ADS Fundamentals – 2009

LAB 9: Final System and Circuit Simulations

Overview – This last lab exercise brings together all the circuits built during the course: the amplifier and filters. They replace the behavioral system models used in the earlier exercise.

OBJECTIVES

- Create a sub-circuit for the 1900 MHz amplifier for use in the system.
- Use the Smart Simulation Wizard.
- Set up and run a CE simulation using a CDMA source.
- Simulate ACPR and power specs using an example data display.
- Program Marker sliders to customize data displays.
- OPTIONAL - Co-simulations with minimal instructions.

Agilent Technologies

TABLE OF CONTENTS

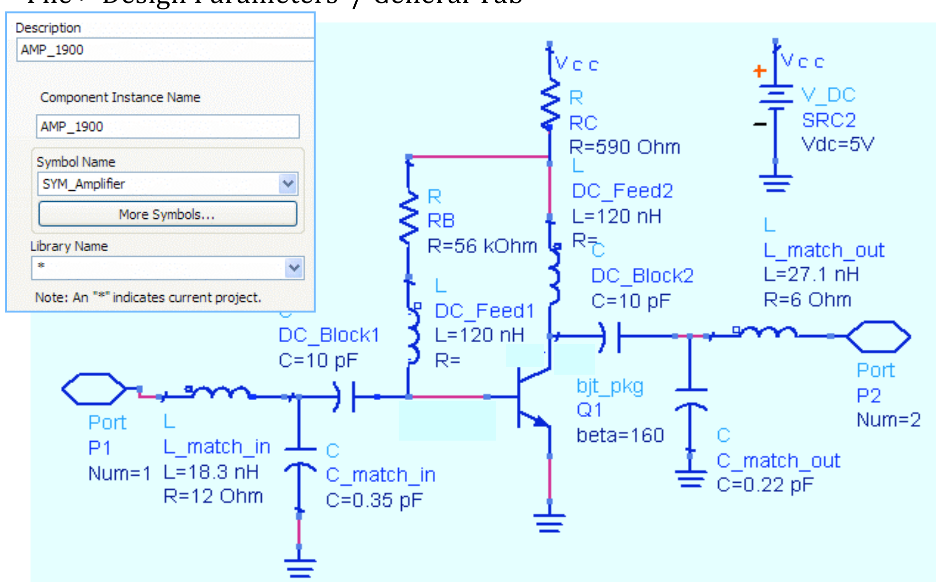
PROCEDURE

1. Create the final AMP_1900 sub-circuit for the library.

a. Save the last amplifier circuit envelope design (ckt_env_gsm) as: **AMP_1900.** This schematic will become the final amplifier design to be used in your system project you created earlier in the course.

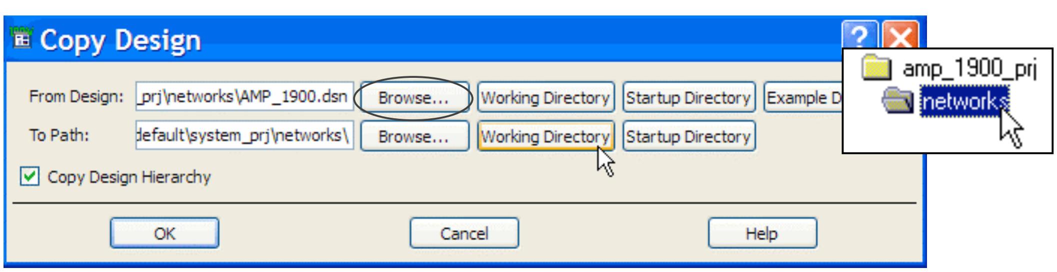
b. As shown here, delete all simulation components, variables, source, etc. Set **Vdc = 5V**. Set **File > Design Parameters** for Component Instance Name**: AMP_1900** and Symbol Name: **SYM_Amplifier**. Also, put **port connectors** 1 and 2 on the input and output. Check the circuit and then **save** and **close** the AMP_1900 design.

File > Design Parameters / General Tab



c. Go to the **ADS Main window** and **open** / **change** to the **system_prj**. This is where you have the rf_sys design and the filters.

d. In system_prj open a new schematic window. Then click **File > Copy Design**. As shown for <u>From Design</u>, click **Browse** and click to the **amp_1900 project / networks** and select **AMP_1900**. In the <u>To Path</u>, click **Working Directory** and click **OK** - the file and its hierarchy (bjt_pkg) will be copied into your system project.



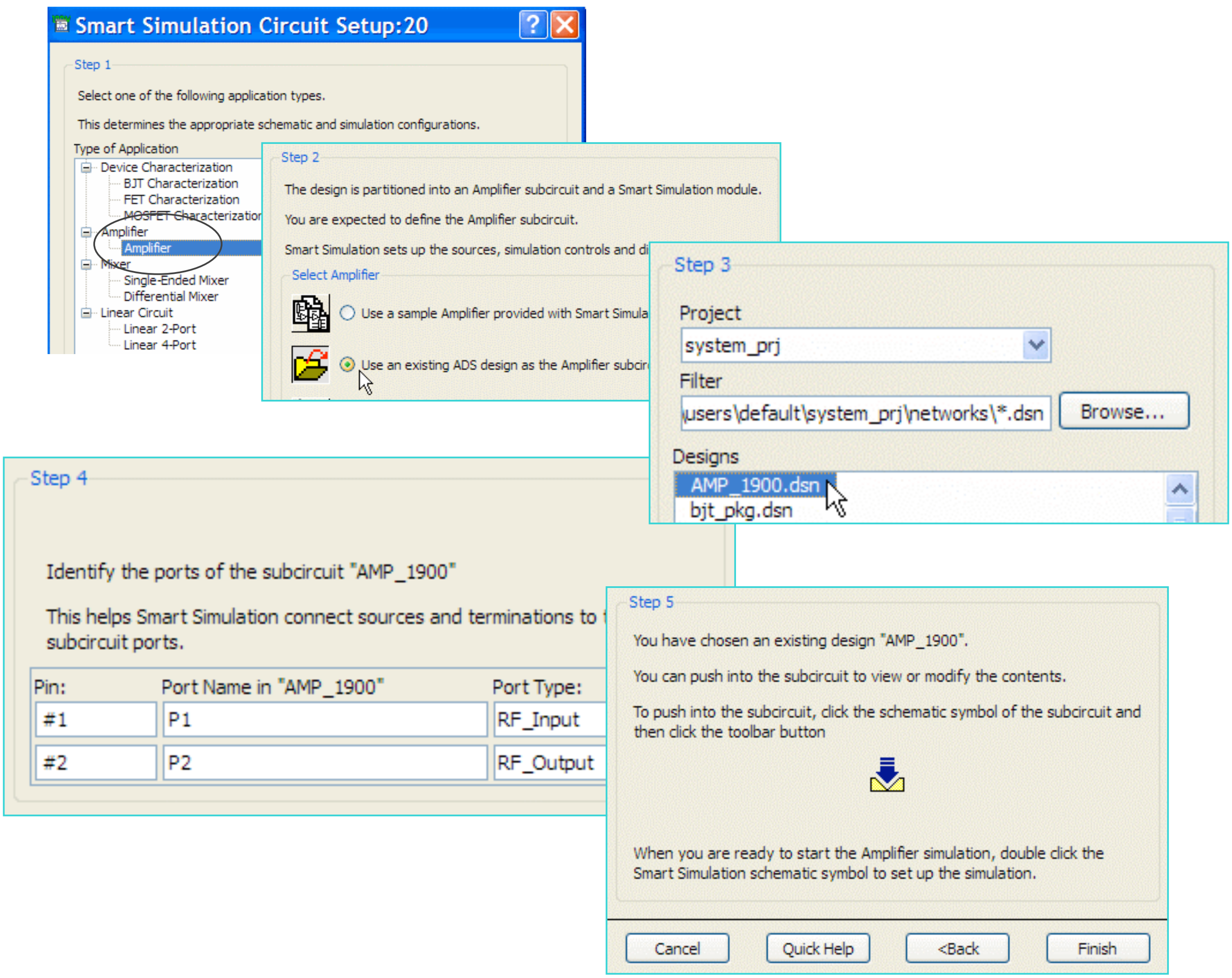
Copying from a different project.

2. Simulate AMP_1900 with the Smart Simulation Wizard.

The wizard has many standard simulation setups. Although this wizard does not take the place of knowing how to use ADS on your own, it is valuable. In these next steps, you will use it to simulate the AMP_1900 where frequency is swept.

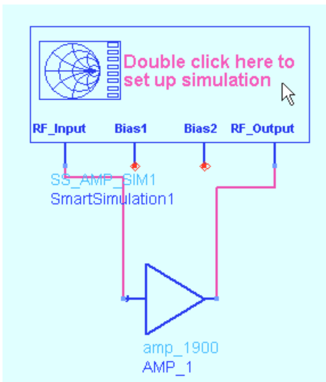
a. In the system _prj, open a new schematic and click **the Smart Simulation Wizard** icon shown here.

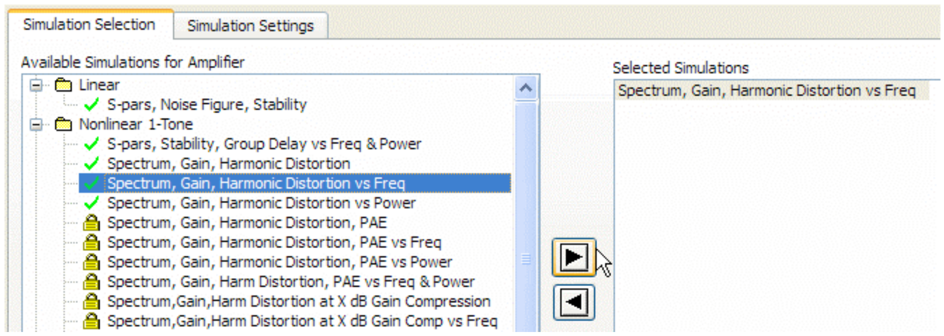
b. Dialogs will appear for the first 5 steps: 1) select **Amplifie**r and **Next**. 2) select **Use an existing ADS design as the Amplifier subcircuit** and **Next**. 3) select **AMP_1900** and **Next**. 4) verify the ports are correct, click **Next** and, 5) click **Finish**.



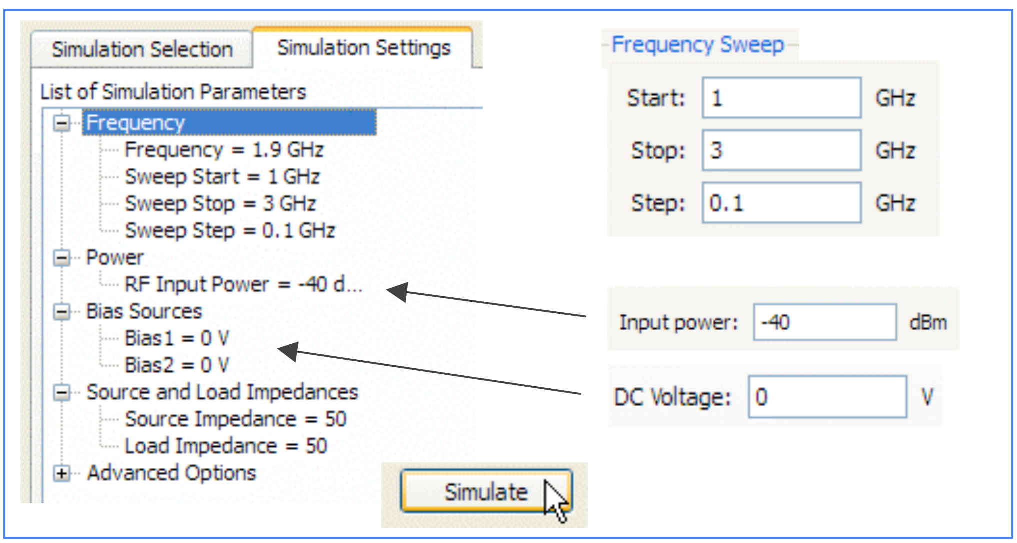
c. In the schematic, double click the simulation setup drawing that looks like a NWA shown here.

d. When the dialog appears, in the Simulation Selection tab, select the Nonlinear 1-Tone **Spectrum, Gain, Harmonic Distortion vs Freq** and click the arrow to add it as shown here.

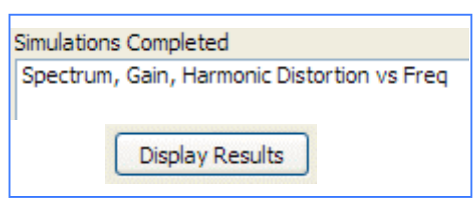




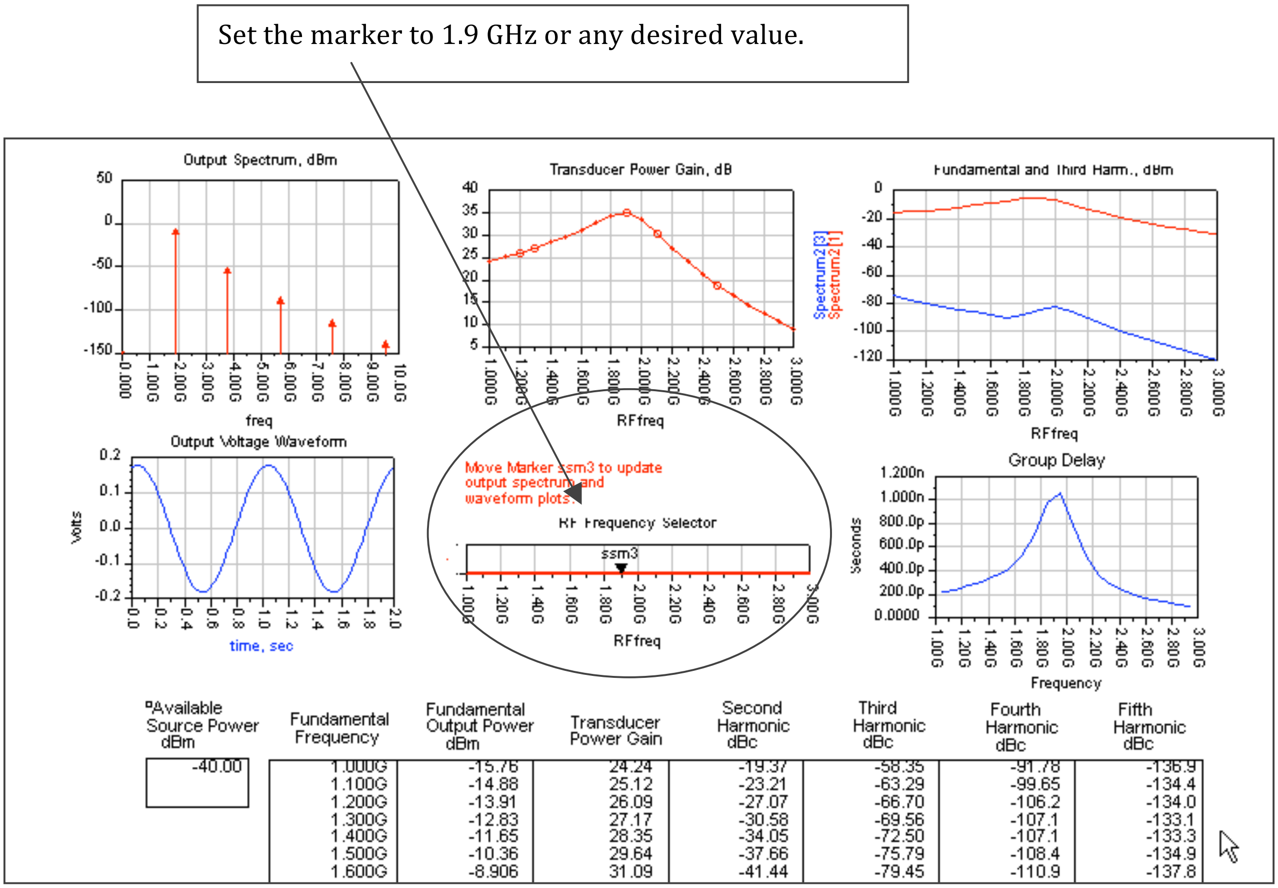
e. In the **Simulation Settings tab**, set the RF frequency to **1.9 GHz** and the sweep from **1 to 3 GHz in 0.1 GHz** steps as shown. Also, click the **RF Input Power** and set it to **–40** dBm - and the bias sources to 0 V.



f. Click the **Simulate** button and when when finished, click **Display Results**. **Results.**



g. The data display will open and the results will be automatically plotted as shown here. Set the marker to the desired frequency.

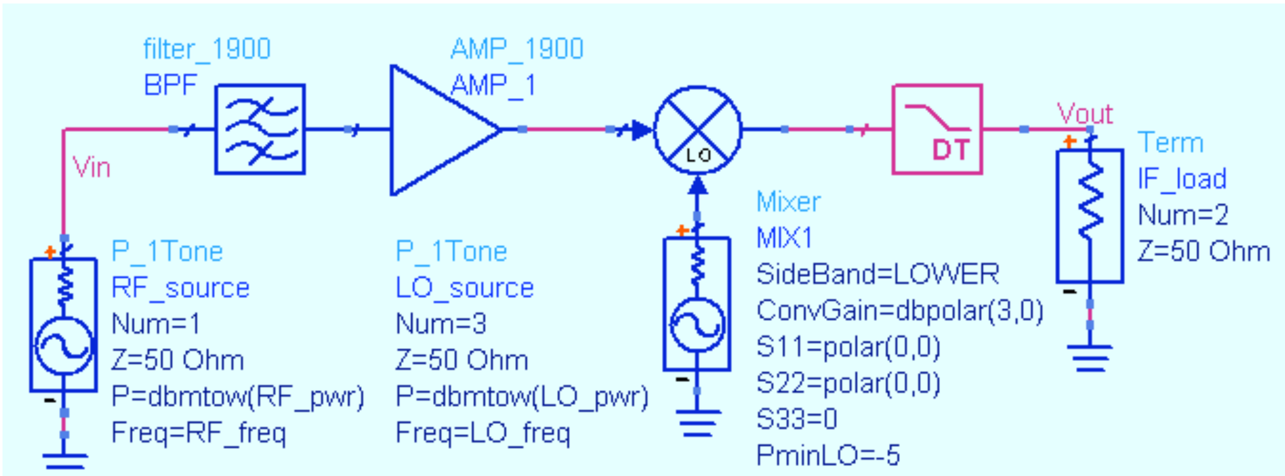


h. As you can see, the wizard can provide quick simulation results for your circuits. Examine the results. You can place markers on other traces and you will see that they match some of the simulations you have already preformed. For now, **close the schematic and data display – no need to save these**. The next steps will be to put AMP_1900 and the filters together in the system design.

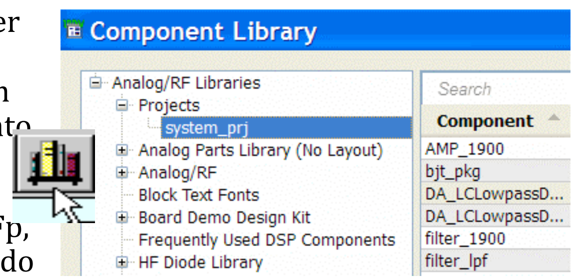
Important Note on the Wizard schematics – Always view the schematics for your design to verify the components used, such as blocking capacitors, variables, etc. The wizard can save you a lot of time and is similar to using templates or design guides. But nothing can replace your knowledge of how ADS operates, especially when results are uncertain or when the wizard setup does not match your topology or configuration.

3. Create the final HB swept LO schematic and equations.

a. Open the existing system_prj design: **rf_sys.dsn**. Now, save it with a new name: **final_hb_lo_swp**. Modify the design by replacing the existing filters and amplifier with your designs as shown here. Step-by-step instructions follow, or try setting it up by referring to this drawing:



b. From the library, replace the behavioral filter behavioral filter with: **filter_1900**. Then Then replace the LPF_Bessel with the Design Design Guide filter: **DA_LCLowpass**. Push into Push into them to verify the circuits.

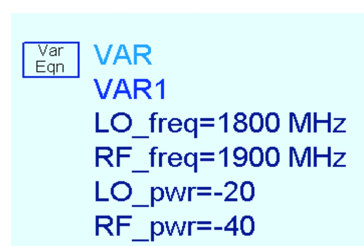


NOTE on DT component text – The parameters (Fp, Fs, etc.) may appear as default values. This is OK – do not change them.

c. Also from the library, replace the system amp with your **AMP_1900** and push into it to check it also.

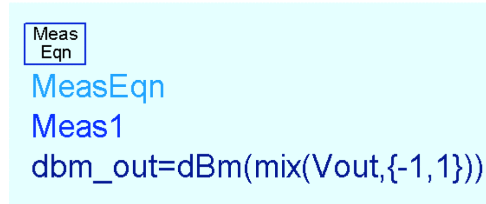
d. Set up two **P_1Tone** sources for the **RF** and **LO** as shown here. Be sure **Num =**, **P=**, and **Freq** = are set with variables as shown.

e. On the Mixer, set the Pmin spec: **PminLO = -5** for a *starvation* effect (mixer diodes not responding). Conversion gain is 3 dB and S 11,22,33 are all set to zeros as shown. No other parameters are necessary.



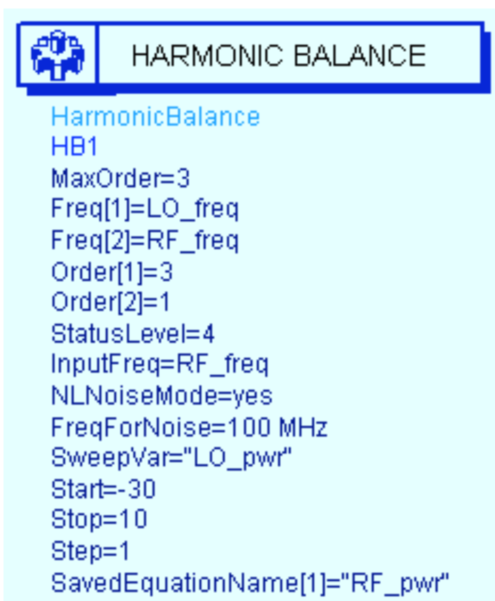
f. Set up **VAR**s for <u>RF and LO Freq and Pwr</u> as shown here. Also, set **Vin** and **Vout** node labels.

g. Write the measurement equation for output power of the IF. Because there is mixing, use the mix function to identify the tone: **dbm_out = dBm (mix (Vout, {-1, 1}))**. Inside curly braces, the index values are: - 1 for the LO and 1 for the RF. The result, dbm_out, is the power of the IF signal at Vout.



4. Final HB simulation: 2 tones with swept LO power and noise.

a. With all other controllers deleted from the schematic, insert and set up a **Harmonic Balance** controller as shown here. You can do this by turning on the display settings first and then typing in the values on screen. Or, you can use each tab to set the values. Either way, go to the **Display tab** first and **turn on the display settings shown here**. The following bullet steps show how to set up the controller using the tabs.

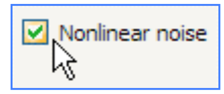


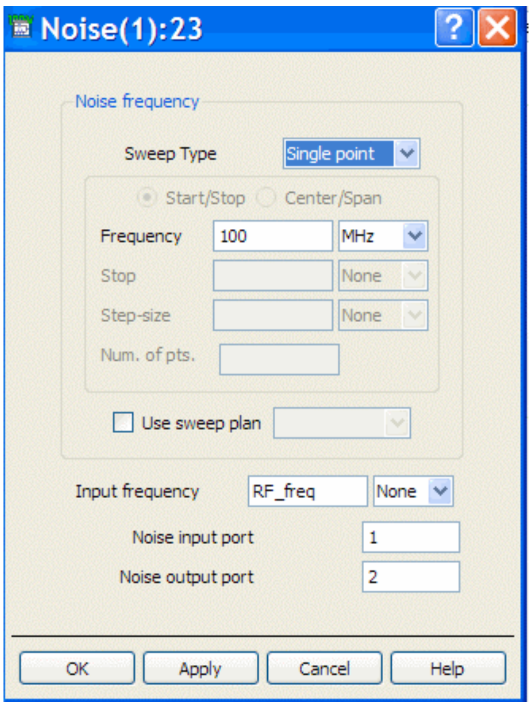
Display tab settings:

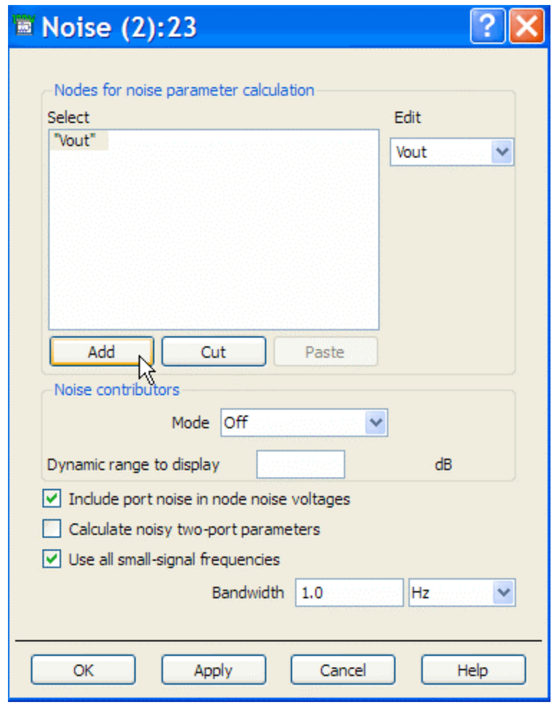
- **Freq tab** – Set MaxOrder (mixing products) = **3**. Set **Freq[1]= LO_freq** with **Order[1]= 3** harmonics. Set **Freq[2]=RF_freq** with **Order[2] = 1** harmonics because its power is low compared to the LO. Also, set the **Status Level = 4** to display more information (status window), including NF and conversion gain.

- **Sweep tab** – Set **LO_pwr** as a linear sweep: **Start = –30** and **Stop = 10**, with **Step = 1**, as shown.

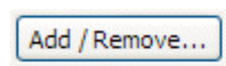
- **Noise Tab** and **Noise [1] and [2]**: Turn on **Nonlinear noise** (bottom of Tab). In Noise[1], set a Single point <u>Frequency</u> to **100 MHz** and <u>Input frequency</u> to **RF_freq**. Also set Noise ports 1 input and 2 output as shown. In Noise[2], use the Edit list box to add **Vout** as the noise node. Leave all other settings in their default as shown.



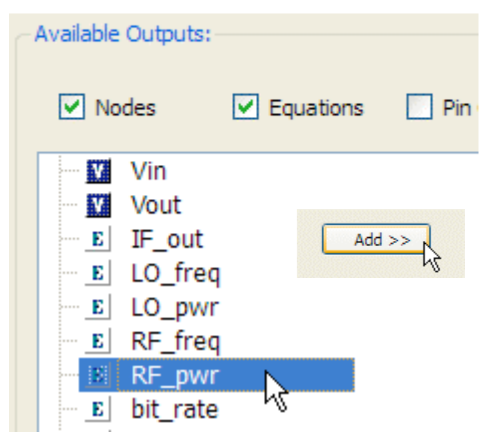




- **Output tab** – Click the Add/Remove button. Then select the **RF_pwr** variable and click the Add button.



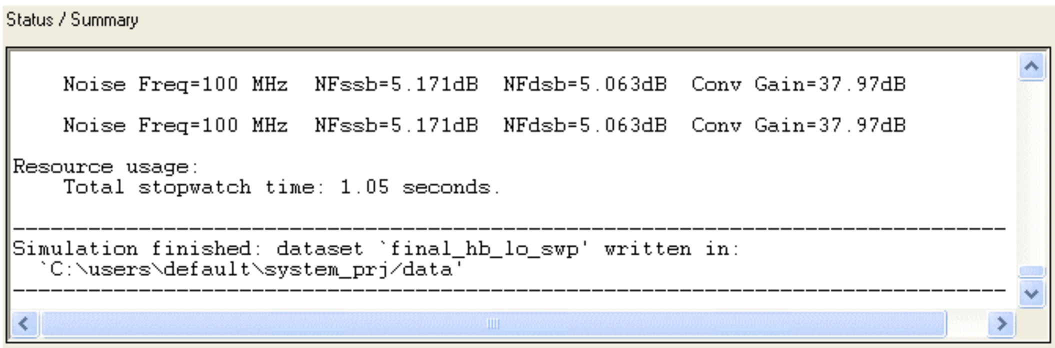
You will use this in the data display to write an equation. You select VarEqns this way because they are not sent to the dataset by default. Only named nodes (pin and wire labels) and measurement equations are output to the dataset by default. Node voltages and MeasEqns set in the Outputs tab will appear on-screen for the SavedEquationName parameter.



b. Check the circuit and HB controller setup a final setup a final time to be sure they are correct (as shown) and then **Simulate**, watching the status window as the power is swept. The simulator information (status level 4) is written into the window – this does take longer than lower status settings. But in this case, you want the mixer conversion gain and noise figure.

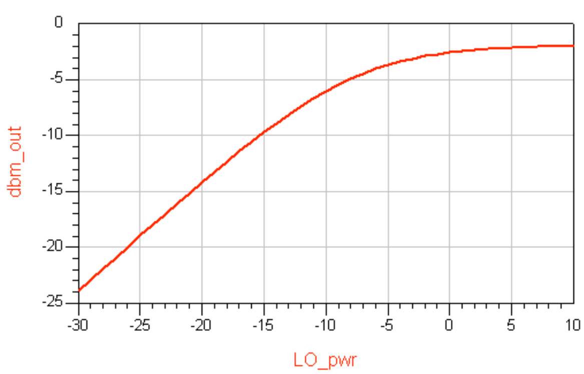
5. Plot the data: NF, Conv Gain, dbm_out, and IF_gain.

a. When the simulation completes, scroll in the status window to see the calculated conversion gain and the noise figures NF as shown here. NOTE on warnings: metal loss message – you can ignore this message.



b. Plot the **dbm_out** equation and you will see the effects of the swept LO power. Notice that near -10 dBm the mixer goes into starvation.

c. Write an equation for **IF_gain** as shown here. By subtracting RF input power from the output power, the result is the gain at all values of the swept LO. Put the equation in a **list** and scroll to see the results.



Eqn IF_gain = dbm_out - RF_pwr[0]

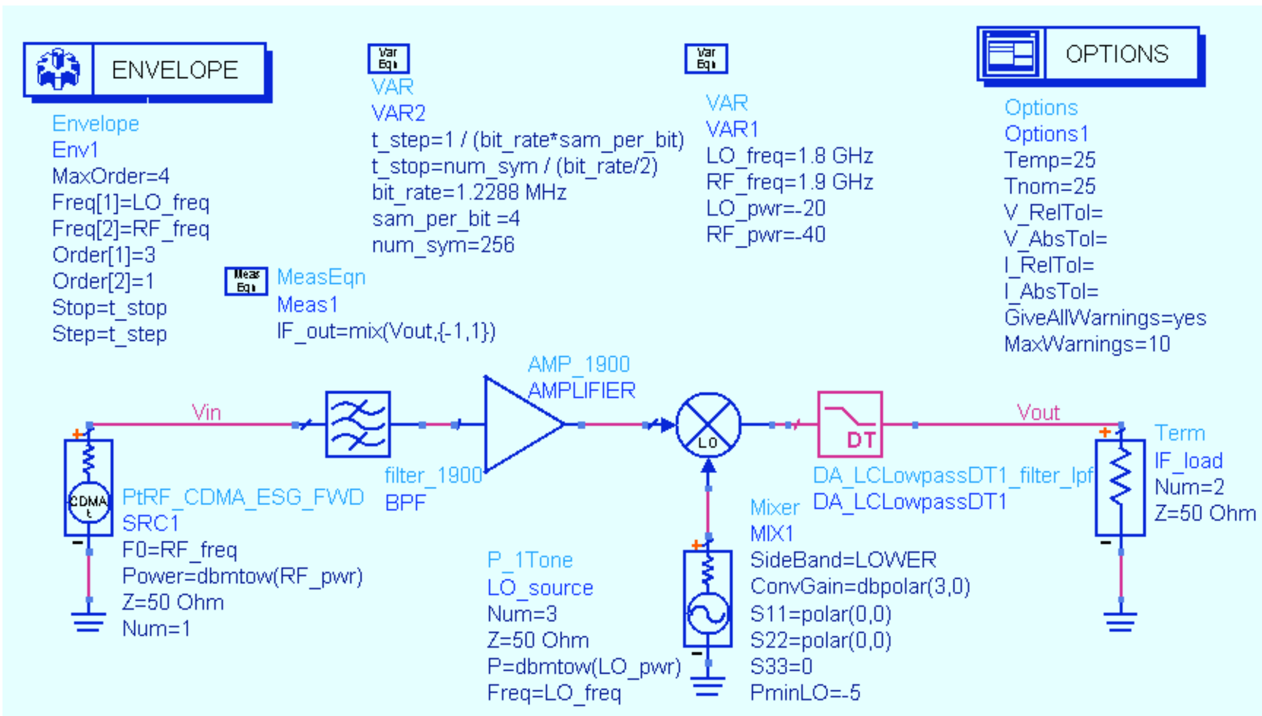
LO_pwr	IF_gain
-6.000	35.954
-5.000	36.313
-4.000	36.622
-3.000	36.883
-2.000	37.103
-1.000	37.286
0.000	37.437

d. **Save** the schematic and data displays.

6. **Final Envelope simulation: CDMA source.**

   a. Save the last design as: **final_env_cdma**.

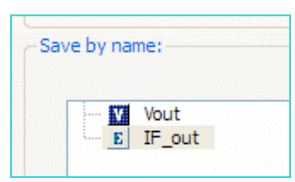
   b. Replace the source with a **PtRF_CDMA_ESG_FWD** source from the Sources-modulated palette – be sure to insert the same CDMA source shown here which is based on a real signal generator. **Set FO = RF_freq** and **Power = dbmtow (RF_pwr)** as shown.



   c. Add a new **VAR** block for t_step, t_stop, bit_rate, sam_per_bit, and num_sym as shown here. Also, add an **Options** block which, by default, forces S-parameters to be used for the linear elements (BPF). If you edit the options block you will see the setting to use S-parameters when possible.

   d. Change the measurement equation to read like the one shown here: **IF_out =mix (Vout, {-1,1})**.

   e. Replace HB with an **Envelope** controller and set it up using using the variables as shown. Also, in the **Output** tab, leave leave both boxes unchecked and select **IF_out** and **Vout** using **Vout** using the Add / Remove buttons so that this is the only the only data in the dataset.

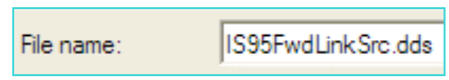


   f. **Simulate**.

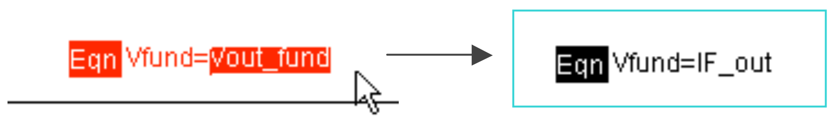
7. Use an Example DDS to plot ACPR and Power for your circuit.

a. When the data display window opens, click: **File > Open** and use the dialog to go to the ADS installation directory to find: **examples / Tutorial / ModSources_prj / IS95_FwdLinkSrc.dds**

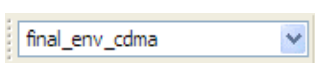
b. **Open** this DDS and then click the DDS command **File > File > Save As** and save it in your system_prj (scroll (scroll using the arrow buttons). Save it with the same same example DDS name.



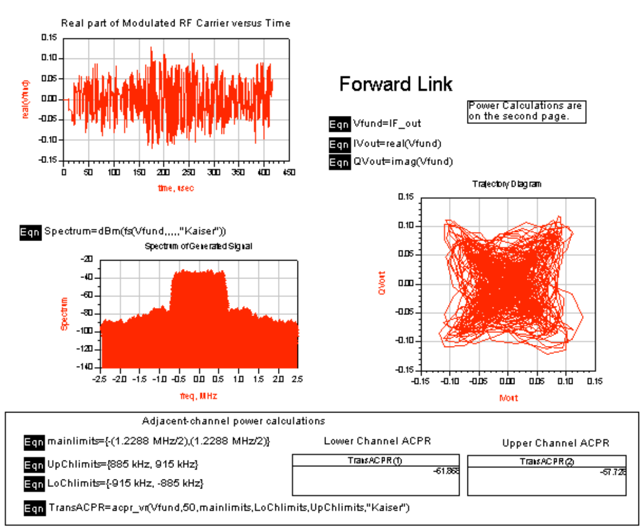
c. Notice the values will be red or invalid without the example data. Change the default dataset to your **final_env_cdma** dataset. Then change the Eqn Vfund to your IF output equation: **Vfund = IF_out**.

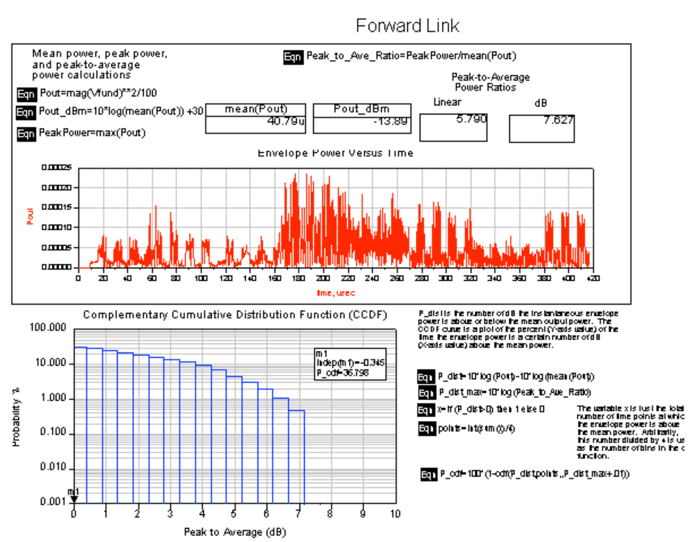


d. The result is the example calculations are now used for your dataset. Examine both pages in the data display (ACPR and Pwr). You can use any example data display for you data in this way like a template.



DDS pages: 1) ACPR and Trajectory and 2) Power Calcs

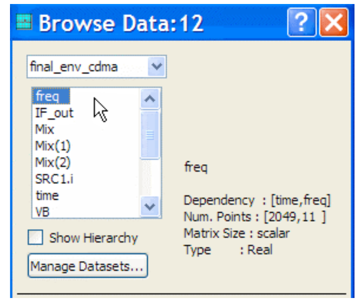




e. Examine the data and then **Save** and **close** the data display window.

8. Plot the spectrum using a programmed marker slider.

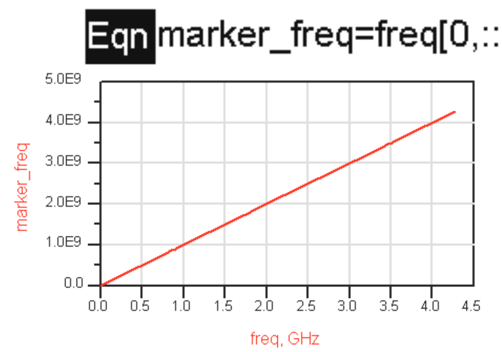
The next several steps will show you how to use powerful expressions to pass a marker value to a function and plot the data whenever the marker is moved.



a. Open a new data display and name it: **Marker_Slider**. **Marker_Slider**.

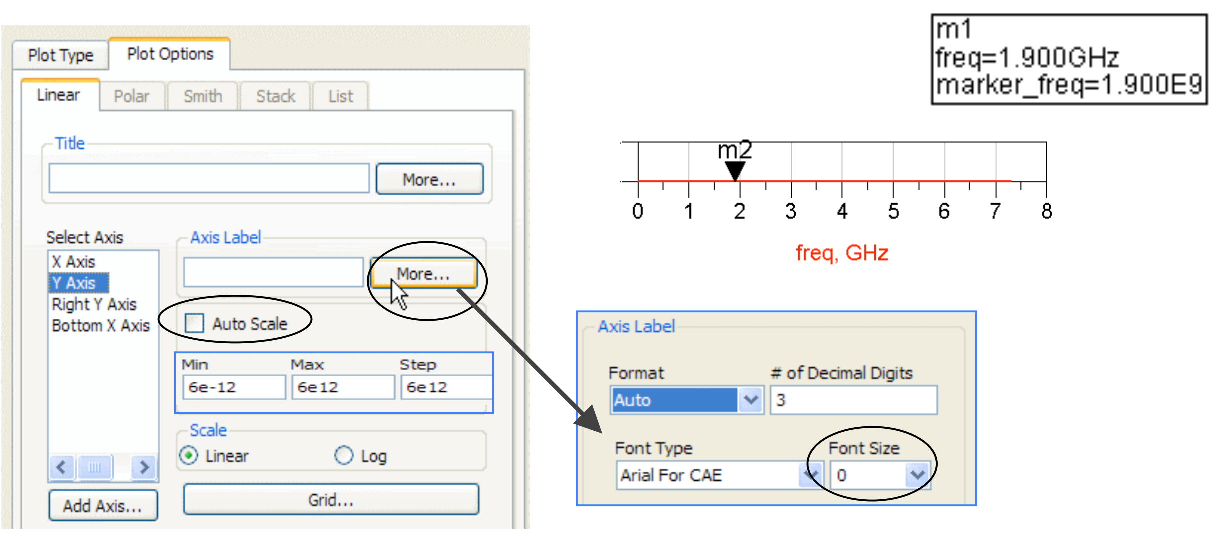
b. **Insert an equation**. Click the Variable Information button. You will see that **freq** is dependent upon time. time. **Close** the dialog.

c. Write the equation **marker_freq** shown here. This will access all frequencies at one point in time. You can use any time point because the number of calculated frequencies are the same at any time point, according to the order and max order you set in the envelope simulation controller. Use zero as shown.



d. <u>Insert a plot</u> of the **marker_freq** equation which is plotted against the independent variable: freq. Next, you will make it look like a slider.

e. <u>Edit the plot</u>. Remove the Auto Scale for the Y-axis. Set the Y-axis Min, Max and Step to: **6e-12**, **6e12**, and **6e12** as shown. Then click the **More** button and set the Y axis **font size** = 0 (type it in). Thicken the trace if you want. Click **OK** and then put a marker on 1900 MHz. Be sure to size it so that it looks like the slider shown here.



f. Write the equation **freq_index** using the ***find_index*** function. The marker value and marker_freq are passed into the argument to return the index value of the marker position. This equation will be the look-up value for the Vout data you want to plot.

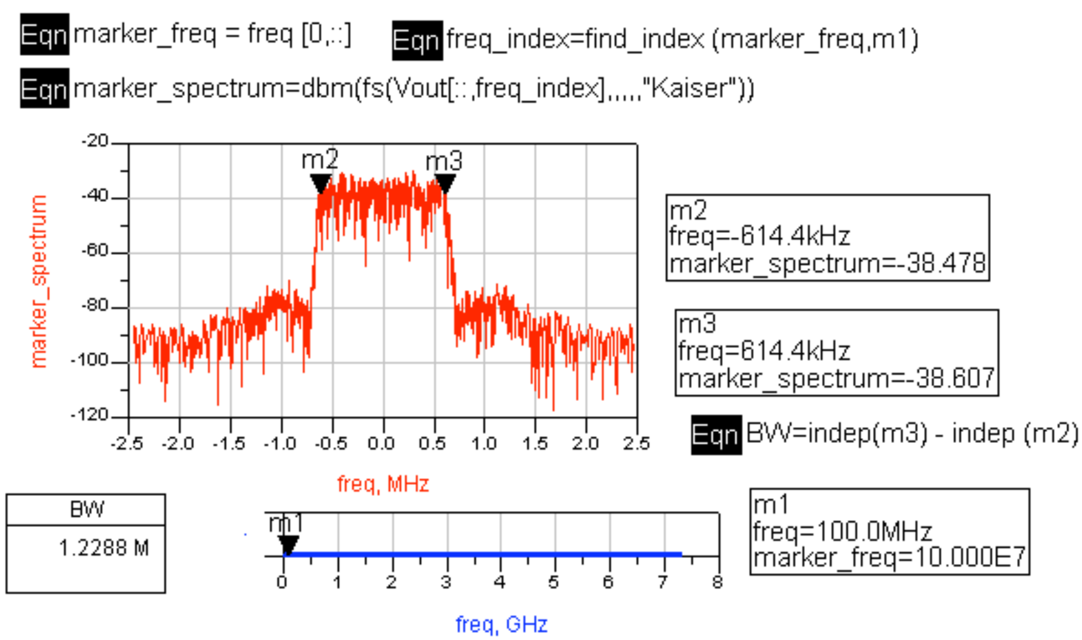
Eqn freq_index=find_index (marker_freq,m1)

g. Write the equation **marker_spectrum** to plot the spectrum around the marker frequency value. The ***fs*** function transforms the envelope time data into frequency - the two colons (**::**) represent all time points and *freq_index* is the index value of the marker frequency. Use 5 commas after the bracket and type in the **"Kaiser"** window function. In all ADS functions, you can disregard any argument by using commas.

Eqn marker_spectrum=dbm(fs(Vout[::,freq_index],,,,,"Kaiser"))

h. Plot the **marker_spectrum** equation and change the **Trace Type** (Trace Options) to **linear**. Then move the slider to **100 MHz**. Put two markers on the spectrum as shown and write an equation, **BW**, using ***indep*** to get the independent variable of the markers (x-axis). Insert a list of **BW** as shown, changing to Engineering format with 4 significant digits and removing the independent data (Plot Options).

i. Move the marker- BW remains the same. Examine your work, **save** the data display and the schematic. **Your circuits have now been simulated in the system and you have completed the course!**

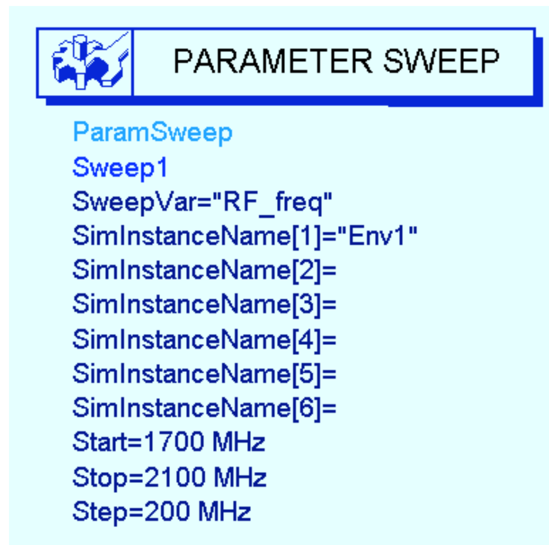


9. CDMA Envelope Simulation with Frequency Sweep

Do this only if you have time…

This simulation shows how to sweep the RF frequency and adjust the LO variable for frequency so that is always producing the same IF output frequency. Afterward, the IF output spectrum can be plotted to show the results for a varying RF input frequency sweep. To do this, you need to set up a parameter sweep of the RF, redefine the LO variable, and then plot the IF output for each of the RF frequencies.

a. Save the current design (final_env_cdma) with a new name: **final_env_cdma_ swp**.



b. Insert a parameter sweep from from any simulation palette and palette and set it up as shown shown here. The quotes will appear if you appear if you edit the controller and enter and enter the values there – if not, be sure be sure to type them onto the schematic. schematic.

SweepVar = RF_freq

SimInstanceName = Env1

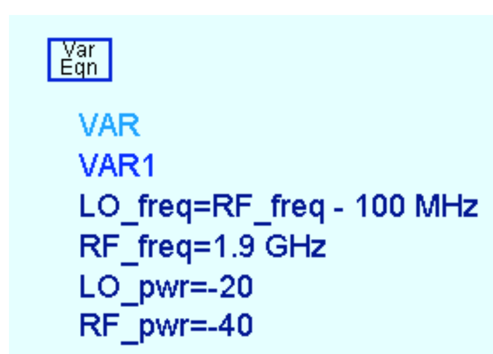
Start = 1700 MHz

Stop = 2100 MHz

Step = 200 MHz

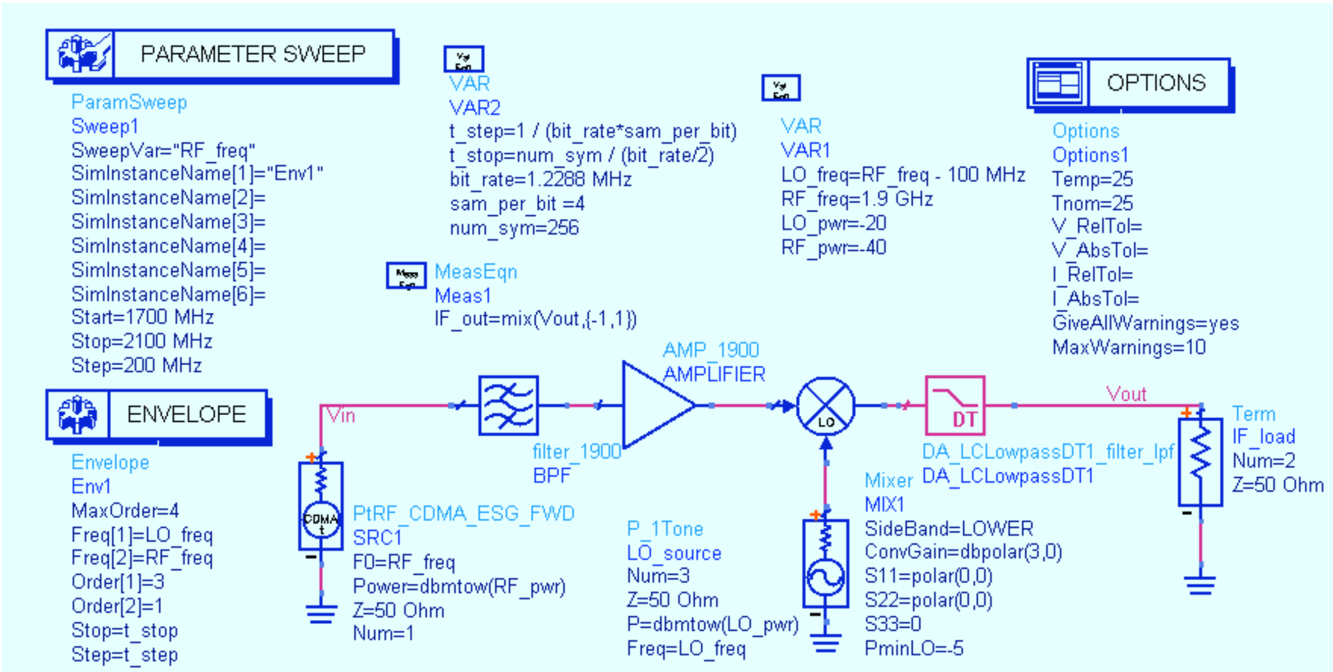
This will sweep the RF signal with 3 frequencies: 1700 MHz will cover the low end and 2100 MHz the high end which are both just outside of the BPF response. Of course, you could sweep it more finely but that would take more time.

c. Change the LO_freq variable to track the track the RF signal sweep by setting it as setting it as shown: LO_freq = RF_freq – RF_freq – 100 MHz. This will make the LO the LO always be 100 MHz less than the RF the RF for any number of steps in the the parameter sweep.

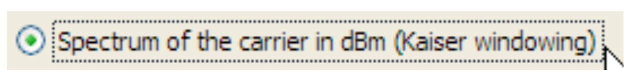


d. Save the design - no other schematic schematic changes are required.

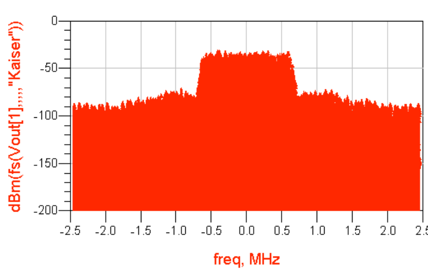
e. Your design should look like the one shown here. If it does, then **Simulate**.

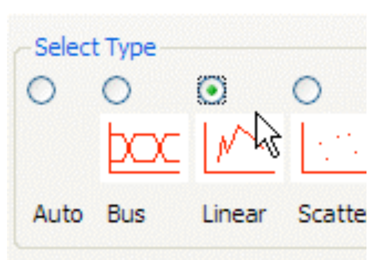


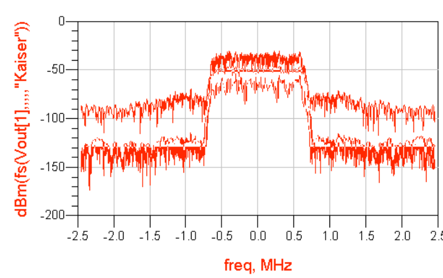
f. After the simulation is finished, insert a rectangular plot and add Vout as the Spectrum of the carrier in dBm dBm with windowing as shown here. here. Click OK and the plot will appear.



g. Edit the trace (double click) and use Trace Options to change the trace to **Linear** to better see the output traces.

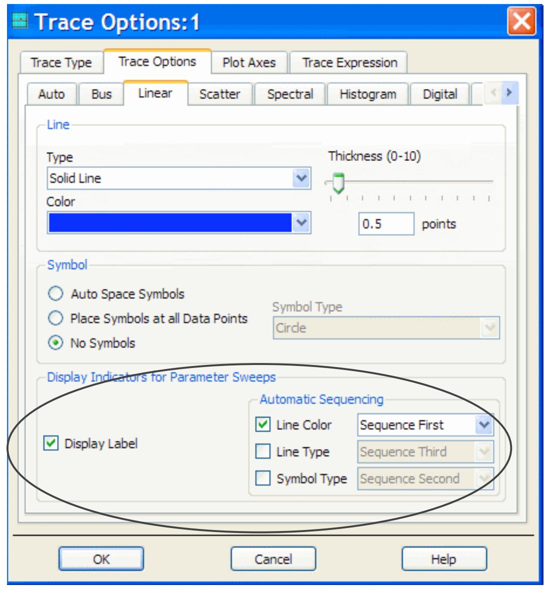




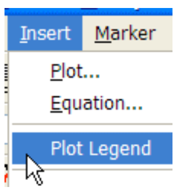


Notice that the spectrum is for Vout[1] which 100 MHz IF tone. However there are three resulting IF tones from the sweep. Now, you need be able to see which trace results from which RF – you will do this in the next step.

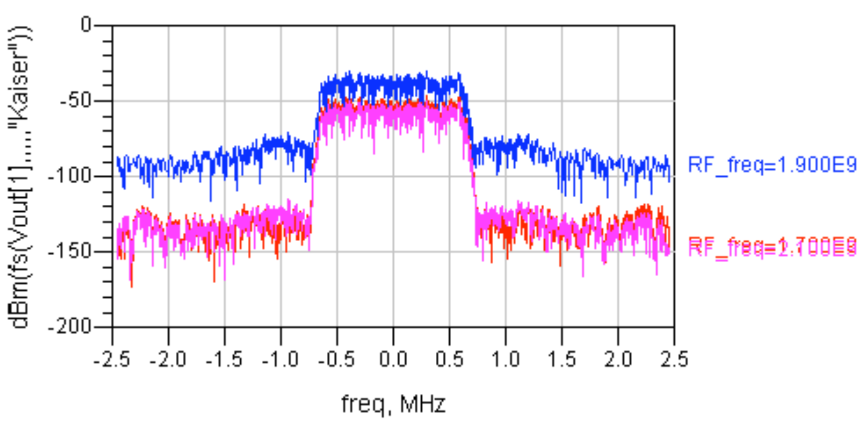
h. Edit the plot one more time and use and use Trace Option and the Trace Trace Expression. Go to the Linear Linear tab and turn on Line Color and Color and Display Label as shown shown here and click OK.

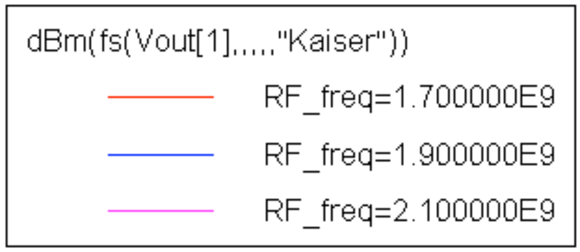


i. Your plot should look like the one one shown here. But to identify the identify the traces even better, use use the command: Insert > Plot Plot Legend.



Finally, you can see the three IF spectral traces that result from each on of the RF tones. This shows how to sweep frequency for a circuit envelope simulation and how the response of the circuit can be analyzed.



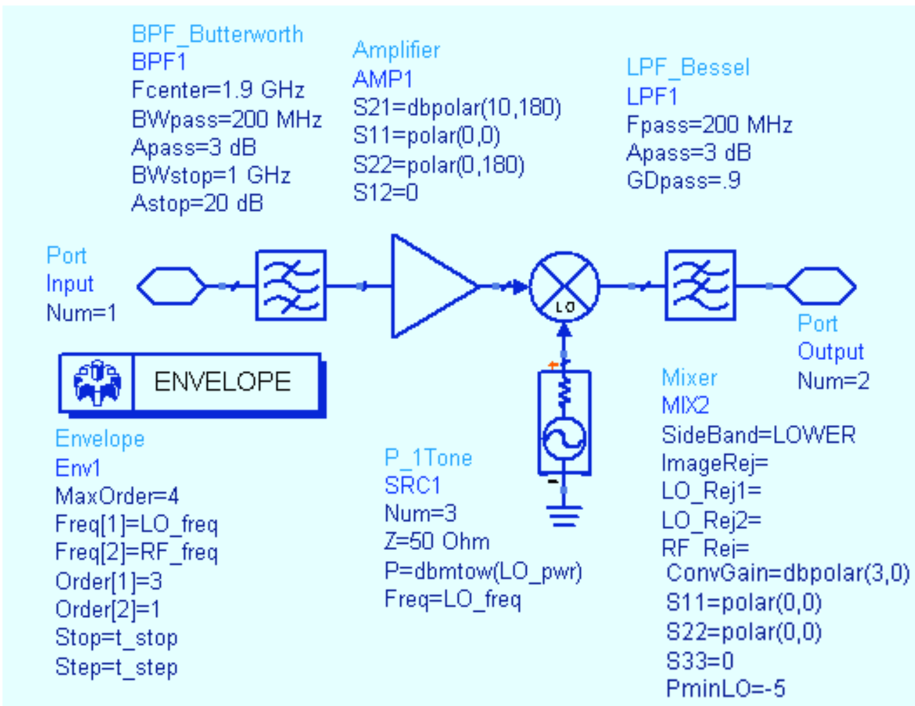


This completes the lab exercise - the remaining steps are optional. Do them only if you have time and have access to the Ptolemy simulator.

10. OPTIONAL - Co-simulation of the behavioral RF system.

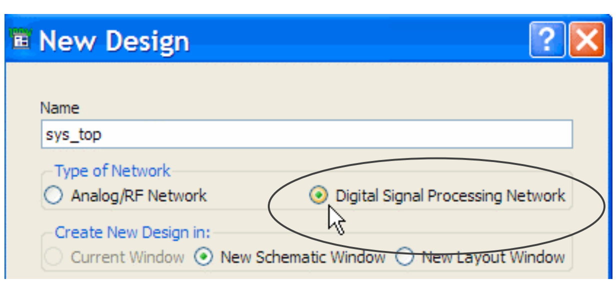
Create two levels of hierarchy: 1) bottom level simulation of the behavioral system with a circuit envelope simulation setup and 2) a top level data flow simulation using the DSP palettes. The steps follow…

a. Open the original rf_sys.dsn you created in lab 2 and save it as **sys_bottom**. Modify the design as follows: Set the Mixer **PminLO= – 5** as shown here. Also, put port connectors on the circuit. Insert an **Envelope** controller and set it as shown. Note that you do not need to insert or declare a VAR here – it will be in the top level.



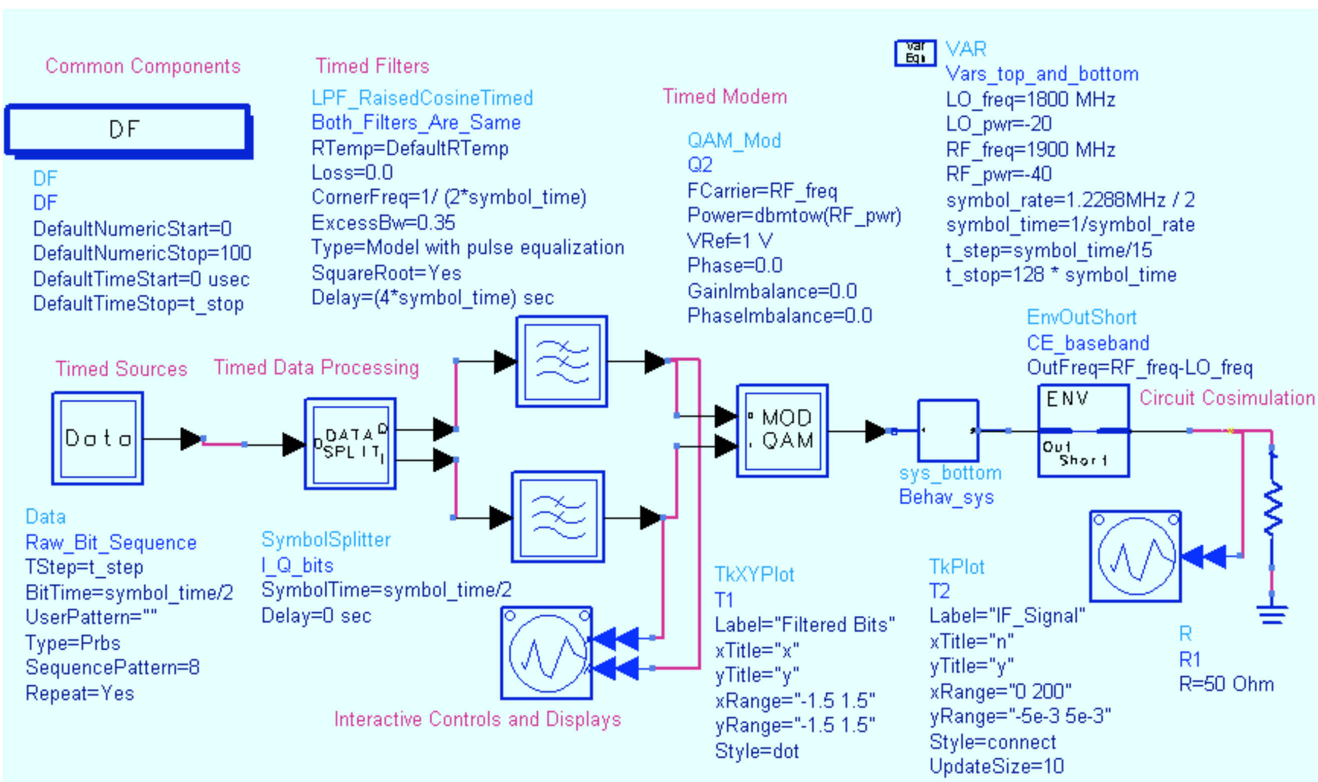
b. Save the design and close it using: **File > Close Design.**

c. Open a new blank schematic window: **File > New Design**. When the dialog appears, type in the name **sys_top**, and **select Digital Signal Processing Network** as the type and then **save** the design.



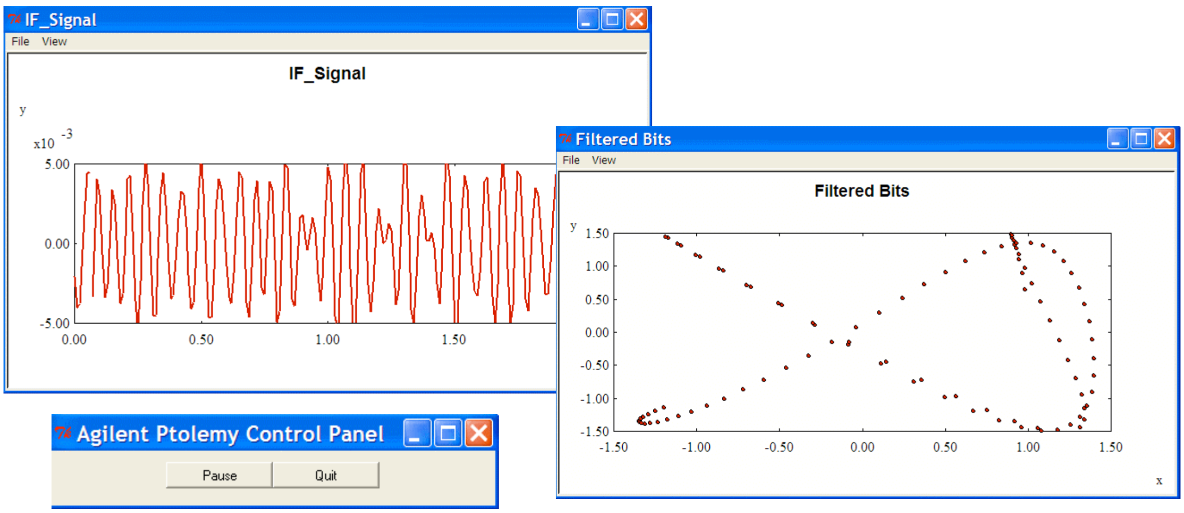
d. In the new design, sys_top, build the design shown here:

- DF (data flow) - is in the Common Components palette. Set only the DefaultTimeStop = t_stop.
- Data - is in the Timed Sources palette. Set only the first two parameters Tstep=t_step and BitTime=symbol_time/2.
- SymbolSplitter – is the Data Splitter, in the Timed Data Processing palette. Set only the two parameters shown.
- LPF_RaisedCosineTimed – These filters are in the Timed Filters palette. Insert one filter, make the settings, and then copy it.
- QAM_Mod – the modulator is in the Timed Modem palette. Set as shown.
- Insert the sys_bottom design from the regular library.
- EnvOutShort – from the Circuit Cosimulation palette, this component captures the IF signal from the sys_bottom design.
- Insert the TK plot and TK-XY plots from Interactive Controls and Displays.
- Insert the resistor and ground and the VAR block.

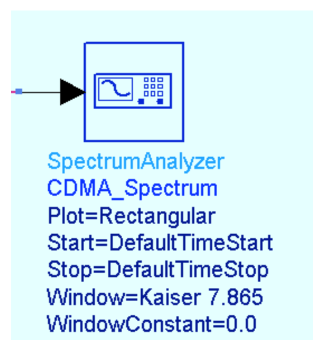


e.

Check all the values and variables to be sure they are correct and **Simulate**. When you do, you will set the two plots in action. Use View> View All to rescale the plots if needed.



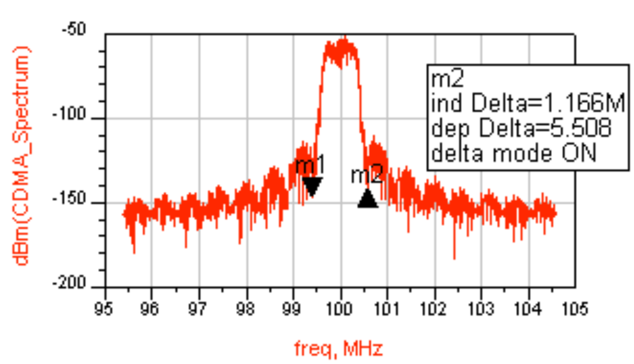
f. Quit the simulation using the control and insert a **Spectrum Analyzer** component from the Sink palette. Connect it to the output and edit it to select the window type as Kaiser shown here. Be sure to set the other values as shown.



g. When the status window shows the data collection is complete for the sink (SpectrumAnalyzer), Quit.

h. Open the data display and add dBm(CDMA_spectrum) to a rectangular plot.

SUMMARY - This data is the result of a co-simulation between Ptolemy and the Envelope simulator. It marks the end of the ADS Fundamentals lab exercises.



END of LAB EXERCISE.