# *LAB 2: System Design Fundamentals*

Overview - This chapter introduces the use of behavioral models to create a system such as a receiver. This lab will be the first step in the design process where the system level behavioral models are simulated to approximate the desired performance. By setting the desired specifications in the system components, you can later replace them with individual circuits and compare the results to the behavioral models.

## OBJECTIVES

- Use the skills developed in the first lab exercise.
- Create a system project for an RF receiver using behavioral models (filter, amplifier, mixer) where: RF = 1900 MHz and IF= 100 MHz.
- Use an RF source, LO with phase noise, and a Noise Controller.
- Test the system: S-parameters , Spectrum, Noise, etc.

Agilent Technologies

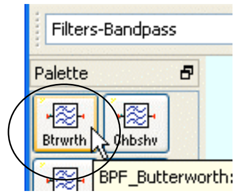
# ***Table of Contents***

## PROCEDURE

### 1. Create a New Project (system) and schematic.

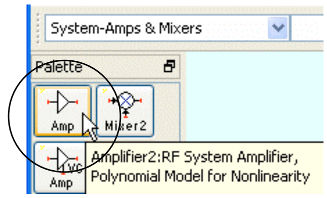
**a.** Use the File > New Project command and name the new project: **system**.

**b.** Open and save a new schematic with the name: **rf_sys**. **rf_sys**.

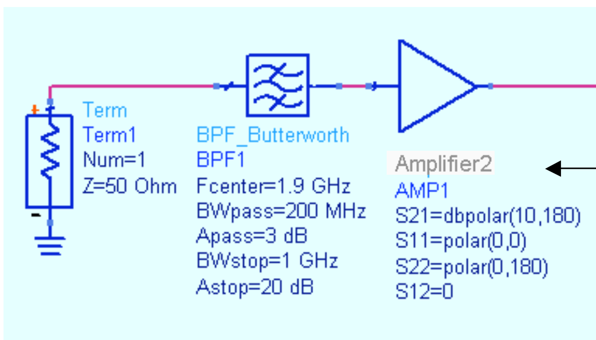



### 2. Build a behavioral RF receiver system.

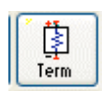
a. **Butterworth filter**: Go to the Component Palette list Palette list and scroll down to **Filters-Bandpass**. **Bandpass**. Insert a Butterworth filter. Set it as shown: Fcenter = 1.9 GHz to represent the carrier carrier frequency. Set BWpass = 200 MHz and and BWstop = 1 GHz.




b. Amplifier: Go to the System-Amps & Mixers palette palette and insert the **Amp**. Set S21 = dbpolar dbpolar (10,180).




For on-screen editing, use the Enter key to step to the next parameter.



c. **<u>Term</u>**: Insert a termination at the input for port 1. Terms are in the **Simulation-S_Param** palette or type in the name Term in the Component History and press Enter.

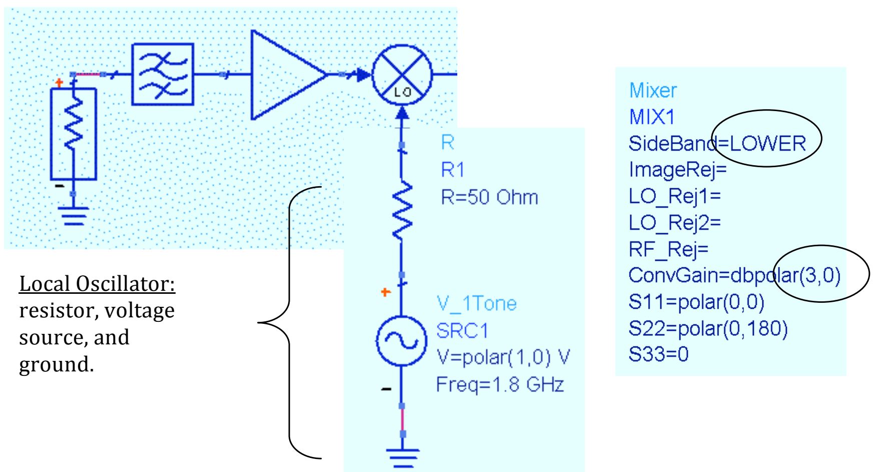
**NOTE on Butterworth filter -** The behavioral Butterworth response is ideal; therefore there is no ripple in the pass band. Later on, when the filter and amplifier are replaced with circuit models, there will be ripple. For system filter modeling with ripple, use the behavioral Elliptical filter.

The next steps will add a behavioral mixer and LO to the RF system.

d. From the System-Amps & Mixers palette, insert a behavioral **Mixer** at the at the amp output - be careful to insert the **Mixer** and not Mixer2. Mixer2 Mixer2 is similar and also for nonlinear analysis but does not work with with the small-signal frequency conversion feature of S-parameter analysis that you will use in this exercise.


e. Set the Mixer **ConvGain = dbpolar (3,0)**. Also, set the Mixer **SideBand = LOWER** by inserting the cursor in front of the default (BOTH) and using the keyboard UP and DOWN arrow keys to toggle the setting to LOWER. Leave all other settings in the default condition.

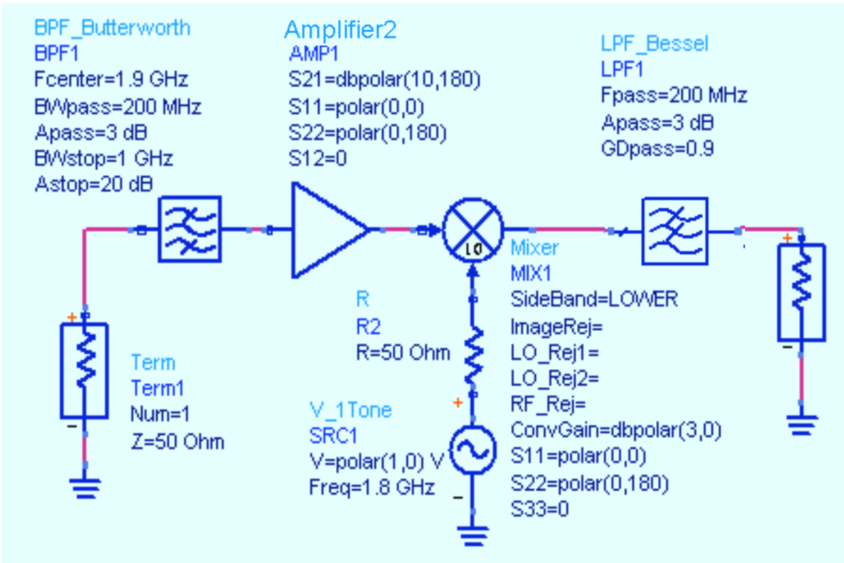
f. **Move component text -** click the **F5** keyboard key and then click on a component to move its text. Do this so that you can clearly see the components.




g. Add the LO by inserting **a 50 ohm resistor** in series with a **V_1Tone** source from the **Sources-Freq Domain** palette. Set the Freq to **1.8 GHz**. This will provide an IF of 100 MHz at the output. Don't forget the ground.

h. Add a low pass Bessel filter at the mixer output as shown here. The filter is in the **Filters-Lowpass** palette. Set **Fpass = 200 MHz**.

i. Insert a **Term** for port 2. The final system circuit should look like the one shown here:




NOTE: You can set the N parameter (order) on the filters but it is not required. By default, ADS will calculate the order (N) based on the specifications. If N is specified, ADS will overwrite the filter specifications.

**Deleted:**

**Formatted:** Bullets and Numbering

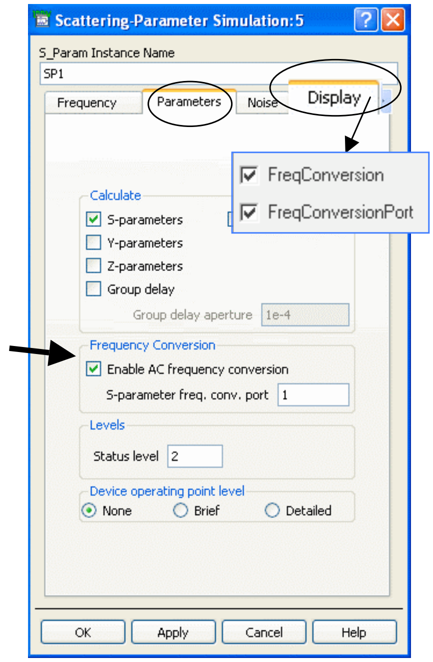
### 3. Set up an S-parameter simulation with frequency conversion.

a. Insert the controller and setup the simulation: 1 GHz to 3 GHz in 100 MHz steps as shown here.

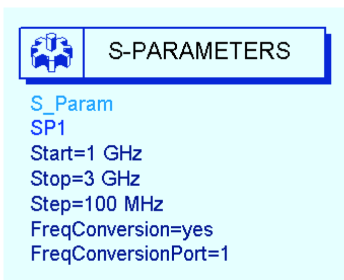
 

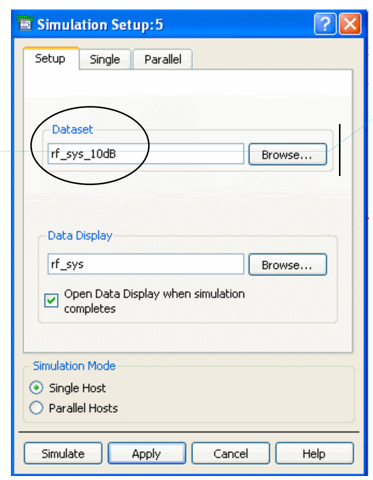
b. Edit the Simulation controller and, in the the Parameters Tab, **Enable AC frequency frequency conversion** by checking the box the box as shown here.

c. Go to the **Display** tab and check the two two boxes to display the settings shown shown here: **FreqConversion** and **FreqConversionPort**. The default (port 1) (port 1) is used because it is the port where where frequencies will be converted using using the mixer settings also. NOTE: this NOTE: this conversion only works with with this ADS mixer.




The S-parameter simulation controller should now look like the one shown here:




**Deleted:**

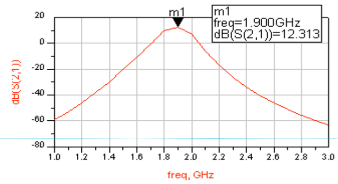
d. Click: **Simulate > Simulation Setup**. When the When the dialog appears, change the default default dataset name to rf_sys_10dB to indicate indicate that this simulation data represents the represents the system with 10dB of amplifier amplifier gain.

e. Click **Apply** and **Simulate**.

**4. Plot the S21 data.**

a. In the Data Display window, insert a rectangular plot of S(2,1).

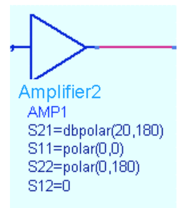
b. Put a marker on the trace near 1.9 GHz with the mouse. Then insert your cursor and type in the value: 1.9 in the readout box. The gain includes mixer conversion gain minus some loss to due mismatches.




**Deleted:** , t

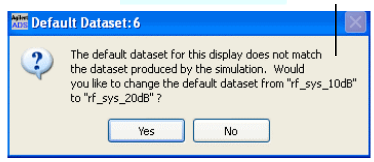
**5. Increase gain, simulate, and add a second trace.**

a. Go back to the schematic and change the amplifier gain S21 from 10 to **20** dB as shown here.




b. In **Simulate > Simulation Setup**, change the dataset name to **rf_sys_20dB**. Click **Apply** and **Simulate**.

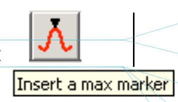
c. When the simulation finishes you will be prompted to change the default dataset – answer: **No**. **No**.




d. Edit the existing plot (double click on it) – this is this is the one with the 10dB trace. When the the dialog appears, click the arrow to see the the available Datasets and Equations (shown here) and select the **rf_sys_20dB** dataset.

e. Select the **S(2,1)** data and **Add** it in **dB**, clicking **OK**. Notice that the entire dataset pathname appears because it is not the default dataset.

f. Put a **Max Marker** on the new trace. **Select both markers** (select the (select the readouts) and click the icon to **Turn on Delta Mode** (select (select either as a reference) to see the 10dB difference between the between the two simulations. Be sure to **save the Data Display**.



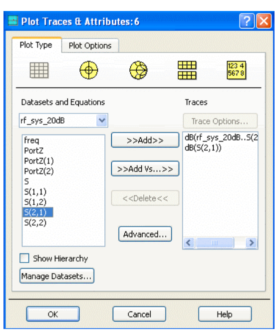

**Deleted:** n

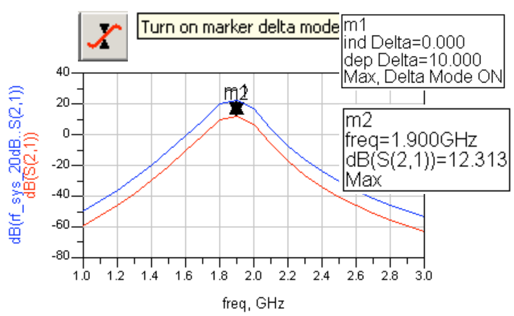
**Deleted:** ew marker

**Deleted:** command **Ma**

**Deleted:** **rker >**

**Deleted:** **On**



**Display**.

6. **Set up an RF source and LO with Phase Noise.**

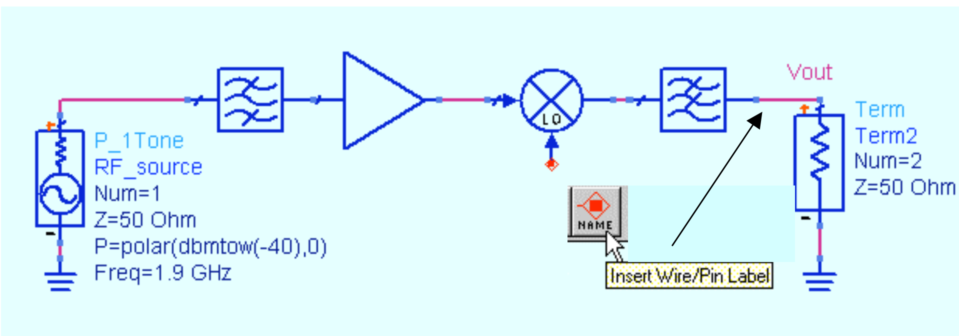
This next step shows how to simulate phase noise, contributed by a behavioral oscillator, using the Harmonic Balance simulator. At this point in the course, it is not required that you understand all the Harmonic Balance settings (covered later).

a. Save the current schematic with a new name. Click**: File > Save Design As** and type in the name: **rf_sys_phnoise**.

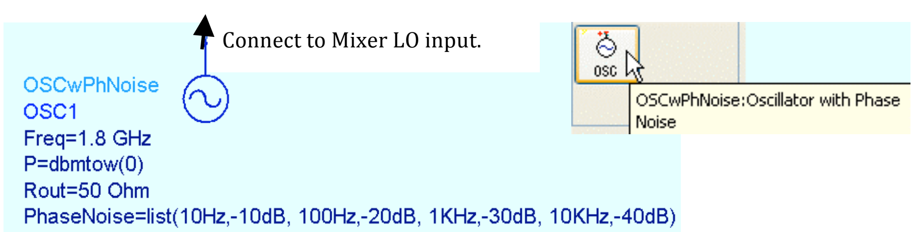
b. In the saved schematic, delete the following components: **S_param simulation controller**, the **V_1Tone** LO source, its **50 ohm resistor** and ground.

c. Replace the port 1 Term with a **P_1Tone** source (Sources-Freq Domain palette) and set the power and frequency as shown: Freq = **1.9 GHz** and P = polar (dbmtow (**-40**), 0). Also, rename the source as **RF_source** and change the Num parameter to **Num =1**.

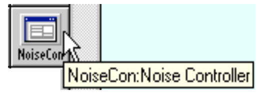
d. Insert a wire label **Vout** (node) and so the schematic looks like the one shown here:



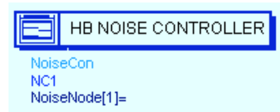

e. Go to **Sources-Freq Domain** palette, scroll to the bottom, select the **OSC** icon and insert the **OSCwPhNoise** - connect it to the mixer. Set **Freq = 1.8 GHz** and change the PhaseNoise **list** as shown. The default value of P is the power in dBm and it has 50 ohms Z (Rout).



### 7. Set up a HB Noise Controller.



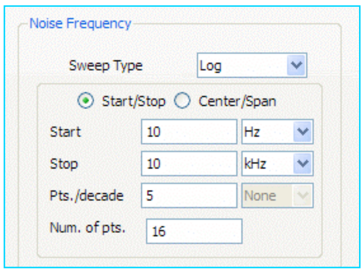

a. Go to the **Simulation-HB** palette and insert a **NoiseCon** (Noise Controller) on the schematic as shown here.

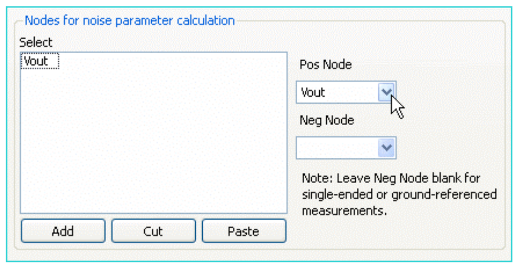



**NOTE on NoiseCon:** This component is used with the HB simulator. It allows you to conveniently keep all noise measurements separate from the simulation controller. Also, you can setup and use multiple noise cons for different noise measurements while only using only one HB controller.

b. **Freq tab -** Edit the Noise Con – go to the **Freq** tab and set the Sweep Type to Log from 10 Hz to 10 KHz with 5 points per decade.

**Deleted:** 10KHz

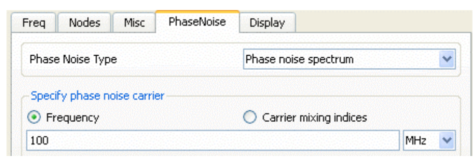



c. **Nodes tab** – Click the **Pos Node** arrow, select the **Vout** node, and click the **Add** button. The noise controller, like other ADS componets, can read and identify node names in the schematic.

d. **PhaseNoise tab** – Set the Phase Noise Noise Type: **Phase Noise spectrum spectrum** and set the **carrier Frequency** to **100 MHz.** This is the IF the IF frequency which has phase noise

phase noise due to the LO.



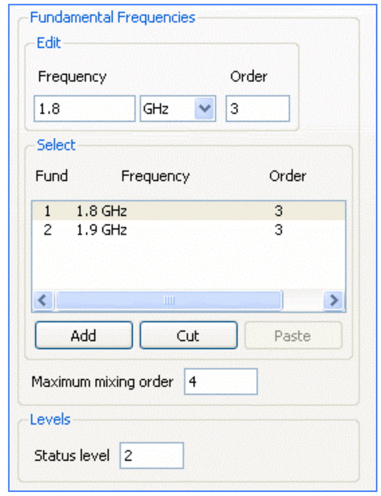

e. **Display tab** – Go to the Display tab and check the boxes for the settings you made (shown here). In the future, you may prefer to display the desired settings first and then edit them on the schematic.

Display these settings:

**8. Set up the HB simulation.**

a. Go to the Simulation-HB palette and insert a **HB** simulation controller on controller on the schematic.

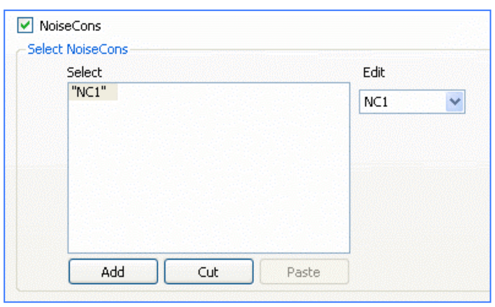

b. Edit the HB controller (double click). In the **Freq Freq** tab, change the default freq setting to **1.8 GHz 1.8 GHz** using the **Apply** button. Then add the RF the RF frequency **1.9 GHz** and click **Apply** again. again.

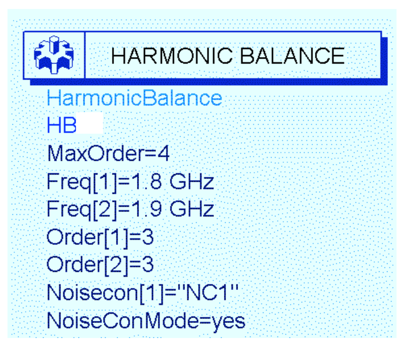



c. In the **Display** tab, check the box to display **MaxOrder** and click **Apply** at the bottom

**NOTE on HB freq settings** - You only need to specify the LO freq (1.8 GHz) and the RF freq (1.9 GHz) in the controller. There is no need to specify any other frequencies because the defaults for Order (harmonics) and Maximum order (mixing products) will calculate the other tones in the circuit, including the 100 MHz IF.

d. Go to the **Noise tab** and check the **NoiseCons** box as shown. Then use the Edit button to select **NC1** which is the instance name of the Noise Con. Click **Add** and **Apply**.

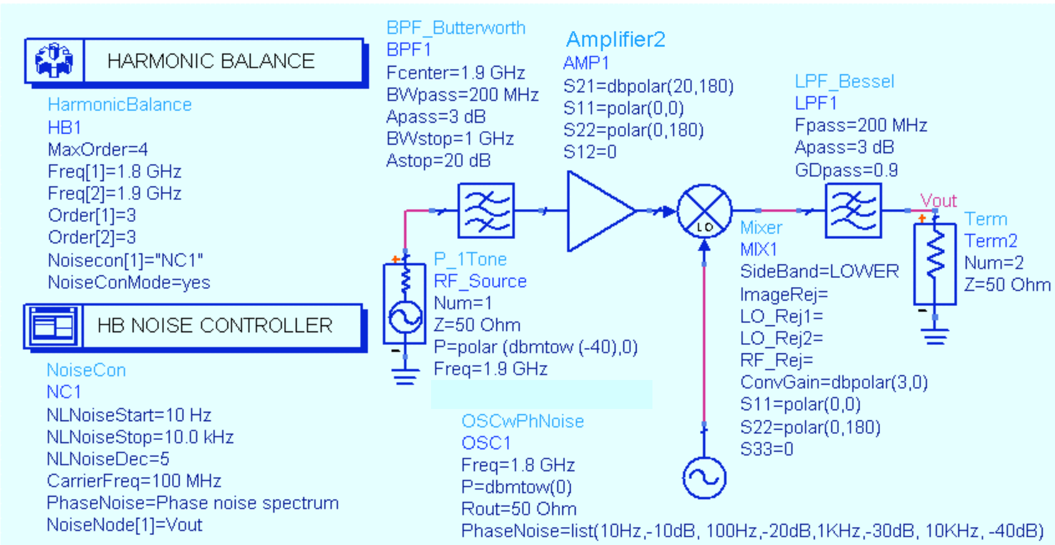



e. Display tab – Go to the HB **Display** tab and check the boxes for the settings shown here. The noise con settings are near the bottom of the list as you scroll down.

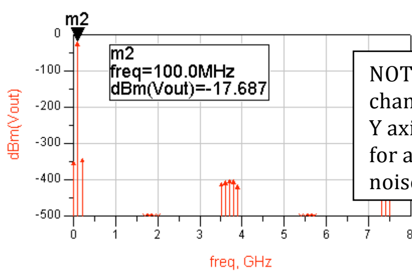
 

The complete schematic for simulating LO Phase Noise at the IF is shown here. Check your schematic before simulating:




**9. Simulate and plot the response: pnmx and Vout.**

a. Insert a rectangular plot of **pnmx**. Use **Plot Options** to set the **X**-axis to **Log** scale. Notice trace shows the decreasing dB values assigned in the oscillator setting (for example: about 30dB at 1 KHz). Also, insert a rectangular plot of **Vout** in **dBm** with a marker on the 100 MHz IF signal. At -40 dBm input, plus about 23 dB of amp and conversion gain, the output should be about –17.7 dBm as shown.


NOTE: You can change the minimum Y axis value to –100 for a more realistic noise floor.

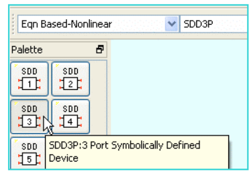
b. **Save** all your work. You have now completed the first step in the design process for the RF receiver. In the following labs, you will build the circuits that will replace the system model components.

## 10. OPTIONAL - SDD (Symbolically Defined Device) simulation

SDDs allow you to write an equation to describe the behavior at the nodes of a component, either linear or nonlinear. For this step, you will write a simple linear equation describing sums and differences that appear at the output of a 3 port SDD.

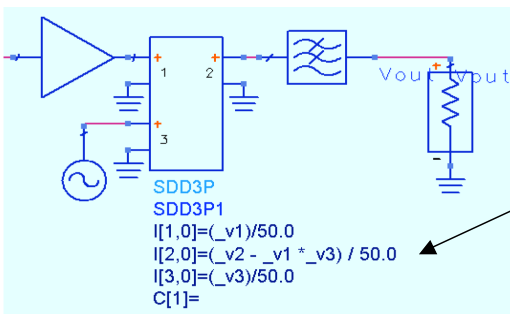
a. Use **Save Design As** to give the current design (rf_sys_phnoise) the name: **rf_sys_sdd**.

b. Delete the behavioral mixer in the circuit.



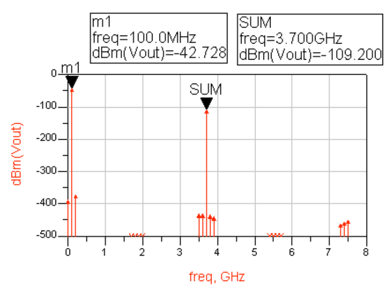

c. Go to the palette **Eqn Based-Nonlinear** and insert the **3** the **3 port SDD** in schematic, in place of the mixer. Connect Connect **grounds** on the negative terminals as shown here. here.

d. Edit the **I[2,0]** value by inserting the cursor directly on the on the text and adding the values shown: **- _ v1 * _v3**. By subtracting the voltage of the mixing terms of the RF (_v1) and LO (_v3), the IF (_v2) voltage remains. The SDD is now a mixer with no conversion gain, and both the sum and the difference frequencies will appear at the output.



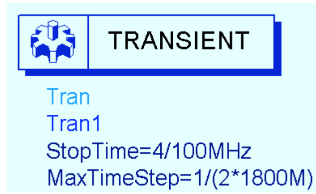

NOTE: SDDs perform numerical operations. This means v1* v3 is a product of the voltages at terminals 1 and 3.

e. **Simulate** and plot the **spectrum of Vout in dBm**. **in dBm**. As you can see, without conversion gain, conversion gain, the IF signal is much lower. lower. Also, both the difference and the sum sum (RF+LO) appear (marker: SUM). Although Although SDDs can be useful to describe behavior, writing the proper equations can be be complicated (requires advanced course). course).



f. Deactivate the HB and NC controllers.

g. Insert a Transient simulation controller and use the setup shown here. Also, use Simulation > Setup to change the dataset name to: rf_sys_sdd_trans.

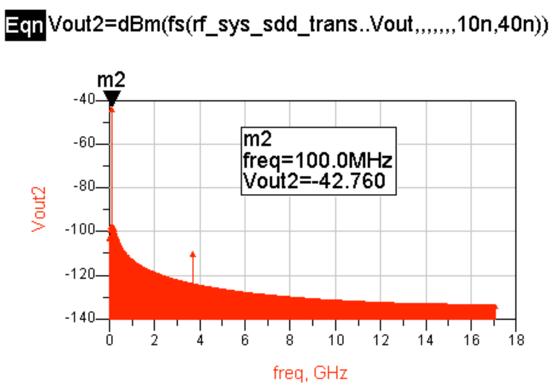



h. Run the Transient simulation.

i. Do not change default datasets in the Data Display.

j. Insert an equation (shown here as Vout2) that uses the **fs** () function to transform the data – be sure to include the 7 commas after Vout (these skip arguments). The 10n argument is the start time of 10 nanoseconds and 40n is the stop time of 40 nanoseconds. NOTE: You could also use Trace Options > Trace Expression on Vout and then modify the expression instead of writing an equation and then plotting it.

k. Insert a plot of the equation. As you can see, the 100MHz tone compares with the HB data extremely well (< 0.1 dB difference).




IMPORTANT NOTE: this step is used only to show how to set up an SDD mixer (especially the multiplier settings). If you use other models in this same setup for a comparison, you may get different results (especially Transient) because such models may have non-causal responses. Also, delay can be added to some filters to eliminate the non-causal effect.

l. Save the design and data.

## *EXTRA EXERCISES*:

1. Using the rf_sys_phnoise design, run a Transient simulation for the system (not using the SDD) and compare the results with the **fs** function.

2. Go back and replace the Butterworth filter with an elliptical filter model shown here and simulate. Try setting different ranges for the Ripple value or try using the tuner to adjust the ripple parameter. Then display the results and look at the ripple in the passband. To do this, you will have to use the zoom commands on the data display.


3. Try tuning various parameters in the design.

4. Enter values of LO and RF rejection to the behavioral mixer and look at the simulation results.