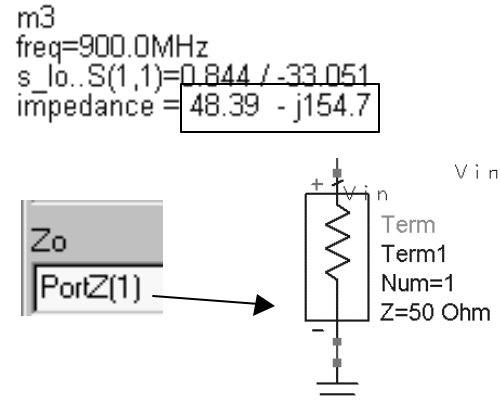
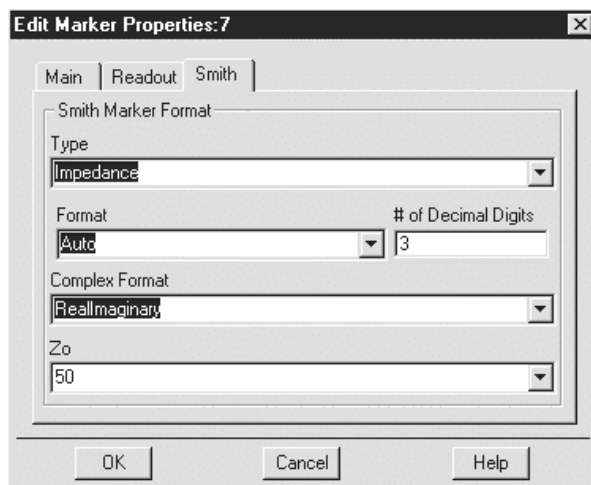
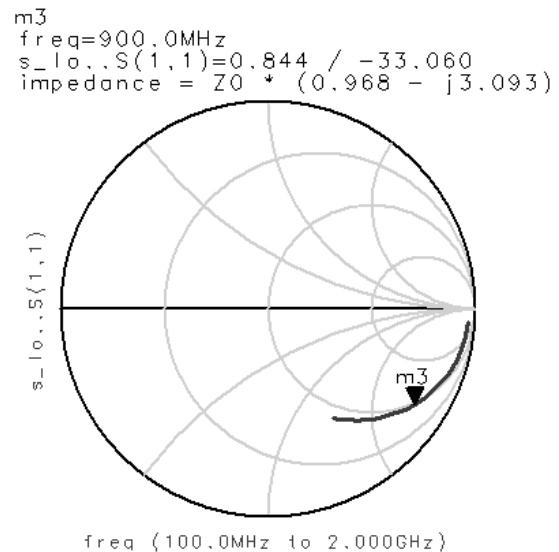
- In the schematic, insert the following components to represent the impedance of the local oscillator: **C=1.0 pF** and **R=50 Ohms** and ground. Insert onto the transistor base.
- Simulate S-21 again** with the frequency range from 100 MHz to 2 GHz (100 MHz steps) and name the dataset: **s_lo**.
- Deactivate the LO components, and simulate again with the dataset name: **s_nolo** to compare results.
- In the data display window, insert a new plot with the two S-21 traces: with the LO and without the LO impedance. The display should look similar to the one shown here. Select the two markers and click **Marker > Delta Mode On**. The delta S-21 is about 0.9 dB.



- Save the data display window as **s_params**. Remember the data display windows are .dds files and the datasets are .ds files in the data directory.

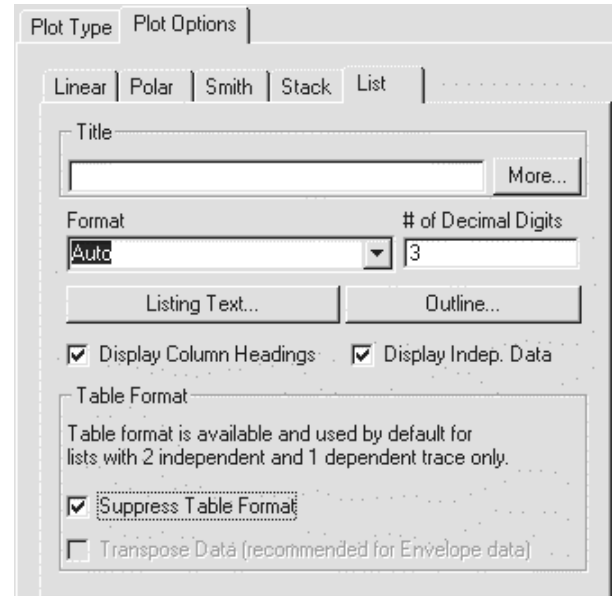
4. Plot the S11 impedance

- In the same data display window, insert a Smith chart with the S-11 trace from the **s_lo** dataset. Insert a marker at 900 MHz and notice that the marker shows the reflection coefficient (magnitude/angle) and also the impedance (real and imaginary). For the design, there is an obvious mismatch here.
- Edit the marker readout (double click on the marker readout). When the dialog appears, change Zo to 50 and click OK. The marker will now give you the real and imaginary value of S-11 in ohms:



- Change the Zo setting to PortZ(1) as shown and you will get the same answer as setting Zo to 50. However, if you did have a different port impedance (for example Z=75 Ohm) then the PortZ setting would calculate the readout using 75.

- d. Insert a **list**. Then select **S** data and **PortZ** and add them. Then use **Plot Options** to **Suppress Table Format**. This will display all four S-parameters in a tabular format as opposed to selecting only one of the S values. In addition, the port impedance of the termination is given too.
- e. In the Data Display window, use the scroll buttons or zoom icons to view the data. Also, try changing the font type and size in the list (you may need this for a presentation) by clicking the command: **Text > Font**.



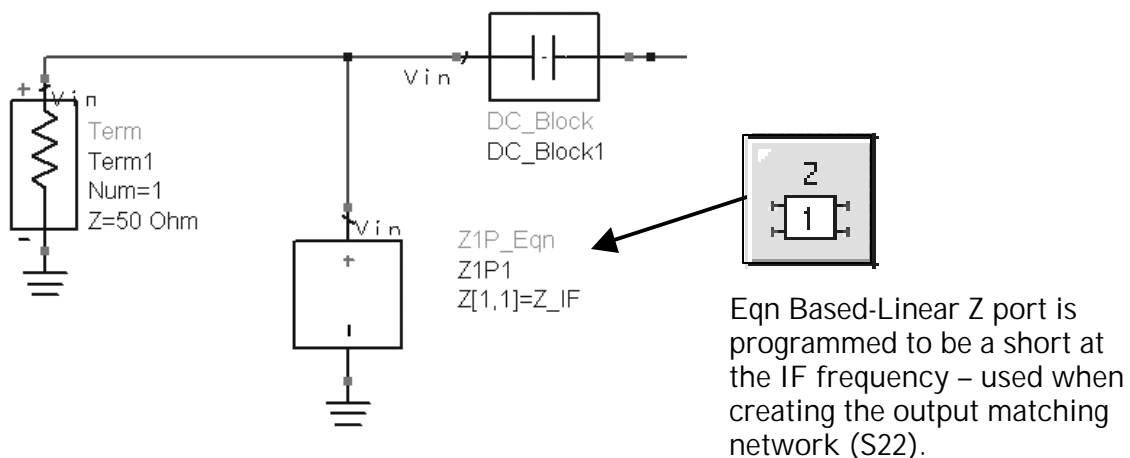
Font			
Larger			
Smaller			
6 point			
8 point			
10 point			
12 point			
14 point			
16 point			
20 point			
24 point			
s_lo..S		freq	s_lo..PortZ
s_lo..S(1,2)		s_lo..PortZ(1)	
100.0MHz	0.967 / -4.013	100.0MHz	50.000 / 0.000
200.0MHz	0.962 / -7.996	200.0MHz	50.000 / 0.000
300.0MHz	0.952 / -11.920	300.0MHz	50.000 / 0.000
400.0MHz	0.940 / -15.759	400.0MHz	50.000 / 0.000
500.0MHz	0.924 / -19.491	500.0MHz	50.000 / 0.000
600.0MHz	0.907 / -23.099	600.0MHz	50.000 / 0.000
700.0MHz	0.887 / -26.568	700.0MHz	50.000 / 0.000
800.0MHz	0.866 / -29.890	800.0MHz	50.000 / 0.000
900.0MHz	0.844 / -33.060	900.0MHz	50.000 / 0.000
1.000GHz	0.821 / -36.077	1.000GHz	50.000 / 0.000
1.100GHz	0.798 / -38.942	1.100GHz	50.000 / 0.000
1.200GHz	0.775 / -41.659	1.200GHz	50.000 / 0.000
1.300GHz	0.753 / -44.234	1.300GHz	50.000 / 0.000
1.400GHz	0.731 / -46.672	1.400GHz	50.000 / 0.000
1.500GHz	0.709 / -48.982	1.500GHz	50.000 / 0.000
1.600GHz	0.688 / -51.171	1.600GHz	50.000 / 0.000
1.700GHz	0.668 / -53.246	1.700GHz	50.000 / 0.000
1.800GHz	0.649 / -55.215	1.800GHz	50.000 / 0.000
1.900GHz	0.630 / -57.086	1.900GHz	50.000 / 0.000
2.000GHz	0.612 / -58.866	2.000GHz	50.000 / 0.000
s_lo..S(1,2)		s_lo..PortZ(2)	
100.0MHz	8.873E-4 / -1.966	100.0MHz	50.000 / 0.000
200.0MHz	8.830E-4 / -3.905	200.0MHz	50.000 / 0.000
300.0MHz	8.761E-4 / -5.792	300.0MHz	50.000 / 0.000
400.0MHz	8.667E-4 / -7.604	400.0MHz	50.000 / 0.000

5. Add frequency sensitive terminations for the RF & IF

Frequency sensitive Z ports from the Equation-based Linear component library allow you to describe how a termination responds to changes in frequency. For example, for S11 matching, the Z port on the output is programmed to be an RF short to ground like a filter that you would use. Conversely, the Z port at the input is programmed to be an IF short to ground. This provides a better approximation of the final S-parameters after creating the matching networks.

NOTE: You will move components around and rewire several items to complete the steps. Take your time and create an organized schematic.

- From the palette, select the **Eqn Based-linear** components and insert a one port **Z Eqn** in parallel with the input.



- Assign the value of $Z[1,1]$ to be a variable: **Z_IF**.
- Insert another **Z 1-port Eqn** in parallel with the output impedance and assign the value of $Z[1,1]$ to be the variable: **Z_RF**.
- Insert a **VAR** (variable equation) and edit it to declare the values of Z_{IF} and Z_{RF} as shown here:



```

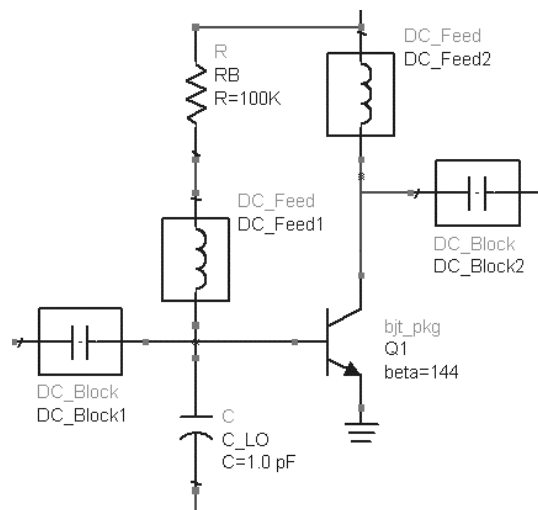
VAR
VAR_Z
Z_IF=if freq <100 MHz then 0.001 else 1e99 endif
Z_RF=if freq >100 MHz then 0.001else 1e99 endif

```


6. Insert ideal DC feed inductors

- Go to the Lumped Components palette and insert a **DC feed** inductor between the transistor base and the resistor RB.
- Insert another **DC feed** between the collector and the resistor RC.

Partial view of schematic shows DC_Feed components used as ideal inductors: short to DC and open to AC

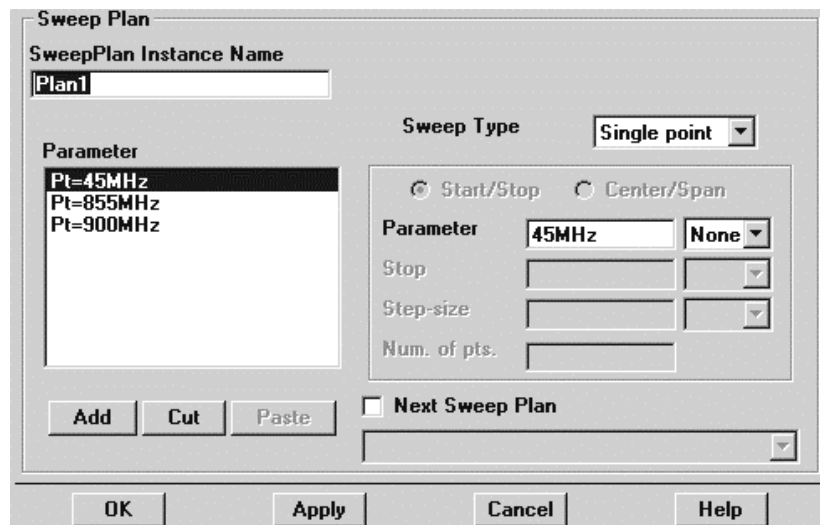


7. Set up a Sweep Plan (SweepPlan)

- Sweep Plans are in simulation palettes - insert a **Sweep Plan**.
- Set the sweep for three single points: RF, IF and LO frequencies. First, click Sweep Type and set it to **Single point**. Then type in the frequency points and click the **Add** button after every entry. Use **Add** or **Cut** to remove or change the position of any unwanted parameters.



- Click **Apply** and **OK** when the dialog has these entries only.



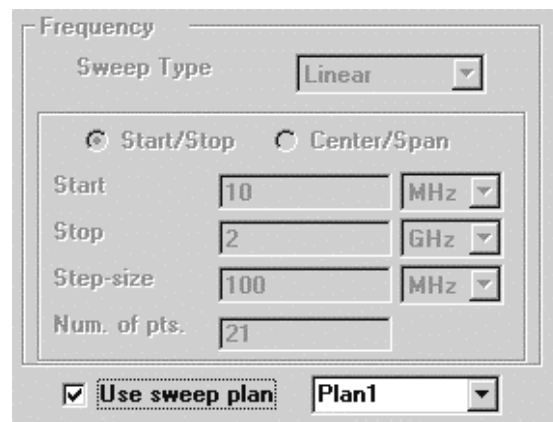
8. Edit the S-Parameter Simulation Controller

- a. Select the S-parameter simulation controller and click the **Edit icon** – this is the same as double clicking on the component.

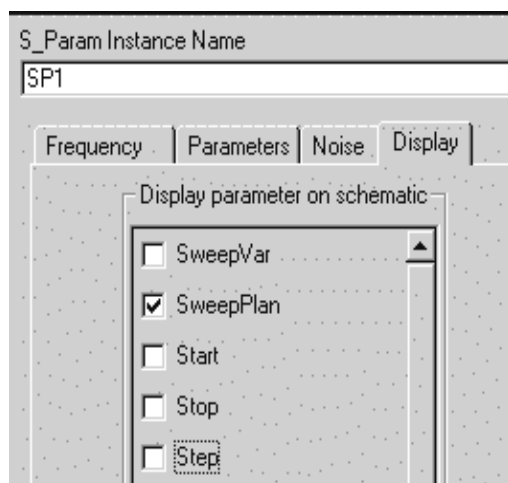


- b. In the **Frequency tab**, click the box **Use sweep plan** and select the sweep plan **Plan 1** which you have already set up – the controller recognizes inserted sweep plans. Click Apply and note that the start and stop settings should now be grayed out.

NOTE: Please be sure that the Sweep Type is **NOT** Single Point. If it is, only one Frequency would be used for simulation.



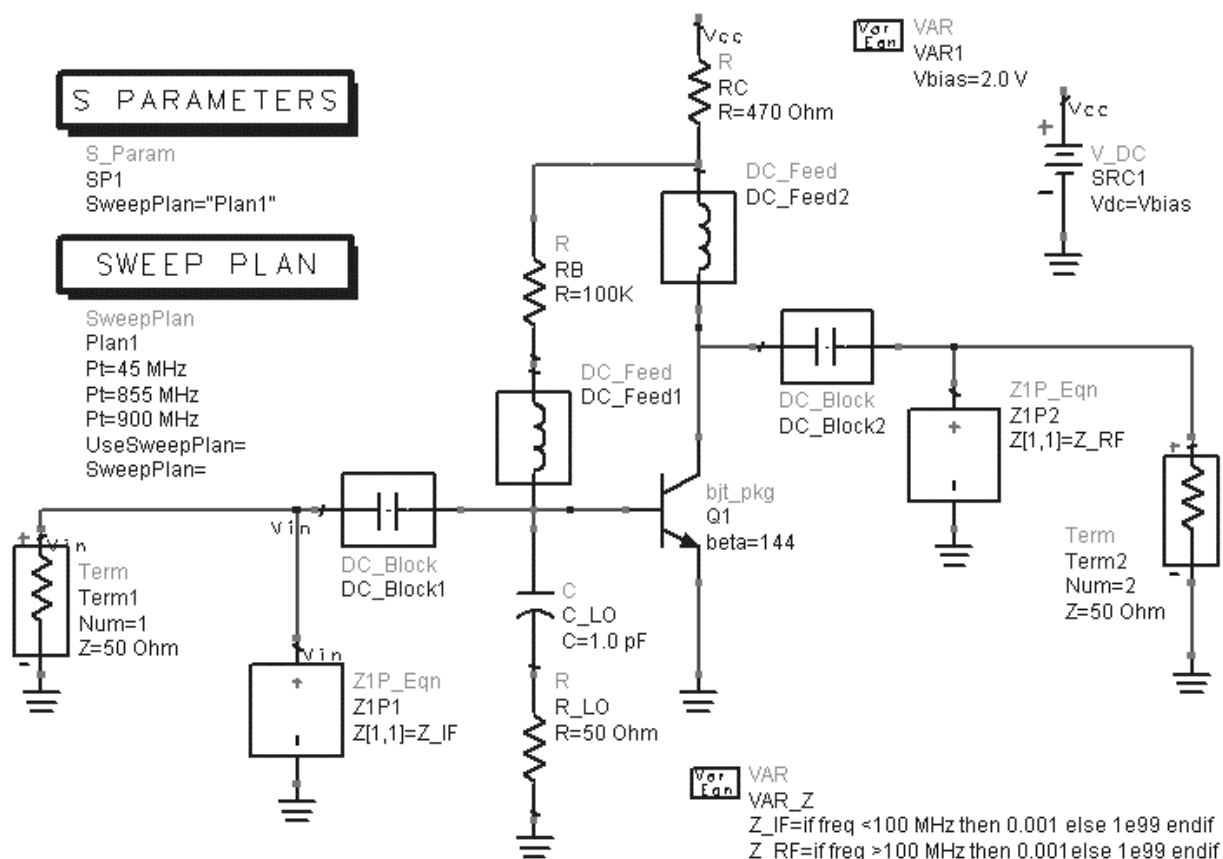
- c. In the Display tab, check the SweepPlan and remove the Start, Stop and Step check boxes. Click OK and the simulation controller should now look like the one here.



S-parameter simulation controller set to assign FREQ to be the sweep plan values.



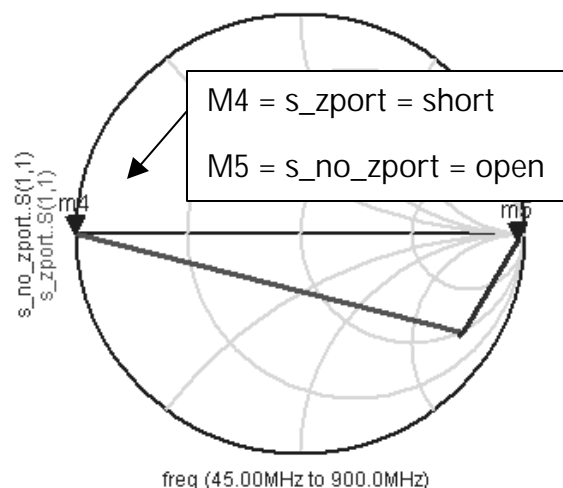
9. Check the circuit - it should look like the one shown here:



- Be sure the LO impedance is activated. Set up a new dataset name: **s_zport** and **Simulate**. Then plot the **S-11** data on a Smith Chart.
- Deactivate the Z-ports**, and simulate again with the dataset name: **s_no_zport**.

- Add the **s_no_zport** trace to the Smith chart and notice the difference at 45 MHz between the two traces by putting new markers at 45 MHz for each trace. As you can see, the IF response is a short with the Z port and almost an open without the Z ports (with opposite phase).

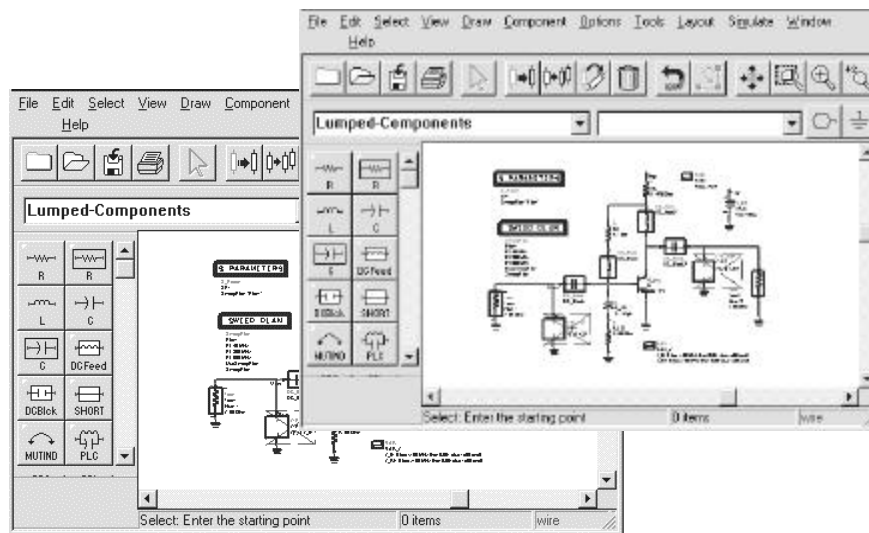
- Save** the data and schematic (s_params).



10. Modify the BJT_PKG sub-circuit (B-C capacitance)

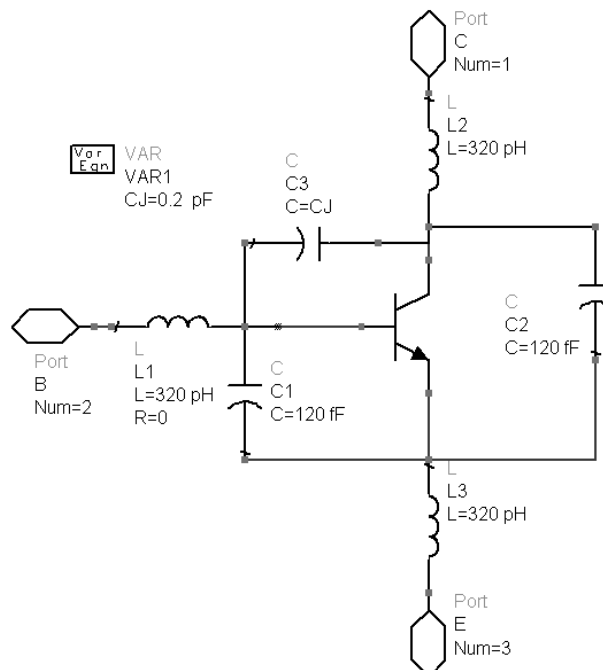
The following steps will further demonstrate the use of Z ports, sub-circuits and simulations in the hierarchy.

- Open a second schematic window** as follows: from the current schematic window (s_params) click Window > Schematic or use the **hot keys: Ctrl+Shift S**. Now you have two windows of the same design. This is good for viewing details of a large schematic while keeping the overall design viewable.



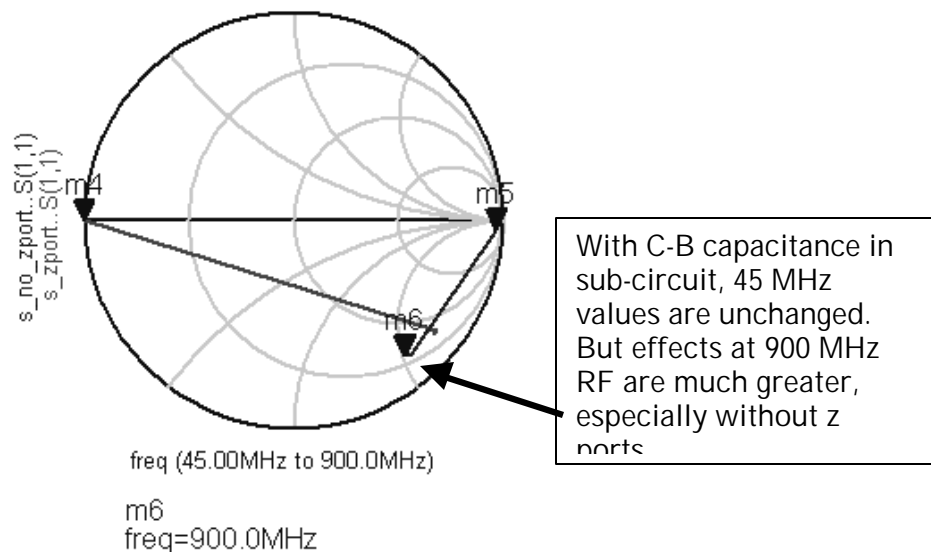
- In the new schematic window, **push into the bjt_pkg**. Now, in this lower level hierarchy, add a capacitor between the collector and base **C=CJ** and add a **VAR = 0.2 pF** being sure to put spaces between the number and the units as shown.

NOTE: This will have a similar effect as modifying the Cjc parameter in the model card.



- c. At this lower level, press the **F7 simulate** key. You should get an error message in the status window because there is no simulation controller.
- d. Move the cursor back to the other schematic window (higher level hierarchy with the simulation controller). Keeping the same dataset name from the previous simulation (**s_no_zport**), and **the z ports deactivated, simulate** and note the change to the S11 data.

DESIGN NOTE: With the added capacitance (base to collector), at 45 MHz there is little or no change. But at 900 MHz, the input impedance is more capacitive as it would be with a real device. Also, the z port (Z_RF) now shows that it can be used to create a mathematical representation of the behavior of a matching network. In this case, it will allow you to start designing the input match with the output already represented, thus eliminating iterations.

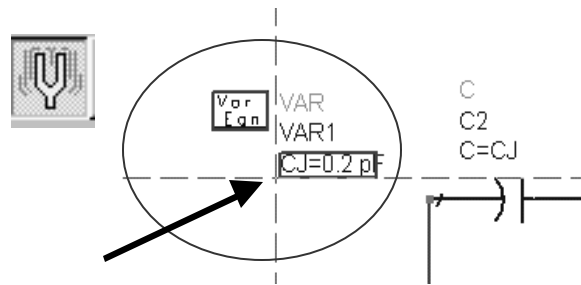


11. Tune the Sub-circuit Variable

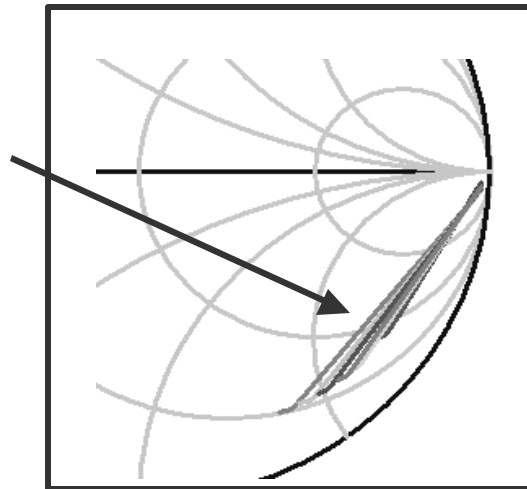
The following step shows how to tune a variable (VAR) in a sub-circuit and keep the data separate from the existing datasets. This will require moving windows and dialogs around on the screen and using both schematic windows. This step is not critical to designing the mixer but it does demonstrate a procedure you may need in your own designs later on.

- a. Setup a new **dataset name: s_cj_tune** and **Simulate** again. This will create a dataset.
- b. **Close the existing data display** window and **open a new data display** window. The default dataset name s_cj_tune should appear. Insert a Smith chart of the S-11 data.

- c. In the upper level schematic, start the **Tune mode** (Simulate> Tuning). You will see the status window and the Tune control dialog appear. Next, move the cursor back to the **bjt_pkg** sub-circuit window and select the **CJ** parameter value.



- d. Position the data display window so you can see the new trace values resulting from the tuning. Here is a case where small changes occur and so you can zoom into the Smith chart.



- e. To easily end tuning, press the keyboard **Esc** key or use the right mouse button End Command.
- f. **Delete the Smith chart** so the data display is empty.
- g. In the subcircuit: **remove the Capacitor and VarEqn**.

NOTE: Save the s_params design. It will be used for the next lab to develop the final input and output matching networks for the mixer.

NOTE on the next 2 steps: Do these only if you have time. They are not required for the mixer design. They demonstrate how to write or read data into ADS.

12. Reading and Writing S-parameter Data with an S2P file

You can read or write data in Touchstone, MDIF, or Citifile formats. ADS can convert supported data into the ADS dataset format. Typically, these data files are put in the project directory but they can also be sent to the data directory. You can control where they reside.

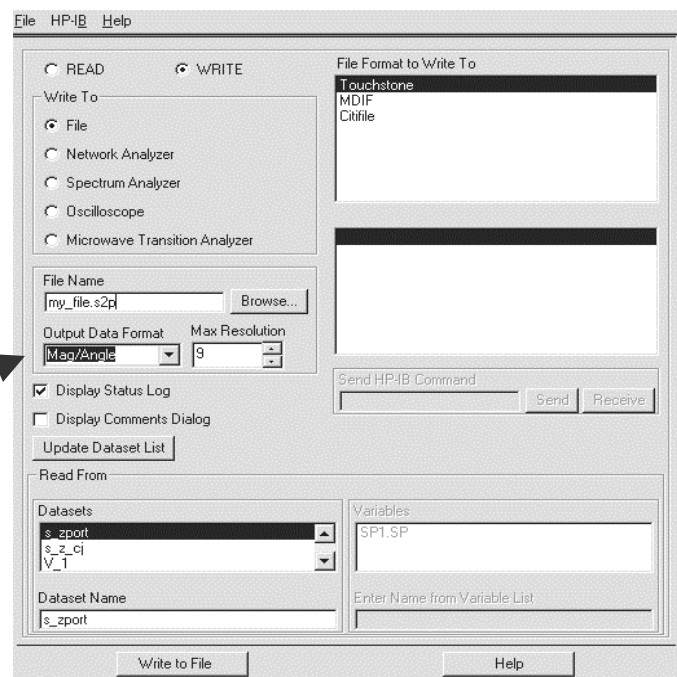
- a. In the data display window (s_params.dds), click on the HP-IB icon (Instrument Server).



- b. When the dialog box opens, click the box to **WRITE** and Write to **File** then select the **Touchstone** format.

IMPORTANT NOTE:

Be sure to select the type of Output Data Format you want whenever you use this feature. For this lab, select Mag/Angle.

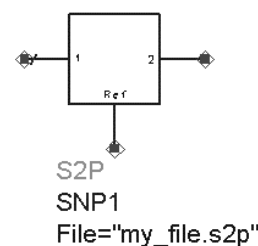


- c. In the File Name field, instead of using the name *default*, type in the name: **my_file.s2p**.
- d. Select the Output Data Format as **Mag/Angle**. In the Datasets field, select the dataset **s_zport** to be translated and select the variables (shown above).
- e. Click the **Write to File** button and then check the Status Window. If everything is correct, you will get a message confirming the dataset write was successful: my_file.s2p is now a Touchstone file.

13. Assign the S2P component to the data file and simulate

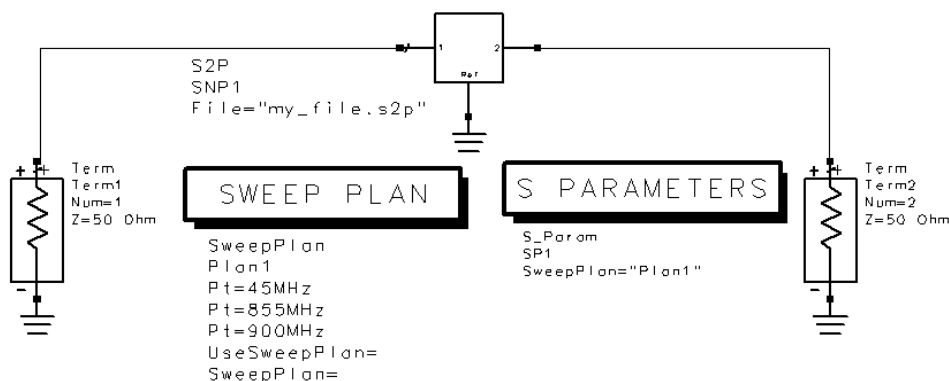
In this step, you will write and read an s-parameter Touchstone file using the s2p component. This is similar to downloading an s-parameter file from the web for use in a simulation.

- Open a new schematic window (untitled) using **Ctrl N** where N means new (schematic).
- Insert an **S2P** component (type it in or get it from the palette **Data Items**. Notice that the component variable (*file=*) is not yet assigned.
- To assign the data, type in the file name or edit the S2P component. Another dialog box will appear. Now, set the browser to look for **All Files (*.*)**.
- Next, **browse** for the file in the directory where the data was written. Then use the **Open** button to select the file: **my_file.s2p**.



NOTE: You can use a text editor to look at or to modify the values.

- In the untitled schematic insert the template: **S-param** (command: File > Insert Template) and wire the S2P component to the ports and insert a reference ground
- Go back to the **s_params** schematic and copy the **Sweep Plan** to the buffer and paste it into the untitled schematic. Set the Simulation controller to use the **Sweep Plan**.
- Simulate** with the dataset name: **my_s2p**. Plot the results in the Data Display window to verify the simulation.



- Save the design and data with the name **s_2p** and close the windows.

EXTRA EXERCISES:

1. Translate data from a Touchstone file into a dataset. Use the **Instrument Server** window and **READ** the file back into the data directory as an ADS dataset. Then simulate the S2P component with another name and save the file.
2. Use a real library device and simulate both with and without the Z ports to actually see the difference in S11. For example, use a device from one of the BJT libraries.
3. Try writing an equation to vary the value of a package parasitic – for example, a value of L that varies with frequency:



L2
 $L = L_0 * (1 + 0.1 * (\text{freq} / \text{freq}_0))$

Var
Eqn

VAR2
 freq_0=1 MHz
 L_0=1 nH

4. Try writing an equation so that the port impedance changes with frequency and then verify that the marker readout calculates the proper value of impedance.
5. Use a Z 2-port and create a conjugate match based on the initial S11 data.