LAB 4: AC Simulations

Overview - This lab continues the amp_1900 project and uses the same sub-circuit as the previous lab. This exercise teaches the basics of AC simulation, including small signal gain and noise. It also shows many detailed features of the data display for controlling and manipulating data.

OBJECTIVES

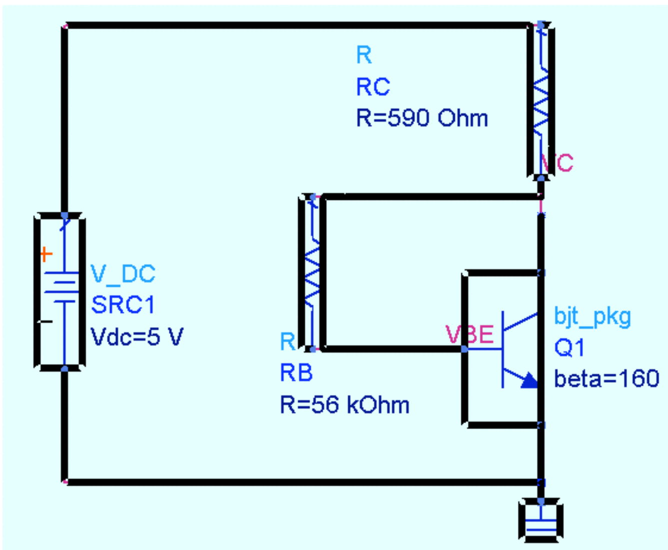
- Perform AC small signal and noise simulations.
- Adjust pin/wire labels.
- Sweep variables and write equations.
- Control plots, traces, datasets, and AC sources.

Table of Contents

PROCEDURE

1. Copy & Paste (Ctrl+C / Ctrl+V) from one design to another.

a. Open the last design (dc_net) and copy the circuit shown highlighted here by dragging the cursor around the area - this is known as rubber banding. With the items highlighted, copy then by using the keyboard keys **Ctrl** + **C** or the **Edit > Copy** command. Using Ctrl + C is preferred because it eliminates mouse clicks.



b. Use the **File> New Design** command to create a new schematic and name it: **ac_sim**. Then use **Ctrl** + **V** or use **Edit > Paste** and insert (ghost image) the copy by clicking into the new schematic.

c. **Save** the **ac_sim** design. You must save it or it will not be written to the database.

d. Click the command **Window > Designs Open**. This command gives you access to designs that are open in memory but not visible in a window or not saved in memory. When the dialog appears, select **dc_net** and click **OK**. Then close dc_net design using **File > Close Design** (no need to save the changes).

e. In the empty schematic window, reopen the ac_sim design using the **File > Open Design icon**. This gives you a list of all the designs in the project. If a design is created but not saved initially, it will not be in this list and you will need to use the command Window > Designs Open to access it.

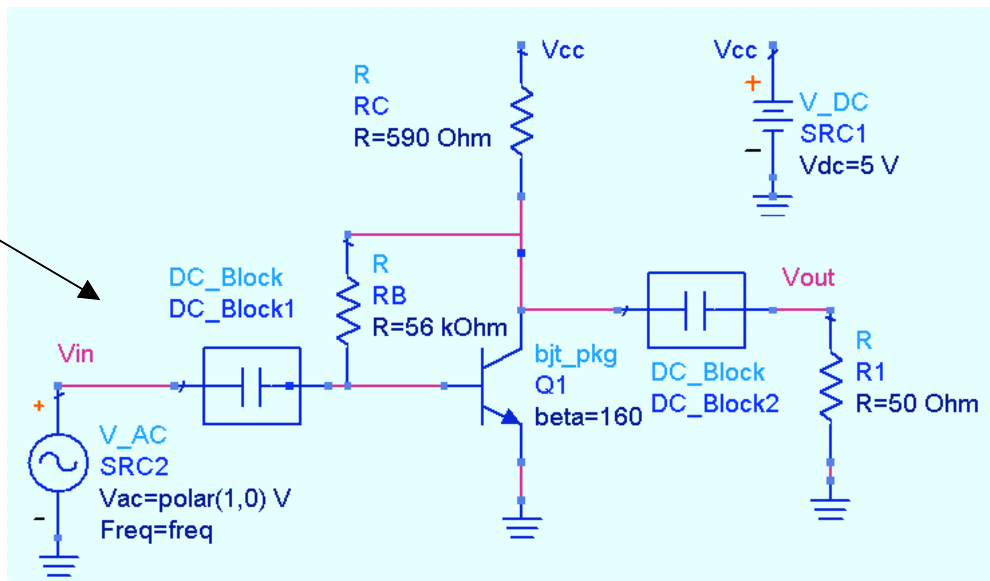
2. Modify the copied circuit and pin labels.

Delete wires, insert new components, and rewire as needed. The steps follow:

a. Disconnect the DC source and move it to the side with a ground.

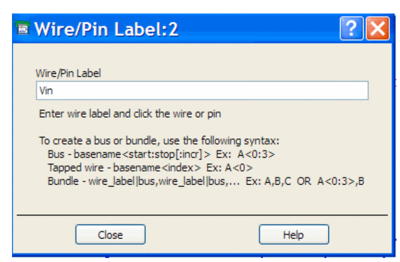
b. Insert two ideal **DC_Block** capacitors from the Lumped-Components palette or use component history.

c. Insert a **V_AC** source from the Sources-Freq Domain palette. Ground the source. Then add a **50 ohm** load resistor and ground to the output.



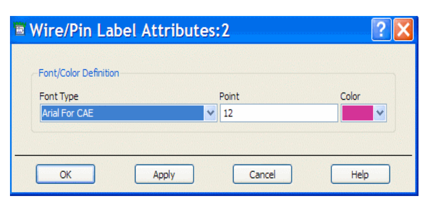
d. Modify the Pin/Wire (node) labels. Click the **Name** icon. Add **Vcc** as a label to both RC and the DC source. This will connect them electrically instead of a wire.

e. Add **Vin** and **Vout** as shown. Also, if you did any OPTIONAL steps in lab 3, <u>remove VC and VBE</u> by by clicking on those labels when the dialog is blank blank (shown here) or use the command**: Edit >Wire/Pin Label** > **Remove Wire/Pin Label.**



f. Verify that the circuit looks like the one shown here. here.

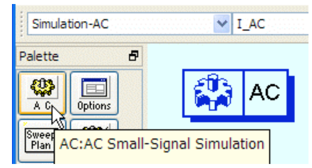
NOTE on Wire/Pin Label Attributes: You can drag labels to move them and you can edit attributes by double clicking on them or by using the command: **Edit >Wire/Pin Label** > **Wire/Pin Label Attributes.**



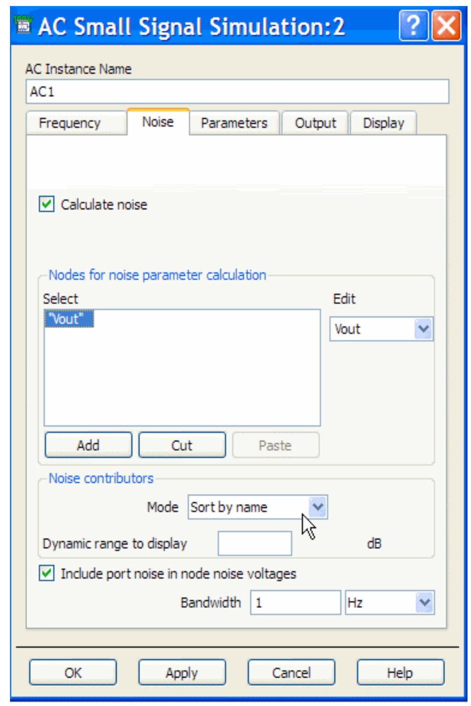
3. Push and pop to verify the sub circuit.

a. Select the bjt_pkg and **push** into the sub-circuit (use the icons) to to check your sub circuit, and then **pop** out again.

4. Set up an AC simulation with Noise.

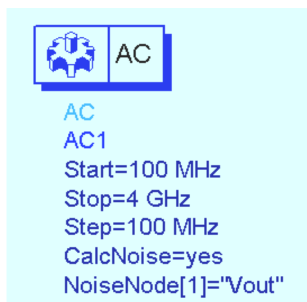


a. Insert an AC Simulation controller. Then edit the start, stop, start, stop, and step frequencies: 100 MHz to 4 GHz in 100 100 MHz steps.



b. In the Noise tab, check the box for **Calculate noise noise** and add the **Vout** node. Set the Mode to **Sort** to **Sort by Name** for each noise contributor. Sort by Sort by value is good for large circuits to see the the largest contributors first. Also, all noise values values will be simulated if a Dynamic range (threshold) is not set.

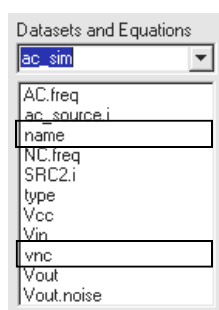
c. Turn on the **Display** for each of the parameters as shown here.



5. Simulate and list the noise data.

a. **Simulate** (F7).

b. In the data display, insert a **list** (icon) of **name** and **vnc** (voltage noise contributors) using the **Ctrl** key to select them both. As shown here, at each frequency, Q1.BJT1 is the total noise voltage for the device and is composed of: Q1.BJT1.ibe and Q1.BJT1.ice. However, these are not correlated voltages but have been added as noise powers: $(V_{total})^2 = (V_{ibe})^2 + (V_{ice})^2$. The total vnc is the same as Vout noise.



NOTE: The index will automatically appear when you list the data.

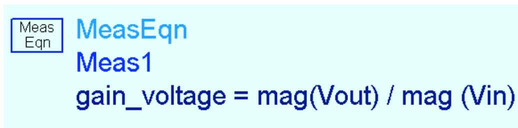
index	vnc	name
freq=100.0MHz		
0	1.728nV	_total
1	1.494nV	Q1.BJT1
2	8.476pV	Q1.BJT1.ibe
3	1.494nV	Q1.BJT1.ice
4	833.3pV	R1
5	24.90pV	RB
6	242.6pV	RC
7	0.0000 V	SRC2

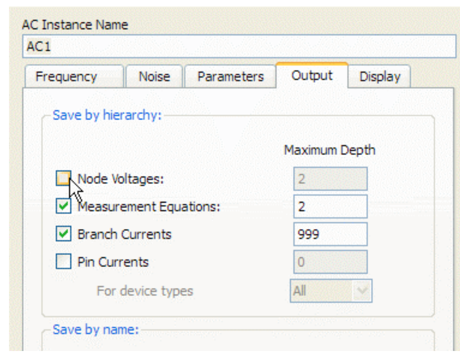
c. **Save** the schematic and data display.

6. Control the output of equations and node voltages.

a. In the ac_sim schematic, insert a **MeasEqn** from any simulation palette. Or, palette. Or, you can type in **MeasEqn** in component history.

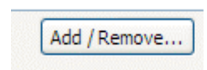
b. Directly on the schematic screen, edit (type) the equation to compute voltage gain using the node (pin) labels Vin and Vout. Use the keyboard arrow key to move across the equal (=) sign.



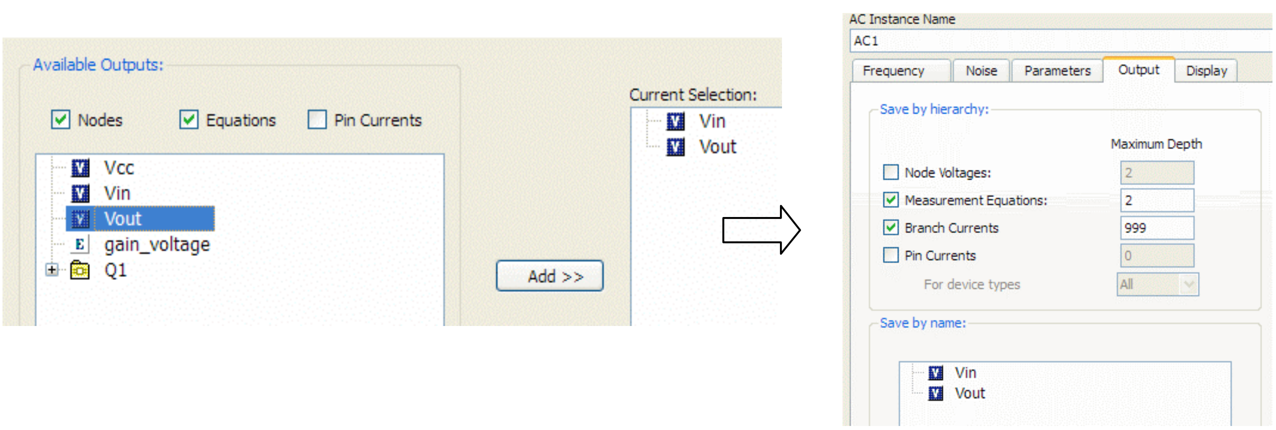


c. Edit the AC simulation controller and go to the **Output** tab. The default is for all labeled node voltages (pin/wire labels) and all Measurement Measurement equations to be reported in the dataset. You will change this in the next steps.

d. **Uncheck** the box for Node Voltages and click on the **Add/ Remove** button.



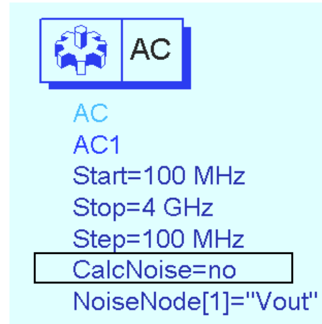
e. Select **Vin** and **Vout** from the list of available outputs and **Add** **Add** them as shown here - then click **OK**. Only those node voltages will be written into the dataset after simulation and <u>Vcc will not</u>. This works for measurement equations also.



f. Click **OK** to dismiss the dialog – you are now ready to simulate.

NOTE on node name display: You can display the node names (Display tab – NodeName check box) but it is not necessary.

7. Simulate without noise.



a. In the schematic, turn off the noise calculation by changing (typing) yes to **no** as shown here. This will save simulation time and memory, especially for large circuits. Of course, this will make your dataset list (name and vnc) invalid.

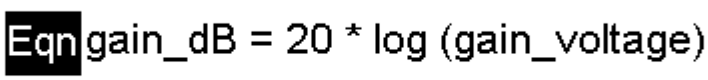
b. Save the schematic and **Simulate** (F7).

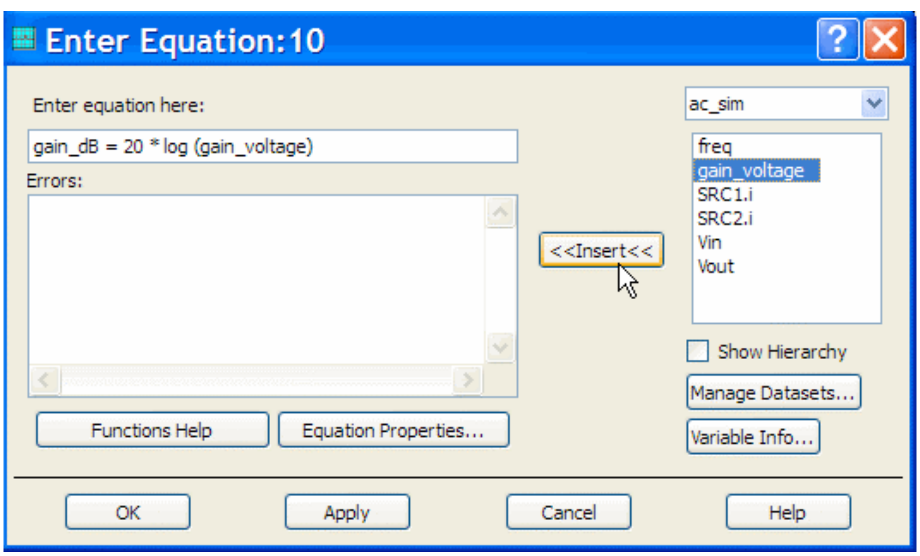
8. Write a data display equation using a measurement equation.

a. In the data display, **delete** the invalid noise listing.

b. Insert a data display equation (use the icon).

c. In the dialog, write an equation for the gain in dB as shown here. Notice that you are inserting the schematic measurement equation into your data display equation and click **OK**:

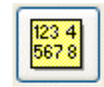




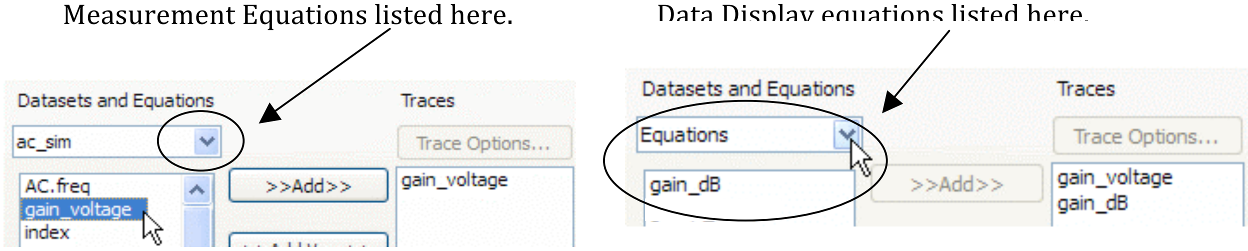
Note on equations - If the measurement equation for voltage gain was not already calculated, you would write the data display equation with all the required values, for example: gain_dB = 20 * log (mag (Vout) / mag (Vin)). However, because that voltage gain was already calculated, it is easier to simply insert it here.

9. Work with measurement and data display equations.

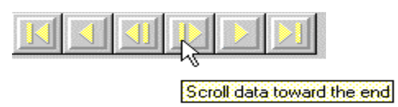
a. Insert a list and add the measurement equation **gain_voltage** and also add also add the data display equation **gain dB** as shown here. Schematic Schematic measurement equations are automatically written to the dataset. dataset. But data display equations are not. Instead, they are stored in the in the data display Equations memory and are selected and added as shown here. Click OK and both equations will appear in the list.



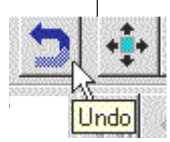
Measurement Equations listed here. Data Display equations listed here.



b. Select the list and scroll down to 1900 MHz using the list the list scroll buttons shown here. Then insert the the cursor directly into the gain_voltage column heading heading and type in the **dB** function as shown – be sure to add **parentheses** so that it reads: **dB (gain_voltage)**. This demonstrates the flexibility of the data display for operating (with ADS functions) directly on data and equations.

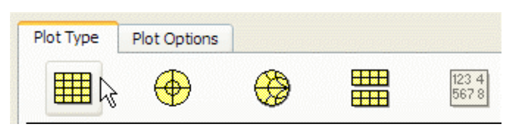


freq	dB(gain_voltage)	gain_dB
1.500GHz	14.736	14.736
1.600GHz	14.649	14.649
1.700GHz	14.559	14.559
1.800GHz	14.465	14.465
1.900GHz	14.368	14.368
2.000GHz	14.268	14.268
2.100GHz	14.165	14.165
2.200GHz	14.061	14.061
2.300GHz	13.953	13.953



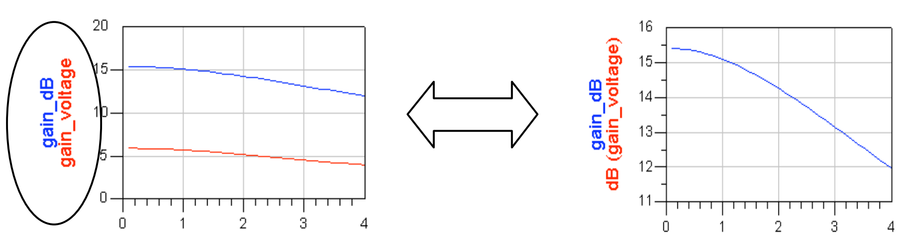
c. Click the data display **Undo** icon remove the dB function.

d. Edit the list (double click) and change it to a a **rectangular plot** by selecting the icon.



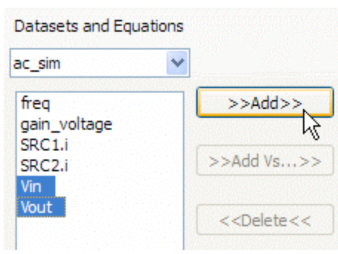
e. Insert the cursor directly onto the **Y-axis** label and change gain_voltage to **dB (gain_voltage)** similar to the way you did in the list. Then **undo** it. Again, this shows the power of functions and the data display.

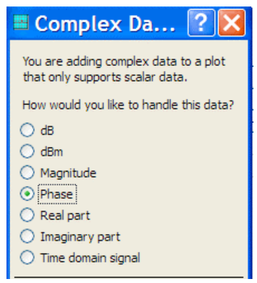
NOTE on dB values – Converting the AC analysis voltage to dB is not the same as S-parameter analysis in dB that uses power (V and I) and also has a 50 ohm source Z.

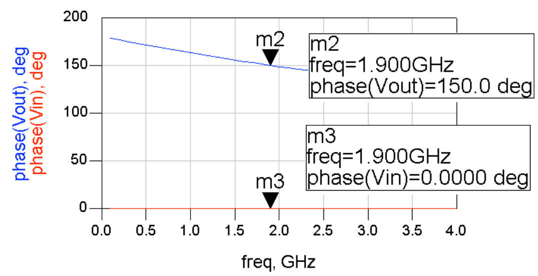


10. Plot the phase and group delay for the ac analysis data

a. Insert a rectangular plot of the **phase** of **Vin** and **Vout** - put markers on 1900 MHz (type in the value). The phase is not 180 degrees due to the bjt_pkg parasitics. Move the markers and see the phase closer to 180 at lower frequencies. You may want to ***Hot Key*** the new marker command using the DDS Options > Hot Key similar to schematic.

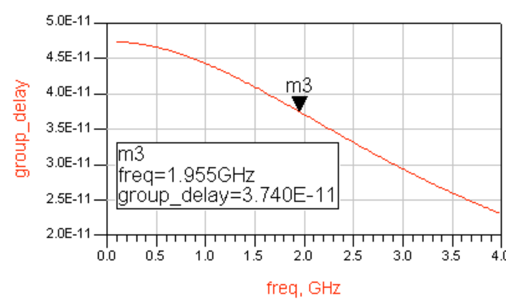






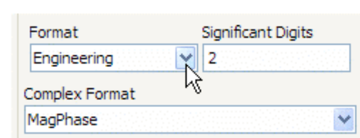
b. Insert a new **equation** to calculate **group delay**. As shown here, use the phase of Vout and the ***diff*** function then **plot** the equation. The ***diff*** function calculates the difference between points on the slope. The minus sign gives the result in decreasing value. Place a marker on the trace and notice that it will be on either side of 1900 MHz (+/- 50 MHz) because of the *diff* function.

Eqn group_delay= (1/360) * (- diff (phase(Vout)))

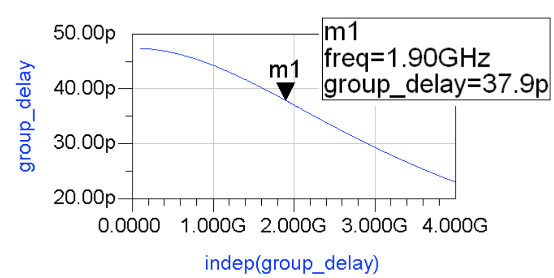


c. Go back to the schematic, change the **step** size to **10 MHz**, **simulate** again and watch the plot update.

d. Edit (double click) the **marker.** In the **Readout** tab, set Format to **Engineering** with **2** with **2** significant digits as shown here. Notice the the marker value changes to pico (pico-seconds) and seconds) and the independent value resolves to 1.90 to 1.90 GHz.



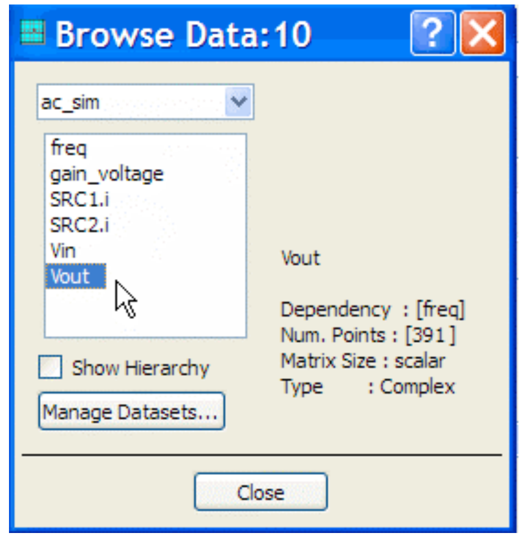
e. OPTIONAL - Try grouping the group group delay equation and the plot so plot so they stay together when you you move them. Use the Shift key and key and select the plot and the equation. Then click**: Edit > Group**. **Group**. They should now move together in the data display.



11. Variable Info and the *what* function.

a. Insert a new **list** (dataset is still **ac_sim**). Add **Vout,** select it, and click on the **Trace Options** button. You can do this in a new page if desired or zoom out by 2 for more room on the display.

b. When the dialog box appears, click on the the **Variable Info** button and another dialog dialog will appear as shown here. Select the the **Vout** data and you will see that the dependency for Vout is 391 frequency points. points. This should be the same for all the items the items in the dataset because only frequency frequency was swept.



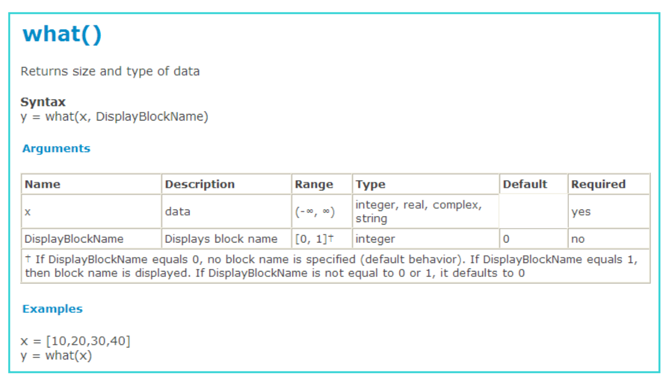
c. Close the dialog, click OK, and go back to the list the list of Vout. Insert the cursor in the **Vout Vout** column and type in the *what* function as function as shown: **what (Vout)**. Notice that that you get the same variable information. information. Later on, you will use this function function to determine how to index into dataset dataset tables with multiple sweeps or mixing products.

freq	Vout
100.0MHz	5.901 / 178.294
110.0MHz	5.901 / 178.123
120.0MHz	5.900 / 177.953
130.0MHz	5.900 / 177.782

→

what (Vout)
Dependency : [freq]
Num. Points : [391]
Matrix Size : scalar
Type : Complex

NOTE on functions: You can read about the *what* function and other ADS functions (abs, real, s_stab_circle, etc.) by clicking the Functions Help button whenever you insert an equation in the data display or whenever you go to Trace Options. When the Help browser appears, scroll down to the function of interest. Try this and look over some of the information to see how ADS functions are described if you have time.



what()

Returns size and type of data

Syntax
y = what(x, DisplayBlockName)

Arguments

Name	Description	Range	Type	Default	Required
x	data	(-∞, ∞)	integer, real, complex, string		yes
DisplayBlockName	Displays block name	[0, 1]†	integer	0	no

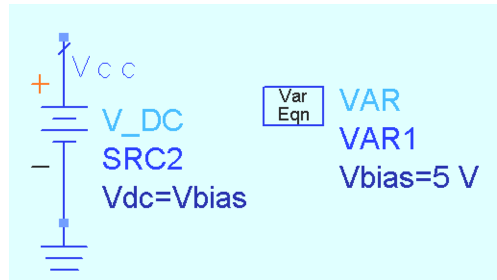
† If DisplayBlockName equals 0, no block name is specified (default behavior). If DisplayBlockName equals 1, then block name is displayed. If DisplayBlockName is not equal to 0 or 1, it defaults to 0

Examples

x = [10,20,30,40]
y = what(x)

12. OPTIONAL - Sweep Vcc (as if the battery voltage is decreasing)

This step will require you to use the skills you already learned in the previous lab exercises. You will set up a parameter sweep for **Vcc** from **2** volts to **5** volts in **0.25** volt steps.

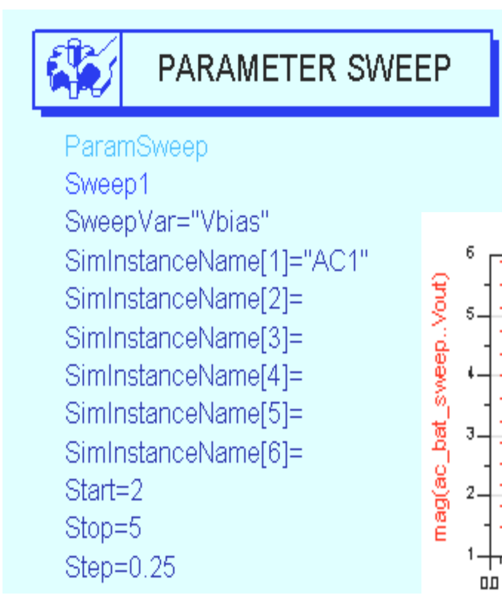


a. In your schematic, insert a **VAR** (variable equation) initializing **Vbias = 5 V**.

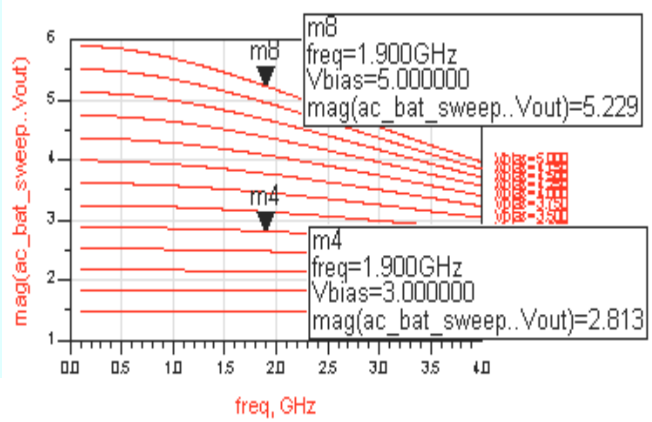
b. Redefine the source: **Vdc = Vbias**.

c. Insert a **Parameter Sweep** from any simulation palette. Then set the **SweepVar** (sweep variable) to be **Vbias**. Be sure the Simulation Instance Name of the AC simulation controller is also set as shown here.

d. Change the **dataset name** = **ac_bat_swp** and **Simulate**. When the simulation is completed and the DDS opens, a dialog will appear asking if you want to change the dataset – answer NO. Then plot the **mag** of **Vout**. A set of curves for each step will appear as shown here.



Trace Options used to Display label of Vbias on right of plot. Trace lines can also be thickened.



e. To display trace labels of Vbias, edit the trace using the **Trace Options** tab and check the **Display Label** box.

f. Insert markers as desired. Remember that you can insert the marker and then, in the marker readout, type in the frequency you want – the marker will then go to that value of freq.

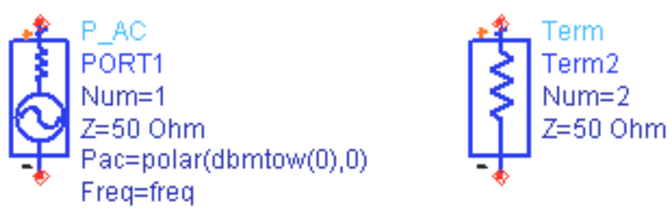
g. **Save all your work**. You can keep the existing schematic window opened – you will use it to start the next lab exercise. But close the data display if it is still opened.

EXTRA EXERCISES:

1. In a new design, simulate with port noise and ports. To do this, use a P_AC source as the input port 1(Num=1) and place a Term on the output as port 2 (Num=2). These two components are shown here with the port numbers.

2. In a new design, insert an I_AC constant current source and simulate.

3. Insert the P_AC source and look at the power gain. Also, sweep another parameter and plot the results.



4. Try using the node settings in the AC simulation palette. You can set initial voltages at nodes using the Node Set or by referring to name nodes using the NodeSetByName component.