cadence®

Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

Product Version 23.1
September 2023

Contents

1

Introduction

This chapter introduces you to the basic building blocks of running successful and effective high frequency simulations using the Cadence® software. The following sections and screenshots explain how to set up your software and environment to run the example circuits in this chapter.

Before you perform the various SpectreRF analyses, you need to set up the component files and start the Cadence® software, as explained in the sections below.

Using SpectreRF from the SPECTRE Hierarchy

The Spectre Circuit Simulator (Spectre) and SpectreRF are present in the SPECTRE release stream.You must download and install the SPECTRE simulator in a separate installation hierarchy than the IC hierarchy you use for the Cadence software. Documentation for new features and most bug fixes are provided exclusively with the SPECTRE release stream.

The SpectreRF examples in this chapter use the `CDSHOME` environment variable to point to the installation hierarchy.

`CDSHOME`	Modify this path as necessary to point to the IC installation directory.

`CDSHOME` may already be set in your environment. If it is already set, then there is no need to reset it. Please check with your Cadence Tool System Administrator for more information.

Accessing the Most Current SpectreRF Documentation

The documentation for the latest features of SpectreRF is always found in the SPECTRE hierarchy.

Important

Note that the help buttons on the forms in ADE Explorer lead you to the IC version of the documentation, which should not be used. Instead, use the SPECTRE documentation located in the SPECTRE hierarchy.

Creating a Local Editable Copy of the ExampleLibRF Library

Perform the following steps to create a copy of the *ExampleLibRF* library and save it to your home or working directory:

1. Navigate to the directory where you want the workshop to be located.
2. Use the UNIX `cp` command to copy `RF_Doc_Database.tar.gz` from the hierarchy to your desired directory.

   `cp <path to>/RF_Doc_Database.tar.gz`
3. Type `tar xfz RF_Doc_Database.tar.gz.`

Work with your system administrator to locate the SPECTRE installation directory at your site. This will be in the SPECTRE hierarchy at `<SPECTRE>/tools/spectre/examples/SpectreRF_workshop/RF_Doc_Database.tar.gz`

Downloading and Using GPDK180

PDK is an abbreviation for Process Design Kit. A PDK is a complete set of technology files to enable analog and mixed signal custom IC circuit design within the Cadence Design System's Custom IC Design Environment. These PDKs are available for download online.

1. Navigate to *<path to>/RF_Doc_Database.*
2. Use the UNIX `ls` command to list the contents in the directory.

   You will see the following files and directories:

   - ❑ `cds.lib*`
   - ❑ `doc/`
   - ❑ `libs/`
   - ❑ `models/`
   - ❑ `readme setup.csh*`
   - ❑ `share/`

- `simulation/`
- `skill/`

This directory structure is organized as follows:

- `cds.lib`: The Cadence library file for the project
- `simulation`: Simulation directory
- `libs`: Directory containing the libraries for the project
- `skill`: Any SKILL code needed
- `models`: Spectre models that are not gpdk specific
- `share`: Directory where gpdk180 is located after being downloaded from the `pdk.cadence.com` site.

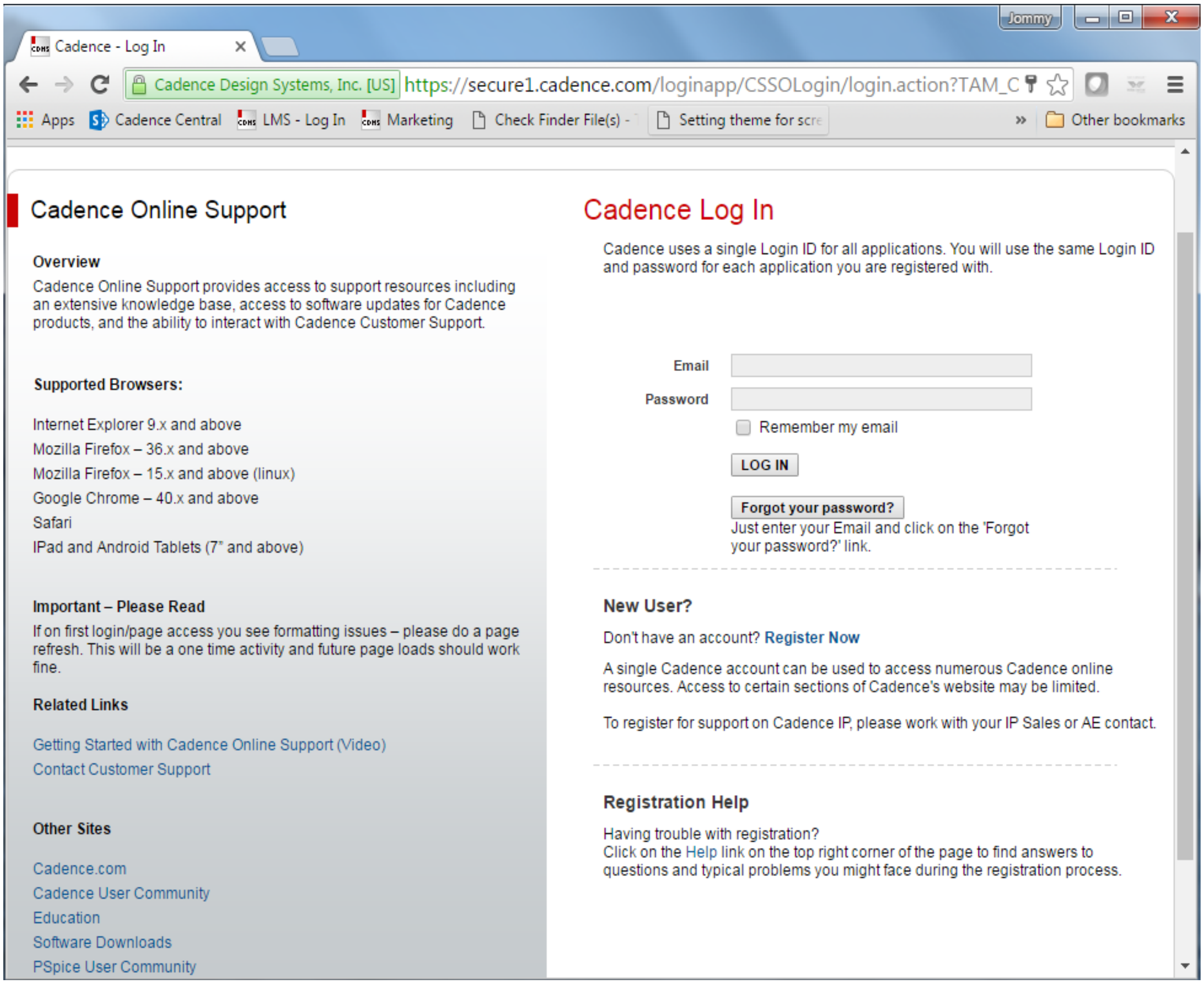
For example, the gpdk needs to be present in the `/share/gpdk180_v3.3` directory.

The models are present in the `/share/gpdk180_v3.3/models/spectre` directory.

To download gpdk180, follow these steps:

1. Open a browser and go to *http://support.cadence.com*. The Cadence Online Support website is displayed, as shown below.

Figure 1-1 Cadence Online Support Website



2. Specify your email and password in the *Email* and *Password* fields and click *LOG IN.*

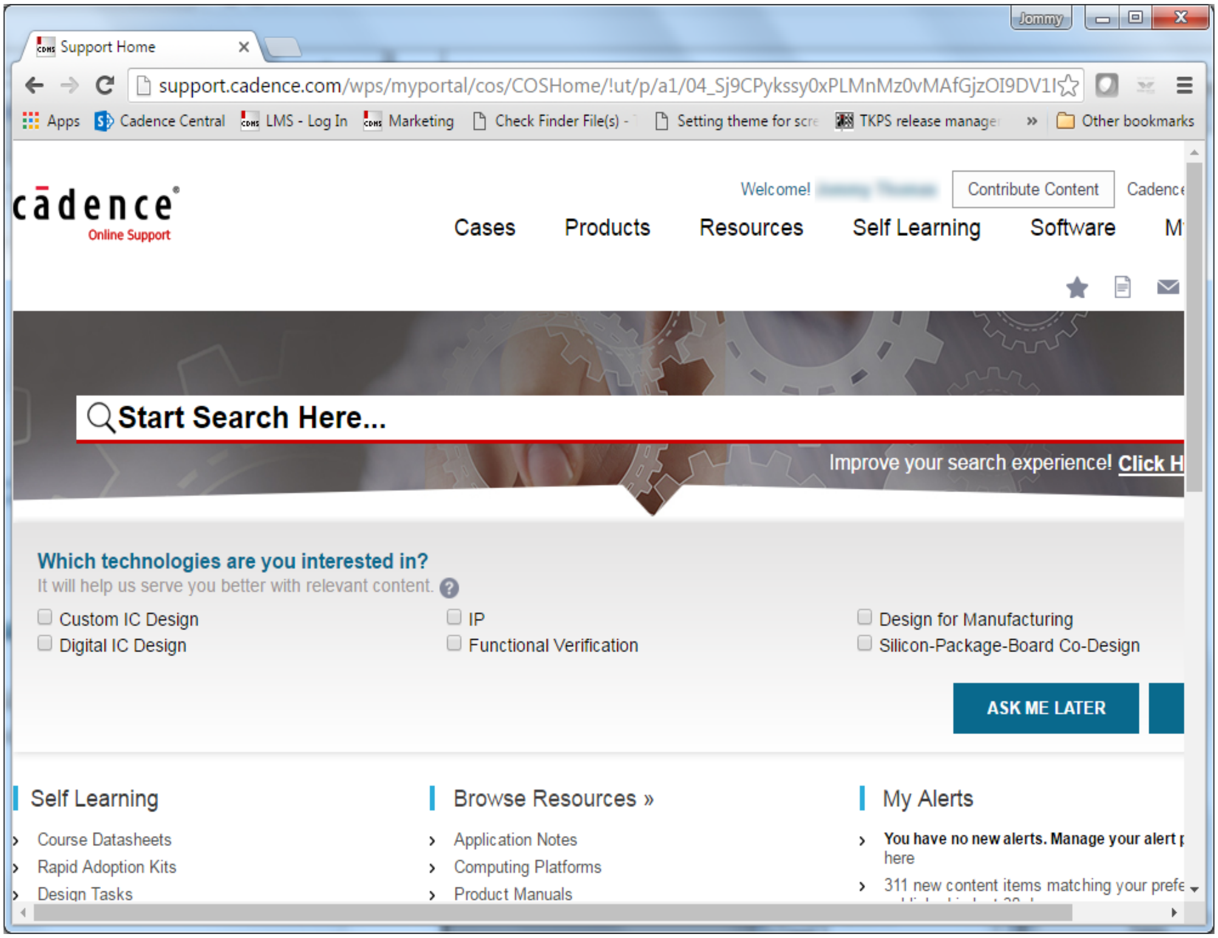
   Note: If you are new user, click *Register Now* and follow the steps to register yourself.

For the example designs in Appendix A, you need to use Cadence's GPDK180 which can be downloaded as follows.

1. Enter your email and password.

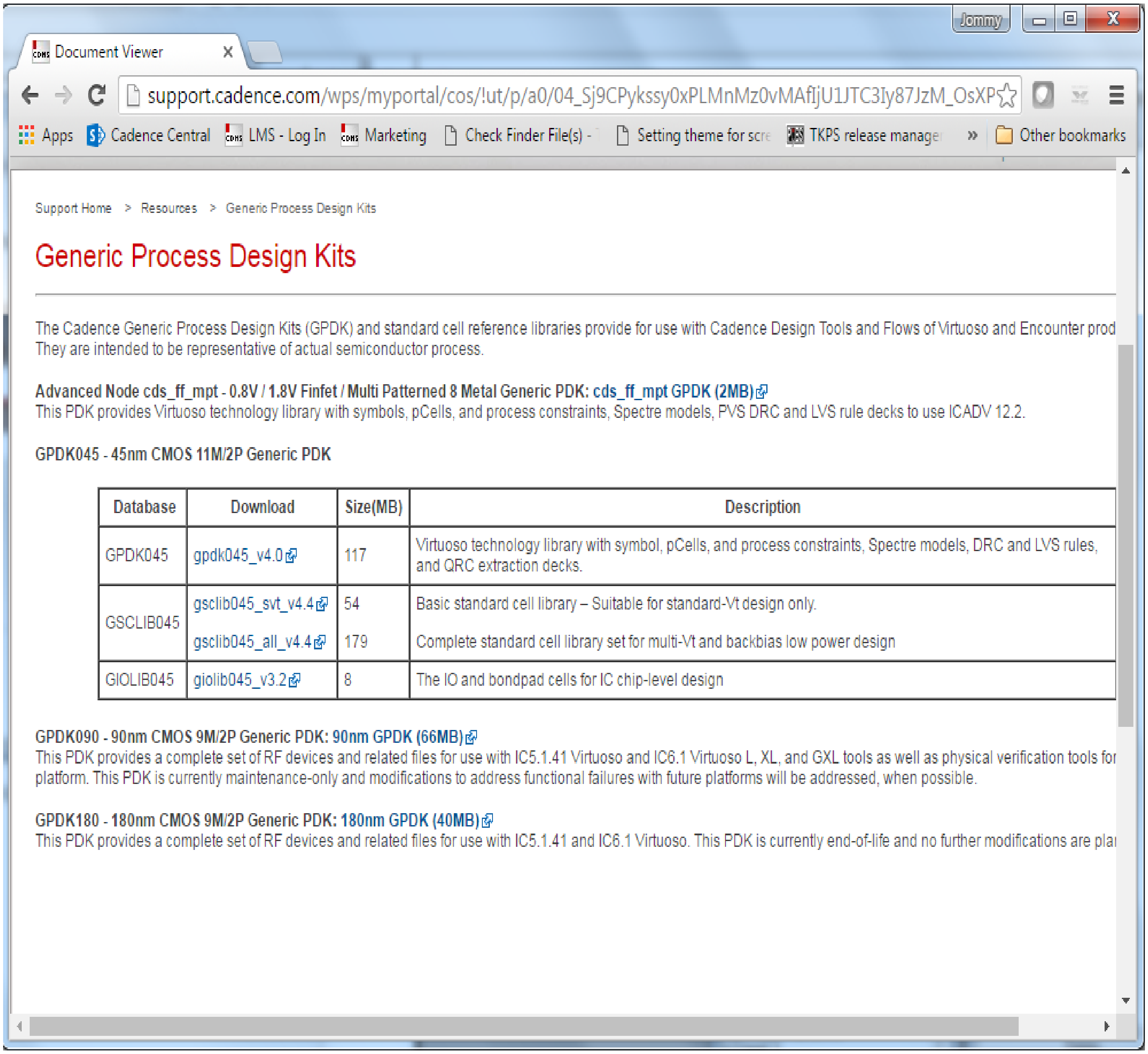
   The *Support Home* web page is displayed, as shown below.

Figure 1-2 Cadence Support Home Page

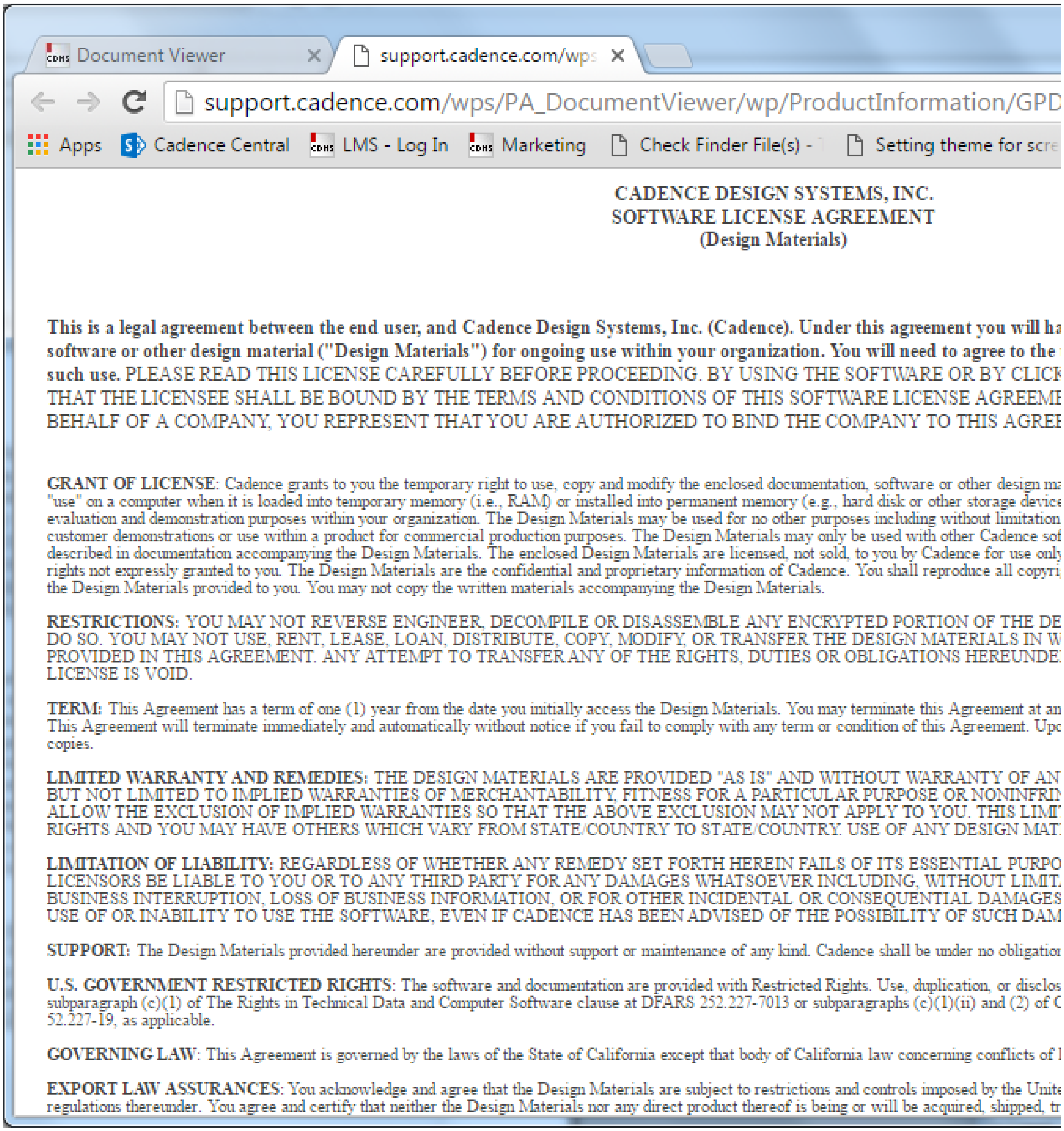


2. Select *Resources - GPDKs*. The *Generic Process Design Kits* Web page is displayed, as shown below.

Figure 1-3 Generic Process Design Kits Web Page

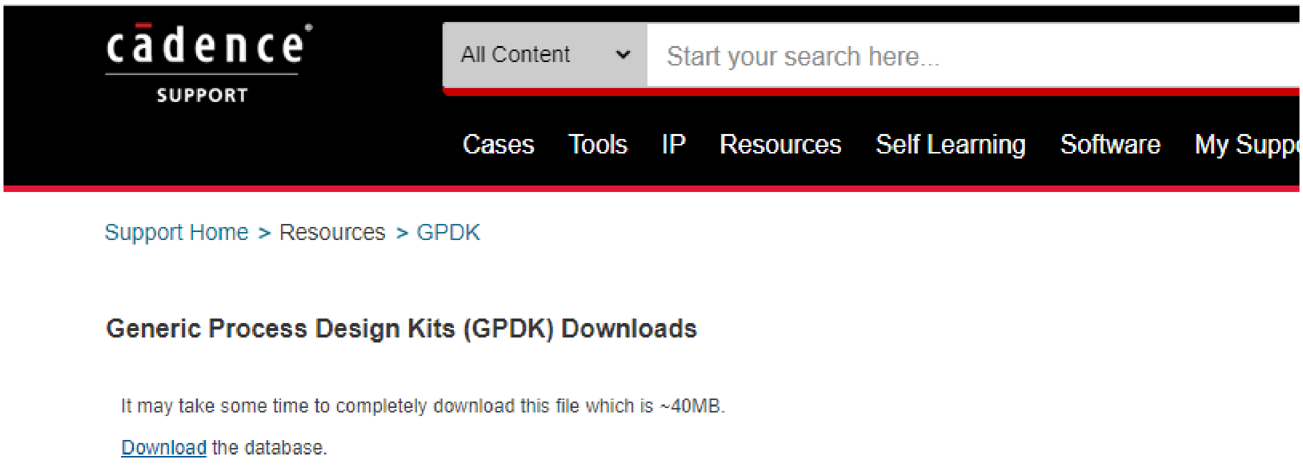


3. Click the *180nm GPDK (40MB)* link. The Software License Agreement page is displayed, as shown below.



4. Scroll down the page and click *I ACCEPT*.

5. The Generic Process Design Kits (GPDK) Downloads Web page is displayed, as shown below.



6. Click *Download the database*.

   The file will start downloading. The time taken to download is about a minute, but that varies depending on the nature and speed of your Internet connection.

7. Save the file `gpdk180_v3.3.tar.gz` to `<path to>/RF_Doc_Database/share`.

8. Navigate to the `RF_Doc_Database/share` directory and untar the `gpdk180_v3.3.tar.gz` file using the following command.

```
tar -xvzf gpdk180_v3.3.tar.gz
```

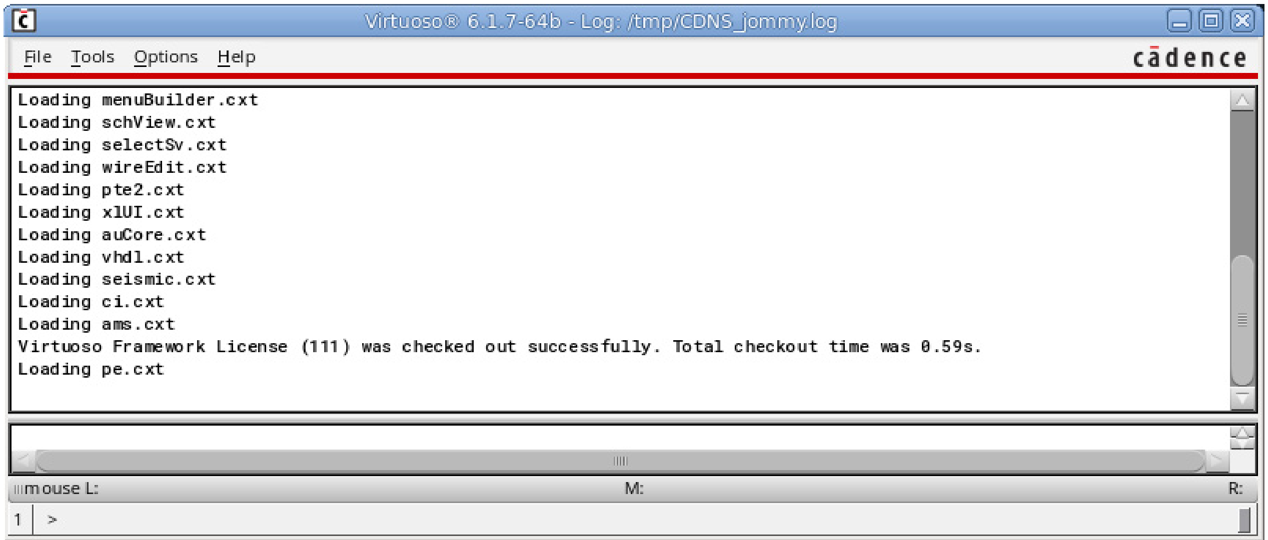
Starting Virtuoso

To access the Library Path Editor, perform the following steps:

1. In a UNIX window, type `virtuoso &` to start the Cadence software.

   The Command Interpreter Window (CIW) is displayed, as shown below.

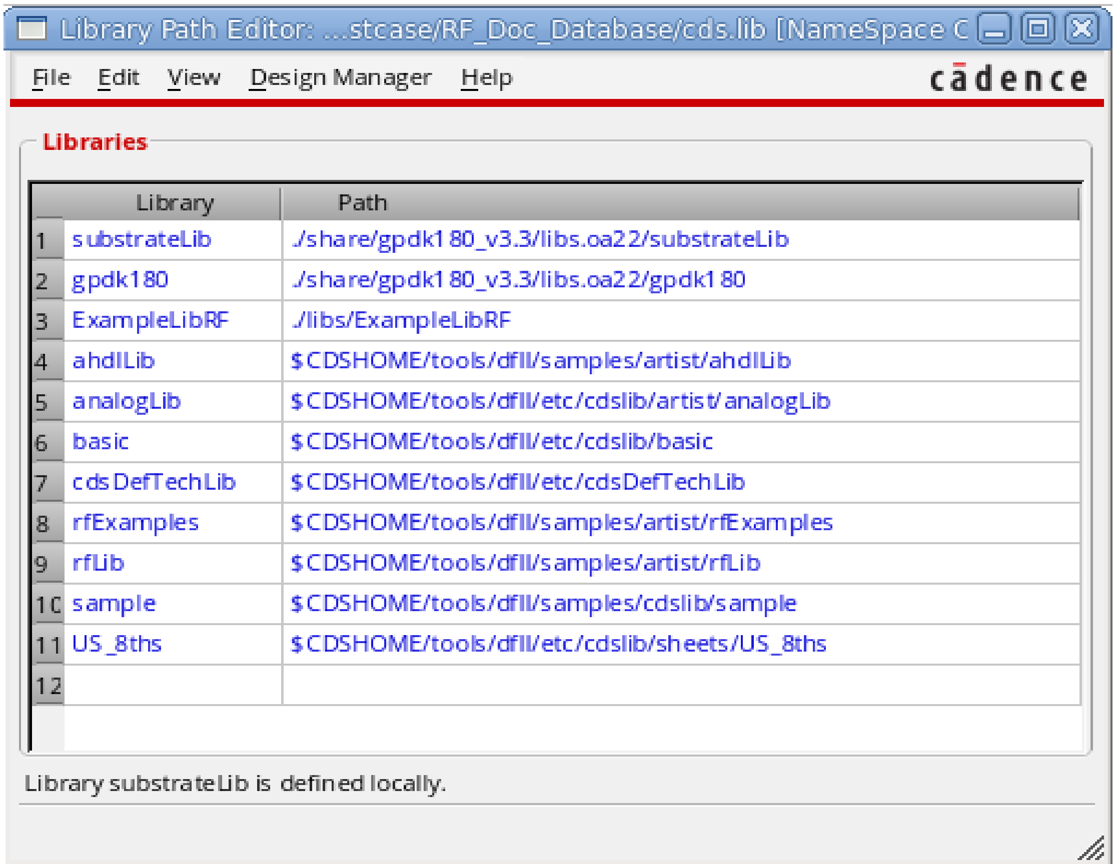
Figure 1-4 Command Interpreter Window



2. In CIW, choose *Tools – Library Path Editor*.
3. The *Library Path Editor* is displayed.

   This next step is to check that all of the libraries necessary are accessible. You will not be changing anything.

Figure 1-5 Library Path Editor Window



Make sure that all of the library names are displayed in blue or green. If any libraries are displayed in red text, there is an error and you need to work with your system administrator to fix the path.

4. Exit the *Library Path Editor*.

2

Simulating Low Noise Amplifiers

The SpectreRF simulator can simulate very linear circuits, such as Low Noise Amplifiers (LNA). This section uses the lna Simple circuit to illustrate how the SpectreRF simulator can determine the characteristics of an LNA design.

The first stage of a receiver is typically an LNA, whose main function is to set the boundary as well as to provide enough gain to overcome the noise of subsequent stages (for example, in the mixer and IF Amplifier). Apart from providing enough gain in addition to introducing as little noise as possible, an LNA should accommodate large signals without distortions, offer a large dynamic range, and provide good matching to its input and output. Good matching is extremely important if a passive band-select and image-reject filter precedes and succeeds the LNA because the transfer characteristics of many filters are sensitive to the quality of the termination.

In the LNA example that follows, you will plot the following characteristics of the low noise amplifier.

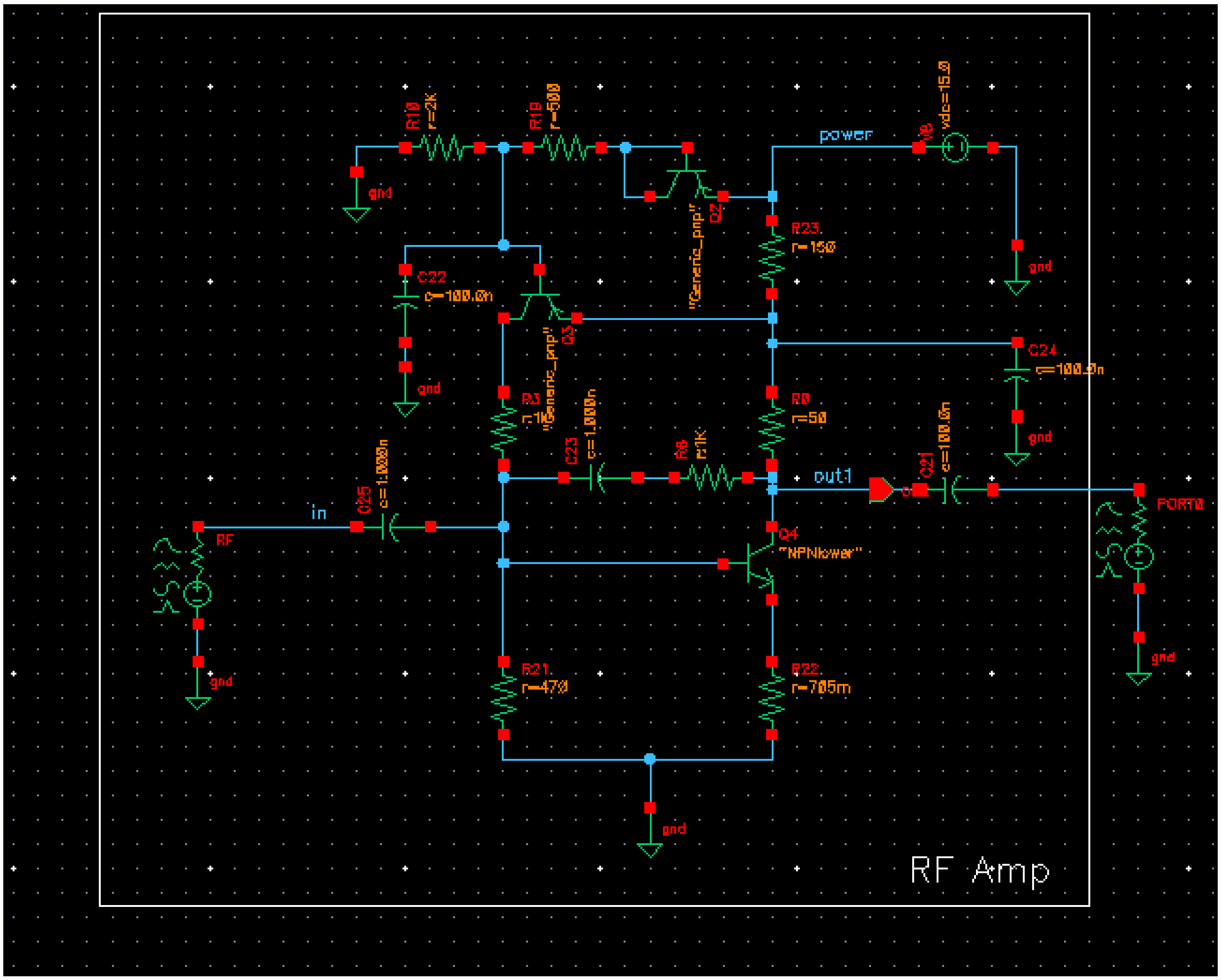
LNA Measurements (lnaSimple	**Analyses**
S-Parameter Analysis, Small Signal Gain	sp
Stability, Stability Circles	sp
Linear two-port Noise Frequency Measurements, Noise Circles	sp+noise
Third-Order Intercept Measurement	hb
Rapid IP3 Using Specialized AC Analysis	ac

To use this section, you must be familiar with the SpectreRF simulator analyses as well as know about LNA designs. For more information about the SpectreRF simulator analyses, refer to the various chapters in this user guide and also the *SpectreRF Simulation Option Theory.*

The lnaSimple Low Noise Amplifier Circuit

The lnaSimple circuit can be found in the *ExampleLibRF* library. Refer to the Introduction chapter for the instructions on accessing the *ExampleLibRF* library. The schematic for the lnaSimple circuit is shown below. It is a differential low noise amplifier.

Figure 2-1 Schematic for the lnaSimple Low Noise Amplifier



The following tables list some measured values for different aspects of the lnaSimple low noise amplifier.

Measurement	Measured
RF frequencies (Hz)	900M
Output frequency (Hz)	900M
RF voltage	200mV peak
RF power	-10 dBm
Gain	*measurement needed*
Stability Factor	*measurement needed*
Noise figure	*measurement needed*
1dB compression point	*measurement needed*
Input IP3 (from swept power)	*measurement needed*
Input IP3 (from AC analysis)	*measurement needed*

Design Variable	Default Value
prf (RF power)	-10 dBm
frf1 (RF frequency)	900M

Setting Up to Simulate the lnaSimple Low Noise Amplifier

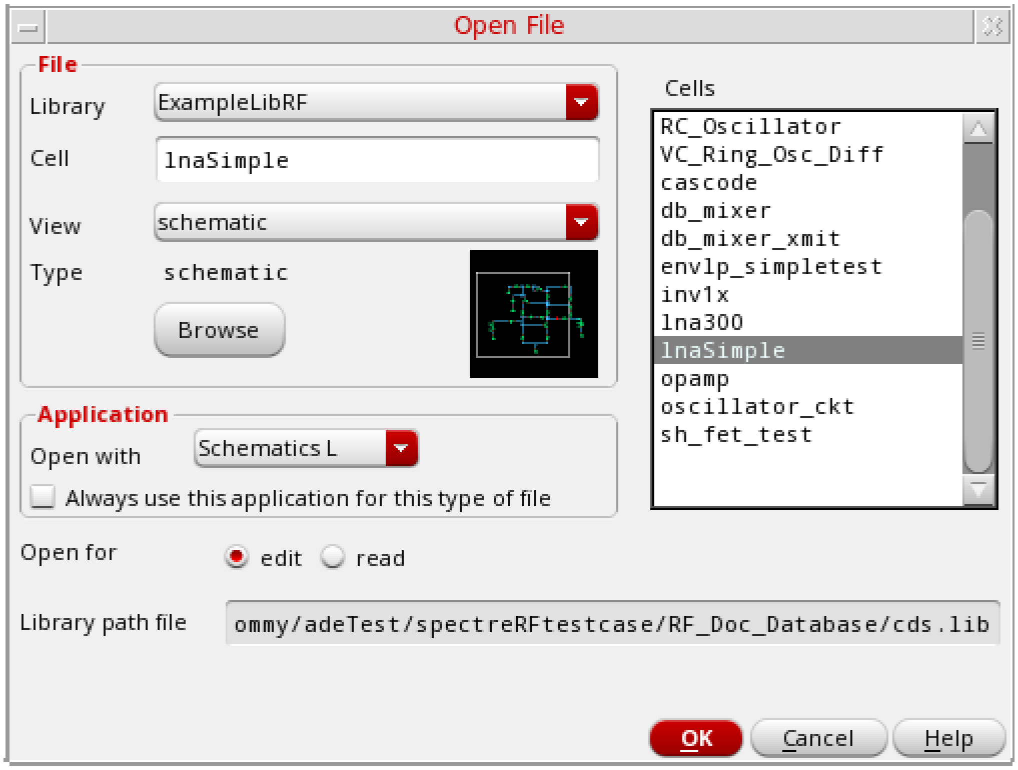
Opening the lnaSimple Circuit in the Schematic Window

1. In CIW, choose *File – Open*.

   The *Open File* form is displayed.
2. Choose *ExampleLibRF from the Library* drop-down list.
3. Choose *lnaSimple* from the *Cells* list box.
4. Choose *schematic* from the *View* drop-down list.

The completed *Open File* form will look like the one below.

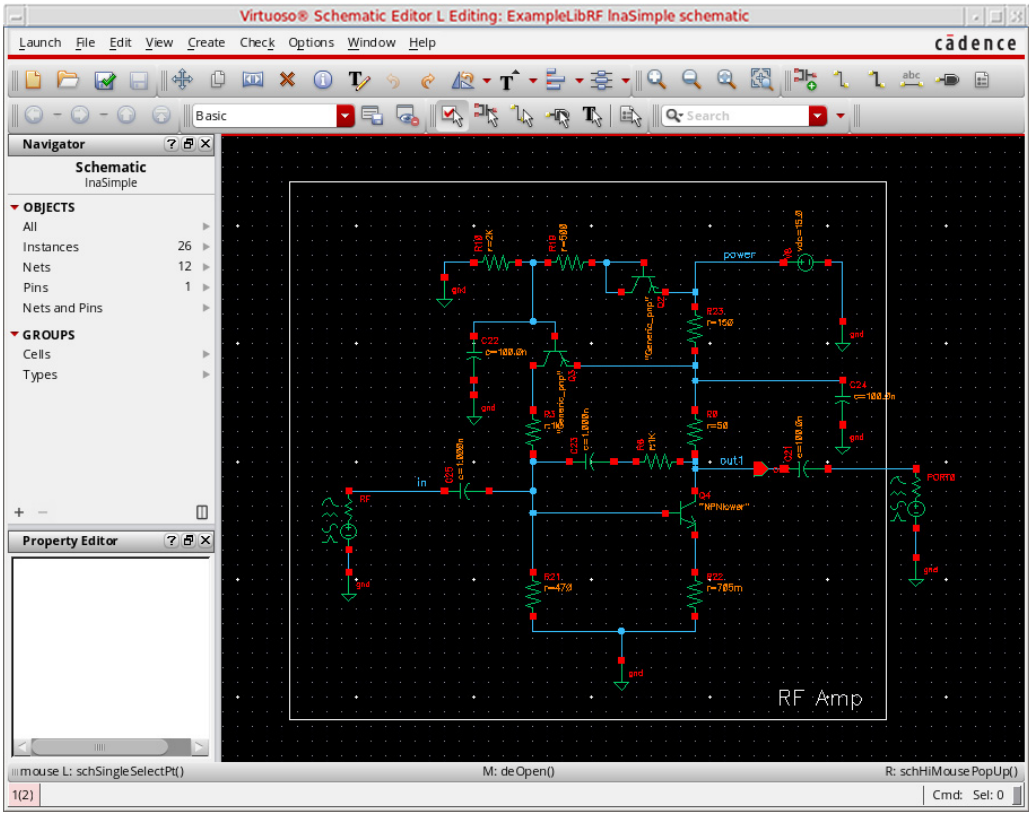
Figure 2-2 Open File Form



5. Click *OK*.

   The Schematic window for the *lnaSimple* circuit is displayed.

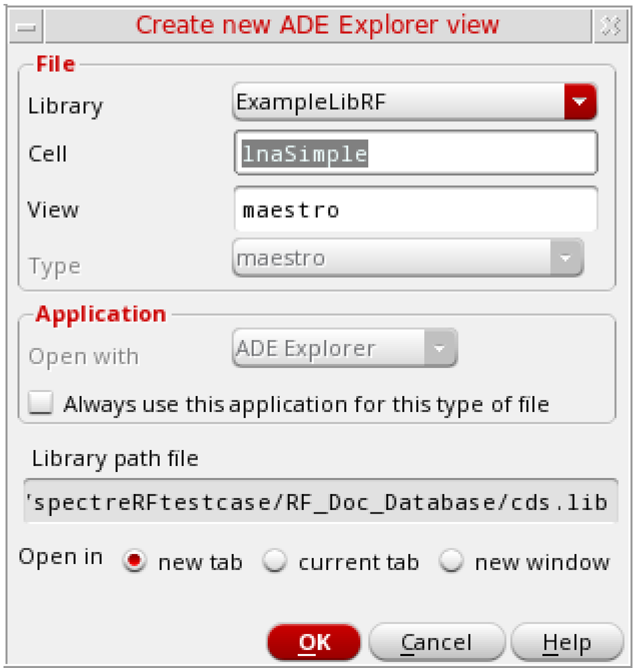
Figure 2-3 InaSimple Schematic



6. In the Schematic window, choose *Launch– ADE Explorer*.
7. In the *Launch ADE Explorer* dialog, select *Create New View*.

   The *Create new ADE Explorer view* form is displayed.

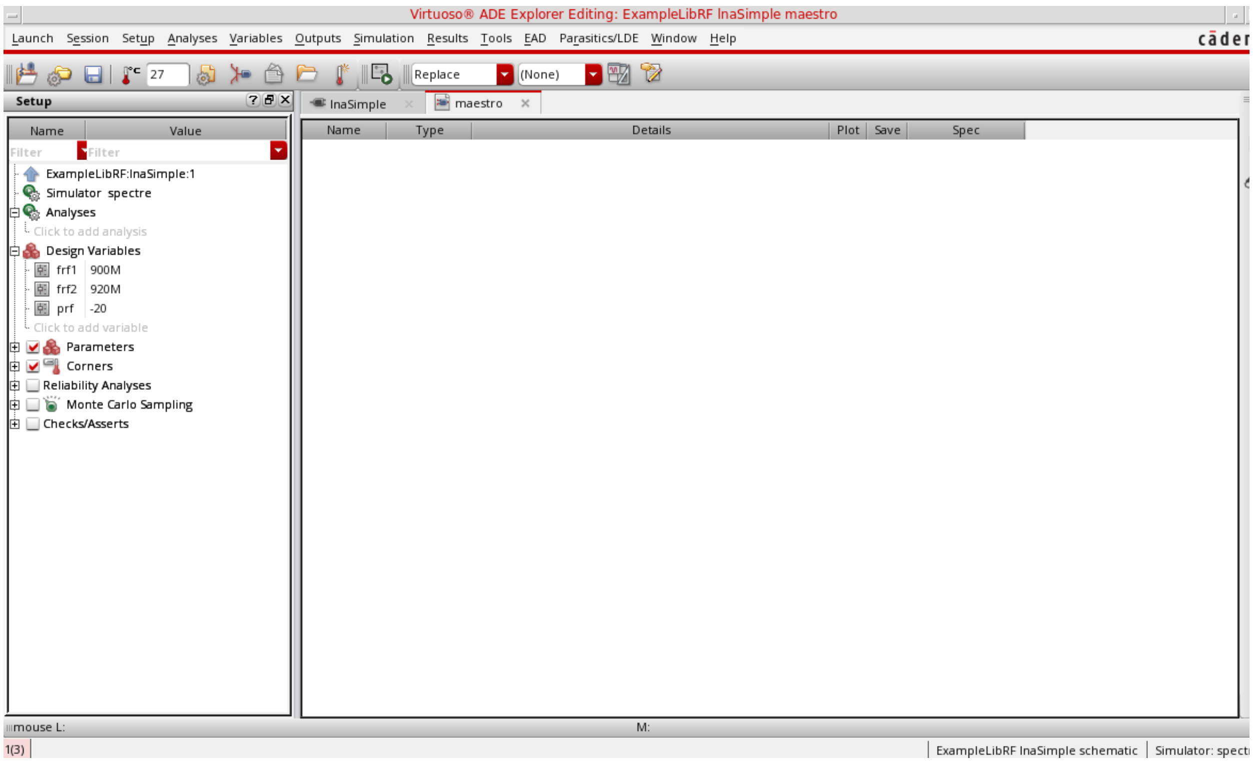
Figure 2-4 Create new ADE Explorer view



8. Leave each option to the default selections and click *OK*.

The *Virtuoso ADE Explorer* window is displayed, as shown below.

Figure 2-5 ADE Explorer Window



Choosing Simulator Options

1. Choose *Setup – Simulator* in the *ADE Explorer*

   The *Choosing Simulator* form is displayed.

Figure 2-6 Choosing Simulator Form

2. Choose *spectre* from the *Simulator* drop-down list.
3. Click *OK*.
4. Now, set up the High-Performance Simulation Options, as follows:

In ADE Explorer, select *Setup-High Performance Simulation*. The *High Performance Simulation Options* window is displayed.

Figure 2-7 High Performance Simulation Options

In the *High Performance Simulation Options* window, select *APS* for *Simulation Performance Mode*. Note that *Auto* is selected for *Multithreading options.* The effect of this is to detect the number of cores on the system (up to 64) and then multi-thread on all the available cores.

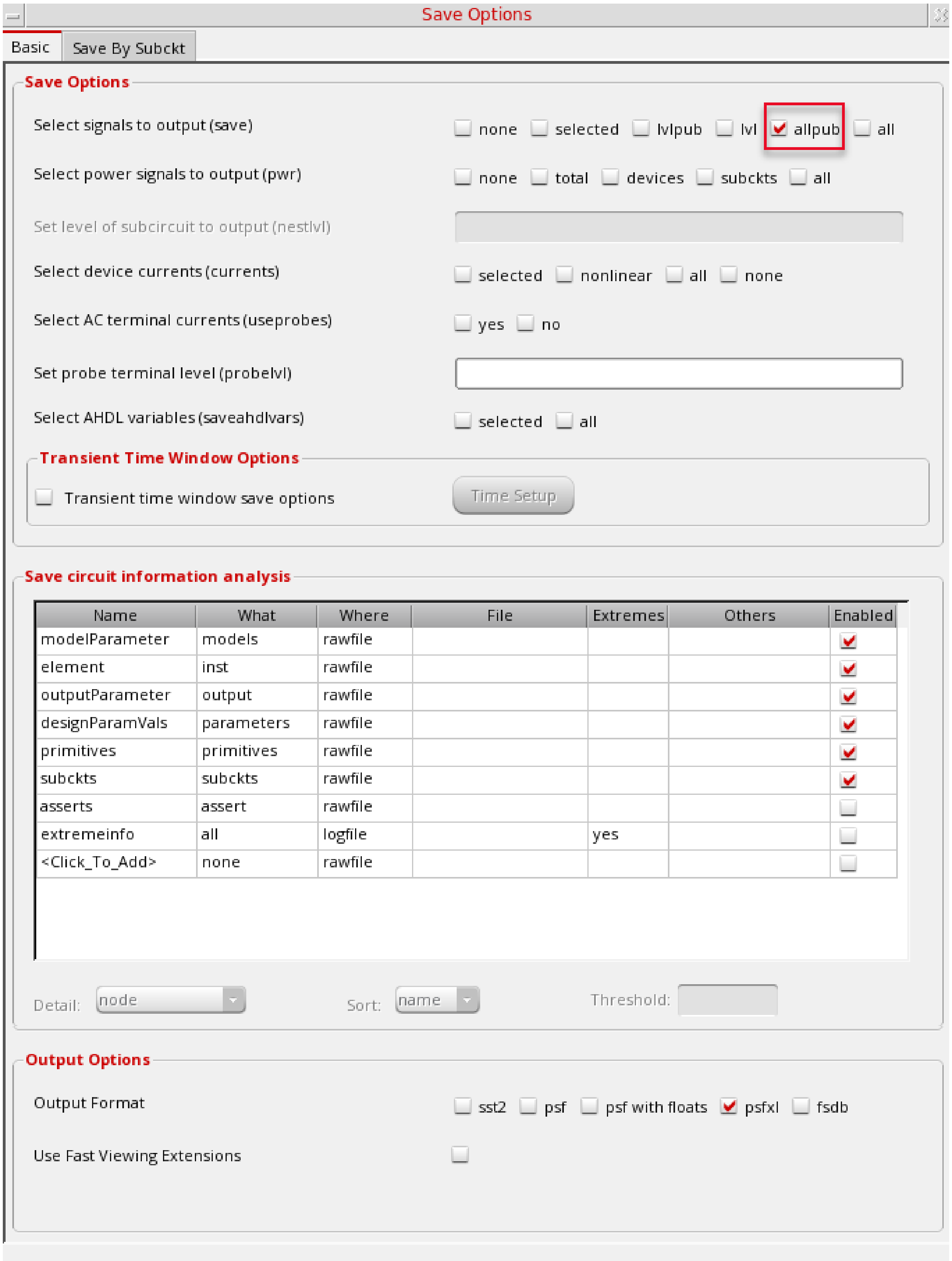
Note: The bigger the circuit, the more threads you should use. For a small circuit such as this, you may want to set the number of threads to 2. Using 16 threads on a small circuit might actually slow things down because of the overhead associated with multithreading. For more information, refer to the Spectre Classic Simulator, Spectre APS, Spectre X, and Spectre XPS.

Click *OK*.

5. In ADE Explorer, select *Outputs – Save All*.

   The *Save Options* form is displayed.

Figure 2-8 Save Options Form



6. In the *Select signals to output(save)* section, ensure that *allpub* is selected.

   This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

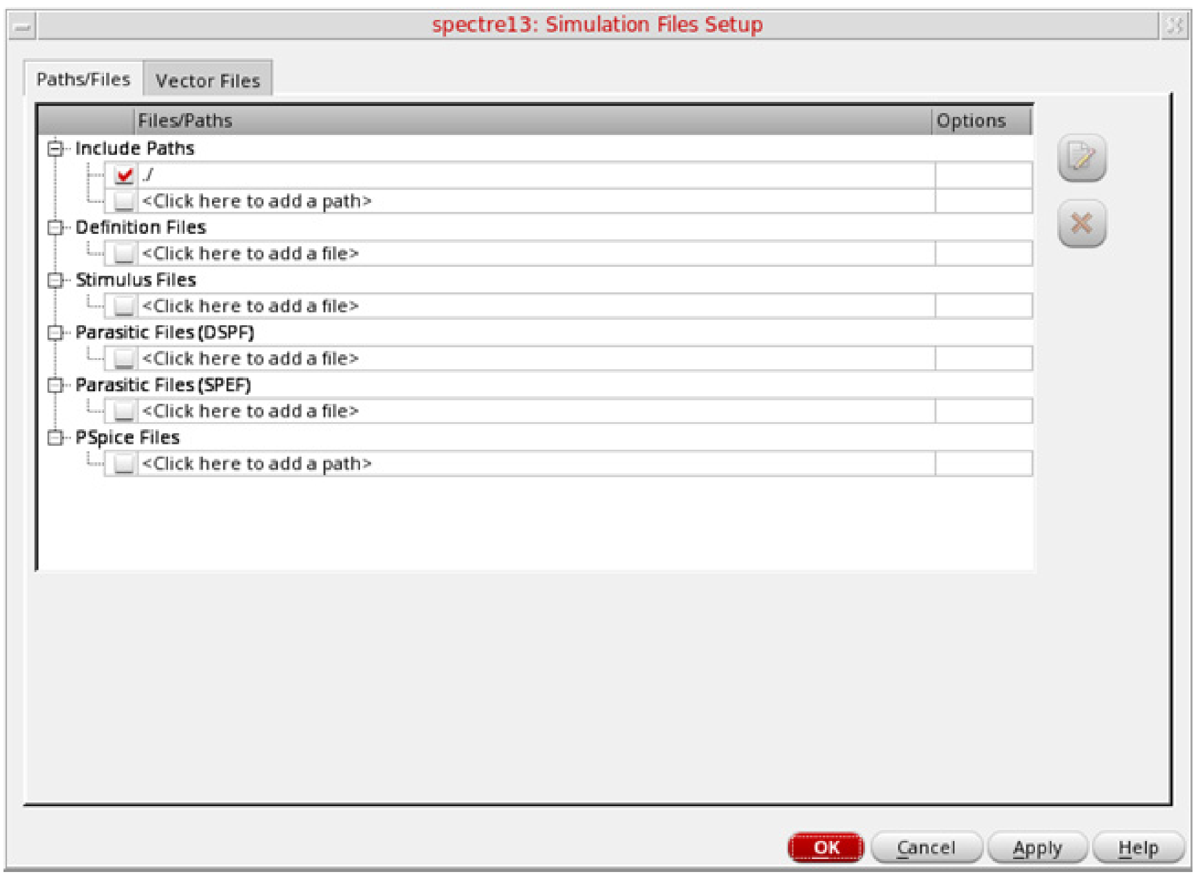
   To save the currents, select *nonlinear* in the *Select device currents (currents)* section if you just want to save the device currents, or select *all* if you want to save all the currents in the circuit.

7. Click *OK* to close the *Save Options* form.

Setting Up Model Libraries

1. In *ADE Explorer*, choose *Setup - Simulation Files*. The *Simulation Files Setup* form is displayed.

Figure 2-9 Simulation Files Setup Form



2. Verify that the *Include Path* is set, as shown above, and click *OK* to close the form.
3. In *ADE Explorer*, choose *Setup – Model Libraries.*

The *Model Library Setup* form is displayed.

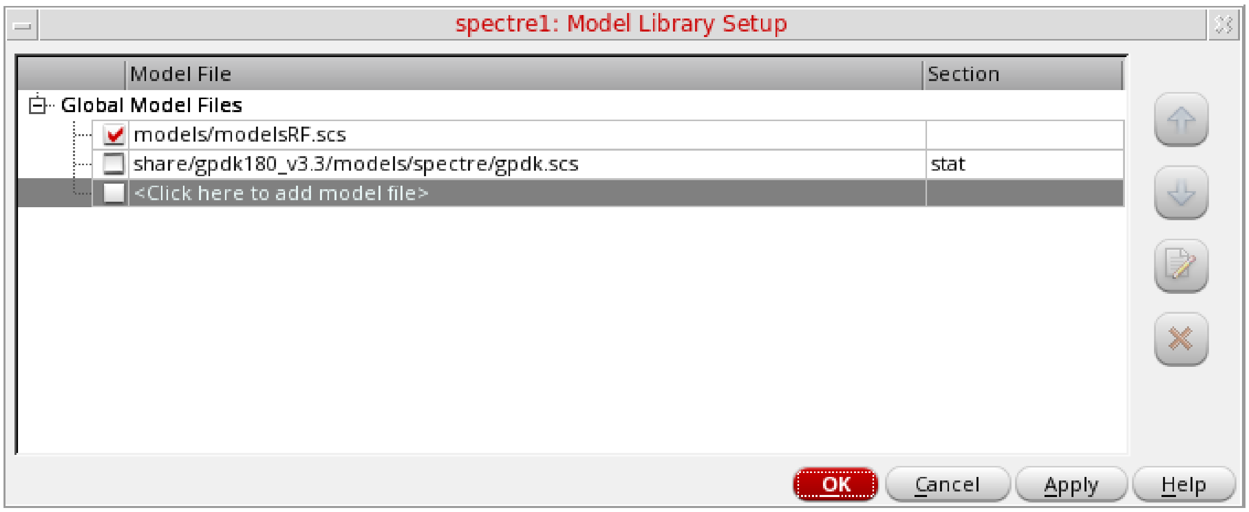
4. Type in the name of the model file, as follows:

   `models/modelsRF.scs`

5. Click *Add*.

   The *Model Library Setup* form will look like the following:

Figure 2-10 Model Library Setup



6. Click *OK*.

SP Analysis and Small Signal Gain

The S-Parameter (sp) analysis is the most useful linear small signal analysis for low noise amplifiers. In this section, you will set up an *sp* analysis by specifying the input and output ports and the range of sweep frequencies.

The S-parameter analysis linearizes the circuit about the DC operating point and computes the S-parameters of the circuit taken as an N-port. In the netlist, the port statements define the ports of the circuit. Each active port is turned on sequentially, and a linear small-signal analysis is performed. Spectre converts the response of the circuit at each active port into S-parameters and outputs these parameters. There must be at least one active port statement in the circuit.

Three power gain definitions appear in the literature and are commonly used in LNA design.

The following gain quantities are valid only for two-port circuits:

- *GA* (available gain) is the power gain obtained by optimally matching the output of the network.
- *GP* (power gain) is the power gain obtained by optimally matching the input of the network.
- *GT* (transducer gain) shows the insertion effect of a two-port circuit. This quantity is used in amplifier design.

Besides these three gain definitions, there are three additional gain definitions you can use to evaluate the LNA design.

- *Gumx* (maximum unilateral transducer power gain)
- *Gmax* (maximum available gain) shows the transducer power gain when a simultaneous conjugate match exists at both ports.
- *Gmsg* (maximum stable gain) shows the gain that can be achieved by resistively loading the two-port such that `k = 1` and then simultaneously conjugately matching the input and output ports. For conditionally stable two-ports, you can approach the maximum stable gain as you reduce the input and output mismatch. If you attempt a simultaneous conjugate match and `k < 1`, the two-port oscillates.

There are also two gain circles that are helpful to the design of input and output matching networks.

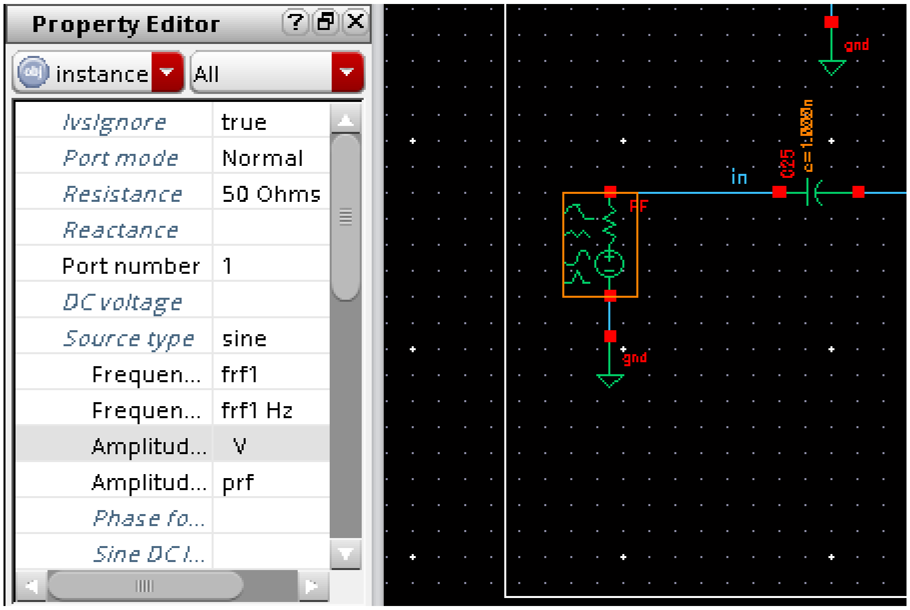
- *GPC*: power gain circle
- *GAC*: available gain circle

The next steps will walk you through these simulations and measurements.

Editing the Schematic

1. In the Schematic window, click the RF voltage source. This is the input port to lnaSimple.

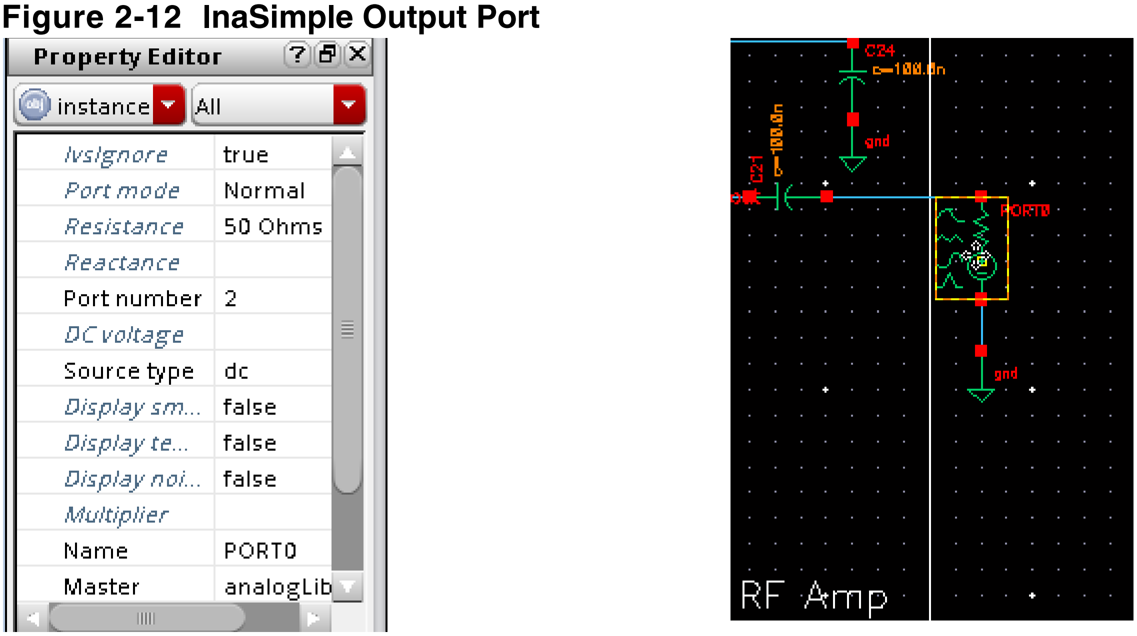
Figure 2-11 RF Voltage Source in lnaSimple



Note that when you select the port, the *Property Editor* populates with the instance properties on the port. Note that the RF port has a *Port Number* of 1 and input *Resistance* of 50 Ohms. Leave the *Source type* parameter set to `sine` and disable the frequency source in the *Design Variables* section of ADE Explorer

2. Next, click the port on the right side of the schematic. This is the output port of lnaSimple. The *Property Editor* shows the instance properties on the output port.

Figure 2-12 lnaSimple Output Port

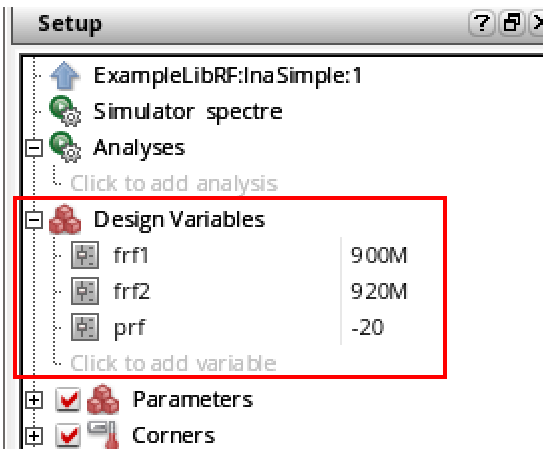


Note that the output *Port Number* is 2 and the reference *Resistance* (real part of the reference impedance) is 50 Ohms. The *Source type* is set to *dc*, as there is no large signal generated on this port.

Setting Design Variables

Perform the following steps to set the design variables to the values required for each simulation. The *Design Variables* section is located in the Setup pane of the ADE Explorer simulation window:

Figure 2-13 Design Variables Section of ADE Explorer Window



1. Change the design variables *frf1* and *frf2* to `0`. To edit the value, simply click the value to the right of the variable name, and type in a value. Then press *Enter*. Setting the input frequency to 0 disables the production of waveforms for the large-signal analyses like tran, pss, and hb (harmonic balance.)

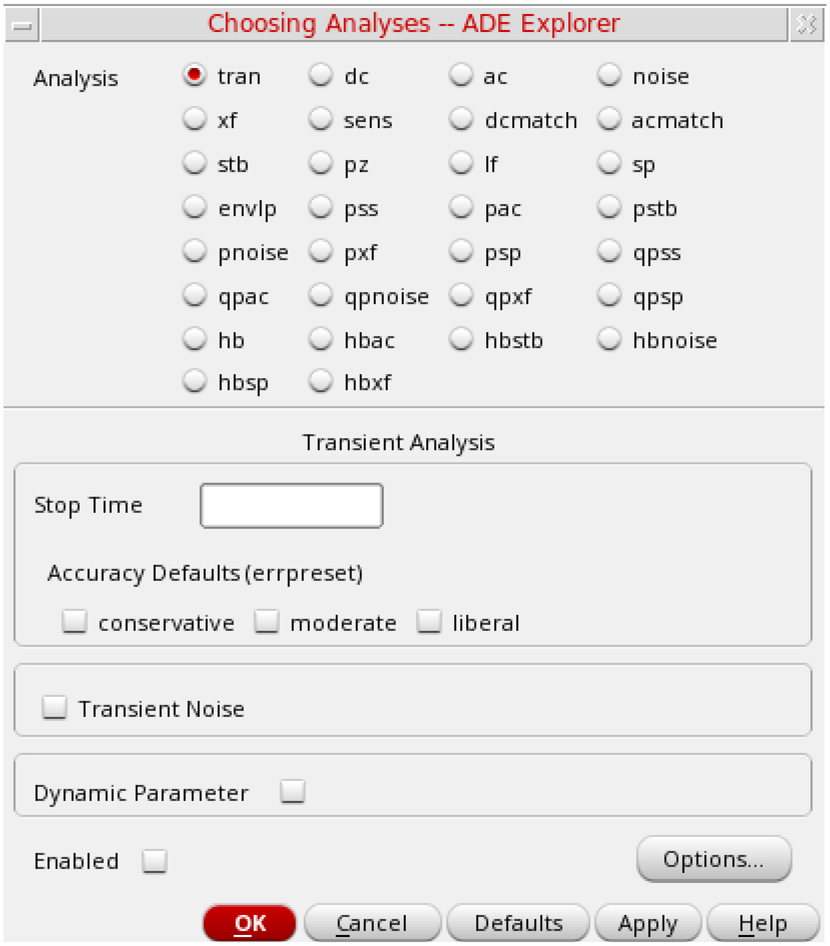
Figure 2-14 Edited Design Variables Section of ADE Explorer Window

The small-signal analysis begins by linearizing the circuit about an operating point. By default, this analysis computes the operating point, if it is not known, or recomputes it if any significant component or circuit parameter has changed.

2. In the ADE Explorer window, select *Analysis - Choose* or click the *Choosing Analyses* icon ( )on the right side of the ADE Explorer window.

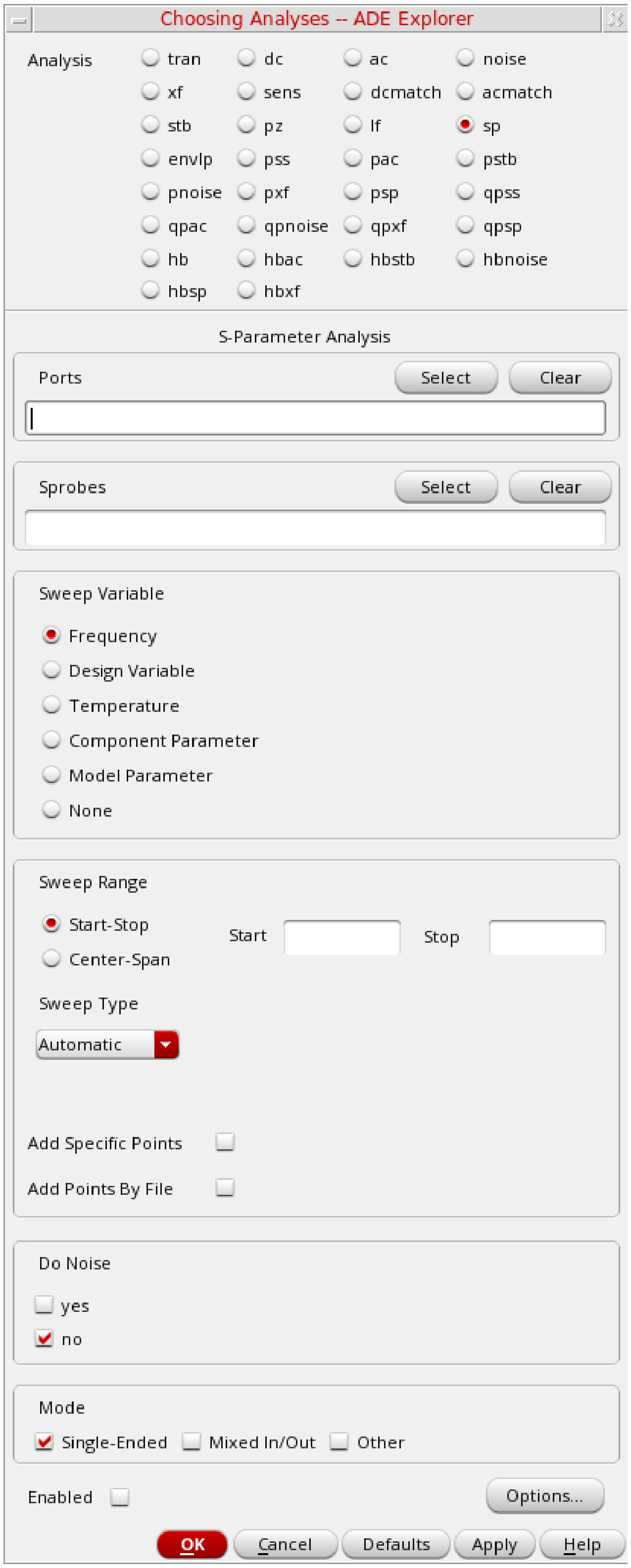
The *Choosing Analyses* form is displayed.

Figure 2-15 Choosing Analyses Form



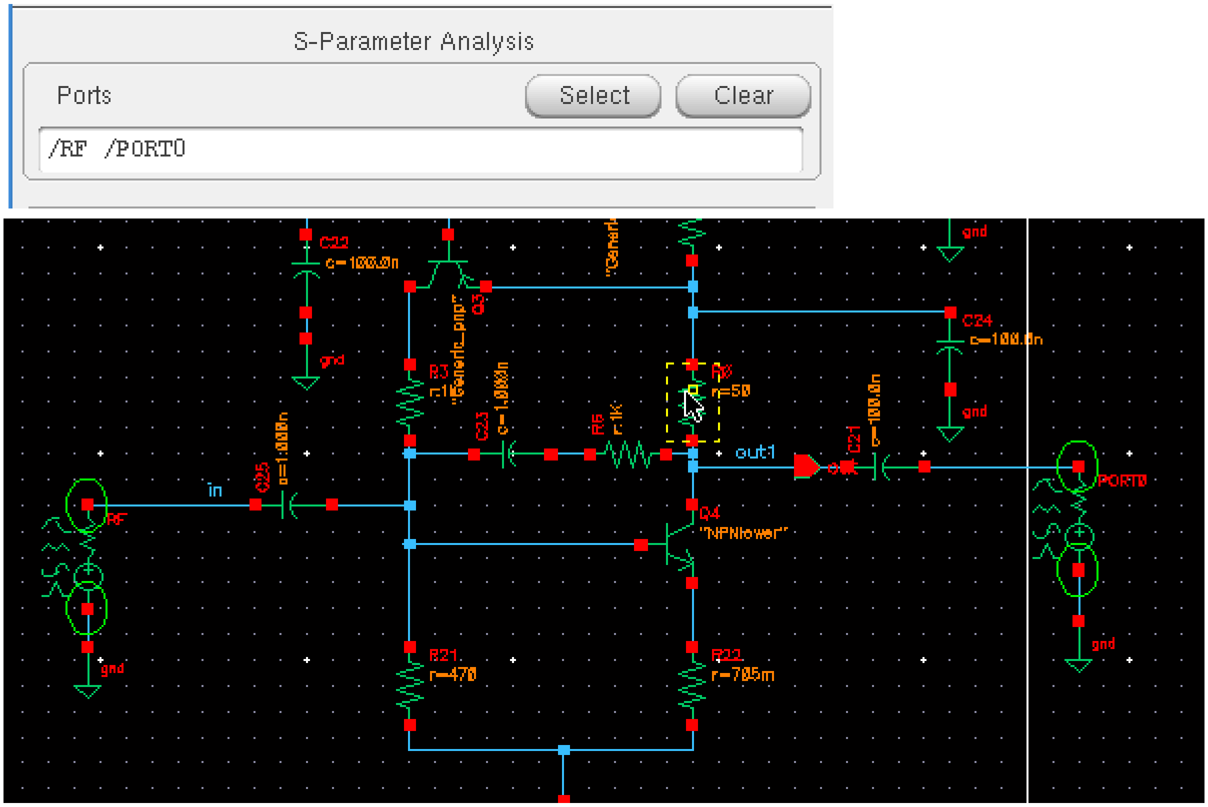
3. In the *Analysis* section, select the *sp* radio button. The form expands.

Figure 2-16 sp Choosing Analyses Form



4. In the *Ports* section, click *Select*. Next, click the input (*RF*) port, followed by the output port (*PORT0*). When finished with the two selections, press the *Esc* key. The form and schematic will look like the following:

Figure 2-17 Choosing Analyses Form with Ports Selected



If the list of active ports is specified with in the Ports field, the ports are numbered sequentially from one in the order given. Otherwise, all ports present in the circuit are active, and the port numbers used are those that were assigned on the port statements in the netlist (or in the *Edit Properties* form).

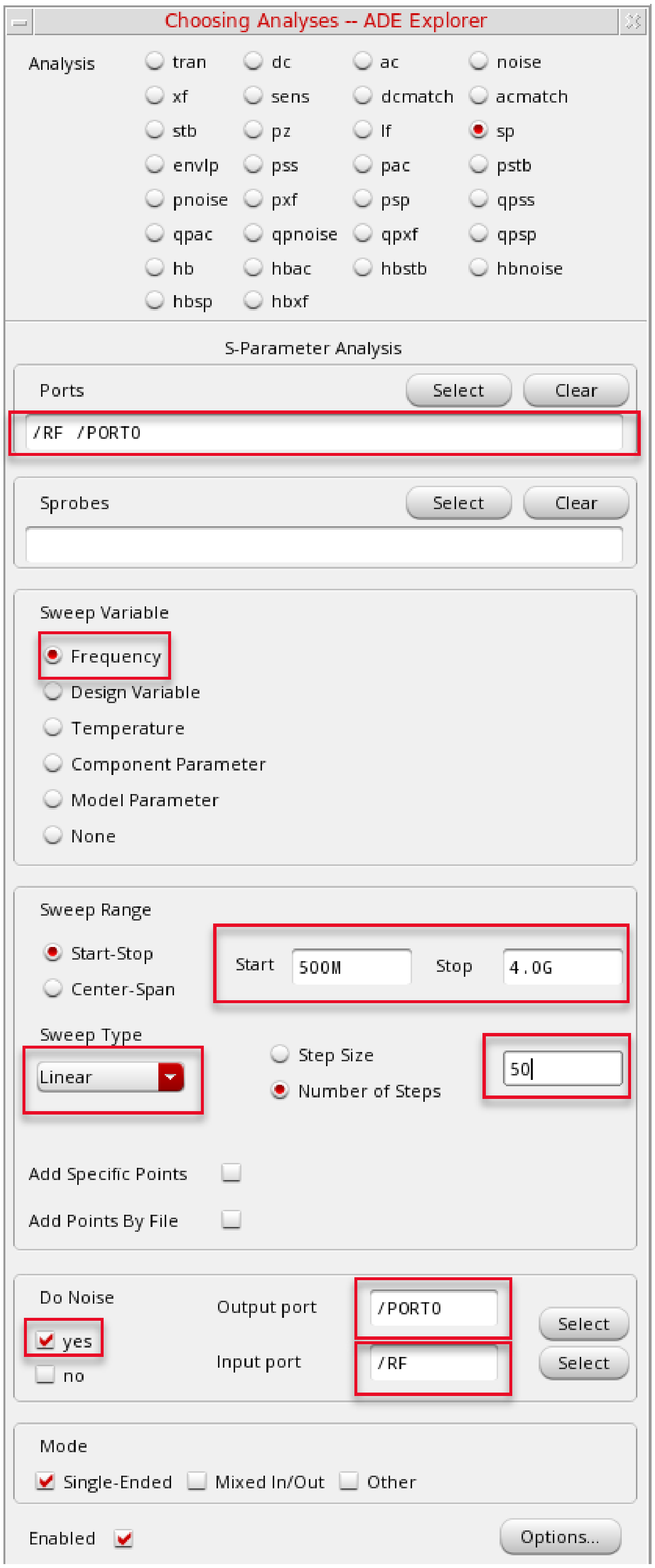
Spectre can perform AC/SP analysis while sweeping a parameter. The parameter can be frequency, temperature, component instance parameter, component model parameter, or netlist parameter. If changing a parameter affects the DC operating point, the operating point is recomputed at each step. After the analysis is complete, the modified parameter returns to its original value.

You can define sweep limits by specifying the end points or the center value and span of the sweep. Steps can be linear or logarithmic, and you can specify the number of steps or the size of each step. If you do not specify a step size parameter, the sweep is linear when the ratio of stop to start values is less than 10 and logarithmic when this ratio is 10 or greater. All frequencies are in Hertz.

5. You will be sweeping frequency in this simulation. In the *Sweep Variable* section of the *Choosing Analyses* form, select *Frequency* (this is the default value).
6. In the *Sweep Range* section, select *Start-Stop*. Enter 500M in the *Start* field and 4.0G in the *Stop* field.
7. Set *Sweep Type* to *Linear*. Select *Number of Steps* and set that to *50*.
8. In the *Do Noise* field, select *yes*. This sets up the small signal (linear 2 port) noise analysis. The small signal assumption is valid when the input power level is low (at least 10dB below the 1dB compression point) and the circuit is operating in the linear range.

   When `donoise=yes` is specified, the noise correlation matrix is computed. If in addition, the output is specified using Output probe (oprobe), the amount that each noise source contributes to the output is computed. Finally, if an input is also specified (iprobe), the two-port noise parameters are computed (F, Fmin, NF, NFmin, Gopt, Bopt, and Rn). When an input port is specified, the two-port noise parameters are computed (F, Fmin, NF, NFmin, Gopt, Bopt, and Rn).
9. Click *Select* to the right of *Output port* and click the output port (PORT0) in the schematic.
10. Click *Select* to the right of *Input port* and click the input port in the schematic (RF). Alternately, you can type `/PORT0` in the *Output port* field and `/RF` in the *Input port* field.
11. In the *Mode* section, select *Single-Ended*. If you are simulating mixed-mode parameters, select the *Mixed In/Out* option. For more information, see the Spectre Circuit Simulator and Accelerated Parallel Simulator User Guide or type `spectre -h sp` at the command prompt.
12. Click *OK* at the bottom of the form. The *Choosing Analyses* form will look like the one below.

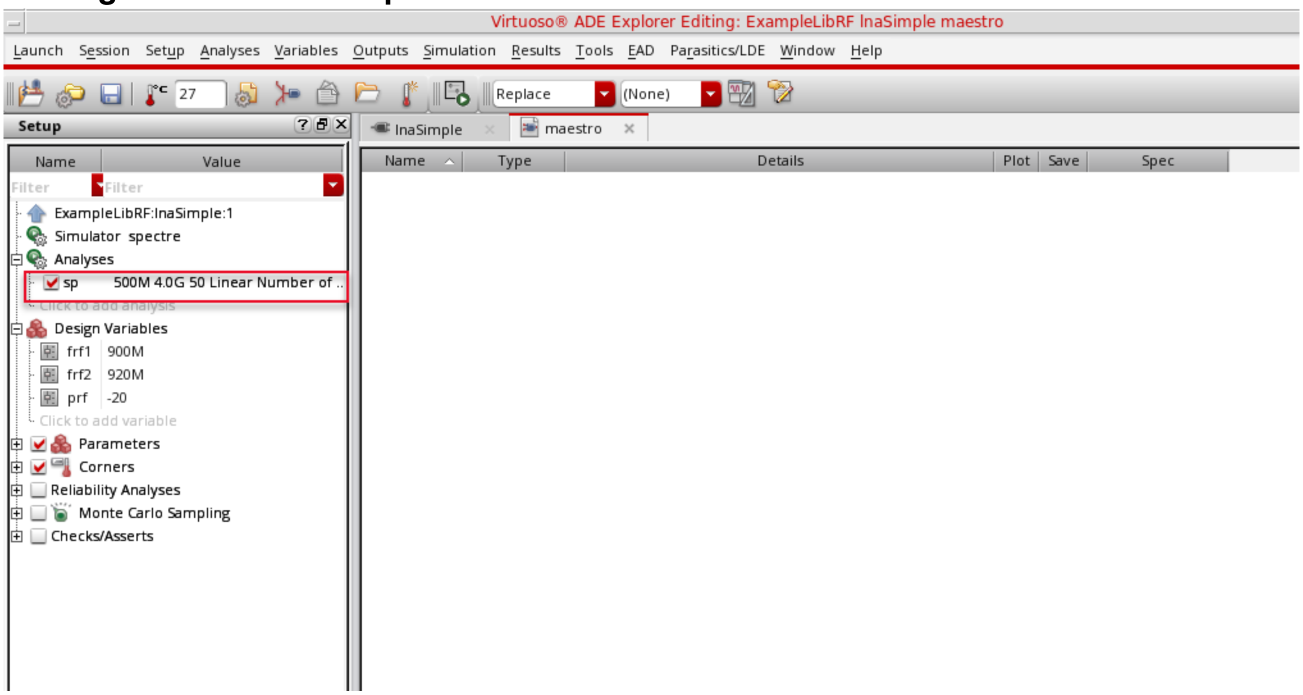
Figure 2-18 sp Choosing Analyses Form



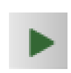
13. Click *OK*.

Your ADE Explorer Setup Assistant will look like the following:

Figure 2-19 ADE Explorer



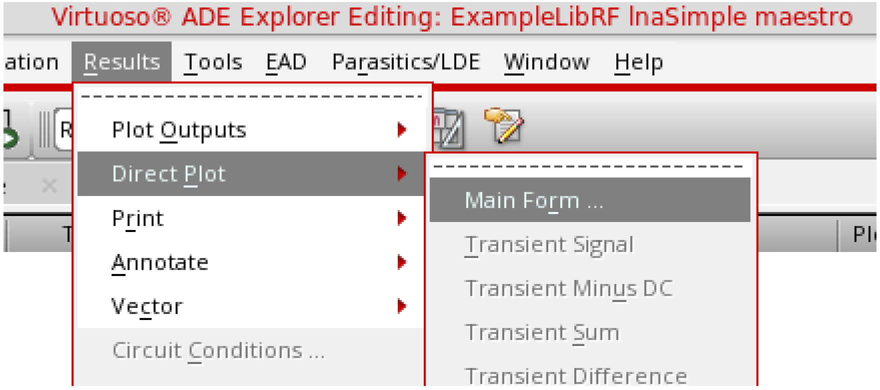
Running the Simulation and Plotting the Results

Start the analyses by clicking the green arrow icon  in ADE Explorer or in the Schematic Editor.

This netlists the design and runs the simulation. A SpectreRF status window is displayed (`spectre.out` log file). When the analysis has completed, you may iconify the status window.

1. In *ADE Explorer*, select *Results - Direct Plot - Main Form*.

Figure 2-20 Invoking the Direct Plot Form



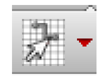
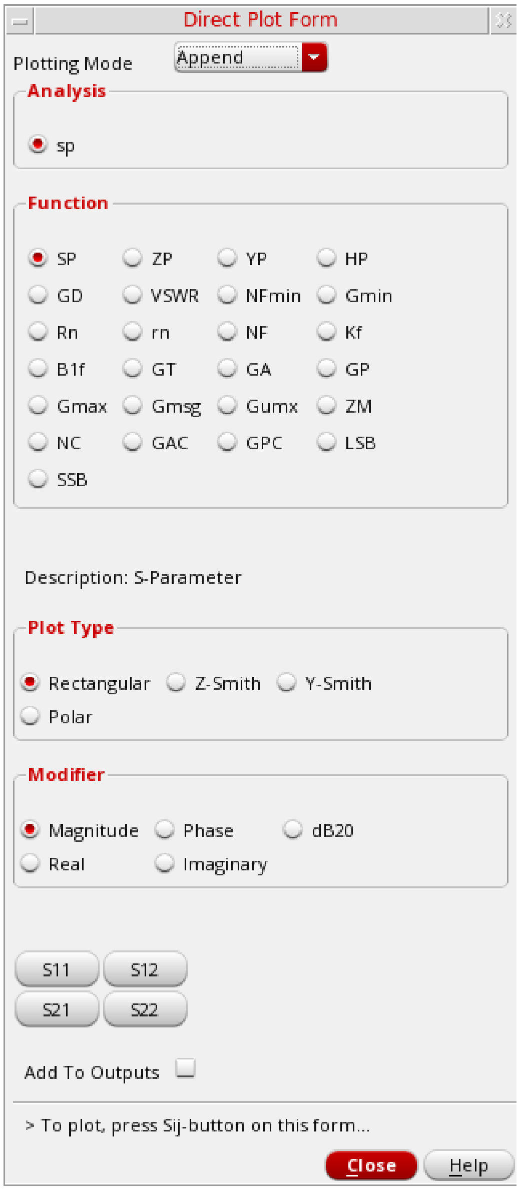
The *Direct Plot Form* is displayed. Alternately, you can click the *Direct Plot* icon  in the Schematic window.

Figure 2-21 sp Direct Plot Form



2. In the *Direct Plot Form*, leave the *Plotting Mode* set to *Append* (this is the default).

3. In the *Function* section, select *GT* (for Transducer Gain). Transducer power gain, *GT*, is defined as the ratio between the power delivered to the load and the power available from the source.

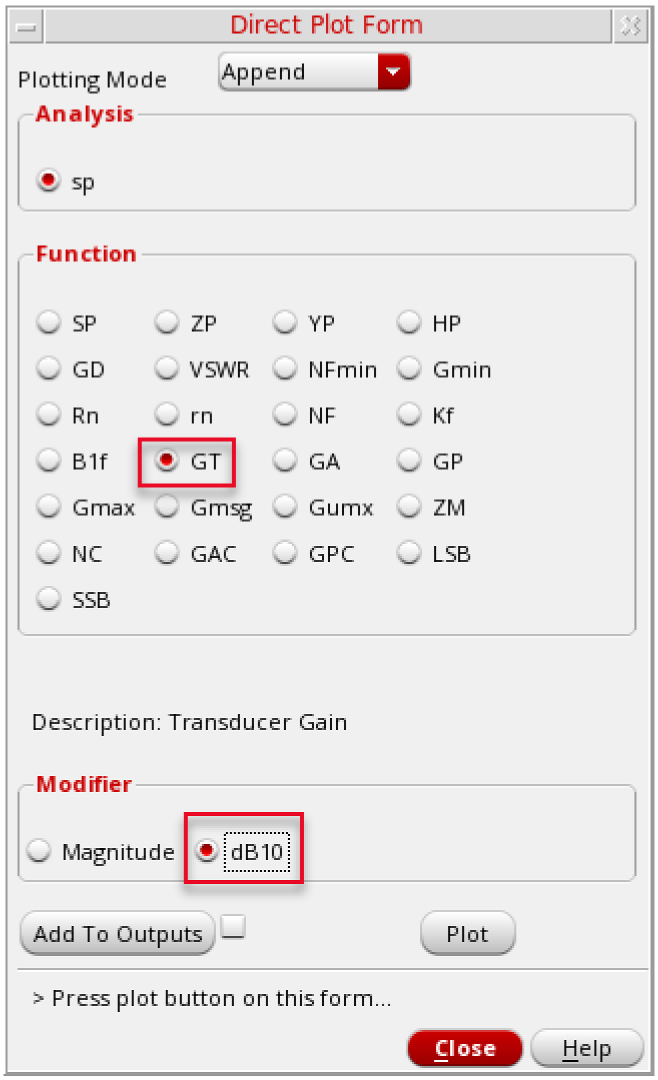
   Note: When using the S-parameter *Direct Plot Form*, the Analog Design Environment assumes that the source (Γ_S) and load (Γ_L) reflection coefficients are zero. *GT*, therefore, plots the insertion gain assuming source and load impedances are matched.

(2-1) Transducer Gain with Source and Load Impedances Matched

$$G_T = |S_{21}|^2$$

4. In the *Modifier* section, select *dB10* because you are plotting power.
5. The sp *Direct Plot Form* will look like the following:

Figure 2-22 sp Direct Plot Form



6. Click *Plot*.
7. In the *Function* section, select *GA* (for Available Power Gain). Available power gain, *GA*, is defined as the ratio between the power available from the network and the power available from the source.

Note: When you use the S-parameter *Direct Plot Form*, Γ_S is set to zero, and therefore available gain (*GA*) is plotted as:

(2-2) Available Gain with T_s=0

$$G_A = \frac{|S_{21}|^2}{1 - |S_{22}|^2}$$

8. Click *Plot* again.
9. In the *Function* section, select *GP* (for Power Gain). Power Gain is defined as the ratio between the power delivered to the load and the power input to the network.

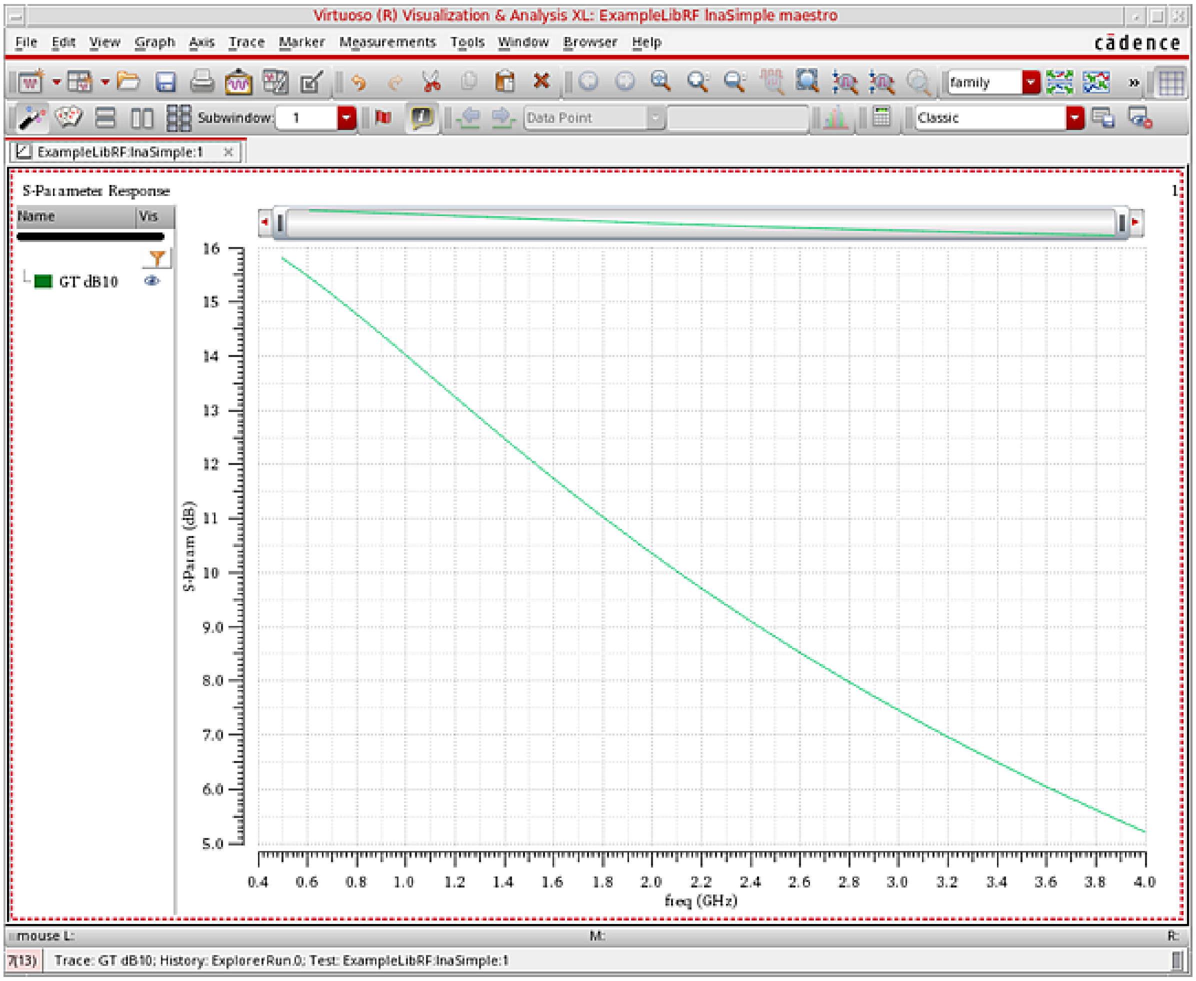
   Note: The ADE Explorer environment assumes that Γ_{in}=0 (the input is matched) so the equation for power gain (*GP*) reduces to the equation below:

(2-3) Power Gain with T_L=0.

$$G_P = \frac{1}{1-|S_{11}|^2}|S_{21}|^2$$

10. Click *Plot* once more. All three gains (*GT, GA, GP*) plots are displayed on one window, as shown below.

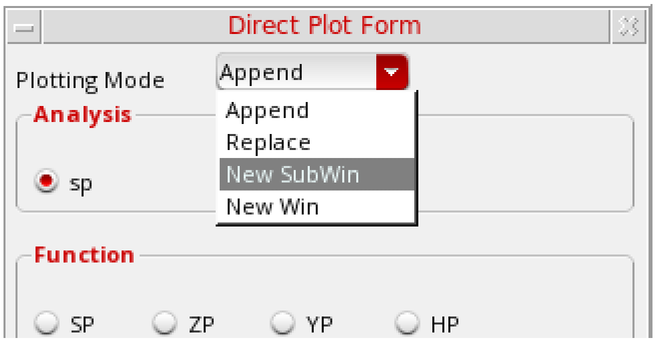
Figure 2-23 Transducer, Available, and Power Gain



Note that *GT* is the smallest gain. Because the power available from the source is greater than the power input to the LNA network, the Power Gain is greater than the Transducer gain (*GP* > *GT*). The closer the two gains are, the better the input matching is. Similarly, because the power available from the LNA network is greater than the power delivered to the load , *GA* > *GP*. The closer these two gains are, the better the output matching is. The power gain *GP* is closer to the transducer gain *GT* than the available gain *GA* which means the input matching network is properly designed. That is, S11 is close to zero.

11. In the *Direct Plot Form*, change the *Plot Mode* to *New SubWin* (new subwindow).

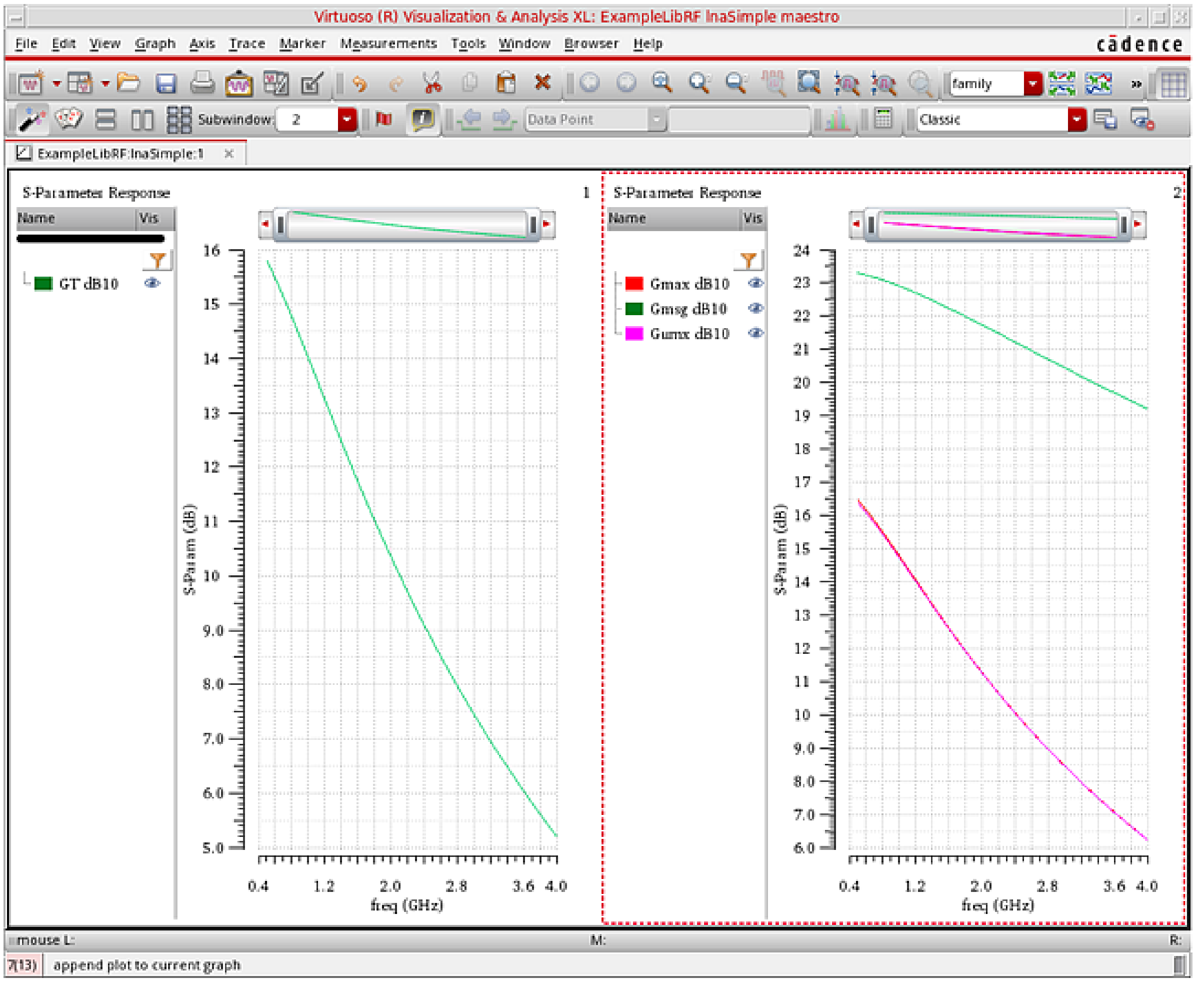
Figure 2-24 Changing Plot Mode to New SubWindow



12. In the *Function* section, select *Gmax* (for maximum Transducer Power Gain) and click *Plot*.
13. Change the *Plotting Mode* to *Append*.
14. In the *Function* section, select *Gmsg* (for Maximum Stability Gain), and Click *Plot*.
15. In the *Function* section, select *Gumx* (for maximum Unilateral Transducer Power Gain), and click *Plot* again.

    The three waveforms are appended to the previous graph, as shown below.

Figure 2-25 Adding Maximum Available, Maximum Stable, and Maximum Unilateral Gain Plots



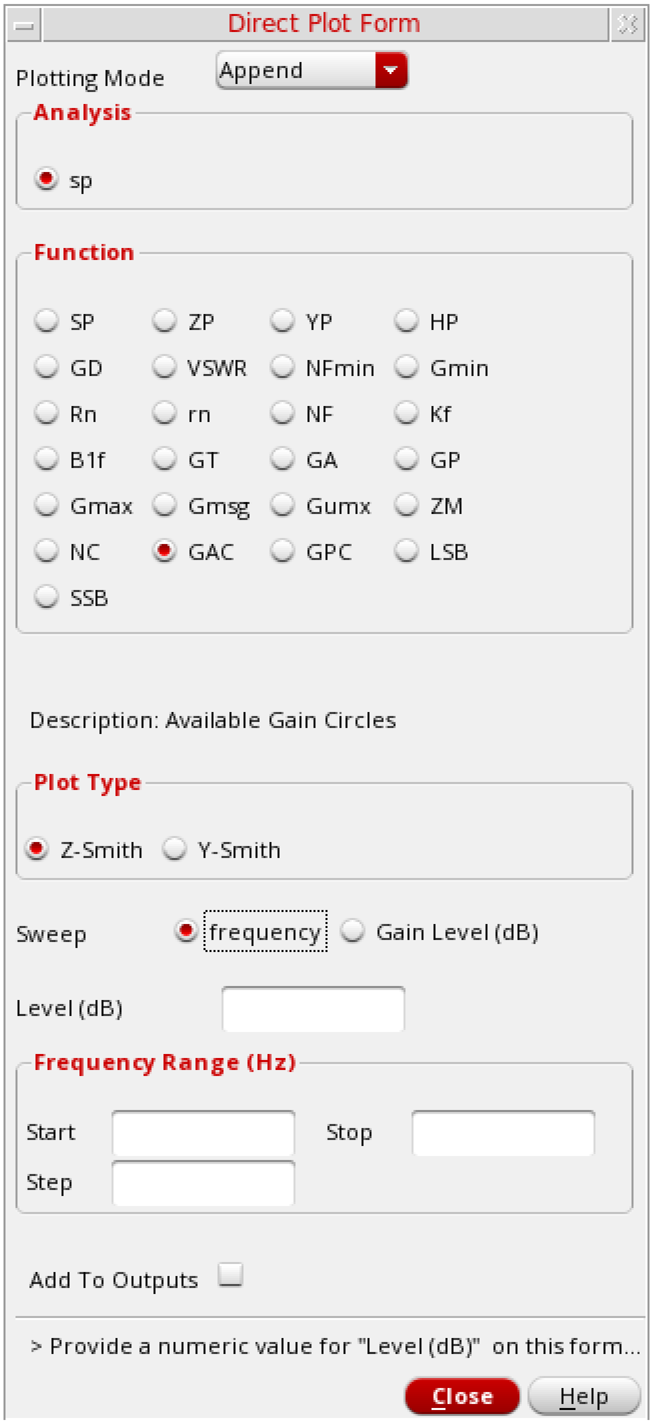
- ❑ Maximum unilateral transducer power gain (*Gumx*) is the transducer power gain when you assume that the reverse coupling of the LNA S_{12}, is zero, and the source and load impedances are conjugately matched to the LNA. That is $S11=\Gamma_s$ and $S22= \Gamma_L$
- ❑ Maximum transducer power gain, *Gmax*, is the simultaneous conjugate matching power gain when both the input and output are conjugately matched.
- ❑ Maximum stable gain, *Gmsg*, is the maximum of *Gmax* when the stability condition, K > 1, is satisfied.

Note: Equations for *Gumx*, *Gmax*, and *Gmsg* and a discussion of the Stability factor *K* are discussed in Chapter 8: AnalogLib Components Used in RF Simulation of the Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide.

In the plots above, the maximum unilateral transducer power gain (*Gumx)* is very close to the maximum transducer power gain (*Gmax)* which means the reverse coupling S12 is small. The maximum stable gain (*Gmsg*) is the largest of the six gains plotted.

1. In the ViVA waveform window, choose *File -Close All Windows* to close the waveform window.
2. Next, you plot the gain circles. There are two types of gain circles: Power Gain Circles and Available Gain Circles. In the *Function* section of the *Direct Plot Form*, select *GAC* (Available Gain Circles). The form expands.

Figure 2-26 Direct Plot Form for Available Gain Circles



Available Gain Circle (GAC)

Available Gain (GA) is solely a function of the source reflection coefficient Γ_S . Thus, you can draw available gain contours on the Smith chart of Γ_S . The location for the peak of the contour corresponds to Γ_S producing the maximum available gain (*GA*). You can move the peak location by changing the design of the input matching network. The best location for the contour peak is at the center of the Smith chart, where Γ_S=0.

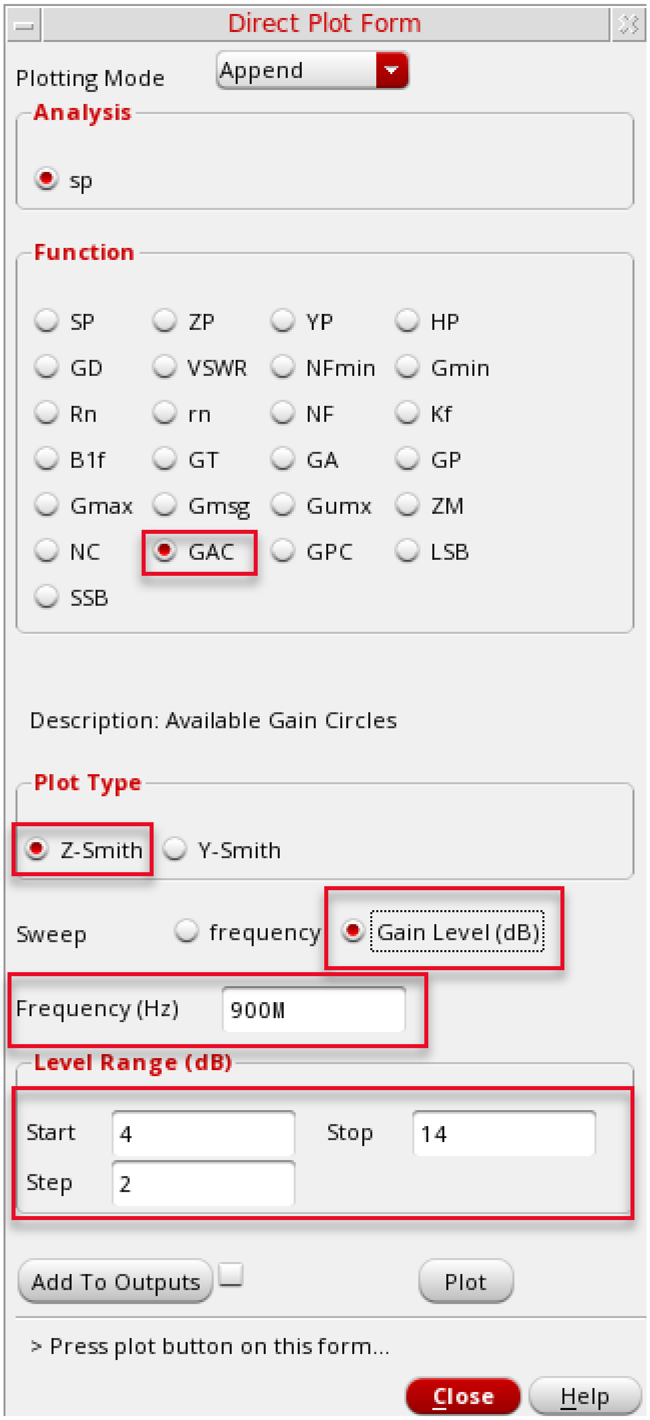
3. In the *Plot Type* section, choose *Z-Smith*. You will be plotting Gain Circles on the Impedance Smith Chart. (*Y-Smith* plots on the Admittance Smith Chart).

4. In the *Sweep* section, you can either choose *Gain Level (dB)* or *Frequency (Hz).* In this case, you will be sweeping Gain Level. The Frequency you specify depends on the operating frequency of your design. Since the lnaSimple circuit operates at 900MHz, enter that value in the *Frequency (Hz)* field.

5. In the *Level Range (dB)* section, set *Start* to 4, *Stop* to 14, and *Step* to 2.

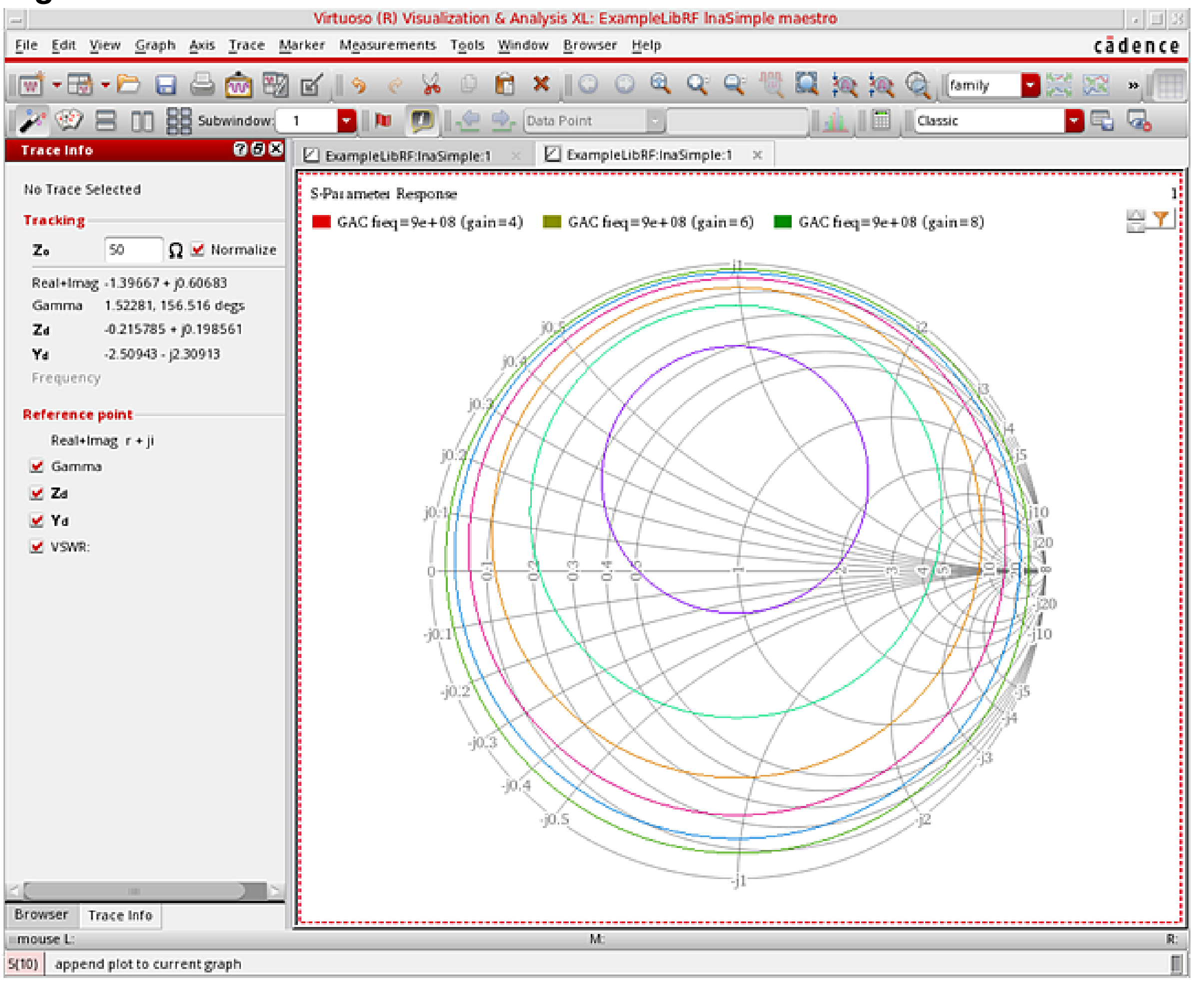
   The *Direct Plot Form* should look like the following:

Figure 2-27 Direct Plot Form for Plotting Available Gain Circles at a Constant Frequency



6. Click *Plot*. The available gain circles are plotted in the waveform window, as shown below.

Figure 2-28 Available Gain Circles on Z-Smith Chart



As you move your cursor around one of the Available Gain Circles, notice that the Tracking Cursor will read out both the Real and Imaginary part or the reflection coefficient directly from the Smith Chart in the *Tracking Info* section on the left side of the Smith Chart. The impedance or admittance at that point is also shown in the *Reference point values* section of the Legend.

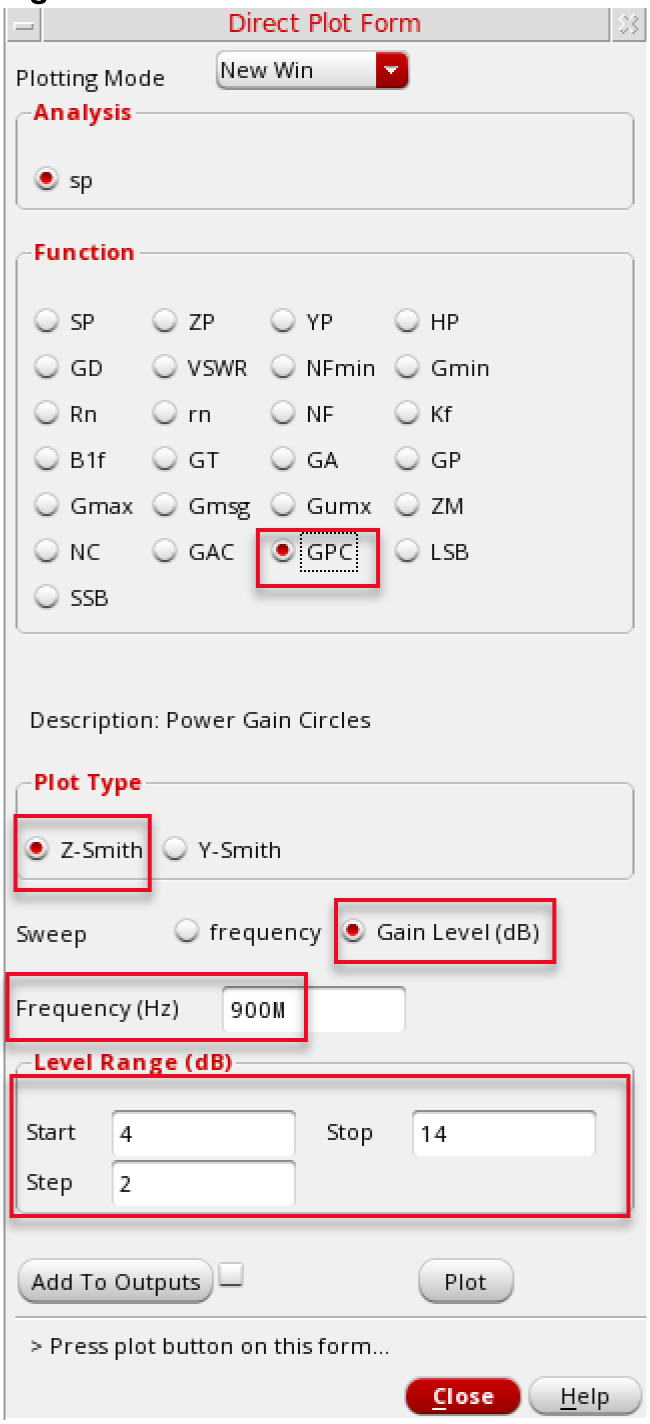
7. In the *Direct Plot Form*, change the *Plot Mode* to *New Window.* Next, plot the Power gain circle (*GPC*).

Power Gain (*GP*) is solely a function of the load reflection coefficient Γ_L. Thus, you can draw the power gain contours on the Smith chart of Γ_L. The location for the peak of the contour corresponds to Γ_L producing the maximum power gain (*GP*). You can move the peak location by changing the design of the output matching network. The best location for the contour peak is at the center of the Smith chart, where $\Gamma_L = 0$.

8. In the sp *Direct Plot Form*, select*GPC* in the *Function* section.

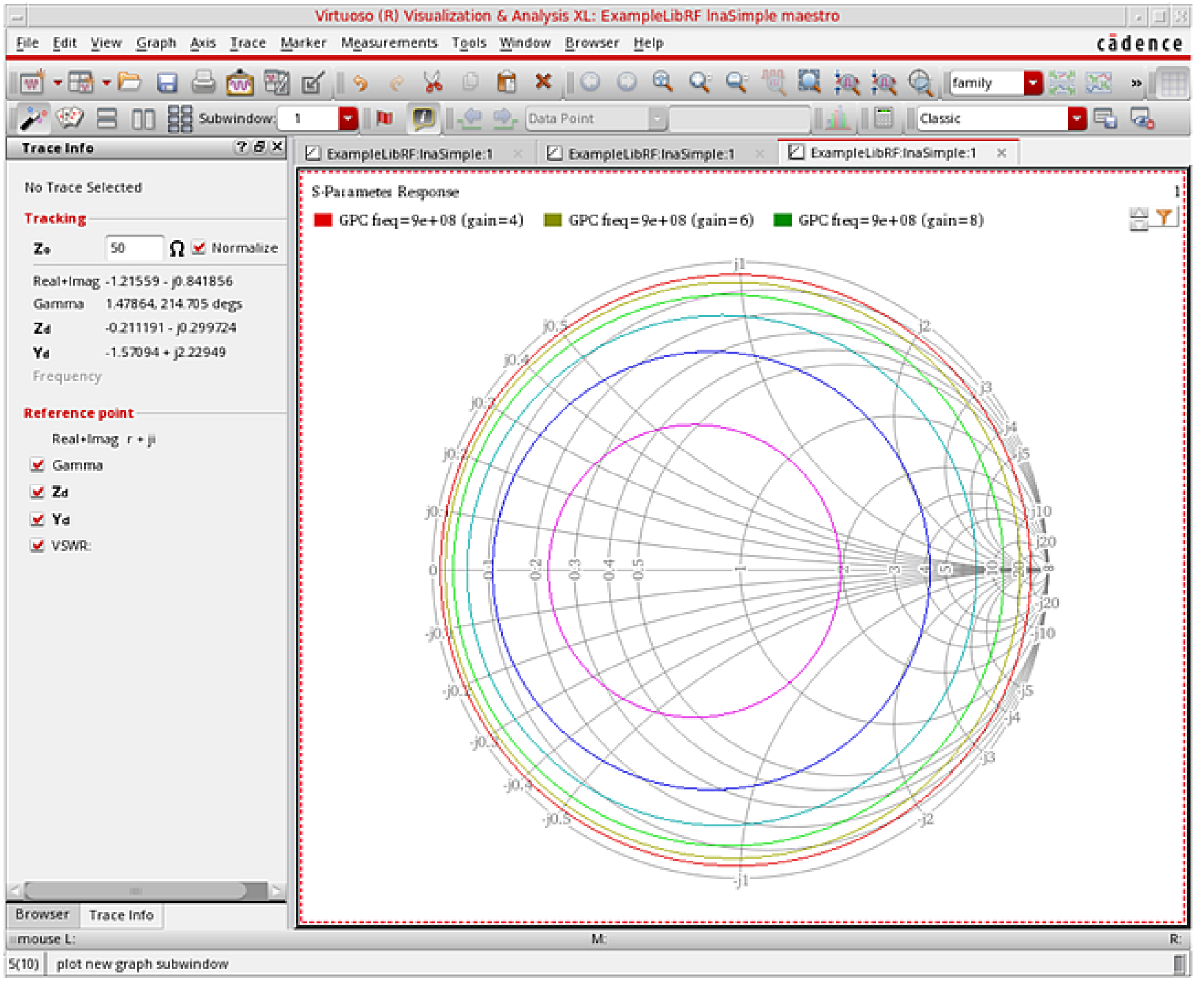
9. In the *Plot Type* section, select *Z-Smith* (it should be the default). You will be plotting Gain Circles on the Impedance Smith Chart.
10. In the *Sweep* section, your choices are *frequency* and *Gain Level (dB)*. Choose *Gain Level (dB)*. Since the lnaSimple operates at 900MHz, enter 900M in the *Frequency (Hz)* field.
11. In the *Gain Level (dB)* section, set *Start* to 4, set *Stop* to 14, and *Step* to 2. The *Direct Plot Form* should look like the following:

Figure 2-29 Direct Plot Form for Power Gain Circles



12. Click *Plot*. The power gain circles are plotted on the Smith Chart, as shown below.

Figure 2-30 Power Gain Circles



Next, you will look at circuit stability and plot Stability Circles.

Stability

After running an sp analysis, you can plot stability factor and stability circles.

Kf, the stability factor, is valid for two-port circuits only. Kf is defined as:

Figure 2-31 Equation for Kf, stability factor

$$K_f = \frac{1 - |S_{11}|^2 - |S_{22}|^2 + |D|^2}{2|S_{21}||S_{12}|}$$

-

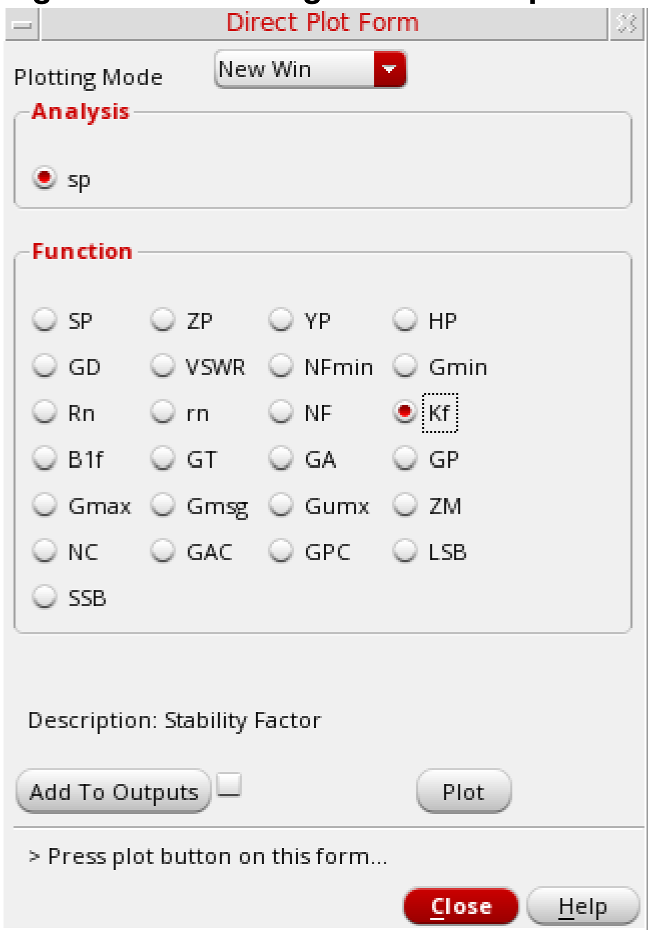
$$D = S_{11}S_{22} - S_{21}S_{12}$$

The above equation is valid for small-signal stability only. Under large signal conditions, the circuit is less likely to be stable. In the presence of feedback paths from the output to the input, the circuit might become unstable for certain combinations of source and load impedances. An LNA design that is normally stable might oscillate at the extremes of the manufacturing or voltage variations, and perhaps at unexpectedly high or low frequencies. When K > 1 and D < 1, the circuit is unconditionally stable. That is, the circuit does not oscillate with any combination of source and load impedances. You should perform the stability evaluation for the S parameters over a wide frequency range to ensure that K remains greater than one at all frequencies.

1. In the *Function* section of the sp Direct Plot form, select *Kf*.
2. Set the *Plotting Mode* to *New Window*.

   The sp *Direct Plot Form* should look like the following:

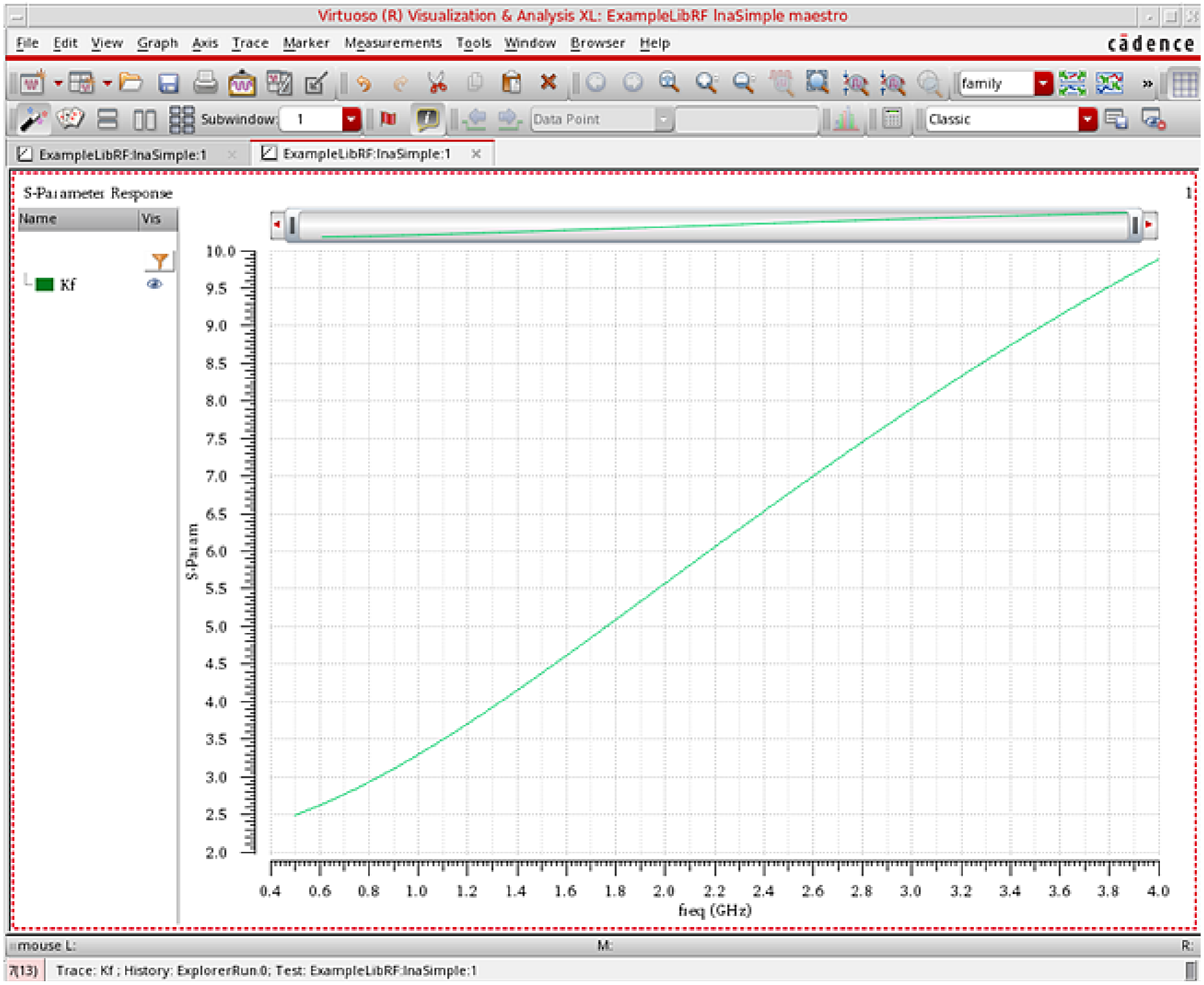
Figure 2-32 Plotting Kf from the sp Direct Plot Form



3. Click *Plot*.

   The waveform window is displayed, as shown below.

Figure 2-33 Plotting Stability Factor K_f vs Input Frequency



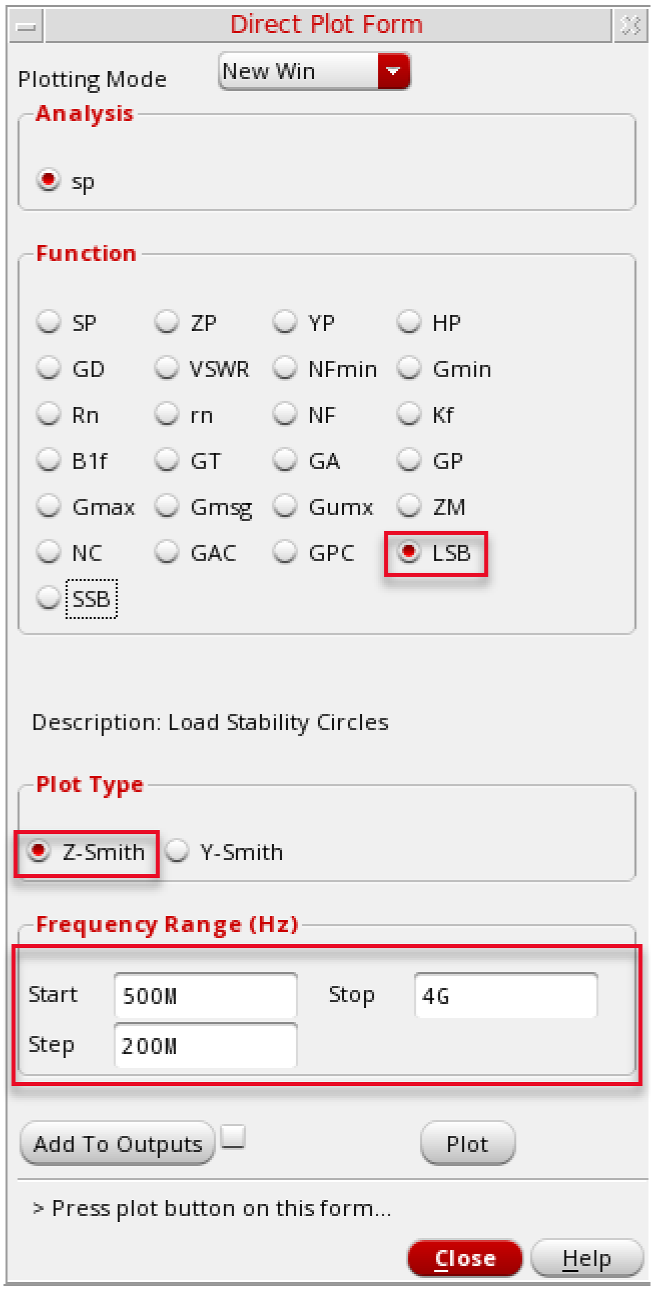
The stability factor *Kf i*s greater than 1 for all frequencies viewed, indicating that the circuit is stable at these frequencies. As the coupling (S12) decreases (reverse isolation increases), stability improves. You might use techniques, such as resistive loading and neutralization to improve stability for an LNA.

Next, plot the Stability Circles.

4. Close the waveform window by choosing *File - Close All Windows* and go back to the *Direct Plot Form*.
5. In the *Function* section, select *LSB* (Load Stability Circles). The form changes.
6. In the *Plot Type* section, choose *Z-Smith* (this is the default). You will be plotting Load Stability Circles (*LSB*) on the Impedance Smith Chart. (*Y-Smith* plots on the Admittance Smith Chart).

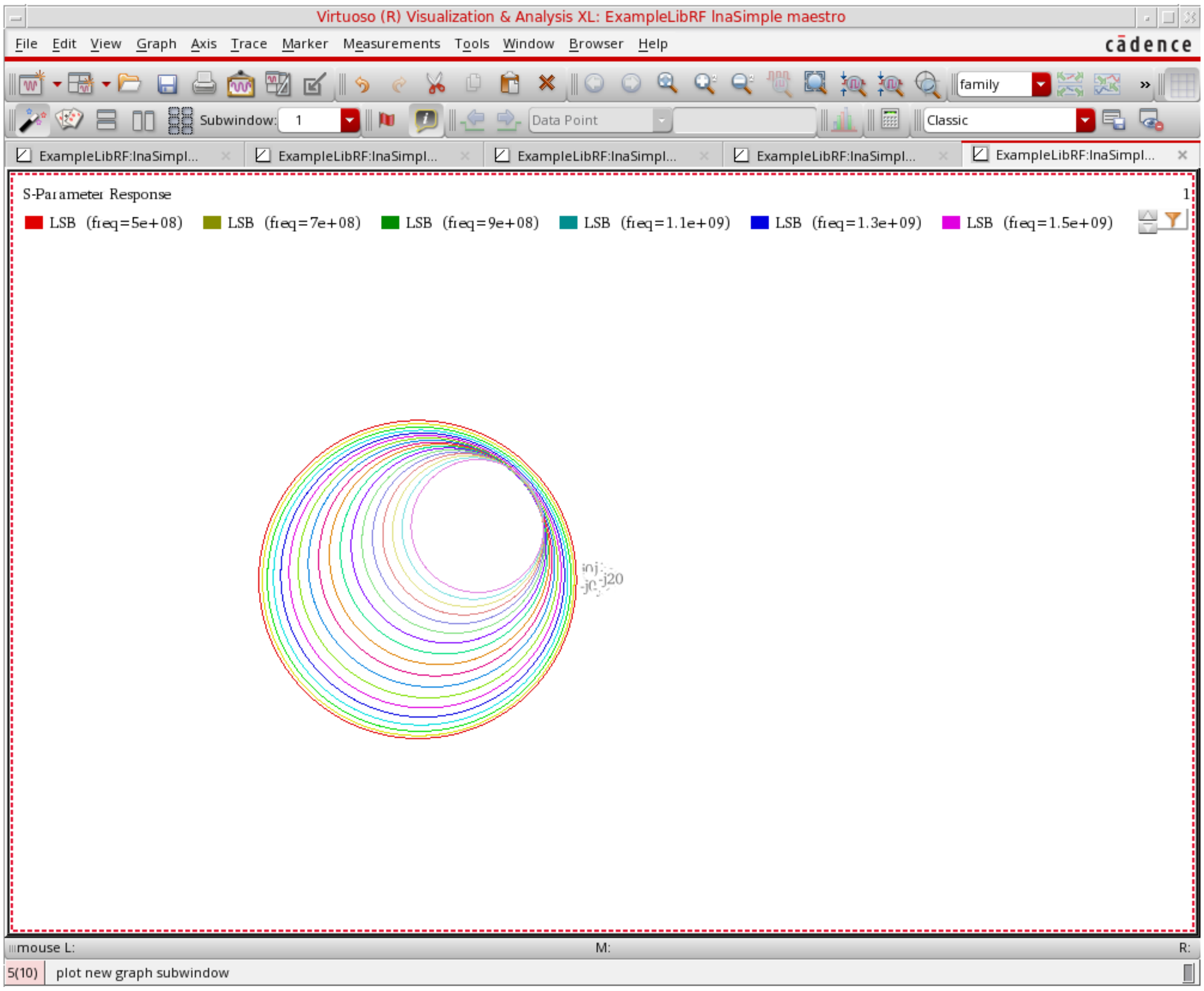
7. In the *Frequency Range (Hz)* section, enter *Start* 500M, *Stop* 4G, and *Step* 200M. The *Direct Plot* form should look like the following:

Figure 2-34 Direct Plot Form for Plotting Load Stability Circles



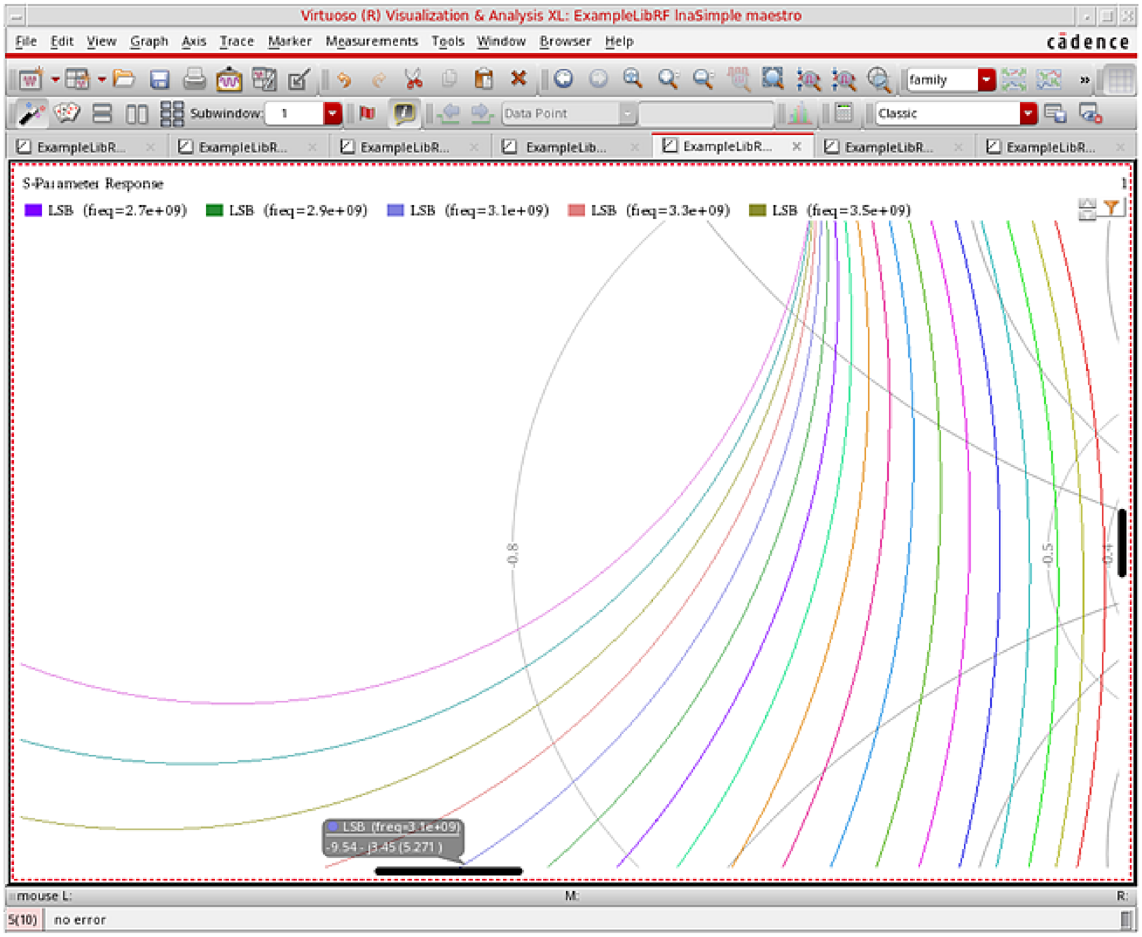
8. Click *Plot*. The Load Stability Circles are plotted, as shown below.

Figure 2-35 Load Stability Circles



By default, the graph is zoomed out to show all traces. Zoom into the Smith Chart by holding down the right mouse button and dragging a square around the section of the Smith Chart you would like to view. When you release the button, the graph redraws. You can also determine which trace belongs to which frequency by clicking on the *+* button to the left of *LSB* in the upper left section of the graph legend. This is shown in the next figure.

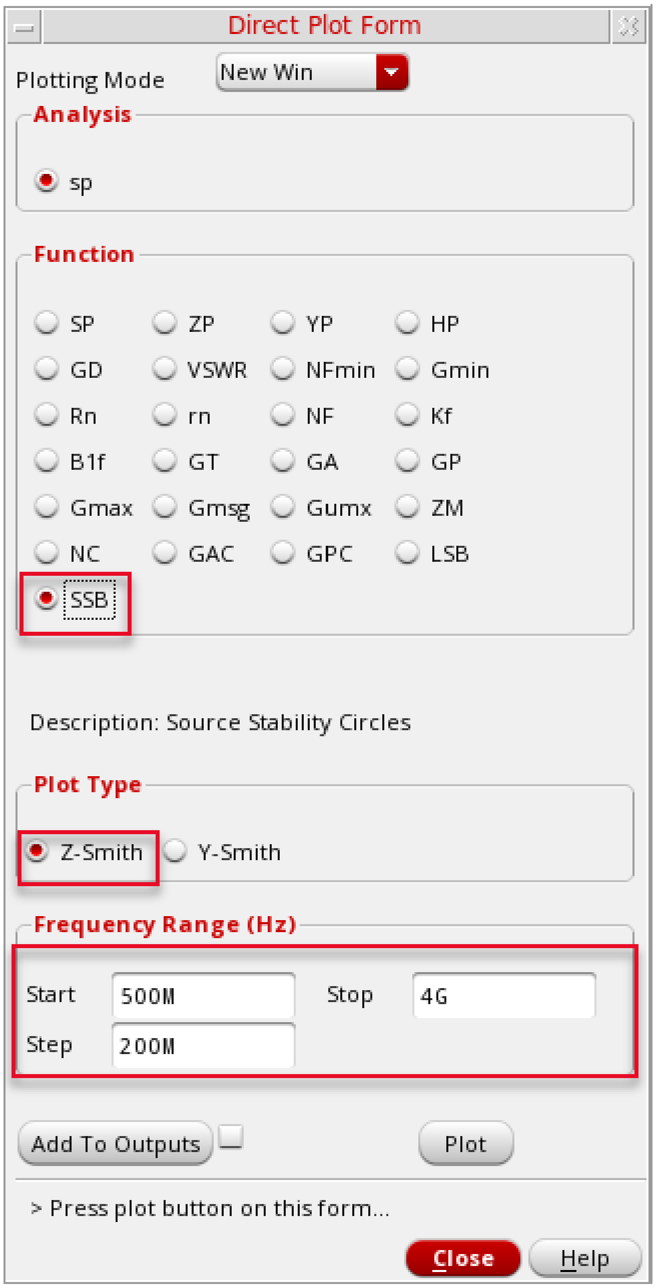
Figure 2-36 Zoomed in Load Stability Circles



Next, you will be plotting the Source Stability Circles.

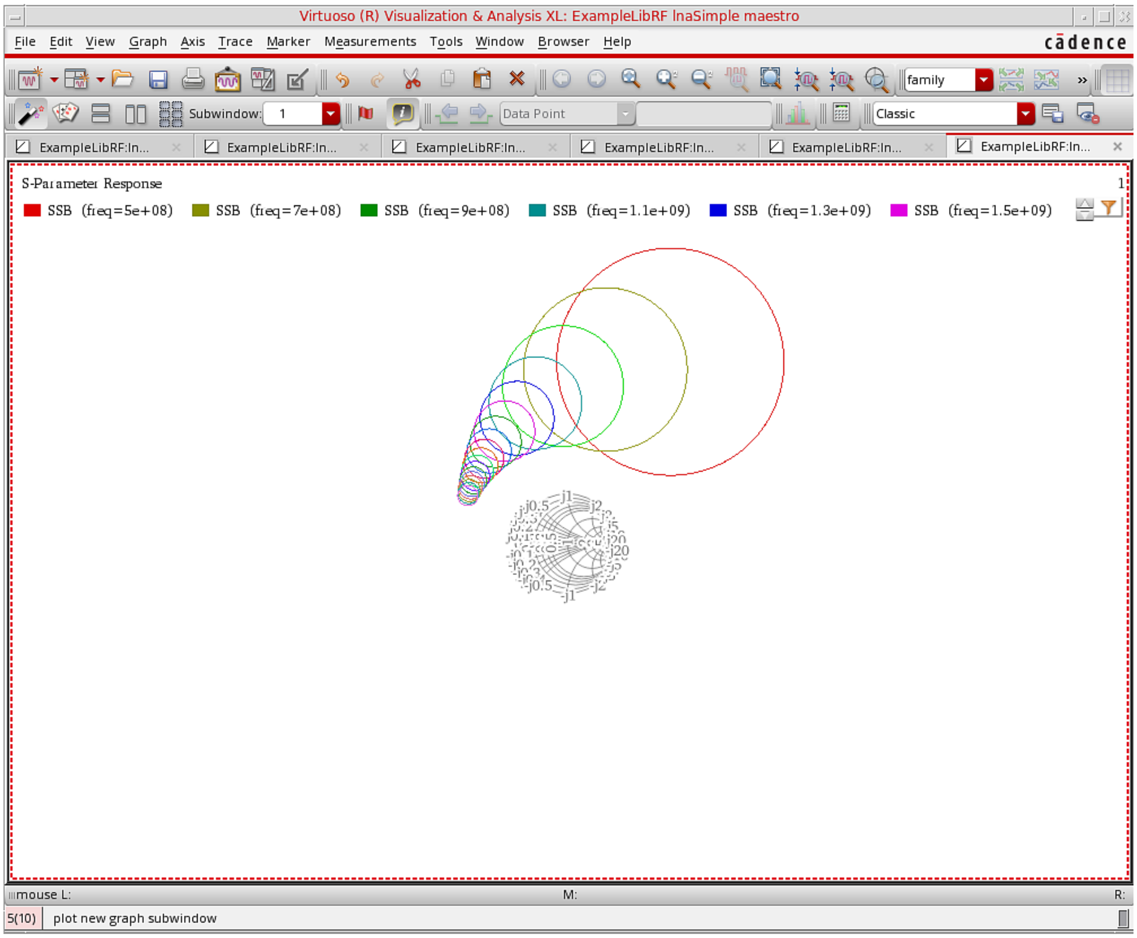
9. In the *Function* section, select *SSB* (Source Stability Circles). The form changes.
10. In the *Plot Type* section, choose *Z-Smith*. You will be plotting Source Stability Circles on the Impedance Smith Chart.
11. In the *Frequency Range (Hz)* section, enter *Start* 500M, *Stop* 4G, and *Step* 200M. The *Direct Plot Form* should look like the following:

Figure 2-37 Direct Plot Form for Source Stability Circles



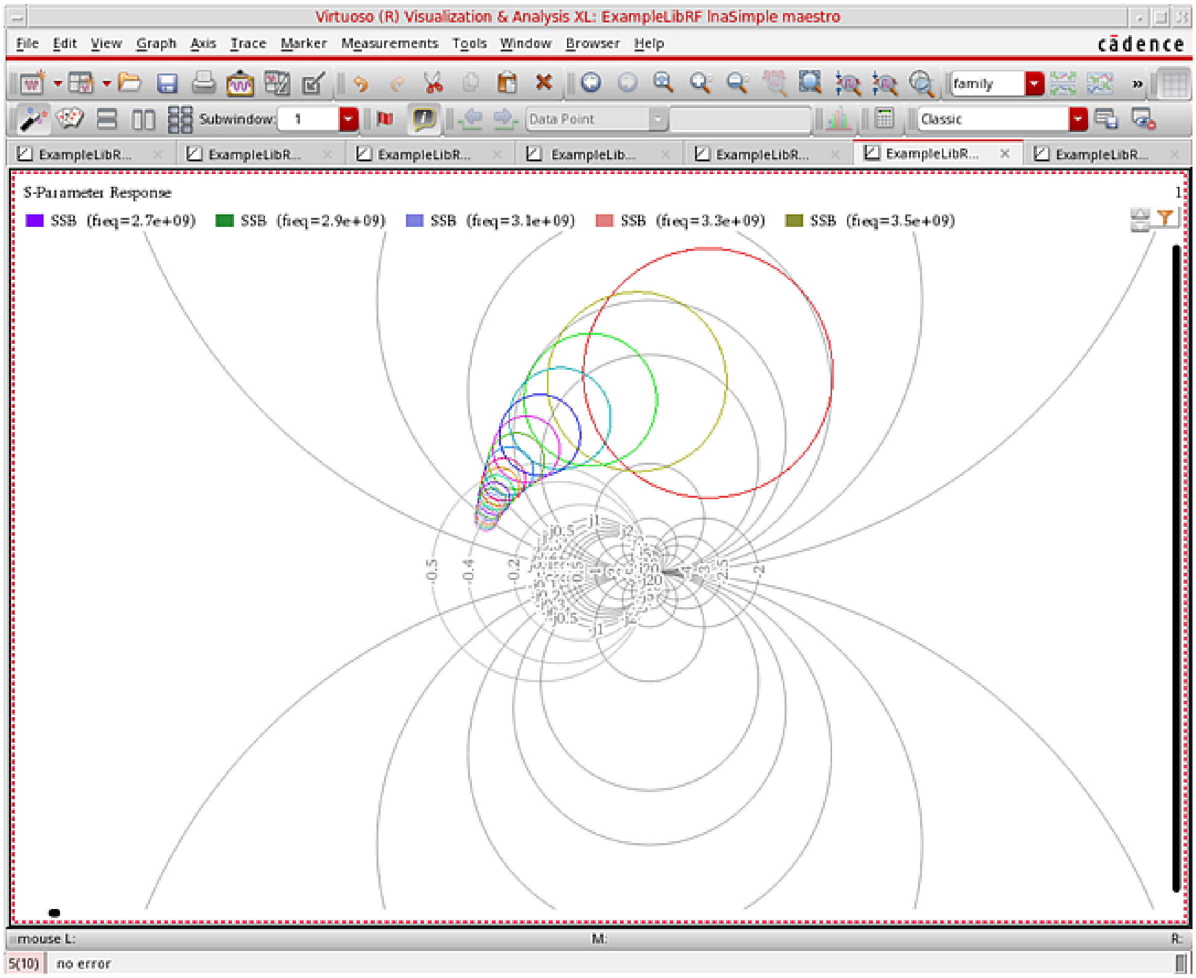
12. Click Plot. The Source Stability Circles are plotted, as shown below.

Figure 2-38 Source Stability Circles for lnaSimple



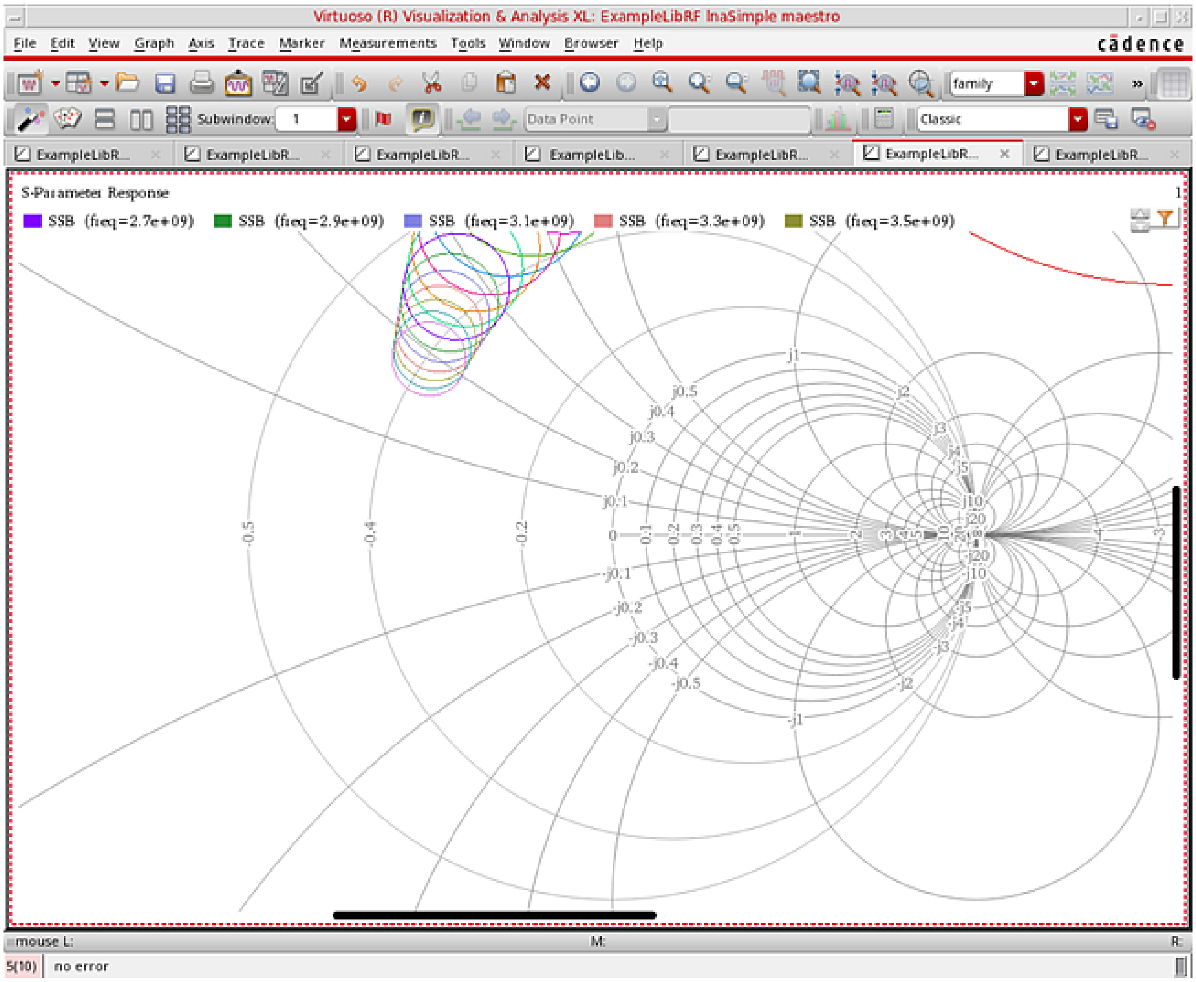
By default, the graph is zoomed out to show all traces. Zoom into the Smith Chart by holding down the right mouse button and dragging a square around the section of the Smith Chart you would like to view. When you release the button, the graph redraws. You can also determine which trace belongs to which frequency by clicking on the + button to the left of *SSB* in the upper left section of the graph legend. This is shown in the next figure.

Figure 2-39 Zoom in Source Stability Circles



When you release the mouse button, the plot redraws, as shown in the next figure.

Figure 2-40 Zoomed in View of Source Stability Circles



The source and load stability circles are also useful for checking for LNA stability. The input stability circle draws the circle $\Gamma_{out} = 1$ out on the Smith chart of Γ_S . The output stability circle draws the circle $\Gamma_{in} = 1$ on the Smith chart of Γ_L.

The non-stable regions of the two circles should be far away from the center of the Smith chart. In fact, it is better if the non-stable regions are located outside the Smith chart circles. This is the case for both Load and Source Stability circles.

13. In the ViVA waveform window, select *File - Close All Windows*. The waveform window closes.

The next measurement you will make is Noise Figure.

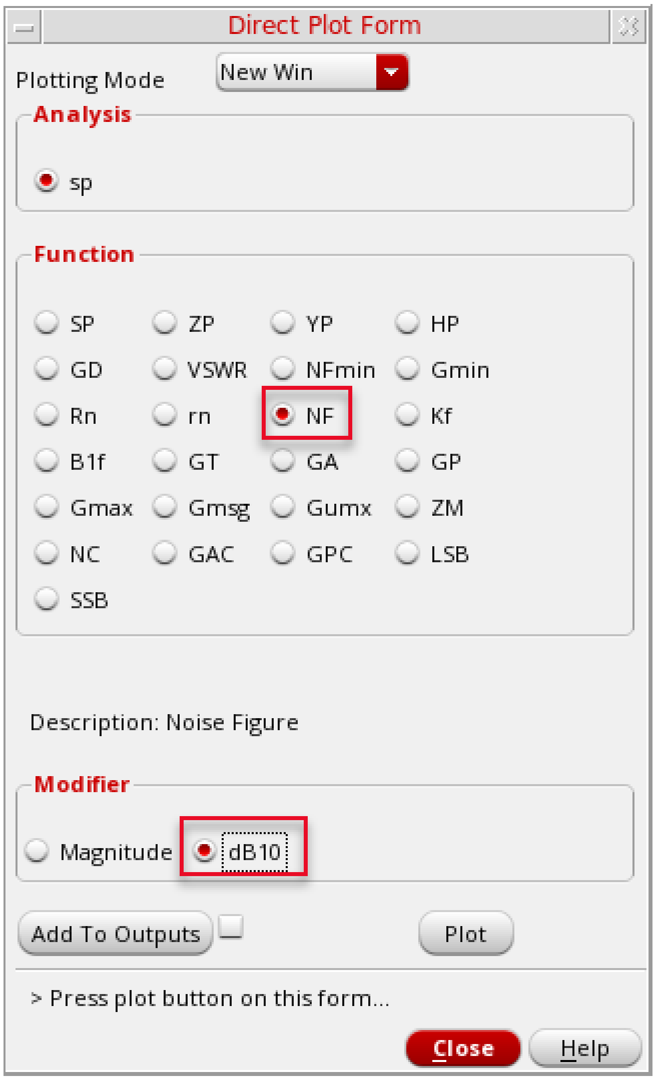
Linear 2-port noise analysis (NF, NFmin) and Noise circles

For cascaded stages, the overall noise figure is mainly determined by the first amplification stage, provided that it has sufficient gain. You achieve low noise performance by carefully selecting the low noise transistor, DC biasing point, and noise-matching at the input. The noise performance is characterized by noise factor, F, which is defined as the ratio between the input signal-to-noise ratio and the output signal-to-noise ratio. For equations to Noise Figure and minimum Noise Figure, see the Noise Calculations in the Simulator section in the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide.

You have already run the simulation for linear two-port noise as part of the sp analysis. Now, you will plot the results.

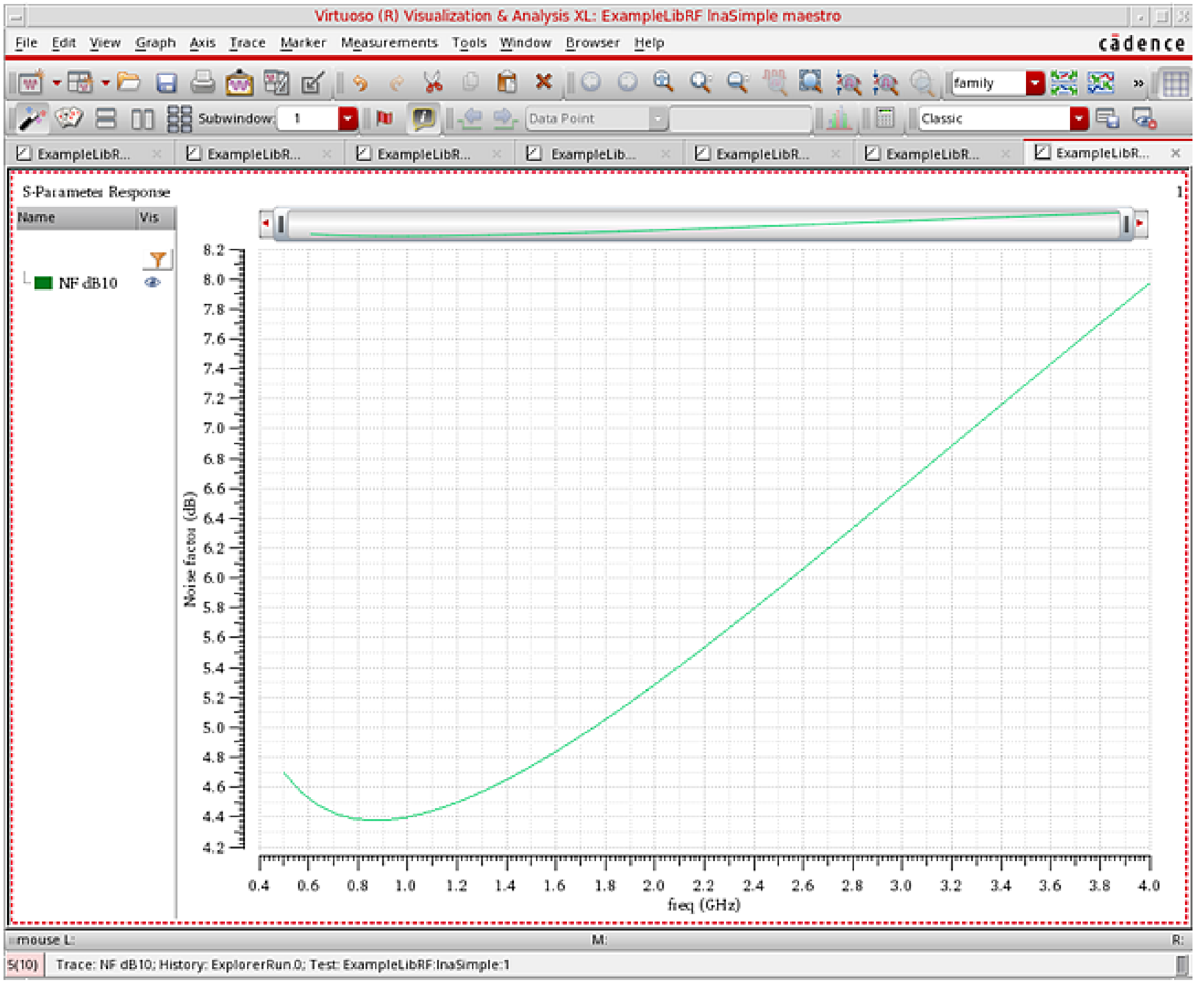
1. In the sp *Direct Plot Form,* select *NF* in the *Function* section. The form changes.
2. In the *Modifier* section, select *dB10* to plot Noise Figure. (If you want to plot Noise Factor instead, select *Magnitude*). The sp *Direct Plot Form* should look like the figure below.

Figure 2-41 sp Direct Plot Form for Plotting Noise Figure



3. Click *Plot*. The noise figure is plotted, as shown below.

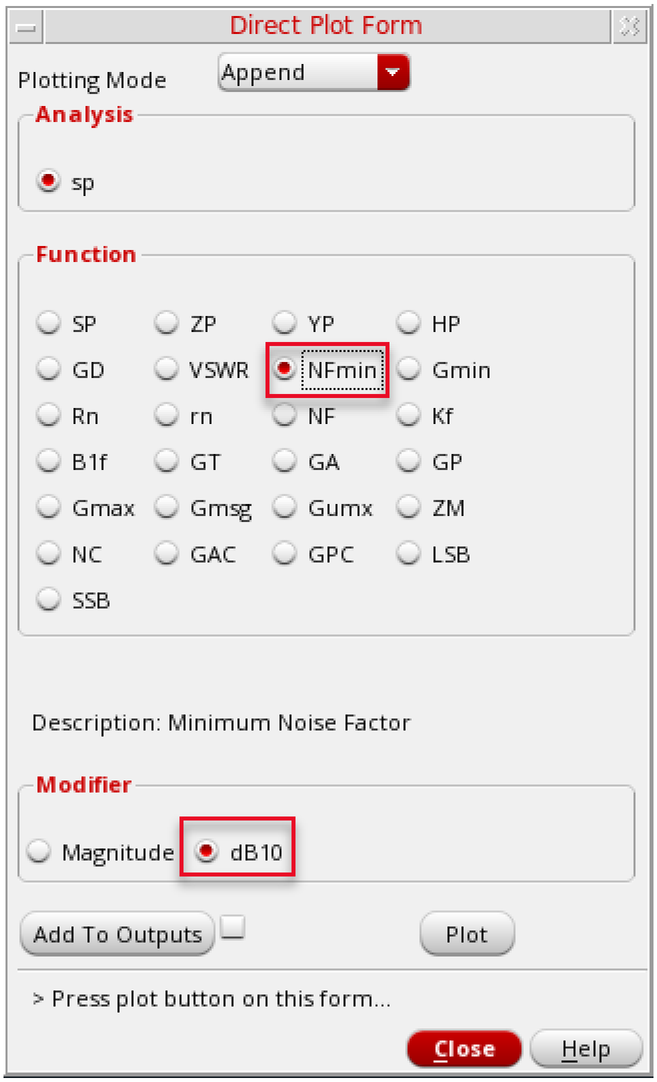
Figure 2-42 lnaSimple Noise Figure Plot



4. In the *Direct Plot Form*, change the *Plotting Mode* to *Append*.
5. In the *Function* section, select *NFmin*. You will be plotting minimum noise figure.
6. Set the *Modifier* to *dB10* to plot minimum noise figure. (Leave the modifier at the default value of magnitude to plot minimum noise factor).

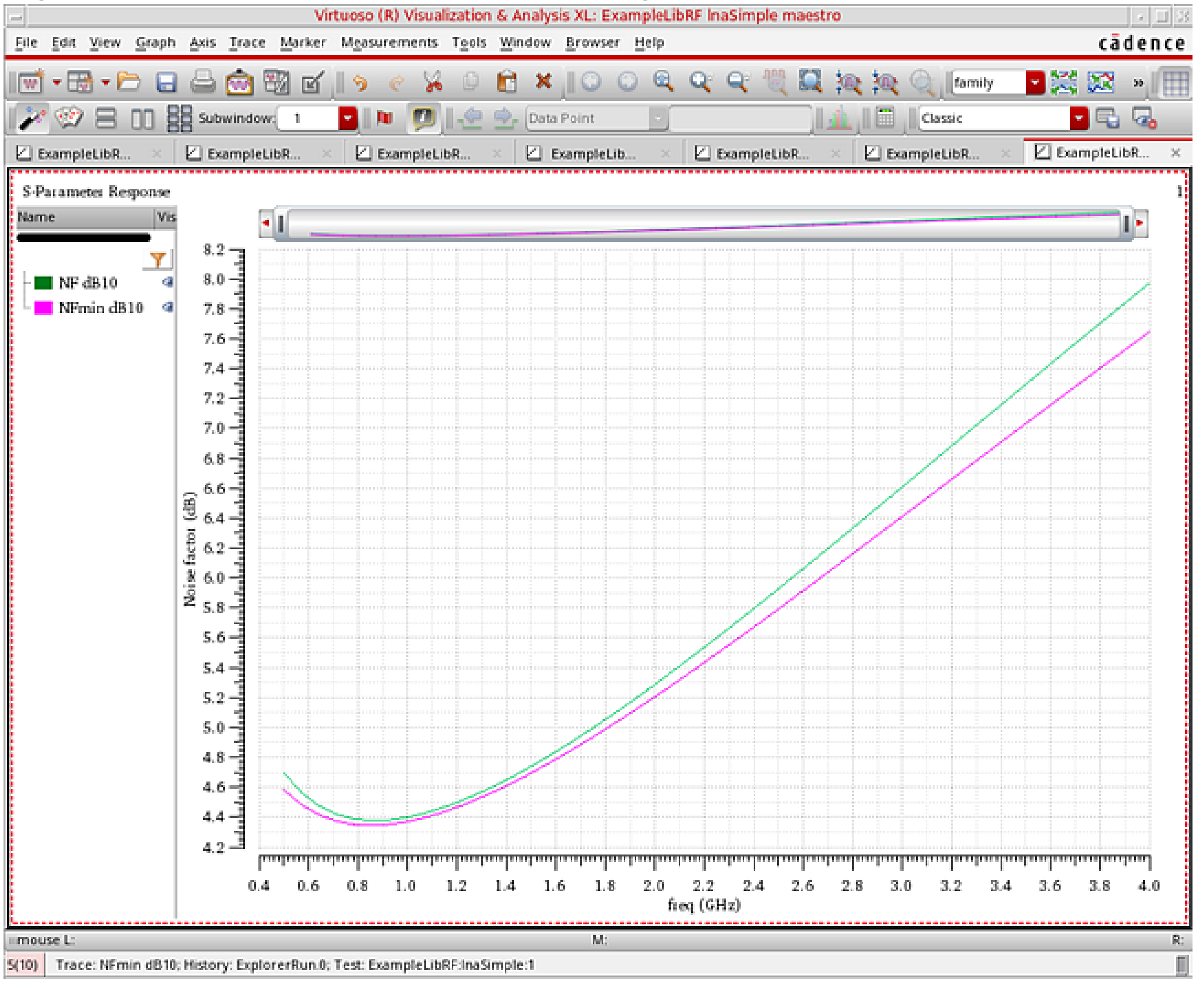
   The *Direct Plot Form* should look like the following figure:

Figure 2-43 sp Direct Plot Form for Minimum Noise Figure



7. Click *Plot* to plot minimum noise figure. The noise figure is plotted, as shown below.

Figure 2-44 lnaSimple Minimum Noise Figure



Note: The y-axis label for Noise Figure and Minimum Noise Figure both show “Noise Factor (dB)”. View the legend at the upper left side of the plot. You will see NF dB10 and NFmin dB10. This shows that noise figure (rather than noise factor) is being plotted.

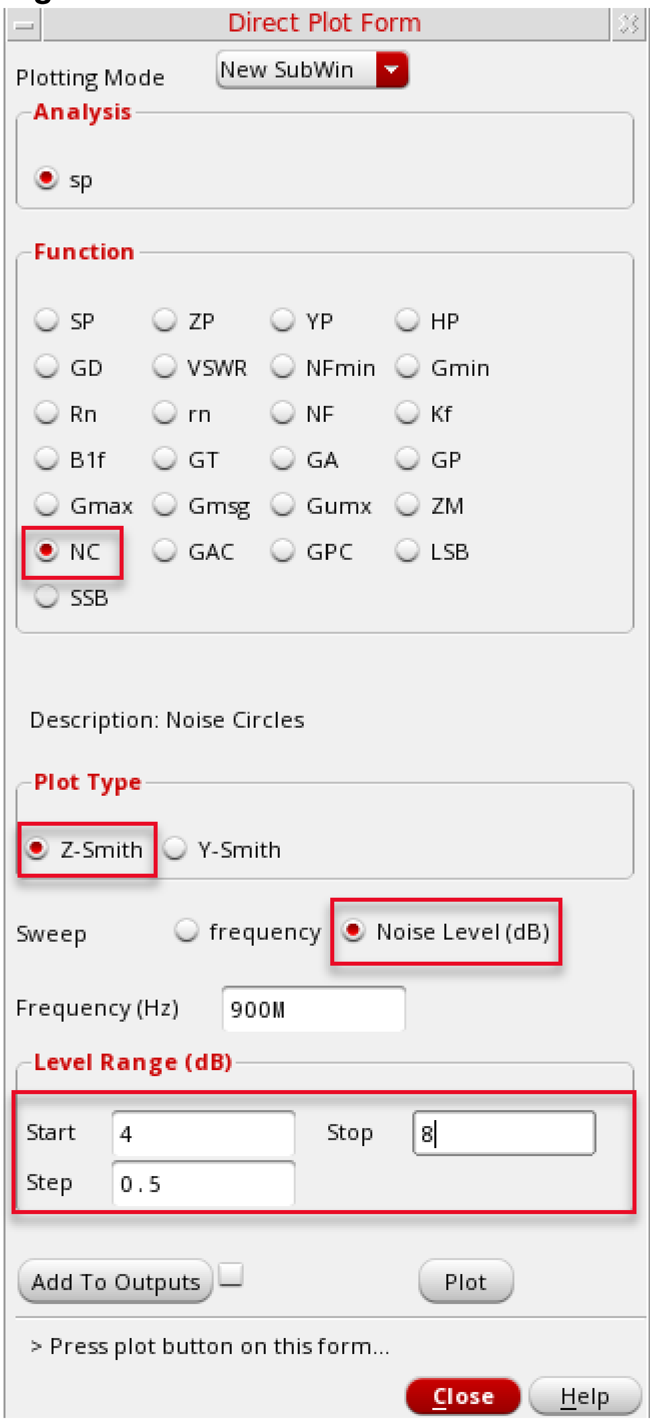
The Noise Figure plots are at a minimum at the frequency of operation.

Next you will plot Noise Circles.

8. In the *Direct Plot Form*, change the *Plotting Mode* to *New SubWin.*
9. In the *Function* section, select *NC*. The form changes.
10. In the *Plot Type* section, select *Z-Smith*. This plots the noise circles on the Impedance Smith Chart (Choosing Y-Smith plots the noise circles on the Admittance Smith Chart.)
11. In the *Sweep* section, select *Noise Level (dB)*. You will be plotting circles of constant noise level at a single frequency.

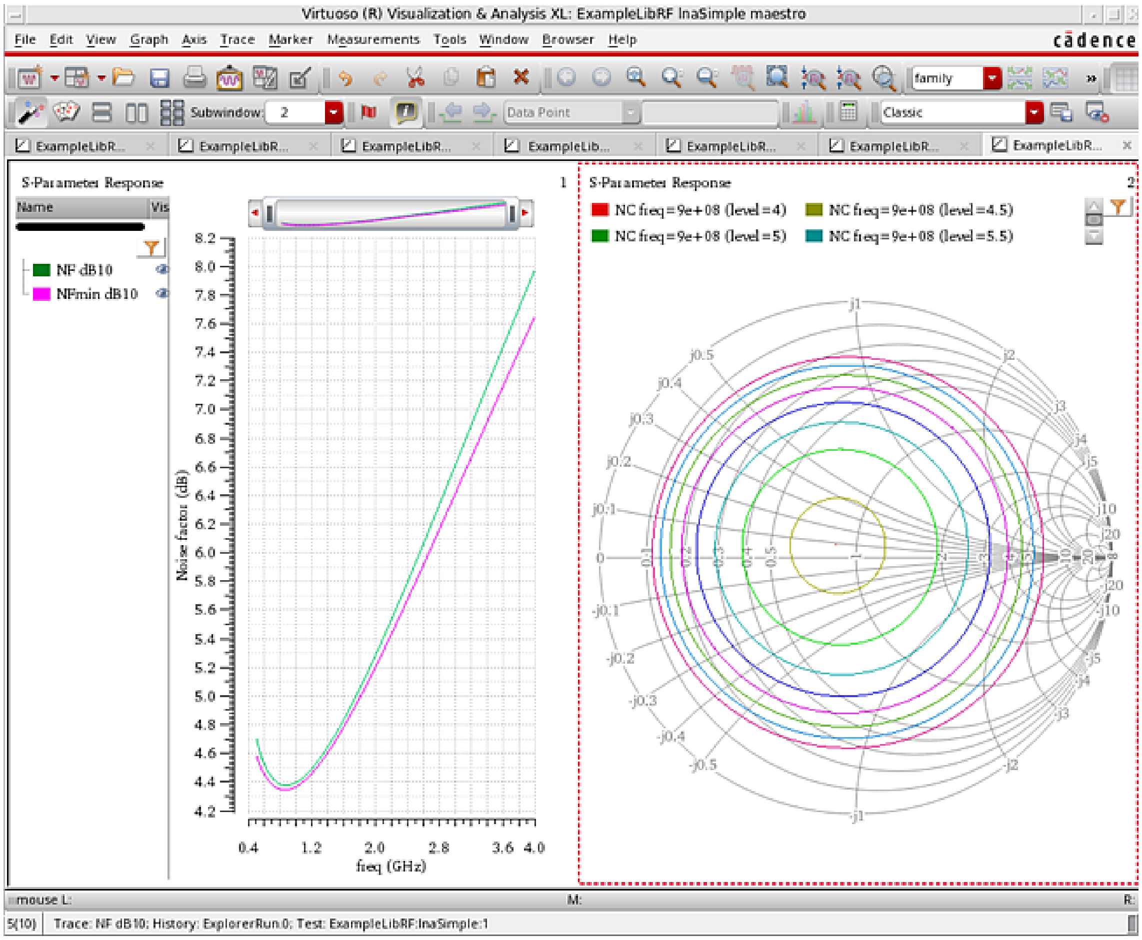
12. In the *Frequency(Hz)* field, type 900M. This is the operating frequency of the lnaSimple circuit.
13. In the Level Range (dB) section, set *Start* to 4, *Stop* to 8, and *Step* to 0.5. The *Direct Plot Form* should look like the figure below.

Figure 2-45 Direct Plot Form for Noise Circles



14. Click *Plot*. The Noise Circle plot is displayed, as shown below.

Figure 2-46 Noise Circle Plot



The optimum location for the center of the noise circle is at the center of the Smith chart. However, it is hard to center both the available gain circle, GAC, and the noise circle, NC, in the Smith chart. When you design an LNA, plot NC, GAC, and the source stability circle, SSB, together in the same plot. Use this plot to trade-off the gain, noise, and stability for the input matching network design.

15. Clean up the screen for the next set of measurements.

a. Close the *Results Display* window by choosing *Window - Close*.

b. Close the *ADE Explorer* window by choosing *Session - Quit.*

c. In the Schematic window, choose *File - Close.*

Summary

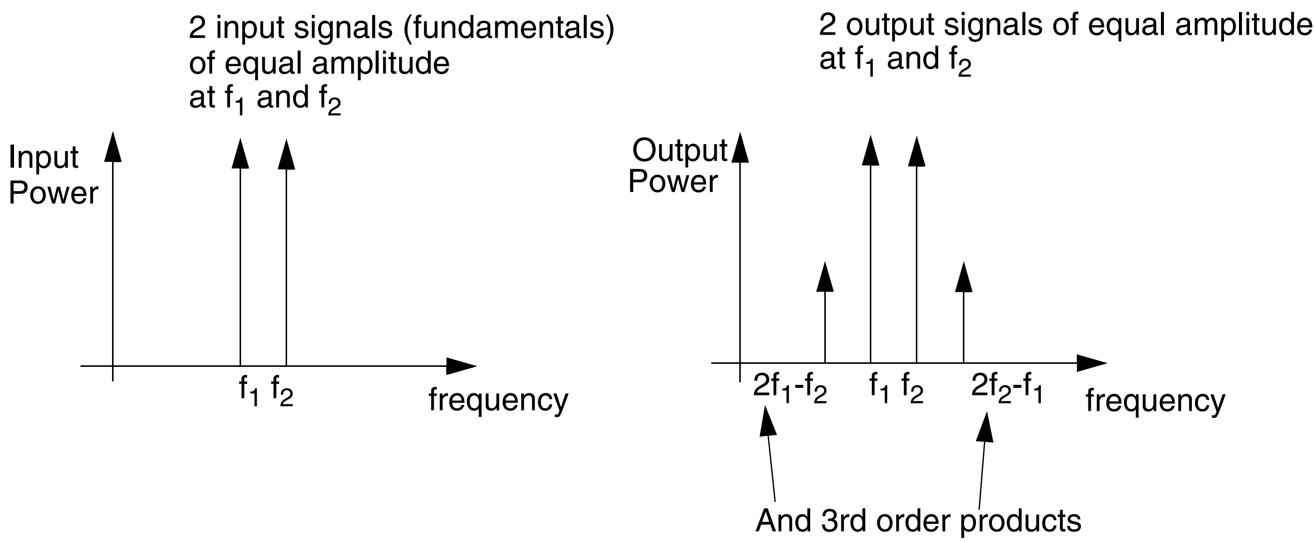
In this section, you have looked at common measurements made on Low Noise Amplifier circuits, specifically Gain, Stability, and Noise. For other examples of Measurements, see Chapter 3: Frequency Domain Analyses: Harmonic Balance in the Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide and the LNA Workshop located in the MMSIM hierarchy.

In the next section, you will measure the 1dB Compression Point and IP3 using two different methods.

Third-Order Intercept measurement with HB (2 tone HB)

In narrowband circuits, distortion is commonly measured by applying two pure sinusoids with frequencies well within the bandwidth of the circuit (call these frequencies f1 and f2). The harmonics of these two frequencies would be outside the bandwidth of the circuit, however, there are distortion products that fall at the frequencies $2f_1$-f_2, $2f_2$-f_1, and so on. These frequencies are within the bandwidth of the circuit and can be used to measure the intermodulation distortion, or IMD, produced by the circuit.

IP3 is an important RF specification. The IP3 measurement is defined as the cross point of the power for the 1st order tones, f_1 and f_2, and the power for the 3rd order tones, $2f_1 – f_2$ and $2f_2 – f_1$, on the load.

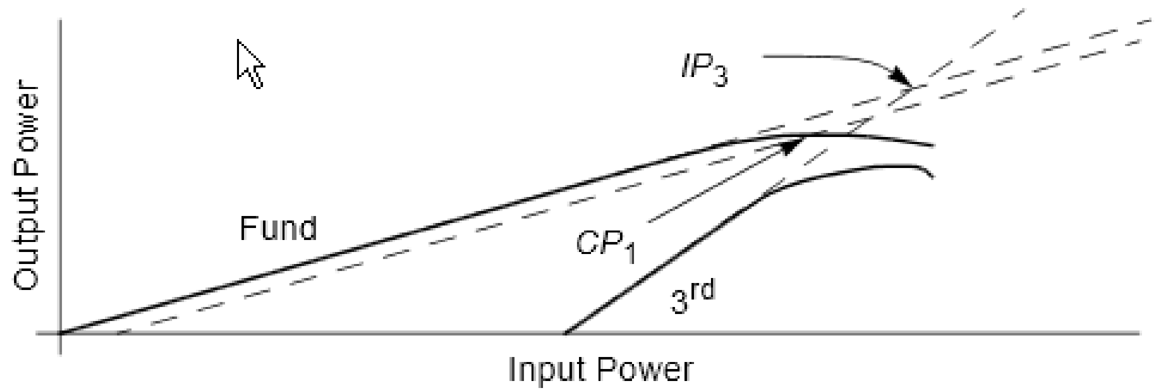


Assuming the input amplitudes are equal, the output first order terms will have the same amplitude and the output third order terms will also have the same amplitude.

As the first order components grow linearly and the third order components grow cubically, they eventually intersect as the input power level increases.

The third order intercept point is where the two output power curves intersect, as shown in the figure below.

Figure 2-47 1dB Compression Point and IP3 Curves



CP_1 is the 1dB compression point
IP_3 is the third order intercept

SpectreRF provides several different ways of simulating IP3 for a low noise amplifier. You will measure IIP3 (input IP3) using two different ways.

The first way is to use hb analysis with two RF tones applied. Generally, the RF power is swept over a range in the small-signal region. Hb is a large-signal simulation that calculates all the harmonics. Because the third-order product is quite small at low-input power, the accuracy of the simulation result must be kept quite high. This usually requires lengthy simulations.

The fastest approach is to select Rapid IP3 from the AC analysis. This is the fastest approach but it is limited to small-signal (at least 10dB below the 1dB compression point) IP3 measurements. In a small-signal application, both techniques produce answers typically within 0.1dB of each other.

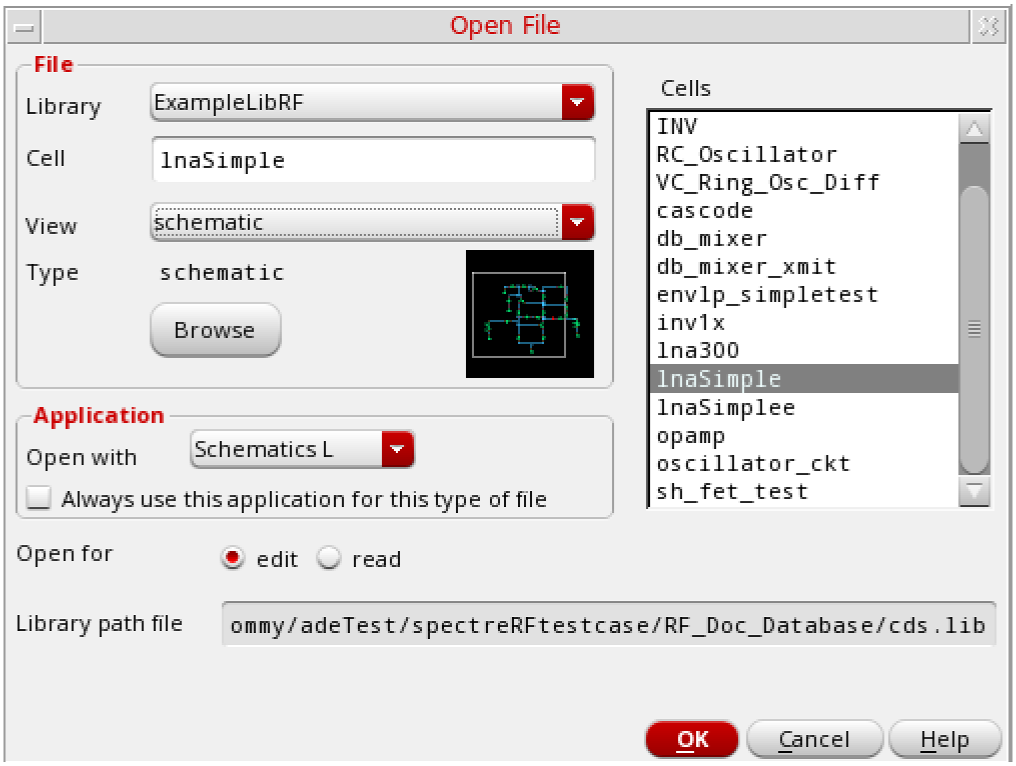
Opening the lnaSimple LNACircuit in the Schematic Window

1. In the CIW, choose *File – Open.*

   The *Open File* form is displayed.

2. Choose *ExampleLibRF* from the *Library* drop-down list.
3. Choose *lnaSimple* from the *Cells* list box. Leave the rest of the fields at their default values.
4. The completed *Open File* form appears like the one below.

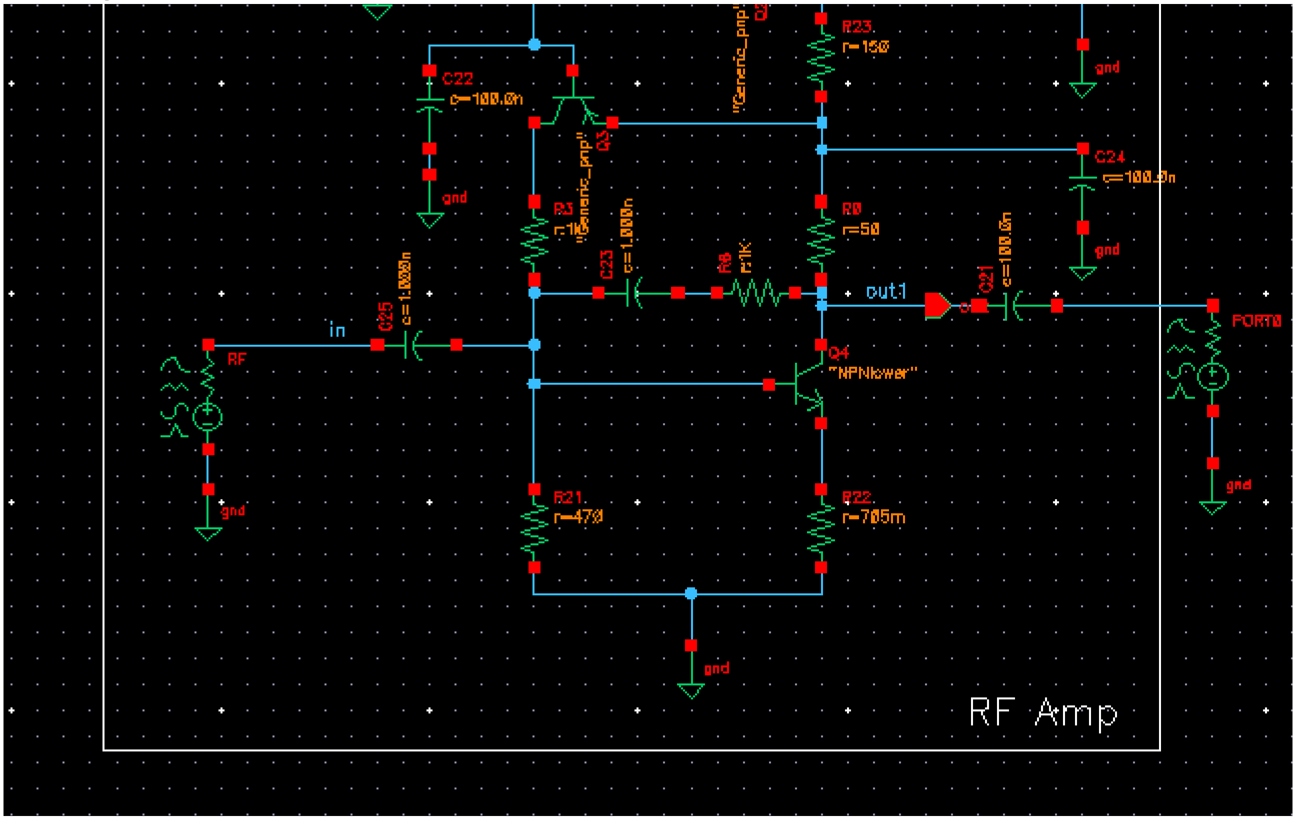
Figure 2-48 Open File Form



5. Click *OK*.

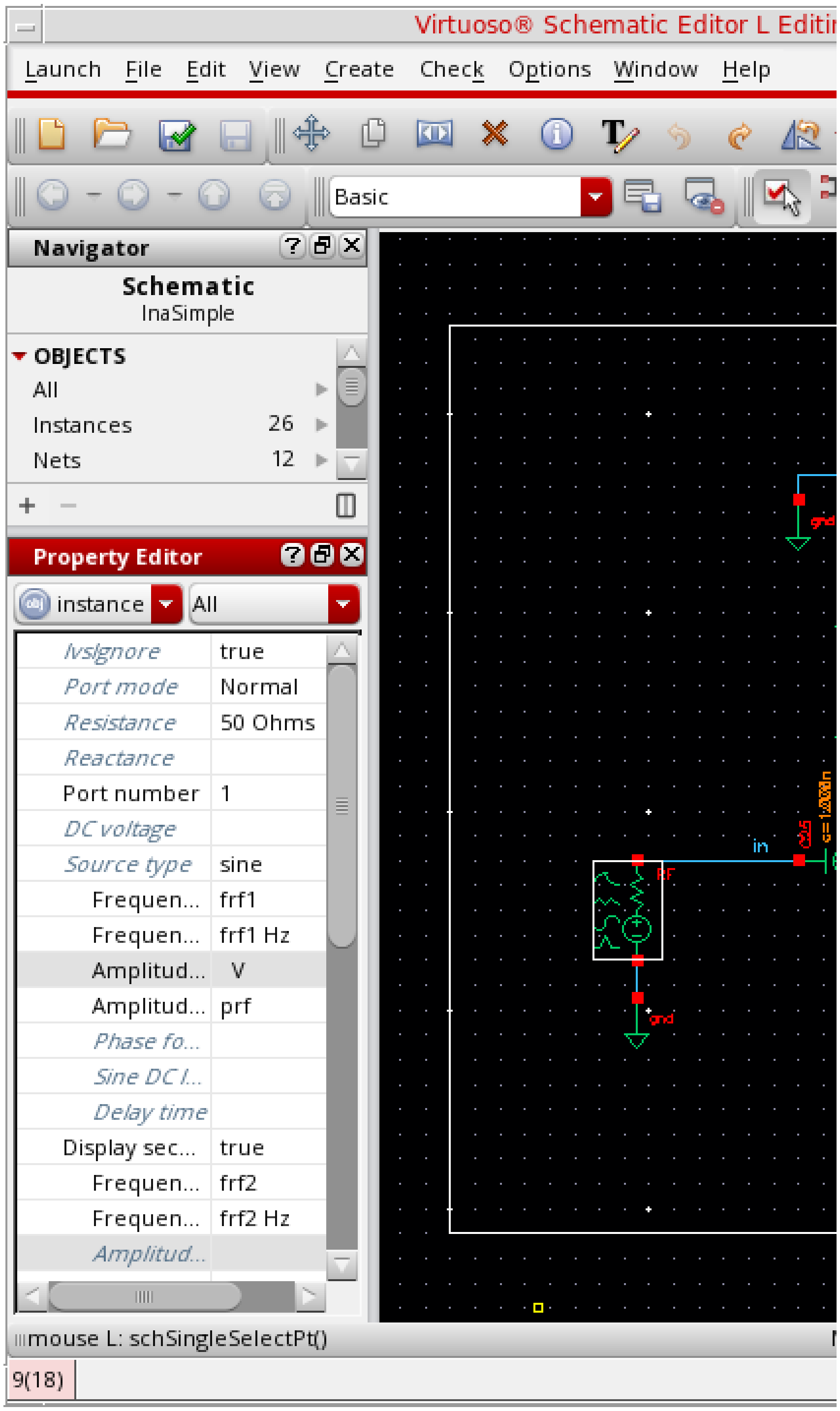
The Schematic window for lnaSimple is displayed.

Figure 2-49 lnaSimple Schematic



6. In the Schematic, click on the RF port on the left side of the schematic.

Figure 2-50 RF Port Selected



Note that when you do this, the RF port properties are populated in the *Property Editor* pane. Examine the input port settings.

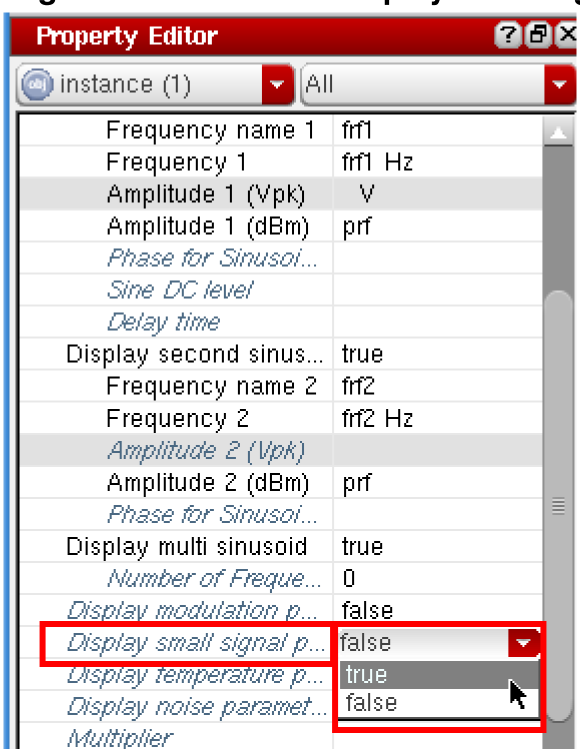
a. *Source type* `sine` means generate large signals for the time domain (transient, pss-shooting, qpss-shooting, envlp-shooting, and so on) or frequency domain (HB or envlp-HB) analyses.

The port can generate one or two large signals. In this example, it is generating two signals. You need to specify a name for all of the large signals. This is done in the *Frequency name 1* and *Frequency name 2* property fields. The frequencies are set to a variable name. The value of the variables is set in ADE Explorer. This allows sweeps to be done in the Analog Design Environment. The amplitudes can be set in volts peak or in dBm. In this case, the amplitudes are set to a variable named *prf*. When the amplitude is set in dBm, the amplitude in volts cannot selected.

The *Display second sinusoid* option is a display function only. If there is an entry there and the display option is off, the second waveform will still be generated. To switch to a single input, remove all the entries, or set the amplitude or frequency to zero in the *Amplitude* field or the *Design Variables* section of ADE Explorer.

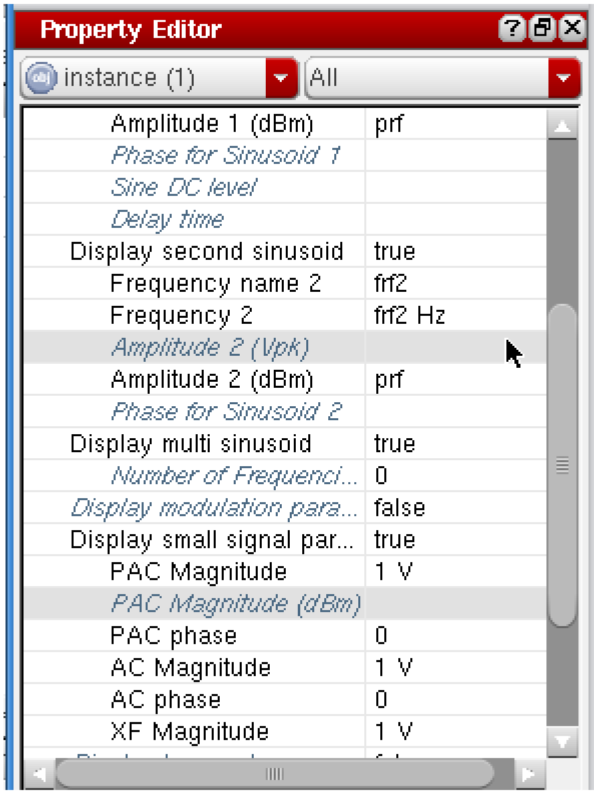
The amplitude for the small-signal analyses AC and PAC are set in the source. To view the small signal values, scroll down the form to the *Display small signal parameter* drop-down list and select *true*. The amplitude for AC and PAC can be either in Volts peak or dBm, but not both at the same time.

Figure 2-51 RF Port Display small signal parameters.



The form expands, as shown below.

Figure 2-52 Expanded RF Port Display Small Signal Parameters



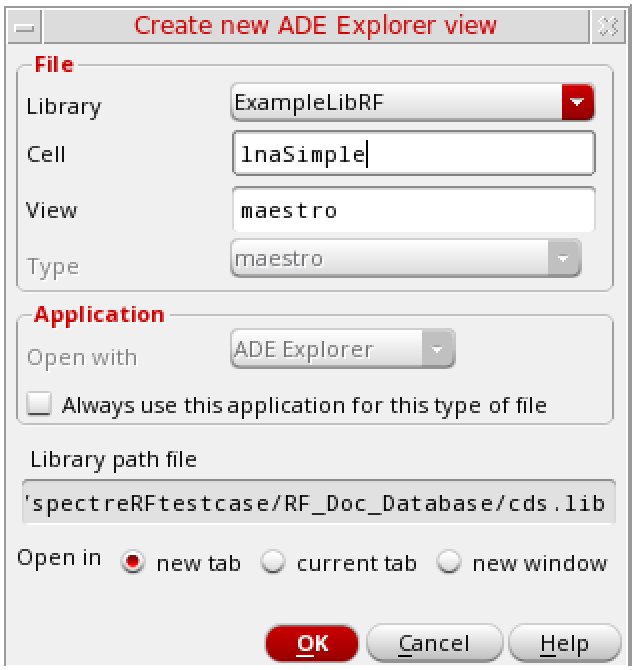
The frequency of these signals will be set in the small-signal setup form in ADE Explorer. The amplitude for PAC can be either in volts peak or in dBm, again with the form not allowing both to be set at the same time.

7. In the Schematic window, choose *Launch – ADE Explorer*.

8. In the *Launch ADE Explorer* dialog, select *Create New View*.

The *Create new ADE Explorer view* form is displayed.

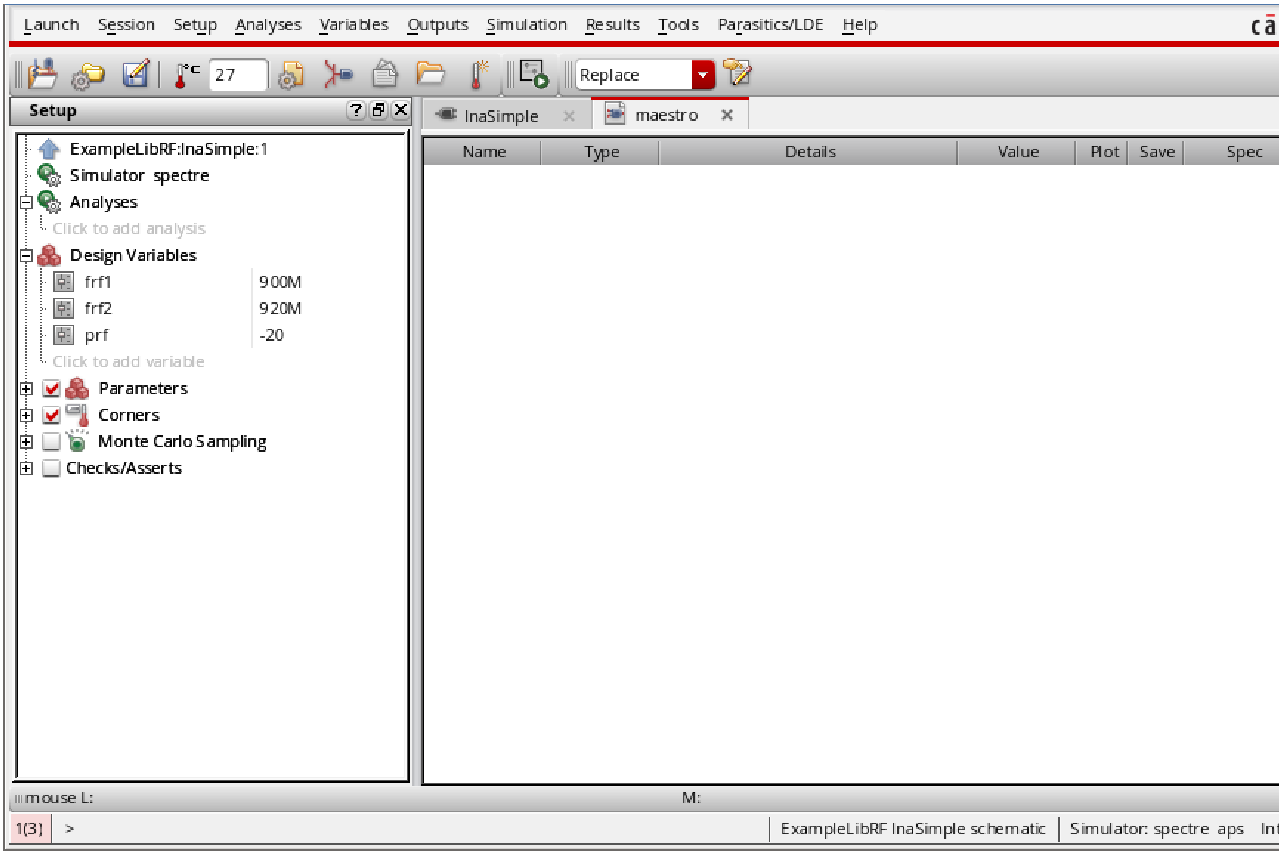
Figure 2-53 Create new ADE Explorer view



9. Leave each option to the default selections and click *OK*.

The Virtuoso ADE Explorer window opens.

Figure 2-54 Analog Design Environment Window



Choosing Simulator Options

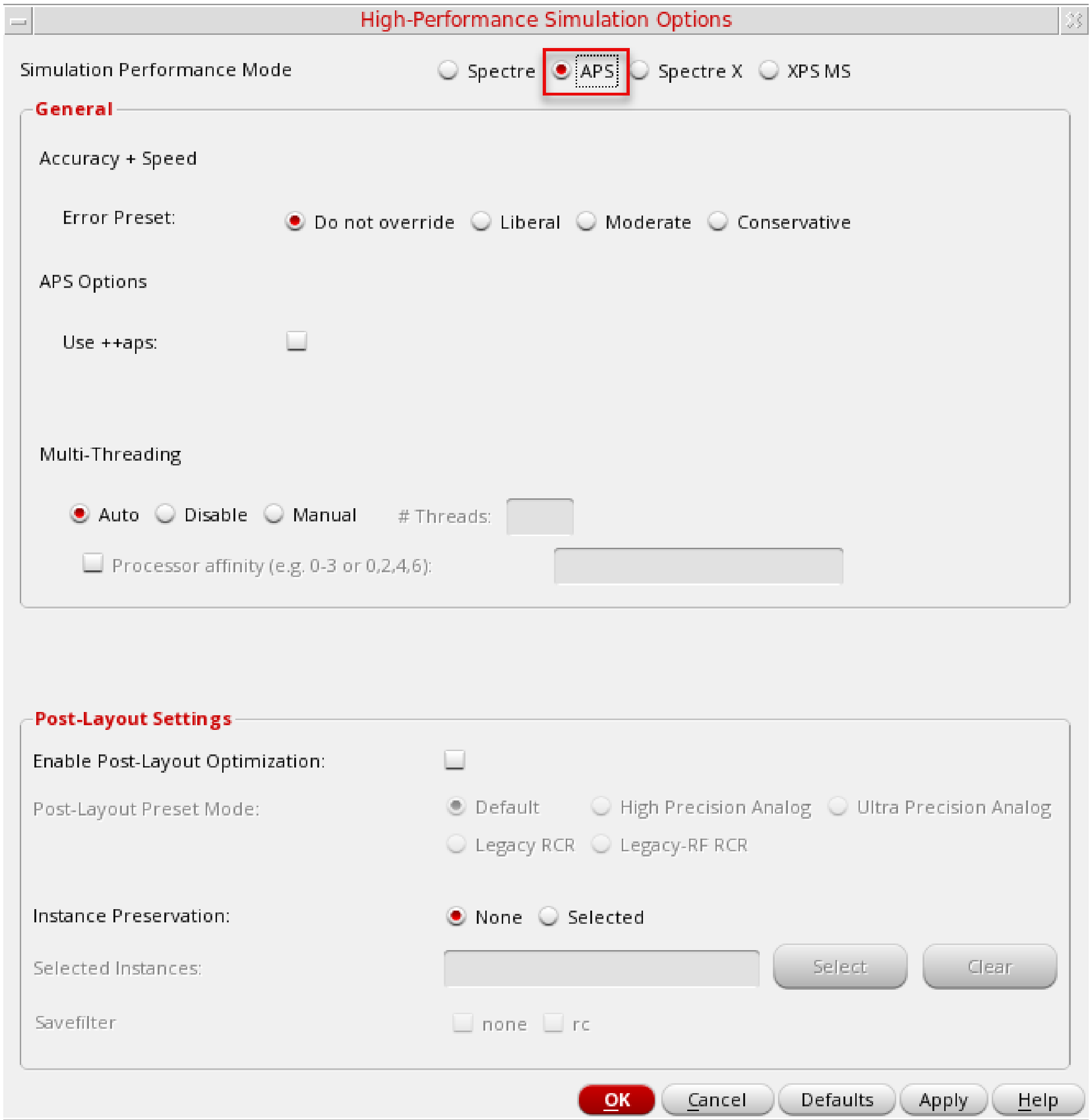
1. Choose *Setup – Simulator* in the Virtuoso Analog Design Environment window.

   The *Choosing Simulator* form is displayed.

Figure 2-55 Choosing Simulator Form

2. Choose *spectre* from the *Simulator* drop-down list.
3. Click *OK*.
4. Select *Setup - High Performance Simulation*. The *High Performance Simulation Options* window is displayed.

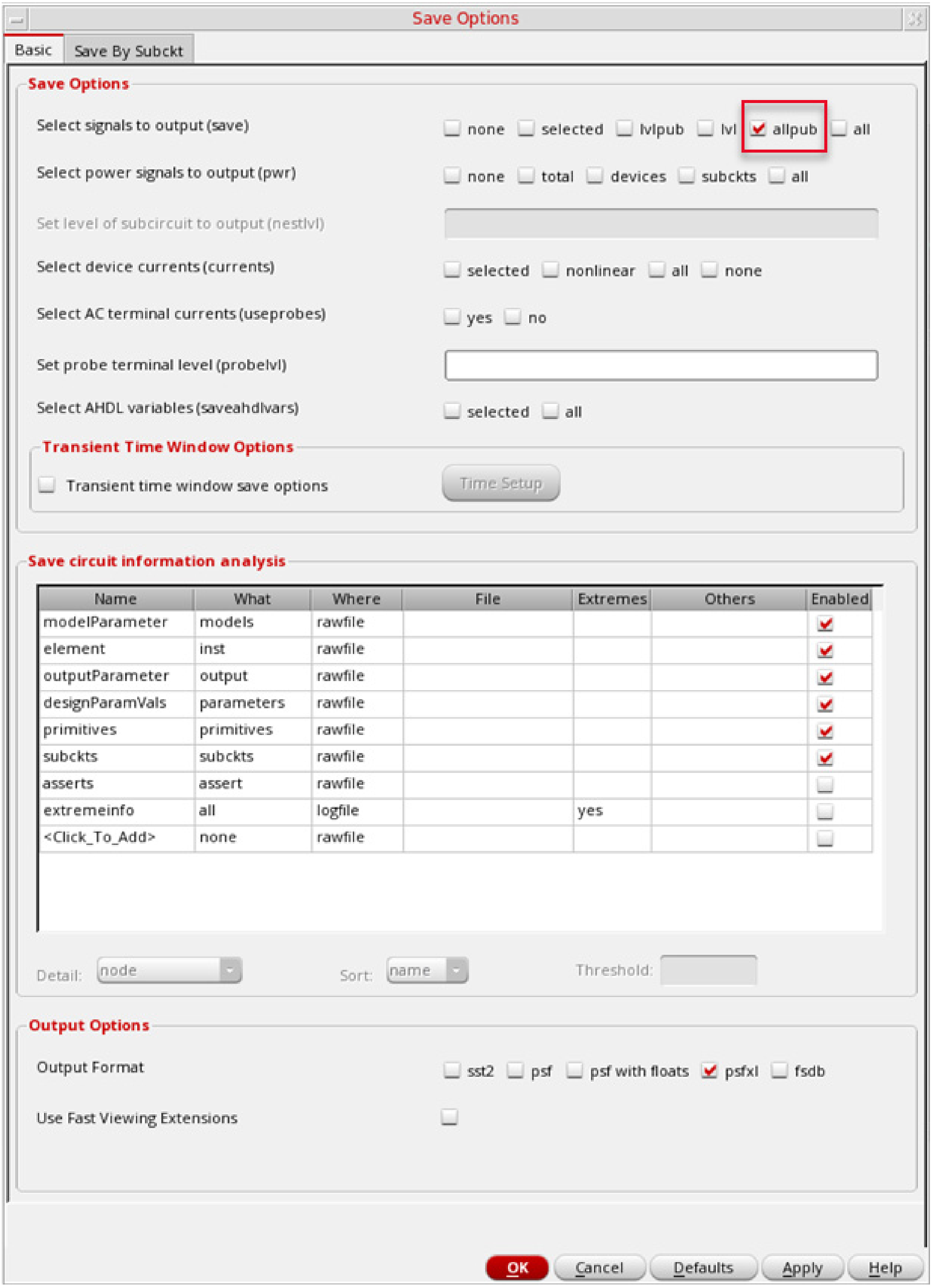
Figure 2-56 High Performance Simulation Options



5. In the *High Performance Simulation Options* window, select *APS* as the *Simulation Performance Mode.* Note that *Auto* is selected for *Multi Threading*. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores.
6. Click *OK*.
7. In ADE Explorer, select *Outputs – Save All*.

   The *Save Options* form is displayed.

Figure 2-57 Save Options Form



8. In the *Select signals to output* section, ensure that *allpub* is selected.

This is the default selection. This saves all the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

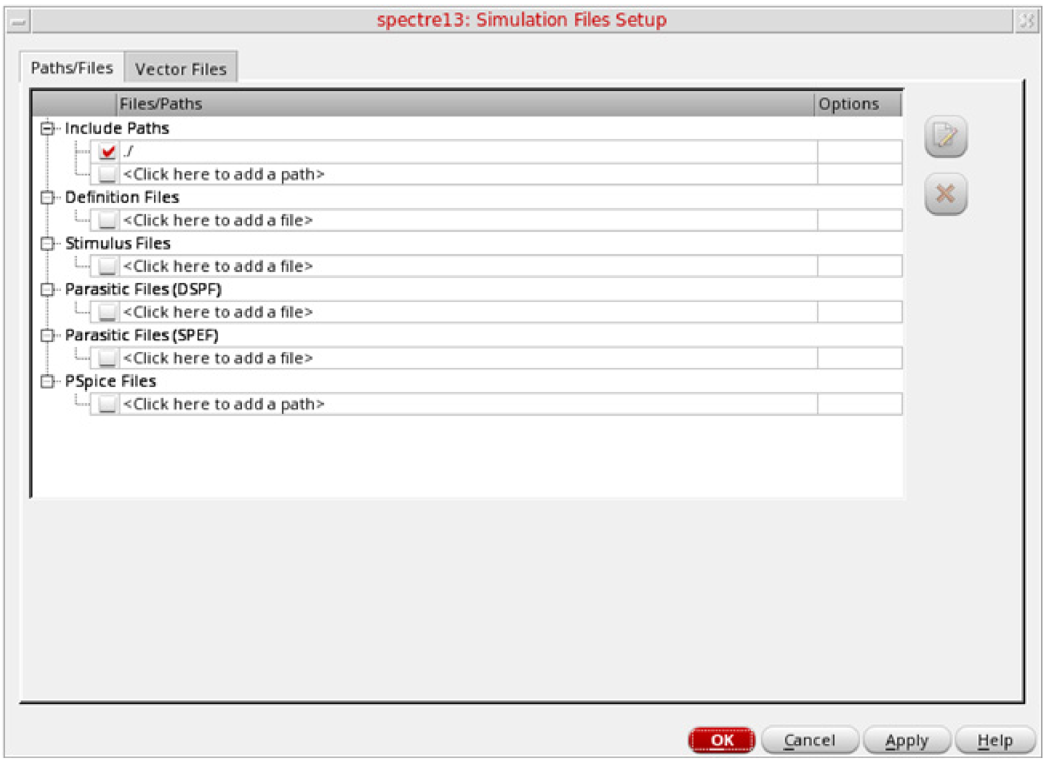
9. Click *OK*.

Setting Up Model Libraries

In ADE Explorer choose *Setup - Simulation Files*.

The *Simulation Files Setup* form is displayed.

Figure 2-58 Simulation Files Setup Form



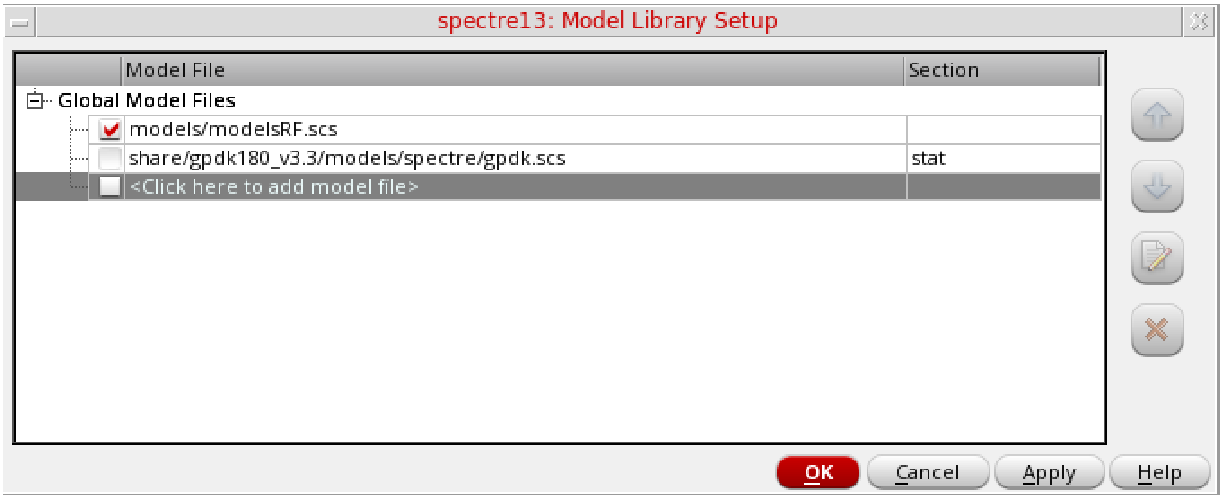
1. Verify that the *Include Path* is set as shown above and close the form.
2. Select *Setup – Model Libraries.*

   The *Model Library Setup* form is displayed.
3. In the *Model Library File* field, type in the name of the model file, as shown below.

   `models/modelsRF.scs`
4. Click *Add*.

   The *Model Library Setup* form looks like the following:

Figure 2-59 Model Library Setup



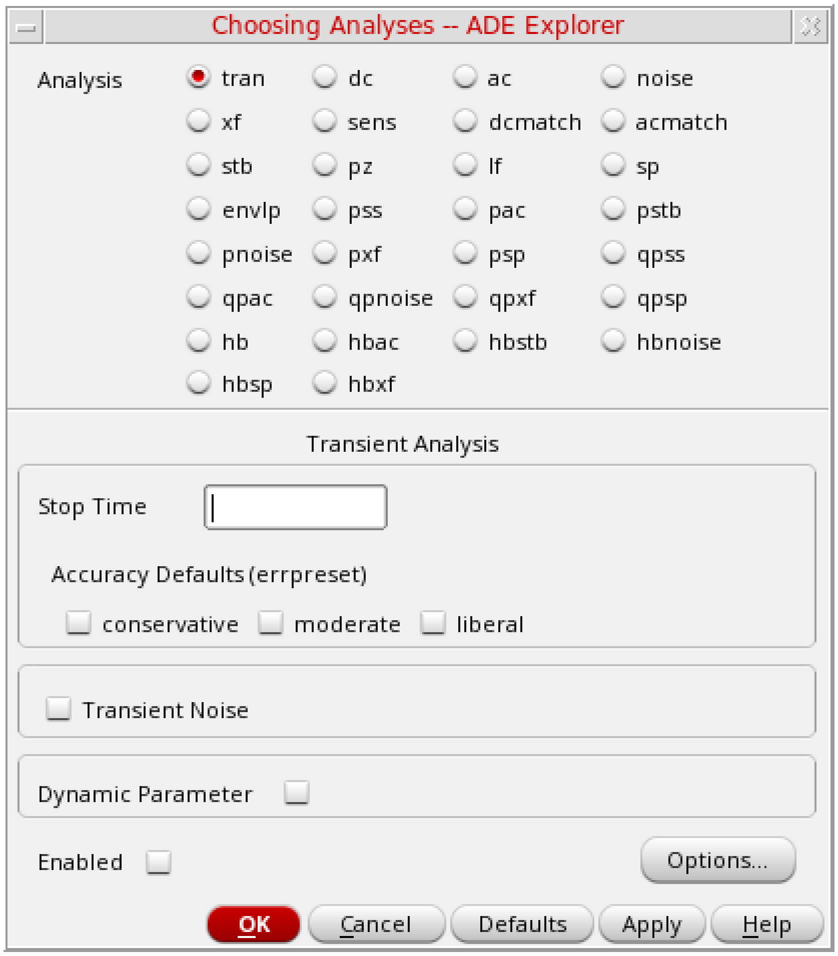
5. Click *OK*.

   Alternately, you can click the *Browse* button and select the `modelsRF.scs` model file.

6. Select *Analyses - Choose*. Alternately, you can click the *Choose Analyses* icon .

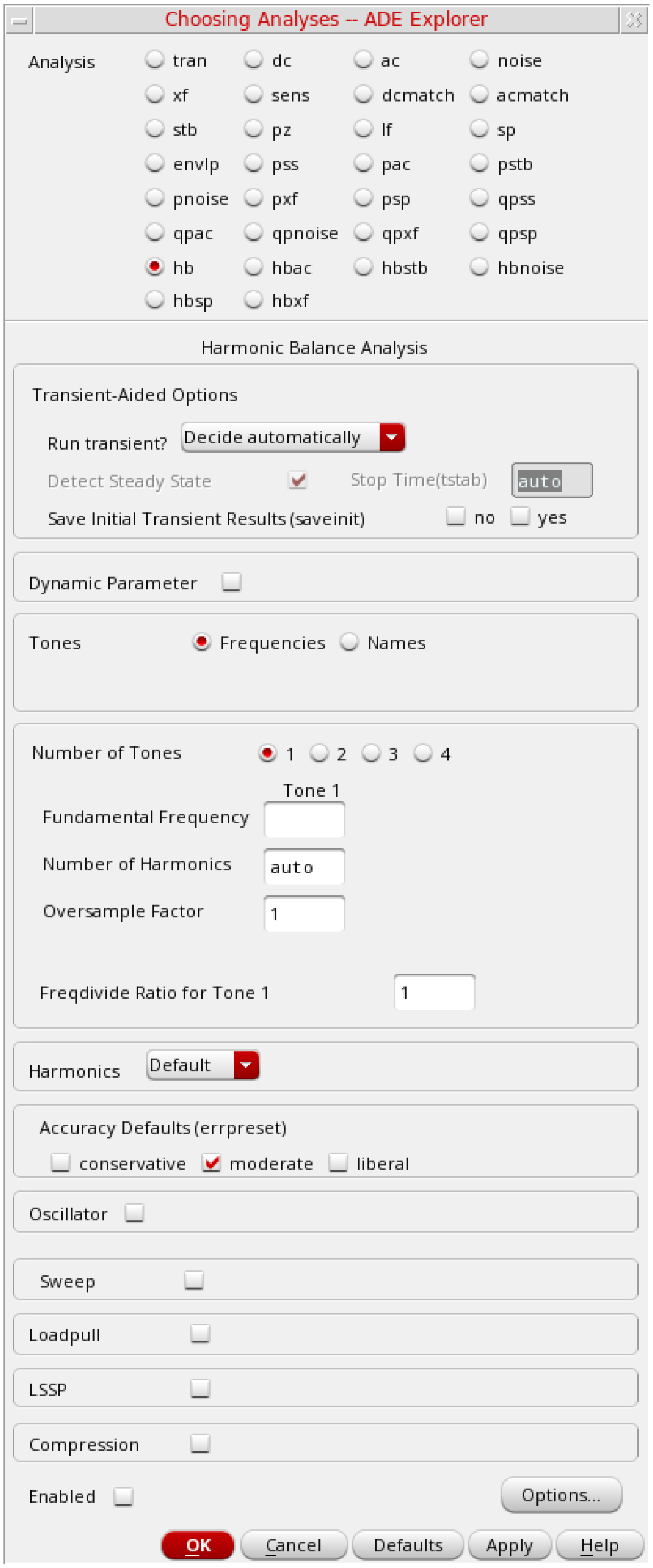
   The *Choosing Analyses* form is displayed.

Figure 2-60 Choosing Analyses Form



7. In the Analyses section, select *hb*. The form expands, as shown below.

Figure 2-61 hb Choosing Analyses Form



Harmonic balance can set the harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. To enable this, select *Decide automatically* or *Yes* from the *Run Transient?* drop-down list in the *Transient-Aided Options* section. This single action will cause a transient analysis to be run until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1* (when *Frequencies* is selected) or for the tone that has tstab enabled (when *Names* is selected).

8. In the Transient-Aided Options section of the form, select the following

   a. For *Run transient?* select *Decide automatically*. (this is the default)

      Run transient? will run the large signal using the transient (In SpectreRF, this is called the *tstab* interval) for a short period of time. At the end of *tstab*, an FFT is performed, and this is used as the starting point in the harmonic balance analysis. Doing this improves the convergence of hb by giving it a better starting point at the cost of a short transient analysis.

   b. For *Stop time (tstab), auto* is automatically populated in the field

      When *auto* is selected for *Stop Time*, a small number of periods of the large signal is run using the transient analysis. During this time, the signal is checked for steady-state conditions. If steady-state is not reached in the initial number of periods, more periods can be added automatically by the simulator. Using this feature allows an accurate FFT for the starting point in the hb iterations.

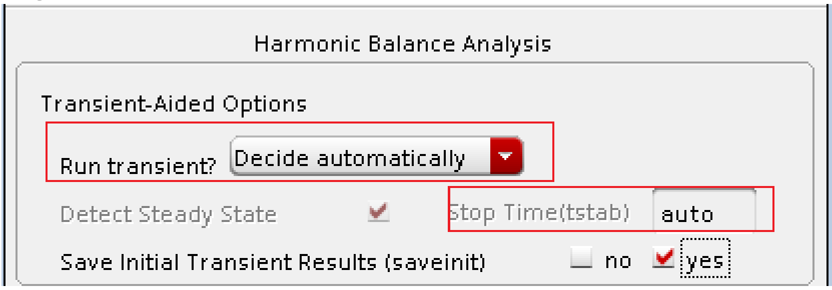
      When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, the simulator stops the transient analysis when steady-state is detected in the *tstab* interval, runs the FFT, and starts iterating in the frequency domain. Using *Decide automatically* simplifies the setup for harmonic balance, and produces correct answers from hb without needing to know how to set hb up manually.

   c. For *Save Initial Transient Results (saveinit),* select `yes`.

      During the transient-assisted HB simulation, a transient simulation runs before the frequency domain iteration of harmonic balance. The large signal in *Tone1* is enabled for this measurement. At the end of the *tstab*, an FFT is run and its result is used as the starting point for the frequency domain iterations

      All the signals are applied and the simulation is done in the frequency domain.Only the signal and its harmonics are calculated.

Figure 2-62 Transient Assisted Harmonic Balance



9. In the *Tones* section, select *Names*. When *Names* is selected, the *Tones* portion of the form expands. All the sources in the top-level schematic are read into the form automatically.

   Note that there are two large tones. The frequency names for all large signal tones are automatically populated from the schematic. You viewed the tones earlier when viewing *Property Editor* for the RF port.

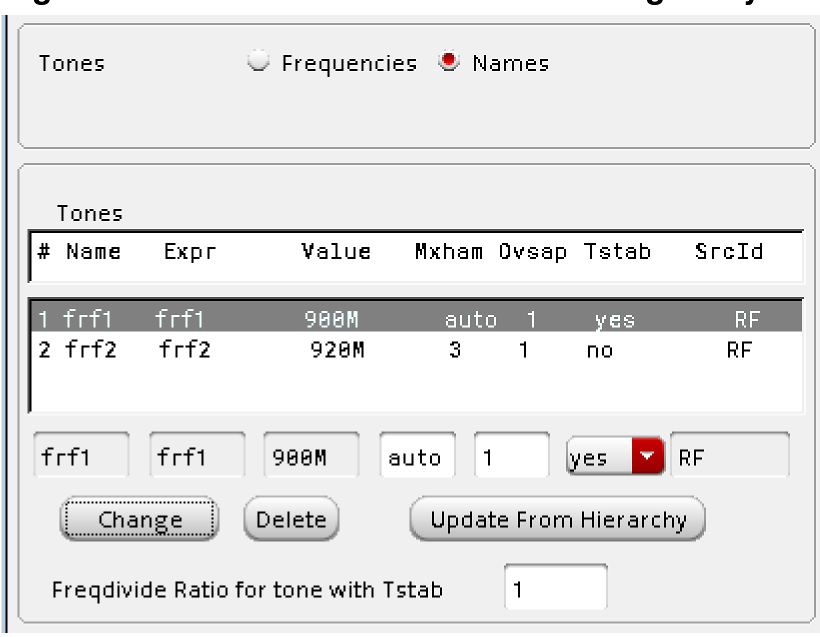
   The frequency values for the frf1 and frf2 tones are set in the *Design Variables* section of the ADE Explorer, which will be shown later.

10. Select the *frf1* tone in the *Tones* field.

11. Set the *Mxham* value to `auto` and click *Change*.

    The form updates. Spectre will choose the appropriate number of harmonics for you.

Figure 2-63 Tones Section of hb Choosing Analyses Form



12. When you set harms to `auto`, leave *OvSap* (oversample factor) set to the default value of `1`.

13. Leave *Tstab* set to the default value of *yes*.

    Because you are using auto-tstab, you do not need to set *Tstab* to `yes` in the *Tones* section for one of the large signal tones. The signal with `tstab=yes` is the signal that is used for transient-assisted harmonic balance. Only one signal can have transient assist, that being the signal with *Tstab* set to `yes`.

    Note: If for some reason you are not using auto-tstab, set *tstab* to `yes` on the signal that causes the largest amount of distortion in the system.

    During the tstab interval, a transient analysis is run before the frequency domain iteration of harmonic balance. At the end of the tstab, an FFT is run and its result is used as the starting point for the frequency domain iterations. All the signals are applied and the simulation is performed in the frequency domain.

14. Leave the *Harmonics* field set to *Default*.

15. In the *Accuracy Defaults* section, verify that *moderate* is selected. For most normal measurements *errpreset* should be set to *moderate*. When you need to measure really small distortions, use *conservative*.

16. To set up a sweep analysis, select the *Sweep* check box and set the value for *Sweep* to 1 (this is the default value).

17. For *Frequency Variable?* select *no*. You will be sweeping input power rather than frequency.

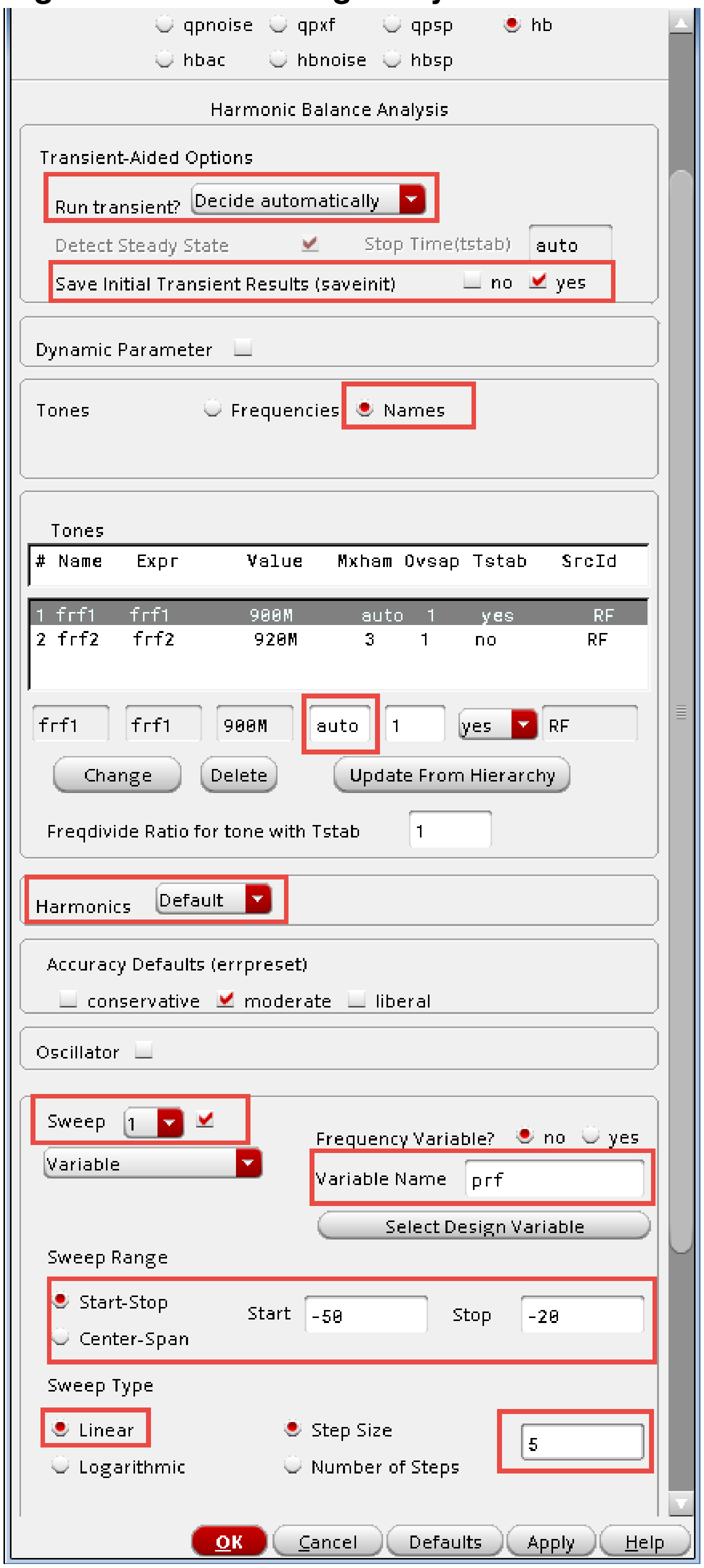
18. Type `prf` in *Variable Name*.

19. In the *Sweep Range*, type -50 in the *Start* field and -20 in the *Stop* field. Typically, you want to choose an input power that is at between 20-40 dB below the 1dB CP.

20. Select *Linear* from the *Sweep Type section* and enter 5 in the *Step Size* field.

21. Leave the rest of the form set to the default values. The hb *Choosing Analyses* Form should look like the following figure:

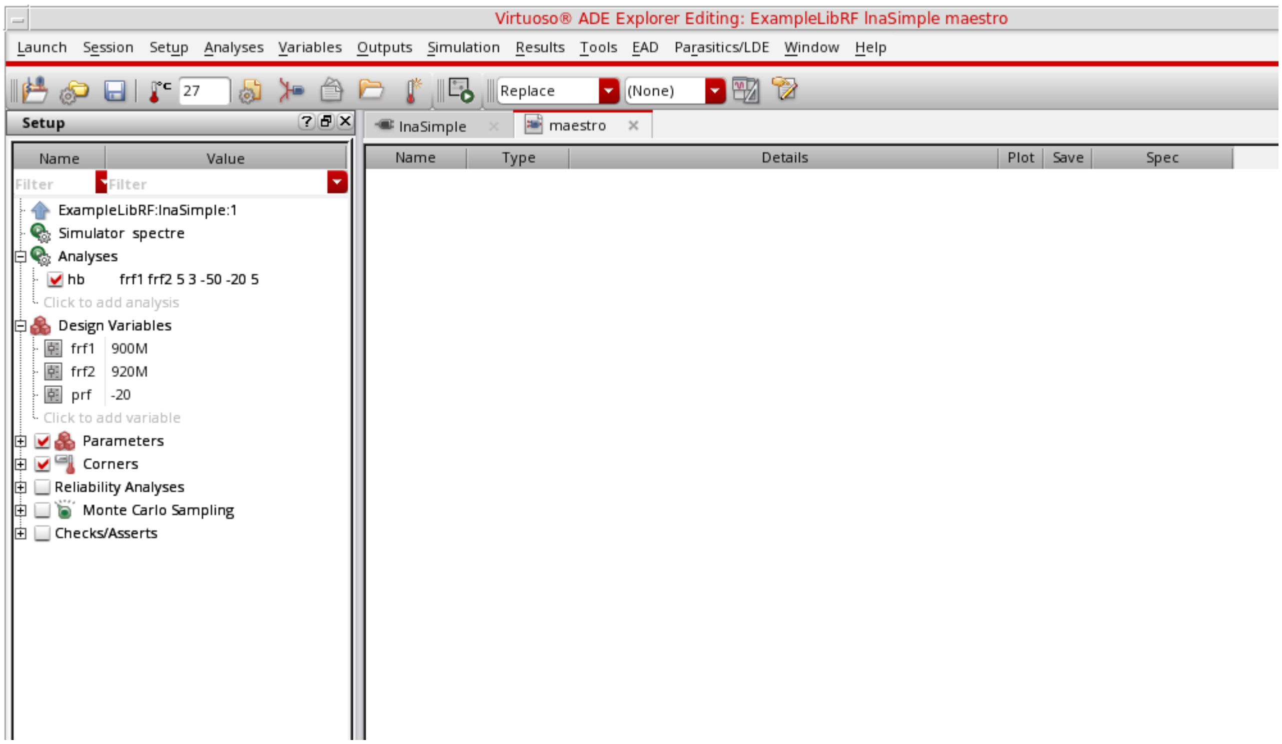
Figure 2-64 Choosing Analyses Form for Two Tone Swept HB Analysis



22. Click *Apply*.

23. View the *Design Variables* section in ADE Explorer. Verify that your *frf1* and *frf2* values are `900M` and `920M` respectively. Your ADE Explorer window should look like the following:

Figure 2-65 Analog Design Environment Window



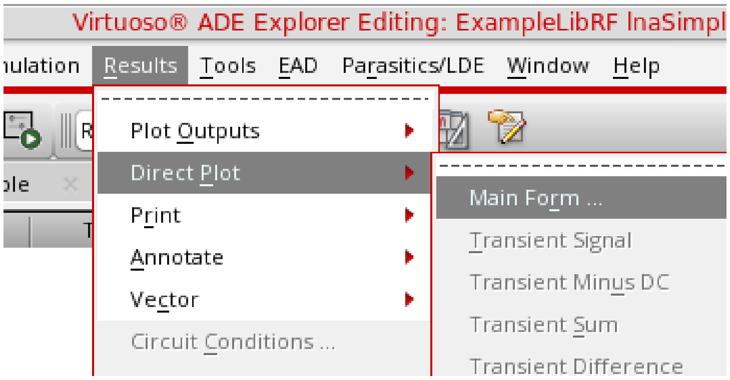
24. Start the simulation by choosing *Simulation - Netlist and Run* or by clicking the green arrow icon  on the right side of the simulation window.

This netlists the design and runs the simulation. A SpectreRF status window appears (spectre.out logfile). Note the simulation time in the Spectre output logfile.

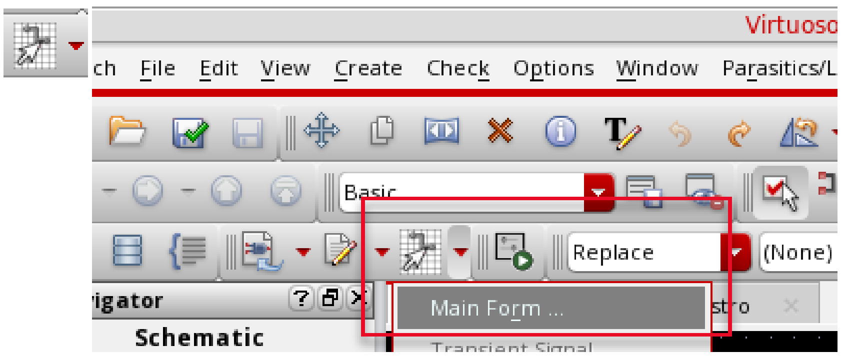
```
Total time required for hb analysis `sweephb-006_hb': CPU = 17.997 ms, elapsed = 49.8779 ms.
Time accumulated: CPU = 232.963 ms, elapsed = 1.02819 s.
Peak resident memory used = 30.8 Mbytes.

Total time required for sweep analysis `sweephb': CPU = 126.98 ms, elapsed = 481.89 ms.
Time accumulated: CPU = 233.963 ms, elapsed = 1.03305 s.
Peak resident memory used = 30.8 Mbytes.
```

25. When the analysis has completed, you may iconify the status window.

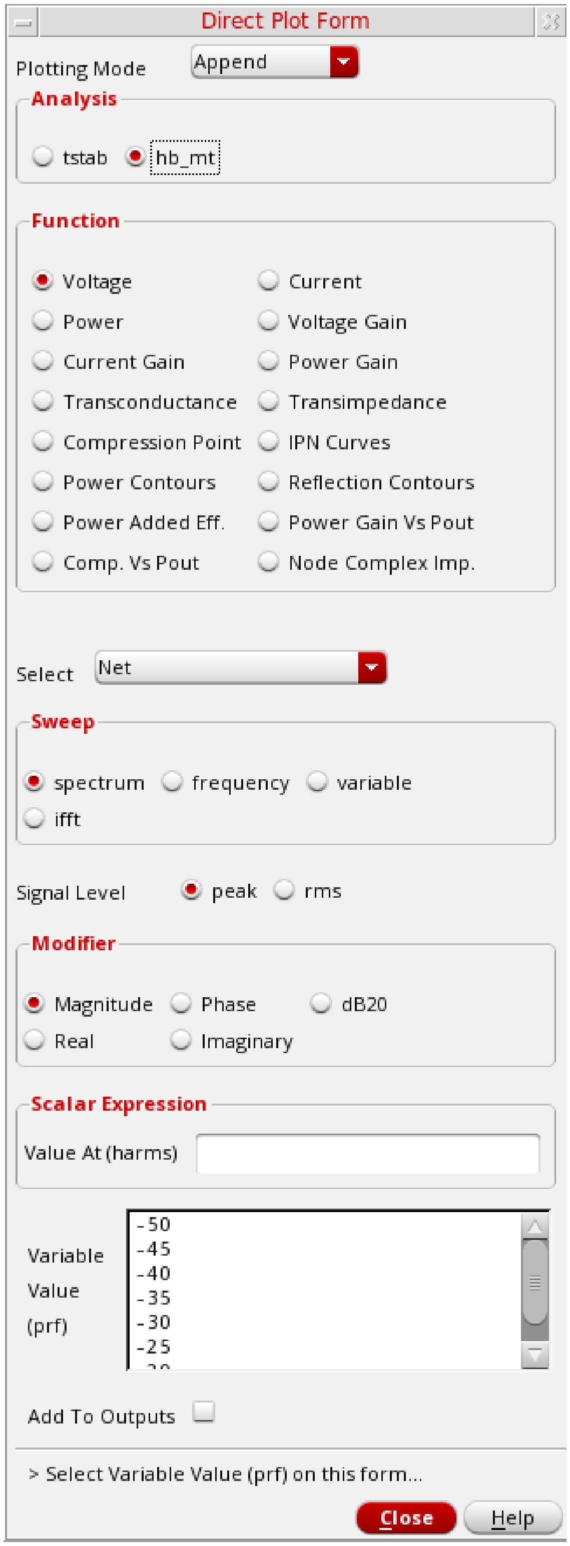
26. In ADE Explorer, select *Results - Direct Plot - Main Form*.



Alternately, you can click the *Direct Plot* icon from the schematic window.



27. The Direct Plot Form is displayed.



28. In the *Analysis* Section, select *hb_mt*. *hb_mt.* It refers to multitone harmonic balance.

29. In the *Function* Section, select *IPN Curves.*

30. Select *Net (Specify R)*. Leave the *Resistance* set to the default value of `50`.

31. Select *Variable Sweep ("prf")* for *Circuit Input Power*.

32. Leave *Input Power Extrapolation Point (dBm)* set to the default (first point in the sweep which is -50). The extrapolation point is where the ideal curves and the data are drawn through the same point. Pick an area where the third order curves have a slope of 3.

33. Select *Input Referred IP3.*

34. *Select 3rd from the Order* drop-down list.

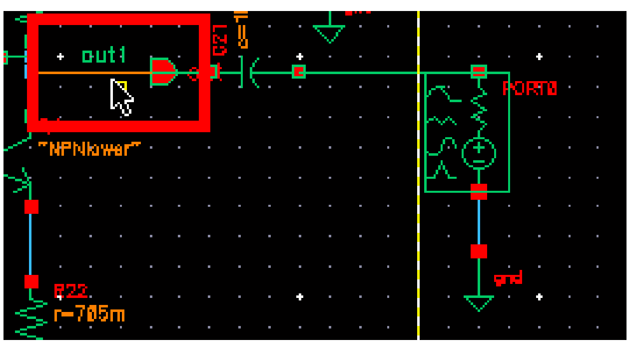
35. For the *3rd Order Harmonic*, select *940M*.

36. For the *1st Order Harmonic,* select *900M*.

Alternately, you can select *880M* and *920M* for the third and first order harmonics, respectively.

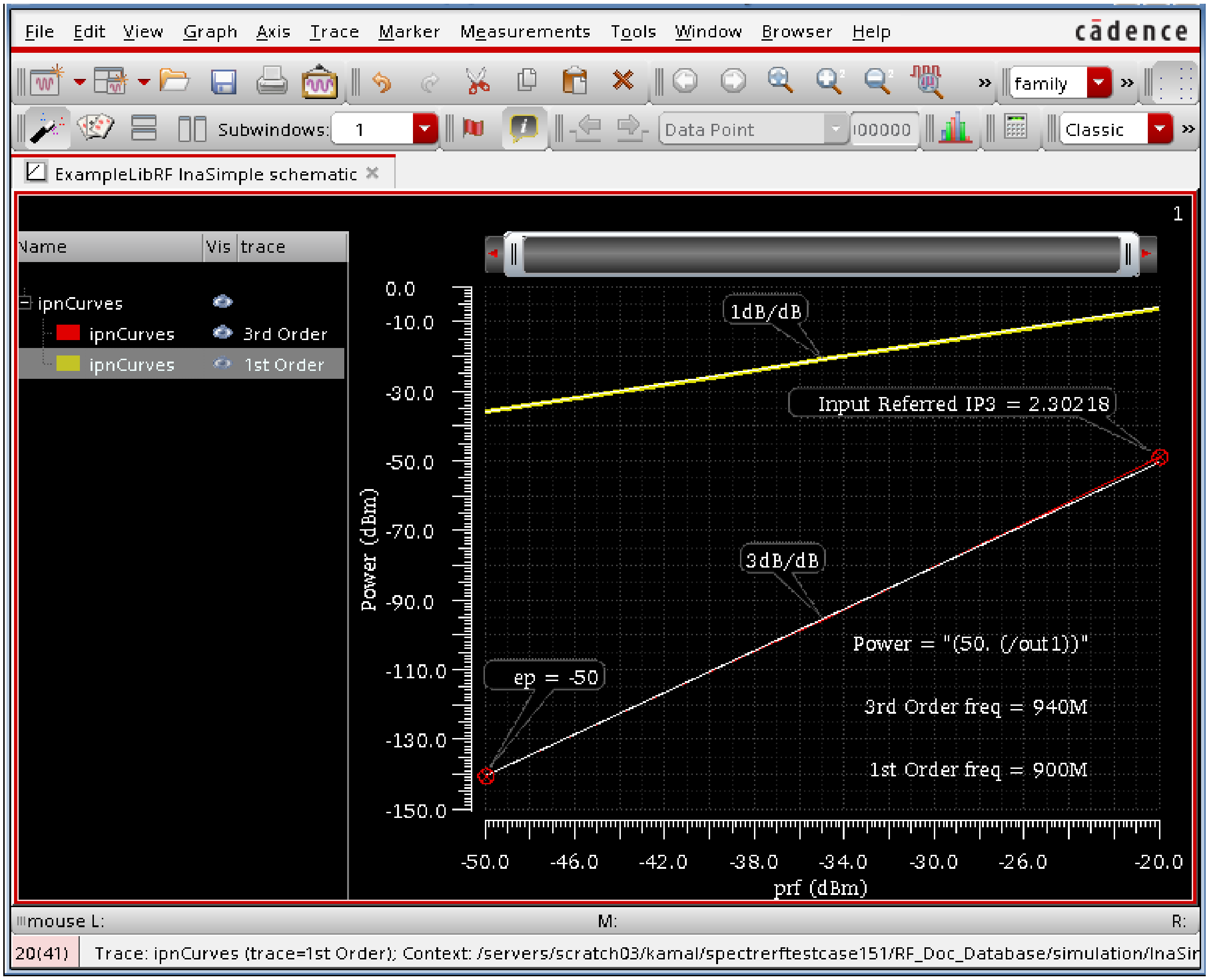
37. Select the *out1* node on the right side of the schematic

Figure 2-66 Selecting the out1 Net on the lnaSimple Schematic.



The IP3 plot is displayed. Note the IP3 readout.

Figure 2-67 The IP3 Plot



Note: If you do not see the IP3 readout, click in the graphics area to deselect the marker, then select and move the *Input Referred IP3* readout so that it is positioned in the visible area of the graph.

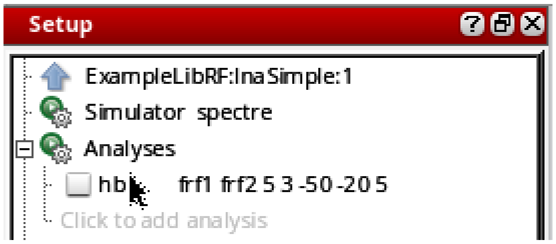
Intermodulation products increase at rates that are multiples of the fundamentals. In the small-signal region, third order terms increase 3dB per dB.

In the above plot, you can see that the circuit is operating within the small signal region. The third order curve is following a 3dB/dB slope. Note the IP3 measurement. In the next section, you will compare this to the IP3 measurement using the Rapid IP3 methodology.

38. In the Waveform window, select *File - Close All Windows*.

39. In ADE Explorer, deselect the hb analysis in the *Setup Assistant*.

Figure 2-68 ADE Explorer Setup Assistant

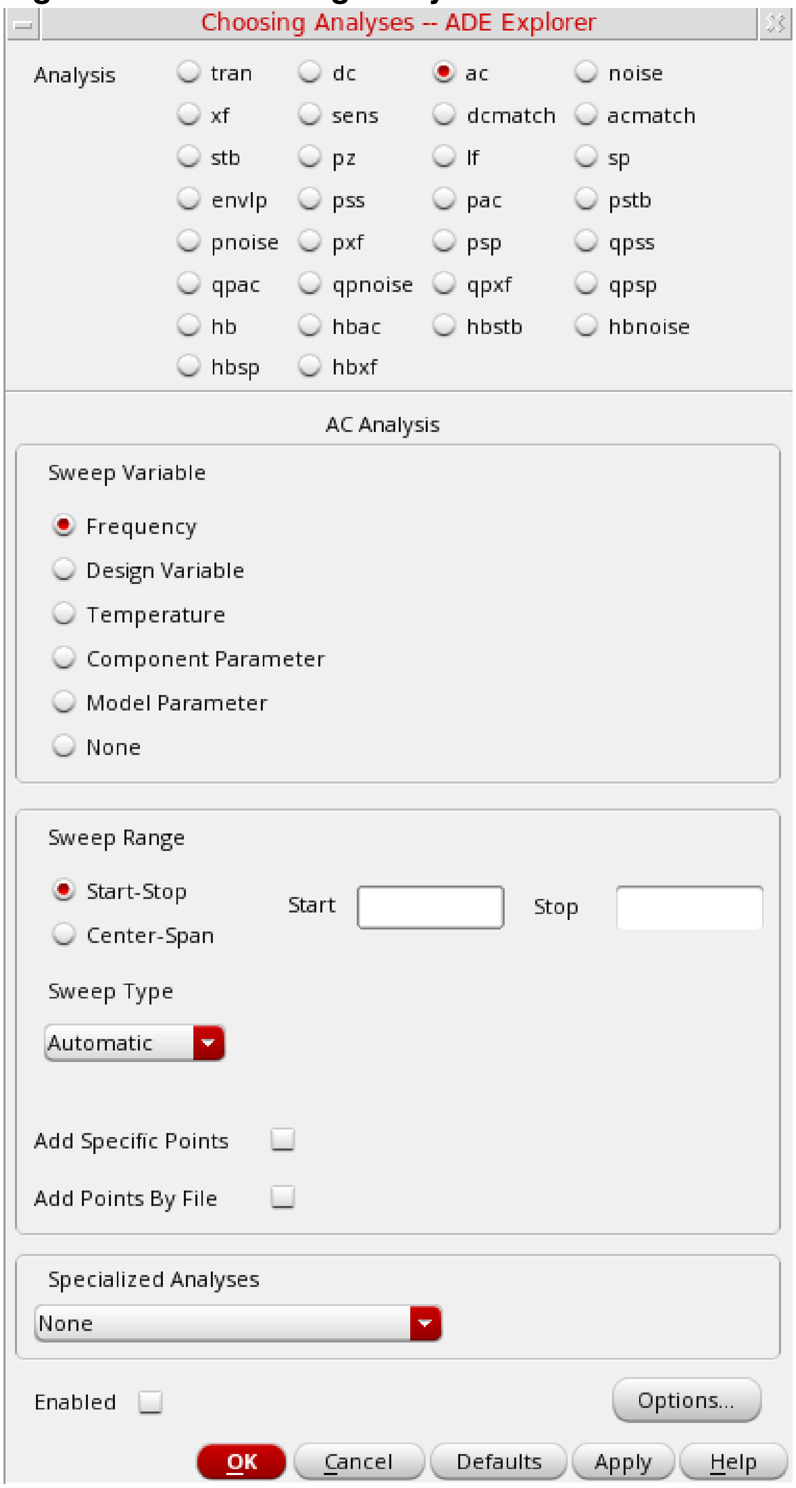


Measuring IP3 with Rapid IP3

1. In ADE Explorer select *Analyses* - *Choose*.

   The *Choosing Analyses* form is displayed.

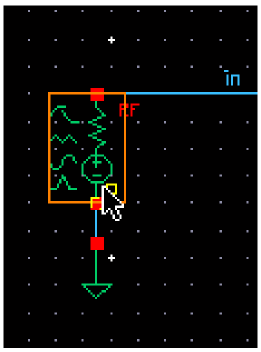
Figure 2-69 Choosing Analyses Form



2. Select *ac* from the *Analysis* section.
3. At the bottom of the form, set *Specialized Analysis* to *Rapid IP3*. The form expands.

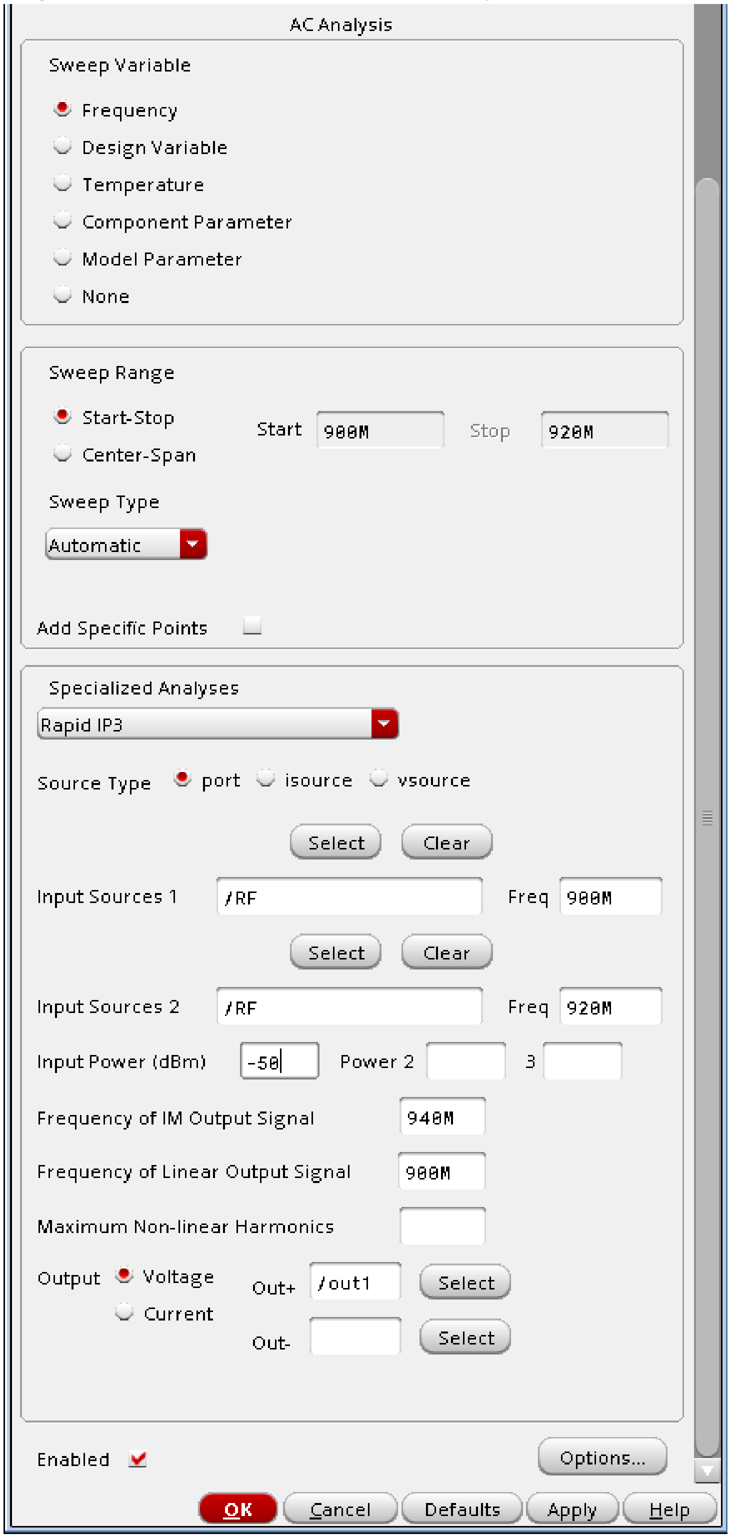
4. Select *port* for *Source Type*.
5. Select the *Input Sources 1* field.
6. Click the *Select* button just above that field.
7. Select the Port source /RF at the left of the schematic.

Figure 2-70 Select Input Port



8. Press the <Esc> key.
9. Type 900M in the *Freq* field to the right of *Input Sources 1*.
10. Type */RF* in the *Input Sources 2* field. (Alternately, you could select the RF port in the schematic by first clicking the *Select* button and then selecting the RF source in the schematic).
11. Type 920M in the *Freq* field the to the right of *Input Sources 2*.
12. Type -50 in the I*nput Power (dBm)* field.
13. Type 940M in the *Frequency of IM Output Signal* field.
14. Type 900M in the *Frequency of Linear Output Signal* field.
15. Type */out1* in the *Out+* field. Because you are leaving the *Out-* field blank, it will default to ground.
16. Leave the rest of the form at their default values. The *Choosing Analyses* form should look like the following:

Figure 2-71 Rapid IP3 AC Analysis Form



17. Click *OK*.

18. Start the simulation by choosing *Simulation - Netlist and Run* or by clicking the green arrow icon on the right side of the simulation window.

This netlists the design and runs the simulation. A SpectreRF status window appears (spectre.out logfile). Note the simulation time in the Spectre output logfile.

```
********************
IP3 measurement `ac'
********************
Input RF1 freq = 900 MHz
Input RF2 freq = 920 MHz
Output IM1 freq = 900 MHz
Output IM3 freq = 940 MHz

DC simulation time: CPU = 0 s, elapsed = 380.993 us.
Linear output:
f_out = f_in_1

IM3 output:
f_IM3 = 2 * f_in_2 - f_in_1
Accumulated DC solution time = 0 s.
Intrinsic ac analysis time =              20 ms.
Total time required for ac analysis `ac': CPU = 18.998 ms, elapsed = 203.026 ms.
Time accumulated: CPU = 118.981 ms, elapsed = 677.548 ms.
Peak resident memory used = 30 Mbytes.
```

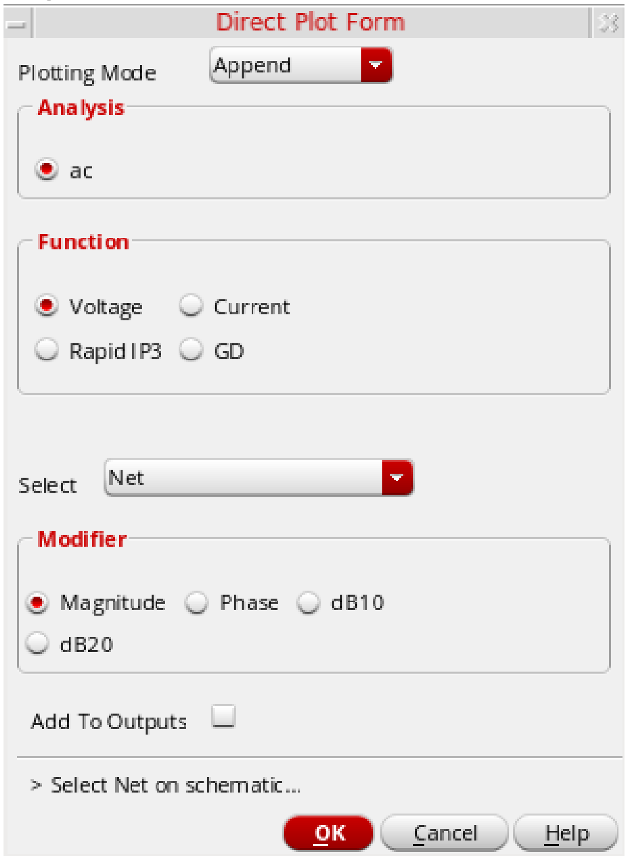
Note that the simulation time is faster than using two tone hb analysis to simulate rapid IP3.

19. When the analysis has completed, you may iconify the status window.

20. In ADE Explorer, select *Results - Direct Plot - Main Form*. Alternately, you can press the *Direct Plot* icon from the schematic window.

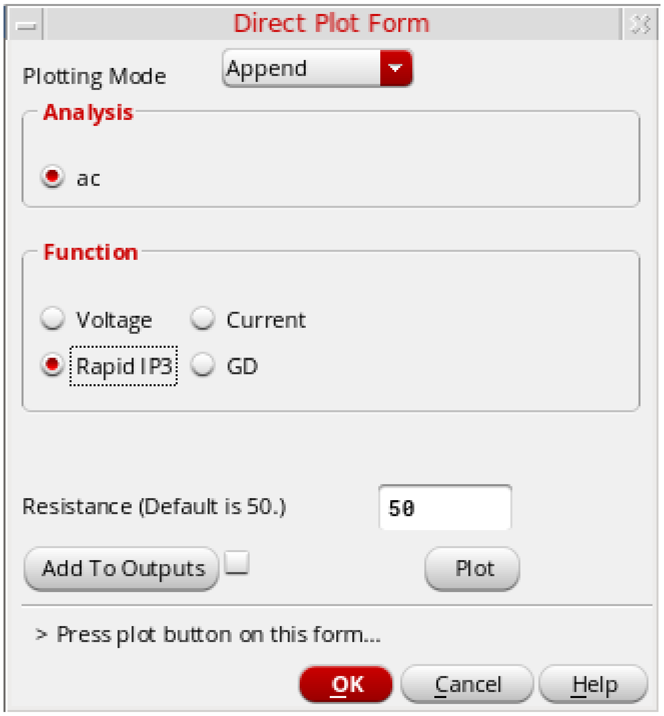
The *Direct Plot Form* is displayed.

Figure 2-72 AC Direct Plot Form



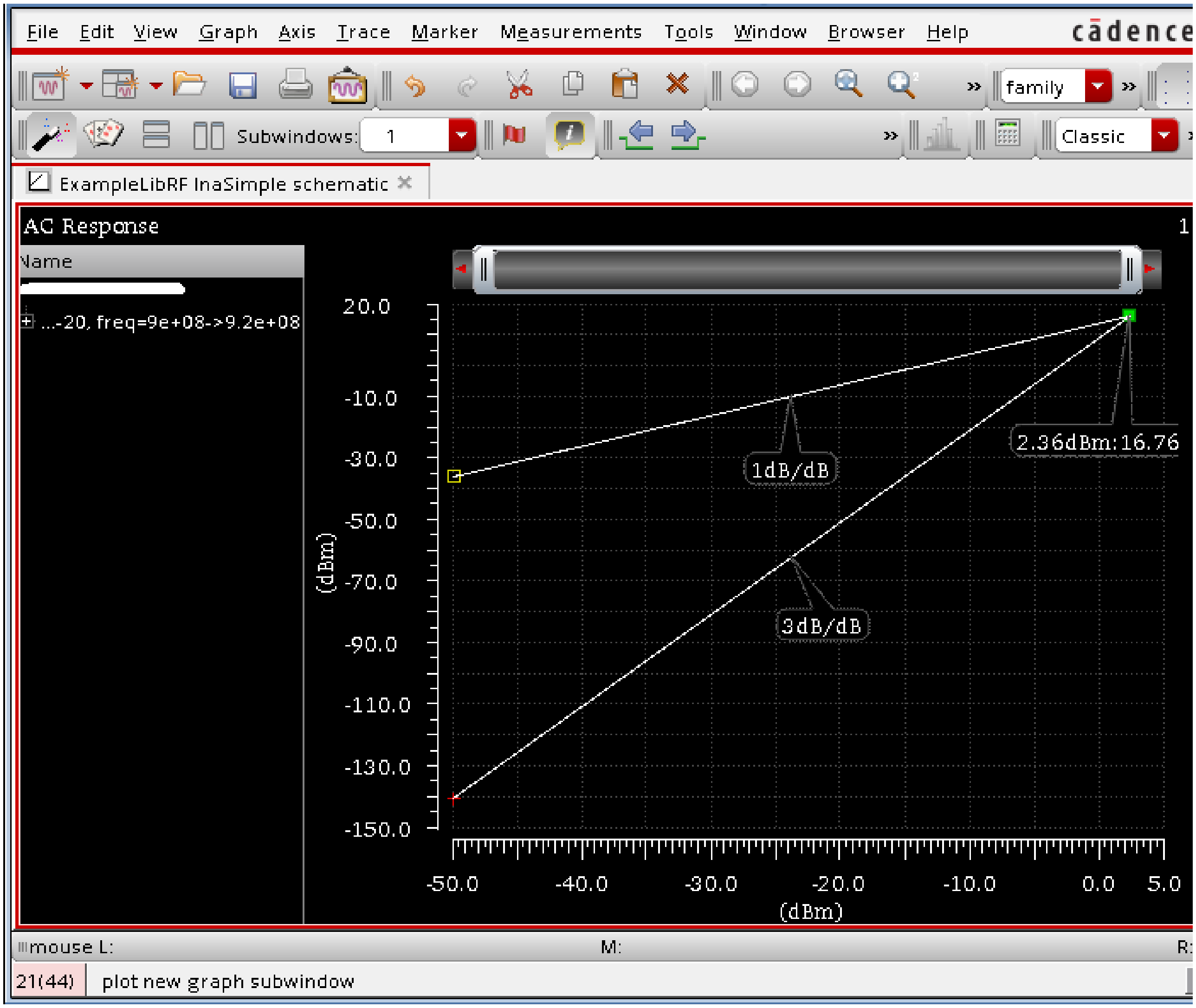
21. In the *Direct Plot Form*, select *Rapid IP3*. The form changes.

Figure 2-73 AC Direct Plot Form - Rapid IP3



22. Leave the rest of the form at the default values. Click *Plot*. The IP3 plot is displayed.

Figure 2-74 IP3 Plot Using Rapid IP3



Note that the value of Rapid IP3 (2.36dBm) is very close to that measured by two toned hb analysis (2.30dBm). Although simulating using both methods was relatively quick, rapid IP3 was faster.

In this section you measured IP3 on a low noise amplifier using two methods: two tone harmonic balance and rapid IP3 using AC analysis. Both gave good accuracy but rapid IP3 was faster.

Summary

The LNA section of Appendix A shows how to simulate and make typical measurements on a low noise amplifier

In the LNA section, the following measurements were shown:

- S-Parameter Analysis, Gain
- Stability, Stability Circles
- Linear 2 port Noise Figure measurements, Noise Circles
- Third Order Intercept measurement using 2 tone harmonic balance
- Rapid IP3 using specialized ac analysis

For more information on simulating low noise amplifiers, please refer to the chapters in the *Virtuoso Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide* and the *Spectre Circuit Simulator RF Analysis Theory* guide.

3

Simulating Oscillators

Oscillators, which are autonomous circuits, are time-invariant circuits with time-varying responses. Thus, autonomous circuits generate non-constant waveforms even though they are not driven by a time-varying stimulus.

You cannot specify the analysis period for autonomous circuits because you do not know the precise oscillation period in advance of the simulation. Instead, you specify an estimate of the oscillation period for the simulation. The PSS analysis uses your estimate to compute the precise period and the periodic solution waveforms.

In the first section of this topic we will concentrate on simulating oscillators as autonomous circuits using mainly Harmonic Balance (HB) method. In the second section we will simulate and do measurements on Ring Oscillator using Shooting method.

Note that there are oscillators like injection locked ring oscillators which are simulated as driven circuits instead of autonomous circuits. However, this class of oscillators will not be discussed.

You will do the following measurements while doing the Oscillator simulation -

Measurements	**Analysis**
Starting and Stabilizing Feedback Oscillators	PSS - HB
Oscillator Loop Gain Measurement	PSS - Shooting, Pstb and stb
Phase Noise Measurement and Noise Summary Table	HB/HBnoise
Oscillator Swept Tuning Range and Phase Noise Measurement	HB/HBnoise
Ring Oscillator Measurements	
Starting and Stabilization of Ring Oscillators	PSS - Shooting/tstab
FM Jitter Measurement using PSS Shooting and Pnoise Jitter Analyses	PSS - Shooting/Pnoise

Measurements	Analysis
Calculating the Swept Tuning Range and Phase Noise for the Ring Oscillator	PSS - Shooting/Pnoise

Simulation Methods

There are two methods you can use to simulate oscillators using SpectreRF:

- Harmonic Balance (HB)
- Shooting Newton

The Harmonic Balance method is recommended for simulating mildly-nonlinear oscillators with resonators, such as LC oscillators and crystal oscillators. Shooting PSS method is recommended for simulating strongly nonlinear resonatorless oscillators, such as ring oscillators or relaxation oscillators.

Phases of Autonomous PSS/HB Analysis

A PSS/HB analysis has two or three phases.

1. If the *Calculate initial conditions (ic) automatically* check box is selected, then after the DC analysis for the time-zero timepoint has been calculated, the oscillator frequency and amplitude are estimated. The beginning of the transient analysis in the tstab interval includes this estimate.
2. A transient analysis phase to initialize the circuit.

   The transient analysis phase is divided into three intervals:

   a. A beginning interval that starts at *tstart*, which is normally 0, and continues through the onset of periodicity for the independent sources and continues through the longest delay time of the periodic sources, or the last point in a PWL waveform, whichever is the longest.

   b. A stabilization interval of length *tstab*

      In the hb *Choosing Analyses* form, you have the option to have Spectre automatically calculate the length needed for tstab.

      For driven circuits, the stabilization interval is optional. For autonomous circuits, *tstab* needs to be set to an interval that provides the circuit a good initial condition.

   c. A final interval that is four times the estimated oscillation period specified in the PSS/HB *Choosing Analyses* form. During the final interval, the PSS/HB analysis monitors the waveforms in the circuit and improves the estimate of the oscillation period. During this final interval, Spectre looks for frequency divider outputs in the circuit.
3. A Shooting or Harmonic Balance phase to compute the periodic steady state solution

During this phase, the circuit is iterated repeatedly over one period. The length of the period and the initial conditions are modified to find the steady state solution.

See *Oscillators and Autonomous PSS Analysis* in *Spectre Circuit Simulator RF Analysis Theory* for more information on the autonomous PSS analysis algorithm.

Phase Noise and Oscillators

Oscillators tend to amplify any noise present near the oscillation frequency. The closer the noise frequency is to the oscillation frequency, the greater the noise amplification. Noise amplified by the oscillator in this manner is called *phase noise*. Phase noise is the most significant source of noise in oscillators, and because phase noise is centered about the oscillation frequency, filtering can never completely remove it.

You can understand phase noise if you recognize that the phase of an oscillator is arbitrary because there is no drive signal to lock to. Any waveform that is a solution to an oscillator can be shifted in time and still be a solution. If a perturbation disturbs the phase, nothing restores the phase, so it drifts without bound. If the perturbation is random noise, the drift is a random walk. In addition, the closer the perturbation frequency is to the oscillation frequency, the better it couples to the phase and the greater the drift. The perturbation need not come from random noise. Noise might also couple into the oscillator from other sources, such as the power supplies.

Starting and Stabilizing Feedback Oscillators

To simulate an oscillator using either HB or Shooting PSS analysis, you must first start the oscillator by supplying either of the following:

1. A brief impulse stimulus

   The stimulus should couple strongly into the oscillatory mode of the circuit and poorly into other long-lasting modes such as bias circuitry. This is usually done by adding a single pulse of current that is connected somewhere in the circuit to start the oscillations.

2. A set of initial conditions (ICs) for the components of the oscillator's resonator.

3. For LC and High Q oscillators, you can select the *Calculate initial conditions (ic) automatically* checkbox in the *Choosing Analyses* form instead of providing any stimulus or initial condition (*ic*).

   This setting tells Spectre to do a variation of the linear stability analysis just before the beginning of the tstab interval. It provides a good estimate of the oscillation frequency amplitude and phase at the beginning of the tstab interval. With a start that is very close, convergence is much easier.

Note that when you set this option then ICs (initial conditions) are automatically ignored.

Regardless of which technique you use to start the oscillator, allow the oscillator to run long enough to stabilize before you start the HB or Shooting phase and compute the steady state solution. Adjust the *tstab* parameter to supply the additional stabilization period. High Q oscillators often need a longer tstab. For extremely high Q circuits, you may need to set tstab to about 50K to 200K periods of oscillation with the default method. However, when setting *Calculate initial conditions (ic) automatically* option you may reduce it to just 10 periods.

In HB, if you set *Run transient* to *Decide automatically* then you do not need to provide any tstab value and the simulator automatically decides how long to run the transient analysis to compute the steady state solution. Also, in HB, while setting the tstab value, you can select the *Detect Steady State* checkbox if you want the transient analysis to stop automatically as soon as the steady state is reached rather than runningthe transient analysis for the whole tstab value.

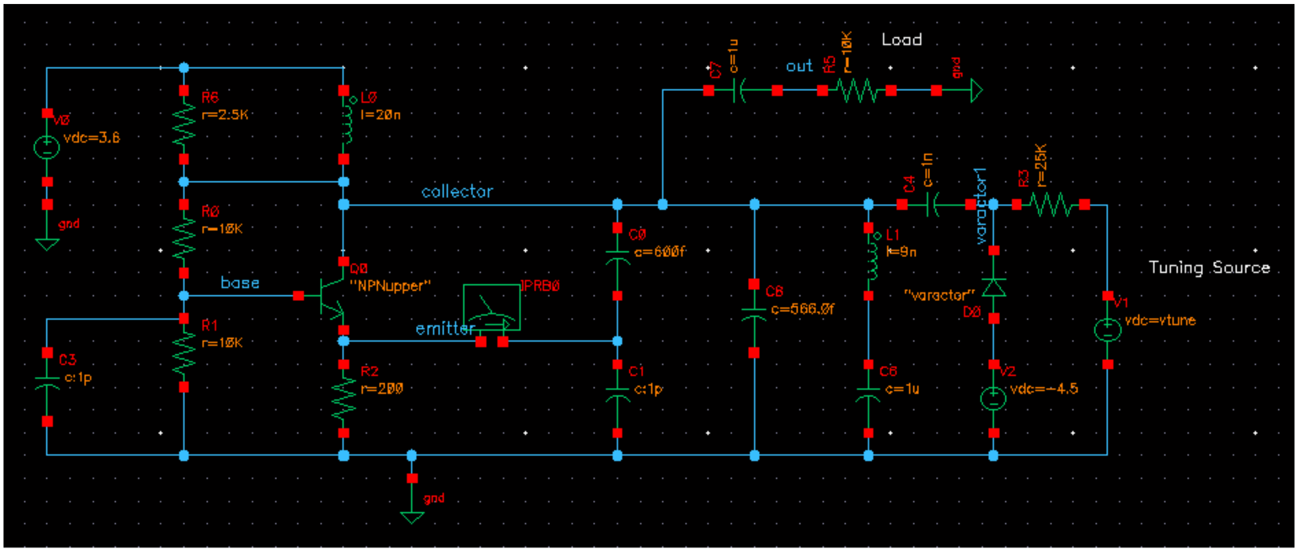
It is recommended to save the initial transient simulation results. This can be done by setting the *Save Initial Transient Results (saveinit)* option to *yes*. You can use this to verify that the oscillator has started up and is stable.

See *Oscillators and Autonomous PSS Analysis* in the *Spectre Circuit Simulator RF Analysis Theory* for more information on oscillator simulation. In addition to this you may also refer to *Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide*.

The Oscillator Circuit

This example computes the periodic steady state solution and the phase noise for the *RF oscillator* circuit, as shown in Figure 3-1.

Figure 3-1 Schematic for the Oscillator Circuit oscillator_ckt

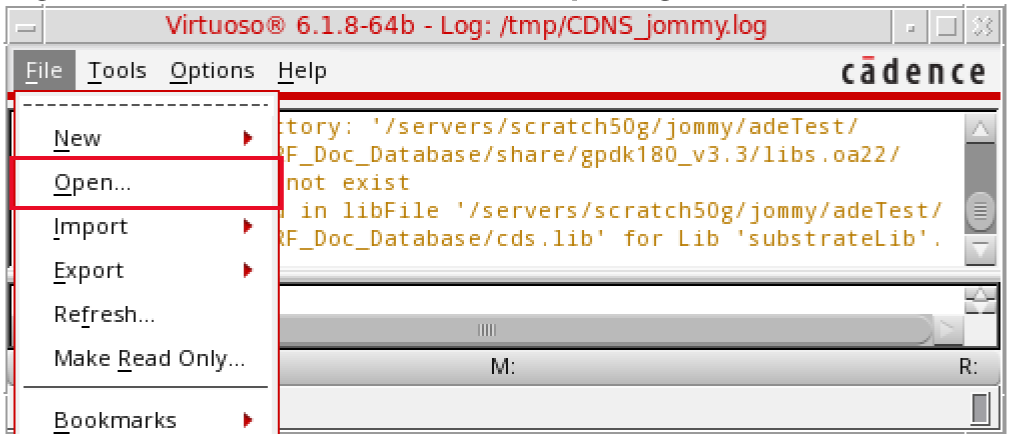


Setting Up to Simulate the Oscillator Circuit

Opening the Oscillator Circuit in the Schematic Window

1. In the Command Interpreter Window (CIW), choose *File – Open*.

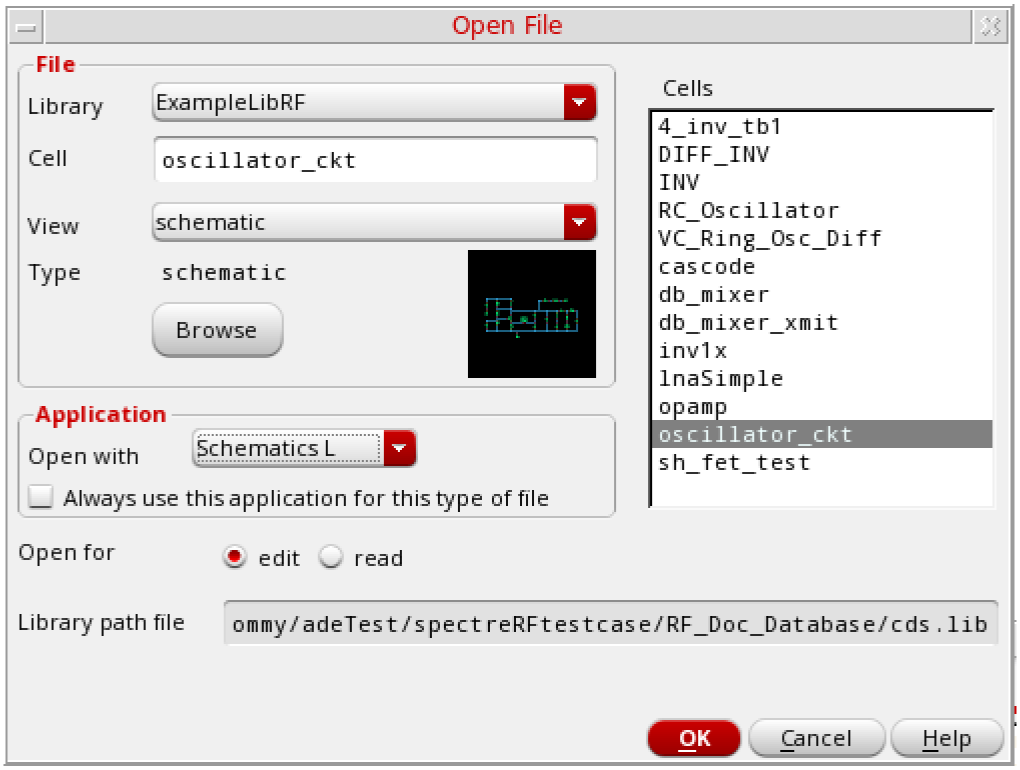
Figure 3-2 Virtuoso CIW Window - Opening Cellview



The *Open File* form is displayed.

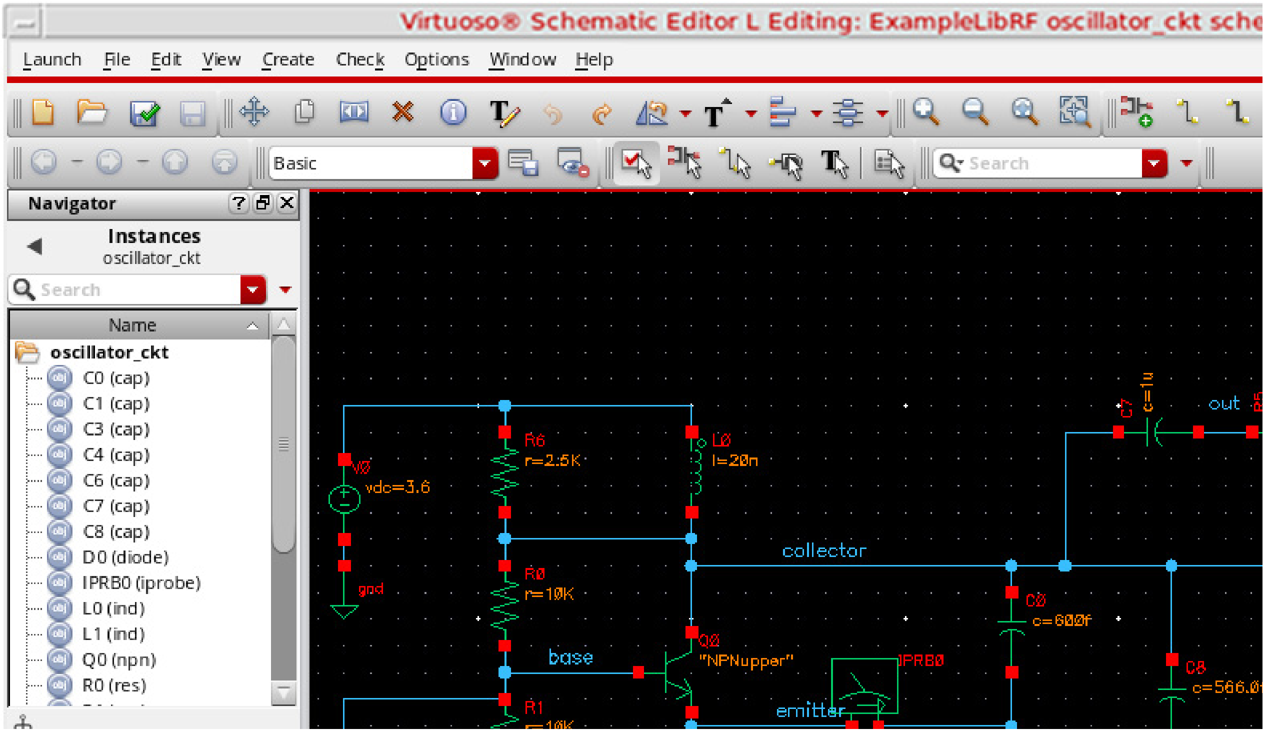
2. Select *ExampleLibRF* from the *Library* drop-down list.
3. In the *Cells* list box, choose *oscillator_ckt*.
4. Select *schematic* from the *View drop-down list*.
5. In the *Application* section, select *Schematic-L* from the *Open with* drop-down list.
6. Leave *Open For* to *edit* (which is set by default).

Figure 3-3 Open File Form to open the oscillator_ckt cell's Schematic View



7. Click *OK* to close the Open File form.
8. This will open *oscillator_ckt* schematic in Virtuoso Schematic Editor L window, as shown below.

Figure 3-4 oscillator_ckt schematic in VSE-L Window

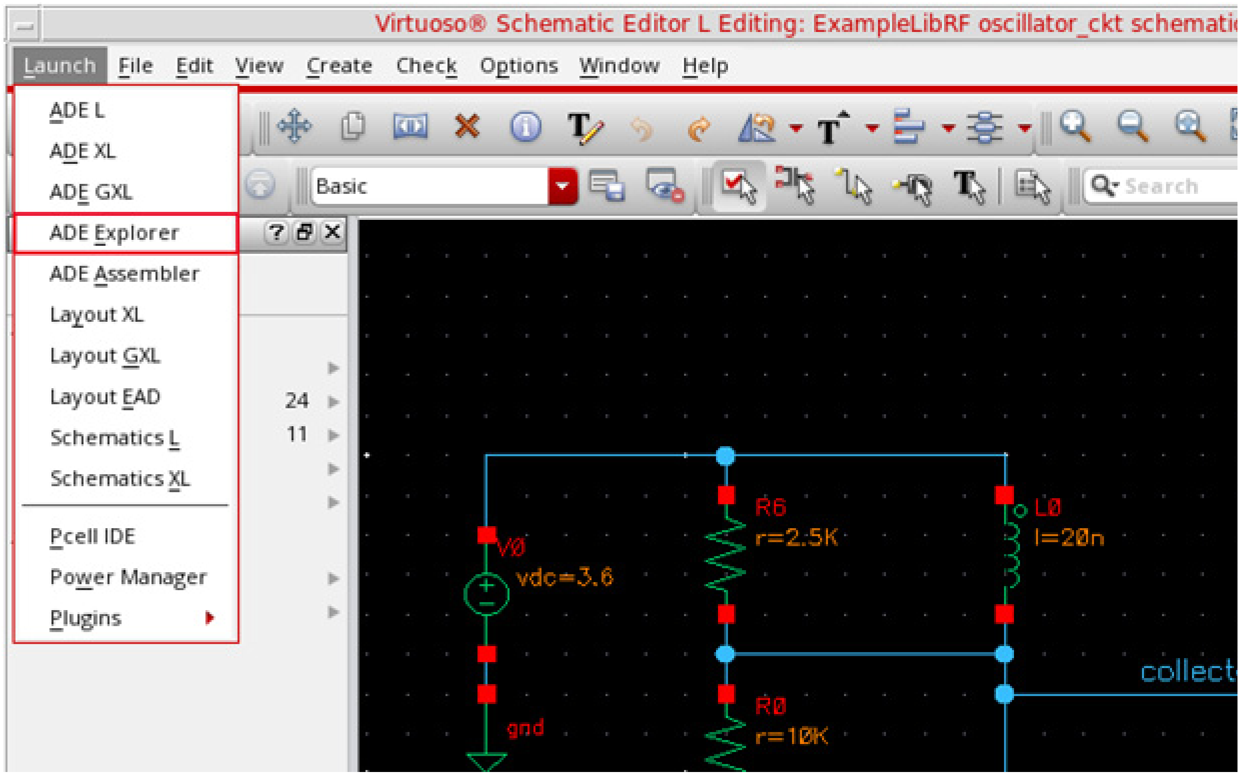


This example computes the periodic steady state solution for the *oscillator_ckt RF* oscillator circuit. You perform a PSS-HB or HB analysis first because the periodic steady state solution must be determined before you can perform any other periodic small-signal analysis like *pstb* or *hbnoise* etc. to determine the stability or phase noise.

Setting up ADE Explorer for Oscillator Simulation

1. In the Schematic Window, choose *Launch - ADE Explorer.*

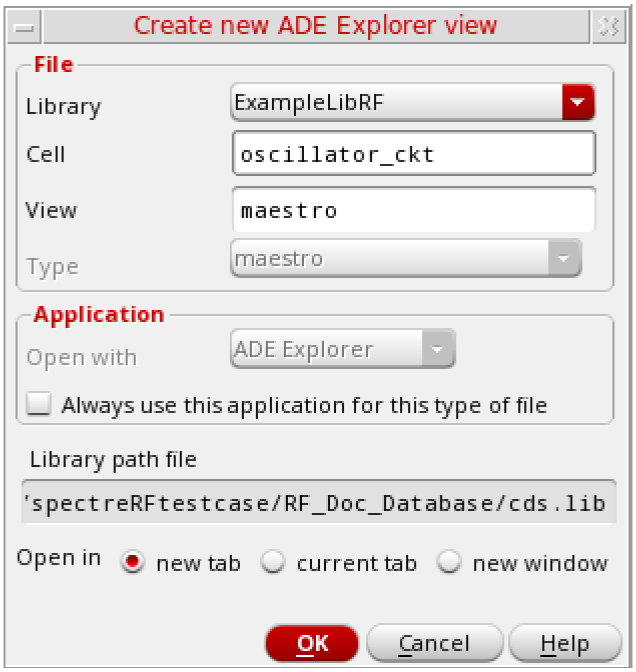
Figure 3-5 Opening ADE Explorer from VSE L window



2. In the *Launch ADE Explorer* dialog, select *Create New View*.

   The *Create new ADE Explorer view* form is displayed.

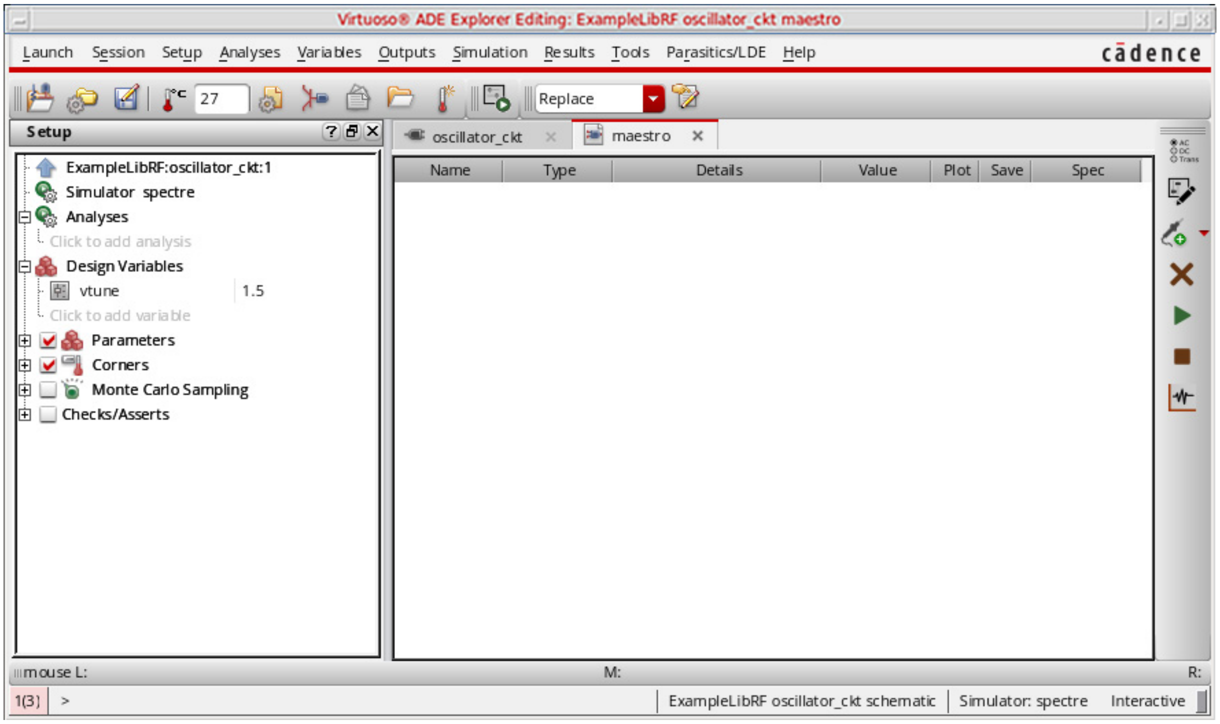
Figure 3-6 Create new ADE Explorer view



3. Leave each option to the default selections and click *OK*.

   The Virtuoso ADE Explorer window is displayed, as shown below:

Figure 3-7 Virtuoso ADE Explorer Window



4. Choose *Setup – Simulator* in ADE Explorer.

   The *Choosing Simulator* form appears.

5. Choose *spectre* for the *Simulator*.

Figure 3-8 Choosing Simulator Form

6. Click *OK*.

7. Set up the High Performance Simulation Options.

   In the ADE window, choose *Setup - High Performance Simulation*. The *High Performance Simulation Options* window is displayed.

   In the *High Performance Simulation Options* window, select *APS*. Note that *Auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores. Usually, it is better to specify the number of threads yourself. Small circuits should use a small number of threads, and larger circuits can use more threads. The overhead of managing 16 threads on a smaller circuit may actually slow the simulation down, compared to two or four threads. Use the number of threads that maximizes the performance.

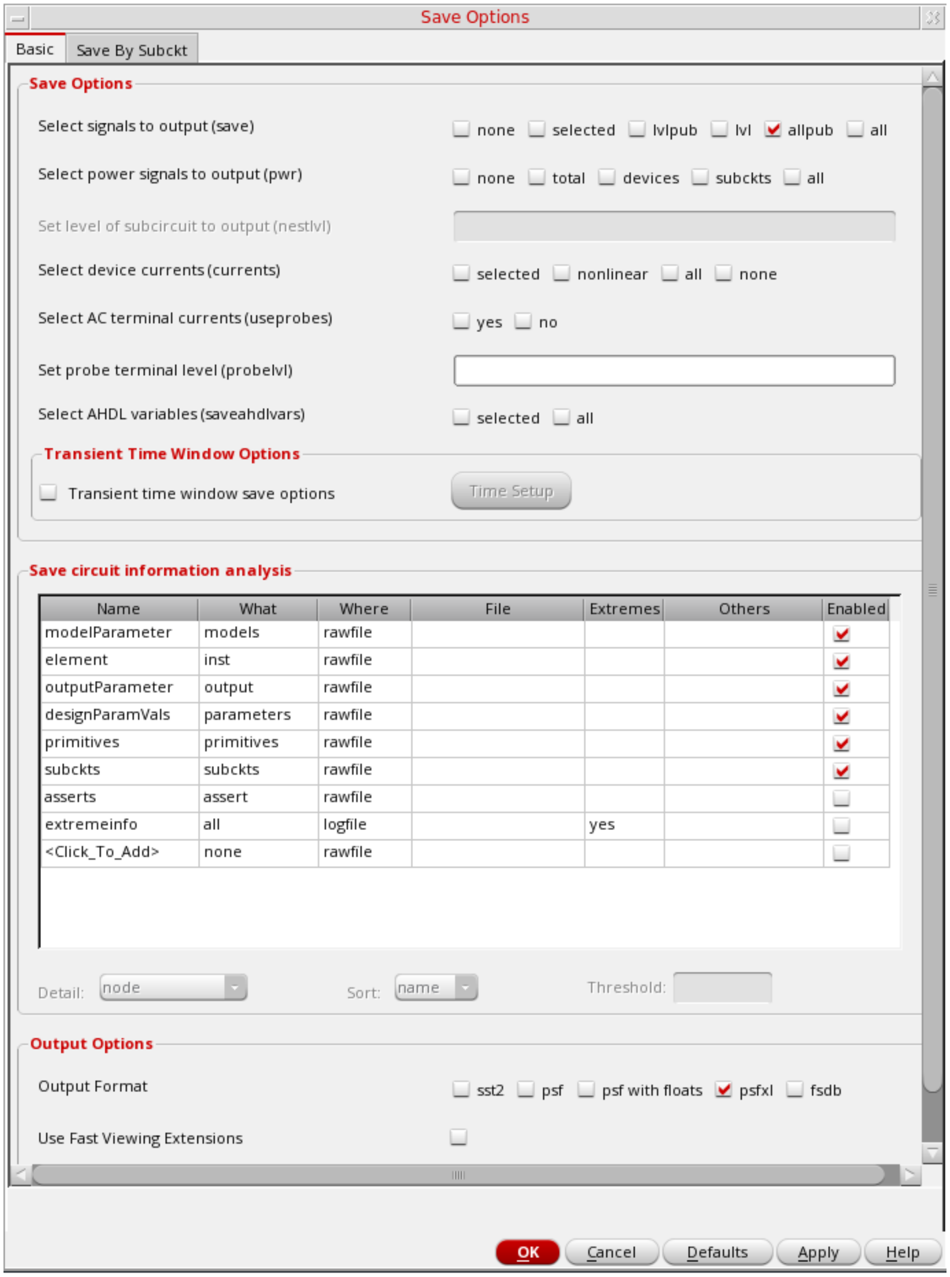
Figure 3-9 High Performance Simulation Options Form

8. Click *OK*.

9. Select *Outputs – Save All*.

   The *Save Options* form is displayed, as shown below.

Figure 3-10 Save Options Form



10. In the *Select signals to output* section, ensure *allpub* is highlighted.

This is the default which saves all node voltages at all levels of hierarchy, but it does not include the node voltages inside the device models.

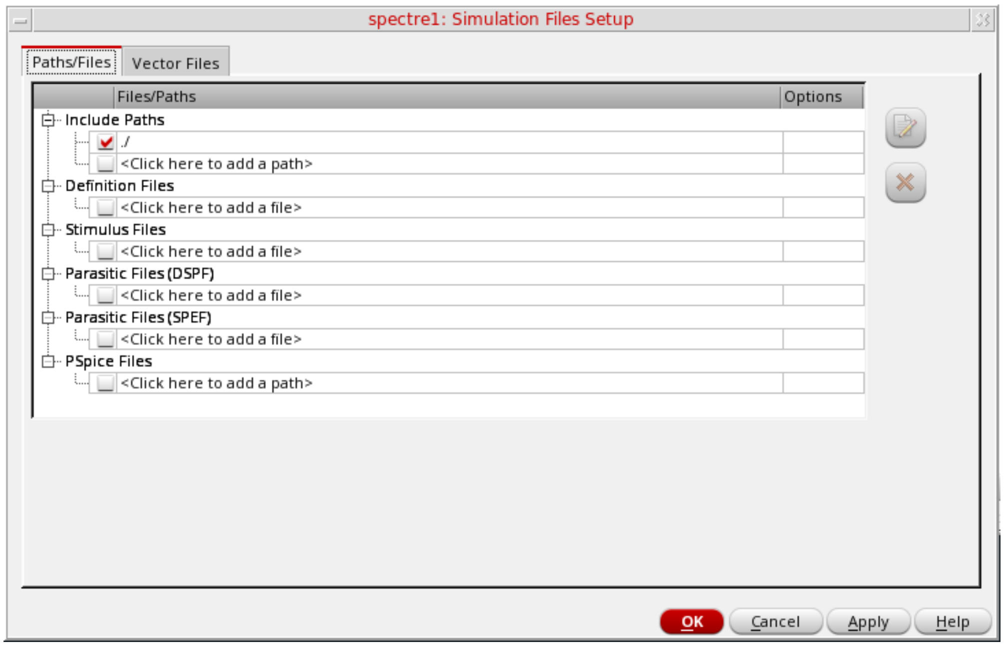
11. To save the currents, choose *nonlinear* for the *Select device currents (currents)* option, if you just want to save the device currents, or choose *all* if you want to save all the currents in the circuit. When you save the currents, more disk space is required for the results file.

12. Click *OK*.

13. Select *Setup - Simulation Files.*

The *Simulation Files Setup* form is displayed.

Figure 3-11 ***Simulation Files Setup* Form**



14. Select *./* by clicking in the *Include Paths* section.

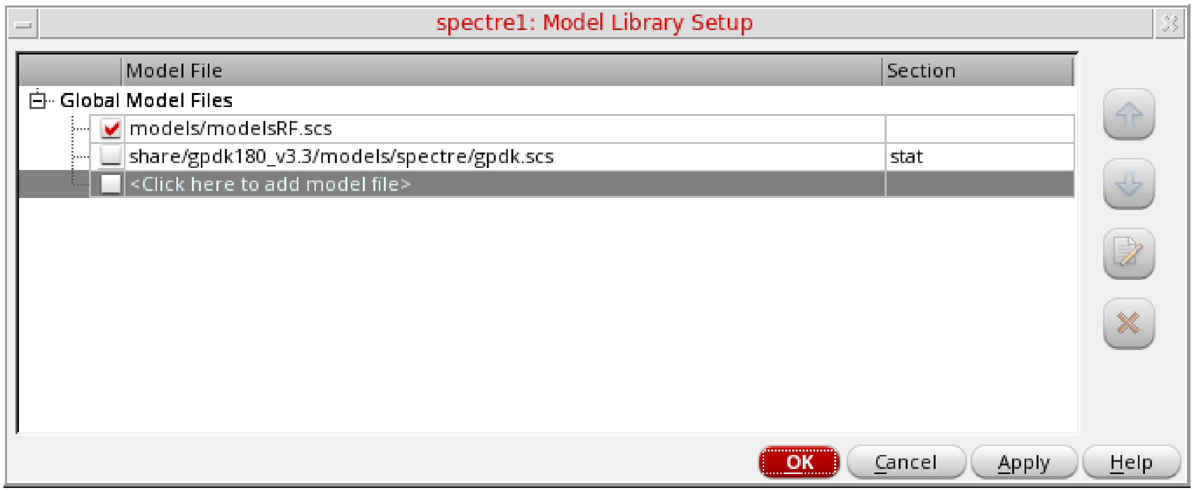
15. Click *OK* to close the *Simulation Files Setup* form.

16. In ADE Explorer, select *Setup – Model Libraries*

The *Model Library Setup* form is displayed.

17. In the *Model File* field, type the path to the model file including the file name, such as *models/modelsRF.scs*.

Figure 3-12 ***Model Library Setup*** **Form**



You can also browse to *modelsRF.scs* file.

18. Click *OK.*

Calculating the Steady-State Solution using PSS Harmonic Balance

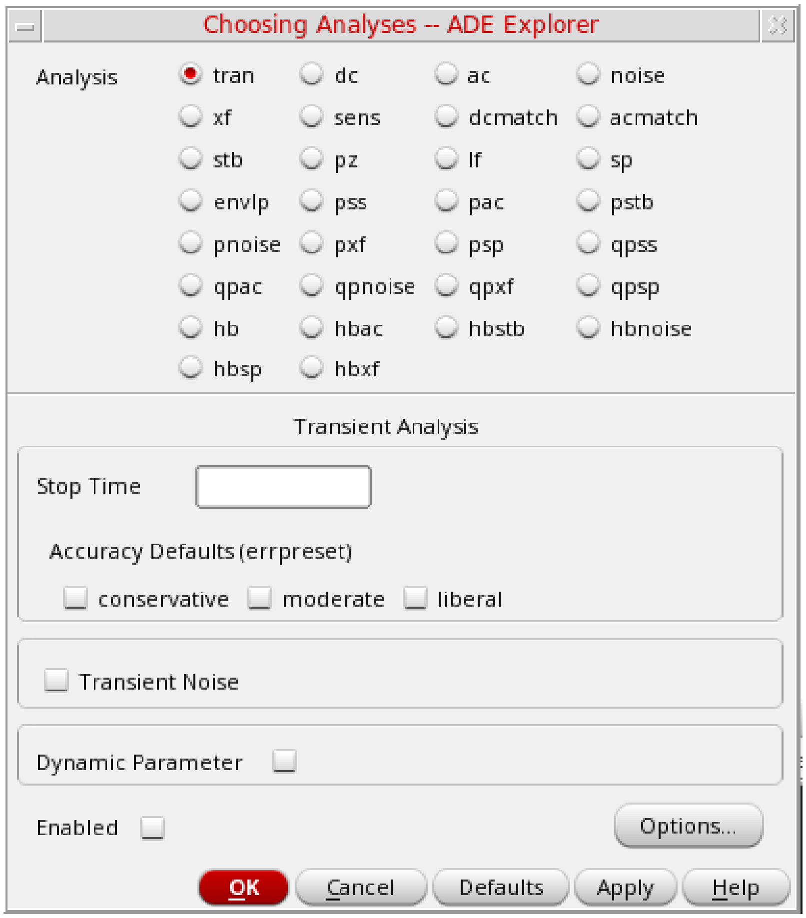
In this very first measurement, you will set up the PSS analysis and then determine the oscillation frequency of the oscillator. You will use the *Harmonic Balance* Engine in the PSS Analysis. You can also use *hb* analysis directly. Here, we have chosen PSS analysis with *Harmonic Balance* Engine because you will perform *pstb* analysis in the next measurement. Since there is no corresponding hbstb analysis available for hb analysis, this setup is done.

Setting up the PSS Analysis

1. Select *Analyses –> Choose* in the Virtuoso Analog Design Environment window.

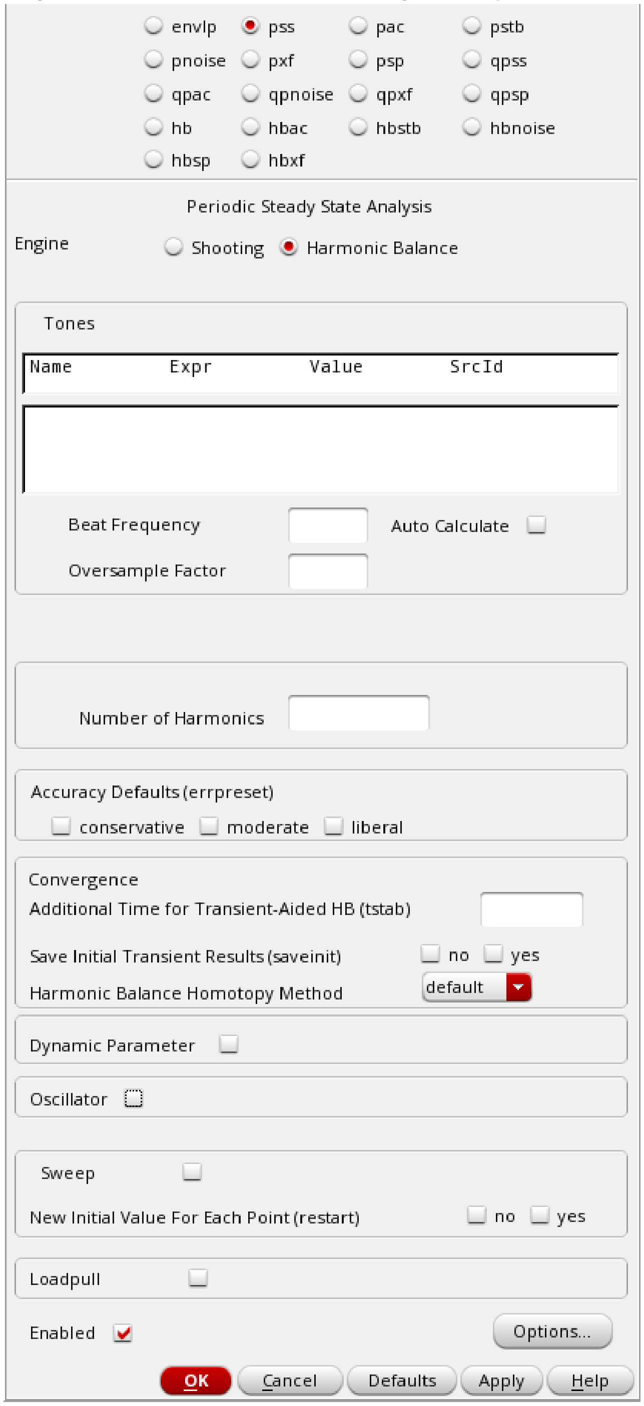
   The *Choosing Analyses* form is displayed, as shown below.

Figure 3-13 The *Choosing Analyses* Form



a. Select *pss* as Analysis. The form expands.

Figure 3-14 The Choosing Analyses Form- Setting PSS Analysis



b. Select *Harmonic Balance* as Engine.

c. In the *Beat Frequency* field, type 1.9G. The frequency entered here is an approximate frequency of oscillation.

d. Leave Oversample Factor (*oversamplefactor*) blank, which means that the Oversample Factor is equal to 1. Since the oscillator has sinusoidal waveforms, an oversample of 1 is appropriate.

e. In the *Number of harmonics* field, type 15.

In general, you want to choose a number that is high enough to capture the nonlinearity of the circuit. Start with 10, and run the simulation. Increase by about 50% to 15 and re-run the simulation. If the harmonics do not change appreciably, then 10 is enough. If they change, increase the number again by about 50%. Use the smallest number of harmonics for the answer to be stable.

f. In the *Accuracy Defaults* (*errpreset*) section, select *conservative*. *conservative* is typically used because very small amplitude phase noise measurements are normally desired. Conservative is recommended for all the oscillators.

g. Type 5n in the *Additional Time for Transient-Aided HB (tstab)* field. *tstab* is typically set to about 10 to 20 periods of the oscillation frequency when the *Calculate initial conditions (ic) automatically* checkbox is selected.

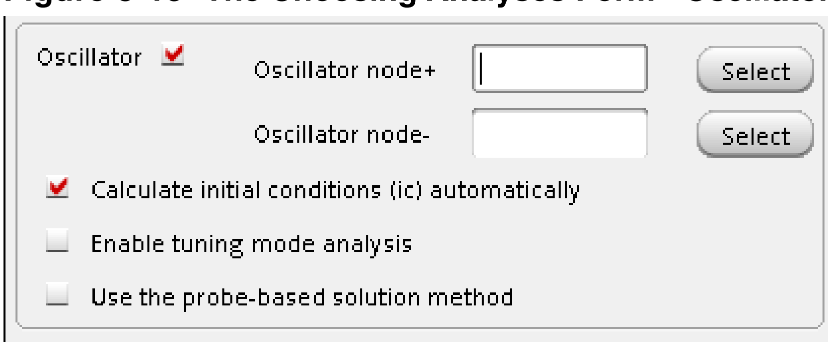
h. Select *yes* for *Save Initial Transient Results*. This will help in visualizing the buildup of the oscillation waveform.

i. Leave the *Harmonic Balance Homotopy Method* as *default* which is *tone*. HbHomotopy (*hbhomotopy*) is one of the HB Convergence Options which helps in the convergence of the simulation of circuit.

You can refer to the earlier chapters for more details.

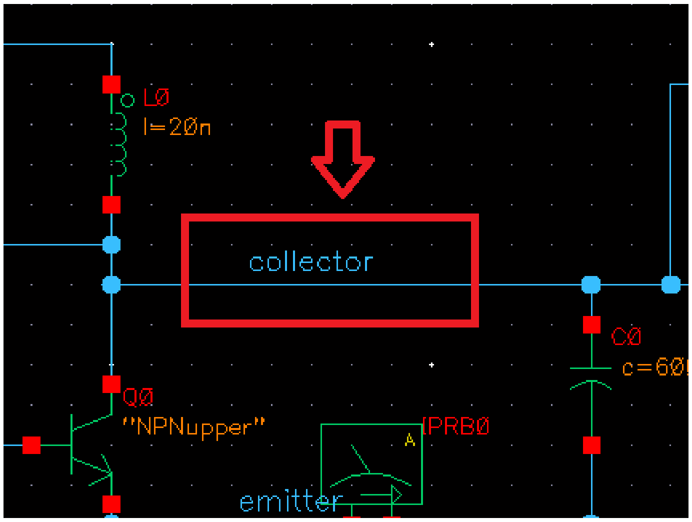
j. Select *Oscillator*. This is required for simulating an autonomous circuit.

Figure 3-15 The Choosing Analyses Form - Oscillator Section



k. In the *Oscillator node* field, click *Select* just to the right. In the schematic, select the *collector* node. This oscillator node will be used by the simulator for the period calculation of the oscillations. It just needs to be a node that has the oscillator signal on it.

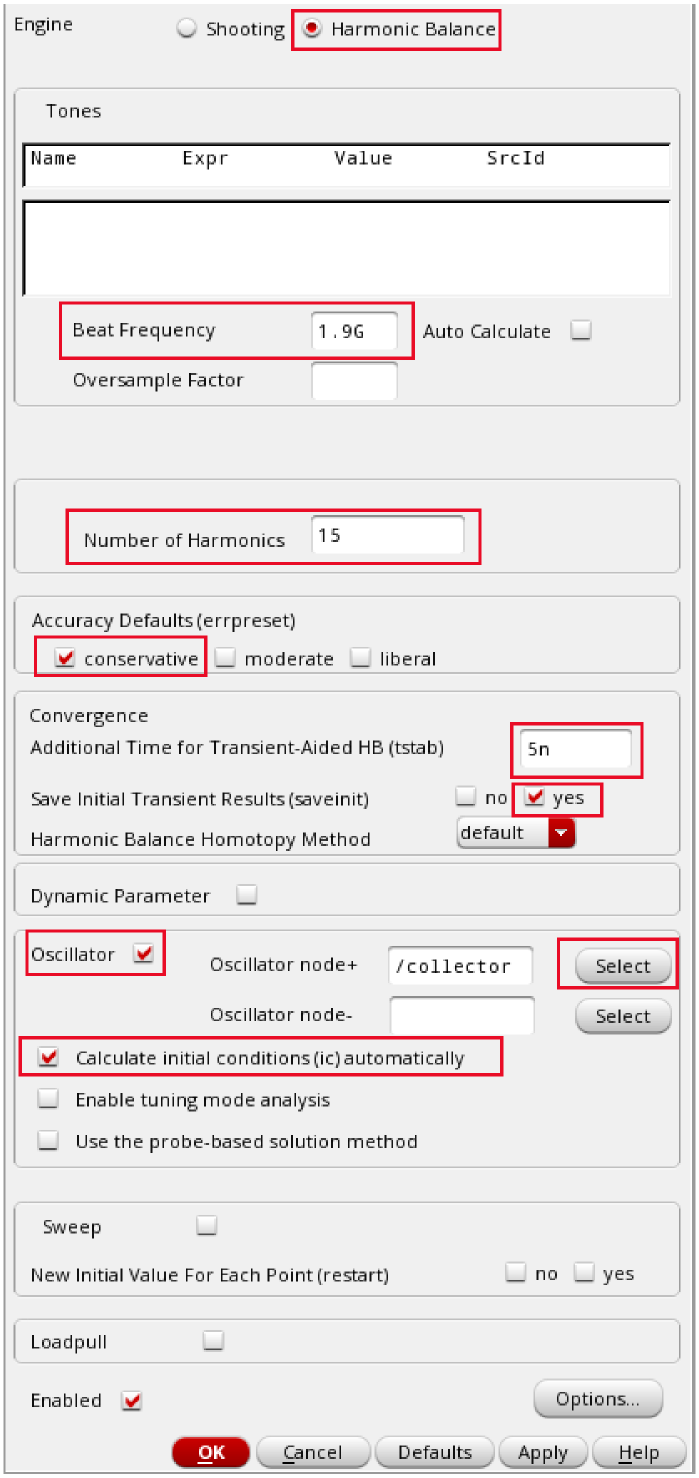
Figure 3-16 Selecting *collector* net on oscillator_ckt schematic



I. If you have an LC oscillator, leave the *Calculate initial conditions (ic) automatically* checkbox selected (this is the default).

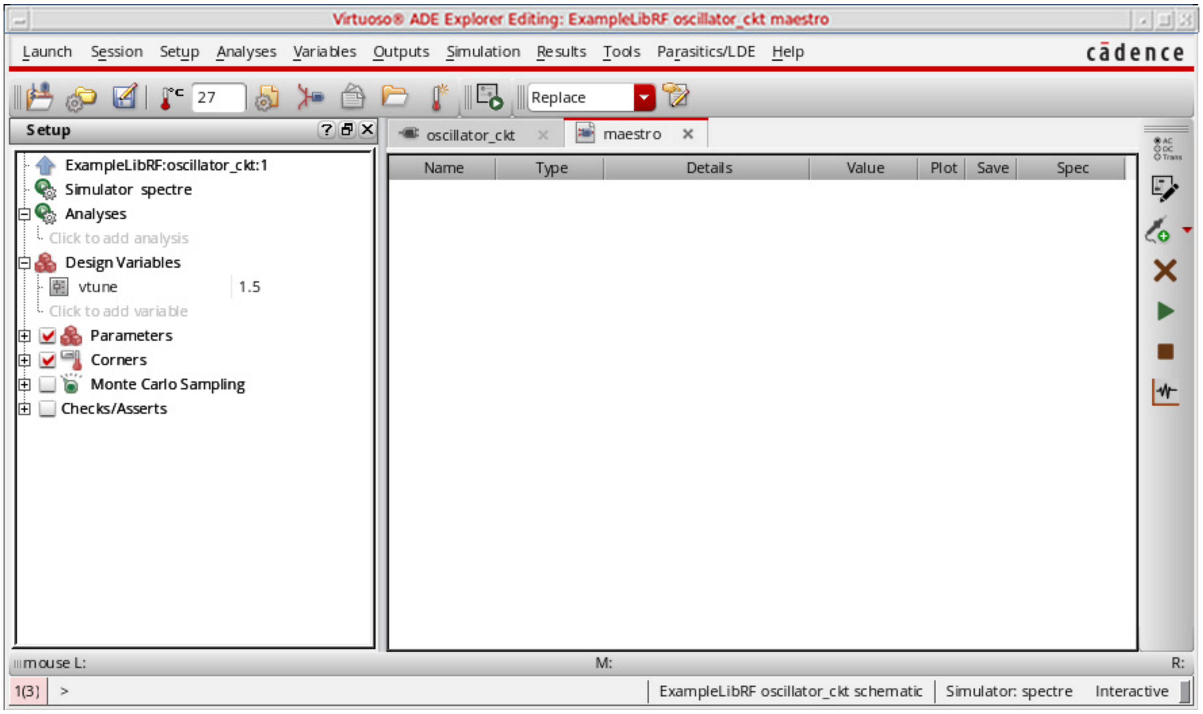
Note that *Calculate initial conditions (ic) automatically* is used to start the oscillator. Other methods to start the oscillator include putting a single current pulse into the resonator, setting initial conditions(ic), or ramping up the power at time = zero plus.

Figure 3-17 Choosing Analyses Form - PSS-Harmonic Balance Setup

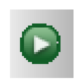


Now click *OK* to close the Choosing Analyses Form and add the *pss* analysis in the *Analyses* section of ADE Explorer as shown below:

Figure 3-18 ADE Explorer Simulation Window - PSS Analysis



Running the PSS analysis

Once finished setting up the PSS analysis click the green icon  on the right hand side of the ADE Explorer window or on the Schematic window to run the simulation.

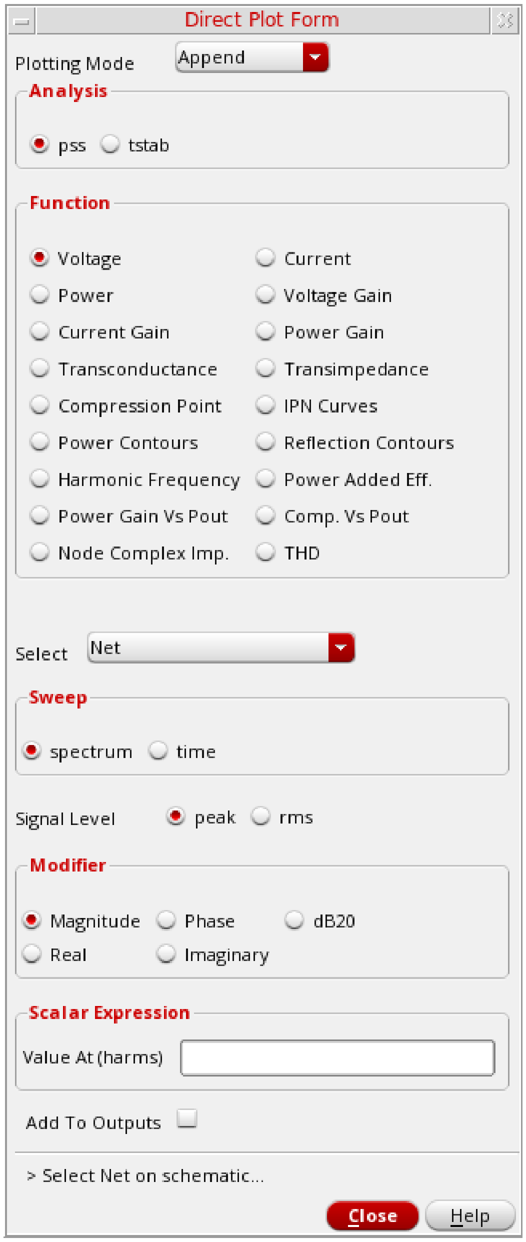
This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` logfile). When the analysis has completed, you may iconify the status window.

Next, you will plot the results.

Plotting the PSS Analysis Results

In ADE Explorer, select *Results - Direct Plot - Main Form*. The *Direct Plot Form* window is displayed.

Figure 3-19 The Direct Plot Form Window

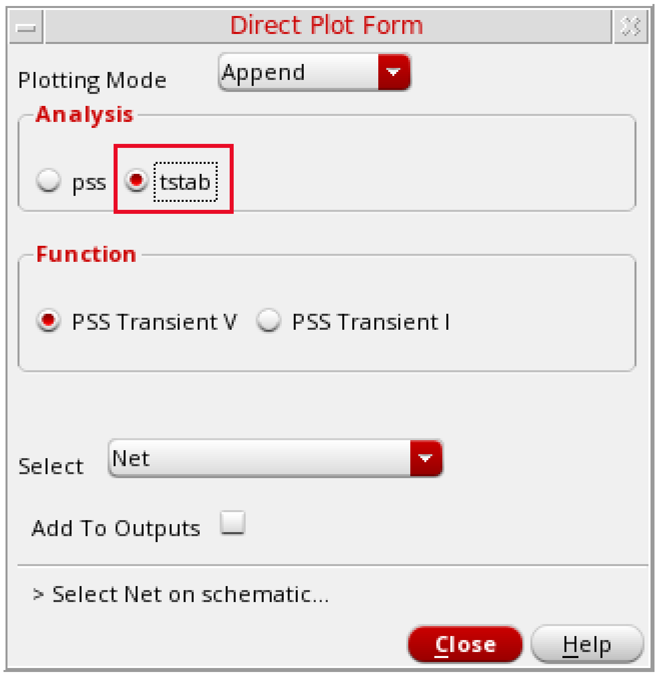


Plot the oscillator startup waveform from tstab run.

1. In the *Direct Plot Form* window, select *tstab* in the *Analysis* section.
2. Leave *Function* as *PSS Transient V* which is set by default.
3. Select *Net* in the center of the form. (This is the default. You can also select differential nets).

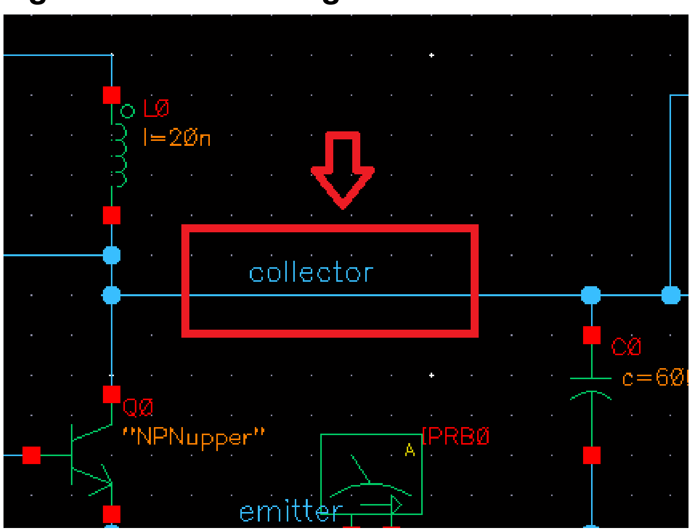
4. The *Direct Plot Form* window should look like the following:

Figure 3-20 PSS Analysis Direct Plot Setup - Initial Transient



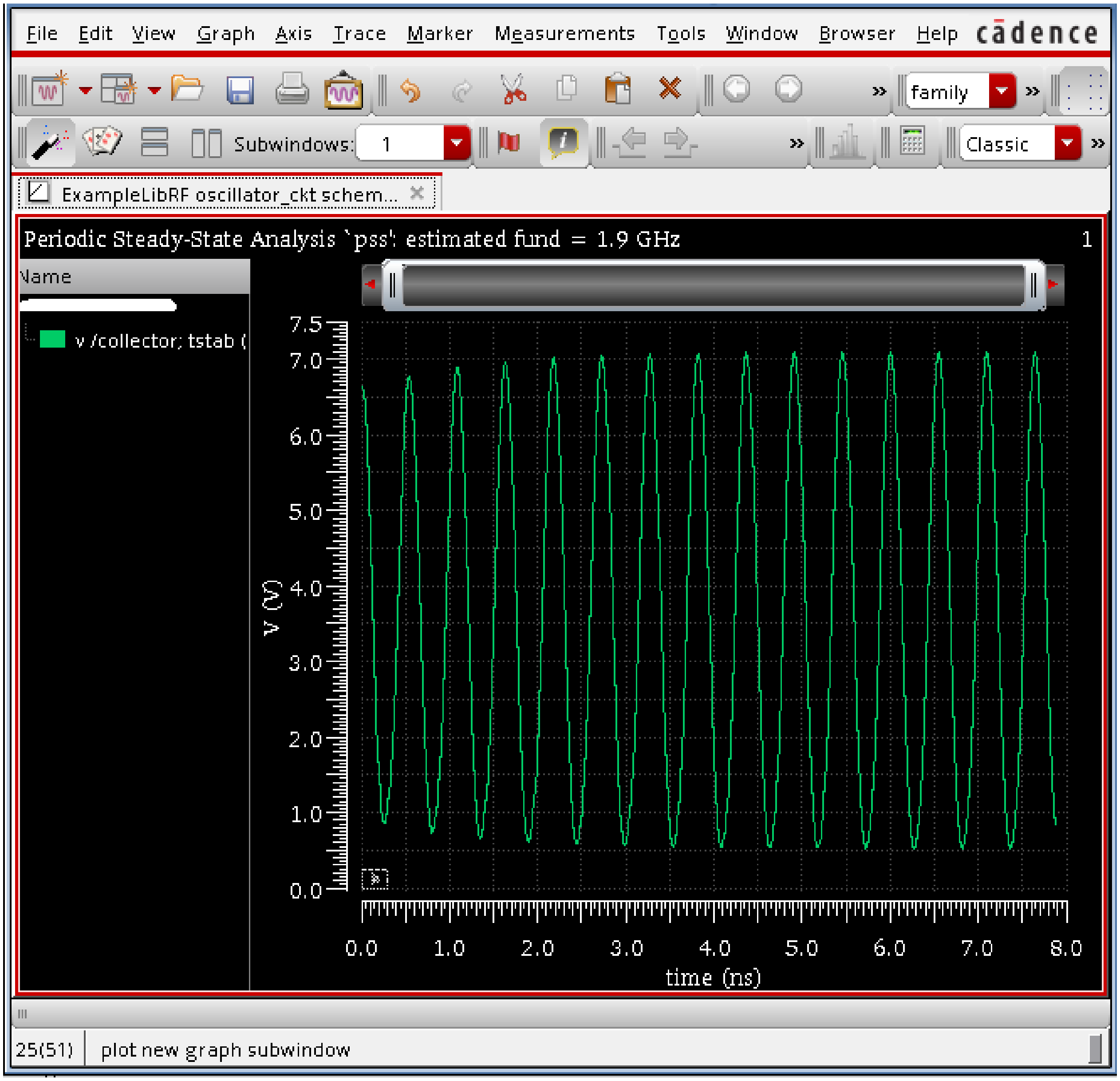
5. Select the *collector* net in the schematic. It is located just below the *collector* label.

Figure 3-21 Selecting *collector* net on oscillator_ckt schematic



The waveform window is displayed.

Figure 3-22 PSS Analysis - Initial Startup Waveform during tstab interval



This plot shows how the initial startup waveform for oscillator gets build up during the tstab interval. Note that the oscillator starts up immediately after time zero. This is because the calculate initial conditions was automatically set in the *Choosing Analyses* form.

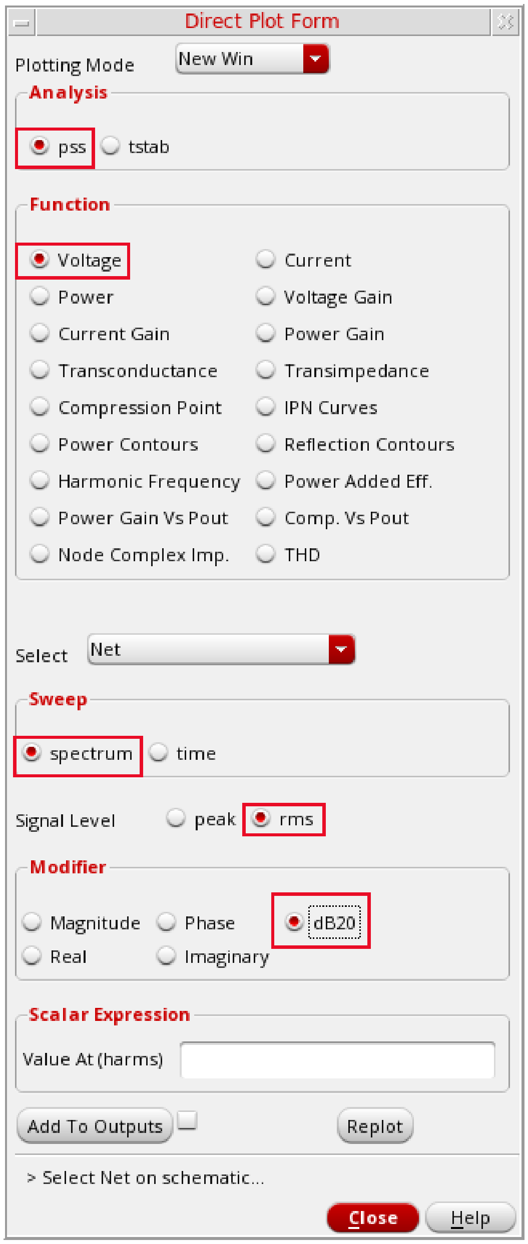
Next you will plot the oscillator output spectrum

Plot the oscillator spectral content, as follows:

1. In the *Direct Plot Form* window, set the *Plotting Mode* to *New Win*.

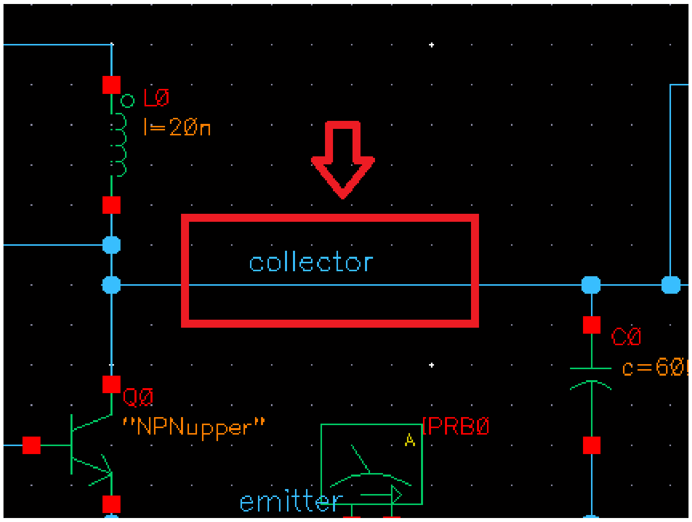
2. Select *pss* as Analysis.
3. Leave *Function* as *Voltage,* which is set by default.
4. Leave the *Select* section of the form set to the default value, *Net* (You can also select differential nets).
5. Select *Sweep* as *spectrum*. (This is the default)
6. Select *rms* as Signal Level (the default is *peak*).
7. Select *dB20* as Modifier.
8. Your *Direct Plot Form* window should like the following:

Figure 3-23 PSS Analysis Direct Plot Form Setup



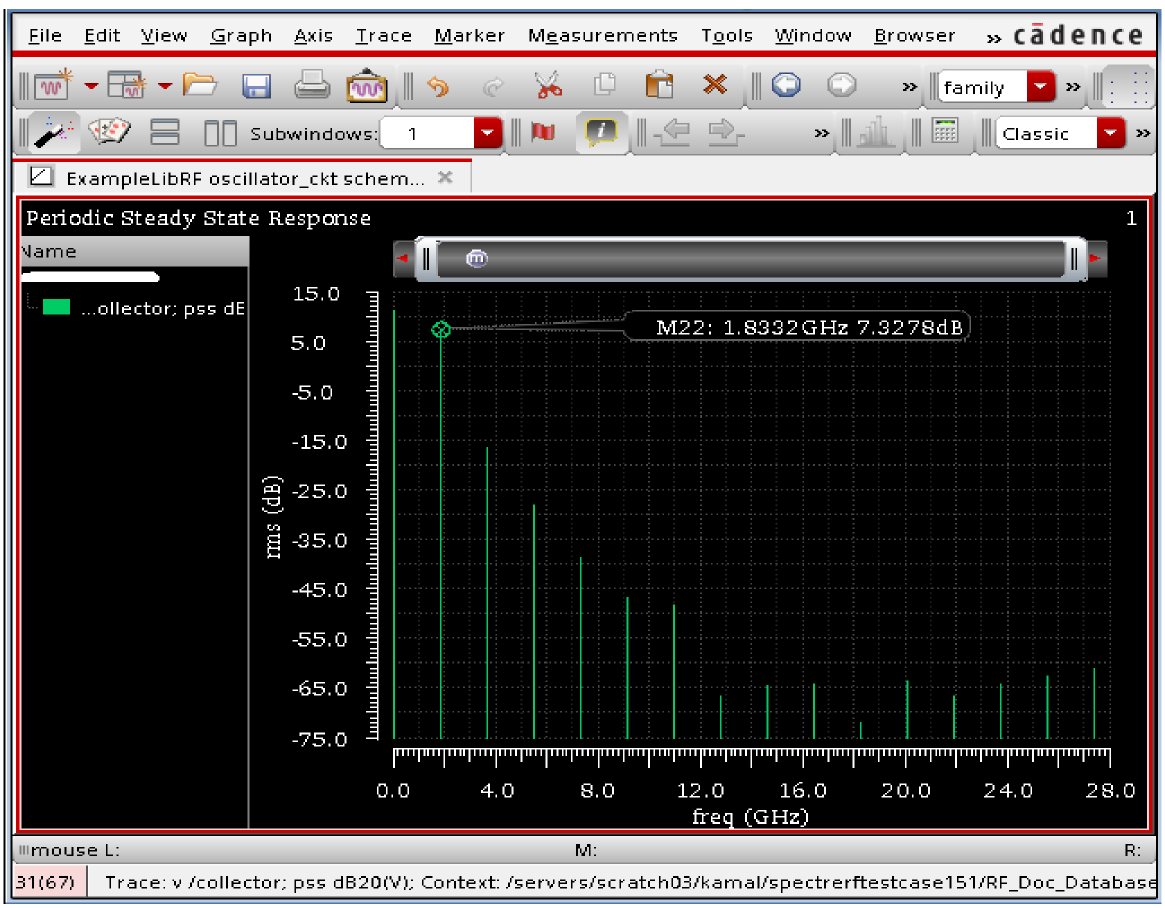
9. Select the *collector* net in the schematic. It is located just below the *collector* label.

Figure 3-24 Selecting *collector* net on oscillator_ckt schematic



The waveform window is displayed, as shown below.

Figure 3-25 PSS Analysis output Graph Window - Voltage Spectrum Plot



10. In the waveform window, position your cursor near the first harmonic, and press the ‘m’ key. Here ‘m’ is the bindkey to place a trace marker on the graph. The first harmonic is chosen as this is the frequency oscillator is designed for.

Note that this frequency is 1.833GHz. This is the frequency of oscillation.

11. In the *Direct Plot Form*, click *Cancel*.

12. In the ViVA window, choose *File* - *Close All Windows*.

13. Clean up the screen for the next set of measurements.

14. Close the Analog Design Environment window by selecting *Session* - *Quit*.

15. In the Schematic window, choose *File* - *Close*.

To summarize, a PSS analysis using Harmonic Balance was setup and simulation was run to determine the oscillation frequency of the oscillator. Next an Oscillator Loop Gain measurement will be performed.

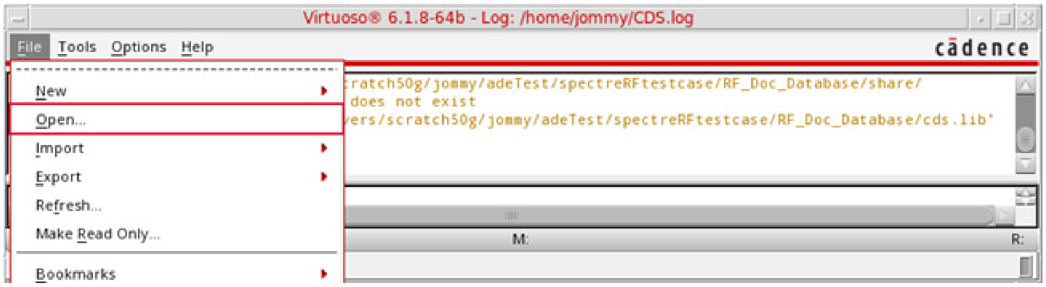
Oscillator Loop Gain Measurement

Oscillator loop gain measurements are typically made with a combination of linear stability (*stb*), Harmonic Balance (*hb*), and HBSTB (*hbstb*) analyses. The *hb* analysis solves for one period of the settled time-domain waveform. Stability Analysis (*stb*) allows the measurement of the loop gain and phase. This is quite useful for the design of the feedback network. Note that, *stb* analysis is a linear analysis and therefore does not take non-linearities of the circuit into account. However, it gives an approximate value of loop gain magnitude and phase and is faster to run. In order to get more accurate values of loop gain and phase of periodically time-varying non-linear circuits, you need to run *hbstb* analysis. The harmonic balance stability analysis (*hbstb*) evaluates the local stability of a periodically time-varying feedback circuit. It is a small-signal analysis, like *stb* analysis, except that the circuit is first linearized about a periodically varying operating point (determined using *hb* analysis) as opposed to a simple DC operating point (which is used in *stb* analysis). Linearizing about a periodically time-varying operating point allows the stability evaluation to include the effect of the time-varying operating point.

Opening the Oscillator Circuit in the Schematic Window

1. In the Command Interpreter Window (CIW), choose *File – Open*.

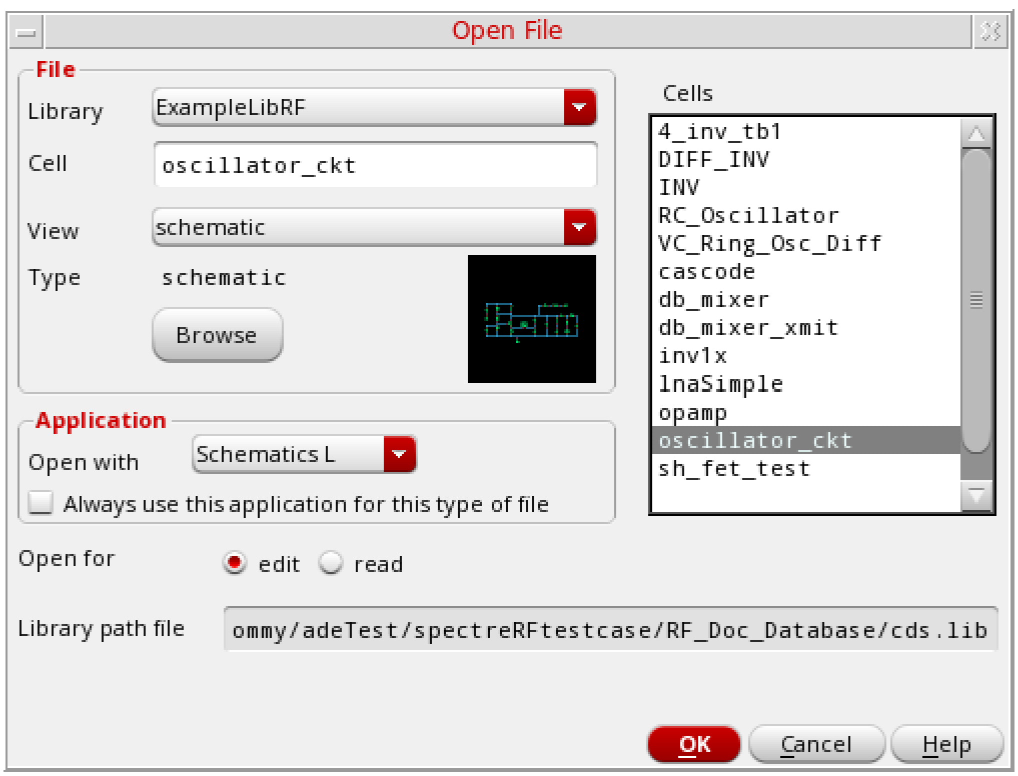
Figure 3-26 Virtuoso CIW Window - Opening Cellview



The *Open File* form is displayed.

2. Select *ExampleLibRF* from the *Library* drop-down list.
3. In the *Cells* field, type *oscillator_ckt*.
4. Choose *schematic* from the *View* drop-down list.
5. In the *Application* section, select *Schematics L* from the *Open With drop-down list.*
6. Leave *Open for* to *Edit* (which is set by default).

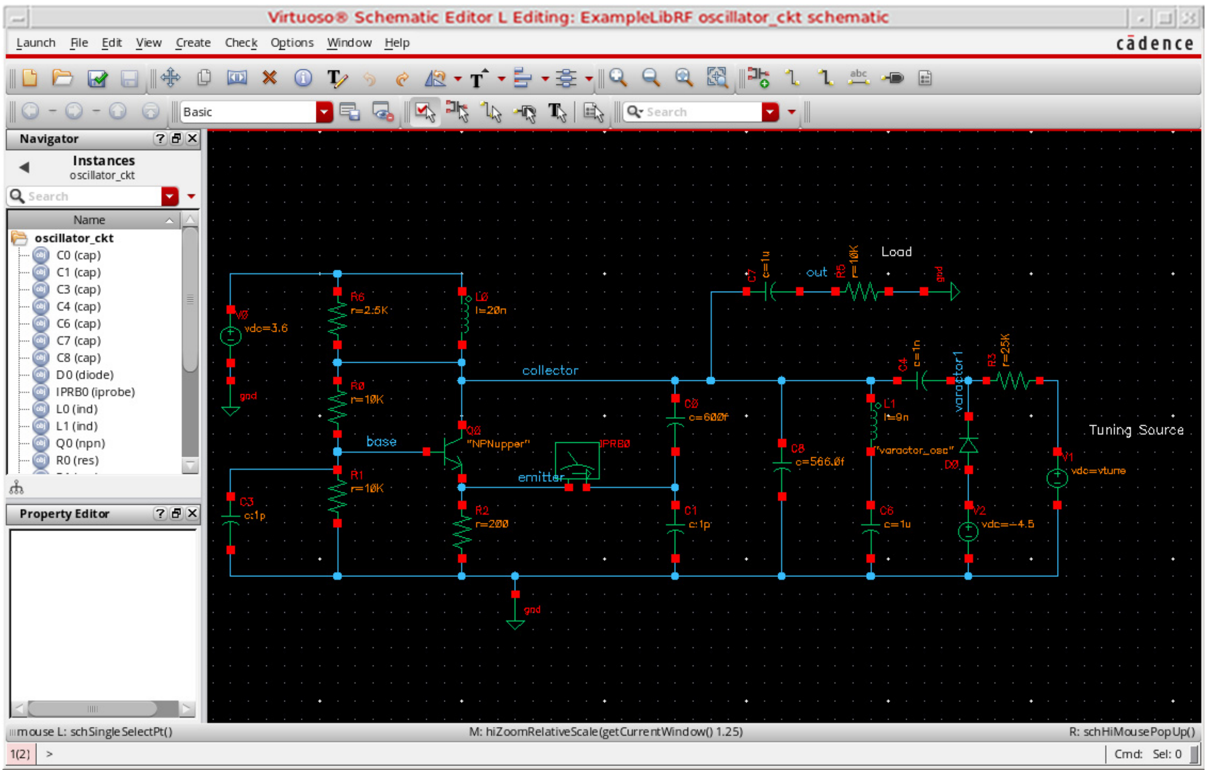
Figure 3-27 Open File Form to open the oscillator_ckt cell's Schematic View



7. Once all the setup is done, click *OK*.

   This will open the oscillator_ckt schematic in Virtuoso Schematic Editor L window, as shown below:

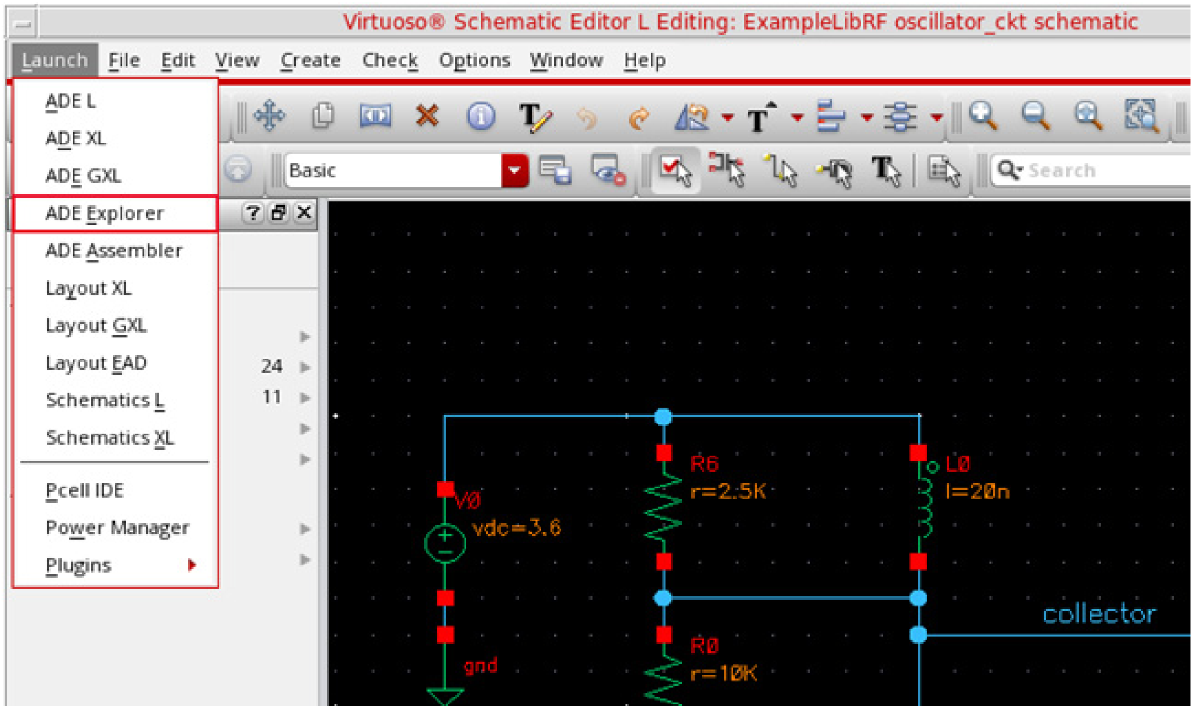
Figure 3-28 oscillator_ckt schematic in VSE L Window



Setting up the stb Analysis

1. In the Schematic Window, choose *Launch - ADE Explorer.*

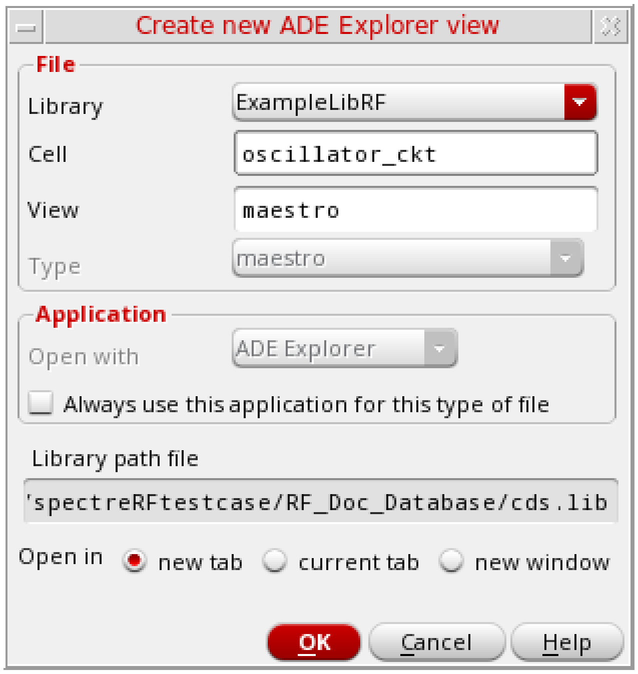
Figure 3-29 Opening ADE Explorer window from VSE window



2. In the *Launch ADE Explorer* dialog, select *Create New View*.

   The *Create new ADE Explorer view* form is displayed.

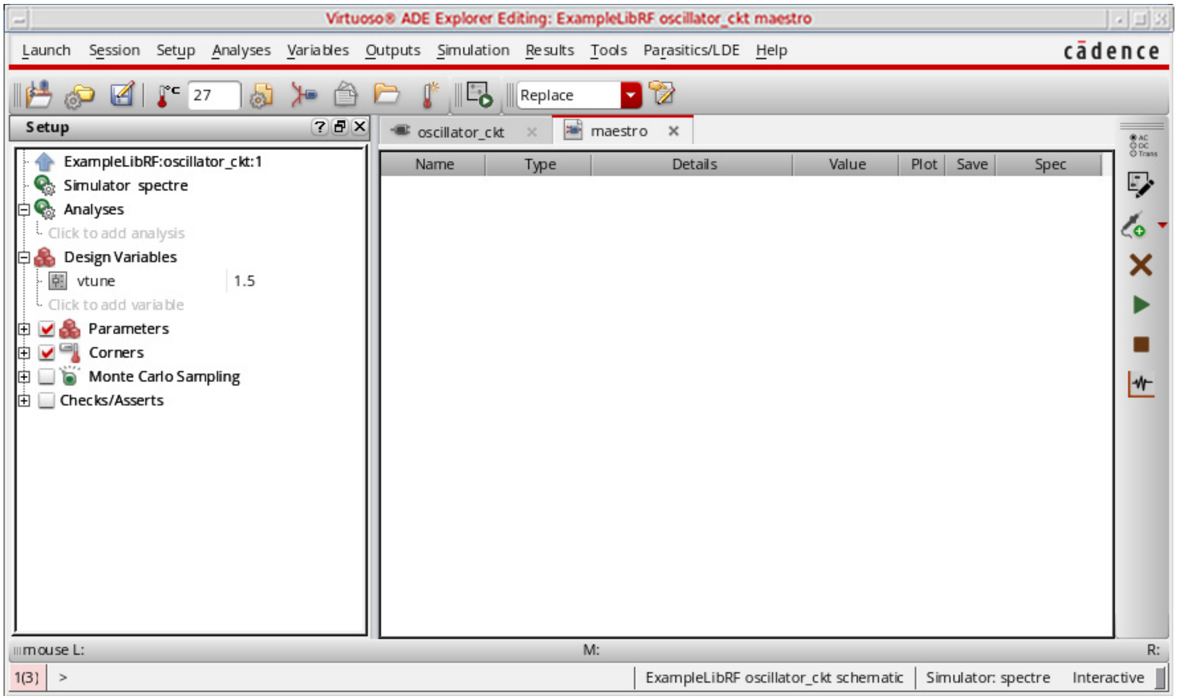
Figure 3-30 Create new ADE Explorer view



3. Leave each option to the default selections and click *OK*.

   Virtuoso ADE Explorer is opened, as shown below.

Figure 3-31 ADE Explorer



4. Select *Setup – Simulator*.

   The *Choosing Simulator* form is displayed.

5. Select *spectre* for the *Simulator.*

Figure 3-32 Choosing Simulator Form

6. Click *OK*.

7. Select *Setup - High Performance Simulation*. The *High Performance Simulation Options* window is displayed.

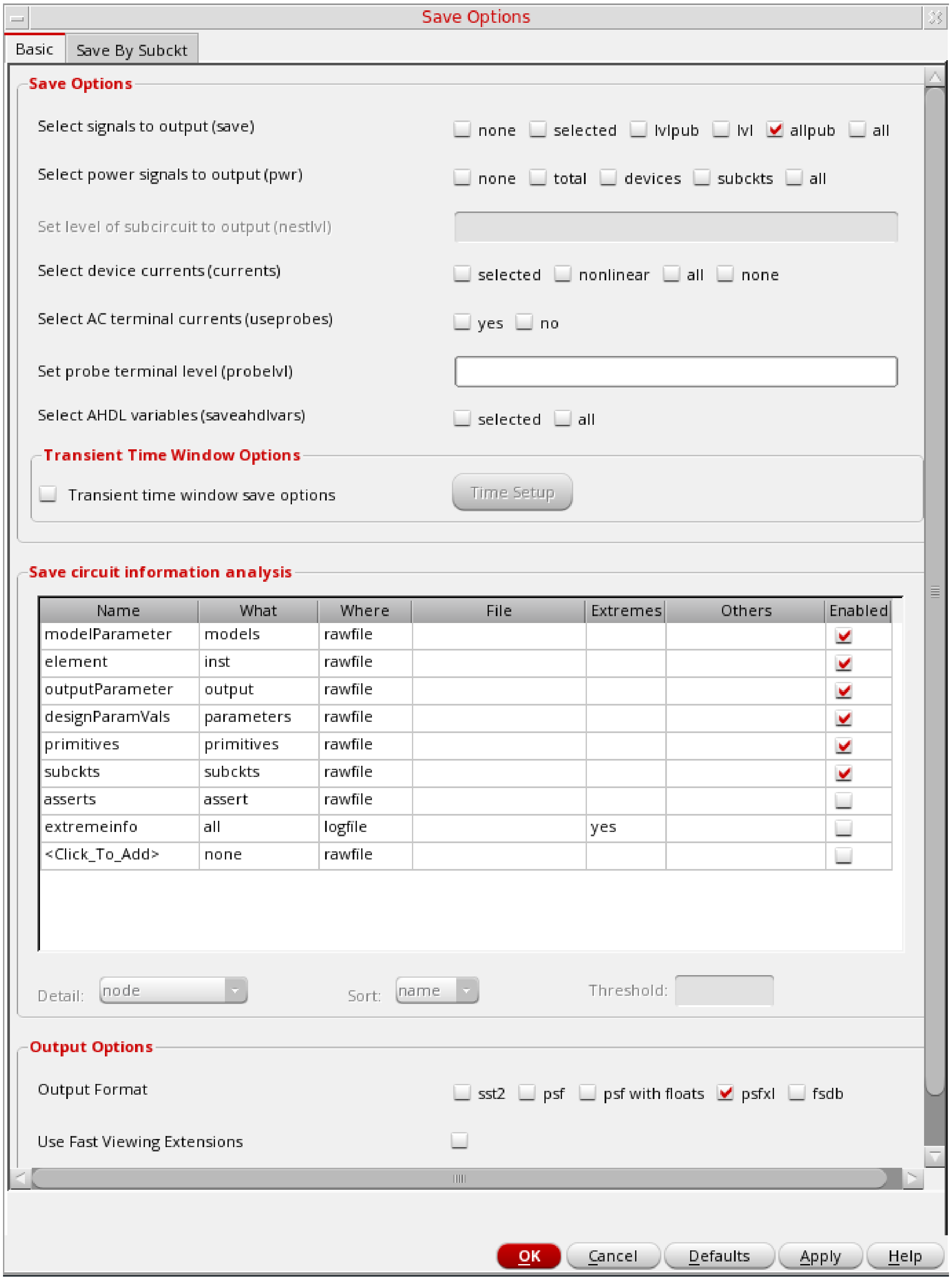
   In the *High Performance Simulation Options* window, select *APS*. Note that *Auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores. Usually it is better to specify the number of threads yourself. Small circuits should use a small number of threads, and larger circuits can use more threads. The overhead of managing 16 threads on a smaller circuit may actually slow the simulation down, compared to two or four threads. Use the number of threads that maximizes the performance.

Figure 3-33 High Performance Simulation Options Form

8. Click *OK*.
9. Select *Outputs – Save All*.

   The *Save Options* form is displayed.
10. In the *Select signals to output (save)* section, make sure that *allpub* is selected.

Figure 3-34 Save Options Form



This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

To save the currents, use the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or all if you want to save all the currents in the circuit. When you save currents, more disk space is required for the results file.

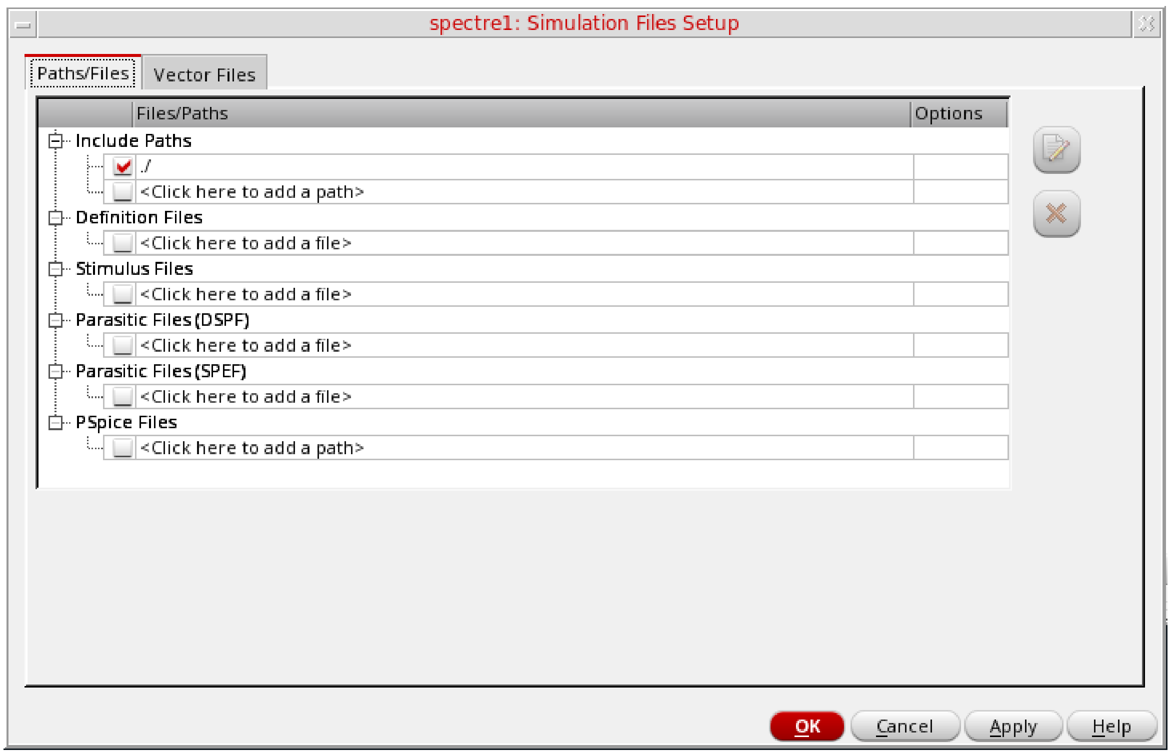
11. Click *OK*.

12. Select *Setup - Simulation Files.*

The *Simulation Files Setup* form is displayed.

13. In the *Simulation Files Setup* form, enter ./ by clicking in the *Include Paths* section. It should look like the following:

Figure 3-35 ***Simulation Files Setup*** **Form**



14. Click *OK* to close the *Simulation Files Setup* form.

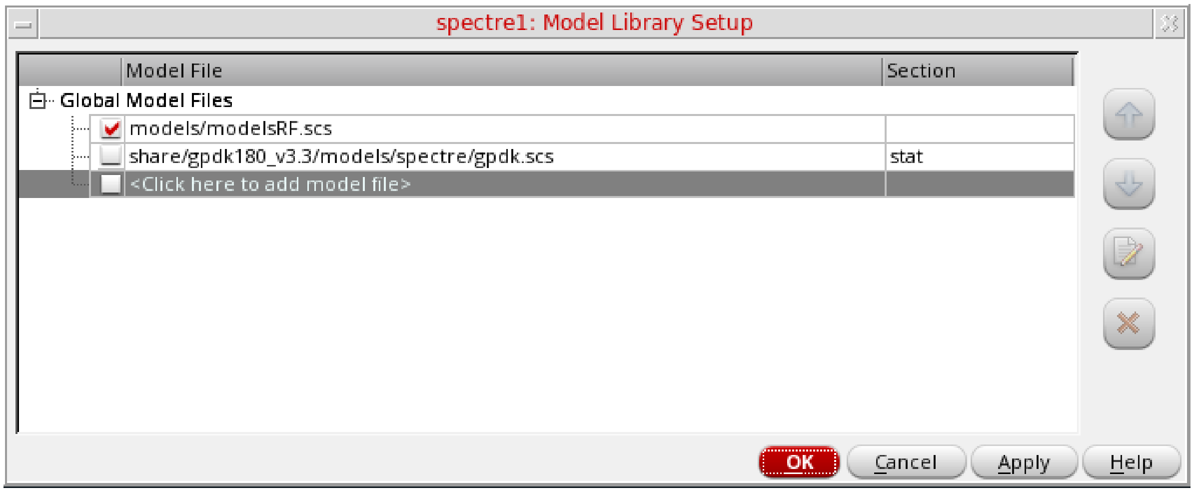
15. Select *Setup – Model Libraries*.

The *Model Library Setup* form is displayed.

16. In the *Model File* field, type the path to the model file including the file name, as shown below.

models/modelsRF.scs

Figure 3-36 *Model Library Setup* Form



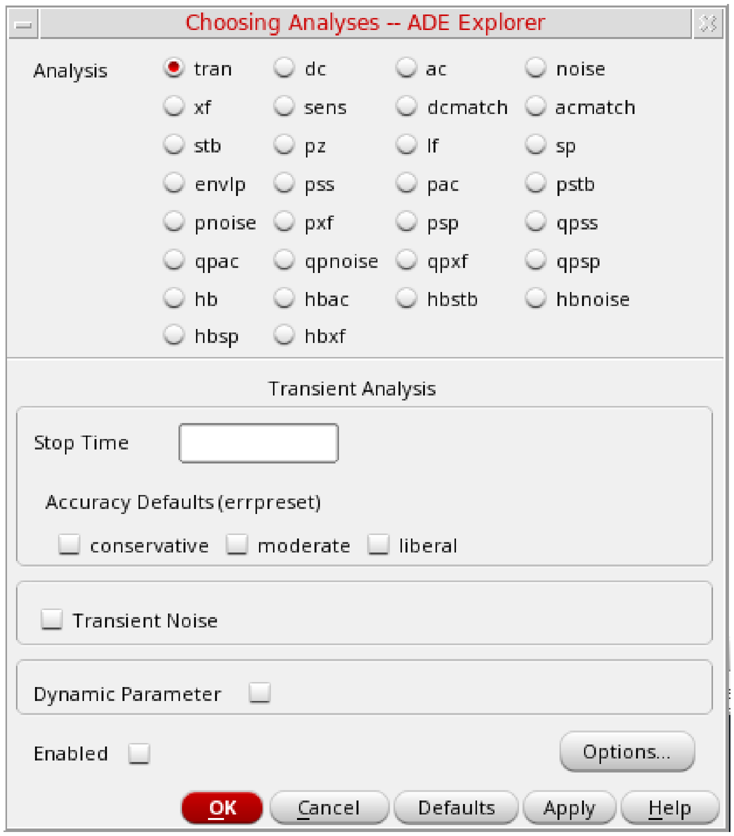
You can also browse to *modelsRF.scs* file.

17. Click *OK*.

18. Select *Analyses - Choose.*

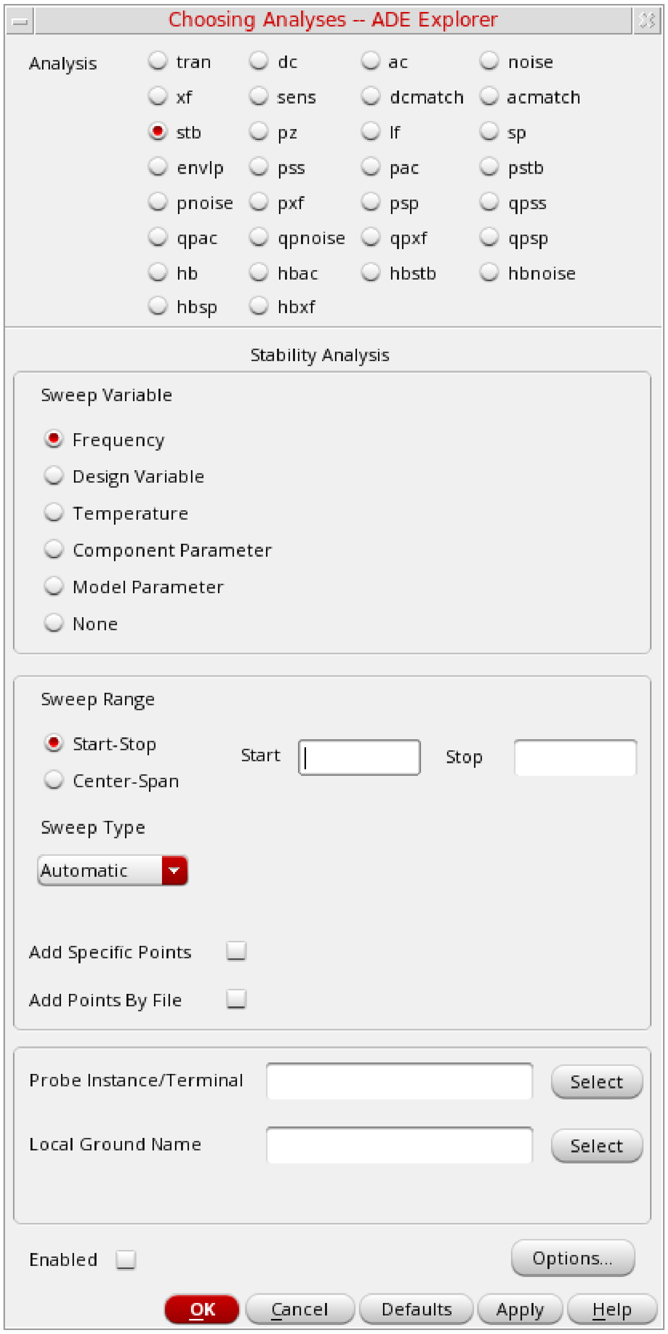
The *Choosing Analyses* form is displayed, as shown below.

Figure 3-37 The Choosing Analyses Form



19. In the *Analysis* section, select *stb*.

Figure 3-38 The Choosing Analyses Form - stb Analysis Setup



20. Type 1m in the *Start* Field. Note the lower case m. This means the start frequency is 0.001 Hz.

21. Type 1T in the *Stop* field. This will set the stop frequency to 1 THz.

The frequency range is deliberately set very wide in order to detect potential parasitic oscillation modes.

22. Select *Logarithmic* sweep.

23. Type 100 for *Points per Decade*. The frequency coverage is set to be very dense as this analysis runs quickly.

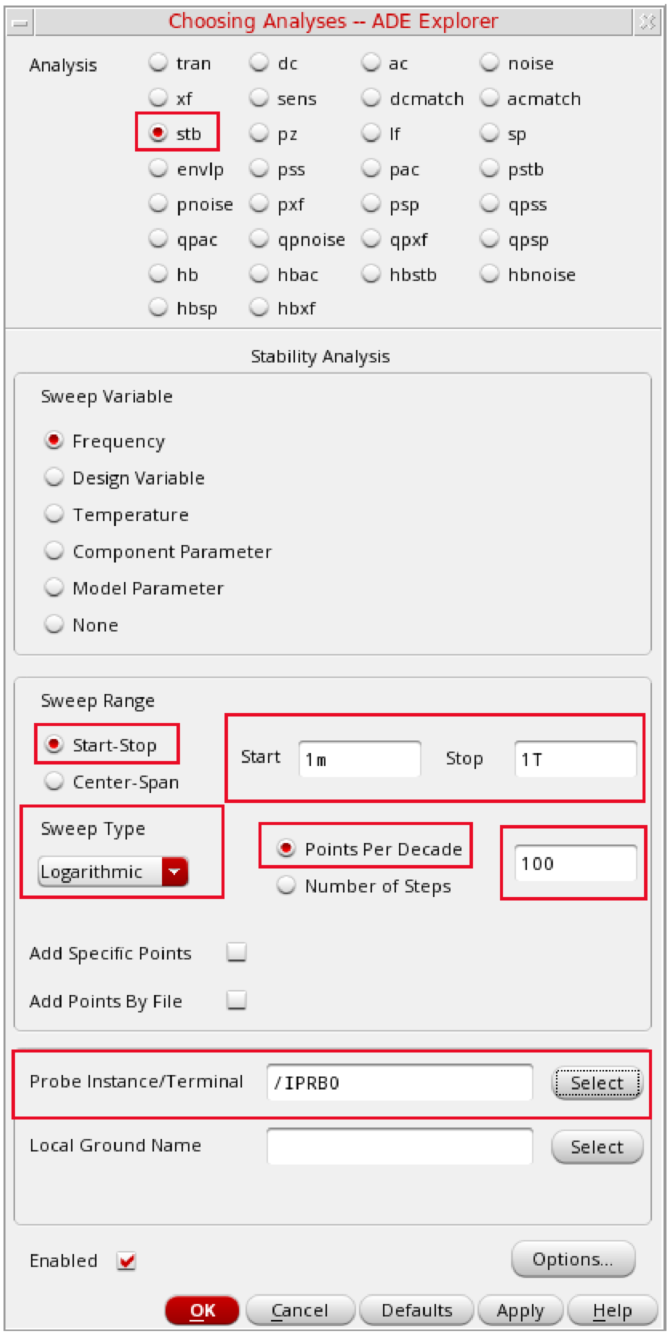
24. Click *Select* to the right of the *Probe Instance/Terminal* field.

25. In the schematic, click the instance that looks like an analog meter in the center of the circuit. This is an iprobe from analogLib.

Note: To use stability analysis, either an iprobe or a vdc set to 0 Volts needs to be added in series with the feedback path. In this case, an iprobe is used. The current probe and the vdc source both have zero resistance. Because of this the loading in the loop is maintained. In the past, the AC part of the loading had to be broken in order to get a loop gain measurement. For a differential circuit, use diffstbprobe from analogLib.

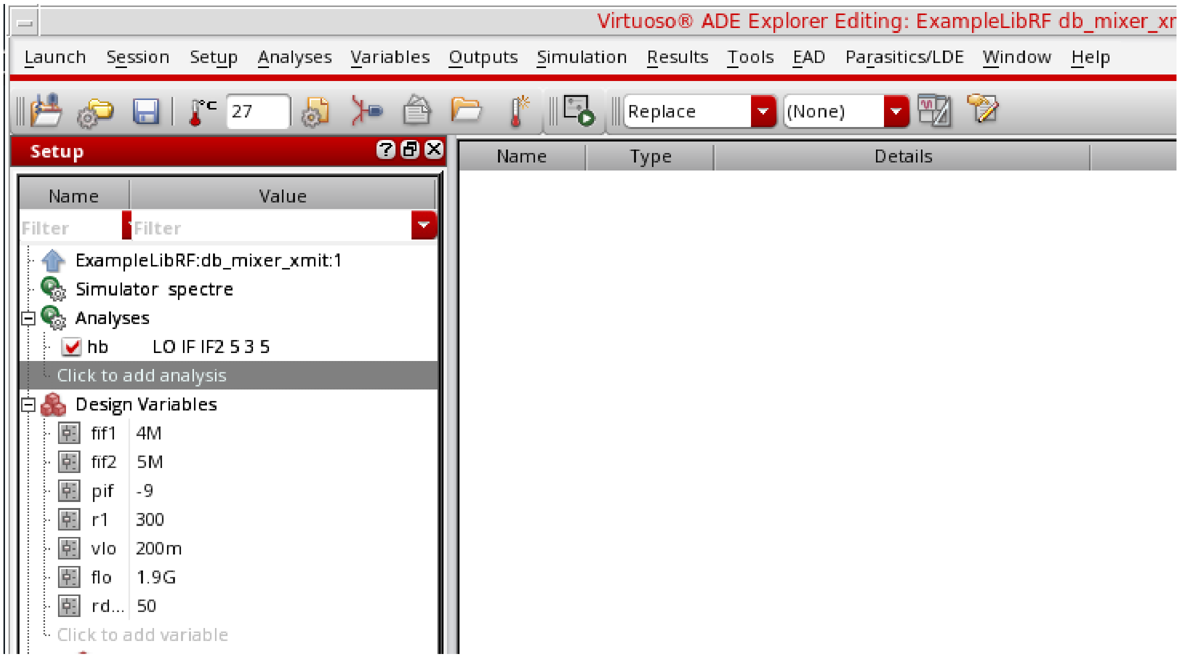
Your Choosing Analyses form should look like this:

Figure 3-39 Choosing Analyses Form - *stb* Analysis Setup



Click *Apply* at the bottom of the *Choosing Analyses* form. This will add the *stb* analysis in the *Analyses* section of ADE Explorer, as shown below:

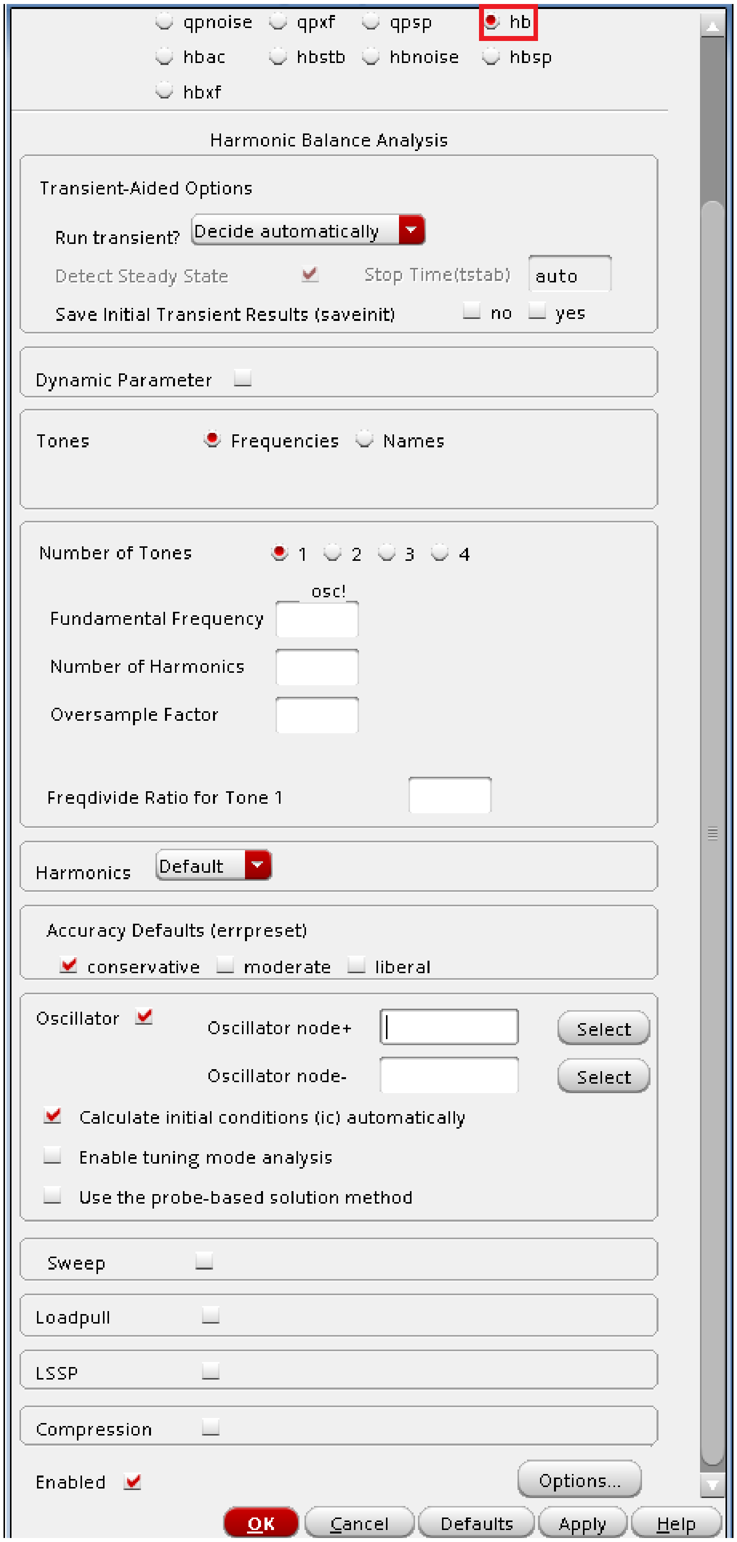
Figure 3-40 ADE Explorer Simulation Window - *stb* analysis setup



Setting up the HB Analysis

1. Select *hb* as *Analysis*. The form expands.

Figure 3-41 The Choosing Analyses Form - Setting HB Analysis



2. Leave the *Run transient* option to the default value of *Decide Automatically*.

3. Select *yes* for *Save Initial Transient Results (savinit)*. This will help in visualizing the buildup of the oscillator waveform.

4. In the *Fundamental Frequency* field, type `1.9G`. The frequency entered here is an approximate frequency of oscillation.

5. In the *Number of harmonics* field, type `15`.

   In general, you want to choose a number that is high enough to capture the nonlinearity of the circuit. Start with 10, and run the simulation. Increase by about 50% to 15 and re-run the simulation. If the harmonics donot change appreciably, then 10 is enough. If they change, increase the number by about 50%. Use the smallest number of harmonics for the answer to be stable.

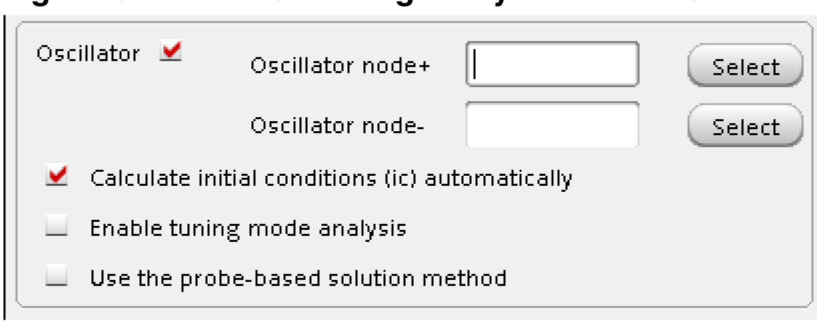
6. Leave Oversample Factor as default (that is, `1`). Since the oscillator has sinusoidal waveforms, an oversample of 1 is appropriate.

7. Leave the *Freqdivide Ratio for Tone 1* option to the default value of `1`.

8. In the *Accuracy Defaults* (*errpreset*) section, select *conservative*. *conservative* is typically used because very small amplitude phase noise measurements are normally desired. Conservative is recommended for all the oscillators.

9. Select *Oscillator*. This is required for simulating an autonomous circuit.

Figure 3-42 The Choosing Analyses Form - Oscillator Section

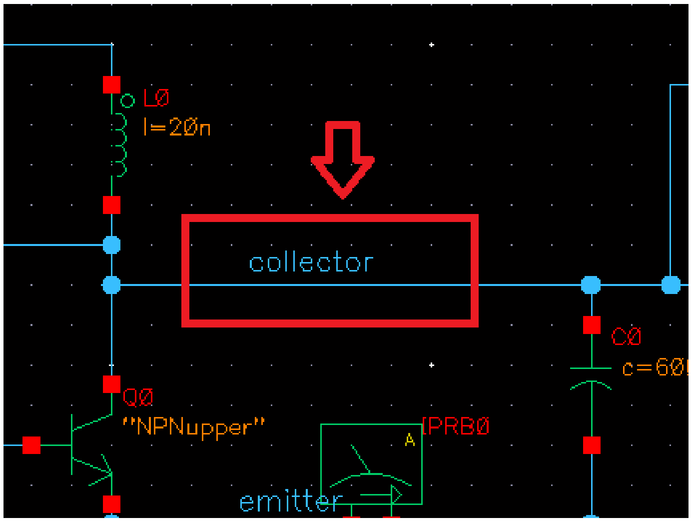


10. In the *Oscillator node+* field, click *Select* just to the right of the field. In the schematic, select the *collector* node. This is the net just below the collector label. This oscillator node will be used by the simulator for the period calculation of the oscillations. It just needs to be a node that has the oscillator signal on it.

11. Leave the *Oscillator node-* field blank.

    Note that if you have a single-ended oscillator, only specify one node. If the second node, (the *r*eference node) is left blank, it will be connected to the global ground node automatically. However, if you have a differential oscillator, you need to specify both the nodes.

Figure 3-43 Selecting *collector* net on oscillator_ckt schematic



12. If you have an LC oscillator, leave the *Calculate initial conditions (ic) automatically* checkbox selected (this is the default).

Note that *Calculate initial conditions (ic) automatically* is used to start the oscillator. Other methods to start the oscillator include putting a single current pulse into the resonator, setting initial conditions(ic), or ramping up the power at time = zero plus.

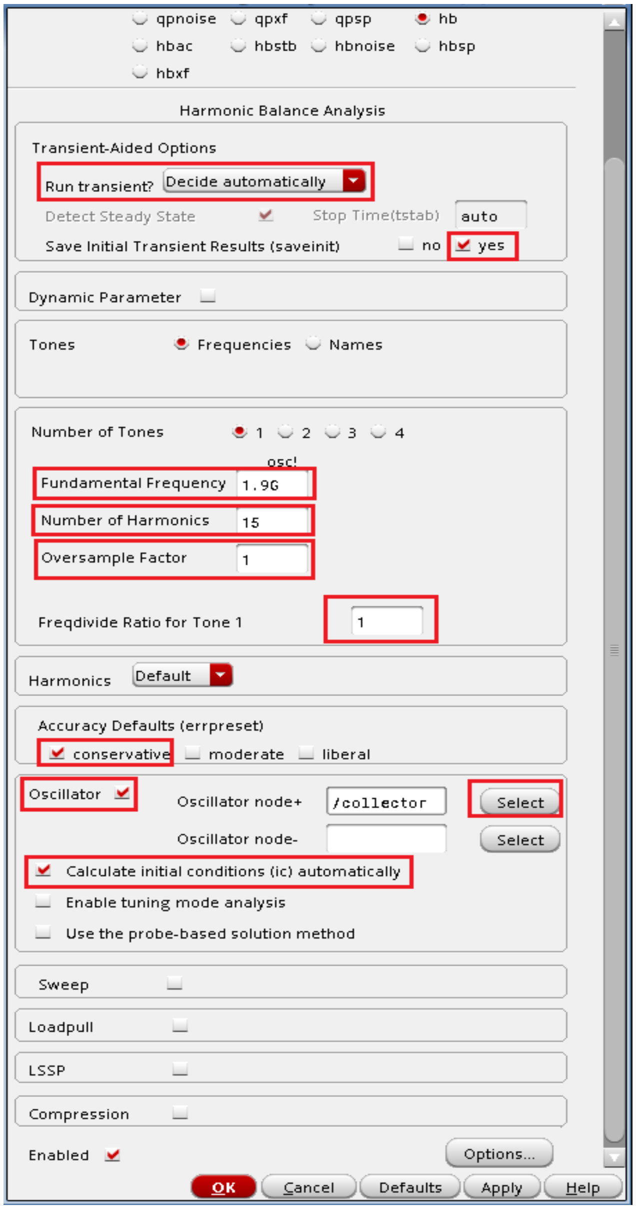
13. Note that by default *Use the probe-based solution method* (*oscmethod*) option is deselected. Spectre will use the *onetier* method.

When *Setting Use the probe-based solution method* (*oscmethod*) option is selected, it iterates for the frequency solution in the outer loop and the amplitude and phase solution in the inner loop. The probe-based method has better convergence but is computationally intensive.

Please refer to *Spectre Circuit Simulator RF Analysis Theory* for more details.

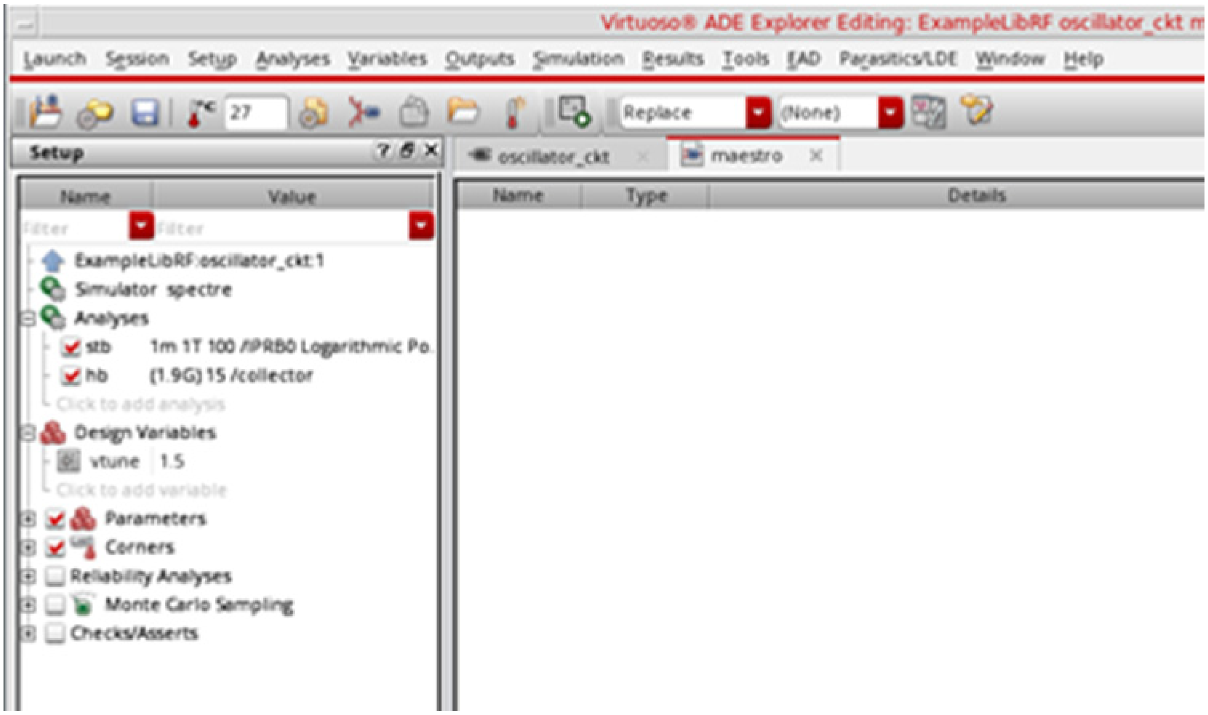
The Choosing Analyses form will look like the following:

Figure 3-44 Choosing Analyses Form - Harmonic Balance Setup



14. Click *Apply* located at the bottom of the form. This will add the *hb* analysis in the Analyses section of ADE Explorer along with *stb* analysis, as shown below:

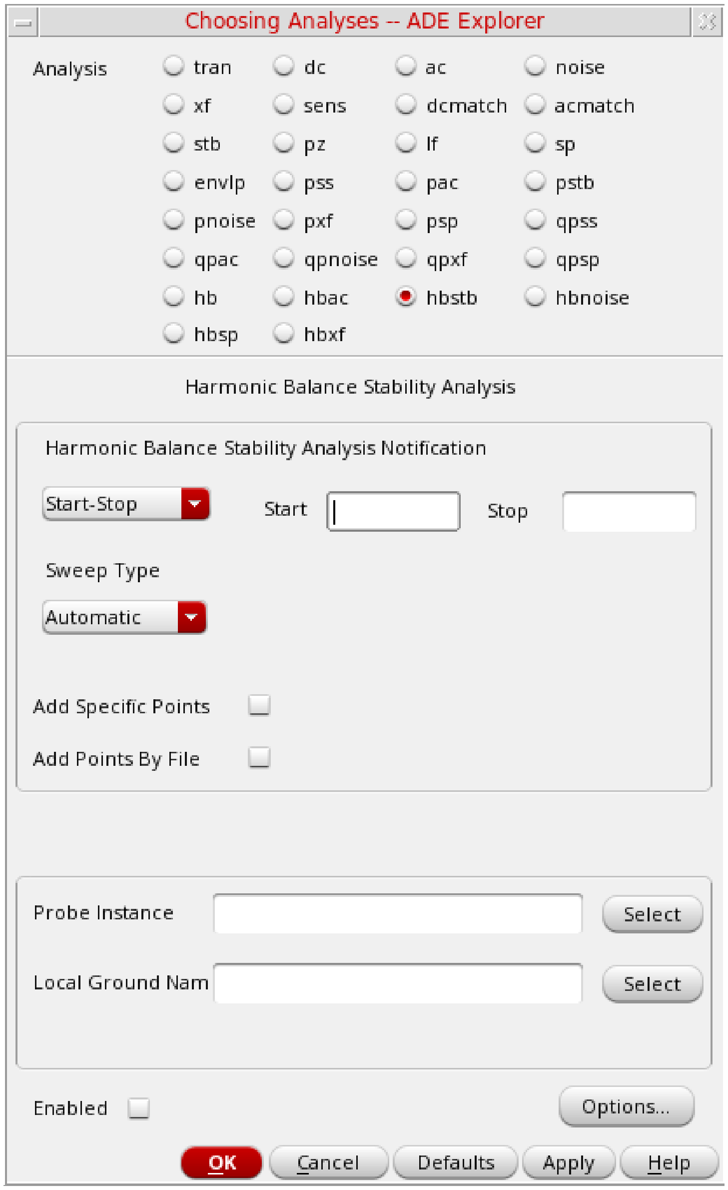
Figure 3-45 ADE Explorer Simulation Window - *stb* and *hb* Analysis setup



Setting up the HBSTB Analysis

1. In the *Choosing Analyses* form, select *hbstb*.

Figure 3-46 The Choosing Analysis Form - *hbstb* Analysis Setup



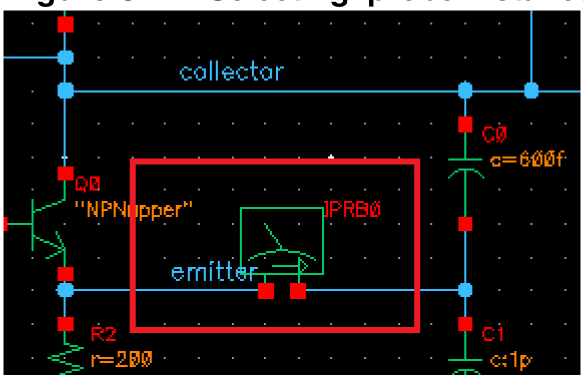
Set your sweep frequency such that the expected oscillation falls within the sweep range.

2. Type 1.7G in the *Start* field.
3. Type 2G in the *Stop* field.
4. Set the *Sweep Type* to *Linear*.
5. Select *Number of Steps*.
6. Type 100 in the *Number of Steps* field.

You need to provide number of steps such that you have enough resolution in frequency. In this case, you have 300MHz/100 = 3MHz step resolution.

7. Click your mouse cursor in the *Probe Instance* field and click *Select* which is just to the right of this field. The *pstb* requires that you place the probe on the feedback loop to identify and characterize the particular loop of interest.
8. In the schematic window, select the *iprobe* in the center of the circuit. It looks like an analog meter. 'iprobe' is required to determine the feedback without breaking the feedback loop.

Figure 3-47 Selecting iprobe instance IPRB0 on the schematic

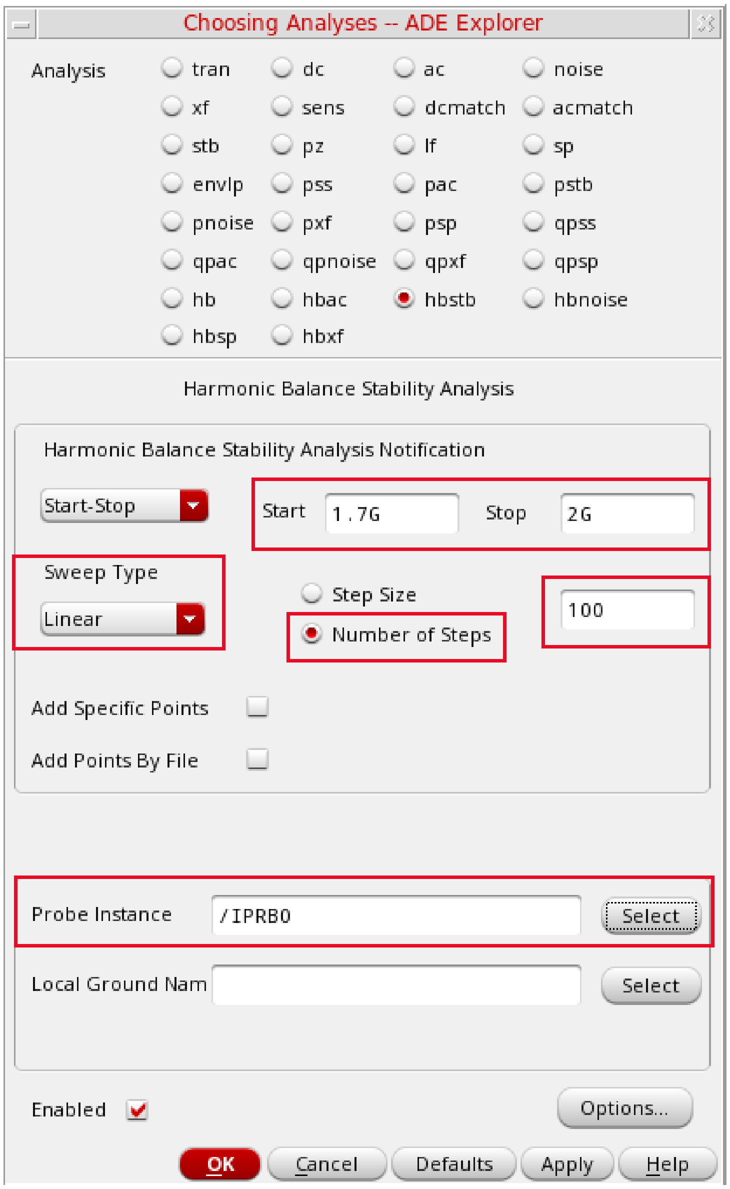


Note: To use pstb analysis, either an iprobe or a vdc set to 0 Volts needs to be added in series with the feedback path. In this case, an iprobe is used. The current probe and the vdc source both have zero resistance. Because of this the loading in the loop is maintained. In the past, the AC part of the loading had to be broken in order to get a loop gain measurement.

Note: For a differential circuit, use the diffstbprobe component from analogLib.

The *Choosing Analyses* form should look like the following.

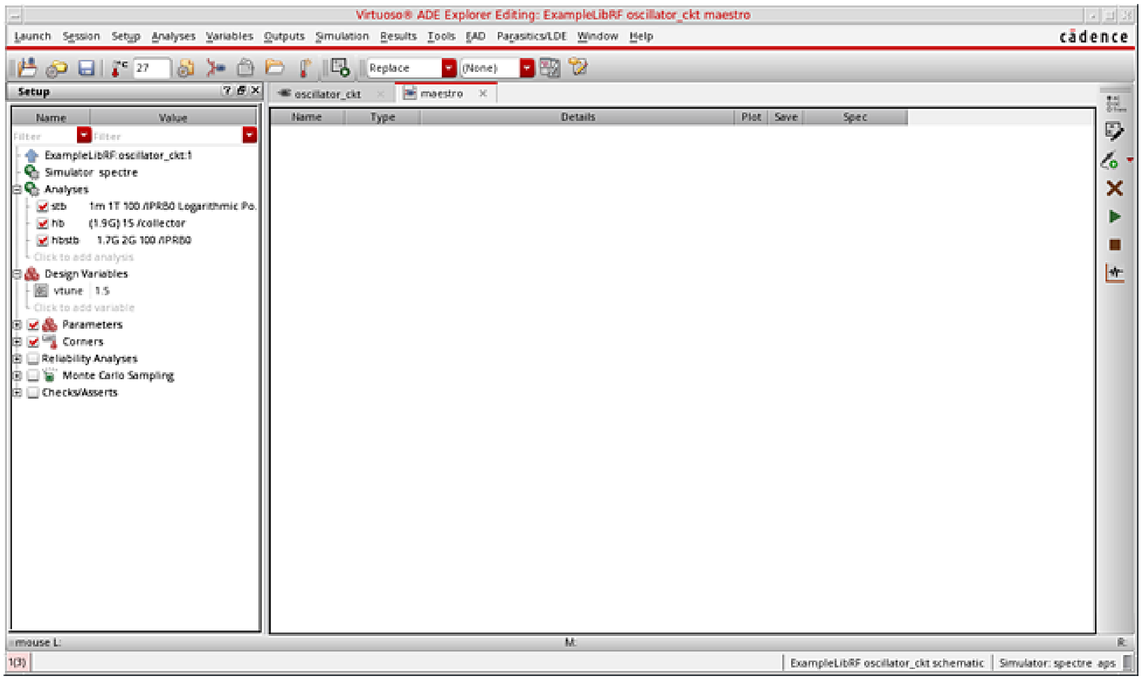
Figure 3-48 Choosing Analyses Form - *hbstb* Analysis Setup



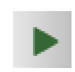
9. Click *OK* button at the bottom of the form to close the *Choosing Analyses* form.

   The *hbstb* analysis gets added along with *stb* and *hb* analysis in the Analyses section of ADE Explorer, as shown below.

Figure 3-49 ADE Explorer Simulation Window - *stb*, *hb* and hbstb setup



Running the HB, stb and HBSTB analysis

Once finished setting up the HB, HBSTB and stb analysis click the green icon  in ADE Explorer or on the Schematic window to run the simulation.

Plotting the results

Plotting the stb analysis results

First plot the Loop Gain magnitude and phase from the stb analysis.

1. In ADE Explorer, select *Results - Direct Plot - Main Form*.

Figure 3-50 The Direct Plot Form - stb, hb and hbstb Analysis

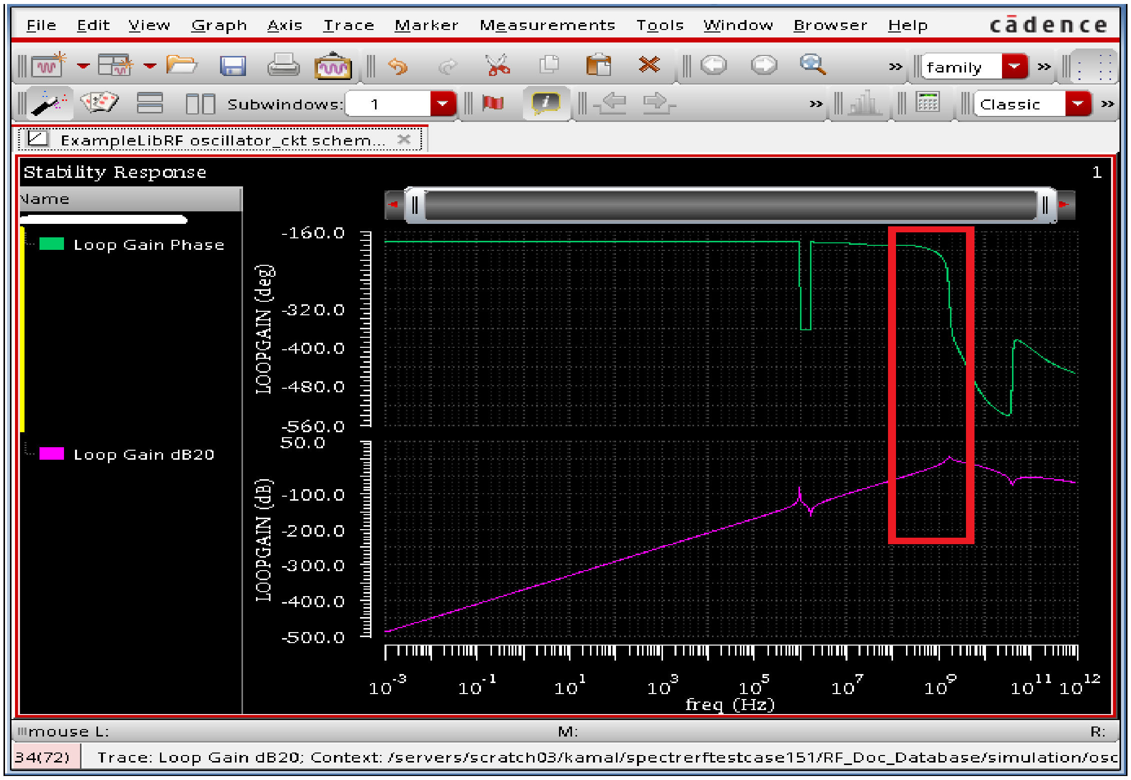
2. Select *stb* as Analysis.
3. Keep the rest of the settings at their default values, that is *Loop Gain* as *Function*, *Magnitude* and *Phase* as *Modifier* and *dB20* as *Magnitude Modifier*. Remember that we are trying to plot the Loop Gain magnitude and phase plot.

   The Direct Plot Form looks like the following:

Figure 3-51 *stb* Analysis Direct Plot Form Setup

4. Click *Plot*.
5. In the waveform window, click the *Split Current Strip* icon  . This will add the Loop Gain and Phase plots in two different strips in the same graph window, as shown below.

Figure 3-52 stb Analysis Ouput Graph Window - Loop Gain Magnitude and Phase Plot



On the top of the graph is a plot of loop gain phase in degrees. On the bottom of the graph is a plot of loop gain magnitude in dB. The area in the red box is the unity gain area.

Note: There is only one place just above 1 GHz where the loop gain approaches unity.

6. Now Zoom in to the unity gain area of the Loop Gain Plot. To do this:

   a. Move your mouse cursor over one of the numbers on the X axis.

   b. Click the right mouse button and select *Axis Properties*.

   c. Click the *Scale* tab and set *Mode* to *Manual*.

   d. Type 1G in the *Min*imum cyclic field. This is near to the frequency before the unity gain area starts.

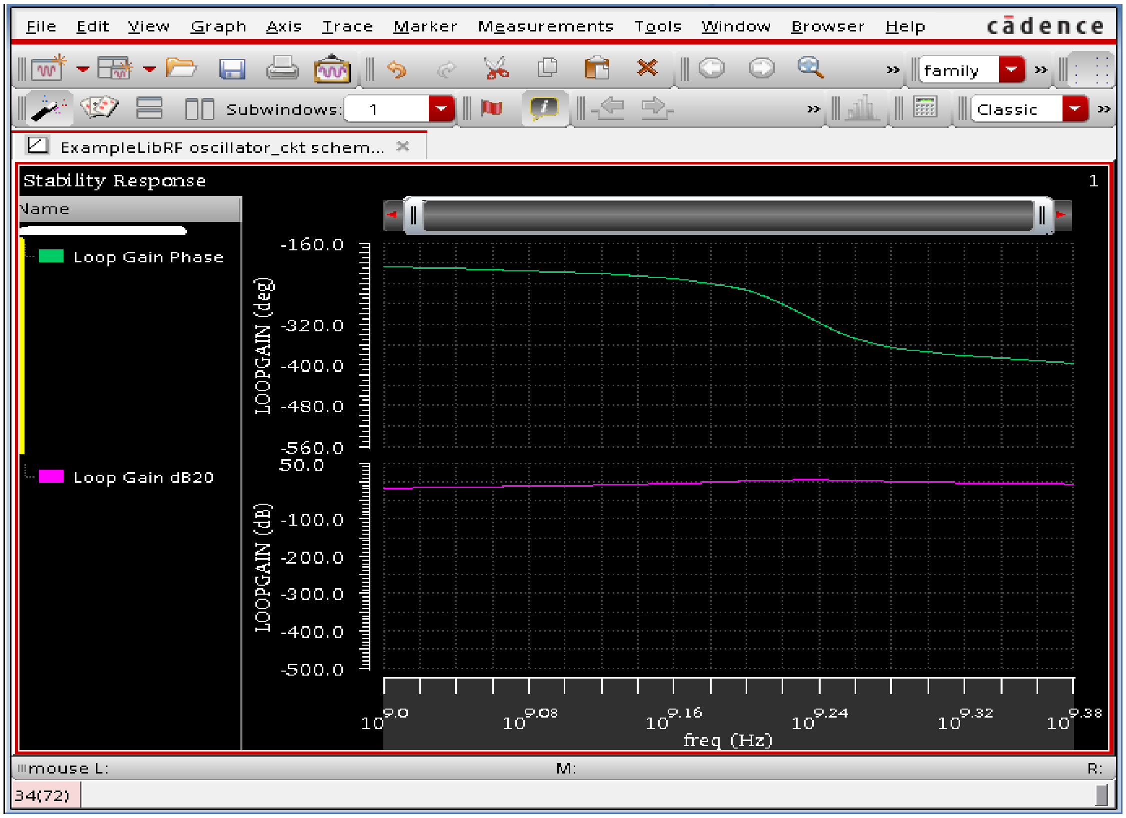
   e. Type 2.4G in the *Maximum cyclic* field. This is frequency near to which the unity gain area ends.

Figure 3-53 Graph Window X-Axis Setup

f. Click *OK*.

This will provide the zoomed in plot of Loop Gain's Magnitude and Phase from 1GHz to 2.4GHz frequency range.

Figure 3-54 ***stb*** **Analysis Loop Gain Magnitude and Phase Zoomed-in Plot**



In traditional control system analysis, an oscillator will oscillate when the loop gain is greater than unity and the phase is 180 degrees. This assumes that there is an inverting summing junction in the feedback loop. In the circuit, we cannot separate out the summing junction, and so the loop gain measurement contains it. In the phase plot, what one looks for is zero or multiples of 360 degrees, not 180 degrees.

1. Place a vertical marker at 360 degrees phase, as follows:

   a. Move the mouse cursor near the 360 degree phase point and type v. This will place a vertical marker.

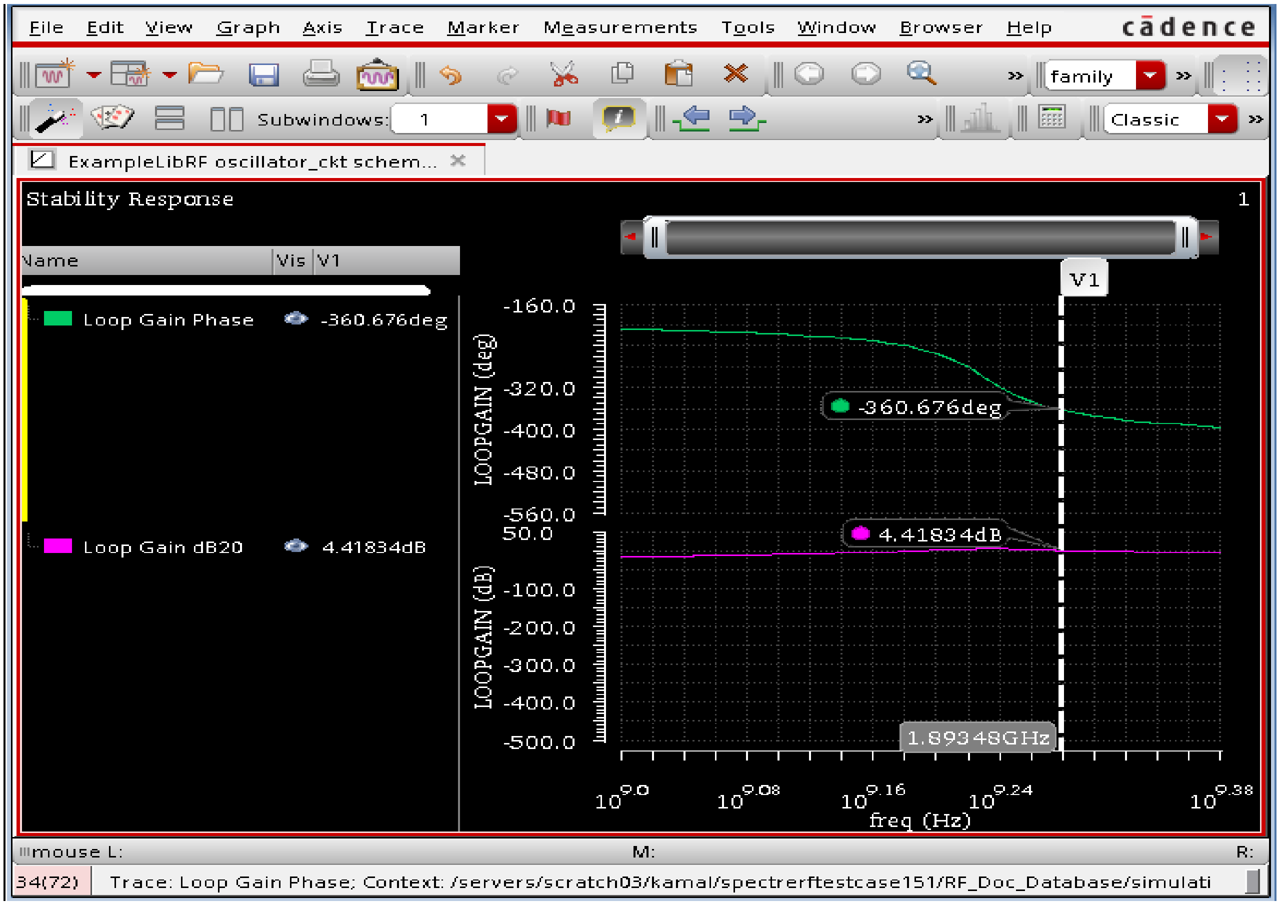
   b. Move your mouse cursor over the vertical marker. The cursor will change shape.

   c. Right-click and select *Intercepts - On*.

   d. Select the vertical marker, click and hold the mouse button, and place it at 360 degrees.

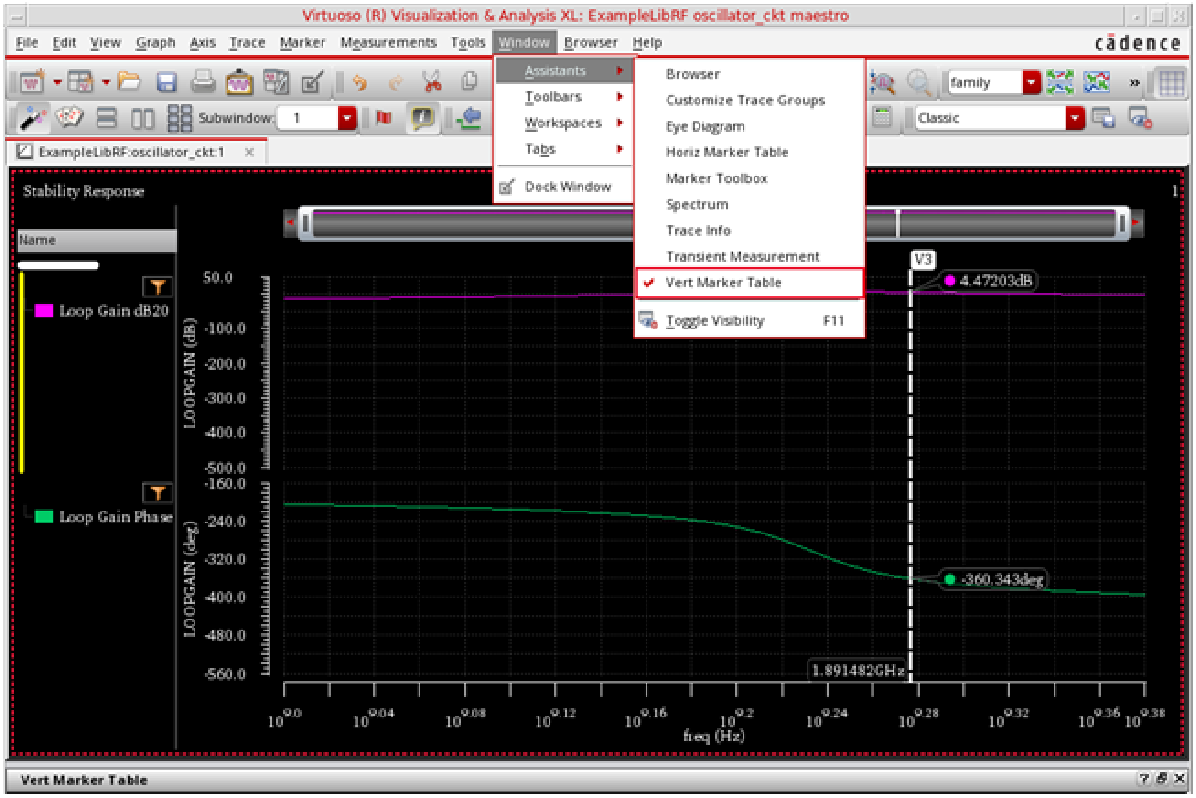
   The graph window will look like the following:

Figure 3-55 stb Analysis - 360 degree Marker Placement



2. Select *Window - Assistants - Vert Marker Table.*

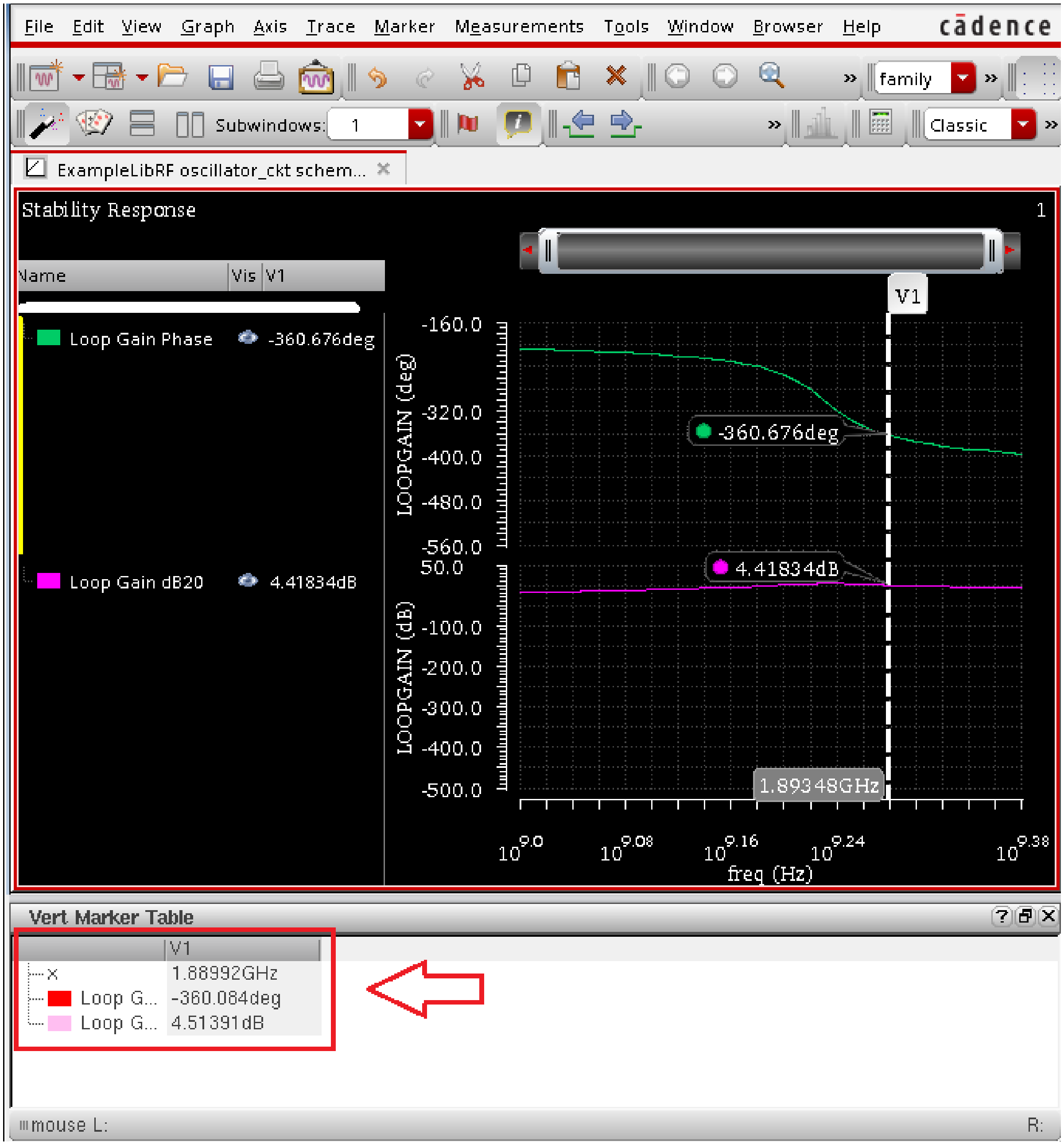
Figure 3-56 ViVA XL - Selecting Marker Toolbox



The Marker Table window appears with the data for the phase and gain curve intercepts along with the frequency.

The Graph Window will look like the following:

Figure 3-57 Oscillation frequency determination from *stb* Loop Gain Plot



Note that the loop gain is just over 4.5 dB on the Loop Gain dB20 curve. (Greater than unity gain.) The oscillator should oscillate.

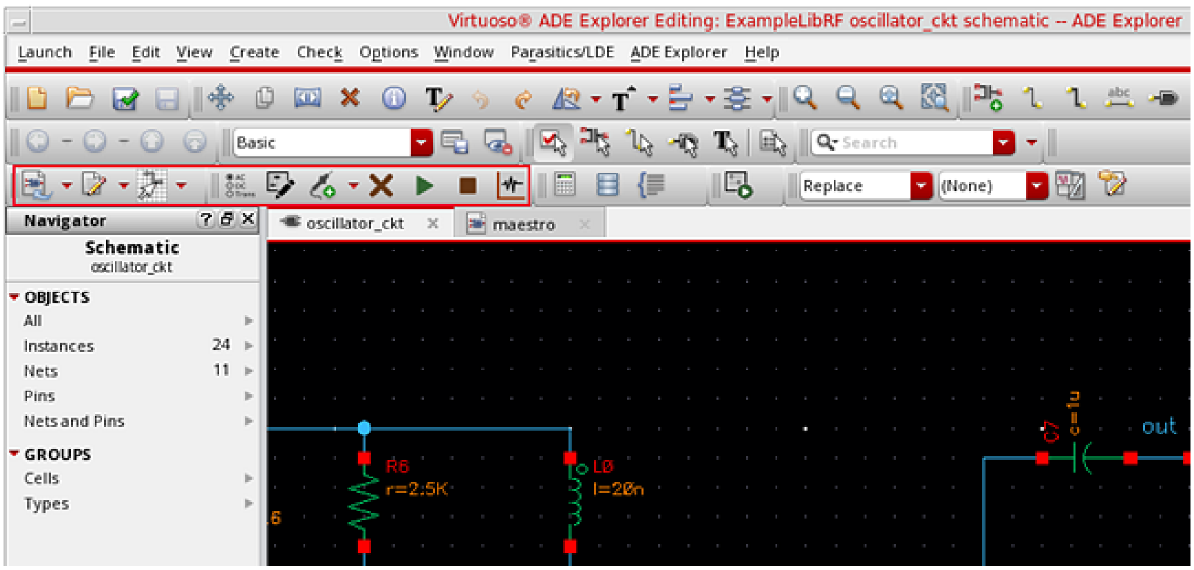
Note that the frequency of oscillation based on linear *stb* analysis is 1.89GHz. You'll refer to this later.

3. In the *Direct Plot Form*, click *Cancel*. In the ViVA window, select *File - Close All Windows*.

Plotting up the hbstb Analysis Results

Next plot the *hbstb* Loop Gain magnitude and phase. This time you will plot the results from the Schematic Window. Once you open an ADE Explorer session/window from Virtuoso Schematic Editor L Window, the Setup and Run and Results toolbar gets added to this Schematic Editor Window, as shown in the figure below:

Figure 3-58 ADE Explorer menu in VSE L Window



You may refer to *Virtuoso ADE Explorer User Guide* for more information regarding these simulation menu icons.

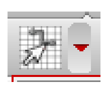
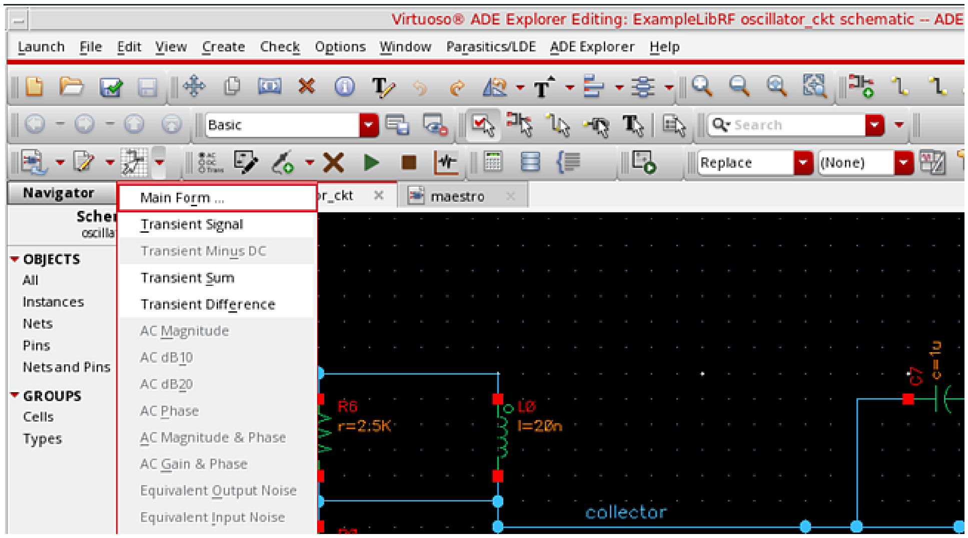
Click the red arrow  in the icon  located at the right. This will open the Direct Plot menu options. Select *Main form*, as shown below:

Figure 3-59 Running Simulation from VSE L Window

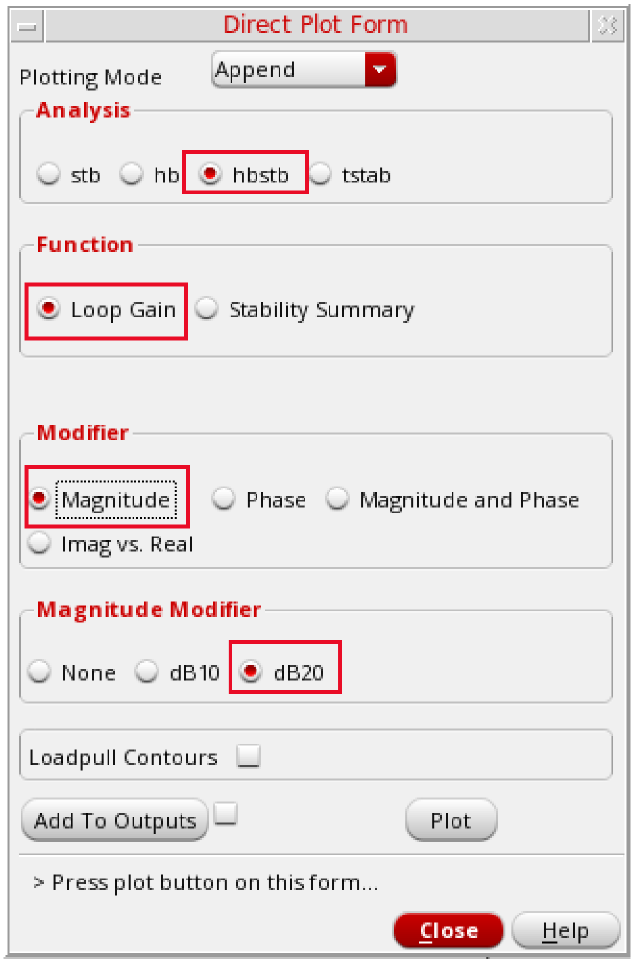


The *Direct Plot Form* is displayed.

1. In the *Direct Plot Form*, select *hbstb* in the *Analysis* section.
2. Select *Loop Gain* in the *Function* section.
3. Select *Magnitude* in the *Modifier* section.
4. Select *dB20* in the *Magnitude Modifier* section.

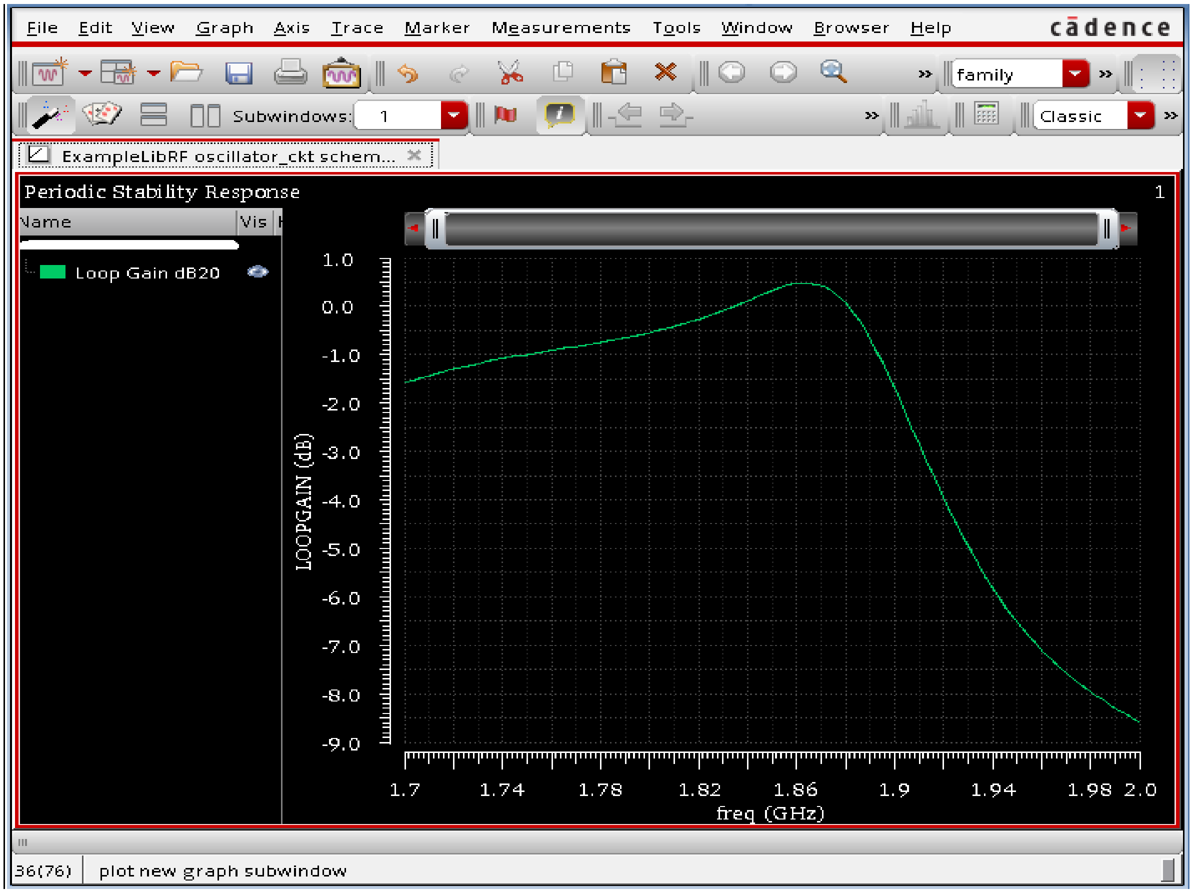
   The *Direct Plot Form* looks like the following:

Figure 3-60 HBSTB Direct Plot Loop Gain Magnitude Setup



5. Click *Plot*. The Plot Window will be displayed, as shown below.

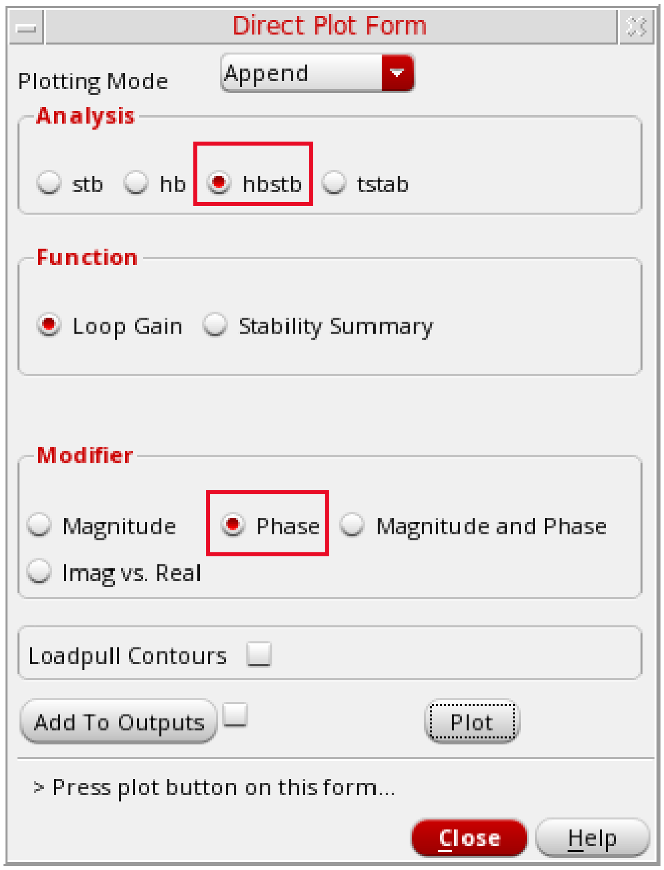
Figure 3-61 HBSTB Loop Gain Magnitude Plot



6. Next, plot the hbstb Loop Gain phase. Select *Phase* in the *Modifier* section with *Loop Gain* selected in the *Function* section.

   The *Direct Plot Form* looks like the following:

Figure 3-62 HBSTB Direct Plot Loop Gain Phase Setup



7. Click *Plot*. The Graph Window will look like the following:

Figure 3-63 *hbstb* Analysis Loop Gain Magnitude and Phase Plot

Note: Loop Gain's Magnitude and Phase are plotted separately as when *Magnitude and Phase* is selected, there is no dB20 modifier for the loop gain.

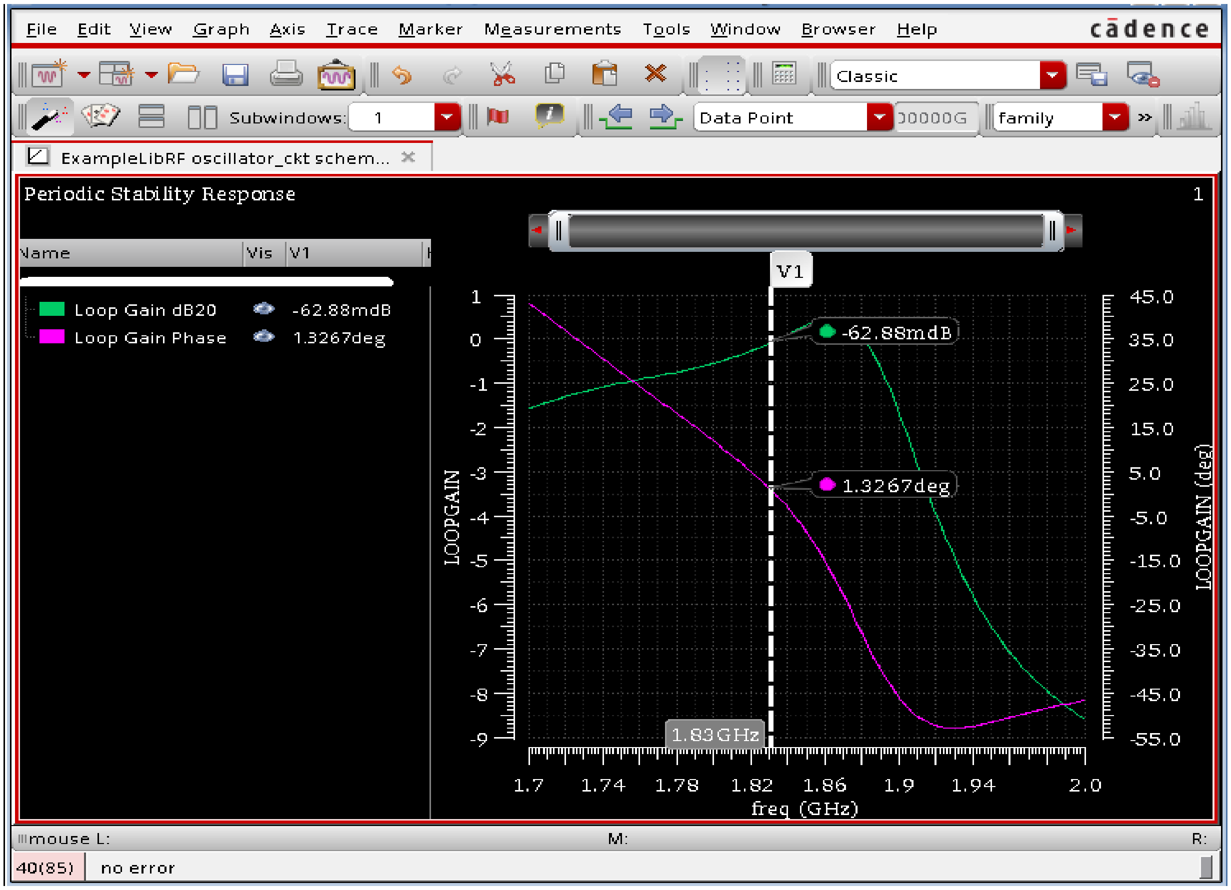
8. Next, you need to place a vertical marker at zero degrees on the plot and get the marker table. To do this:

a. Type v to place a vertical marker.

b. Select the vertical marker, click and hold the left mouse button, and place it at near zero degrees.

The Plot will look like the following:

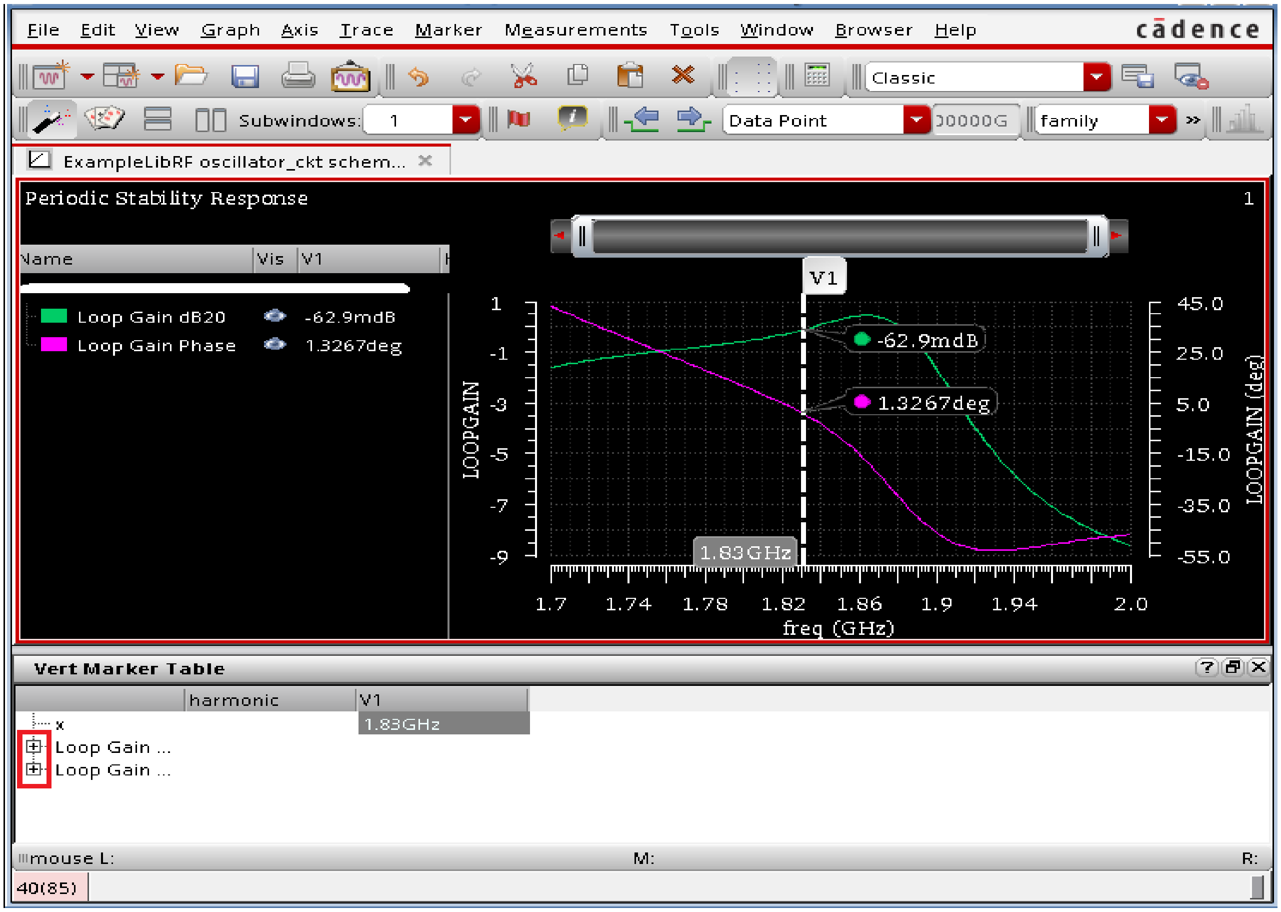
Figure 3-64 hbstb Loop Gain magnitude and phase plot with vertical marker



9. Select *Window - Assistants - Vert Marker Table*.

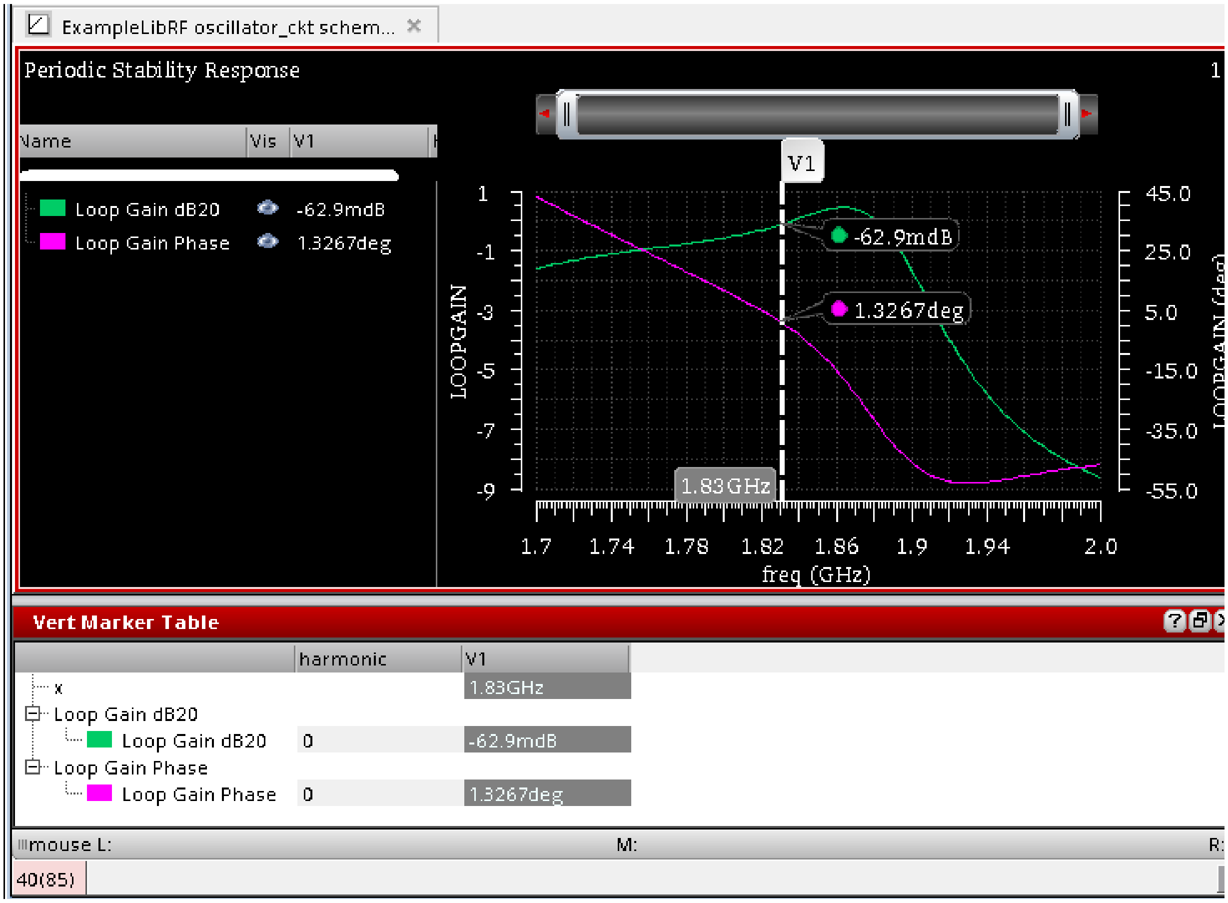
The Marker Table window is displayed with the data for the phase and gain curve intercepts along with the frequency.

Figure 3-65 hbstb Analysis Loop Gain Plot with unexapnded Verticle Marker Table



10. Note that in the plot window, initially only frequency value is visible and the Loop Gain and Phase values are blank. Click the + sign, located to the left of the Loop Gain dB20 & Loop Gain Phase entry to expand them and see their respective values, as shown in the figure below.

Figure 3-66 pstb Analysis Loop Gain Plot with Vertical Marker Table



Note that the Loop Gain magnitude is very near unity gain.

Also note that the oscillation frequency determined from *hbstb* analysis is 1.83GHz, slightly different than the oscillation frequency 1.89GHz determined from linear stability analysis, that is, *stb*. Linearizing about a periodically time-varying operating point in *hbstb* analysis allowed the stability evaluation to include the effect of the time-varying operating point as mentioned earlier.This has changed the oscillating frequency a little in comparison to the one determined using *stb*.

Next, hb Analysis Results will be plotted. You would note that how well pss results agree with the *hbstb* results.

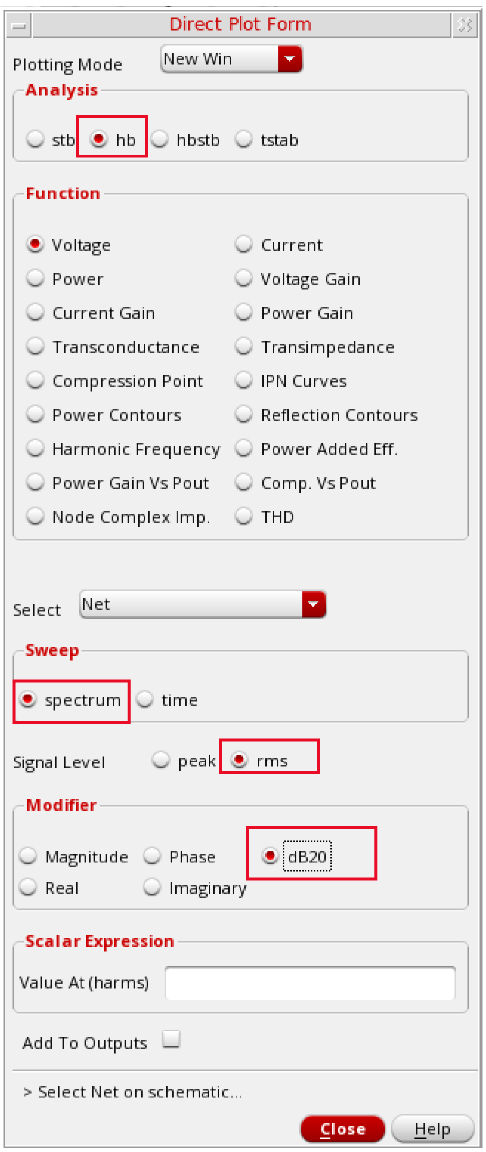
Plotting up the PSS Analysis Results

Next plot the oscillator spectral content -

1. In the Direct Plot Form window, set the *Plotting Mode* to *New Win*.
2. Select *hb* as *Analysis*.

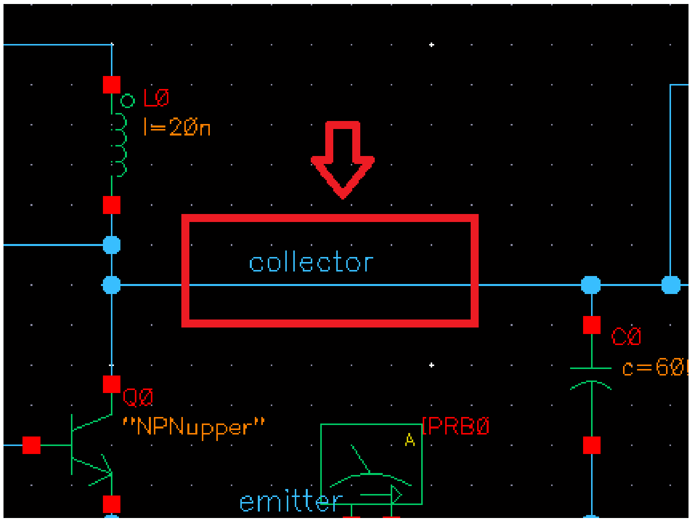
3. Leave *Function* as *Voltage* which is set by default.
4. Select *Net* in the center of the form. (This is the default. You can also select differential nets).
5. Select Sweep as spectrum. (This is the default)
6. Select *rms* as Signal Level (default is *peak*).
7. Select *dB20* as Modifier.
8. Your *Direct Plot Form* should like the following:

Figure 3-67 HB Analysis Direct Plot Form Setup



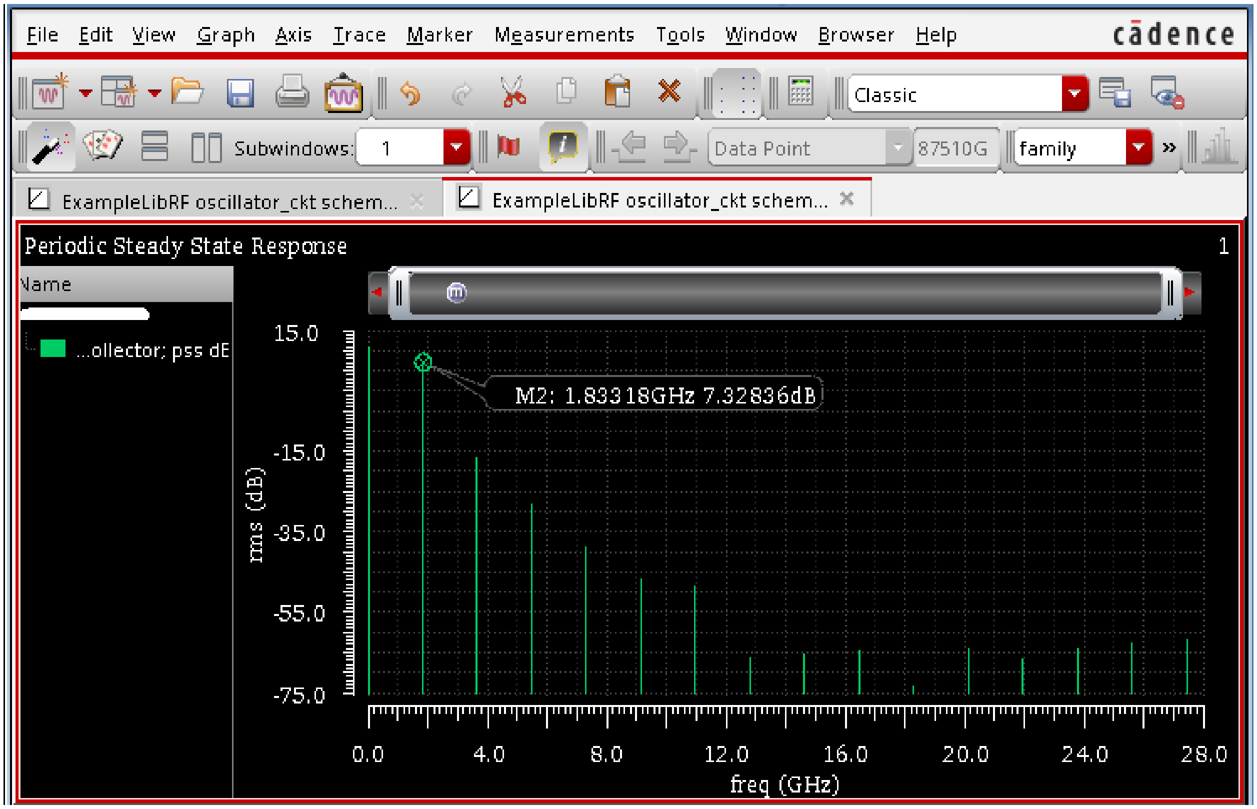
9. Select the *collector* net in the Schematic. It is located just below the *collector* label.

Figure 3-68 Selecting *collector* net on oscillator_ckt schematic



The waveform window is displayed, as shown below.

Figure 3-69 HB Analysis output Graph Window - Voltage Spectrum Plot



10. In the waveform window, position your cursor near the first harmonic, and press the *m* key. Here *m* is the bindkey to place a trace marker on the graph. The first harmonic is chosen as this is the frequency oscillator is designed for.

    Note that this frequency is 1.83075G. This is the frequency of oscillation.

    Note that the oscillation frequency determined from pstb analysis agrees very well with the pss-hb analysis.

11. In the *Direct Plot Form*, click *Cancel*. In the ViVA window, choose *File - Close All Windows*.

12. Close the Analog Design Environment window by selecting *Session - Quit*.

13. In the Schematic window, choose *File - Close*.

In this section, the B magnitude and phase measurements were done using *stb*, *hb*, and *hbstb* analysis. In addition, the oscillation frequency was determined using these measurements.

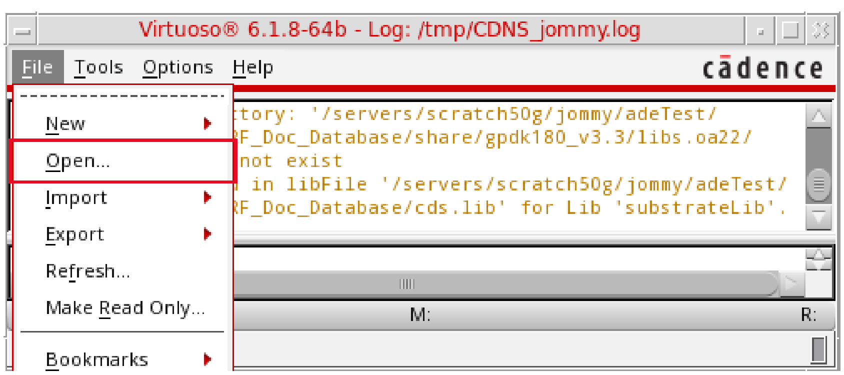
Next, the phase noise measurement which is one of the key measurement in oscillator design will be done. Also, the Noise Summary Table will be obtained.

Phase Noise Measurement and Noise Summary Table

Opening the Oscillator Circuit in the Schematic Window

1. In the Command Interpreter Window (CIW), choose *File - Open*.

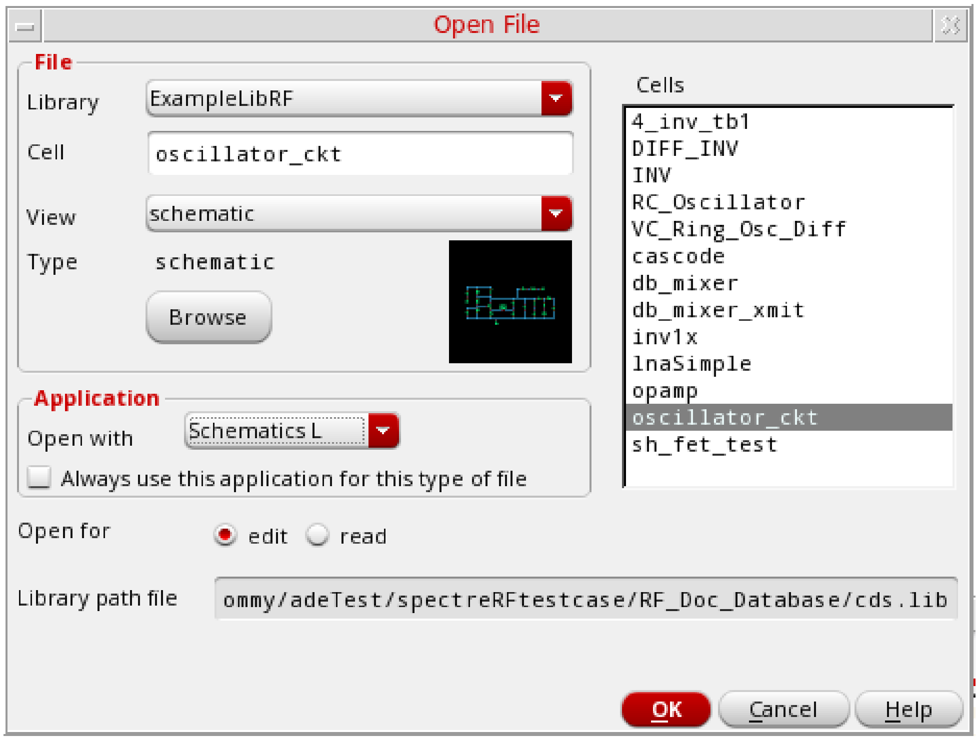
Figure 3-70 Virtuoso CIW Window - Opening Cellview



The *Open File* form is displayed.

2. Select *ExampleLibRF* from the *Library* drop-down list.
3. In the *Cells* field, type *oscillator_ckt*.
4. Select *schematic* from the *View* drop-down list.
5. In the *Application* section, select *Schematic-L* from the *Open With* drop-down list.
6. Leave *Open For* to *Edit* (which is set by default).

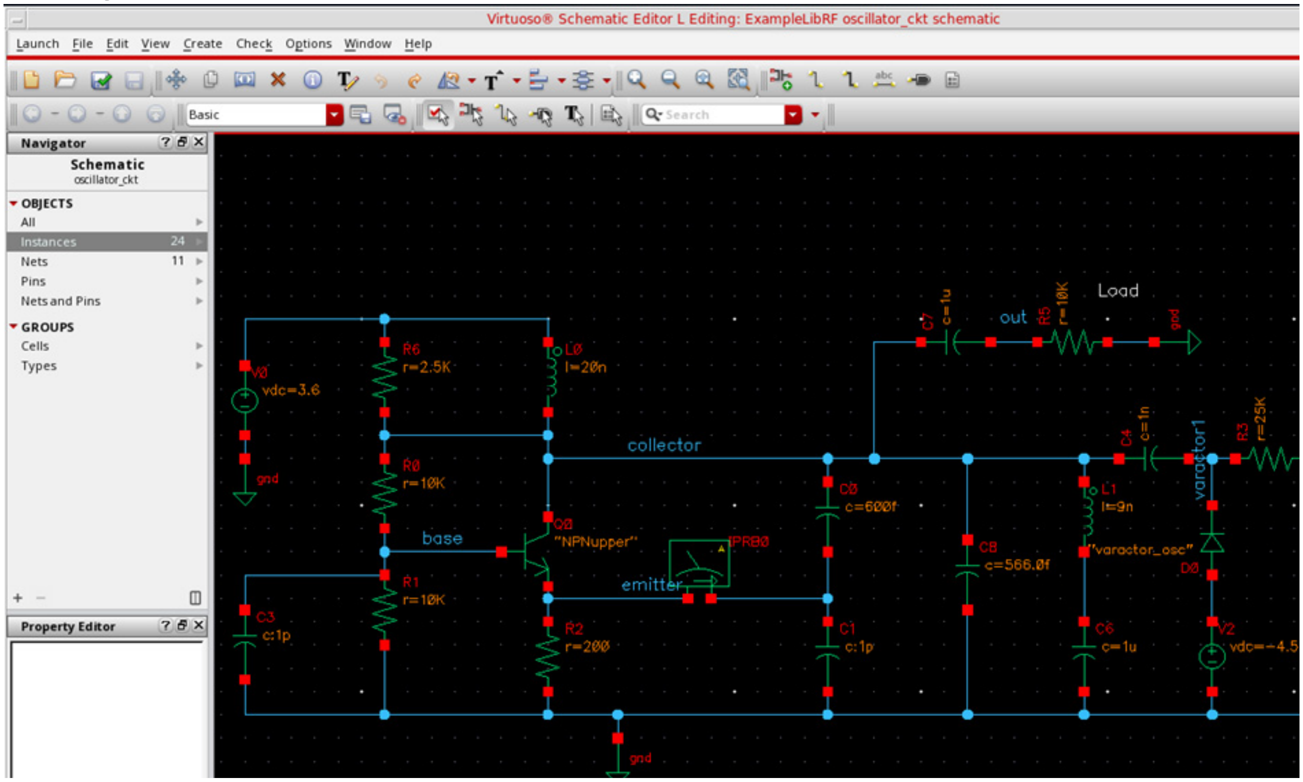
Figure 3-71 Open File Form to open the oscillator_ckt cell's Schematic View



7. Once all the setup is done, click *OK* to close the *Open File* form.

   This will open the oscillator_ckt schematic in Virtuoso Schematic Editor L window, as shown below.

Figure 3-72 Oscillator Schematic opened in Virtuoso Schematic Editor L Window



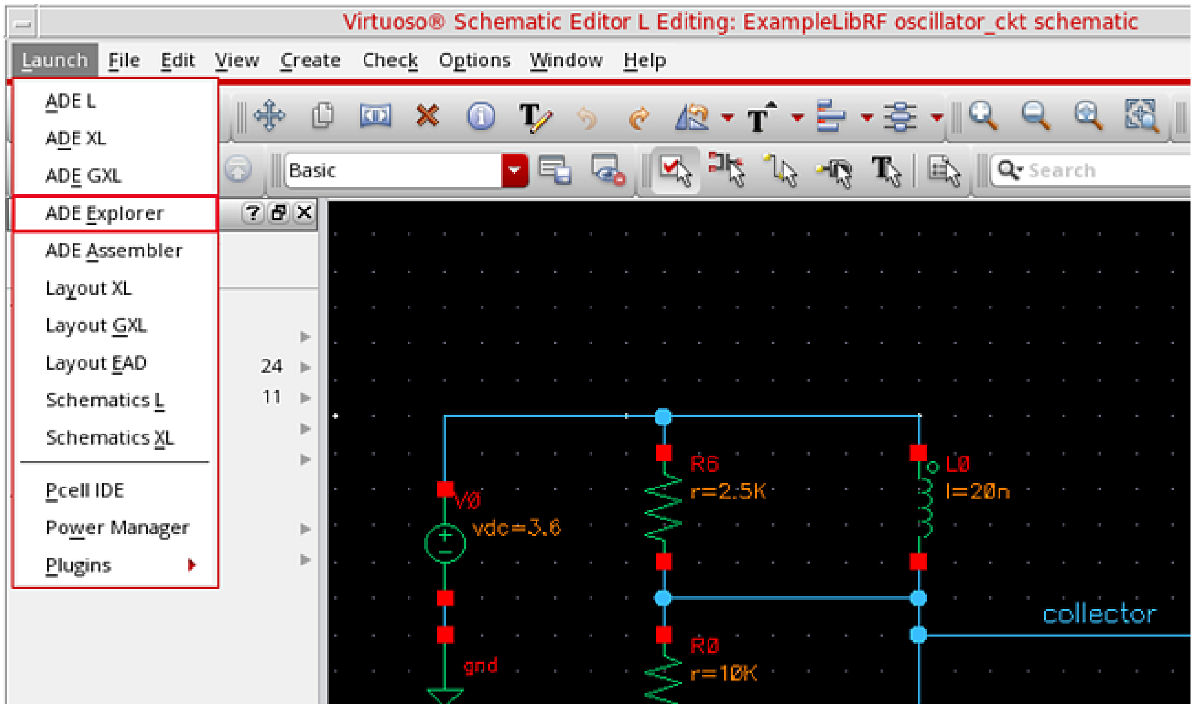
Setting up the HB and HBnoise Analysis

In this section, for performing phase noise measurement, you would be using the *hb* and *hbnoise* analyses.

Setting up the HB Analysis

1. In the Schematic Window, choose *Launch - ADE Explorer.*

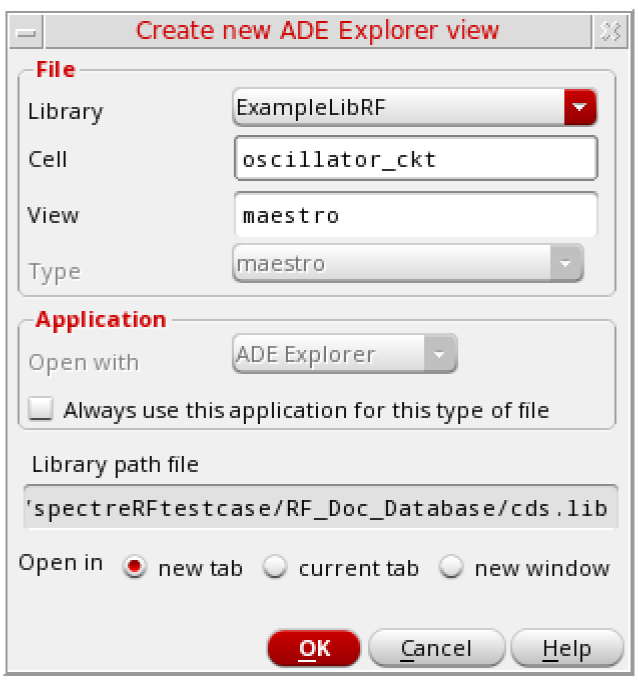
Figure 3-73 Opening ADE Explorer window from VSE window



2. In the *Launch ADE Explorer* dialog, select *Create New View*.

   The *Create new ADE Explorer view* form is displayed.

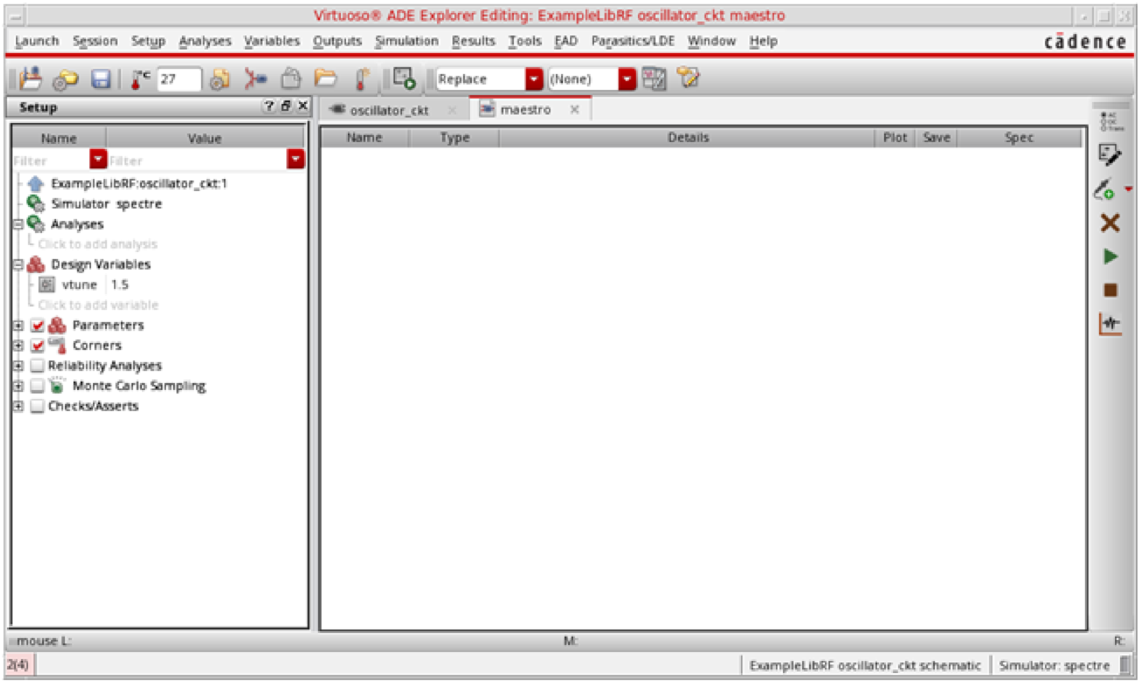
Figure 3-74 Create new ADE Explorer view



3. Leave each option to the default selections and click *OK*.

   The *ADE Explorer* window is displayed, as shown below.

Figure 3-75 Virtuoso ADE Explorer Window



4. In ADE Explorer, select *Setup – Simulator*.

   The *Choosing Simulator* form is displayed.

5. Select *spectre* as the *Simulator.*

Figure 3-76 Choosing Simulator Form

6. Click *OK*.

7. Select *Setup - High Performance Simulation*. The *High Performance Simulation Options* window is displayed.

Figure 3-77 High Performance Simulation Options Form

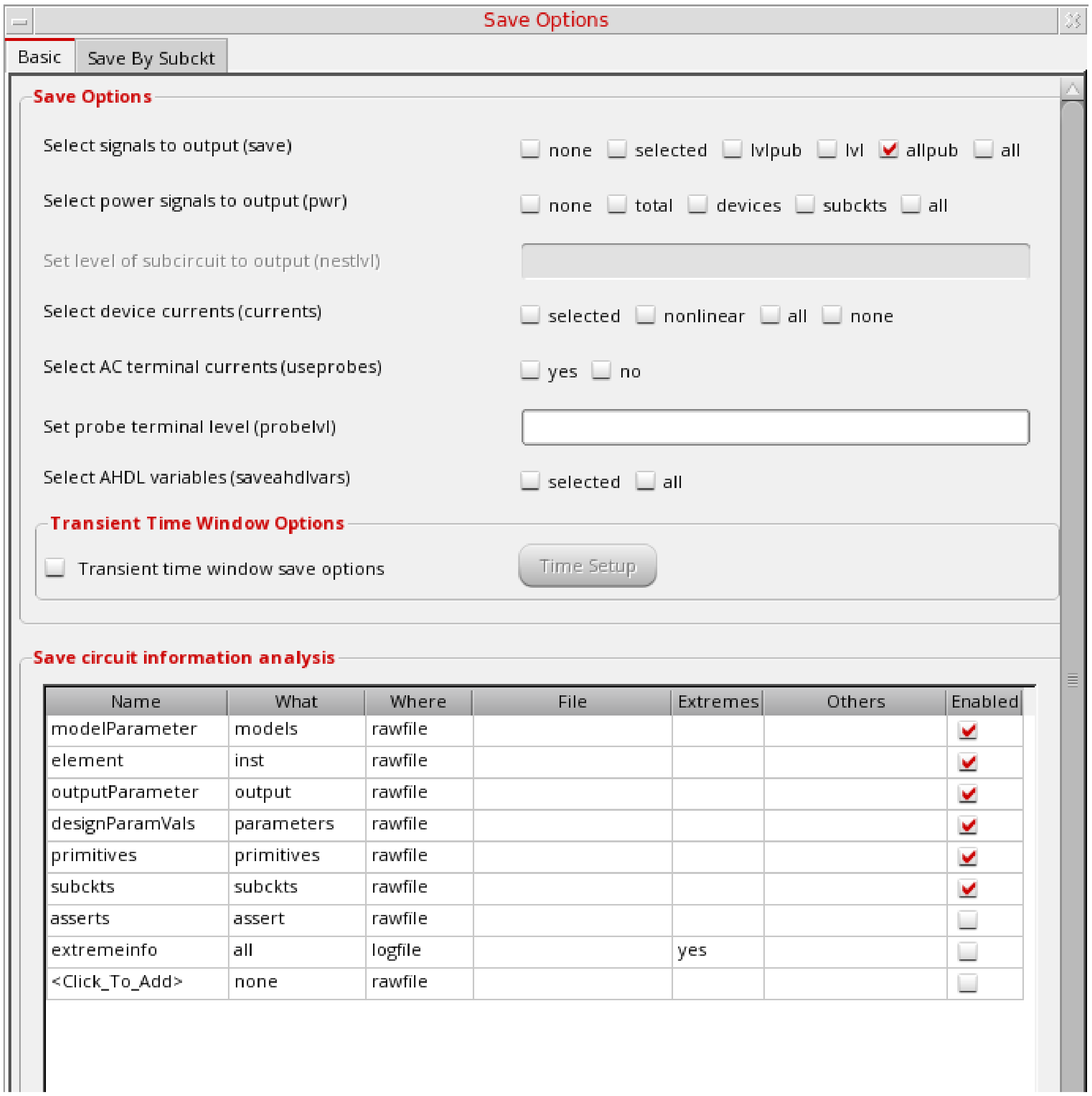
8. In the *High Performance Simulation Options* window, select *APS*. Note that *Auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores. Usually, it is better to specify the number of threads yourself. Small circuits should use a small number of threads, and larger circuits can use more threads. The overhead of managing 16 threads on a smaller circuit may actually slow the simulation down, compared to two or four threads. Use the number of threads that maximizes the performance.

9. Click *OK* to close the *High Performance Simulation Options* form.

10. In ADE Explorer, select *Outputs - Save All*.

The *Save Options* form is displayed.

Figure 3-78 Save Options Form



11. In the *Select signals to output section (save)*, ensure that *allpub* is selected.

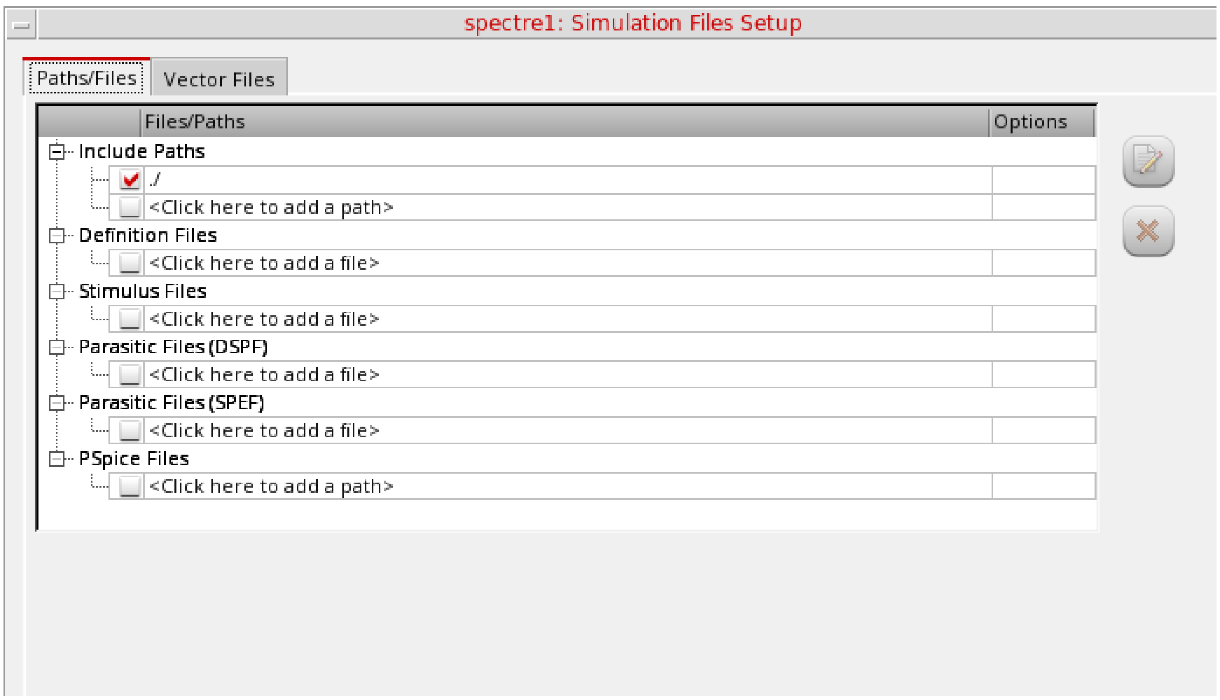
This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

To save the currents, use the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or all if you want to save all the currents in the circuit. When you save currents, more disk space is required for the results file.

12. Click *OK*.
13. Select *Setup - Simulation Files.*

    The *Simulation Files Setup* form is displayed, as shown below.

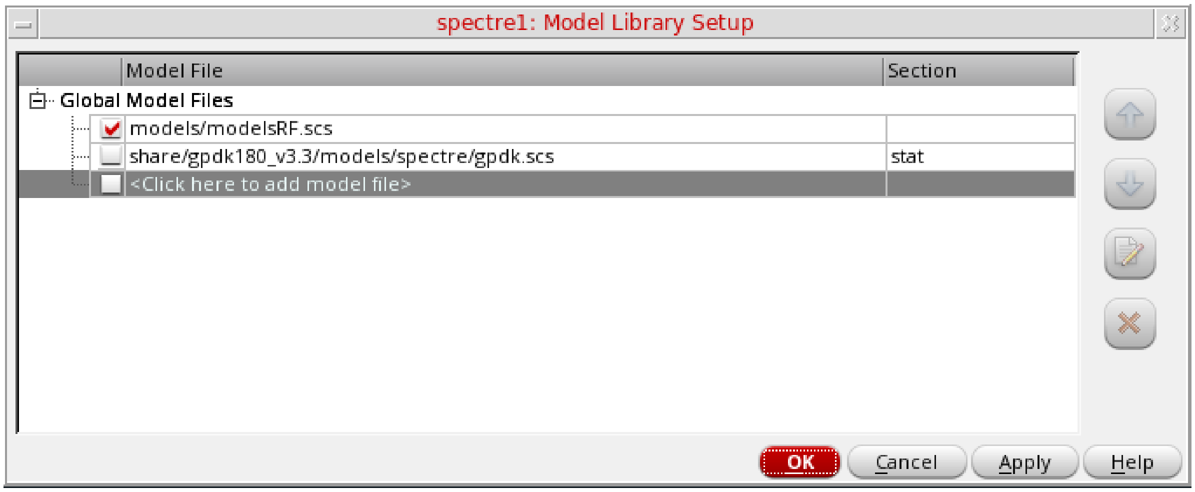
Figure 3-79 *Simulation Files Setup* Form



14. In the *Simulation Files Setup* form, type `./` by clicking in the *Include Paths* section.
15. Click *OK* to close the *Simulation Files Setup* form.
16. Select *Setup – Model Libraries*.

    The *Model Library Setup* form is displayed.

Figure 3-80 *Model Library Setup* Form



17. In the *Model File* field, type the path to the model file including the file name, as shown below:

```
models/modelsRF.scs
```

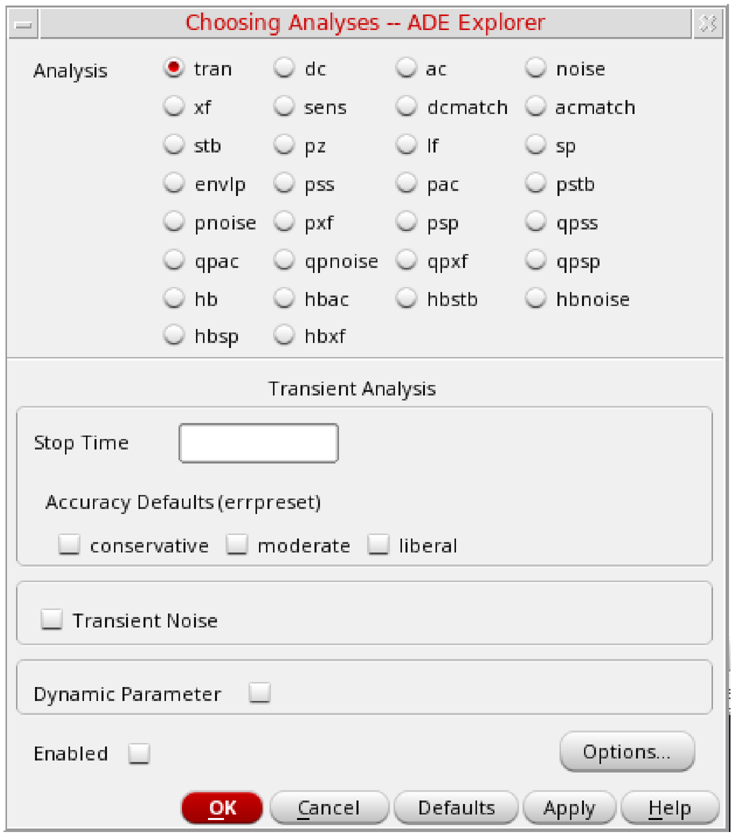
You can also browse to the *modelsRF.scs* file.

18. Click *OK* to close the *Model Library Setup* form.

19. Select *Analyses - Choose*.

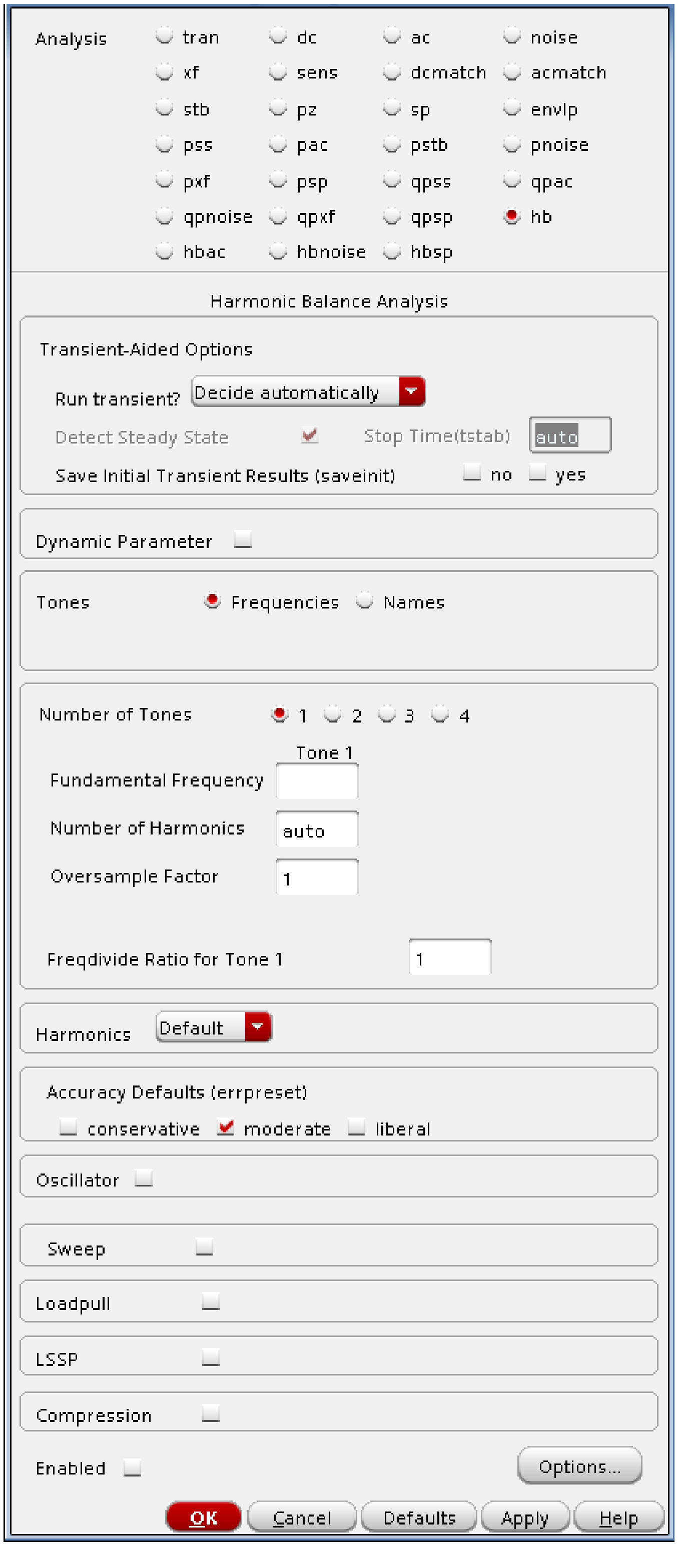
The *Choosing Analyses* form is displayed, as shown below.

Figure 3-81 The Choosing Analyses Form



20. Select *hb* in the *Analysis* section. The form expands, as shown below.

Figure 3-82 The Choosing Analyses Form - Setting *hb* analysis



21. In the *Transient-Aided Options* section, leave *Run Transient?* as *Decide automatically,* which is set by default.

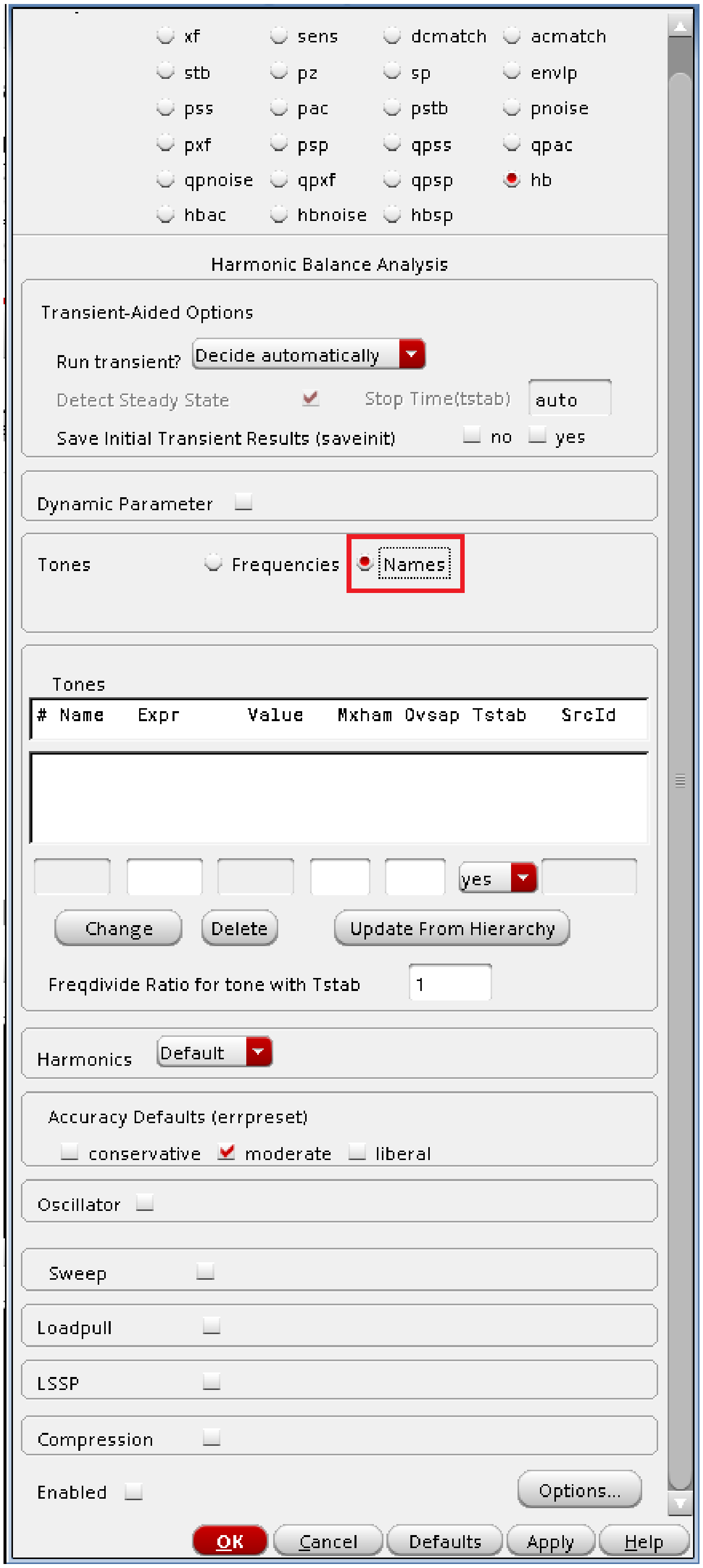
The other choices for this option are *Yes* and *No*. If *Yes* is used then the *Detect Steady State* option becomes active and the *Stop Time(tstab)* field is also activated. This means that you can decide whether you would like the Steady State to be detected automatically during the transient run or not and also specify a *Stop time(tstab)* for that transient run. When you select the checkbox for *Detect Steady State* option, this will run *the* transient analysis until steady-state is detected and then switches to *hb*.

22. Select *yes* for *Save Initial Transient Results* (*saveinit*). This will help in visualizing the buildup of the oscillation waveform.

23. Select *Names* in the *Tones* section. The other option is *Frequencies,* which is selected by default.

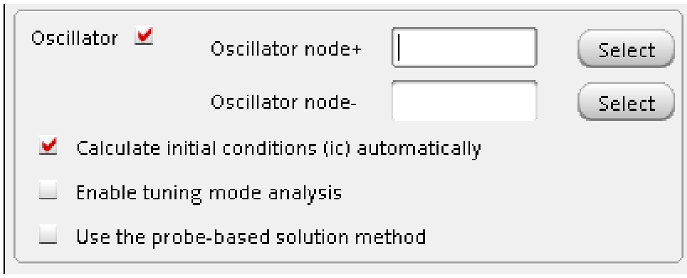
Names is chosen here as it is similar to *pss* Harmonic Balance analysis setup. The form changes, as shown below.

Figure 3-83 The Choosing Analyses Form - Setting *hb* analysis using *Names*



24. Select the *Oscillator* option. This is required for simulating an autonomous circuit.

Figure 3-84 The Choosing Analyses Form - Oscillator Section

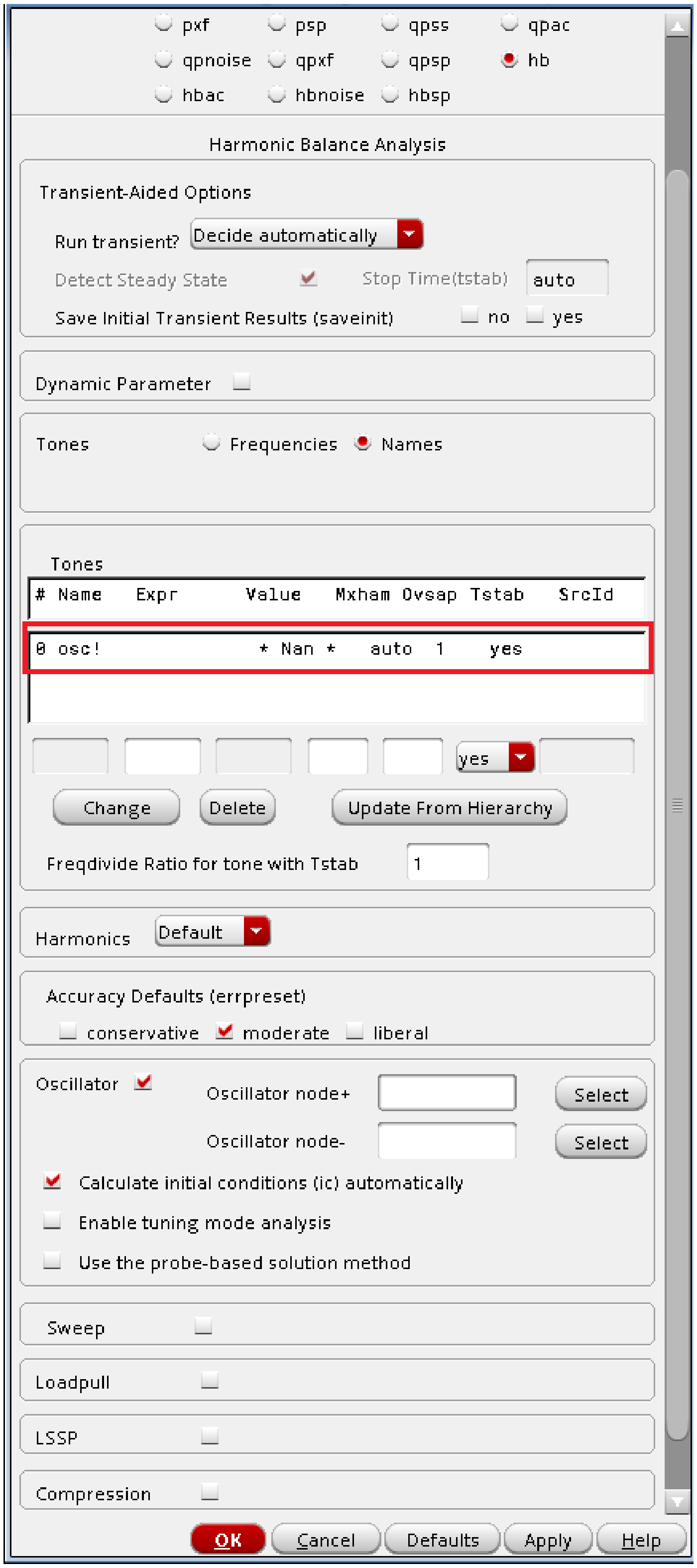


25. A popup window, as shown below appears. This informs about creation of an osc! frequency line entry in the *Tones* field. Click *Close* to close the window.

Figure 3-85 Popup Window - osc! entry creation message

Closing the above popup window will result in creation of the osc! frequency line in the *Tones* section, as shown below.

Figure 3-86 Choosing Analyses Form - *hb* autonomous setup



26. Select the *osc!* row in the *Tones* section.

  a. Enter `1.9G` in the *Expr* field.

  b. Leave the *Number of Harmonics* option set to *auto*. When *Number of Harmonics* is set to *auto,* the simulator calculates harmonics automatically. The calculation is based on Fourier analysis of transient steady-state waveforms.

  c. Leave the *Ovsap* option as set to 1. Since the oscillator has sinusoidal waveforms, an oversample of 1 is appropriate.

27. Enter `1` for the *Freqdivide Ratio for Tone with Tstab* option (which is set by default) as there is no frequency divider in the circuit. If there is a frequency divider in the circuit, then you need to set the *Freqdivide Ratio for Tone with tstab* to the divide ratio of the divider. For example, if the divider is divide-by-two, then the divide ratio is 2. Therefore, you will set *Freqdivide Ratio for Tone with tstab* to 2.

28. *Leave the Harmonics* option set to *Default*.

29. In the *Accuracy Defaults* (*errpreset*) section, select *conservative*.

conservative is typically used because very small amplitude phase noise measurements are normally desired. Conservative is recommended for all the oscillators.

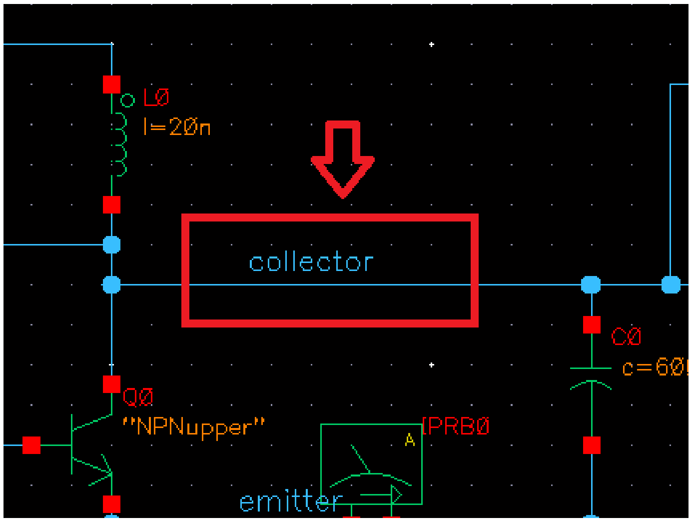
30. In the Oscillator section, in *Oscillator node+* field, click *Select* just to the right of this field. In the schematic, select the *collector* node. Instead of selecting the node from the schematic you can also type `/collector` in the *Oscillator node+* field.

This oscillator node will be used by the simulator for the period calculation of the oscillations. It just needs to be a node that has the oscillator signal on it.

Leave *Oscillator node-* blank.

Note that if you have a single-ended oscillator, only specify one node. If the second node that is, the reference node is left blank, it will be connected to the global ground node automatically. However, if you have differential oscillator, you need to specify both the nodes.

Figure 3-87 Selecting *collector* net on oscillator_ckt schematic



31. If you are simulating an LC oscillator, leave the *Calculate initial conditions (ic) automatically* checkbox selected (this is the default).

Note that *Calculate initial conditions (ic) automatically* is used to start the oscillator. Other methods to start the oscillator include putting a single current pulse into the resonator, setting initial conditions(ic), or ramping up the power at time = zero plus.

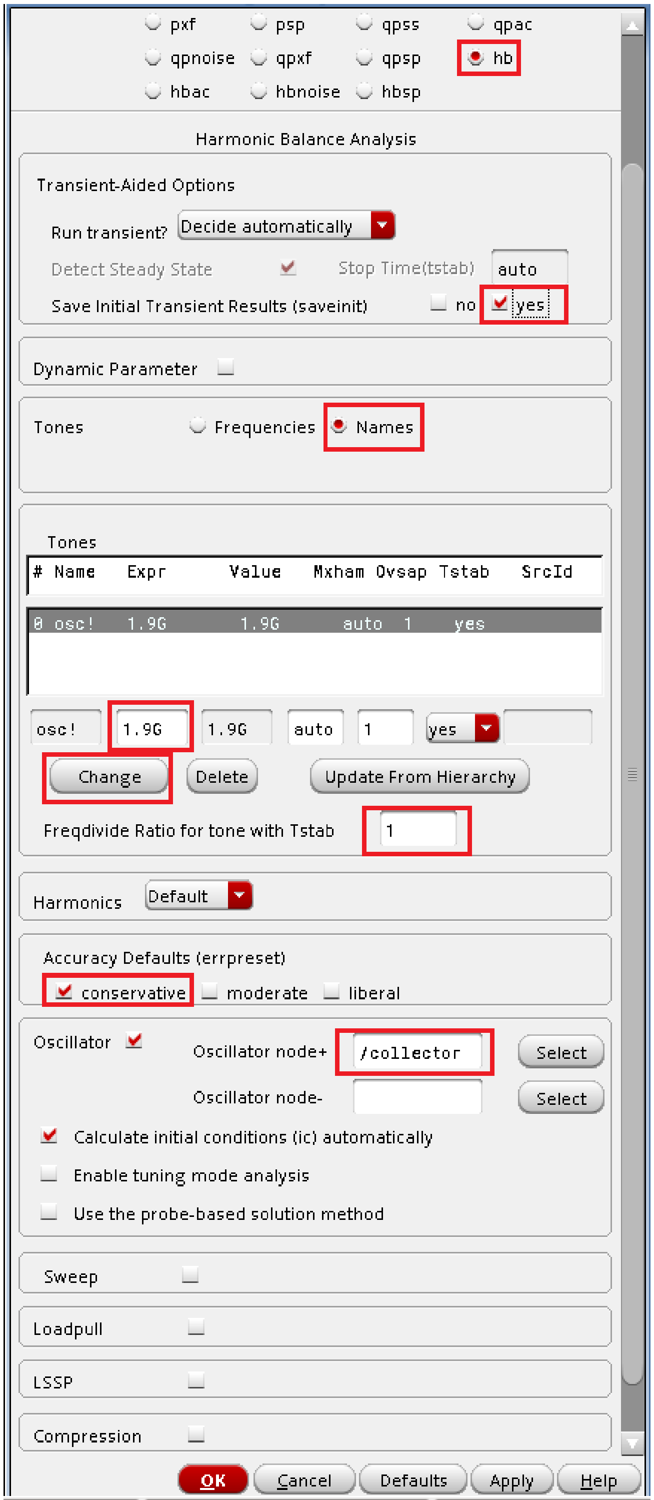
32. Note that, by default, the *Use the probe-based solution method (oscmethod)* option is deselected. Spectre will use the *onetier* method.

When the *Use the probe-based solution method (oscmethod)* option is selected, it iterates for the frequency solution in the outer loop and the amplitude and phase solution in the inner loop. The probe-based method has better convergence but is computationally intensive.

Refer to *Spectre Circuit Simulator RF Analysis Theory* for more details.

33. The *Choosing Analyses* form will look like the following:

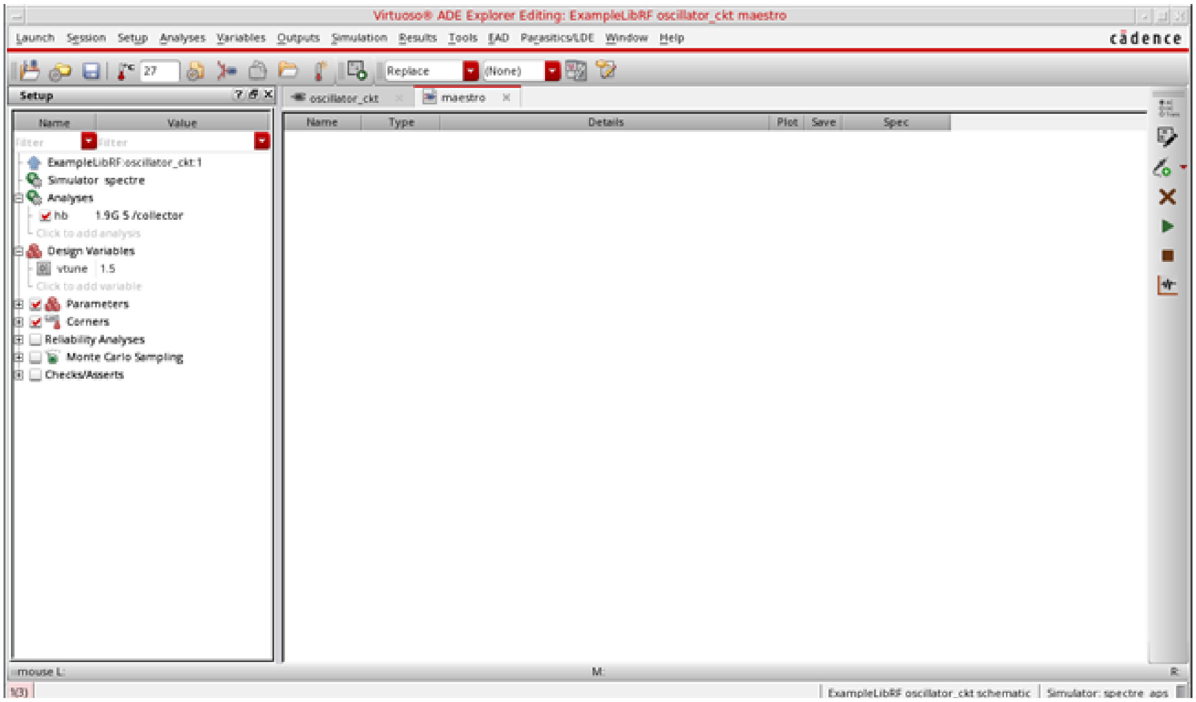
Figure 3-88 *Choosing Analyses* Form - HB Analysis Setup



34. Click *Apply*.

This will add hb analysis to the Analyses section of ADE Explorer, as shown below:

Figure 3-89 ADE Explorer Simulation Window - hb analysis setup



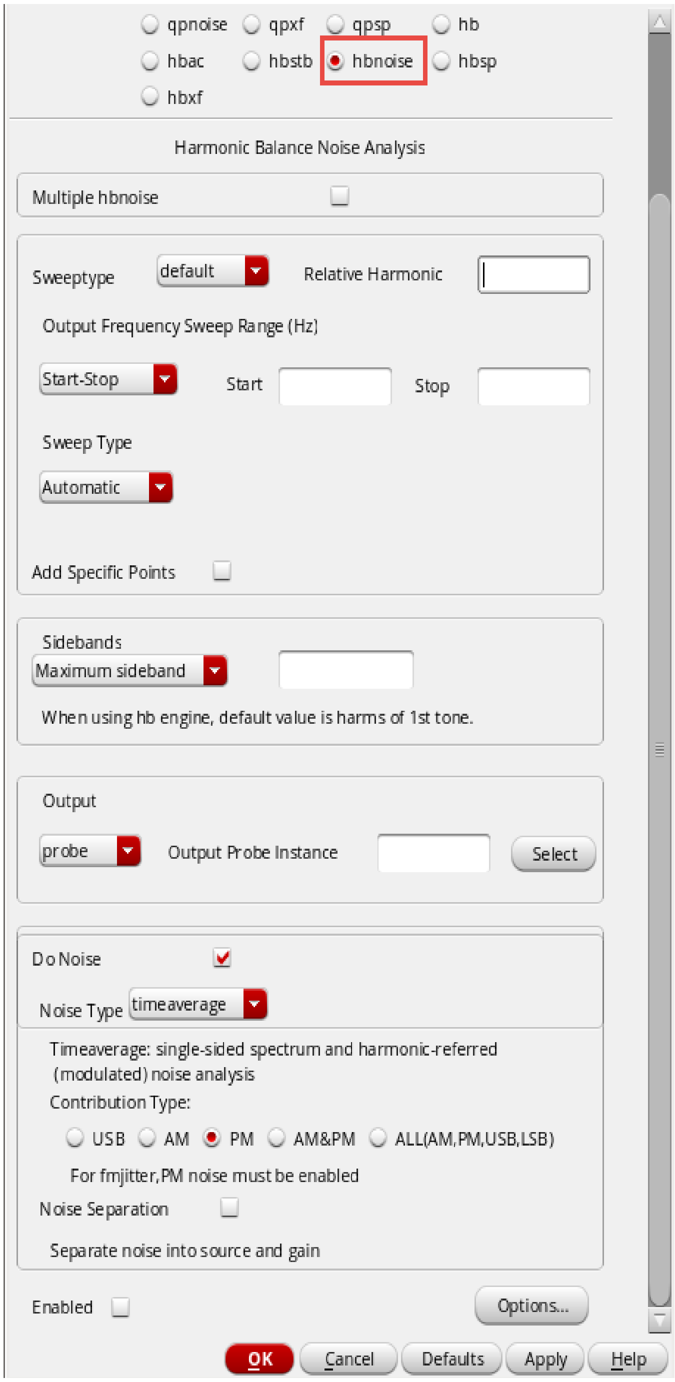
Setting up the *hbnoise* Analysis

Starting with IC617, the noise UI is re-designed. *modulated* has been removed and its functionalities are merged with *timeaverage*, leaving three noise types now. In addition, the FM jitter option is removed. It is calculated through timeaverage PM noise, which should be used when you want to get phase noise.

When *Noise Type* is set to *timeaverage*, USB, LSB, AM and PM noises are directly available.You can choose, for example, PM noise alone, or you can select the ALL(AM,PM,USB,LSB) option, if you need all four types of noise. The available options are *USB*, *AM*, *PM*, *AM&PM*, and *ALL(AM,PM,USB,LSB)*.

1. In the *Choosing Analyses* form select *hbnoise* analysis. The form expands, as shown below.

Figure 3-90 ***hbnoise*** **Choosing Analyses Form**



2. Set the *Sweeptype* to *relative*.

   For oscillators, the *hbnoise/pnoise* frequency range is *relative*. Specify the harmonic number as appropriate for the system you are simulating. If you are simulating an oscillator by itself, then the harmonic number is likely to be 1. If you have an oscillator and a diode frequency doubler, then the harmonic number is likely to be 2. If you have an oscillator with a frequency divider, in the hb/pss form, you should specify the approximate frequency of oscillation for the frequency-divided signal. In the *hbnoise/pnoise* form, if the noise is desired on the frequency divided output, then the relative harmonic is 1. If the noise is desired at the output of the oscillator, the relative harmonic number is the divide ratio. The meaning of *relative* is to take the frequency of the harmonic number specified and add to it the frequencies specified in the *Choosing Analyses* form. If the oscillator has a 1GHz output, and the pnoise had 1M relative to the first harmonic specified, the actual output frequency is 1G + 1M, or 1001M.

a. Type `1` in the *Relative Harmonic* field. You are trying to determine the noise associated with the fundamental frequency of the oscillator.

Next, you will set the output frequency sweep range. Frequency sweep is set for a noise simulation as the noise is spread over the frequency range and it can affect the adjacent frequency channels. Therefore, it is critical to determine its behavior over a frequency range.

b. Type `10` in the *Start* Field in the *Output Frequency Sweep Range (Hz)* section.

c. Type `100M` in the *Stop* Field in the *Output Frequency Sweep Range (Hz)* section.

Set the frequency sweep range as appropriate for your circuit (or application).

d. Set the *Sweep Type* to *Logarithmic*.

e. Type `3` in the *Points Per Decade* field. Typically, 3 to 5 points per decade are a reasonable number to capture the noise behavior of the circuit.

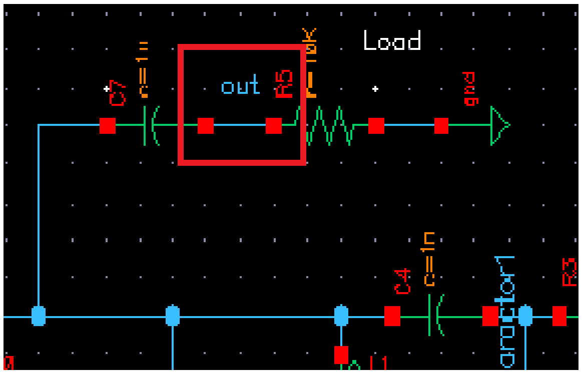
3. Leave the *Maximum Sideband* field blank. In general, *Maximum sideband* needs to be set high enough to include all the frequencies that could mix down to the oscillator output frequency. By default, when this field is left blank, all the mixing with all the *hb* harmonics are present in the *hbnoise* result.

4. Set the *Output* to *voltage*.

a. Type `/out` in the *Positive Output Node* field. You can also select the *out* net from the schematic by clicking the *Select* button on the right of the *Positive Output Node* field and then selecting the net just below the *out* label in schematic.

b. Leave the *Negative Output Node* field blank. If the second node that is the *Negative Output Node* is left blank, it will be connected to the global ground node automatically. However, if you have a differential oscillator, you need to specify both the nodes.

Figure 3-91 Selecting *out* net from the schematic

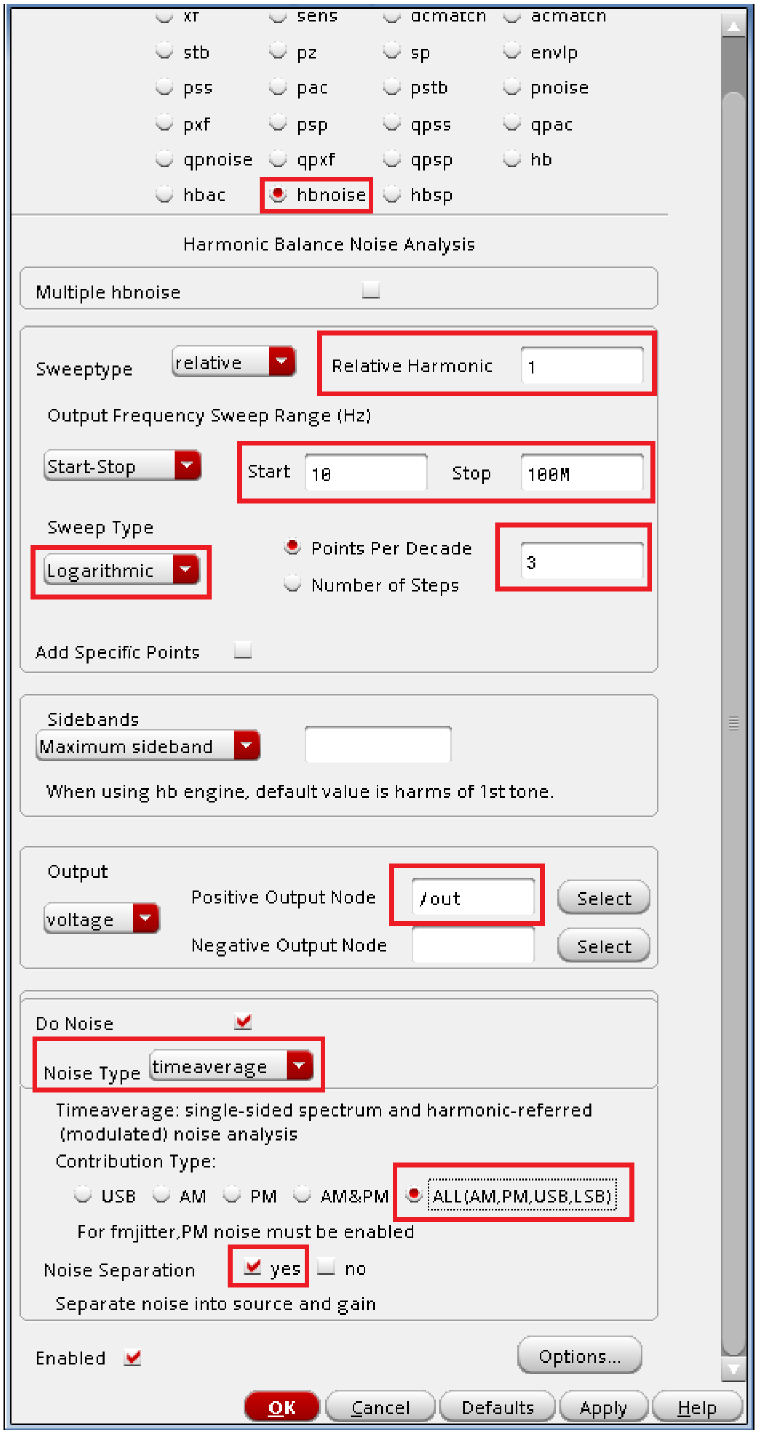


5. The *Do Noise* option is selected by default. Next, you will set the *Noise Type*.
6. Set the *Noise Type* to *timeaverage*. Here, the other available options are *jitter and timedomain*.
7. Select the *ALL(AM,PM,USB,LSB)* option.

   Selecting the *ALL(AM,PM,USB,LSB)* will enable you to view all noise components on the *Direct Plot* form.
8. Set *Noise Separation* to *yes*.

   The *Noise Separation* option enables you to plot the individual noise contributors.
9. The Choosing Analyses form should look like the following:

Figure 3-92 Choosing Analyses Form - *hbnoise* Analysis Setup



10. You now need to set up the *lorentzian* option for the *hbnoise* analysis.

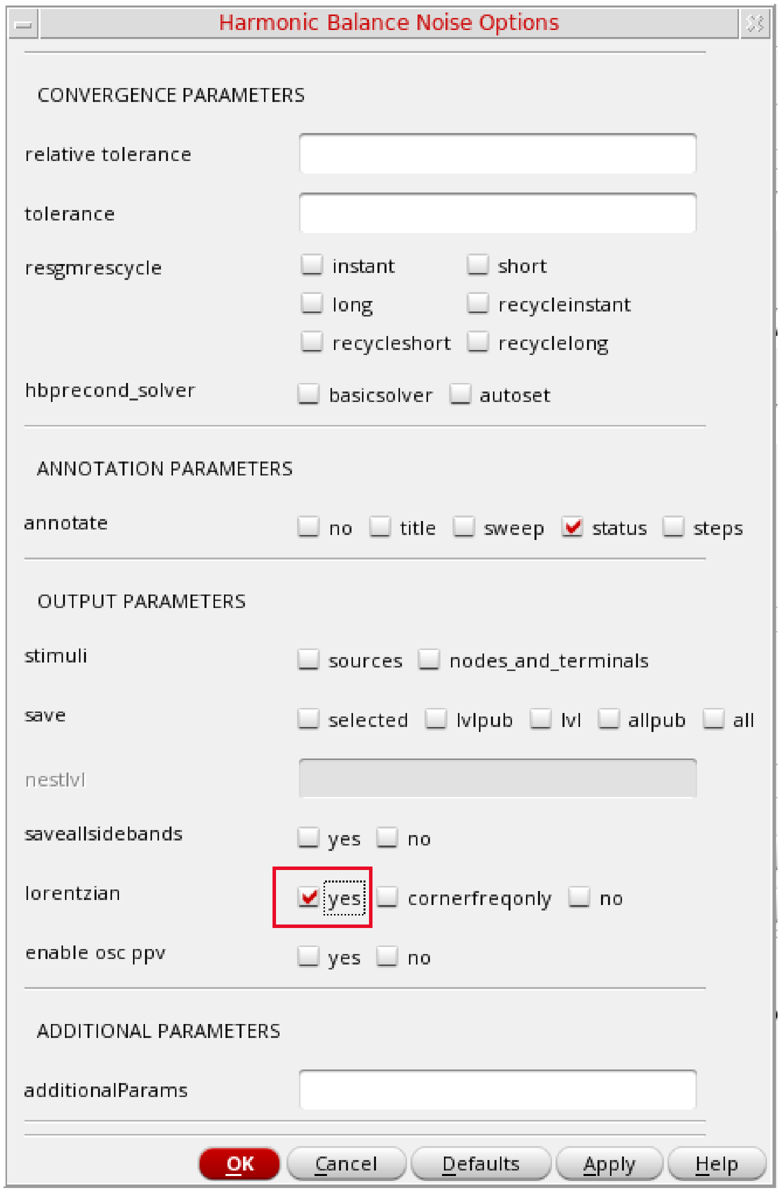
HBnoise/Pnoise is a small-signal analysis and is not limited by large-signal effects, such as clipping or slew-rate limits. As a result, at low offset frequency, the phase noise might be significantly greater than 0dBc/Hz. This indicates that the noise is larger than the oscillations, which is not physically possible. If you want to see the phase noise curve level off at low frequency, set the *lorentzian* option to yes.

a. At the bottom of the *Choosing Analyses* form, select *Options*. This will open *Harmonic Balance Noise Options* form.

b. Select *yes* for *lorentzian* option in the *OUTPUT PARAMETERS* section of this form.

The *Harmonic Balance Noise Options* Form should look like the following:

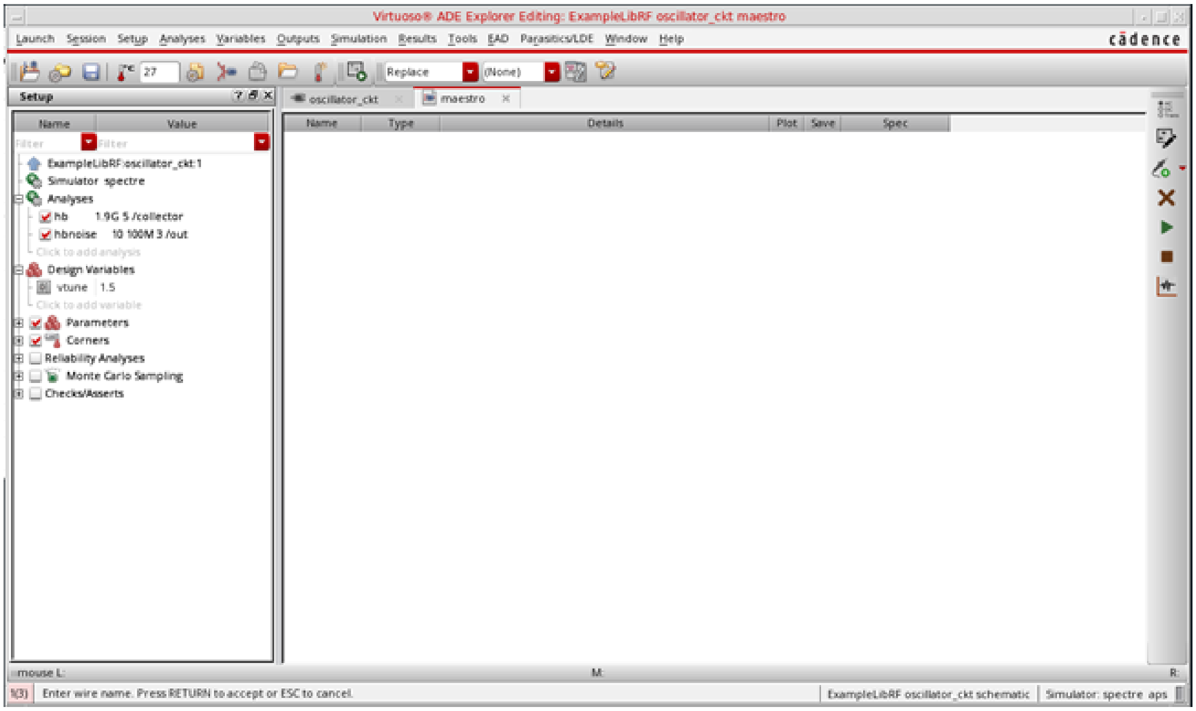
Figure 3-93 ***hbnoise*** **Options Setup Form**



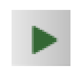
11. Click *OK* to close the *Harmonic Balance Noise Options* form.

12. Next click *OK* to close the *Choosing Analyses* form. This will also add the *hbnoise* analysis in addition to the *hb* analysis in the ADE Explorer *Analyses* section, as shown below.

Figure 3-94 ADE Explorer Simulation Window - *hb* and *hbnoise* analysis setup



Running the simulation

Once finished setting up the *hb* and *hbnoise* analyses. Click the green icon  located on the right side ADE Explorer or on the Schematic window to run the simulation.

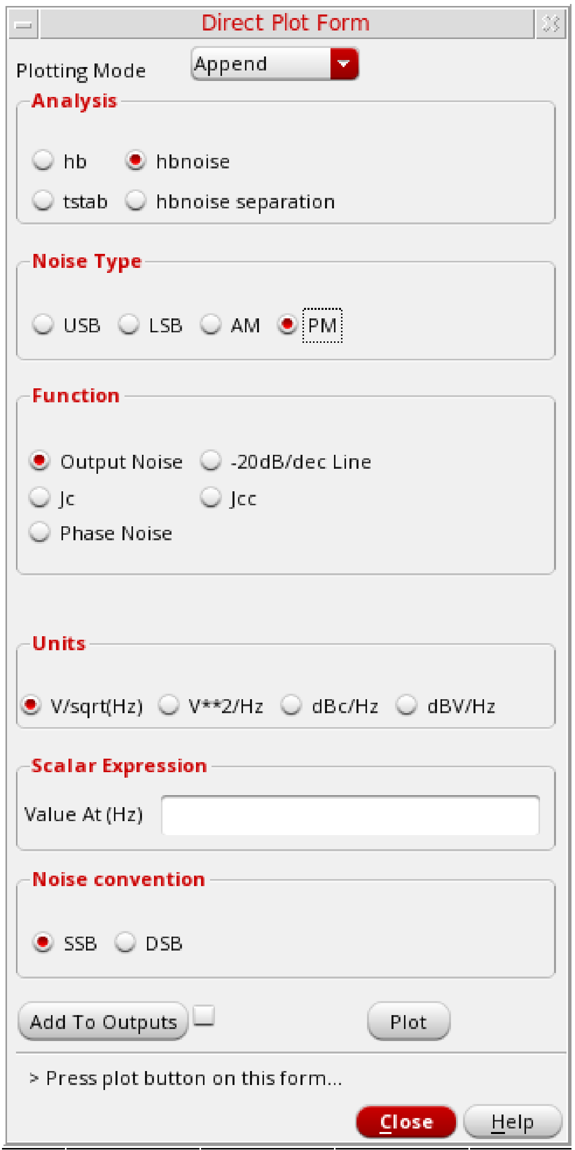
Next, plot the phase noise results and obtain the *Noise Summary* Table once the simulation is finished.

Plotting the results from *hb and hbnoise* analysis

First plot the SSB (single sideband) phase noise, as follows:

1. In ADE Explorer, select *Results - Direct Plot - Main Form*.

Figure 3-95 ***hb*** **and** ***hbnoise*** **Direct Plot Form**

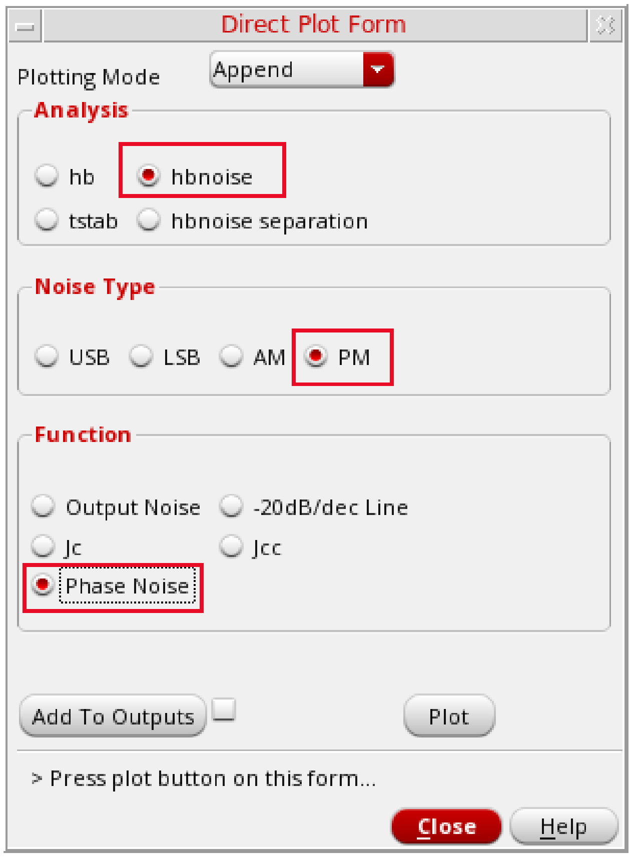


Note that in the Direct Plot form, *hbnoise separation* analysis is added as you have set *Noise Separation* to *yes* in the *hbnoise* analysis setup form and *tstab* is added as you have set *Save Initial Transient Results (saveinit)* to *yes* in the *hb* analysis setup form.

2. Select *hbnoise* as the Analysis.
3. Select *PM* as the Noise Type.
4. Select *Phase Noise* as the Function.

The *Direct Plot Form* Setup should look like the following:

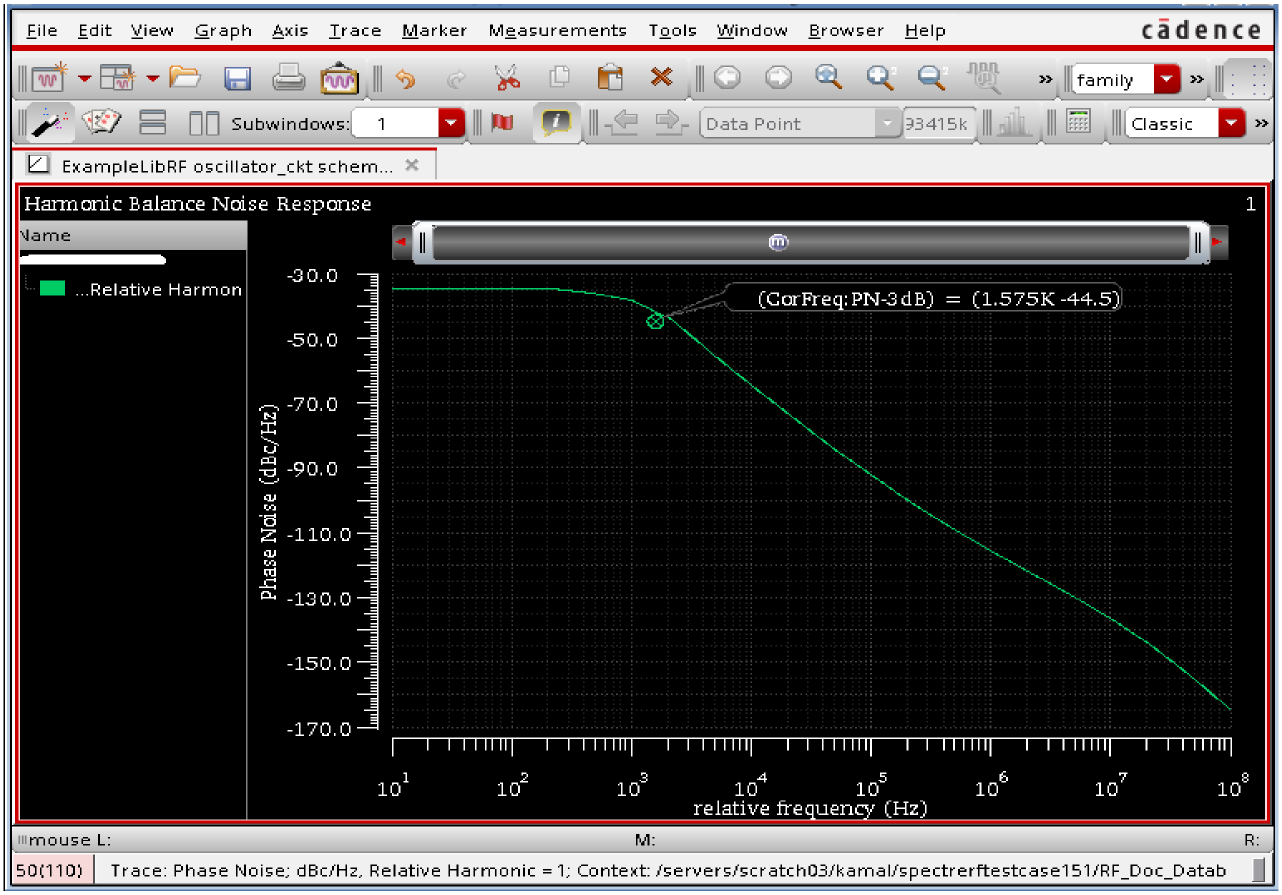
Figure 3-96 *hbnoise* Analysis Direct Plot Form Setup



5. Click *Plot*.

   The phase noise is plotted, as shown below:

Figure 3-97 Phase Noise Plot with CorFreq Label



The label *CorFreq* shows an estimate of the point on the phase noise where the phase noise curve in the physical world levels off instead of continuing its rise.

6. Select *File - Close All Windows* to close the Graph Window in ViVA.

   Next, you will plot the hbnoise Separation.

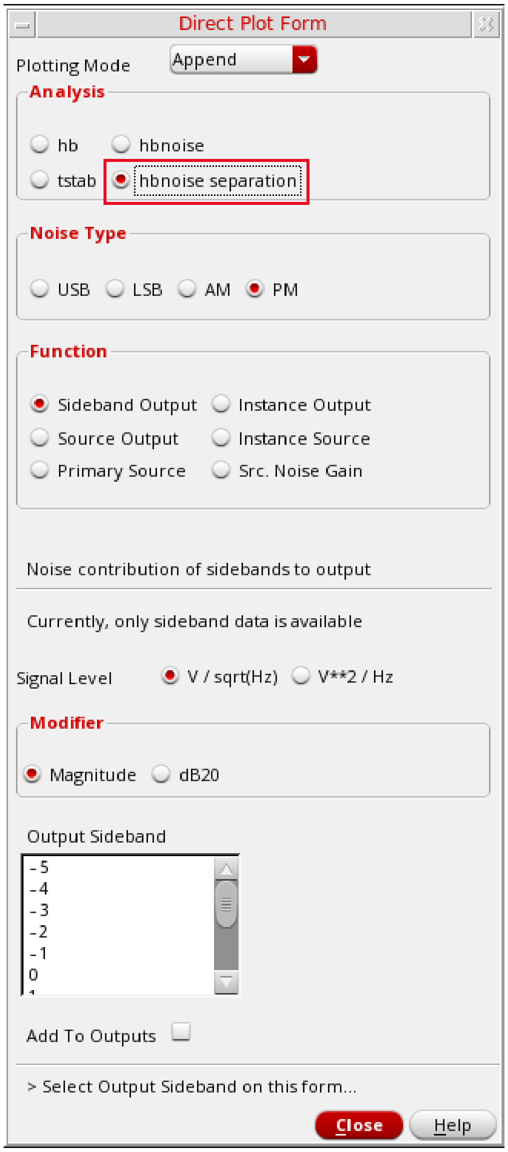
   The key aspect in noise analysis is finding out what is causing the noise problem. Hbnoise provides the total noise from all noise frequencies, and in the noise summary, it provides the noise contributors with all the noise frequencies taken into account. The idea of noise separation is several-fold. First, you can identify the noise frequencies that cause the most noise at the output. Once you know the troublesome frequencies, you can identify the troublesome components. Once you know the troublesome components, you can identify the specific mechanisms within the component that are causing the problem. You can then work upon to find the solution to this problem which may include using different device dimensions or alternate circuit architectures and so on.

To summarize, noise separation is a way of extending standard hbnoise to find out more information about what is causing the problem related to noise. Once the problem is identified, then a solution can be worked upon to fix it.

When *Noise type* is set to *timeaverage* and *Noise Separation* is set to *yes* in the *Choosing Analyses* form while setting *hbnoise*, the *hbnoise separation* feature is included during the simulation and the corresponding results are saved.

7. In the *Direct Plot Form*, select *hbnoise separation* in the *Analysis* section. The *Direct Plot Form* changes, as shown below.

Figure 3-98 Direct Plot Form for hbnoise Separation



8. Leave *Noise Type* selected as *PM*.
9. In the *Function* section, select *Sideband Output*. *Sideband Output* plots the noise contribution of selected sidebands to the output.
10. In the *Signal Level* section, select *V / sqrt(Hz)*.
11. Choose *dB20* as the *Modifier*.
12. Select the *-1*, *0*, and *1 Output Sidebands*.

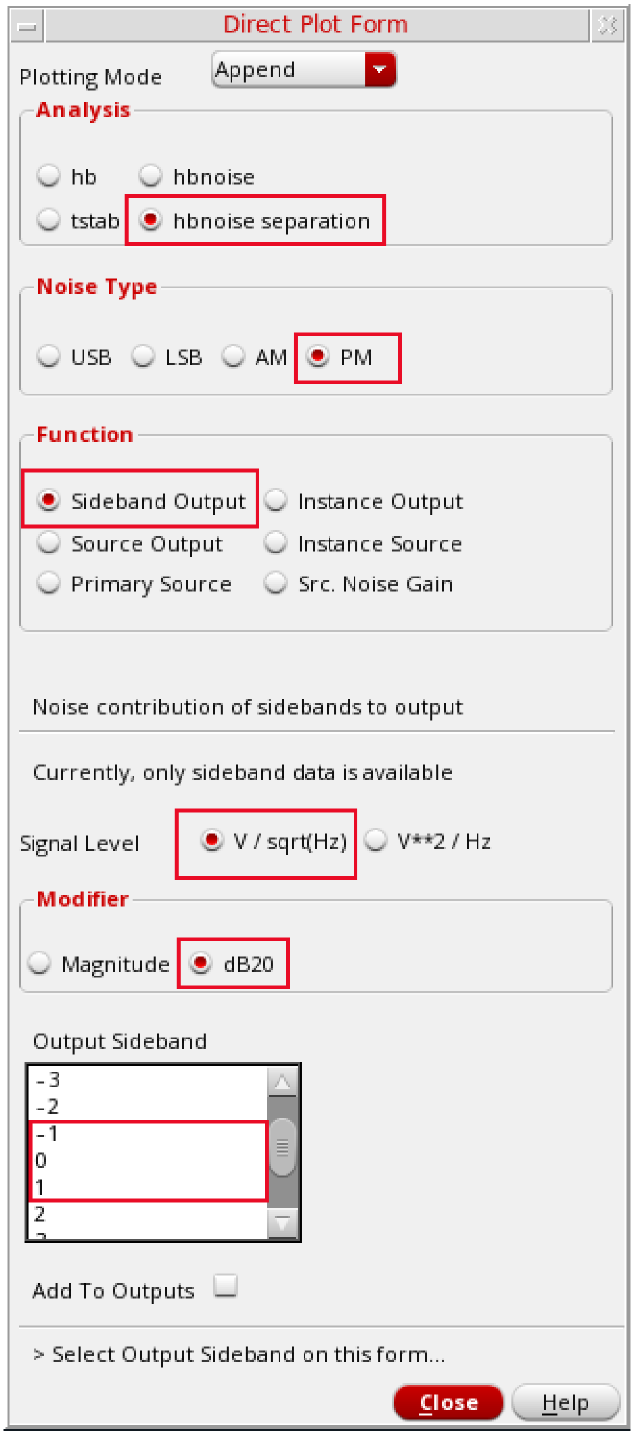
    This will plot the noise contribution of the -1, 0, and 1 sidebands to the output. This allows the identification of which noise frequencies are causing the largest noise contribution at he output of the circuit.

Tip

You can click and hold the mouse over the -1 output sideband and slide the cursor to include the 1 output sideband.

The *Direct Plot Form* should look like the following:

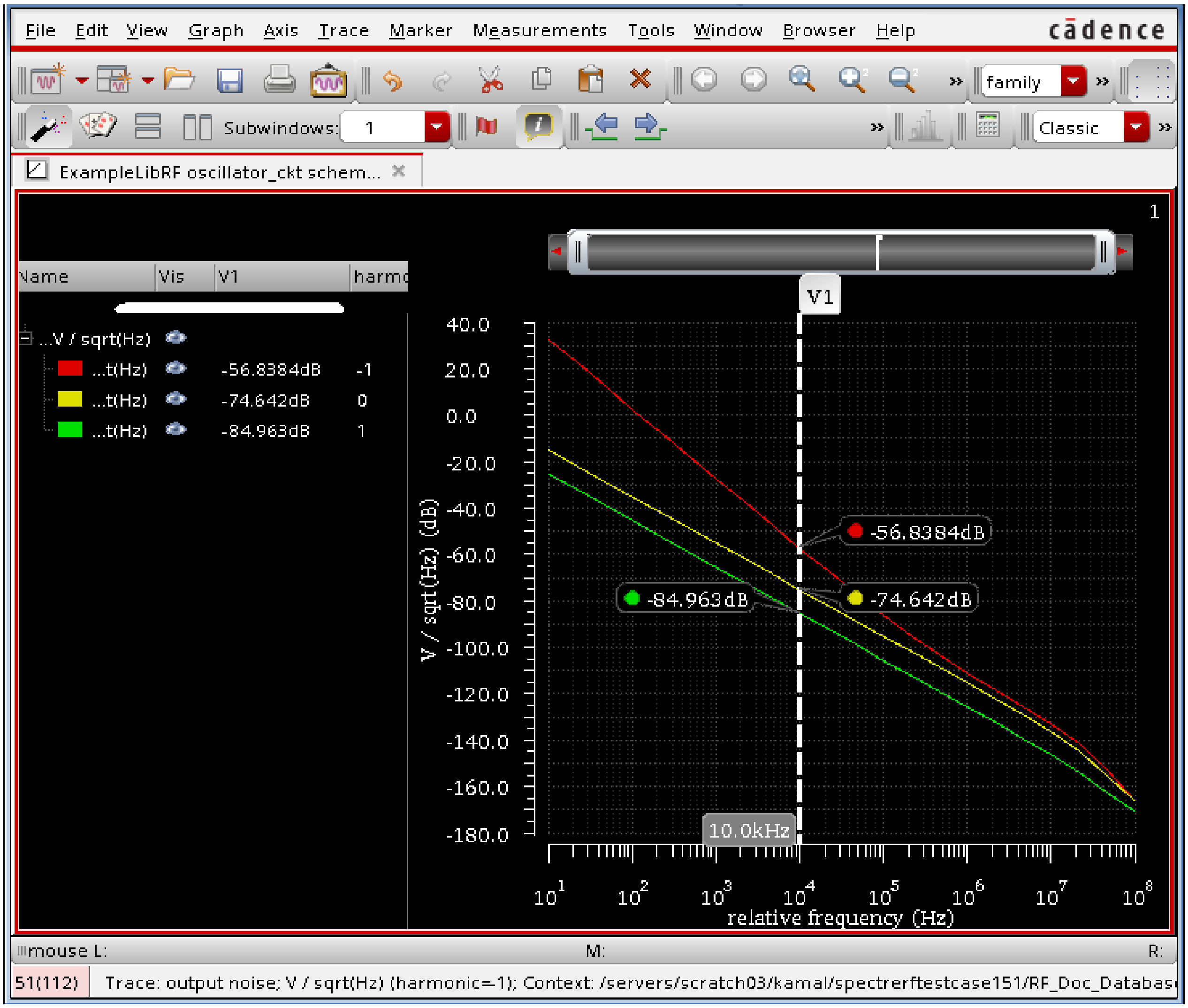
Figure 3-99 Direct Plot Form setup for hbnoise Separation - Sideband Output



13. Click *Plot*.

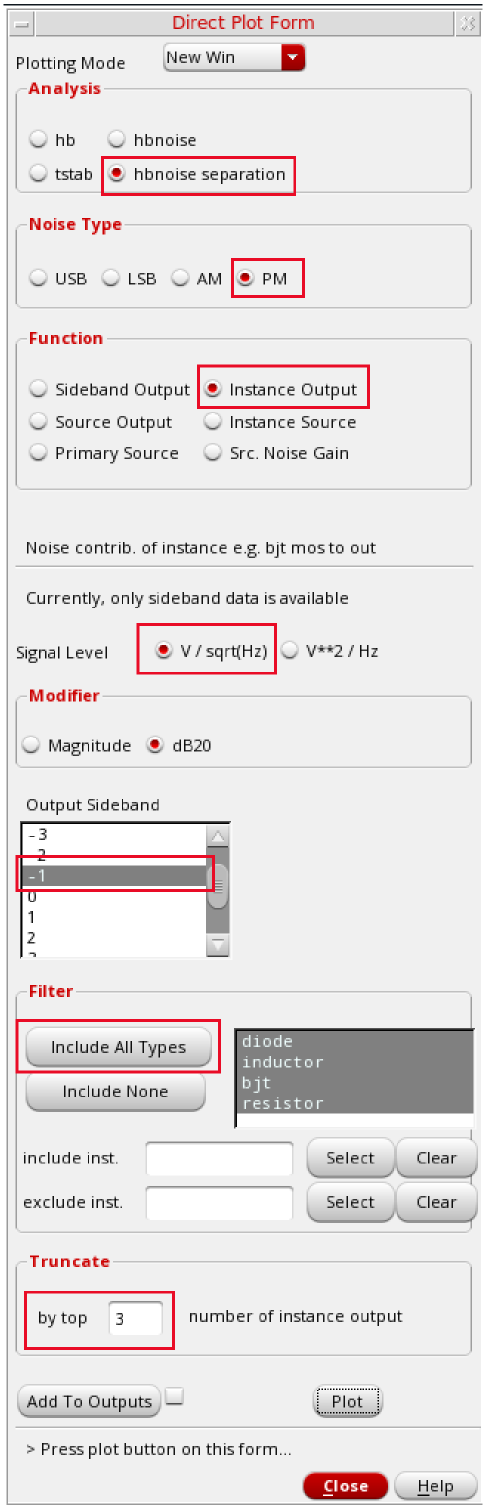
The hbnoise separation plot is shown below. Note that the harmonic numbers plotted are in the left side of the screen.

Figure 3-100 HBnoise Separation, plot of -1, 0, 1 Sideband



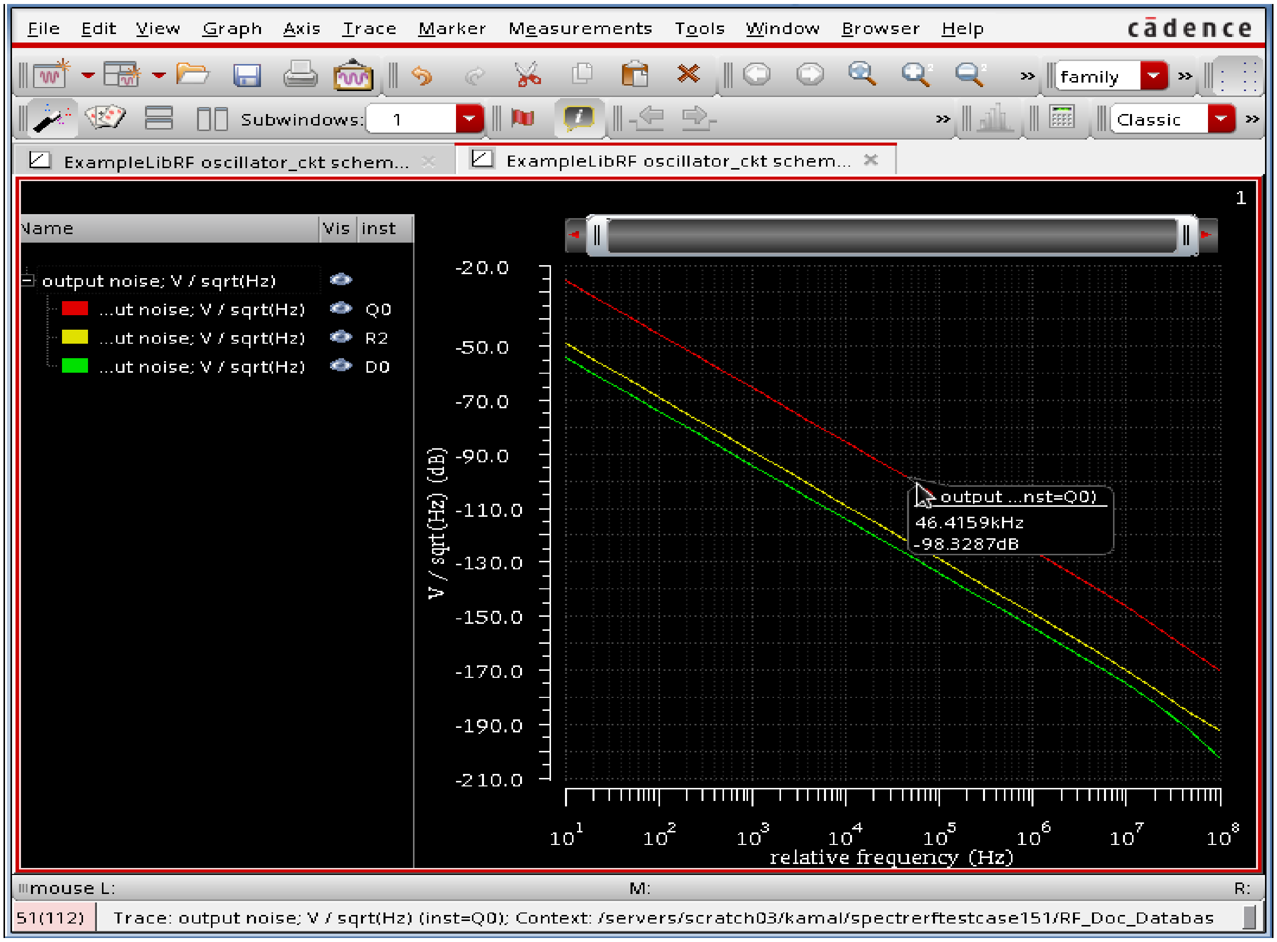
14. Next, plot the Instance output for the largest noise contributor above (It is the -1 sideband in this case). This will show which components contribute the most noise at the output. In the *Direct Plot Form*, first set the *Plotting Mode* to *New Win* then select *Instance Output*. Verify that the Signal Level is set to *V/sqrt(Hz)*. Set the Modifier to *dB20*. Select the *-1* Output sideband. In the *Filter* section, select *Include All Types*. In the *Truncate* section, enter `3` in *by top number of instance output*. The *Direct Plot Form* should look like the one below:

Figure 3-101 Direct Plot Form for Plotting Instance Output for Noise Contribution of instances



15. Click *Plot*. The plot is displayed. There are three plots for the top three noise contributors.

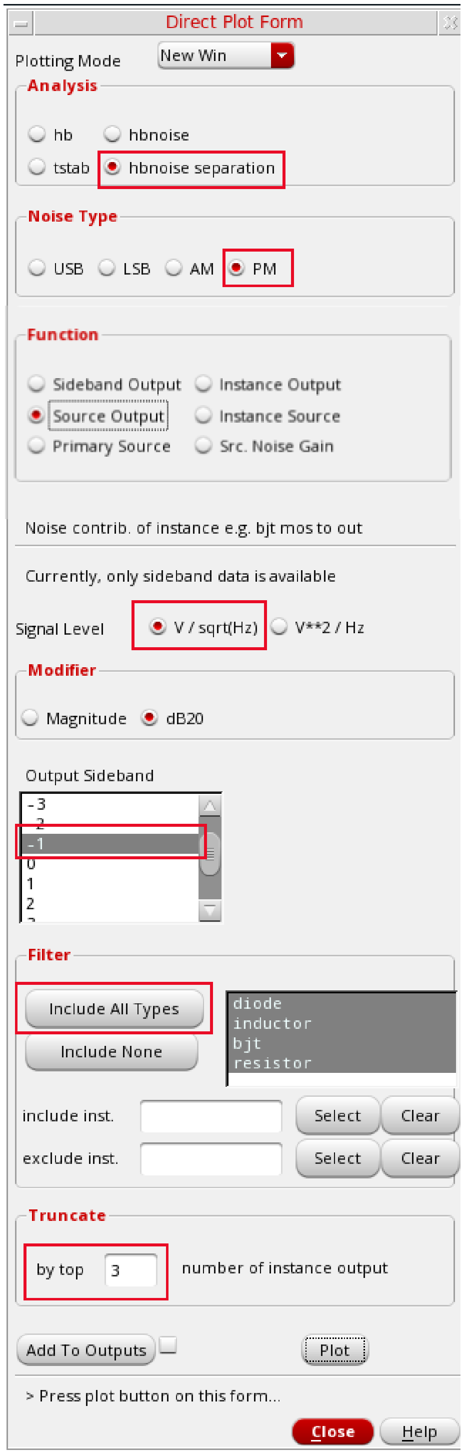
Figure 3-102 Plotting Output Noise Top Instance Contributors



16. If you hover your mouse over a particular trace, it will show which instance contributes the most amount of noise. Above, the red line represents the noise contribution from the instance Q0 (legend shown), the yellow line below represents the noise contribution from instance R2, and the bottom green line represents the noise contribution from instance D0.

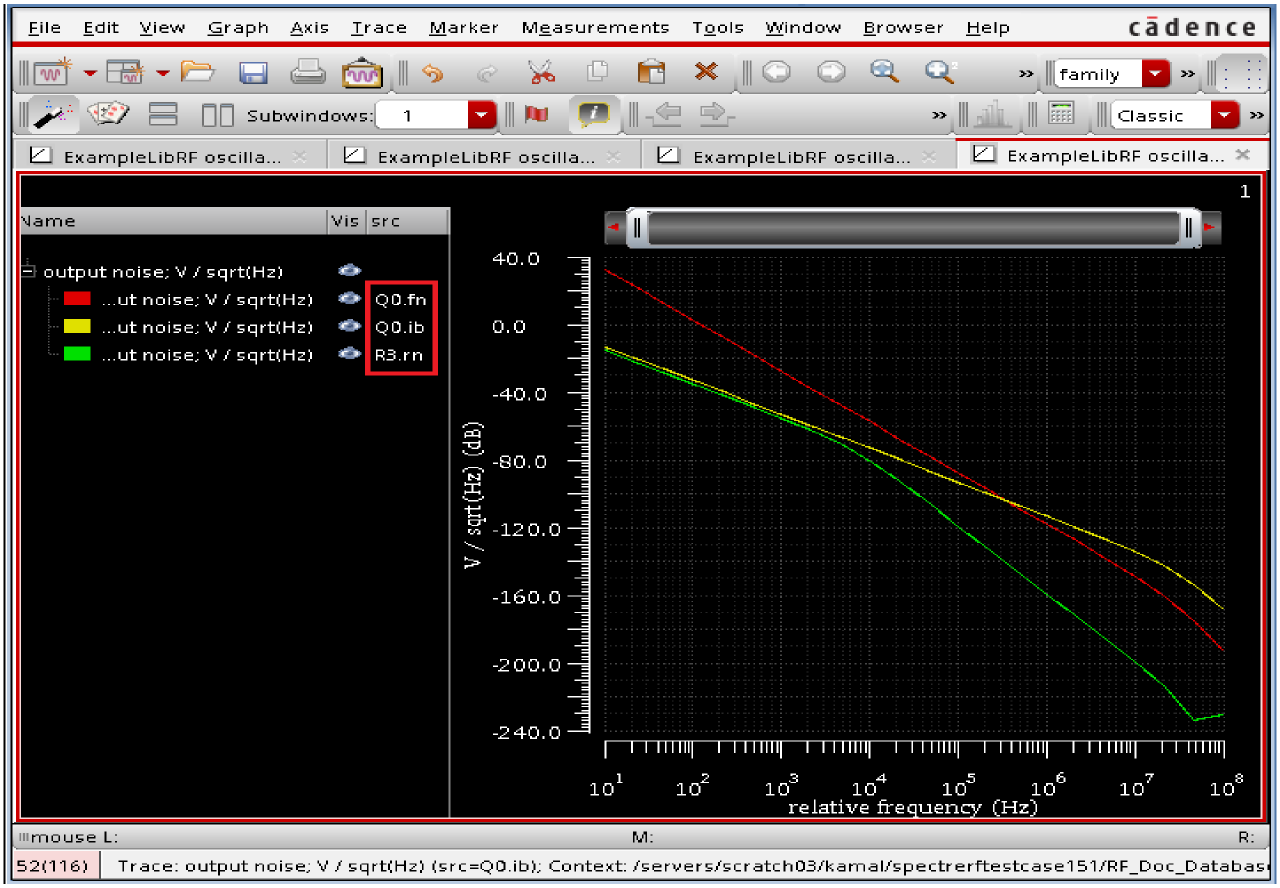
17. Next, plot the *Source Output* so you can see which individual noise sources within the circuit are contributing the most noise. In the *Direct Plot Form*, change the *Function* to *Source Output*. Select *Signal Level* to be *V/sqrt(Hz)*. Set the *Modifier* to *dB20*. Leave the *Output Sideband* at *-1*. In the *Filter* section, select the *Include All Types* option. In the *Truncate* section, type `3` in the *by top* field. The *Direct Plot Form* should look like the one below.

Figure 3-103 Direct Plot Form for Plotting by Top 3 Source Noise Contributors



18. Plot the source output so you can see which individual noise sources within the circuit that are making the most noise.

Figure 3-104 Top 3 Source Noise Contributors



19. Note that the top three Source Output contributors are displayed in the legend on the left side of the graph: Q0:fn, R3:rn and Q0:ib. Here “fn” is the flicker noise, “ib” is the base current shot noise of BJT and “rn” is the resistor thermal noise. See Chapter 3 in this guide for more information on the various noise parameters.

20. In the *Direct Plot Form*, click *Cancel*.

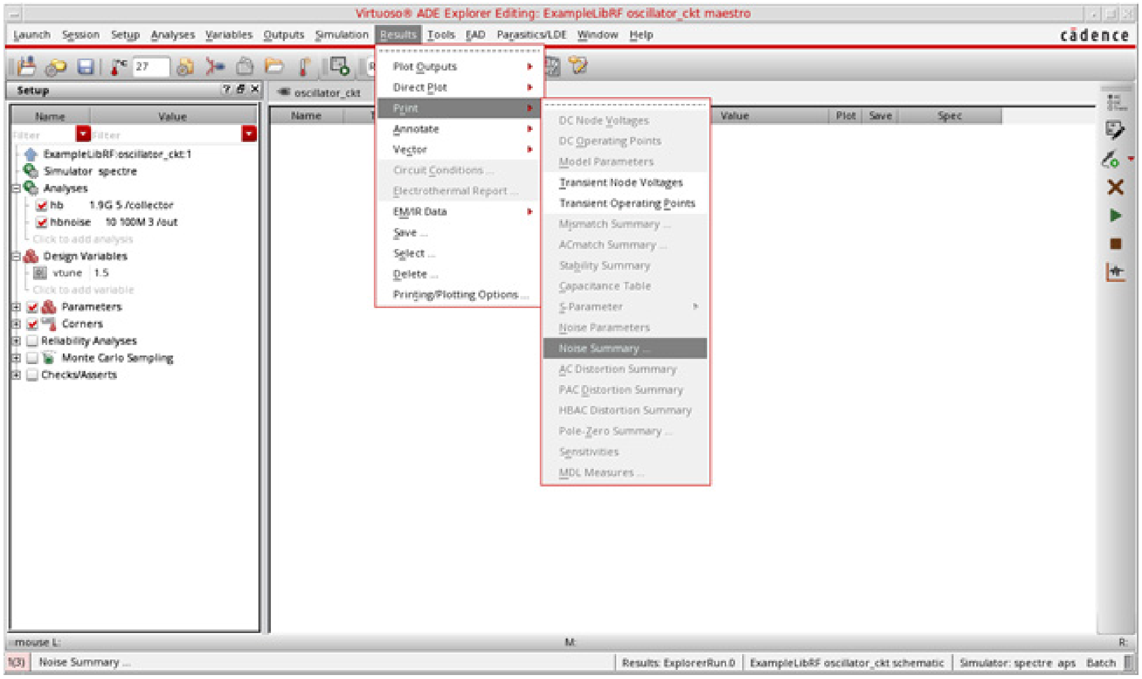
21. In the ViVA window, choose *File - Close All Windows*.

Noise Summary Table

This table provides the list of noise contributors based on the selections made on the Noise Summary Table form.

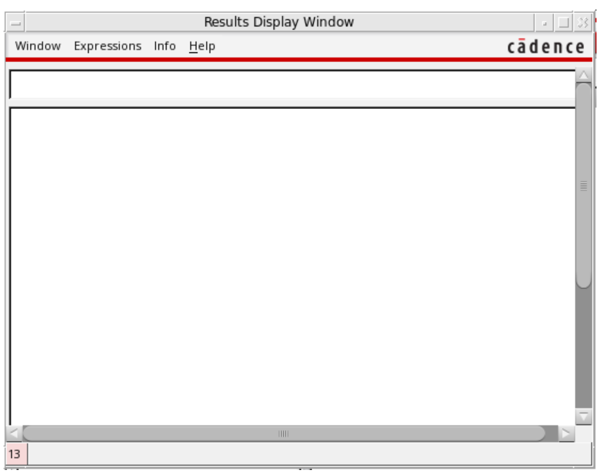
1. In ADE Explorer, select *Results -Print - Noise Summary*.

Figure 3-105 Print Noise Summary in ADE Explorer



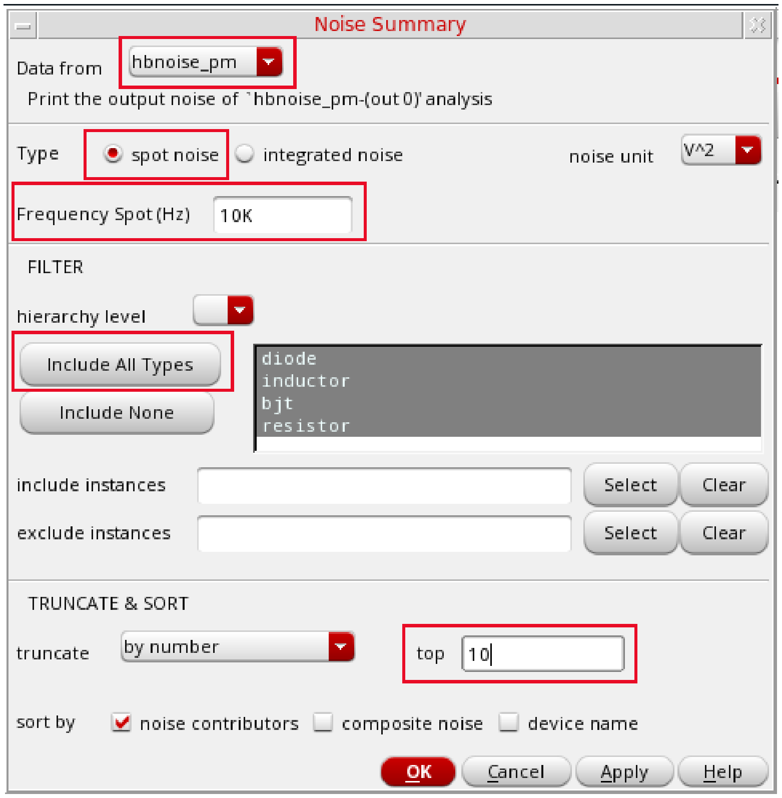
The *Results Display Window* and *Noise Summary* form are displayed. The Results Display Window will currently be blank, as shown below:

Figure 3-106 Results Display Window



The *Noise Summary* form is shown below.

Figure 3-107 Noise Summary Setup Form



2. In the *Noise Summary* window, set *Data from* as *hbnoise_pm*.
3. Set *Type* as *spot noise*.
4. Set *Frequency Spot (Hz)* to `10K`. In spot noise, you will be looking at the noise at one frequency.
5. Click *Include All types* in the *Filter* section. This will include the noise contributions from all devices including resistor, inductor and so on.
6. In the *Truncate and Sort* section, enter `10` in the *top* field. The top 10 noise contributors in the *Results Display* window are displayed.

   Note: You can shorten your noise summary either:

   ❑ *by number* - that is, by specifying the highest contributors to include in the summary,

   or

   ❑ *by relative threshold* - that is, by specifying the percentage of noise a device must contribute to be included in the summary,

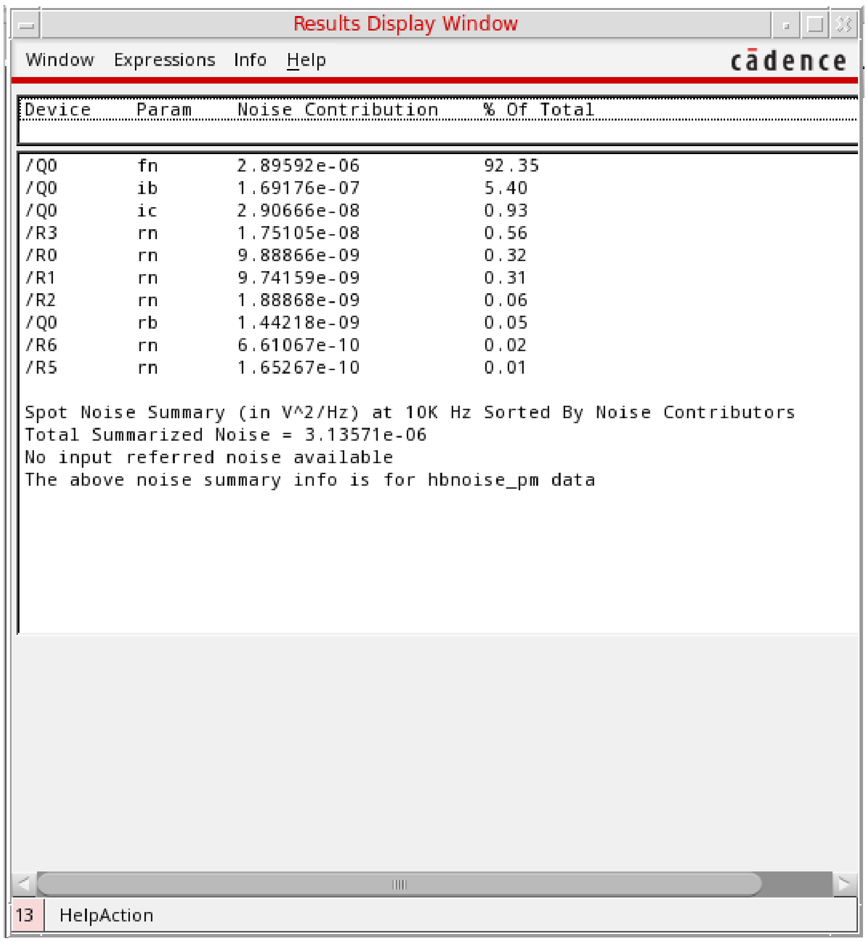
   or

- ❑ *by absolute threshold* - that is by specifying the level of noise a device must contribute to be included in the summary.

7. Click *OK*.

The noise summary results are displayed in the *Results Display Window,* as shown below:

Figure 3-108 Noise Summary Results Display Window



The *Results Display Window* lists the individual contributors, the specific noise mechanism within the semiconductors causing the noise, and the noise contribution. The Total Summarized Noise is shown at the bottom of the Results Display window. Since there is no Input Source present, so there is no input referred noise available. This is usually the case for oscillators.

Note that the output noise include the noise from all the noise contributors, and not just the contributors in the form.

8. In the *Results Display Window,* select *Window - Close* to close the window.

9. Close the Analog Design Environment window by selecting *Session* - *Quit*.

10. In the Schematic window, choose *File* - *Close*.

In this section, one of the key measurements of Oscillator design, that is, phase noise measurement was explained along with obtaining the noise summary.

Next, you will perform Swept HB measurements, that is Tuning Voltage vs. Oscillation Frequency and Phase Noise vs. Tuning Voltage Measurements.

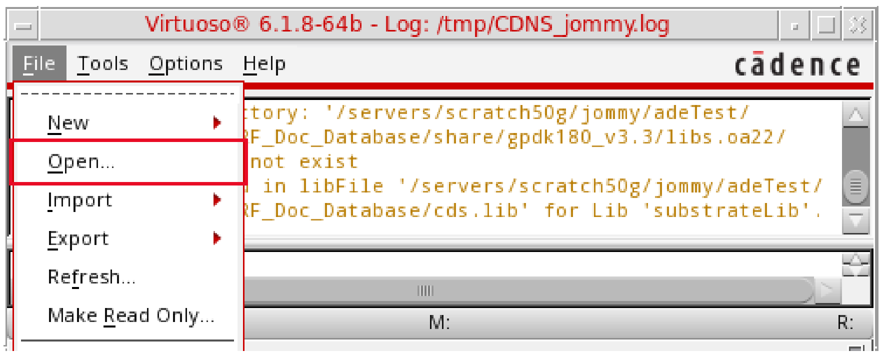
Oscillator Swept Tuning Range and Phase Noise Measurement

In this section, you will sweep the oscillator tuning range while running multiple HB/HBnoise analyses. This will give the information about phase noise vs. tuning range.

Opening the Oscillator Circuit in the Schematic Window

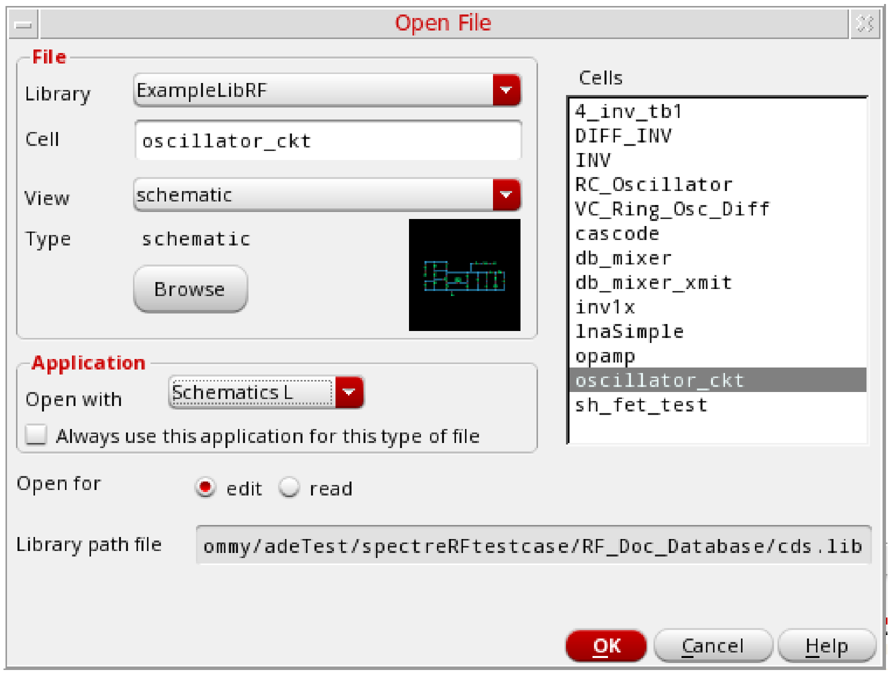
1. In CIW, select *File - Open*.

Figure 3-109 Virtuoso CIW Window - Opening Cellview



The Open File form is displayed.

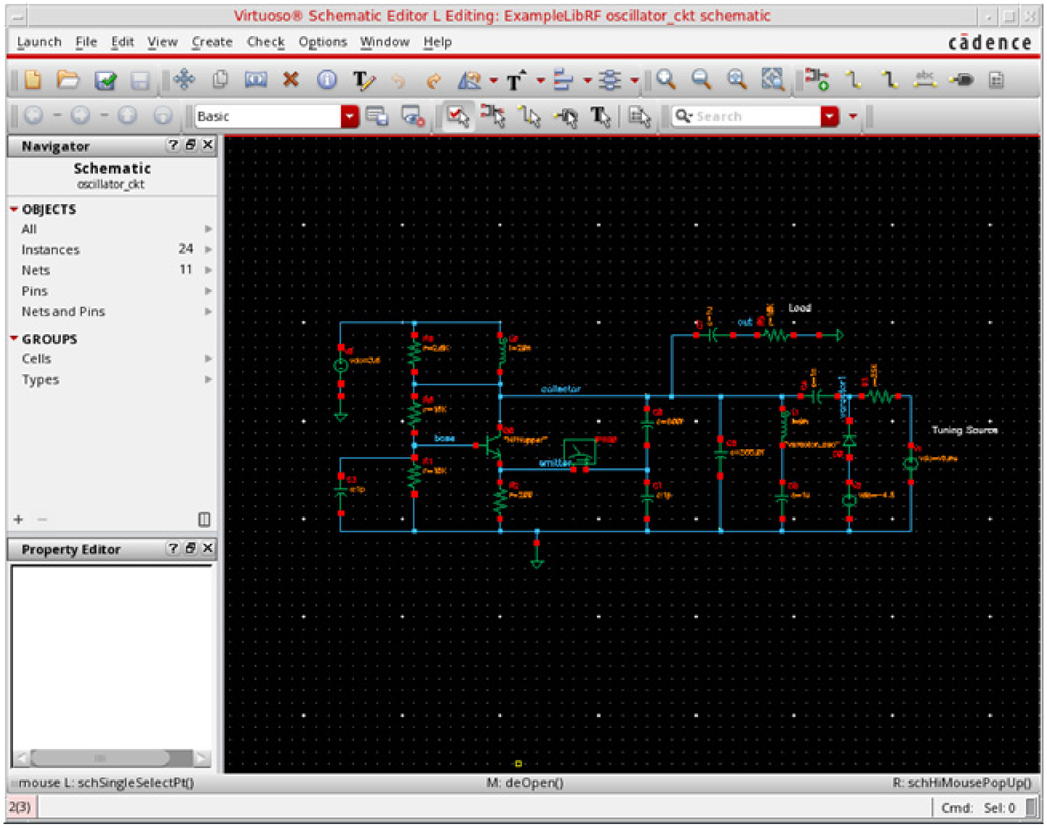
Figure 3-110 Open File Form to open the oscillator_ckt cell's Schematic View.



2. Select *ExampleLibRF* from the *Library* drop-down list.
3. In the *Cells* field, type `oscillator_ckt`.
4. Choose *schematic* from the *View* drop-down list.
5. In the *Application* section, select *Schematic L from the Open With* drop-down list.
6. Leave *Open For* to *Edit* (which is set by default)
7. Click *OK*.

   This will open the oscillator_ckt schematic in Virtuoso Schematic Editor L window.

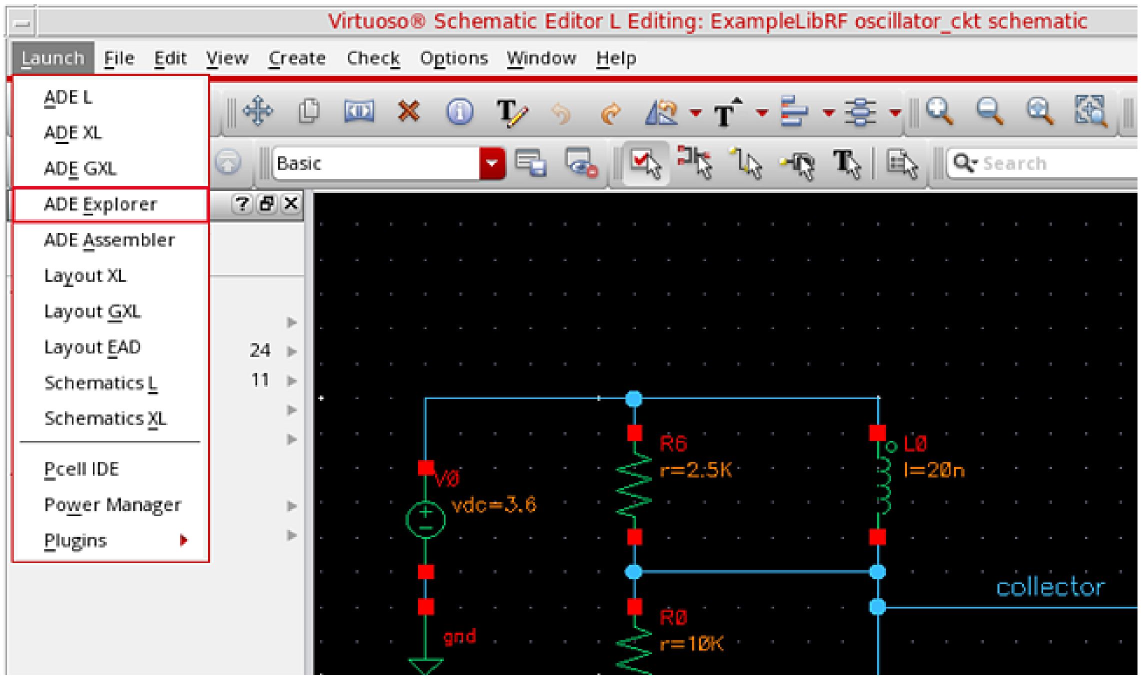
Figure 3-111 Oscillator Schematic in VSE-L Window



Setting up the HB and HBnoise Analysis

1. In the Schematic Window, select *Launch - ADE Explorer.*

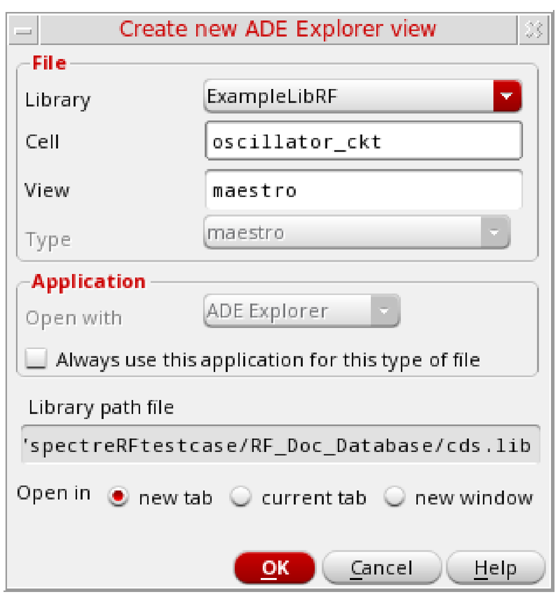
Figure 3-112 Opening ADEL window from VSE window



2. In the *Launch ADE Explorer* dialog, select *Create New View*.

   The *Create new ADE Explorer view* form is displayed.

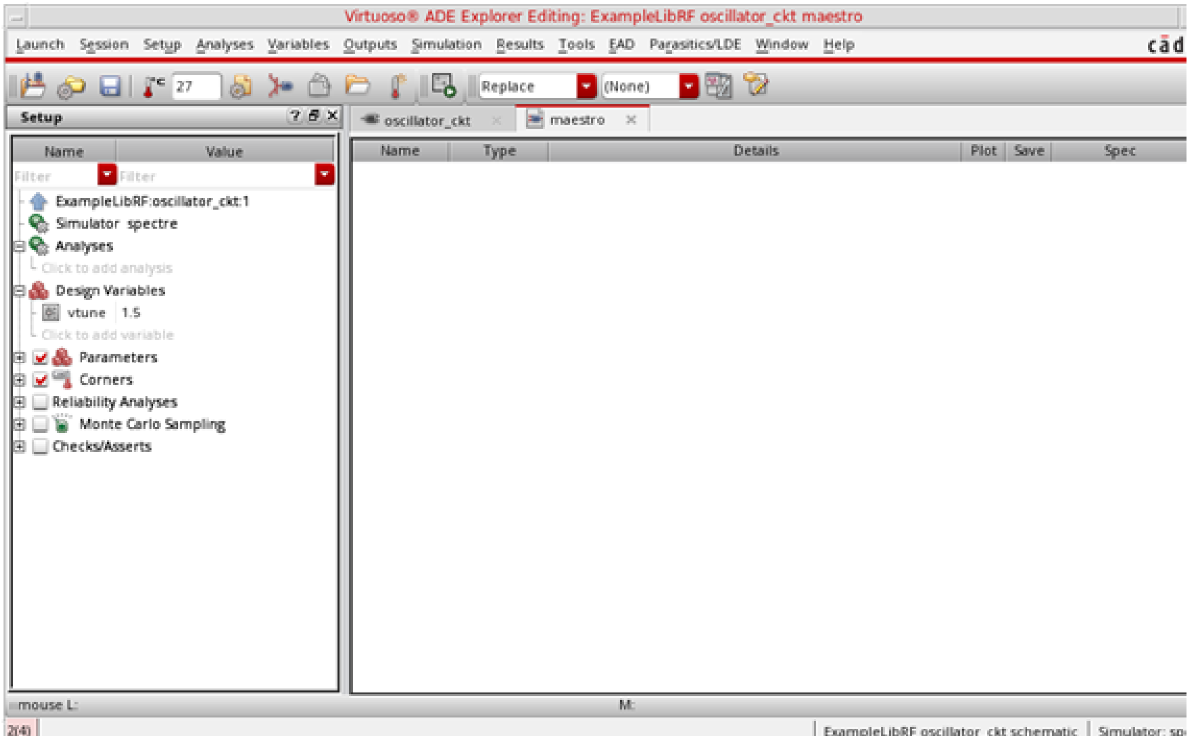
Figure 3-113 Create new ADE Explorer view



3. Leave each option to the default selections and click *OK*.

ADE Explorer is displayed, as shown below.

Figure 3-114 Virtuoso ADE Explorer



4. Select *Setup - Simulator* in the *ADE Explorer*

   The *Choosing Simulator* form is displayed.

Figure 3-115 Choosing Simulator Form

5. Select *spectre* as the *Simulator*.
6. Click *OK* to close the *Choosing Simulator* form.
7. In ADE Explorer, select *Setup - High Performance Simulation*.

   The *High Performance Simulation Options* form is displayed.

Figure 3-116 High Performance Simulation Options Form

8. In the *High Performance Simulation Options* window, select *APS*.

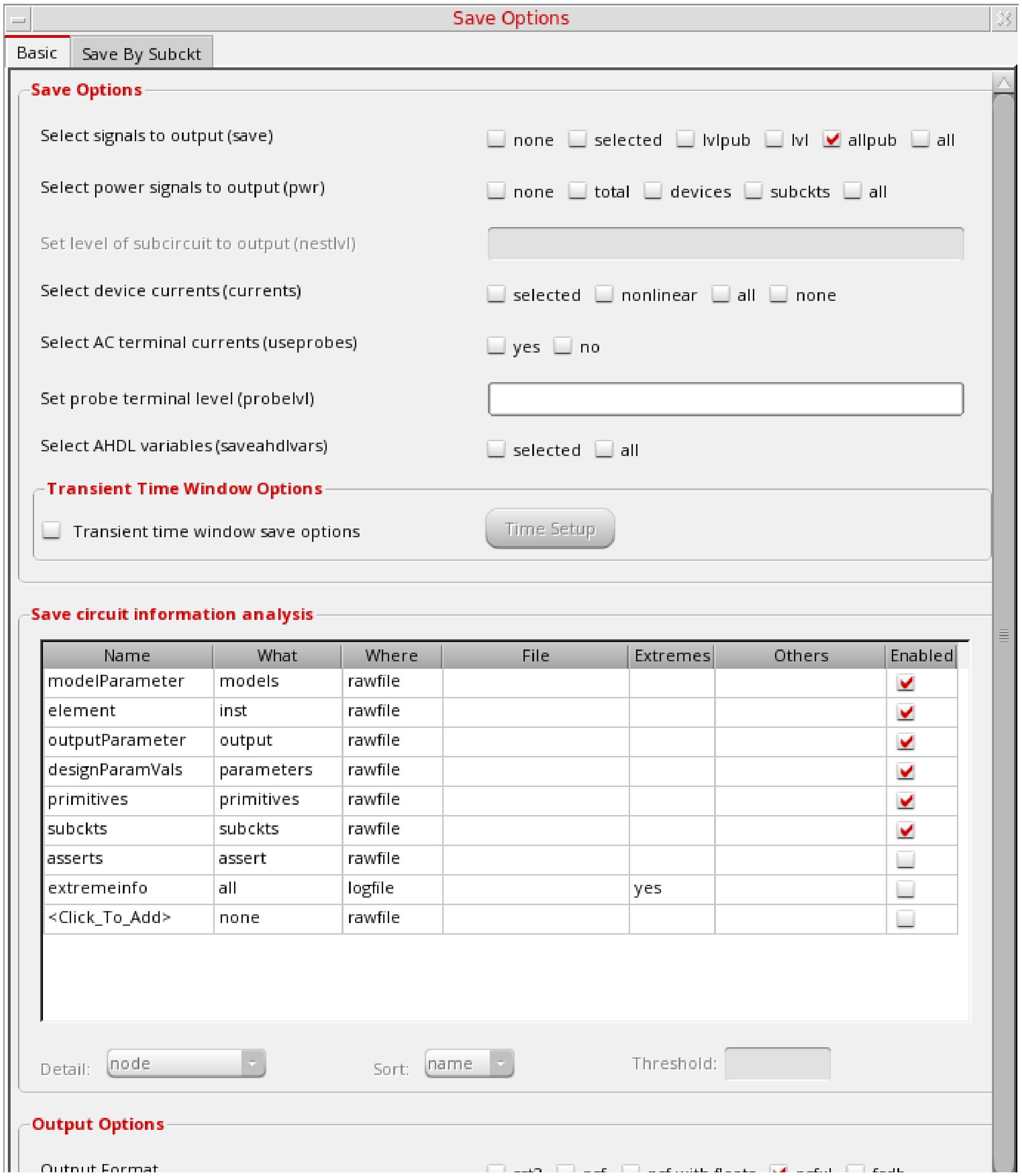
   Note that *Auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 64) and then multi-thread on all the available cores. Usually, it is better to specify the number of threads yourself. Small circuits should use a small number of threads, and larger circuits can use more threads. The overhead of managing 16 threads on a smaller circuit may actually slow the simulation down, compared to two or four threads. Use the number of threads that maximizes the performance.

9. Click *OK* to close the *High Performance Simulation Options* form.

10. In ADE Explorer, select *Outputs - Save All*.

The *Save Options* form is displayed.

Figure 3-117 Save Options Form



11. In the *Select signals to output (save)* section, make sure that *allpub* is selected.

    This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

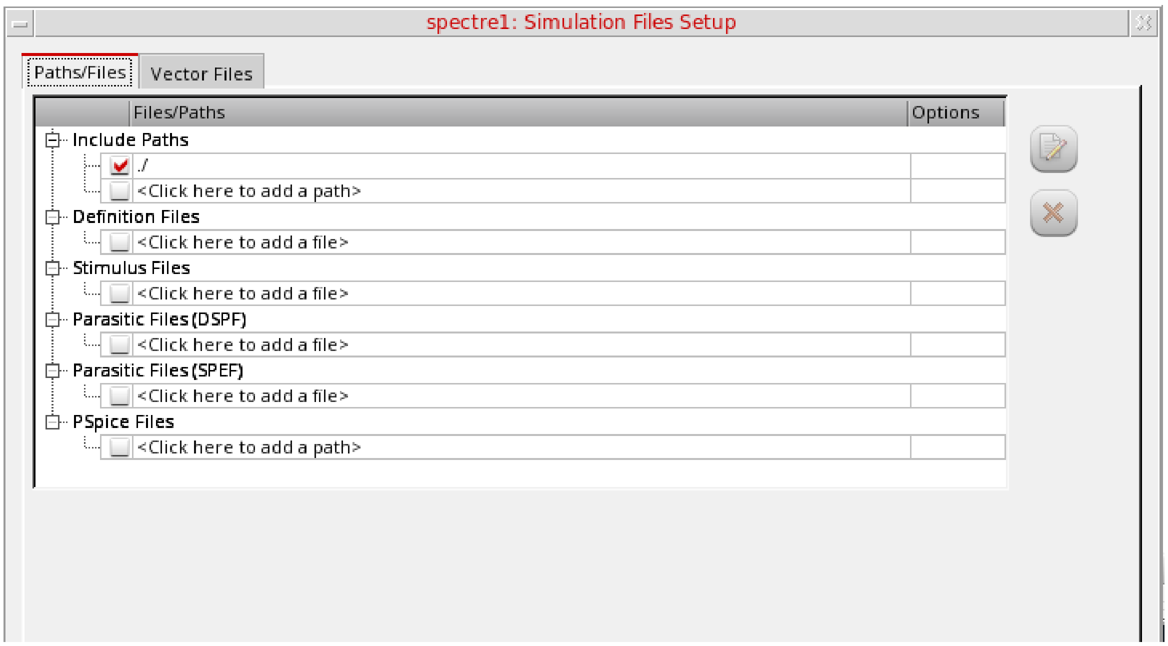
To save the currents, use the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or all if you want to save all the currents in the circuit. When you save currents, more disk space is required for the results file.

12. Click *OK*.

13. In ADE Explorer, select *Setup - Simulation Files.*

The *Simulation Files Setup* form is displayed, as shown below.

Figure 3-118 ***Simulation Files Setup*** **Form**

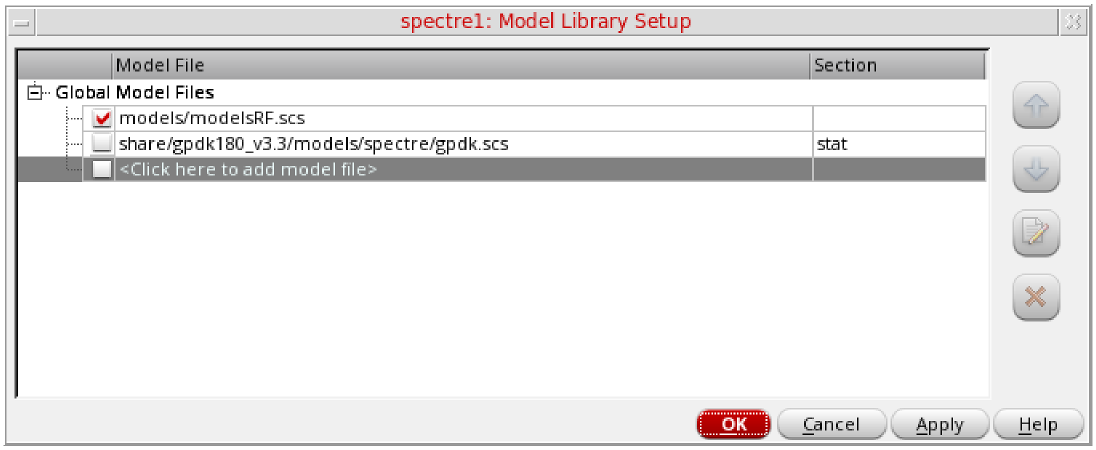


14. In the *Simulation Files Setup* form, enter `./` by clicking in the *Include Paths* section.

15. Click *OK* to close the *Simulation Files Setup* form.

16. Select *Setup – Model Libraries.*The *Model Library Setup* form is displayed.

Figure 3-119 *Model Library Setup* Form



17. In the *Model File* field, type the path to the model file including the file name, as follows:

`models/modelsRF.scs`

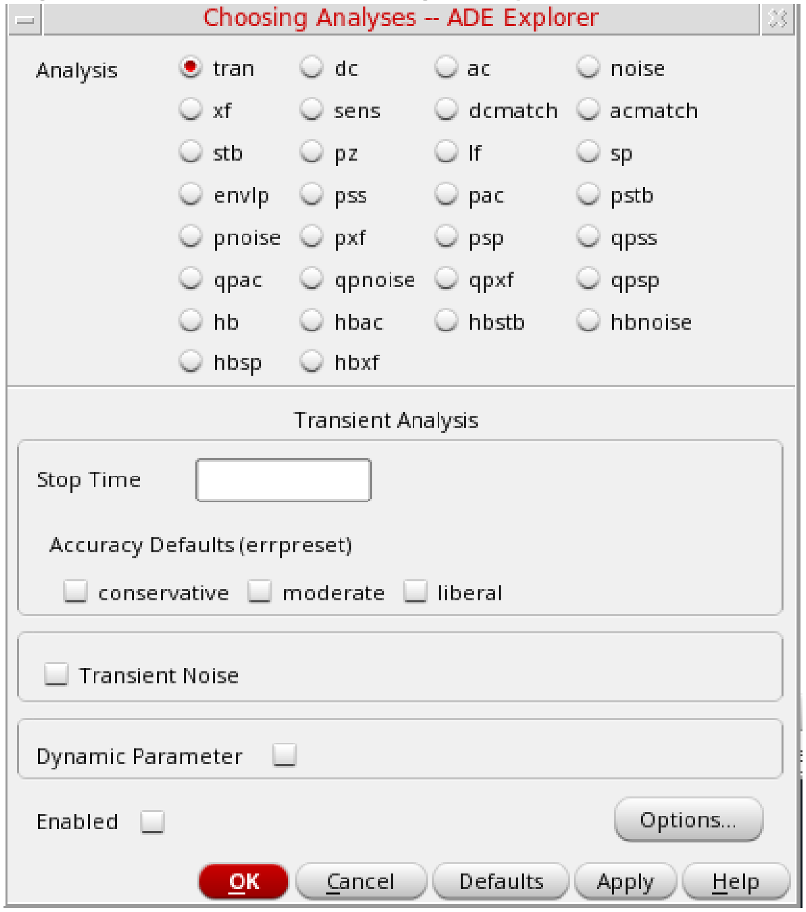
You can also browse to the *modelsRF.scs* file.

18. Click *OK* to close the *Model Library Setup* form.

19. Select *Analyses - Choose*.

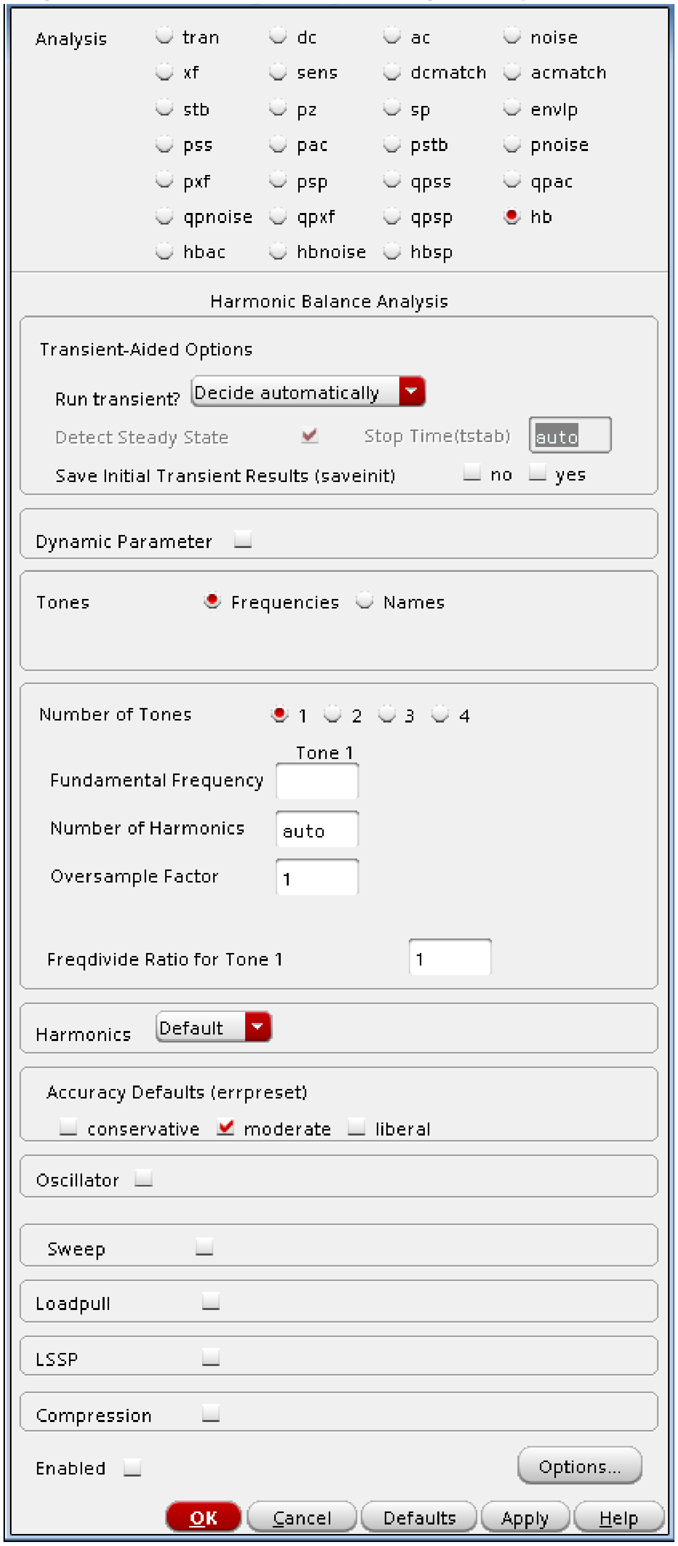
The *Choosing Analyses* form is displayed.

Figure 3-120 The Choosing Analyses Form



20. Select *hb* in the *Analysis* section. The form expands.

Figure 3-121 The Choosing Analyses Form - Setting *hb* analysis



21. In the *Transient-Aided Options,* leave *Run Transient?* as *Decide automatically* which is set by default.

The other choices for this option are *Yes* and *No*. If *Yes* is selected, then the *Detect Steady State* option becomes active and the *Stop Time(tstab)* field is also activated. This means that you can decide whether you would like the Steady State to be detected automatically during the transient run or not and also specify a *Stop time(tstab)* for that transient run. When you select the checkbox for *Detect Steady State* option, this will run *tran* until steady-state is detected and then switches to *hb*.

22. Select *yes* for the *Save Initial Transient Results* (*saveinit*) option. This will help in visualizing the buildup of the oscillation waveform.

23. Select *Frequencies* in *Tones* section (This is selected by default). The other option is *Names*. The *Names* option when selected, is similar to the *pss* Harmonic Balance analysis setup.

24. Enter `1.7G` in *Fundamental Frequency* field.

25. Set the *Number of Harmonics* option to `20`.

When *Number of Harmonics* is set to `auto`, the simulator calculates tone-1 harmonics automatically. The calculation is based on Fourier analysis of transient steady-state waveforms.

26. Leave the *Oversample Factor* option to `1` by default. Since the oscillator has sinusoidal waveforms, an oversample of 1 is appropriate.

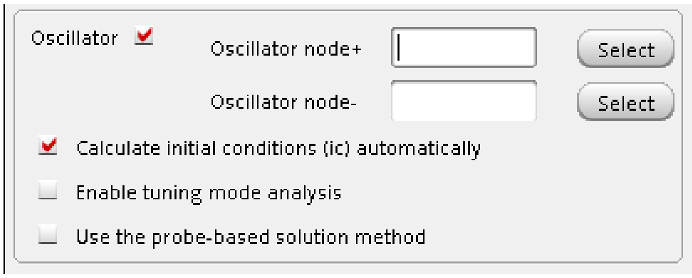
27. Set the *Freqdivide Ratio for the Tone 1* option to `1` as there is no frequency divider in the circuit. If there is a frequency divider in the circuit then you need to set the *Freqdivide Ratio for Tone 1* to the divide ratio of the divider. For example, if the divider is divide-by-two then the divide ratio is 2. Therefore, you will set *Freqdivide Ratio for Tone 1* to 2.

28. *Leave the Harmonics* option as is which is set to *Default*.

29. In the *Accuracy Defaults* (*errpreset*) section, select *conservative*. *conservative* is typically used because very small amplitude phase noise measurements are normally desired. Conservative is recommended for all the oscillators.

30. Select *Oscillator*. This is required for simulating an autonomous circuit.

Figure 3-122 The *Choosing Analyses* Form - Oscillator Section



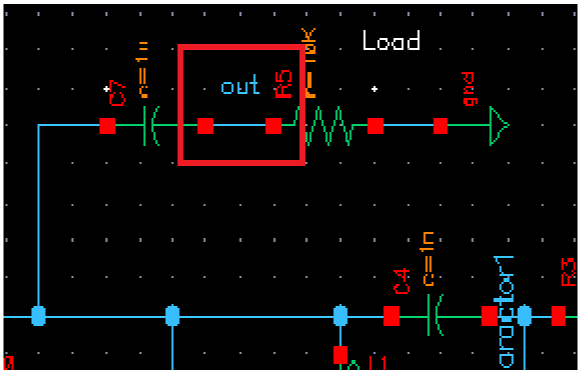
Selecting the Oscillator button changes the Tone1 name in the Tones section to osc!.

31. In the Oscillator section, in *Oscillator node+* field, click *Select* just to the right of this field. In the schematic, select the *out* node. Instead of selecting the node from the schematic you can also type `/out` in the *Oscillator node* field. This oscillator node will be used by Spectre for the period calculation of the oscillations. It just needs to be a node that has the oscillator signal on it.

32. Leave the *Oscillator node-* blank.

Note: If you have a single-ended oscillator, only specify one node. If the second node, that is, *Oscillator node-* is left blank, it will be connected to the global ground node automatically. However, if you have differential oscillator, you need to specify both the nodes.

Figure 3-123 Selecting *out* net from the schematic



33. If you have an LC oscillator, leave the *Calculate initial conditions (ic) automatically* checkbox selected (this is the default).

Note that *Calculate initial conditions (ic) automatically* is used to start the oscillator. Other methods to start the oscillator include putting a single current pulse into the resonator, setting initial conditions(ic), or ramping up the power at time = zero plus.

34. Note that by default *Use the probe-based solution method* (oscmethod) option is deselected. Spectre will use the *onetier* method. In *onetier* method, the frequency and voltage spectrum are solved simultaneously in one single set of nonlinear equations.

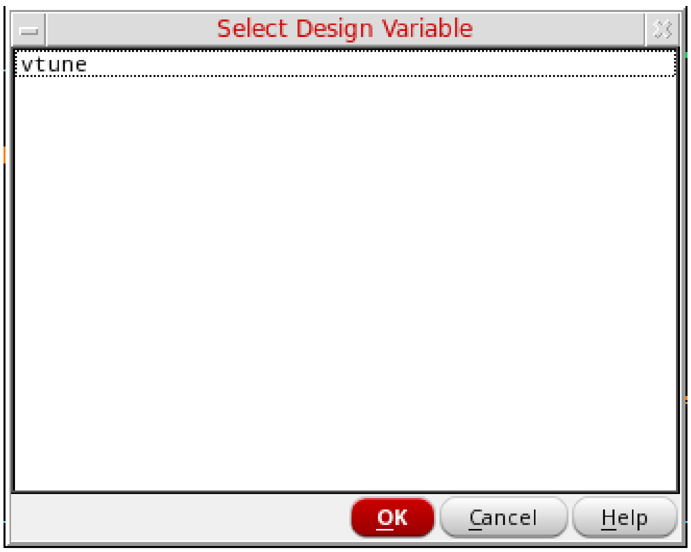
Note: When *Use the probe-based solution method* option is selected/enabled, it iterates for the frequency solution in the outer loop and the amplitude and phase solution in the inner loop. The probe-based method has better convergence but is computation intensive.

Refer to *Spectre Circuit Simulator RF Analysis Theory* for more details.

35. Click *Sweep*. This will enable the sweeping of tuning voltage (in this case it is named as vtune) which is explained below.

36. Click *Select Design Variable*. The *Select Design Variable* window is displayed, as shown below.

Figure 3-124 Choosing vtune in Select Design Variable Form during HB Analysis setup



37. In the Select Design Variable window, select *vtune*.

38. Click *OK* to close the *Select Design Variable* form.

39. In the *Sweep Range* section, type `0.2` in the *Start* field.

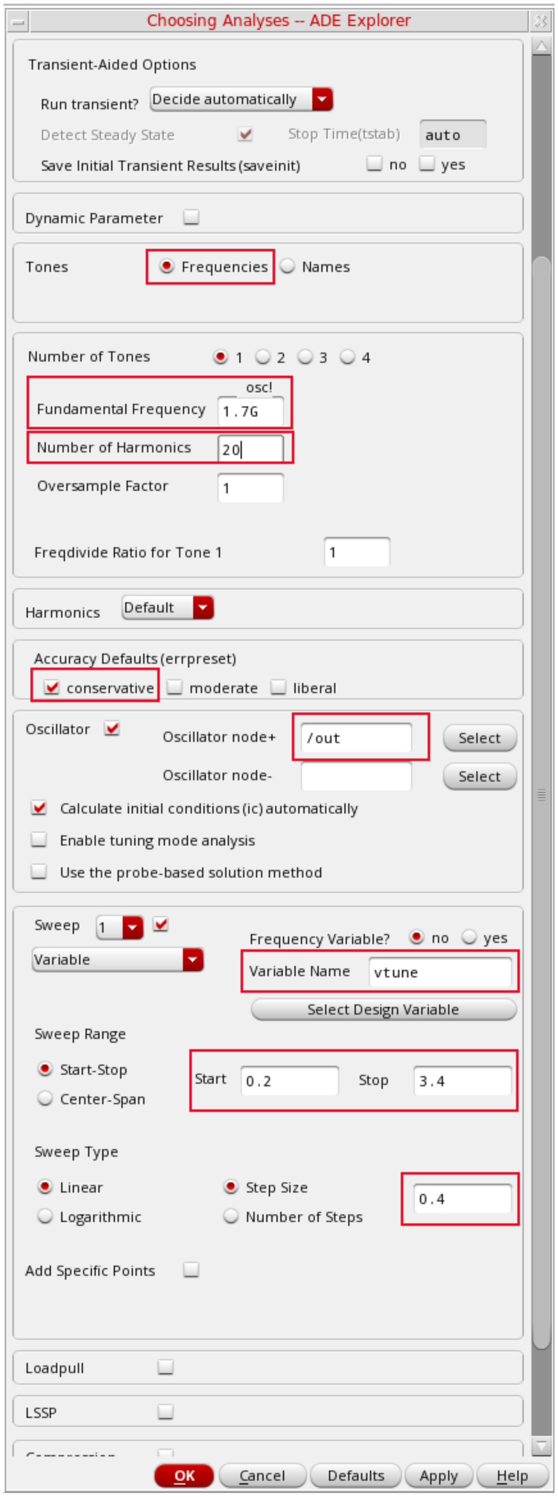
40. Type `3.4` in the *Stop* field.

41. By default, the *Sweep Type* is set to *Linear*.

42. Type `0.4` in the *Step Size* field.

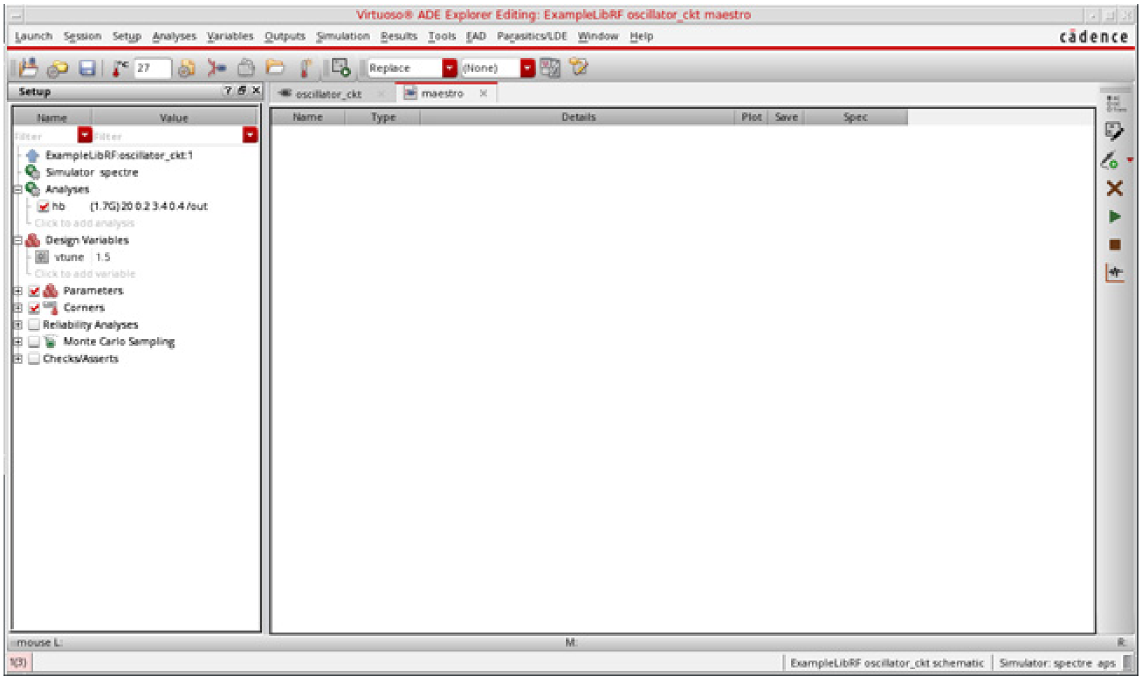
The *Choosing Analyses* form should like the figure below:

Figure 3-125 Choosing Analysis Form - swept *hb* Analysis Setup



Click *Apply* at the bottom of the *Choosing Analyses* form. This will add *hb* analysis with sweep setup in the *Analyses* section of ADE window, as shown below.

Figure 3-126 ADE Explorer Simulation Window - *hb* analysis setup

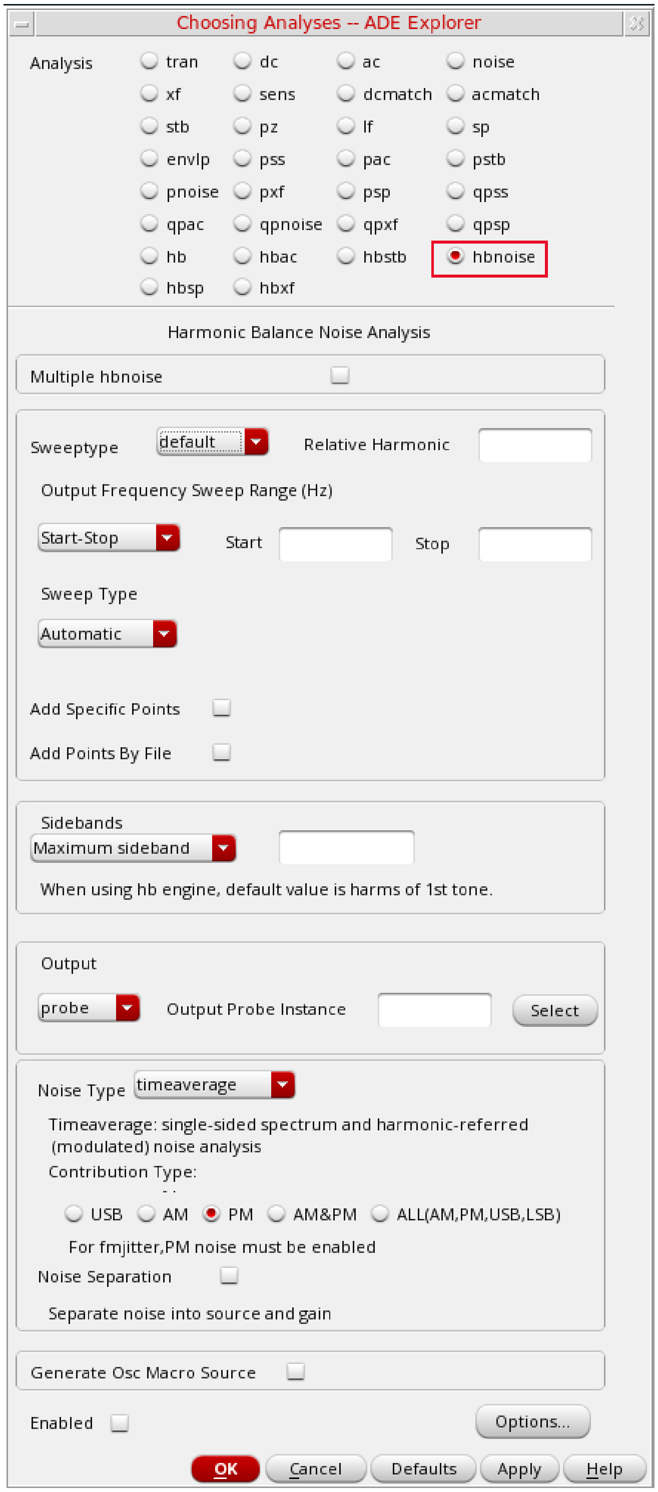


Setting up the HBnoise analysis

This analysis is set to do the phase noise measurement. *hbnoise* analysis is a small signal analyses run after *hb* analysis.

1. In the *Choosing Analyses* form, select *hbnoise*. The form expands.

Figure 3-127 The Choosing Analyses Form - *hbnoise* Analysis Setup



2. Set the *Sweep Type* to *relative*.

For oscillators, the *hbnoise/pnoise* frequency range is *relative*. Specify the harmonic number as appropriate for the system you are simulating. If you are simulating an oscillator by itself, then the harmonic number is likely to be 1. If you have an oscillator and a diode frequency doubler, then the harmonic number is likely to be 2. If you have an oscillator with a frequency divider, in the hb/pss form, you should specify the approximate frequency of oscillation for the frequency-divided signal. In the *hbnoise/pnoise* form, if the noise is desired on the frequency divided output, then the relative harmonic is 1. If the noise is desired at the output of the oscillator, the relative harmonic number is the divide ratio. The meaning of *relative* is to take the frequency of the harmonic number specified and add to it the frequencies specified in the *Choosing Analyses* form. If the oscillator had a 1GHz output, and the pnoise had 1M relative to the first harmonic specified, the actual output frequency is 1G + 1M, or 1001M.

a. Type `1` in the *Relative Harmonic* field as you are simulating an oscillator by itself.

b. Select *Single Point* for *Output Frequency Sweep Range.*

c. Type `1M` in the *Freq* Field. In general, the hbnoise frequency would be set as appropriate for your application.

3. Leave the *Maximum Sideband* field blank.

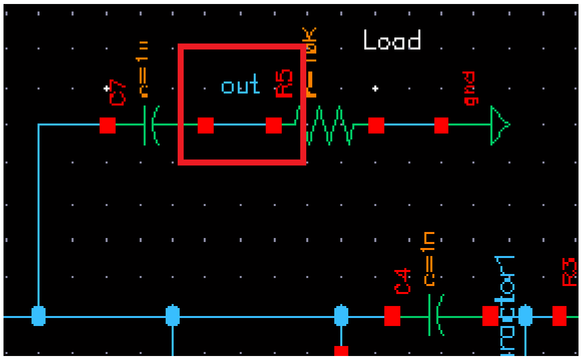
   In general, *Maximum sideband* needs to be set high enough to include all the frequencies that could mix down to the oscillator output frequency. By default, it should match the number of harmonics set in HB analysis.

4. Set the *Output* to *voltage*.

   a. Type /out in the *Positive Output Node* field. You can also select *out* net from schematic by clicking the *Select* button on the right of the *Positive Output Node* field and then selecting the net just below the *out* label in schematic.

   b. Leave the *Negative Output Node* field blank. If the second node, that is, the *Negative Output Node* is left blank, it will be connected to the global ground node automatically. However, if you have differential oscillator, you need to specify both the nodes.

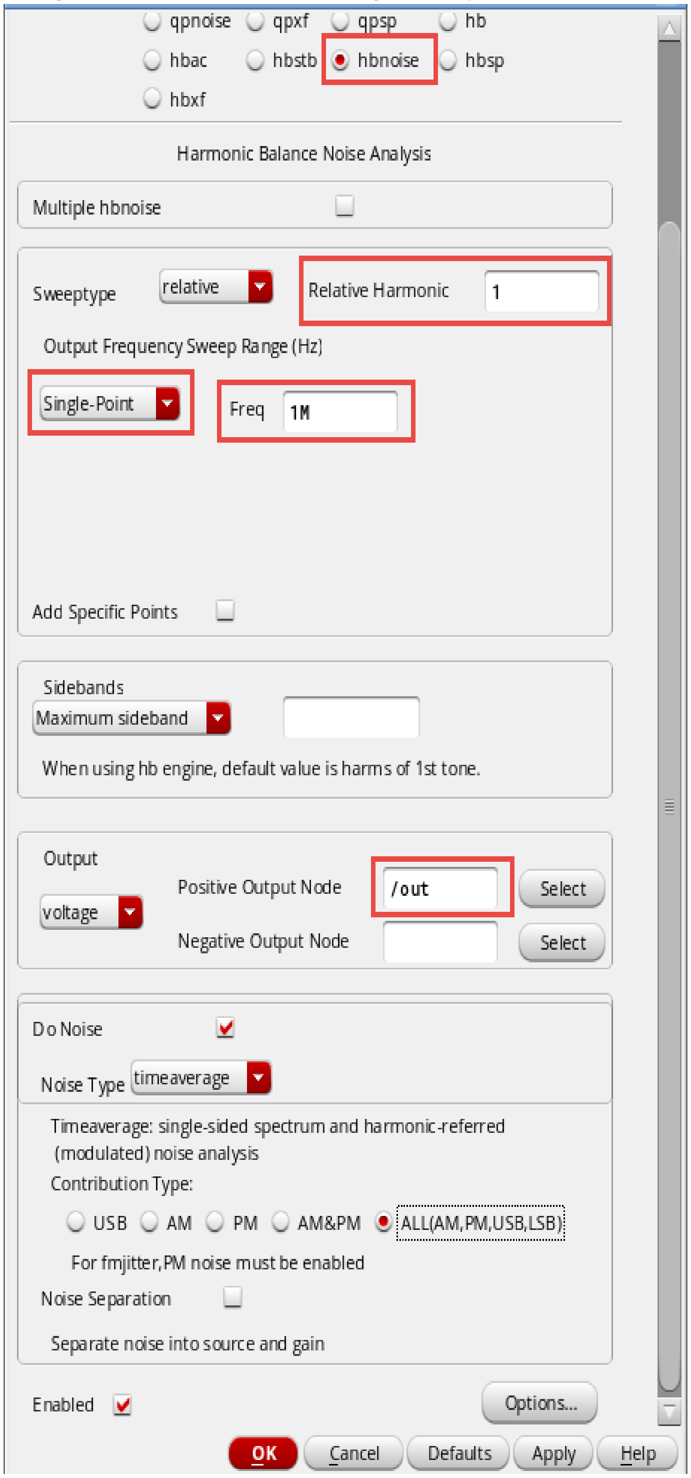
Figure 3-128 Selecting *out* net from the schematic



5. Select the *ALL(AM,PM,USB,LSB)* option.

   The *Choosing Analyses* form should look like the following:

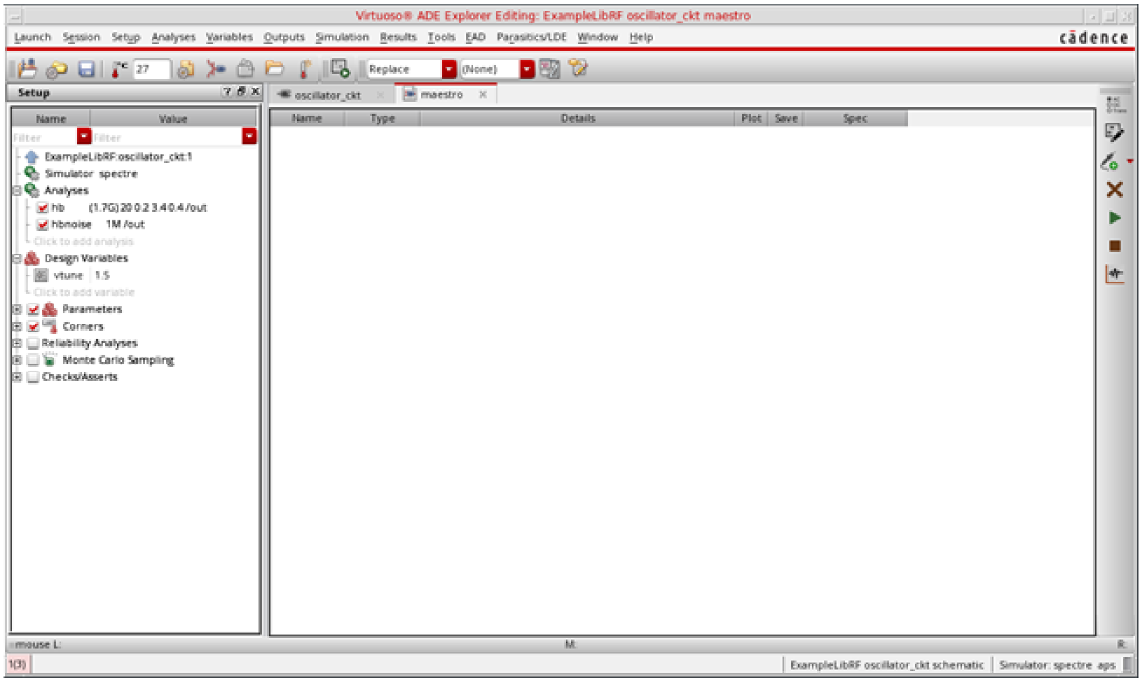
Figure 3-129 Choosing Analysis Form - *hbnoise* Analysis Setup



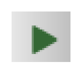
6. Click *OK* to close the *Choosing Analyses* form.

This will add *hbnoise* analysis along with *hb* analysis in the *Analyses* section of ADE Explorer, as shown below:

Figure 3-130 ADE Explorer Simulation Window - *hb* and *hbnoise* analysis setup



Running the simulation

Once finished setting up the *hb* and *hbnoise* analyses click the green icon  on the right of the *ADE Explorer* window or on the Schematic window to run the simulation.

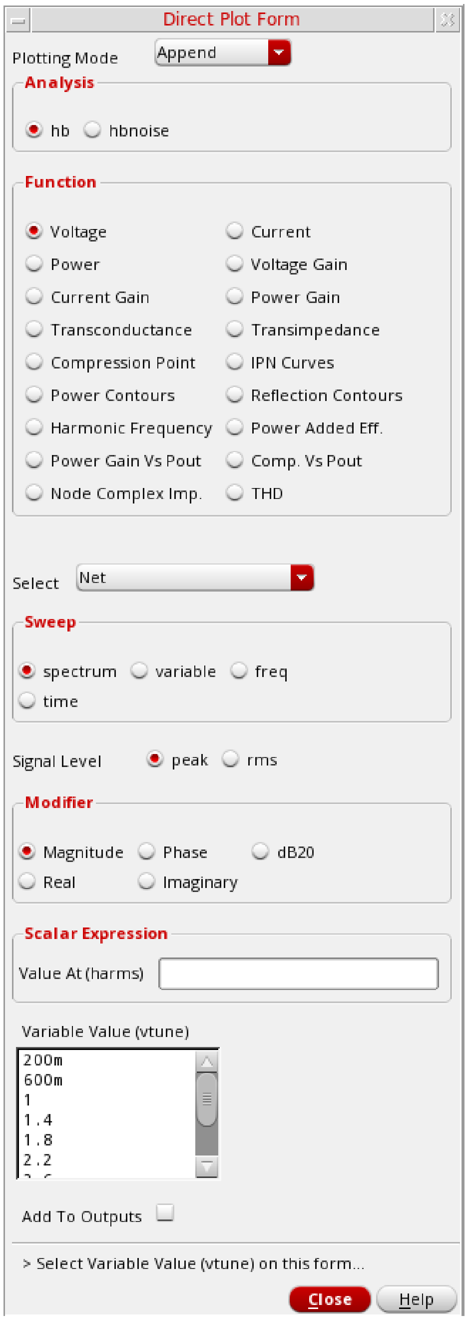
Next, plot the results when the simulation is finished.

Plotting the results

First plot the oscillator output frequency.

1. In ADE Explorer, select *Results - Direct Plot - Main Form*.

Figure 3-131 Swept *hb* and *hbnoise* Analysis Direct Plot Form

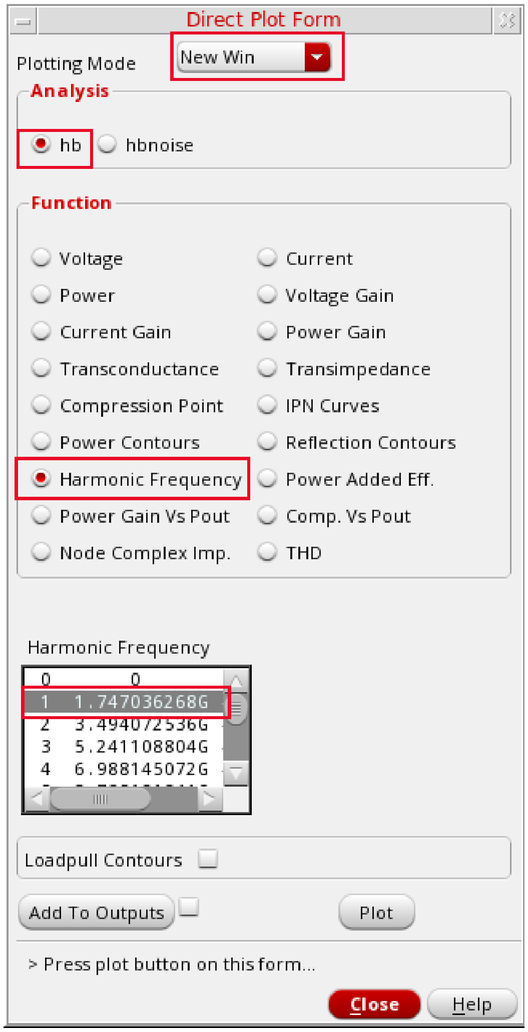


2. In the *Direct Plot Form*, select *New Win*.
3. Select *hb* in the *Analysis* section. This is selected by default.
4. Select *Harmonic Frequency* in the *Function* section.

5. In the *Harmonic Frequency* section, select the first harmonic. This will plot only the change in first harmonic of the oscillation frequency vs. change in oscillator's tune voltage.

   The *Direct Plot Form* setup should look like the following:

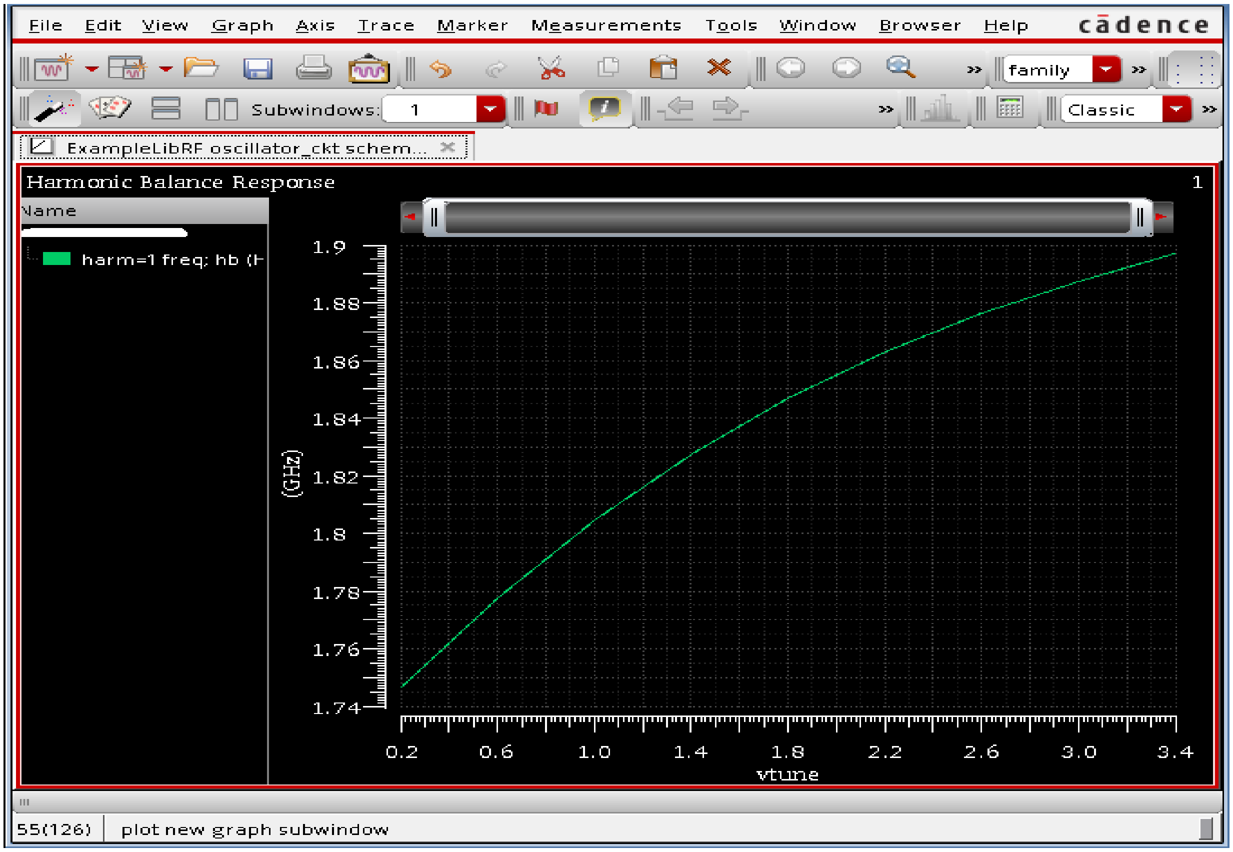
Figure 3-132 Swept *hb* Analysis Direct Plot Form Setup



6. Click *Plot*.

   The oscillator tuning range is plotted. You can see from the plot that as you increase the tuning voltage, the oscillator frequency increases.

Figure 3-133 Swept HB Measurement Plot - vtune vs. osc frequency variation plot



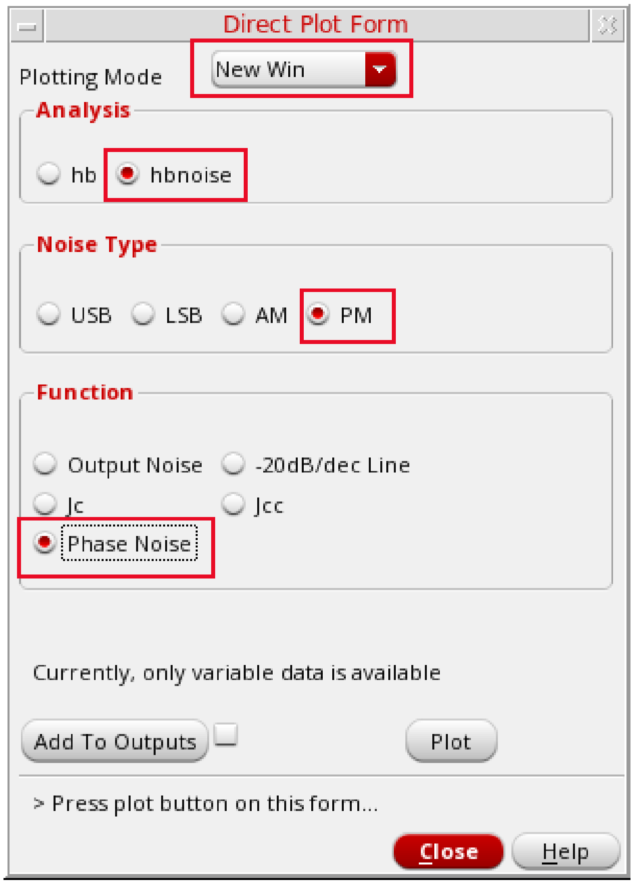
7. Select *File - Close All Windows* to close ViVA.

Next plot the phase noise.

1. In the *Direct Plot Form*, select *New Win*.
2. Select *hbnoise* in the *Analysis* section.
3. Select *PM* in the *Type* section.
4. Select *Phase Noise* in the *Function* section.

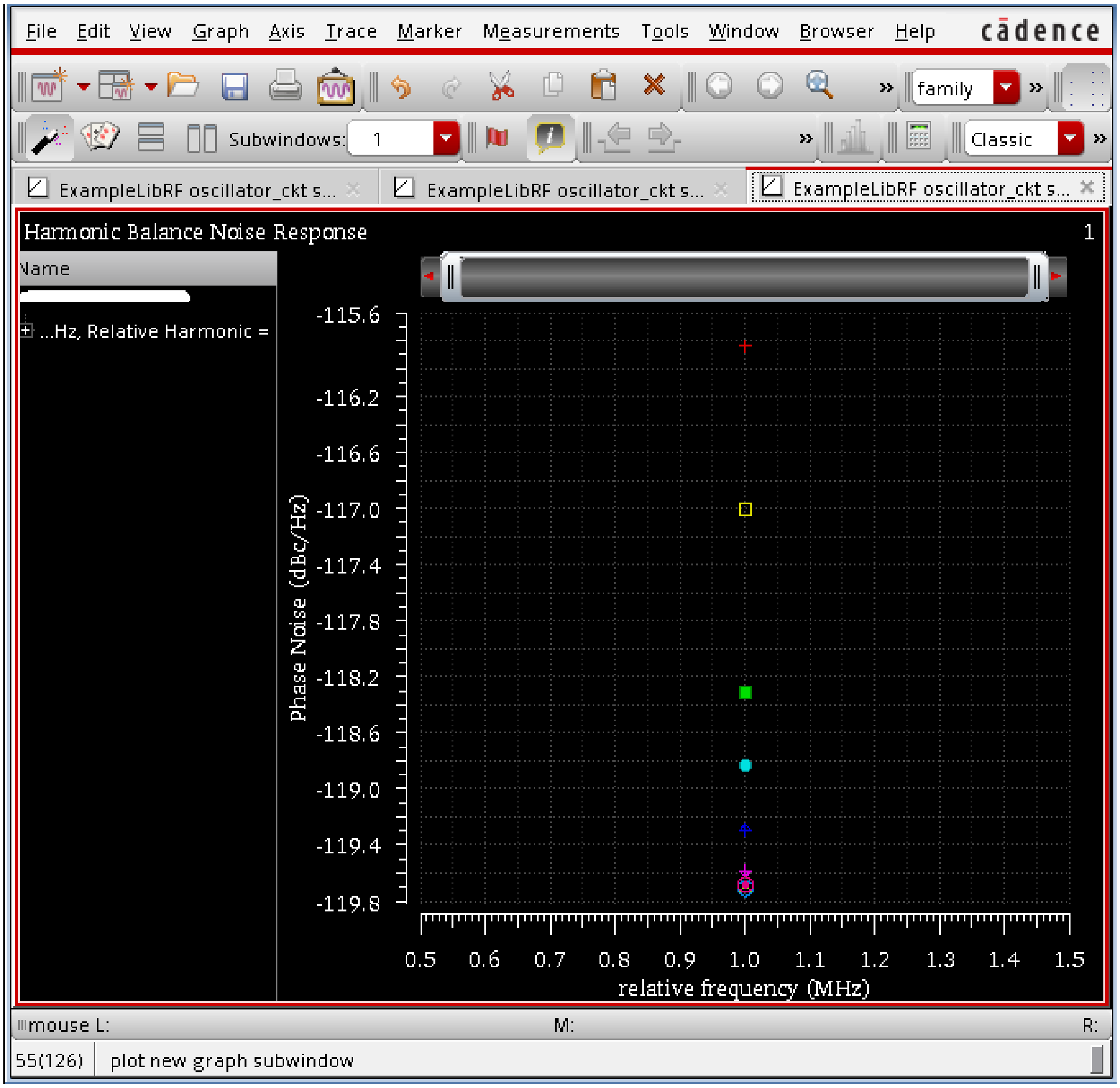
   The *Direct Plot Form* should look like the following:

Figure 3-134 *hbnoise* Direct Plot Setup



5. Click *Plot*. This will plot the Single Sideband Phase Noise vs. Relative Frequency graph, as shown below.

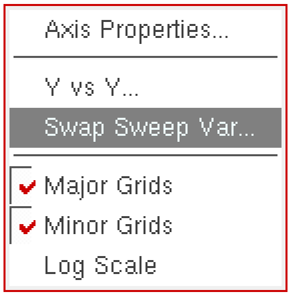
Figure 3-135 Phase Noise vs. Relative Frequency Plot



In the above plot, you have a graph of phase noise vs. relative frequency. Next, you will modify the phase noise plot to provide more useful information. In the next few steps you will plot phase noise vs. tuning voltage.

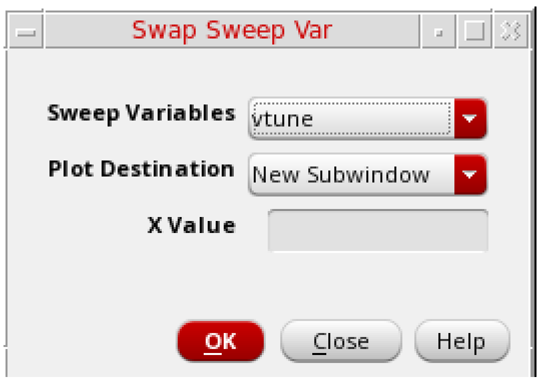
6. In the plot window, right-click one of the numbers on the X Axis, and select *Swap Sweep Var* from the context menu.

Figure 3-136 X-Axis - Right Mouse Button (RMB) - Object Menu



7. In *Select Sweep Var* form which opens, select *vtune* from the *Sweep Variables* drop-down list. Keep *Plot Destination* as *New SubWindow*.

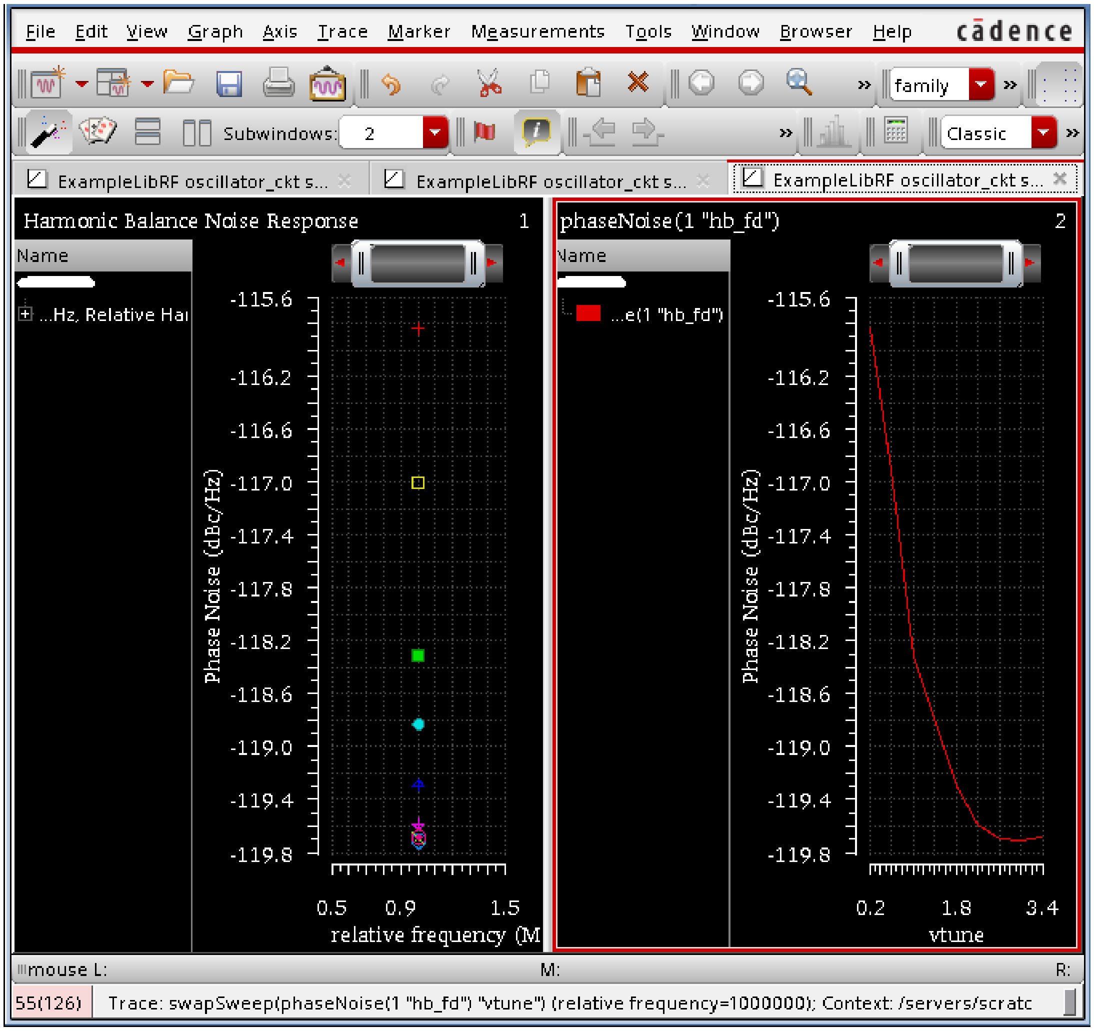
Figure 3-137 Swap Sweep Var Dialog Box Window



8. Click *OK* to close the Swap Sweep Var form.

   The waveform tool draws a new subwindow.

Figure 3-138 vtune vs. Phase Noise Plot



If you look at the graph on the right, you will see that the phase noise is worse at the lower end of the tuning range.

9. In the *Direct Plot Form*, click *Cancel*.
10. In the waveform window, choose *File - Close All Windows*.
11. Clean up the screen for the next set of measurements.
    a. Close ADE Explorer by selecting *Session - Quit*.
    b. In the Schematic window, select *File - Close*.

Summary

In this section, Tuning Voltage vs. Oscillation Frequency and Phase Noise vs. Tuning Voltage Measurements were done.

Ring Oscillator Measurements

In this section, you will perform measurements on Ring Oscillator.

Starting and Stabilization of Ring Oscillators

To simulate a Ring Oscillator, it is recommended to use Shooting PSS analysis as they are highly non-linear in nature. Recent improvements in harmonic balance allows ring oscillators to be simulated in hb as well. In this case, set the oversample factor to 4. In the tstab interval, or in the transient analysis, measure the fastest slew rate for the rising and falling edges, and calculate the zero to 100% transition time based on this slew rate measurement. Set the number of harmonics to at least the period of the oscillations divided by the transition time. You can first start the oscillator by supplying initial conditions(ICs). You should choose initial conditions such that a transient analysis displays the correct oscillation frequency (and amplitude) at the output of the ring oscillator. This is the actual check that your initial conditions are working to start the oscillator. As a suggestion, in a ring oscillator, you set the output of one stage high or low. If the oscillator configuration is differential, then set the output of 1 stage high on one differential net, and low on the other differential net for the same stage.

Regardless of which technique you use to start the oscillator, allow the oscillator to run long enough to stabilize before you start the Shooting phase and compute the steady state solution. Adjust the *tstab* parameter to supply the additional stabilization time, a value of 2-3 periods is recommended.

It is recommended to save the initial transient simulation results. This can be done by setting *saveinit=yes*. You can use this to verify that the oscillator has started up and is stable.

In the case of Injection Locked Ring Oscillator where you apply a signal (vsource) to your ring oscillator which causes the oscillator to oscillate at the injection frequency makes it a driven circuit - as it is driven by the injection source. Thus, in this case, you do not use an autonomous PSS analysis. In order to do that, you do not click the *Oscillator* option on the PSS *Choosing Analyses* form for such oscillators. SpectreRF considers this type of circuit a driven circuit and will error out if you click the *Oscillator* option on the PSS *Choosing Analyses* form.

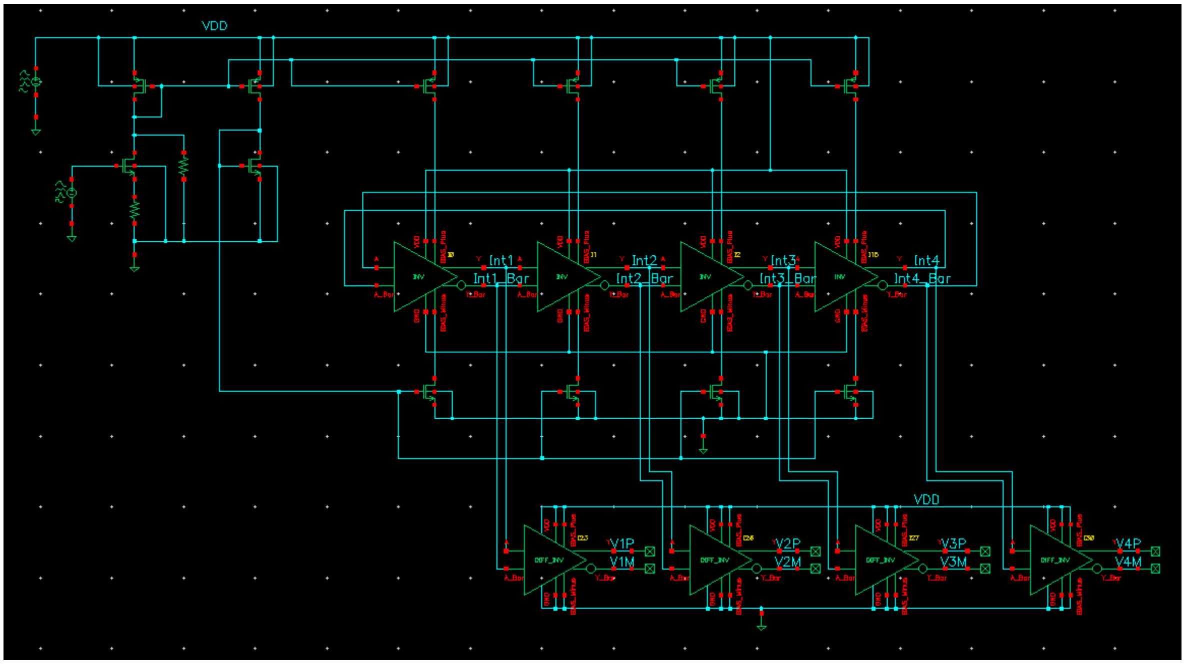
See *Oscillators and Autonomous PSS Analysis* in the *Spectre Circuit Simulator RF Analysis Theory* for more information on oscillator simulation.

The Oscillator Circuit

This example computes the periodic steady state solution and the phase noise for the *Ring oscillator* circuit shown in Figure 3-139.

This is a four stage fully differential ring oscillator designed at 1.9GHz.

Figure 3-139 Schematic for the Ring Oscillator Circuit ringOsc_1900M

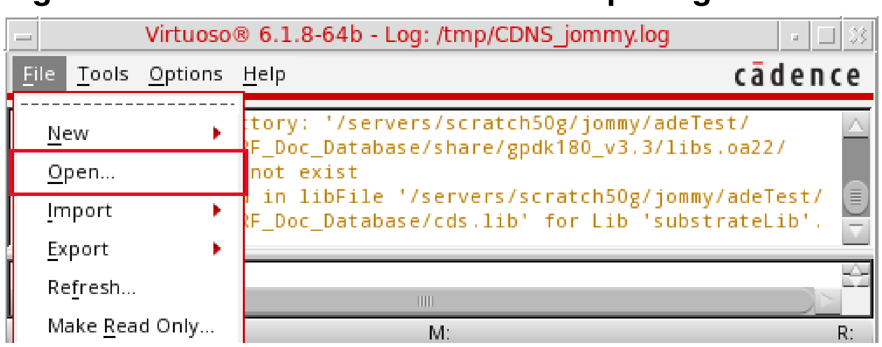


Simulating the Oscillator Circuit

Opening the Oscillator Circuit in the Schematic Window

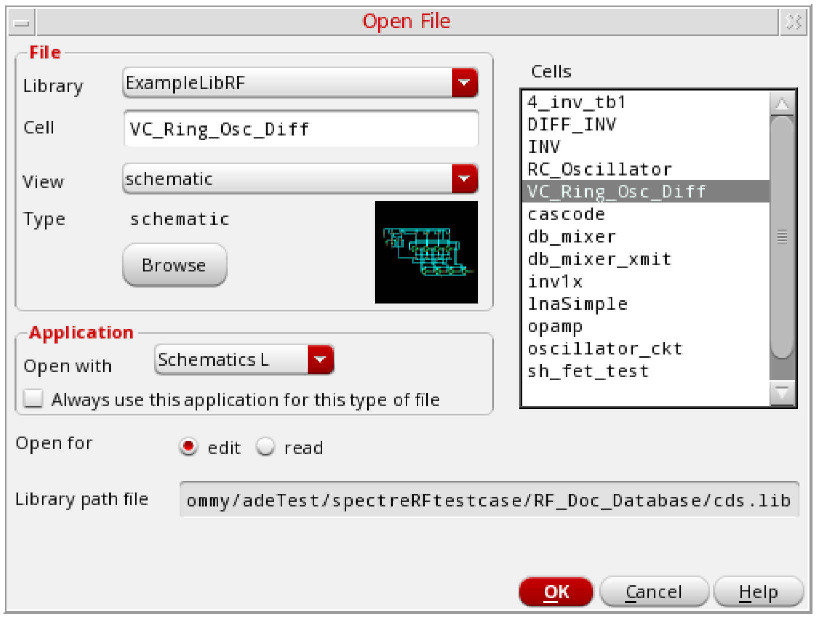
1. In the Command Interpreter Window (CIW), choose *File –> Open*.

Figure 3-140 Virtuoso CIW Window - Opening Cellview



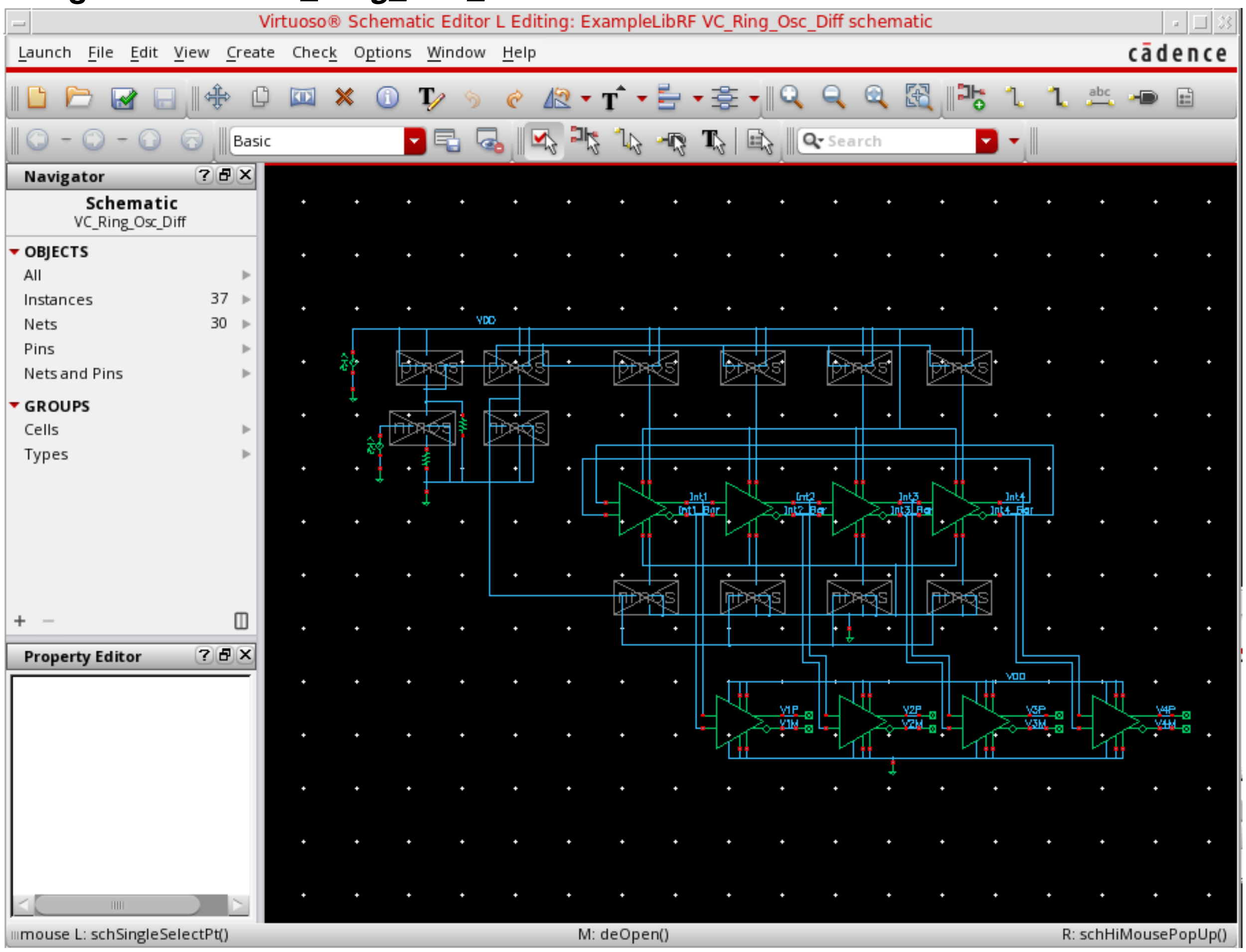
The *Open File* form is displayed.

Figure 3-141 Open File Form to open the testbench_1900M cell's Schematic View



2. Select *ExampleLibRF* from the *Library* drop-down list.
3. In the *Cells* field, type *VC_Ring_Osc_Diff*.
4. Choose *schematic* from the View drop-down list.
5. In the *Application* field, select *Schematic L* from the Open *With* drop-down list.
6. Leave *Open For* to *Edit* (which is set by default).
7. Once all the setup is done, click *OK*.
8. This will open the *VC_Ring_Osc_Diff* schematic in Virtuoso Schematic Editor L window, as shown below:

Figure 3-142 ***VC_Ring_Osc_Diff*** **schematic in VSE-L Window**



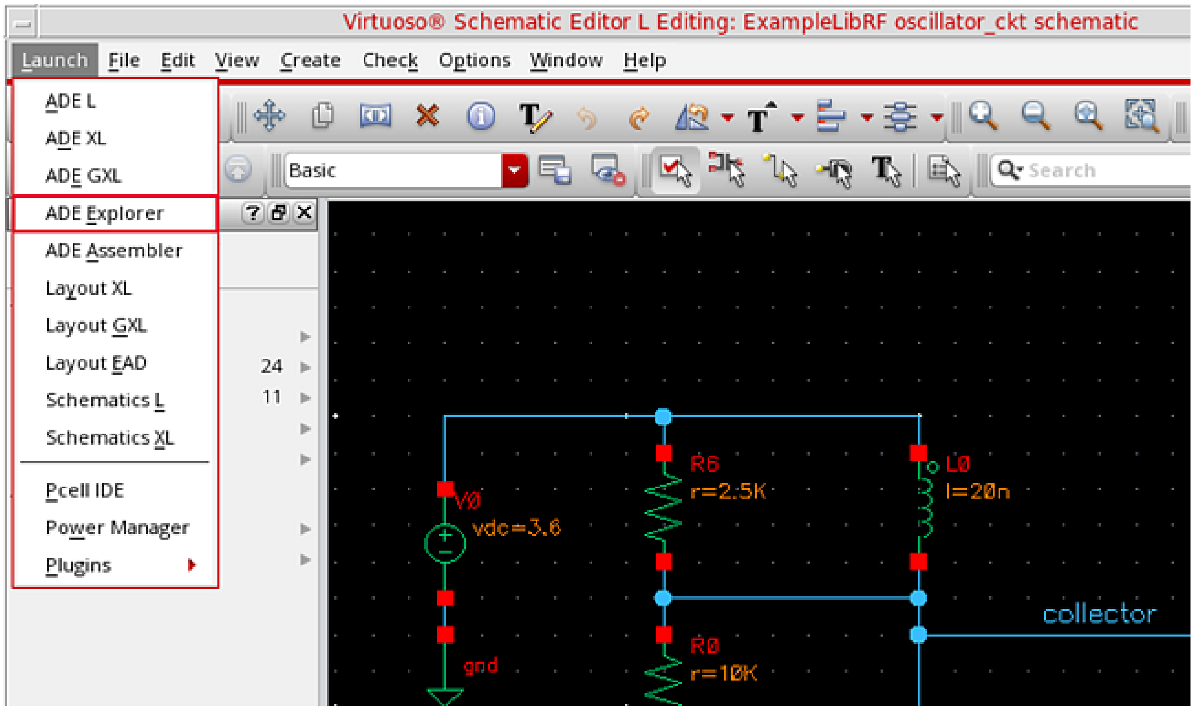
Calculating the Steady-State Solution using PSS Shooting Analysis

This example computes the periodic steady state solution for the *VC_Ring_Osc_Diff* oscillator circuit. You perform a PSS-Shooting analysis first because the periodic steady state solution must be determined before you can perform any other periodic small-signal analysis, such as pnoise to determine phase noise.

Setting up the PSS Analysis

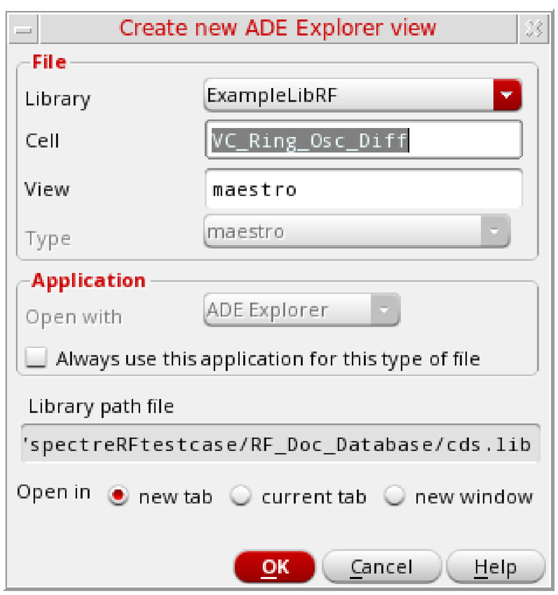
1. In the Schematic Window, select *Launch - ADE Explorer.*

Figure 3-143 Opening ADE Explorer window from VSE window



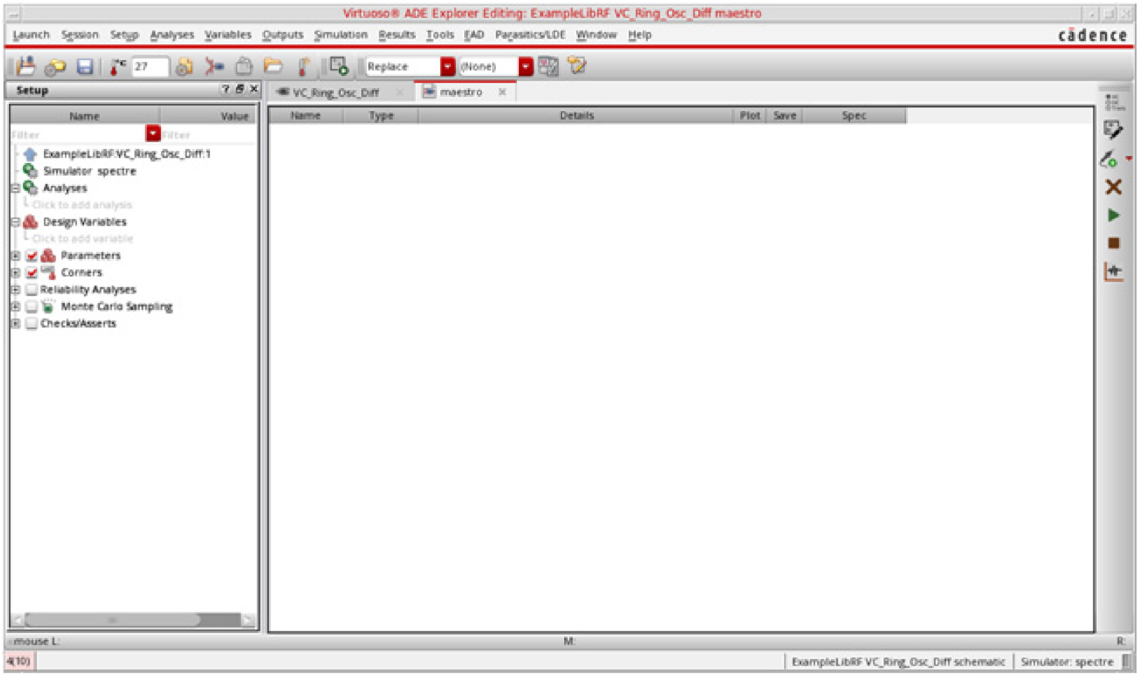
2. Click OK in the Create new ADE Explorer view form.

Figure 3-144 Create new ADE Explorer view Form



ADE Explorer Window opens, as shown below.

Figure 3-145 Virtuoso ADE Explorer Window



3. Select *Setup – Simulator*.

   The *Choosing Simulator* form is displayed.

Figure 3-146 Choosing Simulator Form

4. Select *spectre* from the *Simulator* drop-down list.
5. Click *OK* to close the *Choosing Simulator* form.
6. Set up the High Performance Simulation options.

   In ADE Explorer, select *Setup - High Performance Simulation*. The High Performance Simulation Options form is displayed.

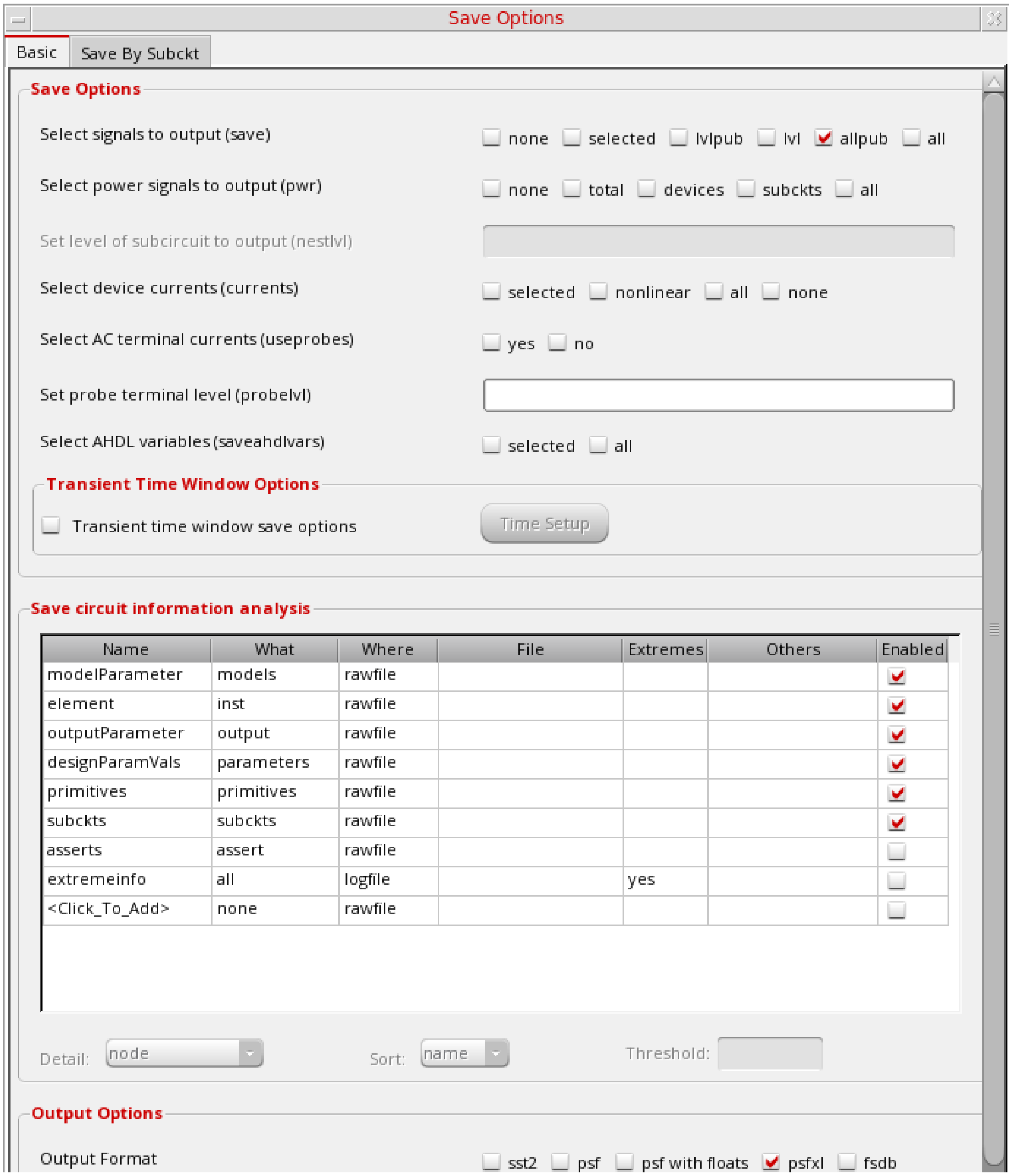
   In the *High Performance Simulation Options* form, select *APS*. Note that *Auto* is selected for Multithreading options. The effect of this is to detect the number of cores on the system (up to 64) and then multi-thread on all the available cores. Usually it is better to specify the number of threads yourself. Small circuits should use a small number of threads, and larger circuits can use more threads. The overhead of managing 16 threads on a smaller circuit may actually slow the simulation down, compared to two or four threads. Use the number of threads that maximizes the performance.

Figure 3-147 High Performance Simulation Options Form

7. Click *OK* to close the *High Performance Simulation Options* form.
8. Select *Outputs - Save All.*

   The *Save Options* form is displayed.
9. In the *Select signals to output(save)* section, make sure that *allpub* is selected.

Figure 3-148 Save Options Form



This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

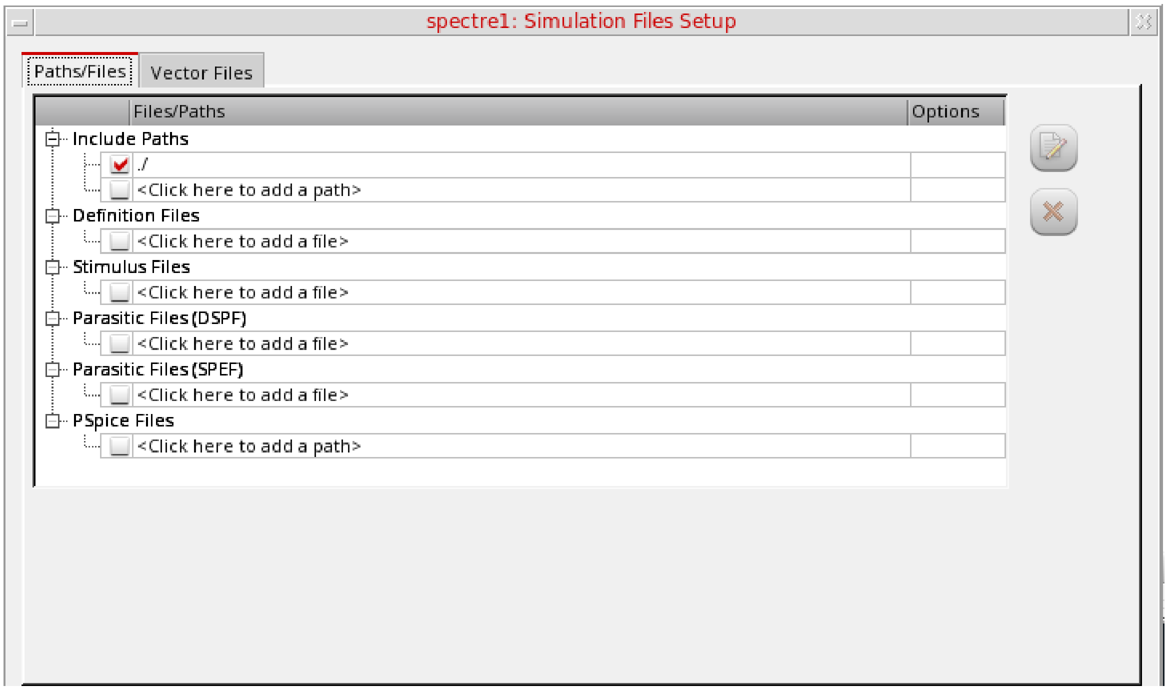
To save the currents, select the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or all if you want to save all the currents in the circuit. When you save currents, more disk space is required for the results file.

10. Click *OK*.

11. Select *Setup - Simulation Files.*

12. In the *Simulation Files Setup* form which gets opened, enter `./` by clicking in the *Include Paths* section. The form should look like the following:

Figure 3-149 ***Simulation Files Setup*** **Form**



13. Click *OK* to close the *Simulation Files Setup* form.

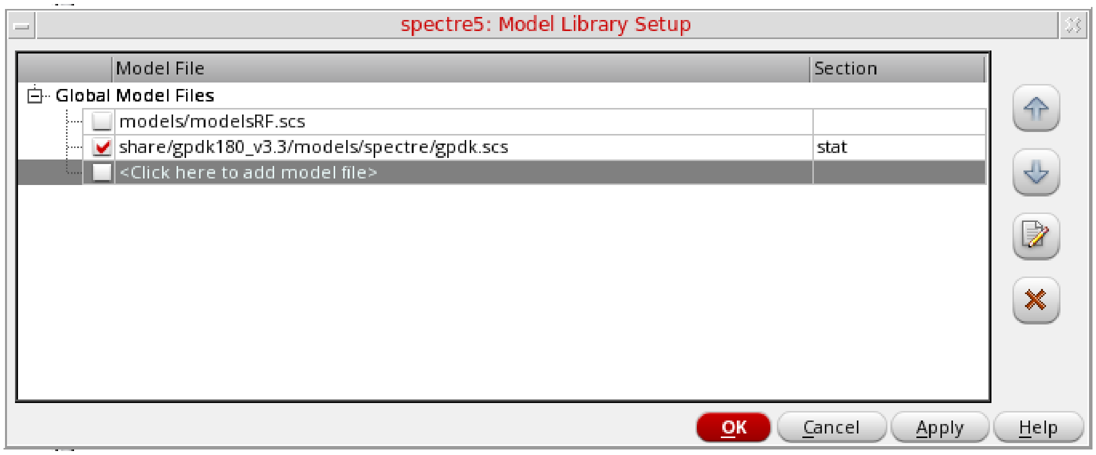
14. Select *Setup - Model Libraries*.

The *Model Library Setup* form is displayed.

15. In the *Model File* field, type the path to the model file including the file name, as shown below.

```
share/gpdk180_v3.3/models/spectre/gpdk.scs.
```

Figure 3-150 ***Model Library Setup*** **Form**



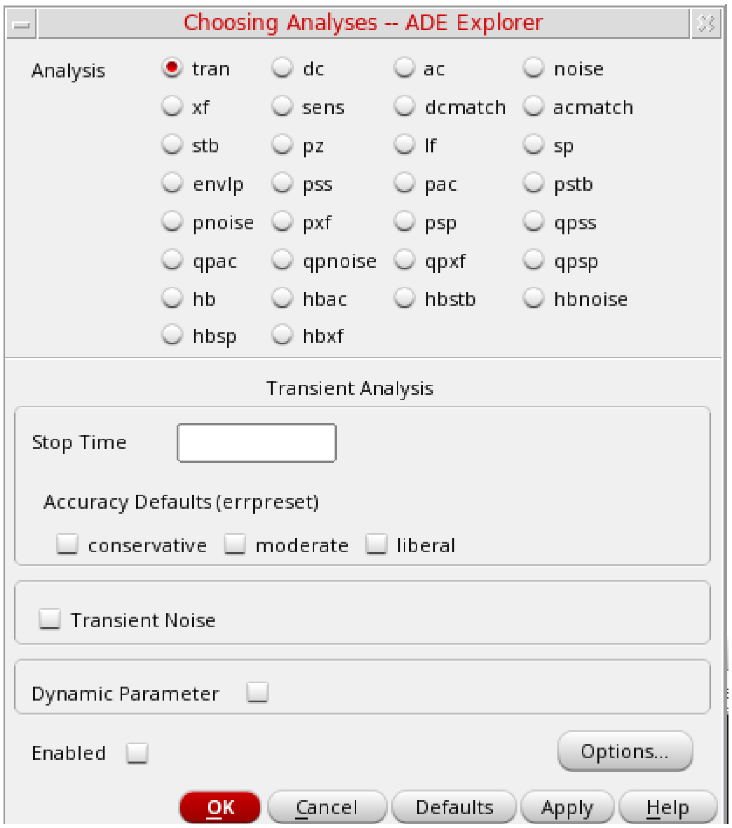
You can also browse to *gpdk.scs* file and then set the *Section* to *stat*.

16. Click *OK* to close the *Model Library Setup* form.

17. Click *Analyses - Choose...* in ADE Explorer.

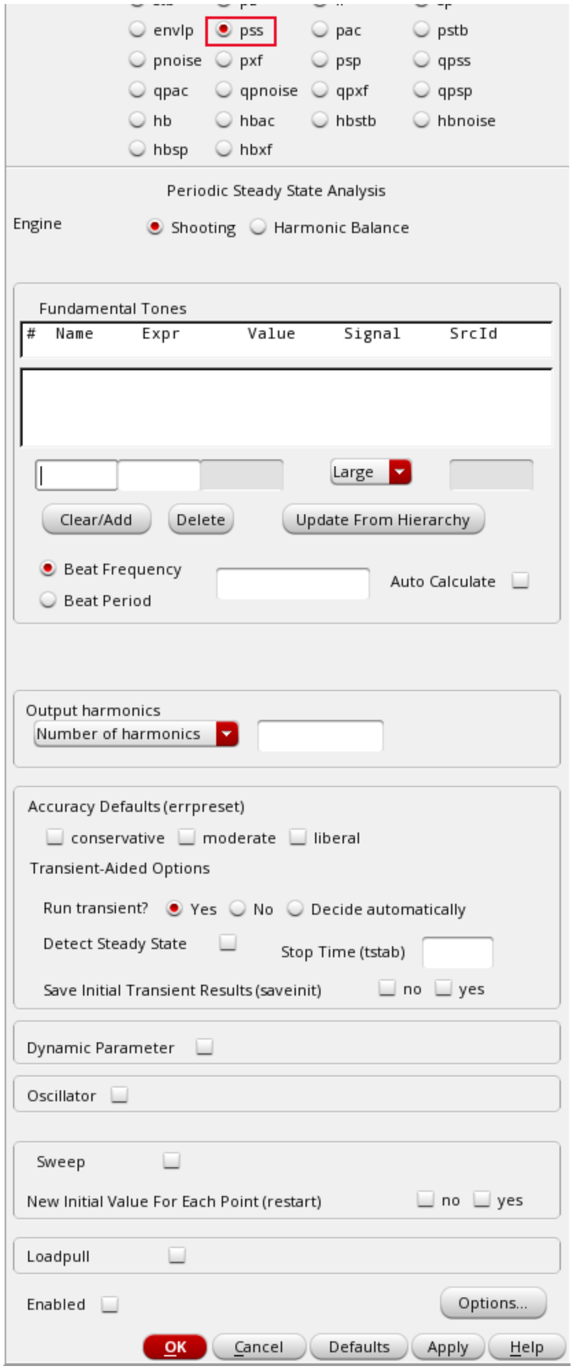
The *Choosing Analyses* form is displayed.

Figure 3-151 The Choosing Analyses Form



18. In the *Analysis* section, select *pss*. The form expands, as shown below.

Figure 3-152 The Choosing Analyses Form- Setting PSS Analysis



19. In the *Engine* section, verify that *Shooting* is selected (this is the default).

20. In the *Beat Frequency* field, type `1.9G`. The frequency entered here is an approximate frequency of oscillation.

21. In the *Number of harmonics* field, type `20`.

In general, you want to choose a number that is high enough to capture the nonlinearity of the circuit.

22. In the *Accuracy Defaults* (*errpreset*) section, select *conservative*.

conservative is typically used because very small amplitude phase noise measurements are normally desired. *conservative* is recommended for all oscillators.

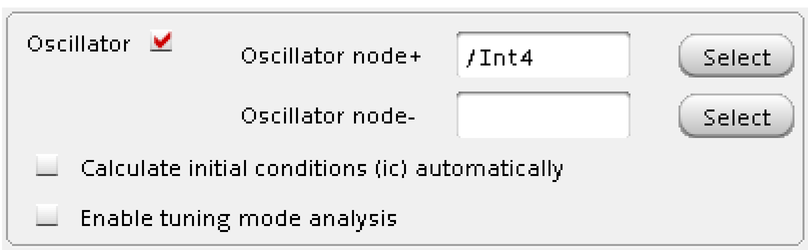
23. Select *yes* for *Run Transient*.

24. Type `2n` in the *Stop Time (tstab)* field. *tstab* is typically set to about 2-3 periods of the oscillation frequency for ring oscillator circuits. The tstab should be long enough so that the oscillator must reach near the steady-state behavior during this phase of simulation. This would result in better convergence.

25. Select *yes* for Save Initial Transient Results. This will help in visualizing the buildup of the oscillation waveform.

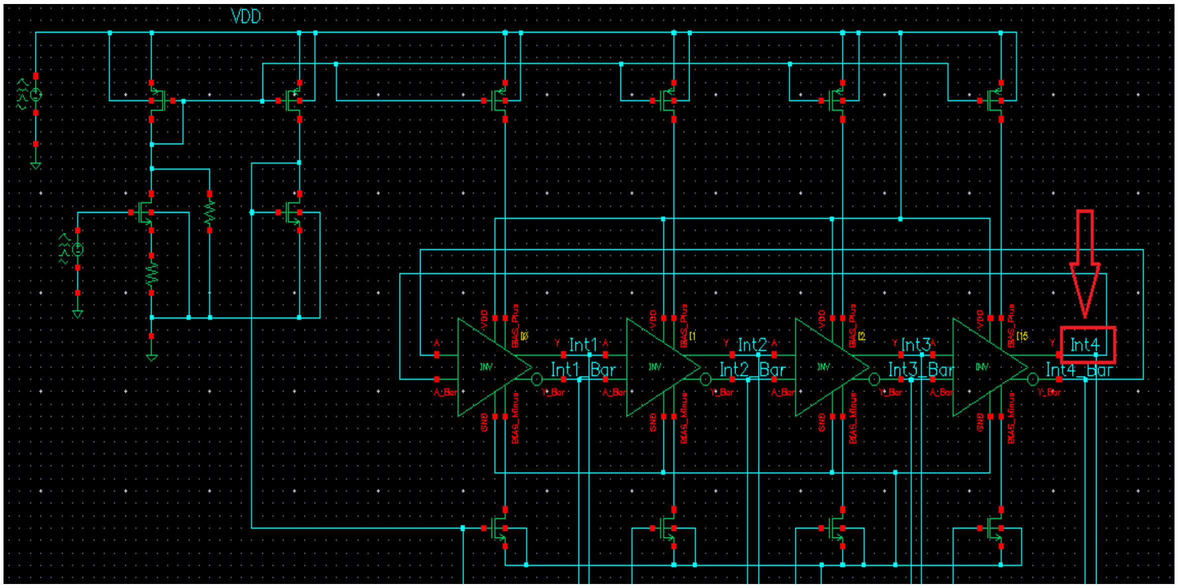
26. Select the *Oscillator* option. This is required for simulating an autonomous circuit. The oscillator section expands, as shown below.

Figure 3-153 The Choosing Analyses Form - Oscillator Section



27. In the *Oscillator node+* field, click *Select* to the right. In the schematic, select the *Int4* node. This oscillator node will be used by the simulator for the period calculation of the oscillations. It just needs to be a node that has the oscillator signal on it.

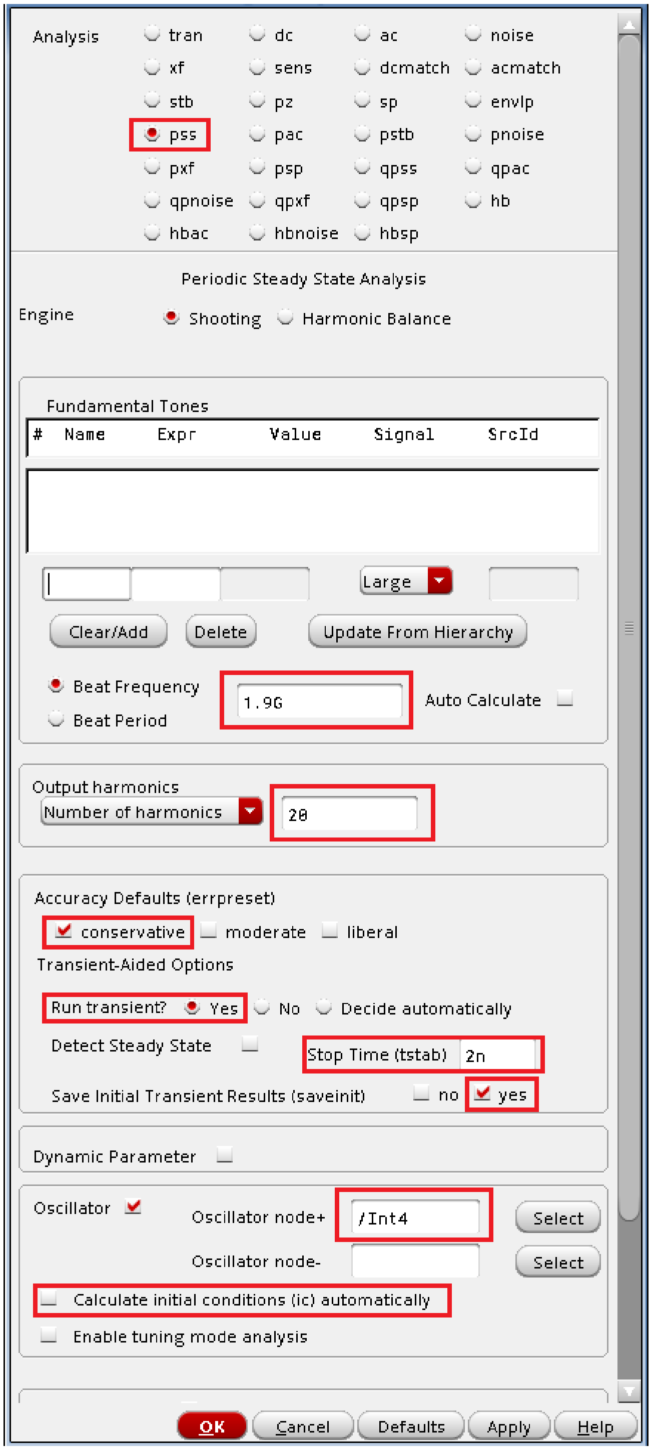
Figure 3-154 Selecting *Int4* net on schematic



28. Deselect the *Calculate initial conditions (ic) automatically* checkbox. This is because selecting this checkbox only works for feedback oscillators while ring oscillator is not a feedback oscillator. Therefore, selecting this checkbox will not be able to find an oscillatory state for ring oscillator.

The *Choosing Analysis* form should look like the following:

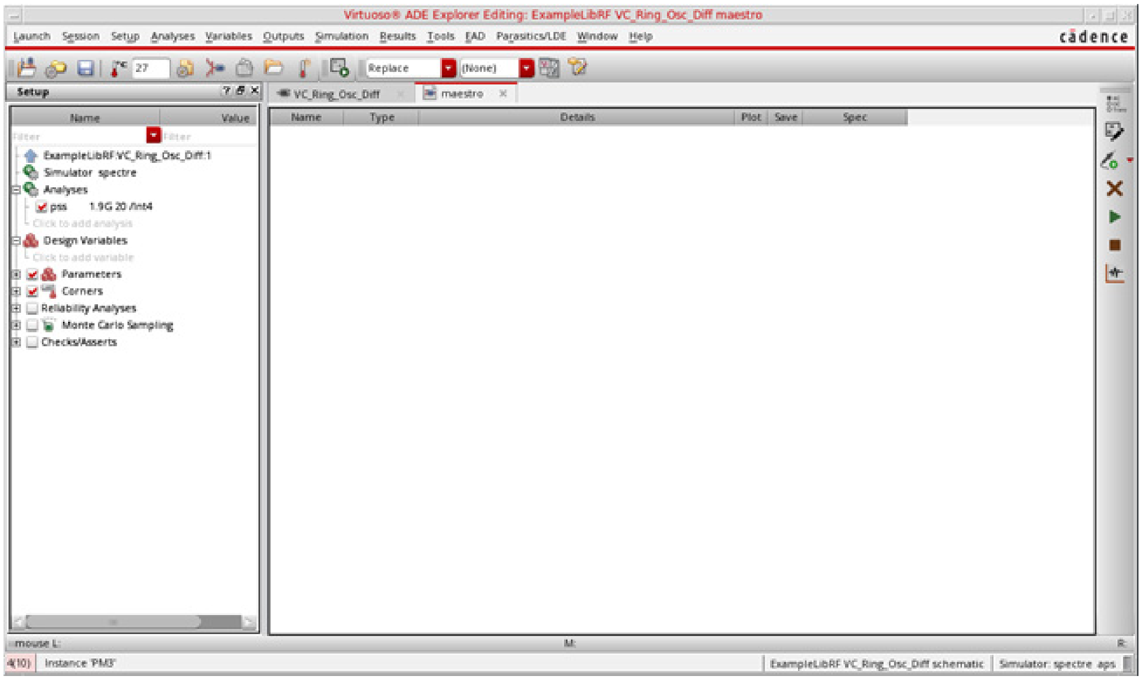
Figure 3-155 Choosing Analyses Form - PSS-Shooting Setup



29. Click *OK*.

This will close the *Choosing Analyses* form. In addition, this will add the *pss* analysis in the *Analyses* section of ADE Explorer, as shown below:

Figure 3-156 ADE Explorer Simulation Window - PSS Analysis



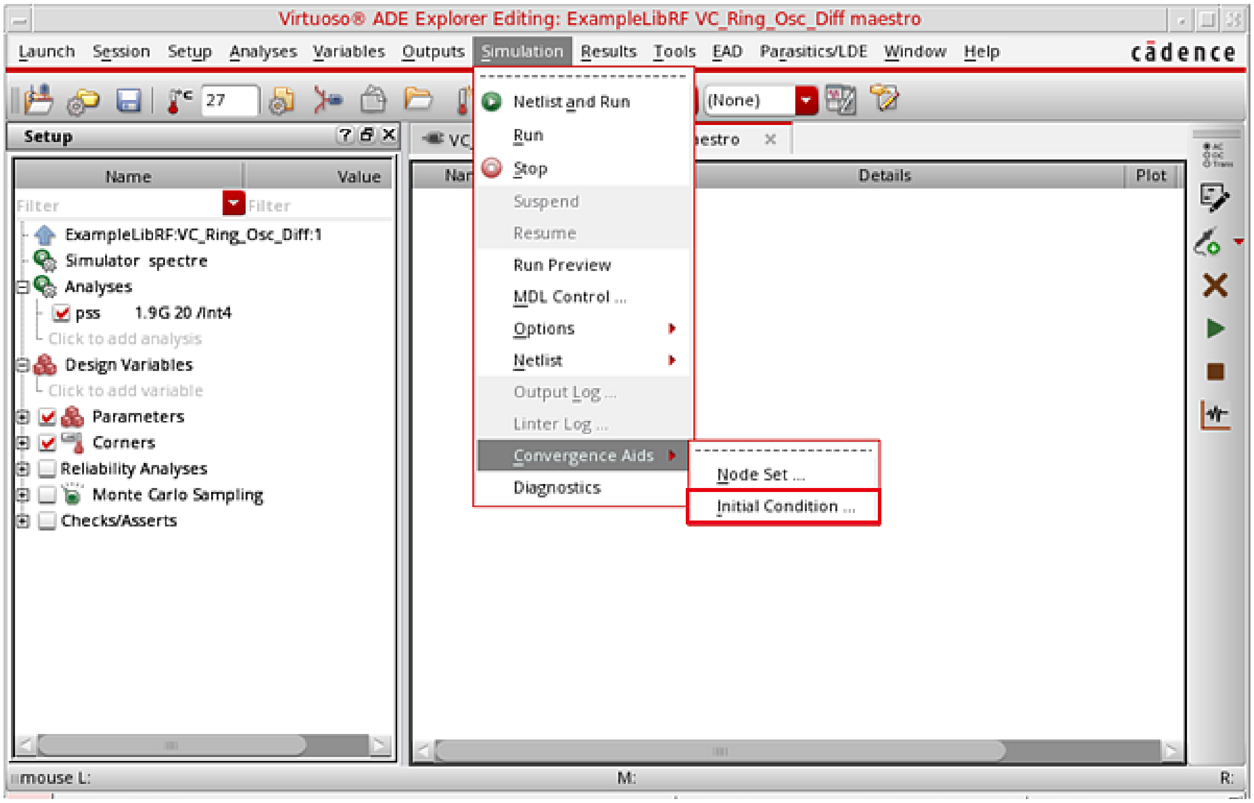
Setting initial conditions forces the circuit to a specific state at the time zero timepoint, and then removes that force after the time zero point is calculated.

In this case, we will set one stage of the oscillator so that one differential output is high, and one differential output (in the same stage) is low.

This defines all the nodes in the circuit, and the nodes that are connected to the nodes that are forced are pulling as hard as they can to get the forced nodes to change to the other state. When the forcing condition is removed, this action starts the oscillations.

Set the initial conditions by choosing *Simulation - Convergence Aids - Initial Condition* in ADE Explorer, as shown below -

Figure 3-157 PSS Analysis - Setting Initial Condition in ADE



The *Select Initial Condition Set* form is displayed.

30. Type `0` (zero) in the *Node Voltage* field.

31. Select *Int4_Bar* node in the schematic. Note that the node highlights in the schematic.

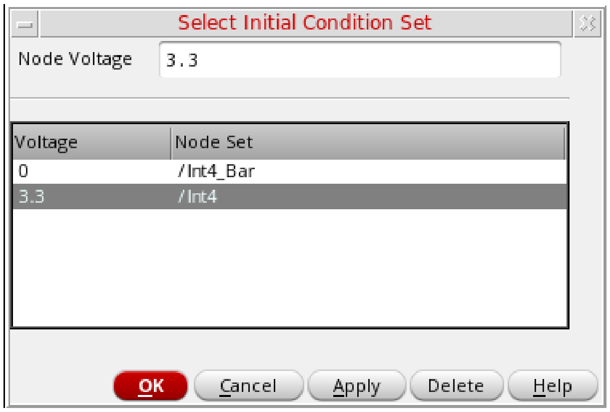
32. Click *Apply*.

33. Type `3.3` in the *Node Voltage* field.

34. Select the *Int4* node in the schematic. Note that the node highlights in the schematic.

The populated *Select Initial Condition Set* dialog box would look like the following:

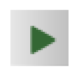
Figure 3-158 Setting Initial Condition Form



35. Click *OK*.

This finishes the setting of PSS Analysis with setting up of Initial Conditions.

Running the PSS analysis

Once finished setting up the PSS analysis, click the green icon  on the right side of ADE Explorer or on the Schematic window to run the simulation.

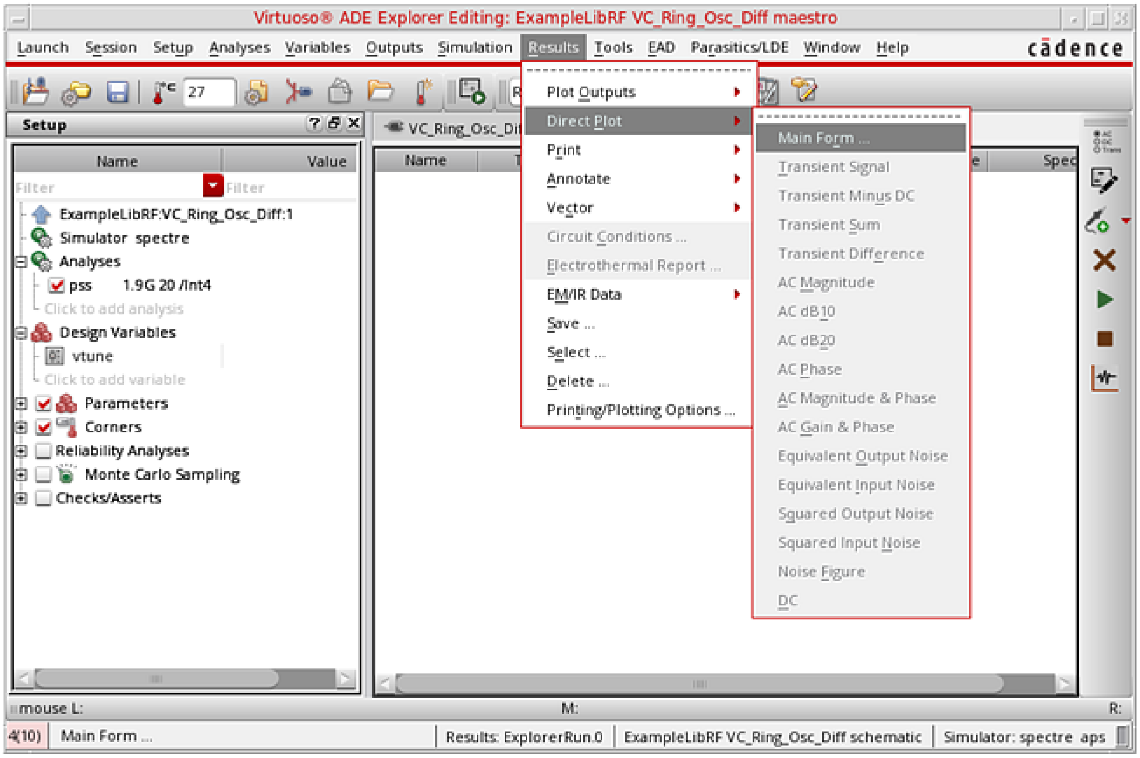
This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` logfile). When the analysis has completed, you may iconify the status window.

Next, you will plot the results.

Plotting the PSS Analysis Results

1. In ADE Explorer, select *Results - Direct Plot - Main Form*.

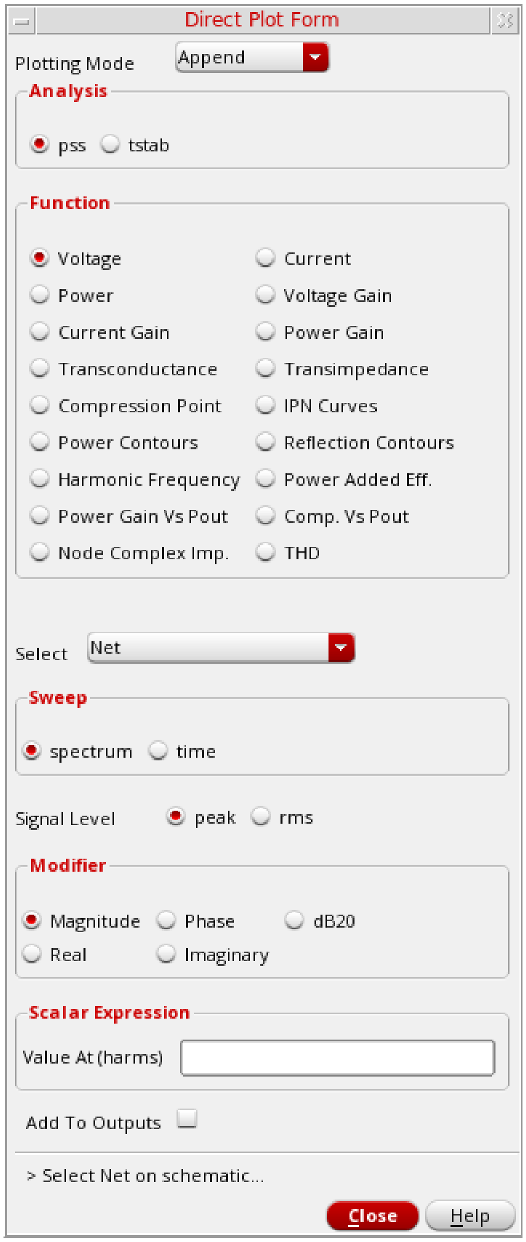
Figure 3-159 Invoking Direct Plot Form



The *Direct Plot Form* is displayed.

You will see that there is pss analysis and tstab analysis in the *Analysis* section although you have only run pss analysis. The tstab analysis gets added as part of pss analysis run and is used to plot the initial transient waveforms.

Figure 3-160 The Direct Plot Form

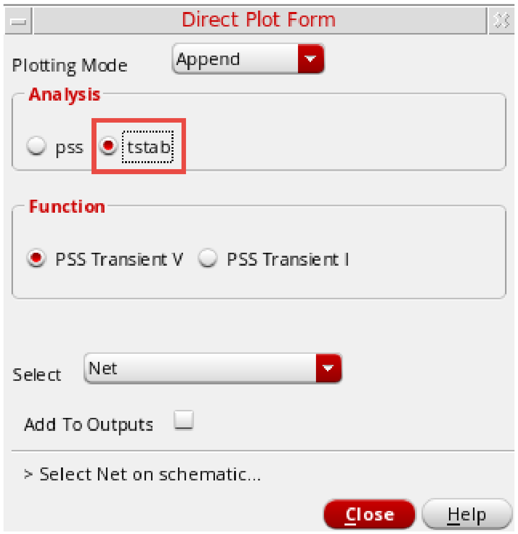


Plot the oscillator startup waveform from the tstab run.

2. In the *Direct Plot Form*, select *tstab* in the *Analysis* section.
3. Leave *Function* as *PSS Transient V* which is set by default.
4. Select *Net* in the center of the form. (This is the default. You can also select differential nets).

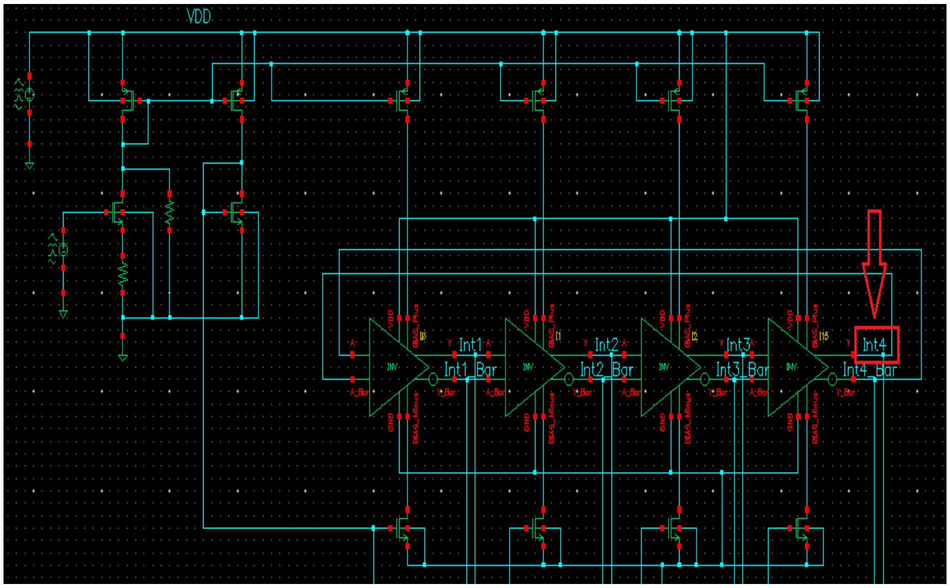
The *Direct Plot Form* window should like the following:

Figure 3-161 PSS Analysis Direct Plot Setup - Initial Transient



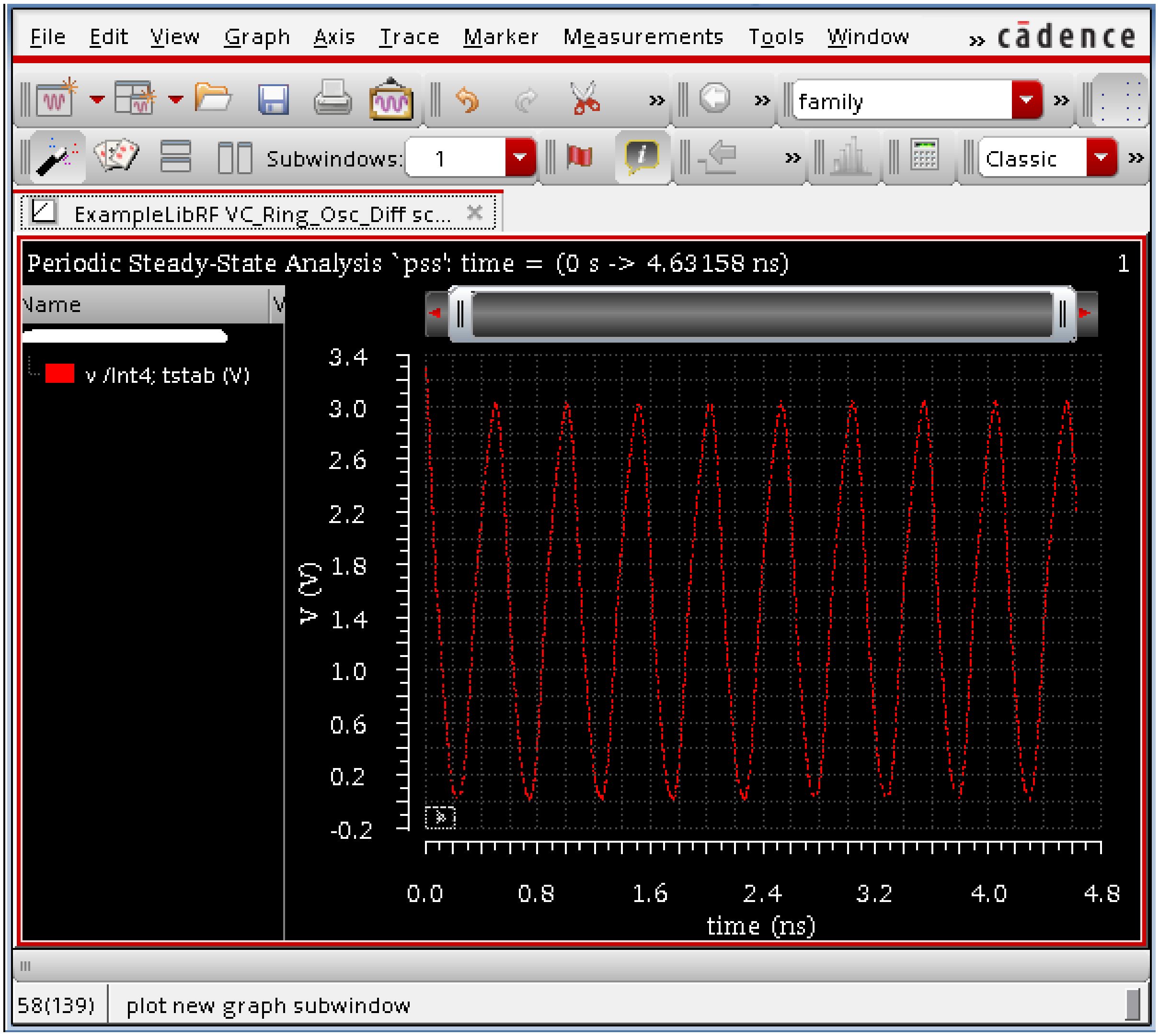
5. Select the *Int4* net in the schematic. It is located just below the *Int4* label.

Figure 3-162 Selecting *Int4* net on schematic



The waveform window is displayed, as shown below.

Figure 3-163 PSS Analysis Initial Transient Voltage Waveform



You can see from the plot that the initial transient waveform is for 4.63158ns as mentioned in the Output log window. The oscillator is in steady state.

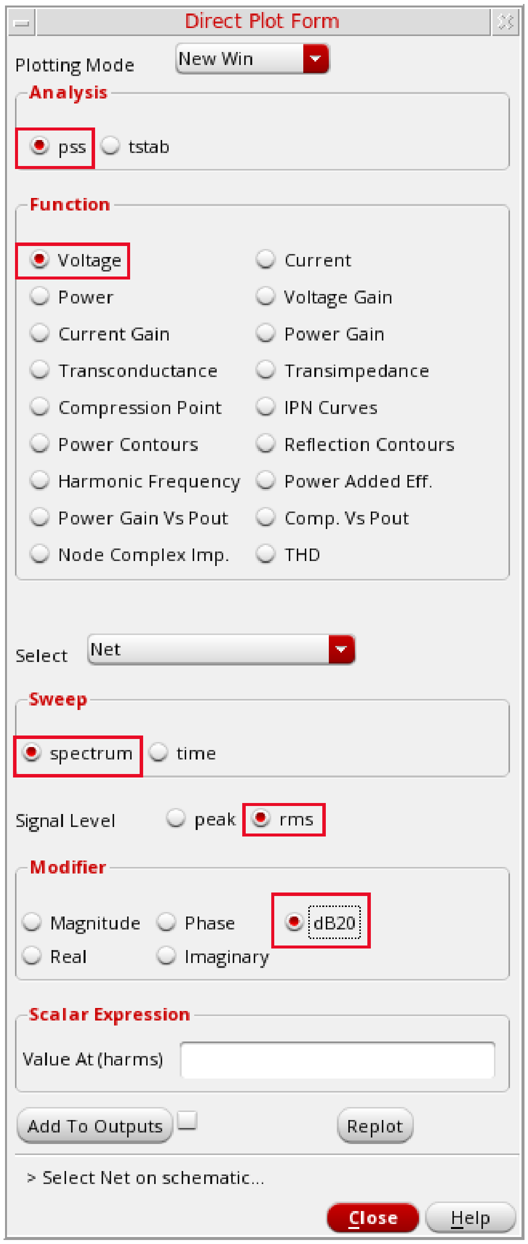
Next, you will plot the oscillator output spectrum.

1. In the *Direct Plot Form*, set the *Plotting Mode* to *New Win*.
2. Select *pss* in the *Analysis* section.
3. Leave *Function* as *Voltage*, which is set by default.
4. Select *Net* in the center of the form. (This is the default. You can also select differential nets).

5. Select *spectrum* in the Sweep section. (This is the default)
6. Select *rms* in the *Signal Level* section (the default is *peak*).
7. Select *dB20* in the *Modifier* section.

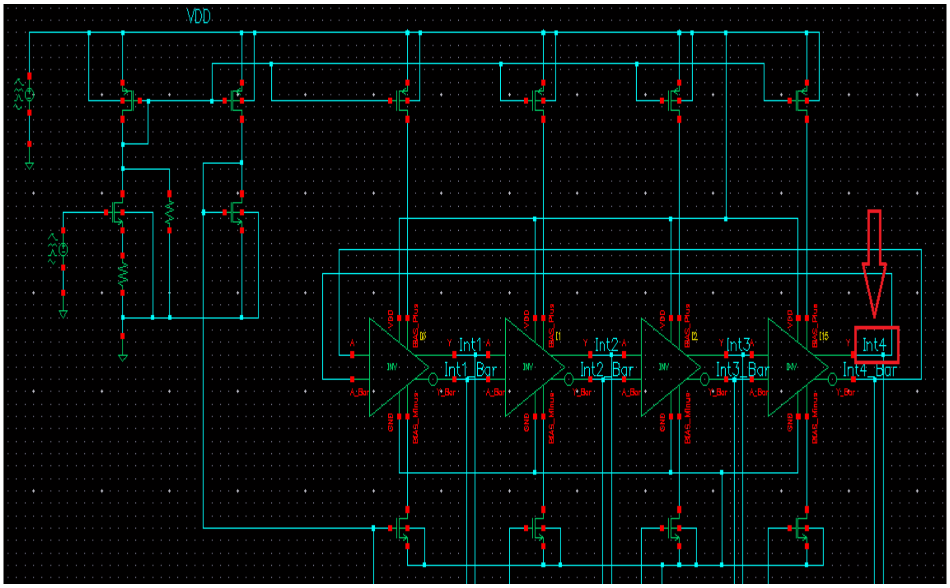
   The *Direct Plot Form* window should like the following:

Figure 3-164 PSS Analysis Direct Plot Form Setup



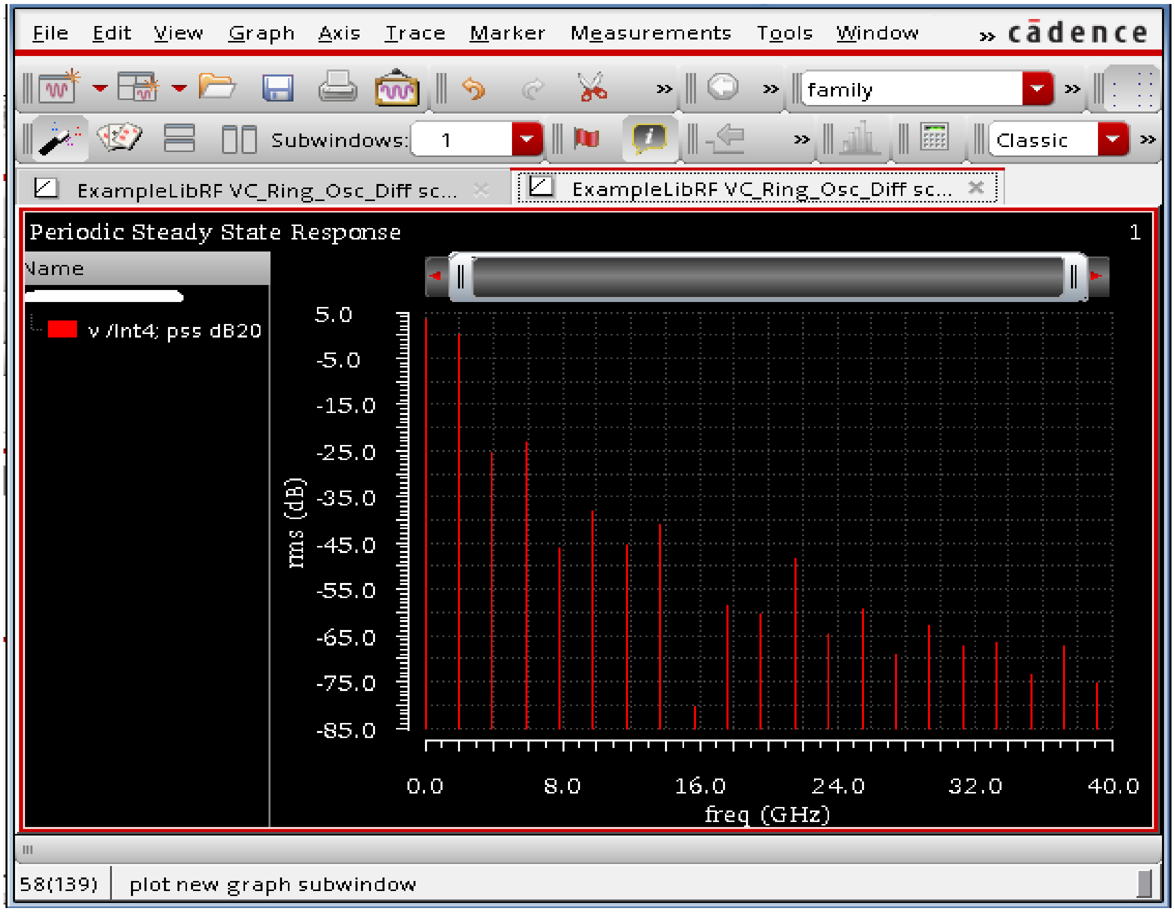
8. Select *Int4* net in the schematic. It is located just below the *Int4* label.

Figure 3-165 Selecting *Int4* net on schematic



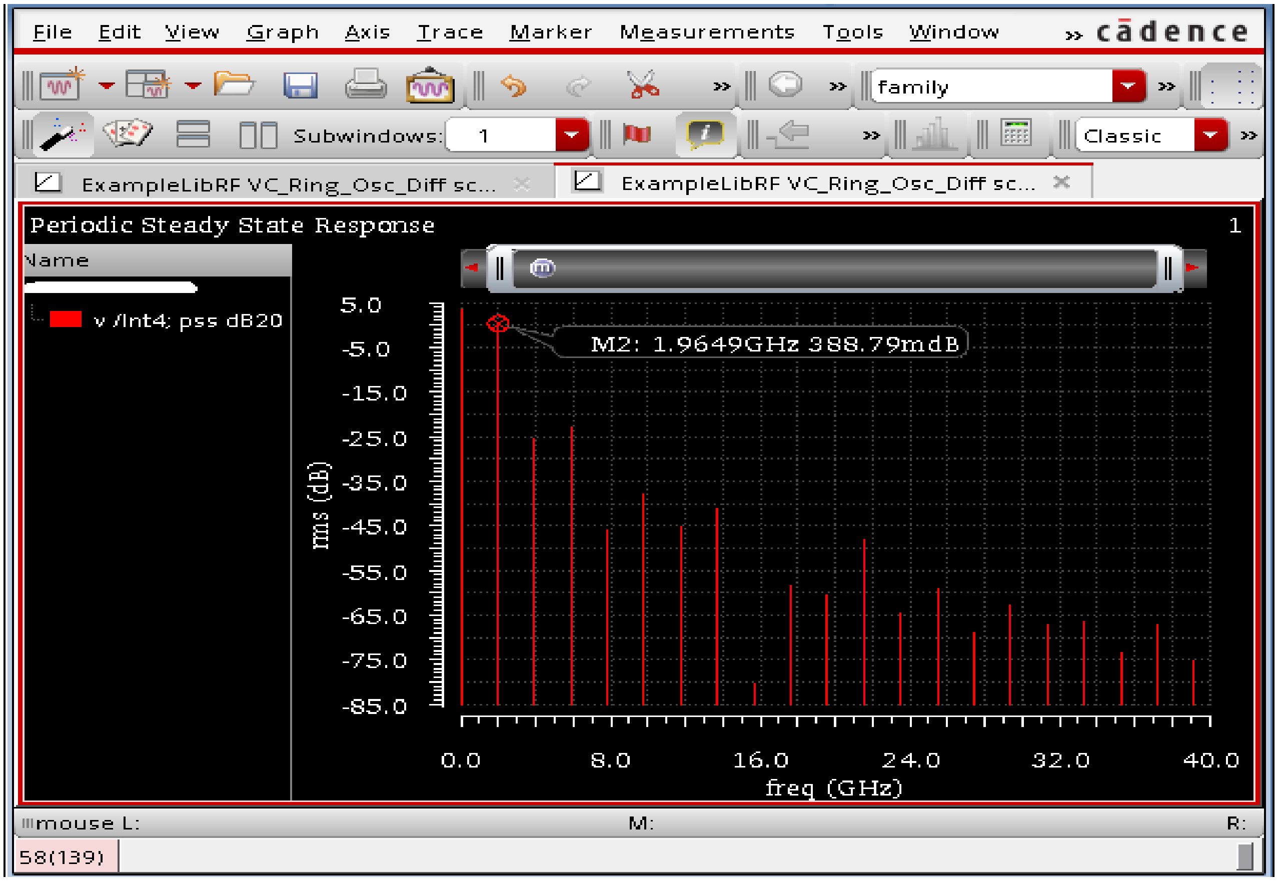
The waveform window is displayed, as shown below.

Figure 3-166 PSS Analysis output Graph Window - Voltage Spectrum Plot



9. In the waveform window, position your cursor near the first harmonic, and press the *m* key. Here *m* is the bindkey to place a trace marker on the graph. The first harmonic is chosen as this is the frequency oscillator is designed for.

Figure 3-167 PSS Analysis output Graph Window - Voltage Spectrum Plot (with marker placed)



10. Note that this frequency is 1.9649GHz. This is the frequency of oscillation.

11. In the *Direct Plot Form*, click *Cancel*. In the waveform window, choose *File - Close All Windows*.

12. Clean up the screen for the next set of measurements.

   a. Close the ADE Explorer by selecting *Session - Quit*.

To summarize, a PSS analysis was set up using the Shooting Method and a simulation was run to determine the oscillation frequency of the oscillator.

Next, you will perform the FM jitter value measurements.

FM Jitter Measurement using PSS Shooting and Pnoise Jitter Analyses

This example computes the periodic steady state solution using the shooting method for the *VC_Ring_Osc_Diff* oscillator circuit. It then runs a periodic small-signal analysis pnoise to determine the FM jitter values of the oscillator. You perform a pss-shooting analysis first because the periodic steady state solution must be determined before you can perform any other periodic small-signal analysis like pnoise, pxf etc. to determine phase noise or transfer function and so on.

Determining FM Jitter

FM jitter calculates a standard averaged phase noise measurement, and also the AM and PM components from the modulated analysis, and adds the ability to integrate the phase noise curve to calculate the cycle jitter or the cycle-to-cycle jitter. The jitter calculations are integrated into the *Direct Plot Form* in ADE Explorer. All the measurements are averaged over the oscillator cycle.

PM Jitter is like the timedomain jitter measurement. You specify a threshold voltage and the timing jitter is calculated. In PM Jitter, a quantitative jitter measurement can be made from the direct plot form.

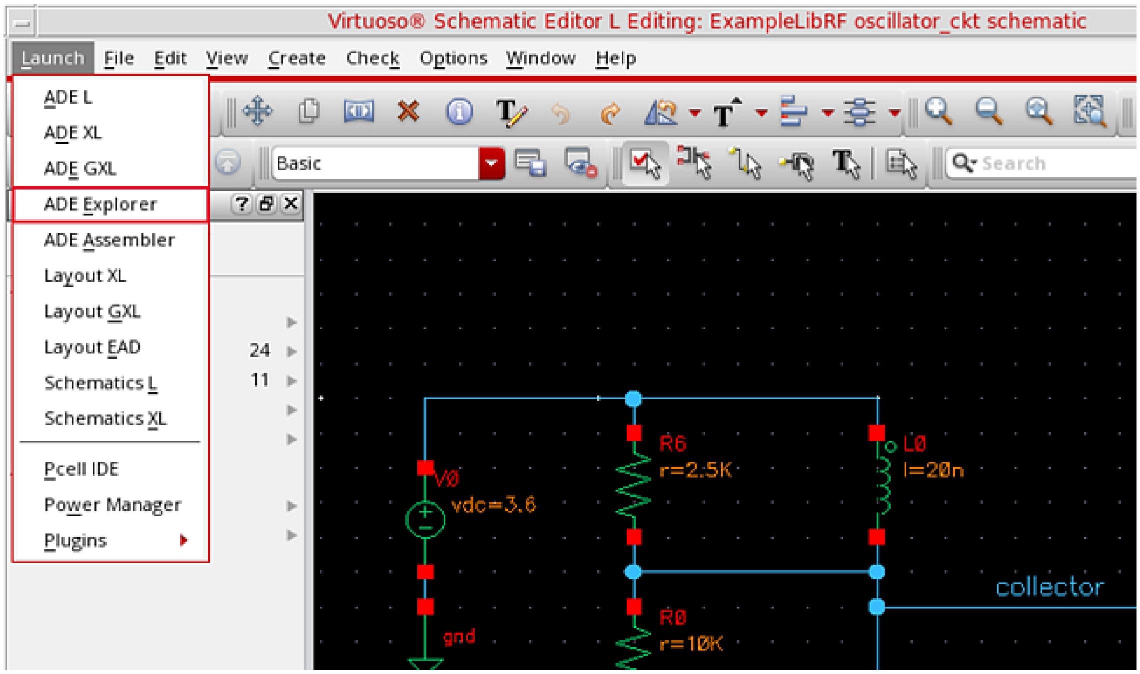
You may also refer to Chapter 3: Frequency Domain Analyses: Harmonic Balance in the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer User Guide for more details on AM and PM Jitter.

In this measurement, you will perform the FM jitter measurement and plot the cycle jitter Jc.

Setting up the PSS Analysis

1. In the Schematic Window, choose *Launch - ADE Explorer.*

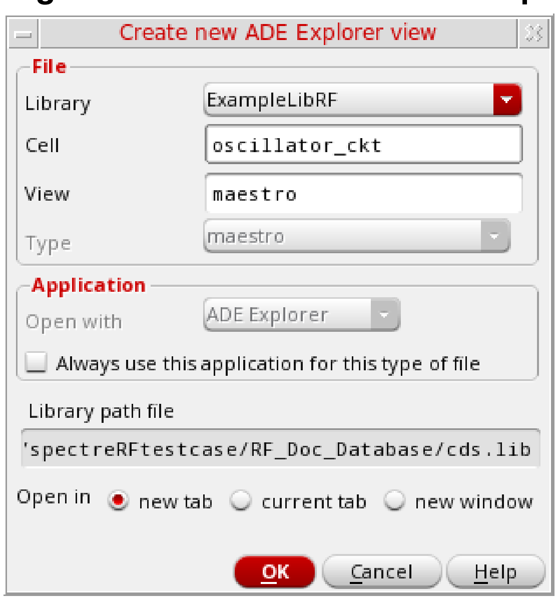
Figure 3-168 Opening ADEL window from VSE window



2. In the *Launch ADE Explorer* dialog, select *Create New View*.

   The *Create new ADE Explorer view* form is displayed.

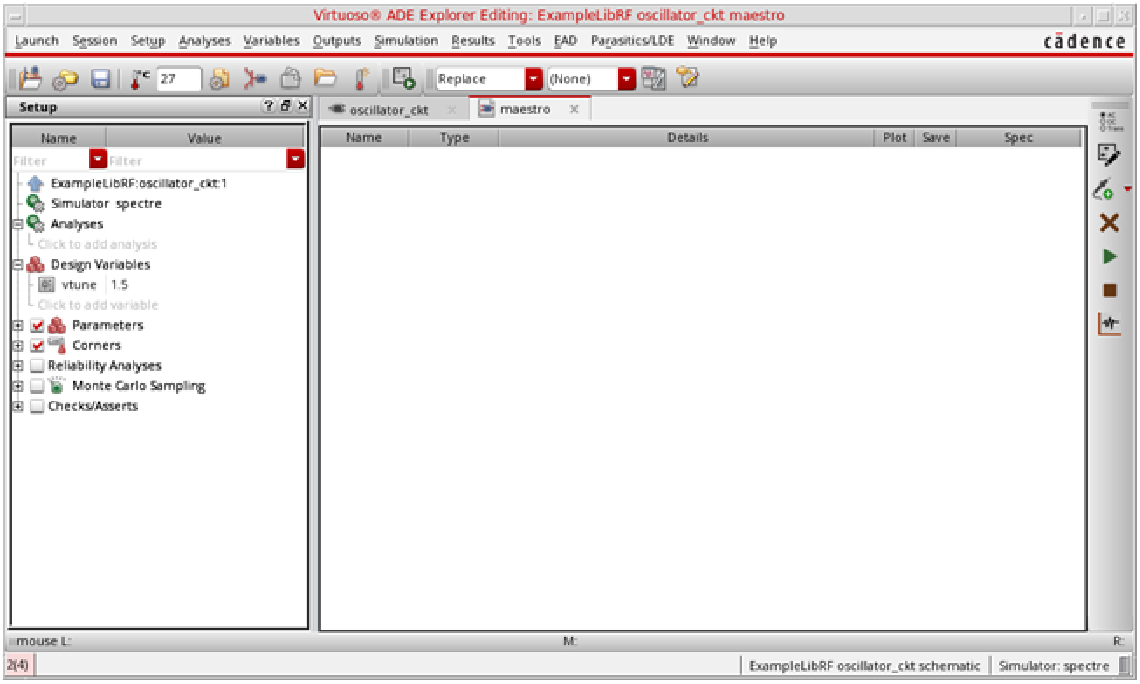
Figure 3-169 Create new ADE Explorer view



3. Leave each option to the default selections and click *OK*.

   ADE Explorer is displayed, as shown below.

Figure 3-170 Virtuoso ADE Explorer Window



4. Select *Setup – Simulator* in ADE Explorer.

   The *Choosing Simulator* form is displayed.

5. Select *spectre* for the *Simulator*.

Figure 3-171 Choosing Simulator Form

6. Click *OK* to close the *Choosing Simulator* form.

7. Set up the High Performance Simulation Options.

   In the ADE Explorer, select *Setup - High Performance Simulation.* The High Performance Simulation Options window is displayed.

   In the *High Performance Simulation Options* window, select *APS*. Note that *Auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 64) and then multi-thread on all the available cores. Usually it is better to specify the number of threads yourself. Small circuits should use a small number of threads, and larger circuits can use more threads. The overhead of managing

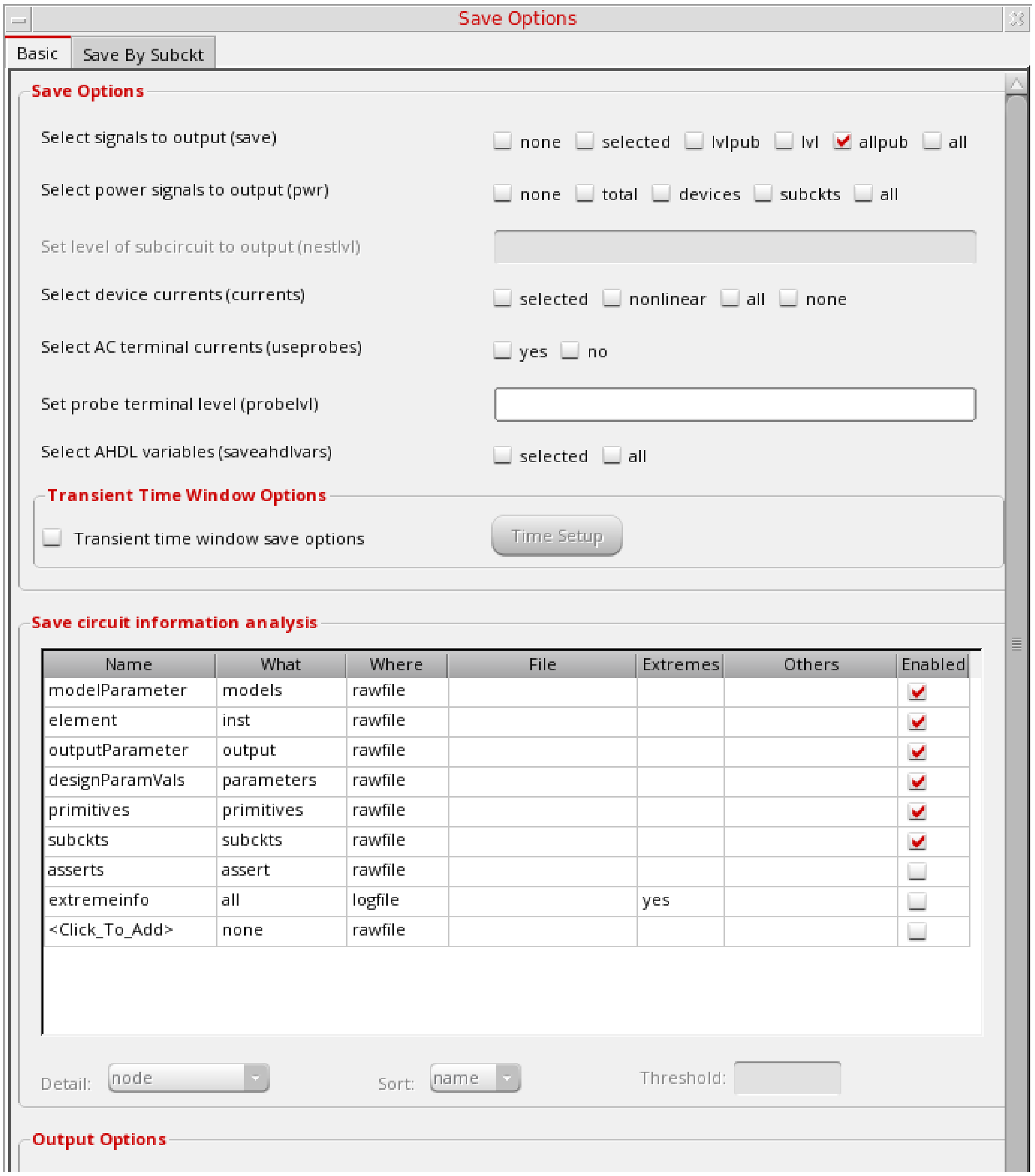
16 threads on a smaller circuit may actually slow the simulation down, compared to two or four threads. Use the number of threads that maximizes the performance.

Figure 3-172 High Performance Simulation Options Form

8. Click *OK* to close the *High Performance Simulation Options* form.
9. Select *Outputs - Save All*.

   The *Save Options* form is displayed.
10. In the *Select signals to output(save)* section, make sure that *allpub* is selected.

Figure 3-173 Save Options Form



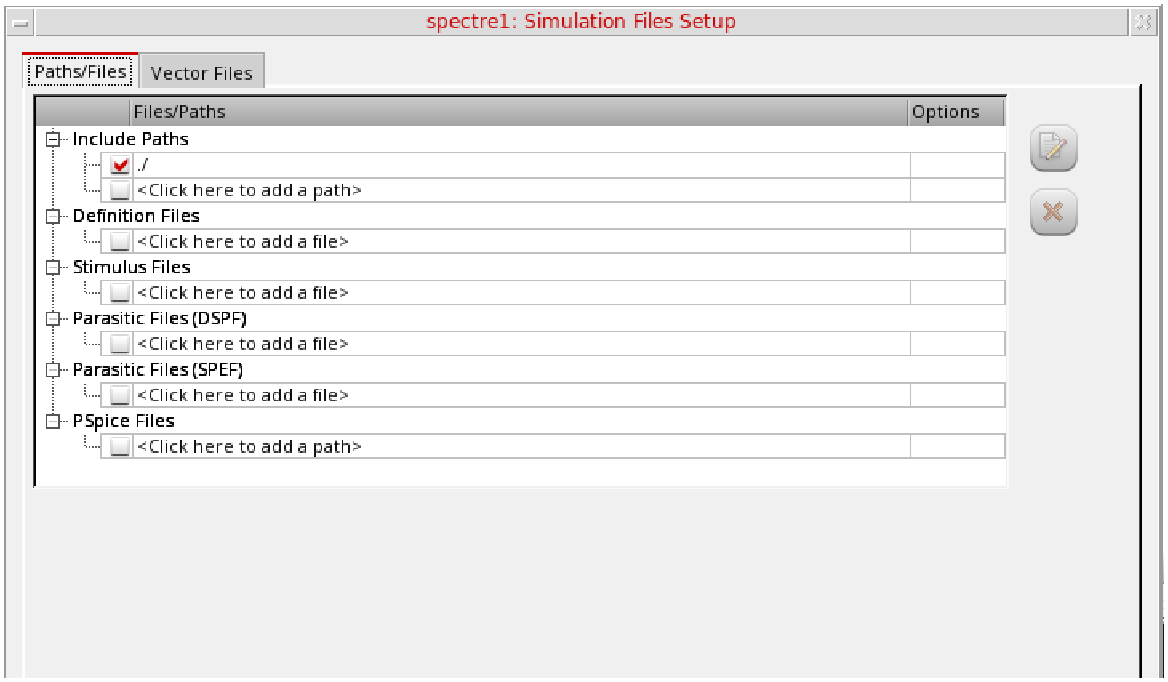
This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

To save the currents, use the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or all if you want to save all the currents in the circuit. When you save currents, more disk space is required for the results file.

11. Click *OK*.

12. Select *Setup - Simulation Files.*
13. In the *Simulation Files Setup* form which gets opened, enter `./` by clicking in the *Include Paths* section. It should look like as shown below:

Figure 3-174 *Simulation Files Setup* Form

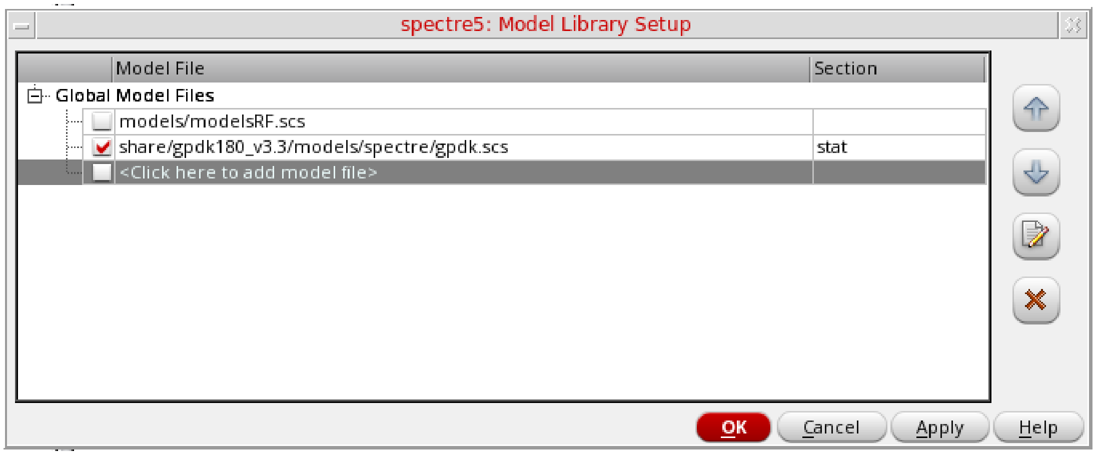


14. Click *OK* to close the *Simulation Files Setup* form.
15. Select *Setup – Model Libraries.*

    The *Model Library Setup* form is displayed.
16. In the *Model File* field, type the path to the model file including the file name, as follows:

    `share/gpdk180_v3.3/models/spectre/gpdk.scs`.

Figure 3-175 *Model Library Setup* Form

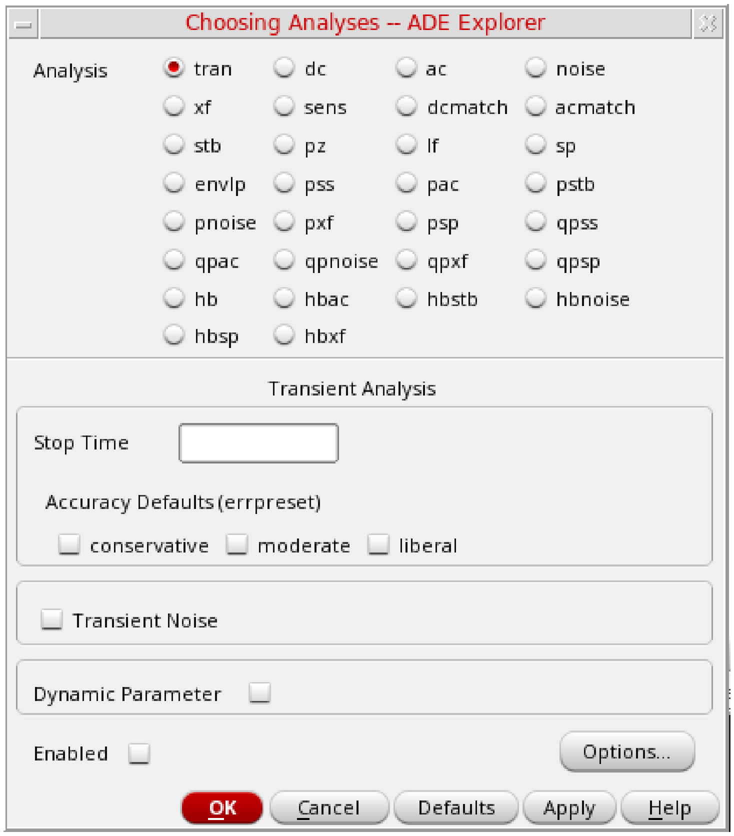


You can also browse to *gpdk.scs* file.

17. Set the *Section* to `stat`.
18. Click *OK* to close the form.
19. In ADE Explorer, select *Analyses - Choose*.

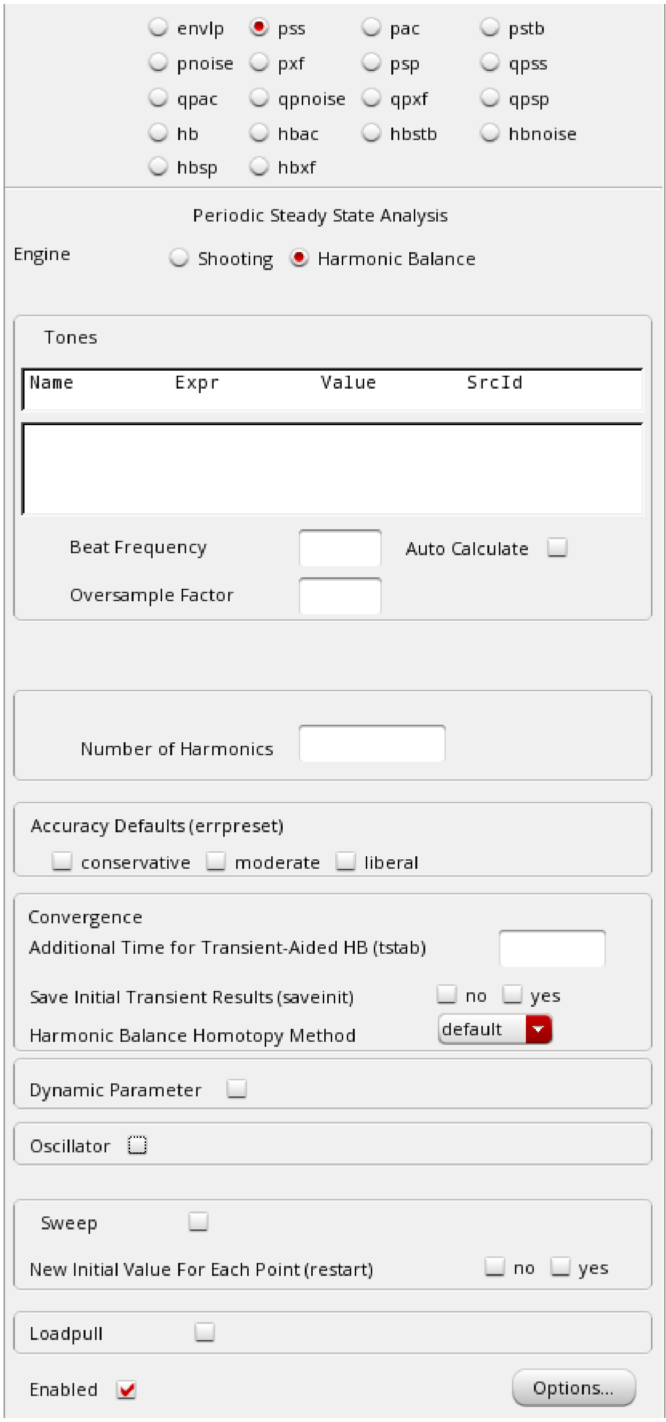
    The *Choosing Analyses* form is displayed, as shown below.

Figure 3-176 The Choosing Analyses Form



20. Select *pss* as *Analysis*. The form expands, as shown below.

Figure 3-177 The Choosing Analyses Form- Setting PSS Analysis



21. In the *Engine* section, verify that *Shooting* is selected (this is the default).

22. In the *Beat Frequency* field, type `1.9G`. The frequency entered here is an approximate frequency of oscillation.

23. In the *Number of harmonics* field, type `20`.

    In general, you want to choose a number that is high enough to capture the nonlinearity of the circuit. Start with 10, and run the simulation. Increase by about 50% to 15 and re-run the simulation. If the harmonics do not change appreciably, then 10 is enough. If they change, raise the number again by about 50%. Use the smallest number of harmonics for the answer to be stable.

24. In the *Accuracy Defaults* (*errpreset*) section, select *conservative*.

    conservative is typically used because very small amplitude phase noise measurements are normally desired. *conservative* is recommended for all the oscillators.

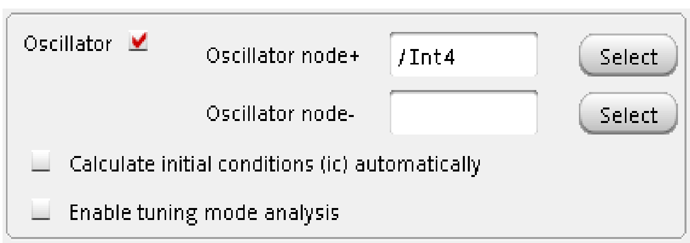
25. Select *yes* for *Run Transient*.

26. Type `2n` in the *Stop time (tstab)* field. *tstab* is typically set to about 2-3 periods of the oscillation frequency for ring oscillator circuits.

27. Select *yes* for *Save Initial Transient Results*. This will help in visualizing the buildup of the oscillation waveform.

28. Select the *Oscillator* option. This is required for simulating an autonomous circuit. The oscillator section expands, as shown below.

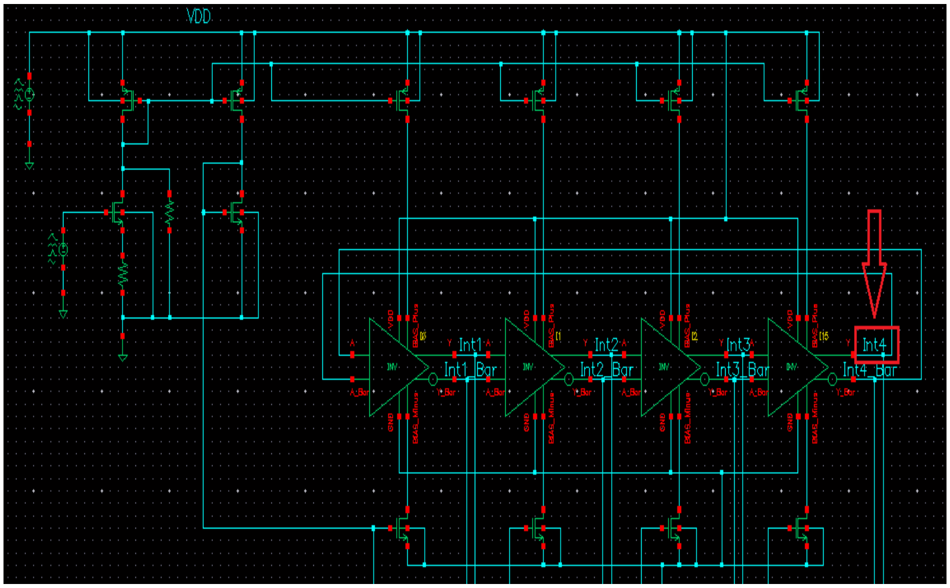
Figure 3-178 The Choosing Analyses Form - Oscillator Section



29. In the *Oscillator node+* field, click *Select* just to the right. In the schematic, select the *Int4* node. This oscillator node will be used by the simulator for the period calculation of the oscillations. It just needs to be a node that has the oscillator signal on it.

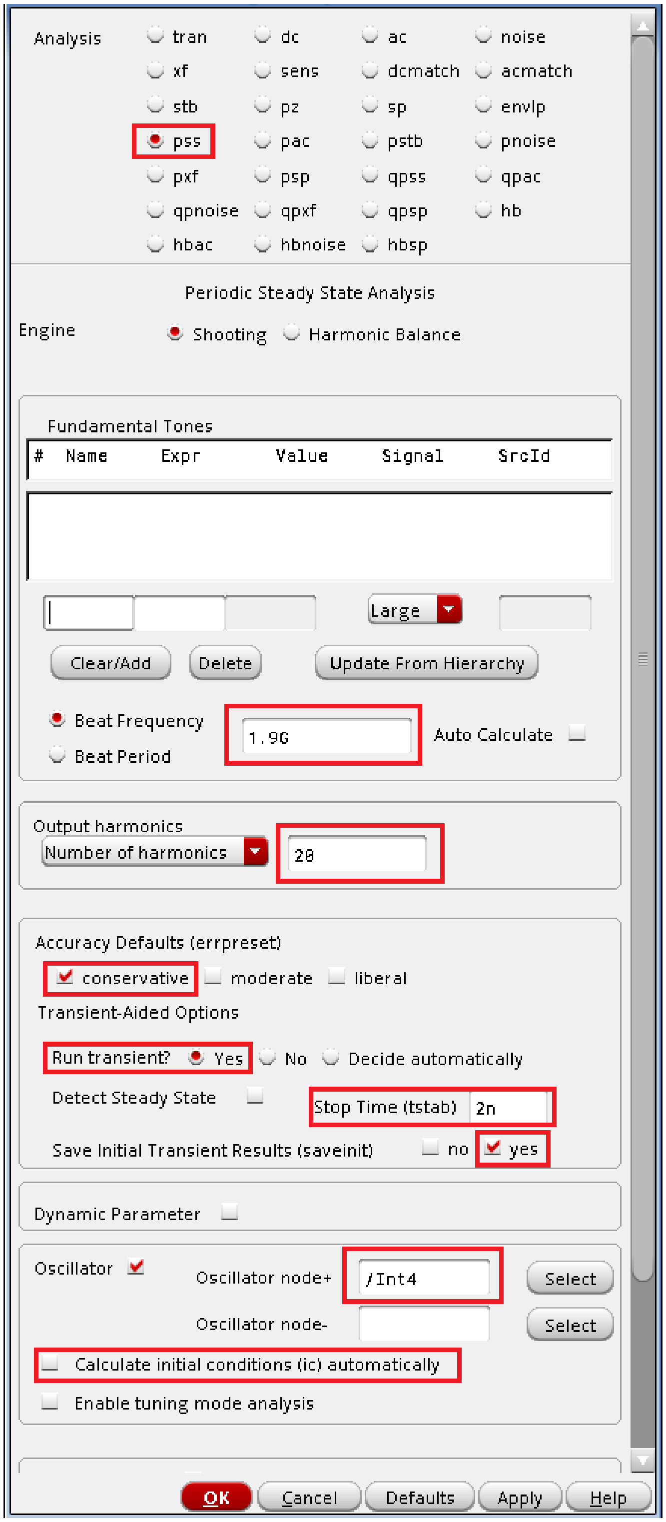
30. Deselect the *Calculate initial conditions (ic) automatically* checkbox. This is because selecting this checkbox only works for feedback oscillators while ring oscillator is not a feedback oscillator. Therefore, selecting this checkbox will not be able to find an oscillatory state for ring oscillator.

Figure 3-179 Selecting *Int4* net on schematic



The *Choosing Analyses* form should looks like the following:

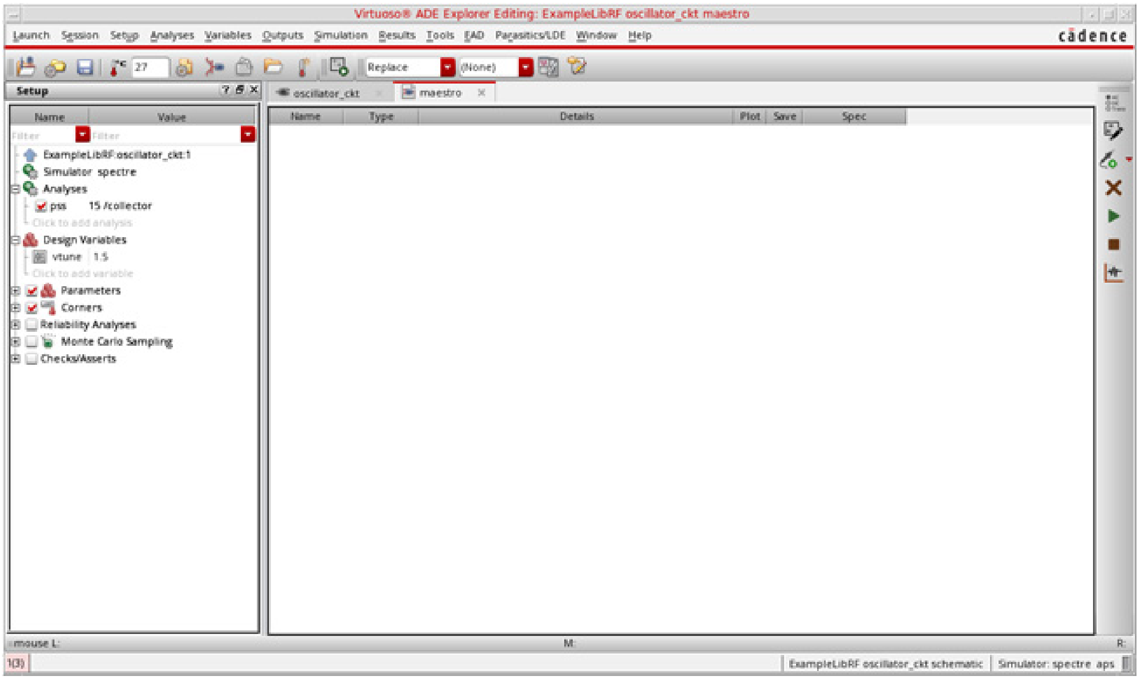
Figure 3-180 Choosing Analyses Form - PSS-Shooting Method Setup



31. Click *Ok*.

This will close the *Choosing Analyses* form. In addition, this will add the *pss* analysis in the *Analyses* section of ADE Explorer, as shown below.

Figure 3-181 ADE Explorer Simulation Window - PSS Analysis



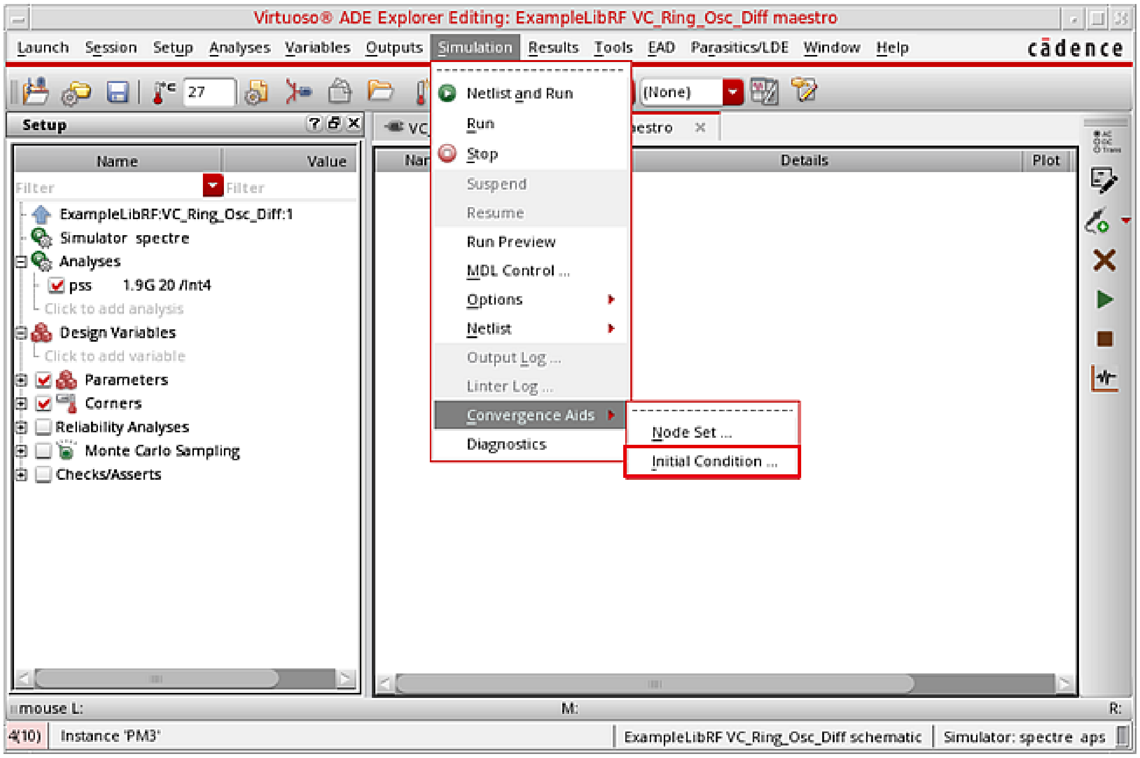
Next, set the initial conditions, which forces the circuit to a specific state at the time zero timepoint, and then removes that force after the time zero point is calculated.

In this case, we will set one stage of the oscillator so that one differential output is high, and one differential output (in the same stage) is low.

This defines all the nodes in the circuit, and the nodes that are connected to the nodes that are forced are pulling as hard as they can to get the forced nodes to change to the other state. When the forcing condition is removed, this action starts the oscillations.

Set the initial conditions by choosing *Simulation - Convergence Aids - Initial Condition* in ADE Explorer, as shown below.

Figure 3-182 PSS Analysis - Setting Initial Condition in ADE Explorer



This will open the *Select Initial Condition Set* dialog box.

32. Type `0` (zero) in the *Node Voltage* field.

33. Select the *Int4_Bar* node in the schematic. Note that the node highlights in the schematic.

34. Click *Apply*.

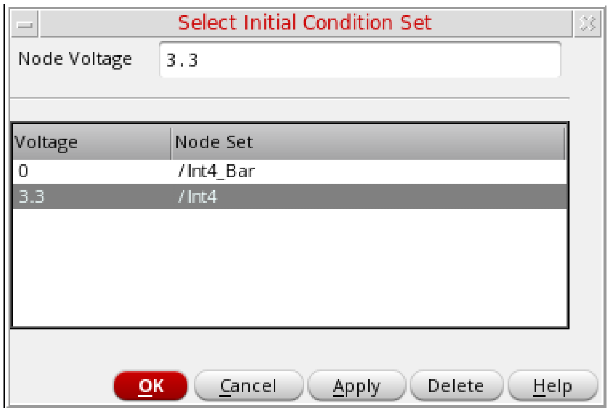
35. Type `3.3` in the *Node Voltage* field.

36. Select the *Int4* node in the schematic. Note that the node highlights in the schematic.

37. Click *OK*.

The populated *Select Initial Condition Set* dialog box would look like the following:

Figure 3-183 Setting Initial Condition Form



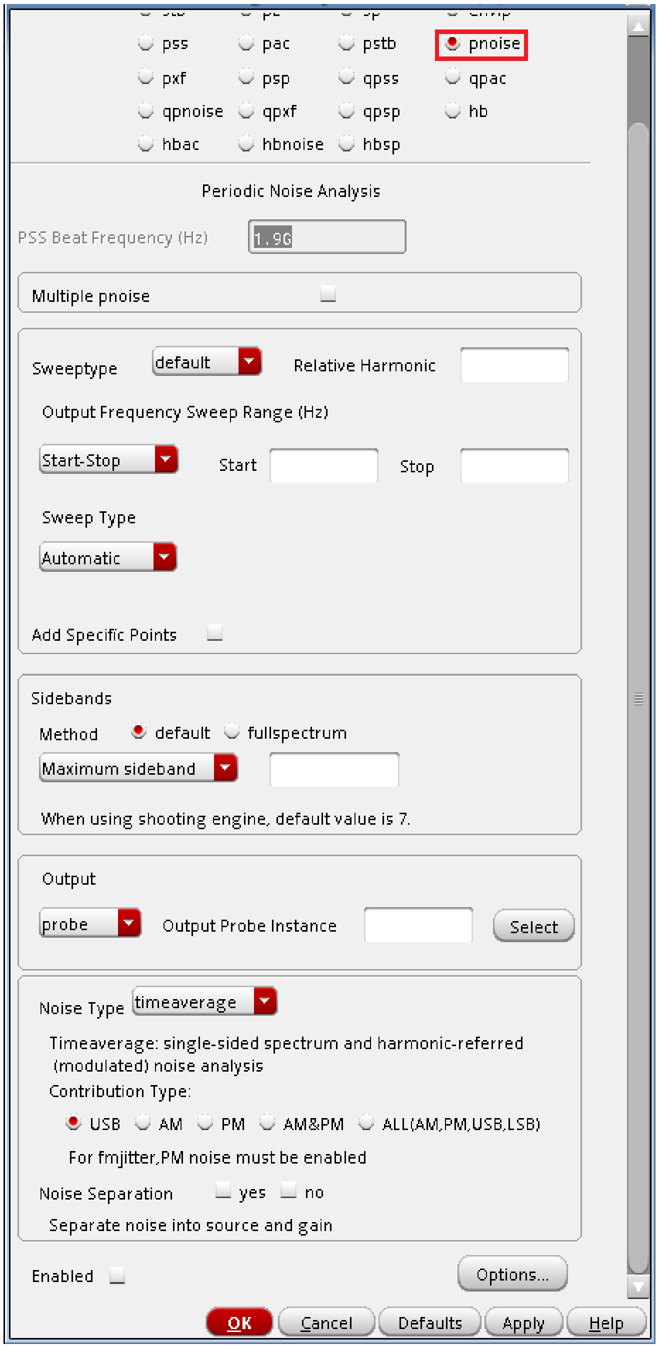
This finishes the setting of PSS Analysis with setting up of Initial Conditions.

Setting up the Pnoise analysis

This analysis is set to do the phase noise measurement. It is run after *pss* analysis.

1. In the *Choosing Analyses* form, select *pnoise*. The form expands, as shown below.

Figure 3-184 The Choosing Analyses Form - *pnoise* Analysis Setup



2. Set the *Sweep Type* to *relative*.

For oscillators, the *hbnoise/pnoise* frequency range is *relative*. Specify the harmonic number as appropriate for the system you are simulating. If you are simulating an oscillator by itself, then the harmonic number is likely to be 1. If you have an oscillator and a diode frequency doubler, then the harmonic number is likely to be 2. If you have an oscillator with a frequency divider, in the hb/pss *Choosing Analyses* form, you should specify the approximate frequency of oscillation for the frequency-divided signal. In the *hbnoise/pnoise Choosing Analyses* form, if the noise is desired on the frequency divided output, then the relative harmonic is 1. If the noise is desired at the output of the oscillator, the relative harmonic number is the divide ratio. The meaning of *relative* is to take the frequency of the harmonic number specified and add to it the frequencies specified in the *Choosing Analyses* form. If the oscillator had a 1GHz output, and the pnoise had 1M relative to the first harmonic specified, the actual output frequency is 1G + 1M, or 1001M.

a. Type `1` in the *Relative Harmonic* field as you are simulating an oscillator by itself.

   Next, you will set the output frequency sweep range. Frequency sweep is set for a noise simulation as the noise is spread over the frequency range. Oscillator phase noise adds phase uncertainty for phase modulated signals. Note that BPSK, QPSK, and all QAM signals have phase information in the constellation. If the LO signal has phase noise on it, it makes it harder to demodulate the signals because there is more spread in the phase of the received signal.Therefore, it is critical to determine its behavior over a frequency range.

b. In the *Output Frequency Sweep Range (Hz)* section, type `10K` in the *Start* Field.

c. Type `1G` in the *Stop* Field.

   Set the frequency sweep range as appropriate for your circuit (or application).

d. Set the *Sweep Type* to *Logarithmic*.

e. Type `4` in the *Points Per Decade* field. Typically, 3 to 5 points per decade are a reasonable number to capture the noise behavior of the circuit.

3. In the Sidebands section, choose *fullspectrum* as the *Method*.

4. Leave the *Maximum Sideband* field blank. Since *fullspectrum* is set, and the oscillator frequency is well above 100KHz, the *Maximum sideband* field should be left blank.

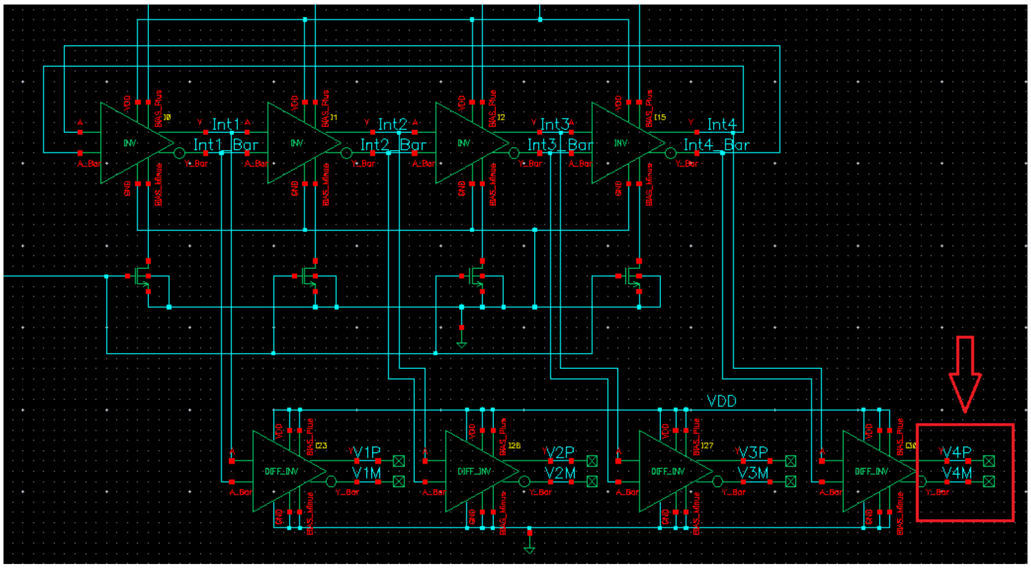
5. Set the *Output* to *voltage*.

   a. Type `/V4P` in the *Positive Output Node* field. You can also select *V4P* net from schematic by clicking *Select* on the right of the *Positive Output Node* field and then selecting the net just below the *out* label in schematic.

   b. Type `/V4M` in the *Negative Output Node* field. If the second node, that is, the *Negative Output Node* is left blank, it will be connected to the global ground node automatically. However, if you have differential oscillator, you need to specify both the nodes.

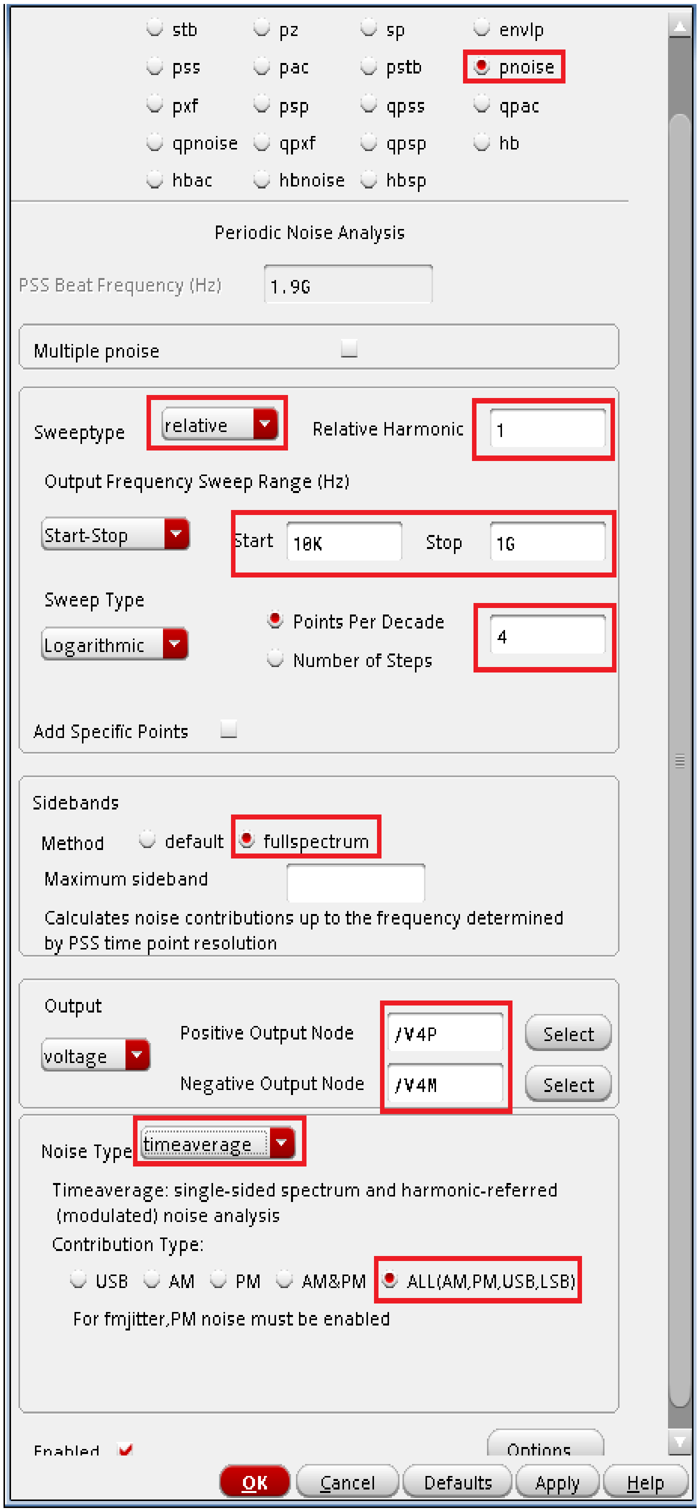
6. Select *timeaverage* from the *Noise Type* drop down list.
7. Select the *ALL(AM,PM,USB,LSB)* option.

Figure 3-185 Selecting *V4P and V4M* net from the schematic



The *Choosing Analyses* form should look like the following:

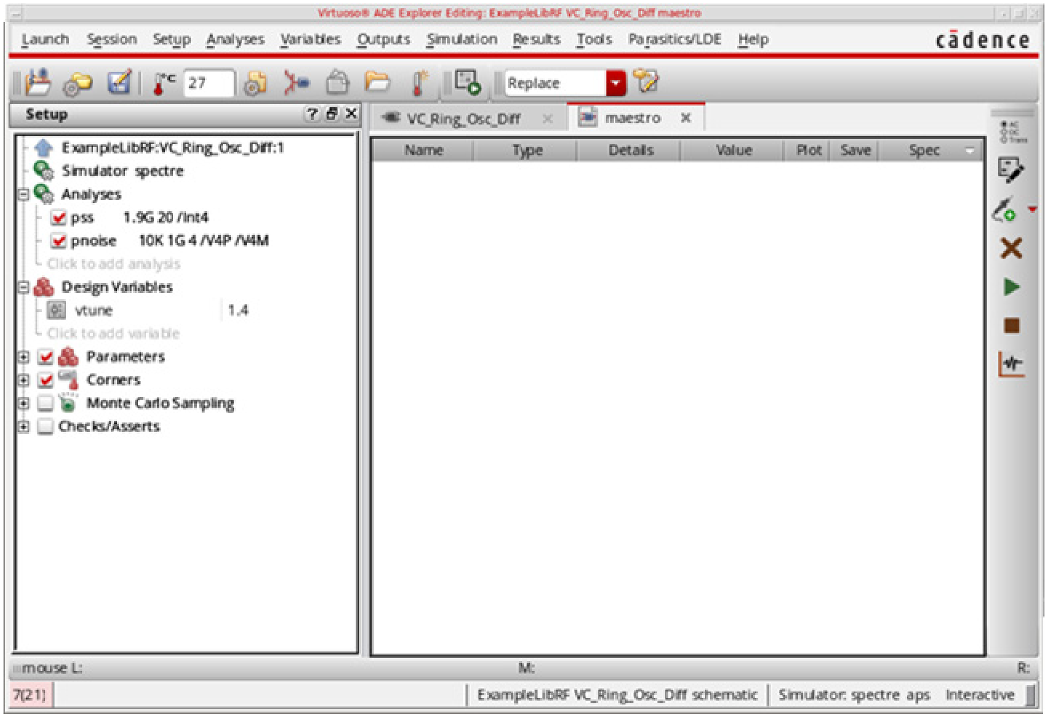
Figure 3-186 Choosing Analysis Form - *pnoise* Analysis Setup



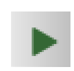
8. Click *OK* to close the *Choosing Analyses* form.

This will add the *pnoise* analysis along with *pss* analysis in the *Analyses* section of ADE Explorer, as shown below:

Figure 3-187 ADE Explorer Simulation Window - *pss* and *pnoise-jitter* analysis setup



Running the PSS and Pnoise analysis

Once finished setting up the PSS and Pnoise Analyses click the green icon  on the right of ADE Explorer or on the Schematic window to run the simulation.

This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` logfile). When the analysis has completed, you may iconify the status window.

Next, you will plot the results.

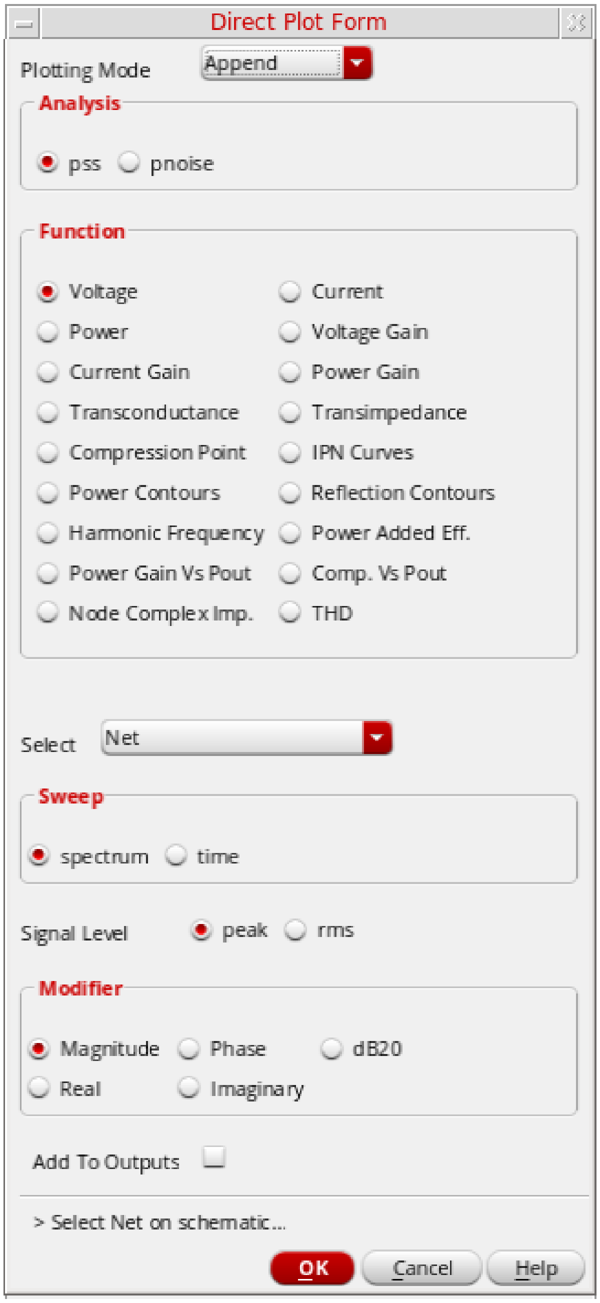
Plotting the results

First Plot the oscillator phase noise based on *pnoise* analysis -

1. In ADE Explorer, select *Results - Direct Plot - Main Form*.

   The *Direct Plot Form* is displayed, as shown below.

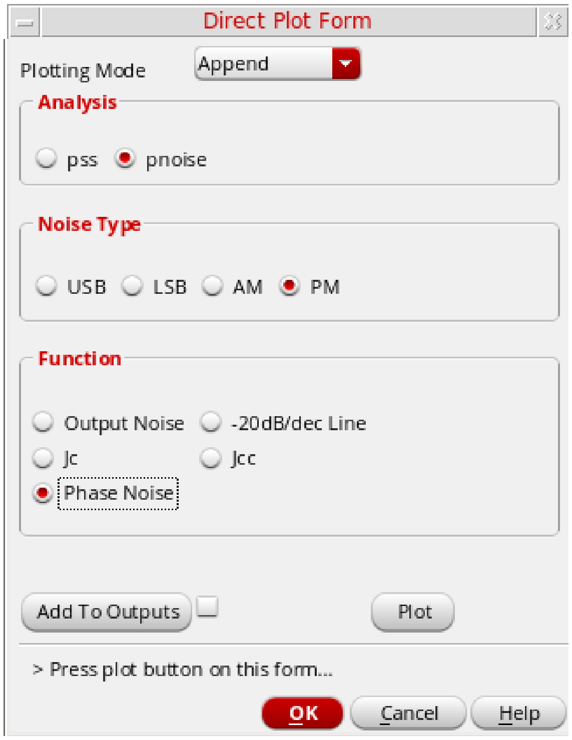
Figure 3-188 ***pss*** **and** ***pnoise*** **Analysis Direct Plot Form**



2. In the *Direct Plot Form*, select *pnoise* in the *Analysis* section. The form changes.
3. Select *PM* in the *Type* section.
4. Select *Phase Noise* in the *Function* section.

   The *Direct Plot Form* Setup should look like the following:

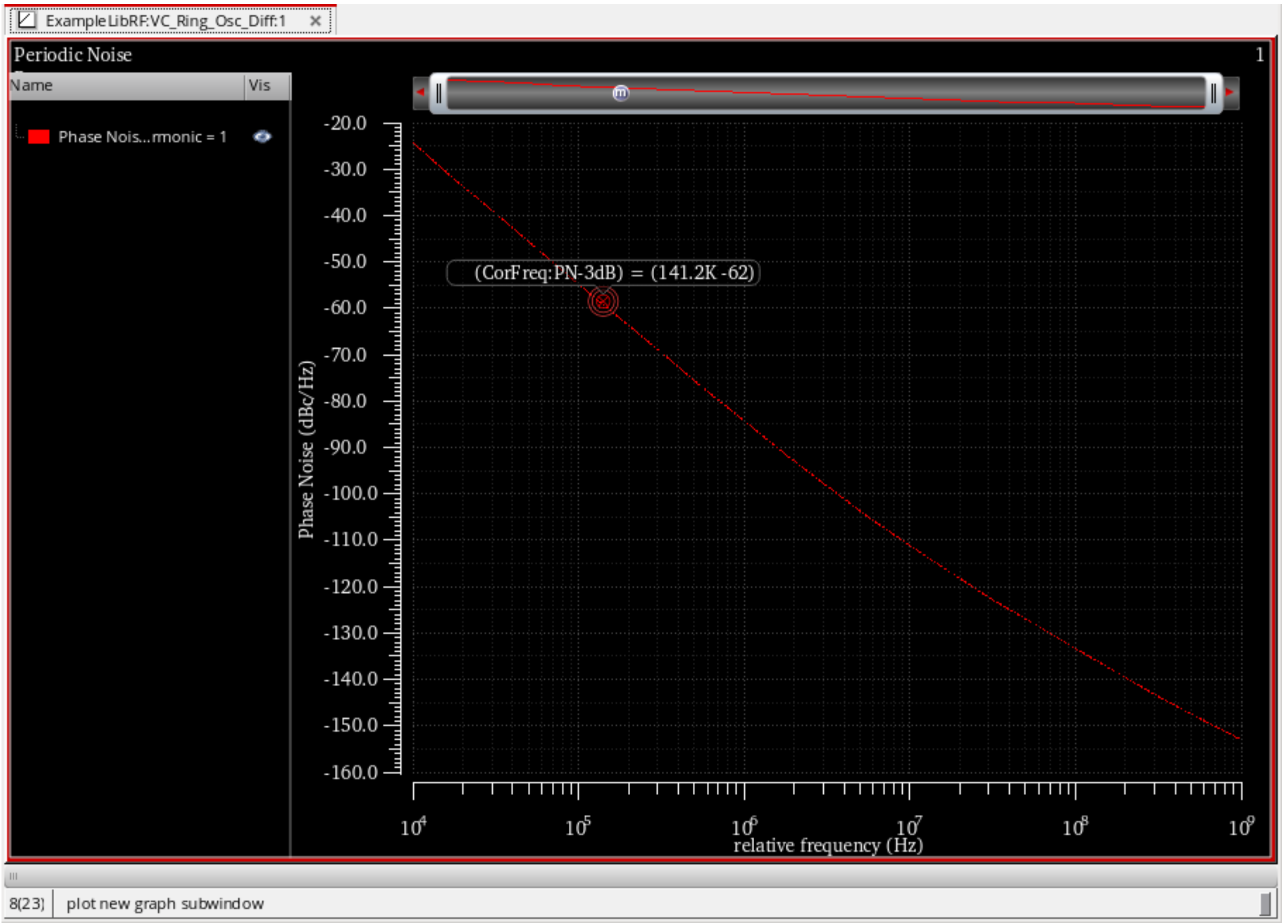
Figure 3-189 *pss* and pnoise FM jitter Analysis Direct Plot Form Setup



5. Click *Plot*.

The oscillator phase noise curve is plotted. This is a Single Sideband (SSB) Plot.

Figure 3-190 Phase Noise (SSB) plot from pnoise Analysis



Next, plot the Jc (cycle or period) jitter value.

1. In the *Direct Plot Form*, select *Jc* in the *Function* section.

   The form expands. Jc is cycle or period jitter while Jcc is cycle-to-cycle or period-to-period jitter.

2. Leave *Number of Cycles (k)* as `1` (which is set by default).

   Number of cycles determines whether one period or k-periods jitter will be computed. In this example, you are going to determine only one cycle Jc (cycle) jitter.

3. Leave *Signal Level* as *rms* (this is selected by default). *rms* jitter is typically used.

4. In the *Modifier* section, leave the default selection as *Second*.

   Here, the other options are *UI* which is Unit interval, and *ppm* which is parts per million. Select the *Modifier* value based on the units you are using in your design specifications.

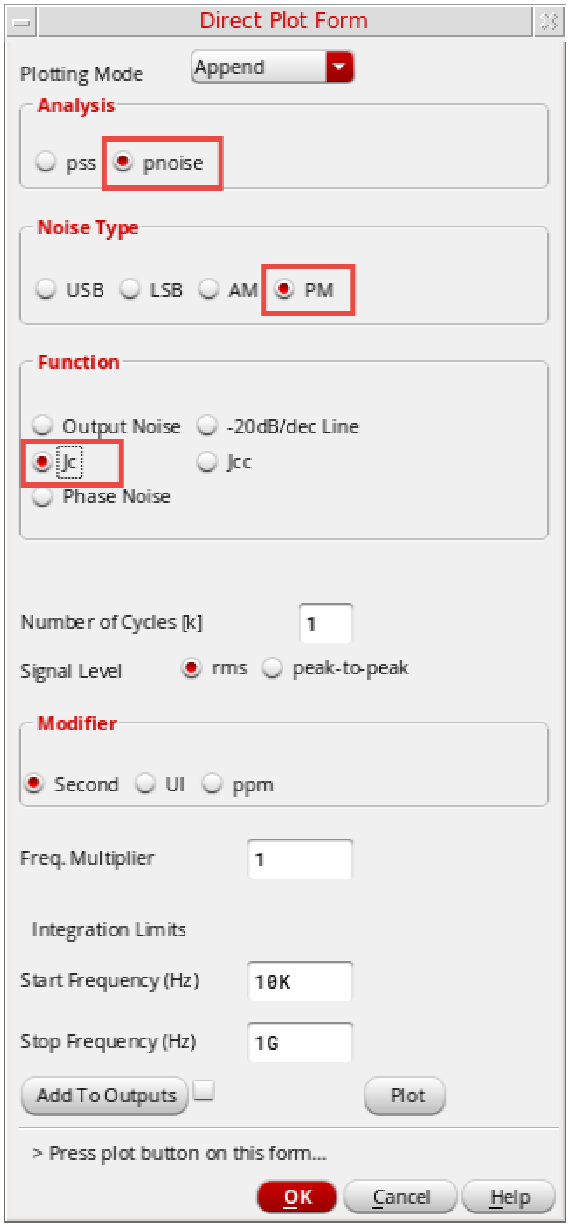
5. Leave the *Freq. Multiplier* set to `1` (which is the default value).

   Use the *Freq. Multiplier* parameter when the selected output is not at the PSS fundamental/beat frequency.

6. Fill in the *Start Frequency (Hz)* and *Stop Frequency (Hz)* fields. These are the limits of integration to determine jitter. The appropriate values are included either in the jitter specifications or in the verification methodology. The best values to use also depend on the circuit properties. For a small *Number of Cycles [k]*, such as a short observation time of one or a few periods, the lower frequency noise does not contribute much to either Jc or Jcc jitter. For a larger *Number of Cycles [k]*, the lower frequency noise is important.

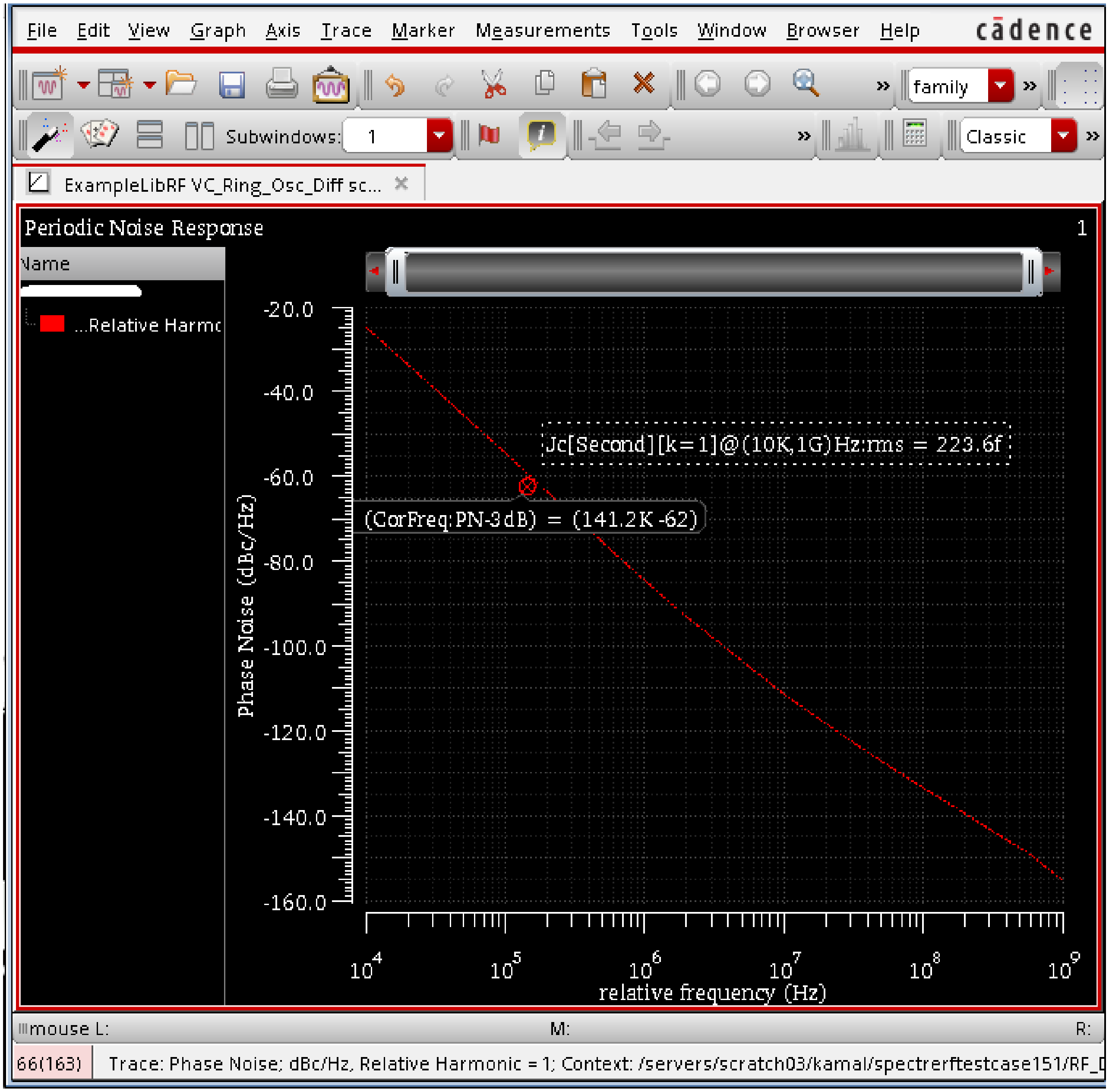
   Refer to the *Jitter Measurements Using SpectreRF* Application Note for more information on how to set up the lower limit of the integration.

Figure 3-191 Direct Plot pnoise jitter Jc Setup



a. Click *Plot*. This will calculate the Jc (cycle jitter) value and the value of Jc will appear on the plot as a label, as shown below.

Figure 3-192 Jc value as label on Phase Noise Plot



Since the Jc value appears as a label on the plot, it is important that there is a plot window already open when the Jc value is plotted/calculated.

To summarize, PSS analysis using Shooting Engine and Pnoise was setup and simulation was run to determine the FM jitter of the oscillator.

Next you will determine the tuning range and phase noise of the Ring Oscillator.

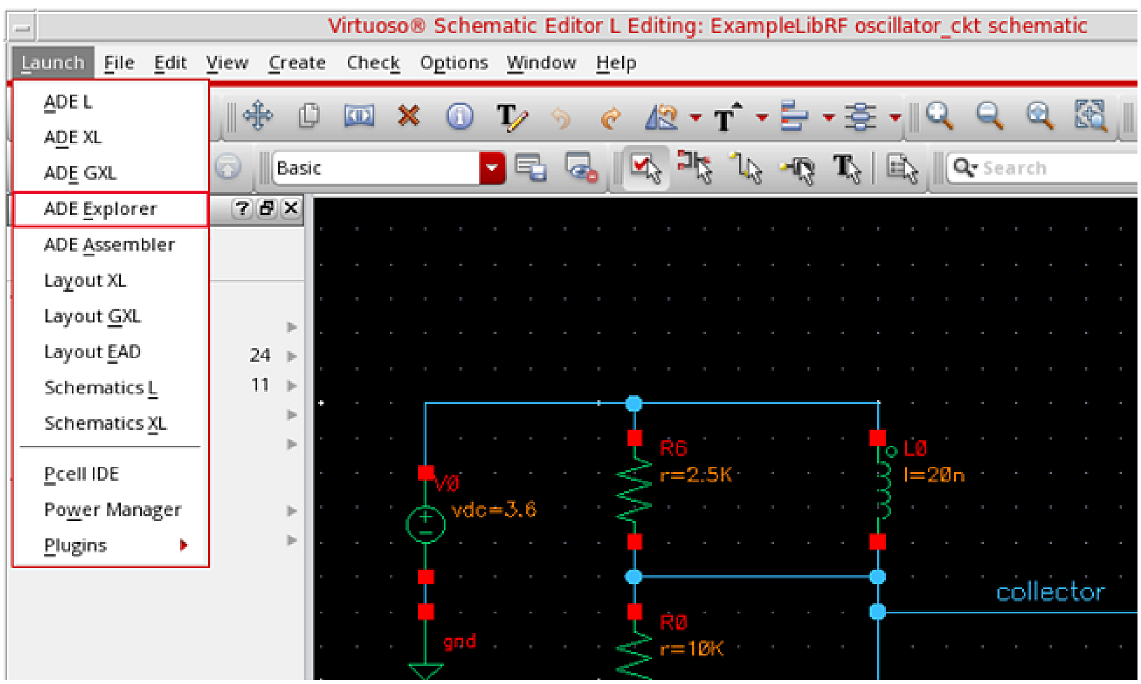
Calculating the Swept Tuning Range and Phase Noise for the Ring Oscillator

This example computes the swept periodic steady state solution using the shooting method for the *VC_Ring_Osc_Diff* oscillator circuit. You will sweep the tuning voltage to determine its effect on oscillation frequency. It then runs a periodic small-signal analysis pnoise to determine the phase noise of the oscillator. You perform a PSS-Shooting analysis first because the periodic steady state solution must be determined before you can perform any other periodic small-signal analysis, such as pnoise pxf to determine phase noise or transfer function and so on.

Setting up the PSS Analysis

1. In the Schematic Window, choose *Launch - ADE Explorer.*

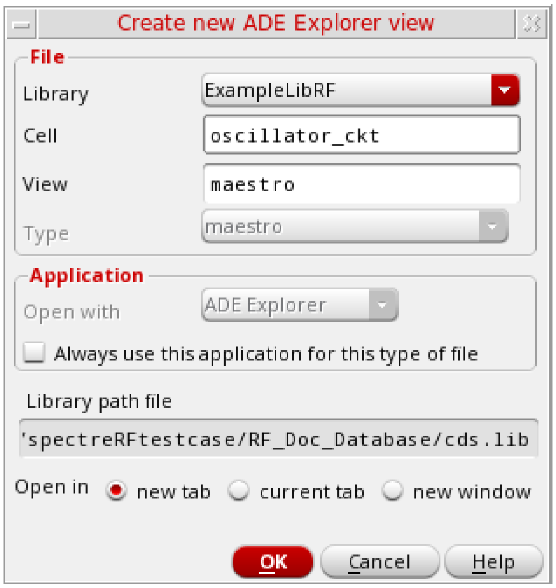
Figure 3-193 Opening ADE Explorer window from VSE window



2. In the *Launch ADE Explorer* dialog, select *Create New View*.

   The *Create new ADE Explorer view* form is displayed.

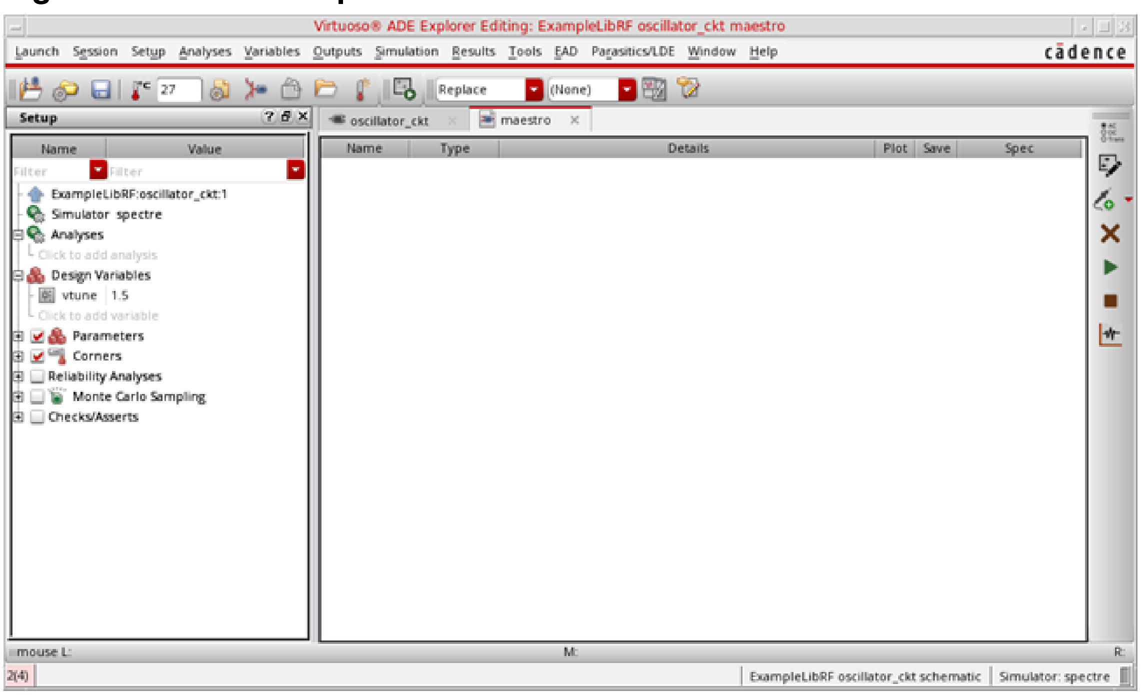
Figure 3-194 Create new ADE Explorer view



3. Leave each option to the default selections and click *OK*.

   ADE Explorer is displayed, as shown below.

Figure 3-195 ADE Explorer Window



4. Select *Setup – Simulator* in ADE Explorer.

The *Choosing Simulator* form is displayed.

5. Select *spectre* as the *Simulator*.

Figure 3-196 Choosing Simulator Form

6. Click *OK* to close the *Choosing Simulator* form.

7. Set up the High Performance Simulation Options, as follows:

   In ADE Explorer, select *Setup - High Performance Simulation*. The *High Performance Simulation Options* window is displayed.

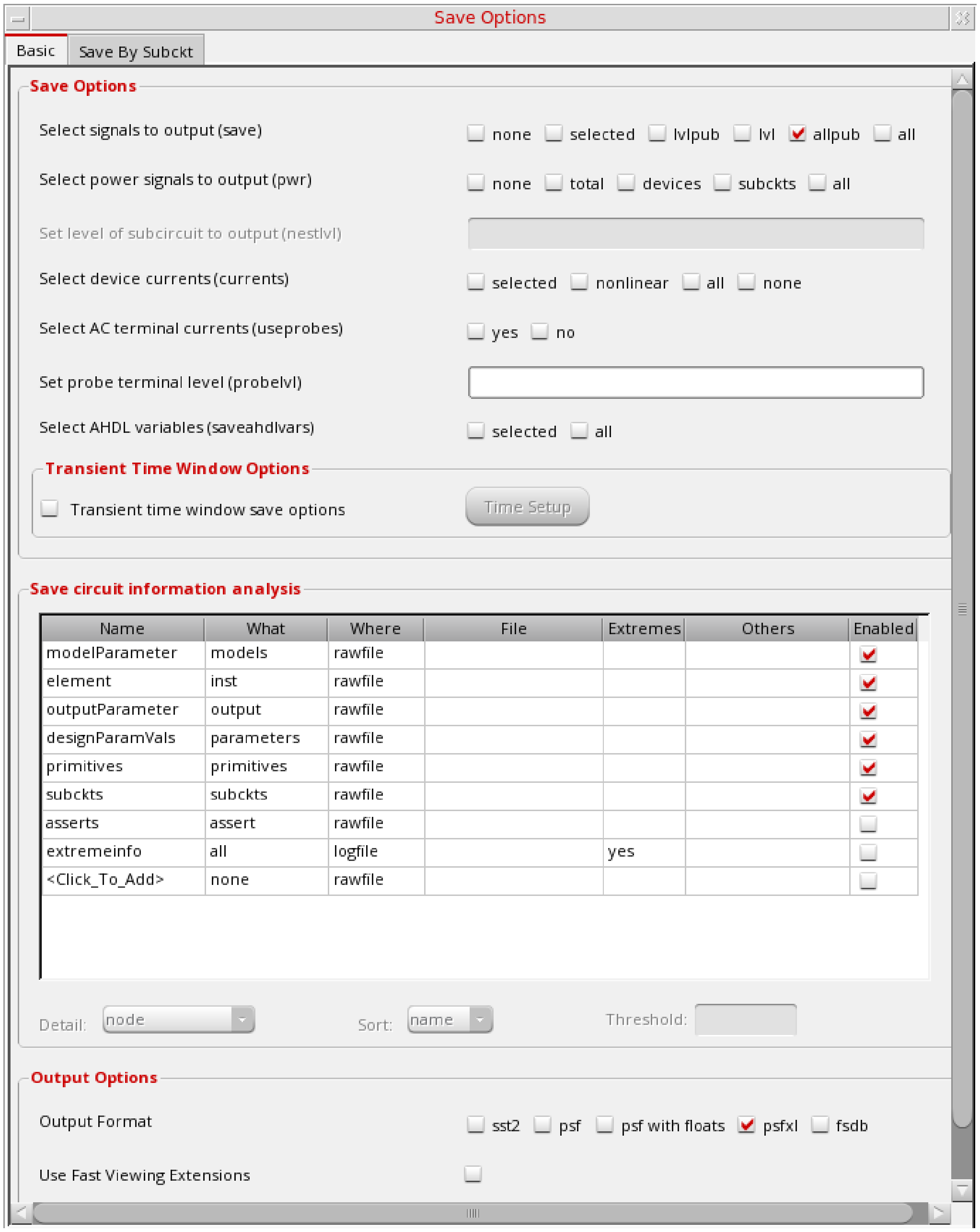
   In the *High Performance Simulation Options* window, select *APS*. Note that *Auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 64) and then multi-thread on all the available cores. Usually it is better to specify the number of threads yourself. Small circuits should use a small number of threads, and larger circuits can use more threads. The overhead of managing 16 threads on a smaller circuit may actually slow the simulation down, compared to two or four threads. Use the number of threads that maximizes the performance.

Figure 3-197 High Performance Simulation Options Form

8. Click *OK* to close the *High Performance Simulation Options* form.
9. Select *Outputs - Save All*.

   The *Save Options* form is displayed.
10. In the *Select signals to output(save)* section, make sure that *allpub* is selected.

Figure 3-198 Save Options Form



This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

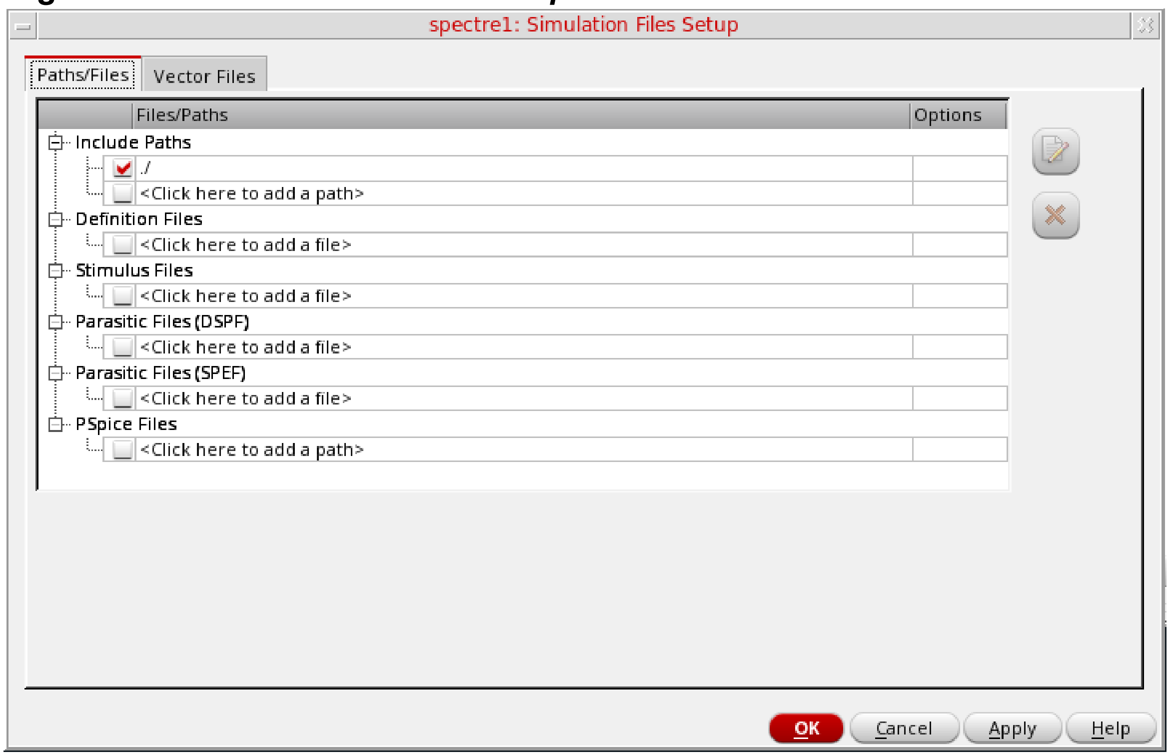
To save the currents, use the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or select *all* if you want to save all the currents in the circuit. When you save currents, you require more disk space for the results file.

11. Click *OK*.

12. Select *Setup - Simulation Files.*

13. In the *Simulation Files Setup* form which opens, enter `./` by clicking in the *Include Paths* section. The *Simulation Files Setup* form should look like the following:

Figure 3-199 *Simulation Files Setup* Form



14. Click *OK* to close the *Simulation Files Setup* form.

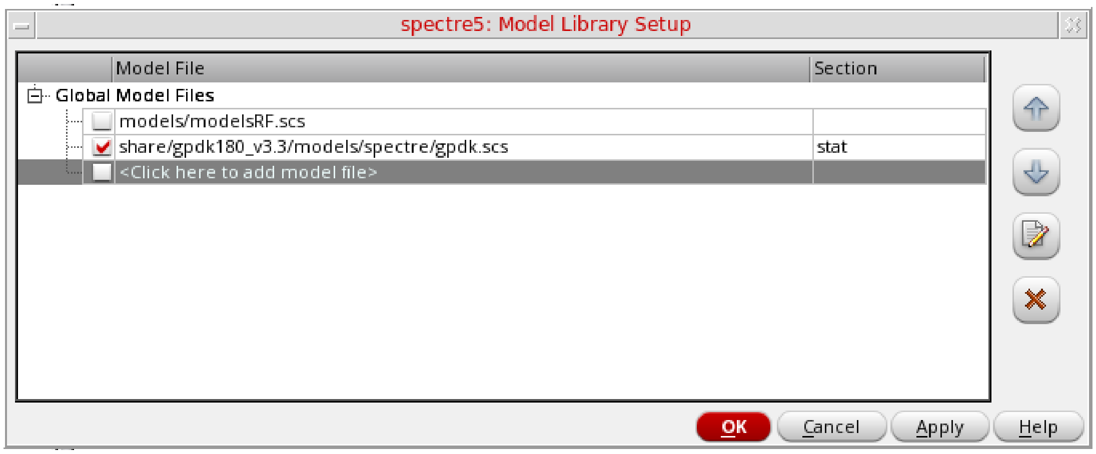
15. Select *Setup – Model Libraries*.

The *Model Library Setup* form is displayed.

16. In the *Model File* field, type the following path to the model file including the file name:

`share/gpdk180_v3.3/models/spectre/gpdk.scs.`

Figure 3-200 Model Library Setup Form



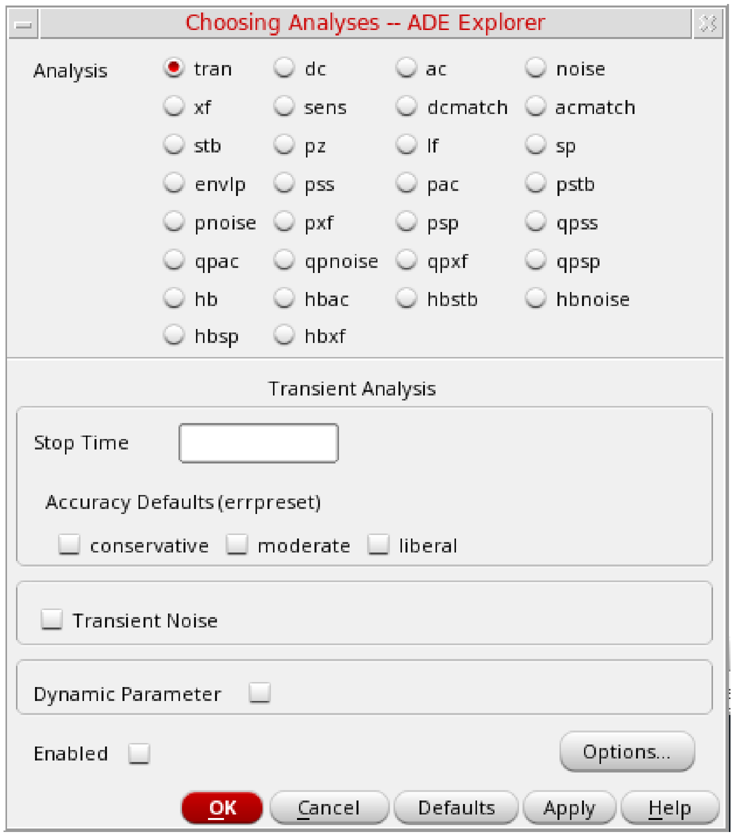
You can also browse to *gpdk.scs* file. Set the Section to *stat*.

17. Click *OK* to close the *Model Library Setup* form.

18. Select *Analyses - Choose*.

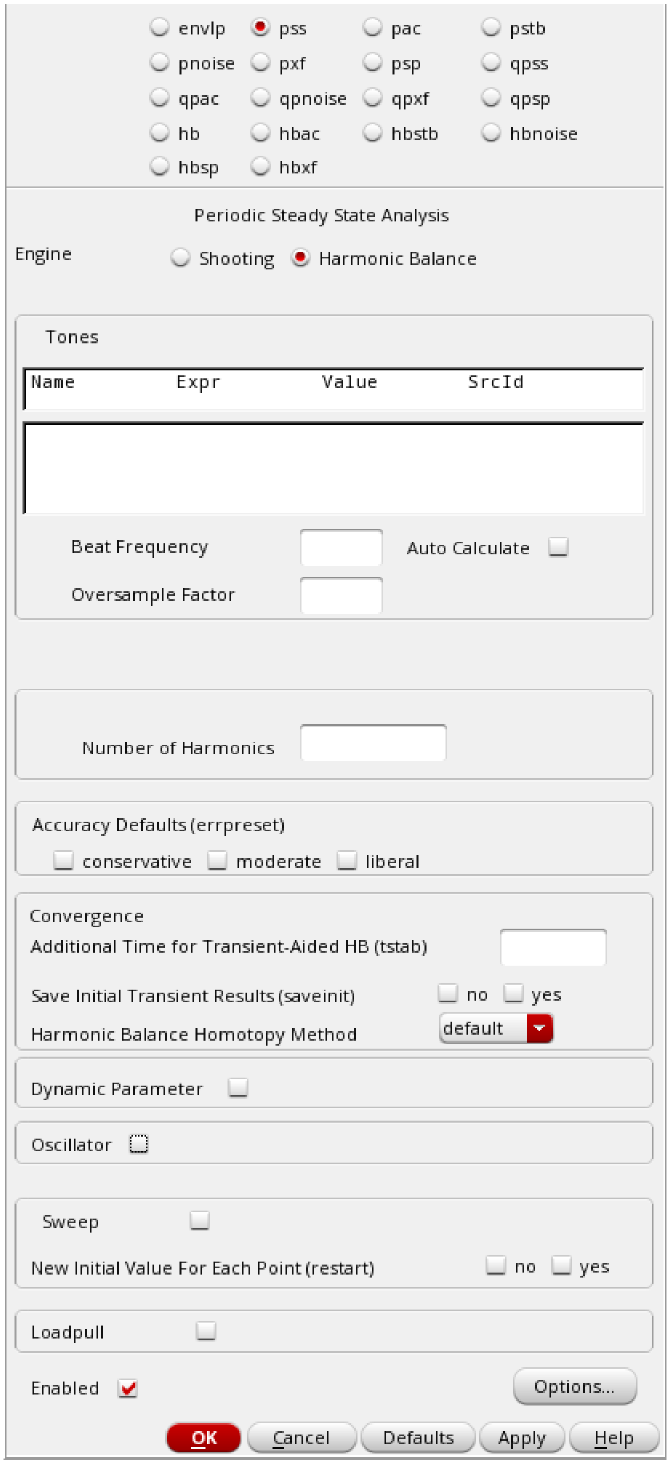
The *Choosing Analyses* form is displayed.

Figure 3-201 The Choosing Analyses Form



19. In the *Analysis* section, select *pss*. The form expands, as shown below.

Figure 3-202 The Choosing Analyses Form- Setting PSS Analysis



20. In the *Engine* section, ensure that *Shooting* is selected (this is the default).

21. In the *Beat Frequency* field, type `1G`.

The frequency entered here is an approximate frequency of oscillation. Normally, the oscillation frequency is set near the actual oscillation frequency. In this case, we have a very wide tuning range for this oscillator. A value near the middle frequency is chosen so that pss can converge on all the frequencies that are produced in the sweep.

22. In the *Number of harmonics* field, type `20`.

In general, you want to choose a number that is high enough to capture the nonlinearity of the circuit.

For example, in this measurement you would be running fullspectrum pnoise. For fullspectrum pnoise, start with 10 pss harmonics, select *fullspectrum,* and select *APS*. Run the simulation, and plot the noise result. Now increase the number of harmonics which forces more pss timepoints, and run the simulation again. If the pnoise result did not change, then you had enough harmonics to begin with. If the noise result did change, then increase the number of harmonics and run the simulation again till your pnoise result do not change.

In this measurement 20 number of harmonics gives stable pnoise results.

23. In the *Accuracy Defaults* (*errpreset*) section, select *conservative*.

conservative is typically used because very small amplitude phase noise measurements are normally desired. *conservative* is recommended for all the oscillators.

24. Select *yes* for the *Run Transient* option.

25. Type `10n` in the *Stop Time(tstab)* field.

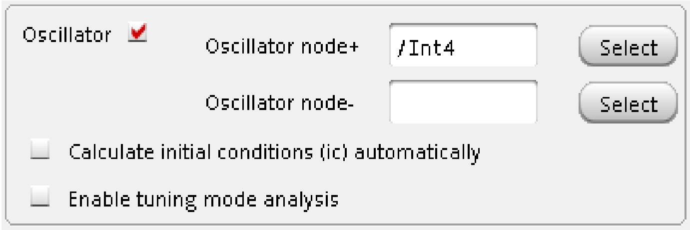
tstab is typically set to about 2-3 periods of the oscillation frequency for ring oscillator circuits. Note that this needs to apply for the lowest frequency that is produced. For that reason, 10n is set.

26. Select *yes* for *Save Initial Transient Results*. This will help in visualizing the buildup of the oscillation waveform.

27. Select the *Oscillator* option.

This is required for simulating an autonomous circuit. The oscillator section expands, as shown below.

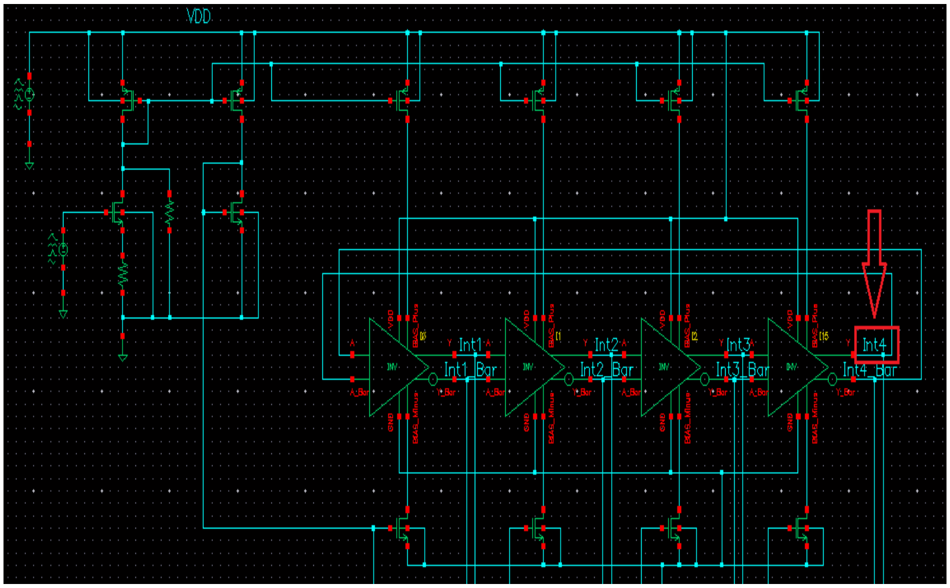
Figure 3-203 The Choosing Analyses Form - Oscillator Section



28. In the *Oscillator node+* field, click *Select* on the right. In the schematic, select the *Int4* node. This oscillator node will be used by the simulator for the period calculation of the oscillations. It just needs to be a node that has the oscillator signal on it.

29. Deselect the *Calculate initial conditions (ic) automatically* checkbox. This is because selecting this checkbox only works for feedback oscillators while the ring oscillator is not a feedback oscillator. Therefore, selecting this checkbox will not be able to find an oscillatory state for ring oscillator.

Figure 3-204 Selecting *Int4* net on schematic

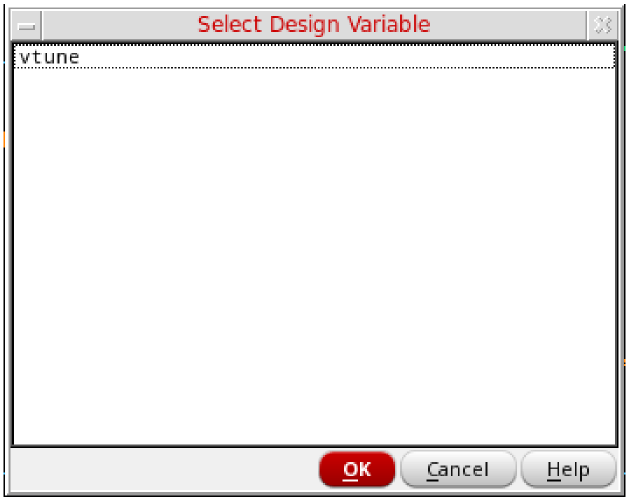


30. Select the *Sweep* option. This allows you to sweep the tuning voltage (vtune), as explained below.

a. Click the *Select Design Variable* button. The *Select Design Variable* window is displayed.

b. In the *Select Design Variable* window, select *vtune*.

Figure 3-205 Choosing vtune in Select Design Variable Form during PSS Analysis setup



c. Click *OK* to close the *Select Design Variable* Form.

d. In the *Sweep Range* section of the form, type `0.0` in the *Start* field.

e. Type `3.2` in the *Stop* field.

f. By default, the *Sweep Type* is set to *Linear*.

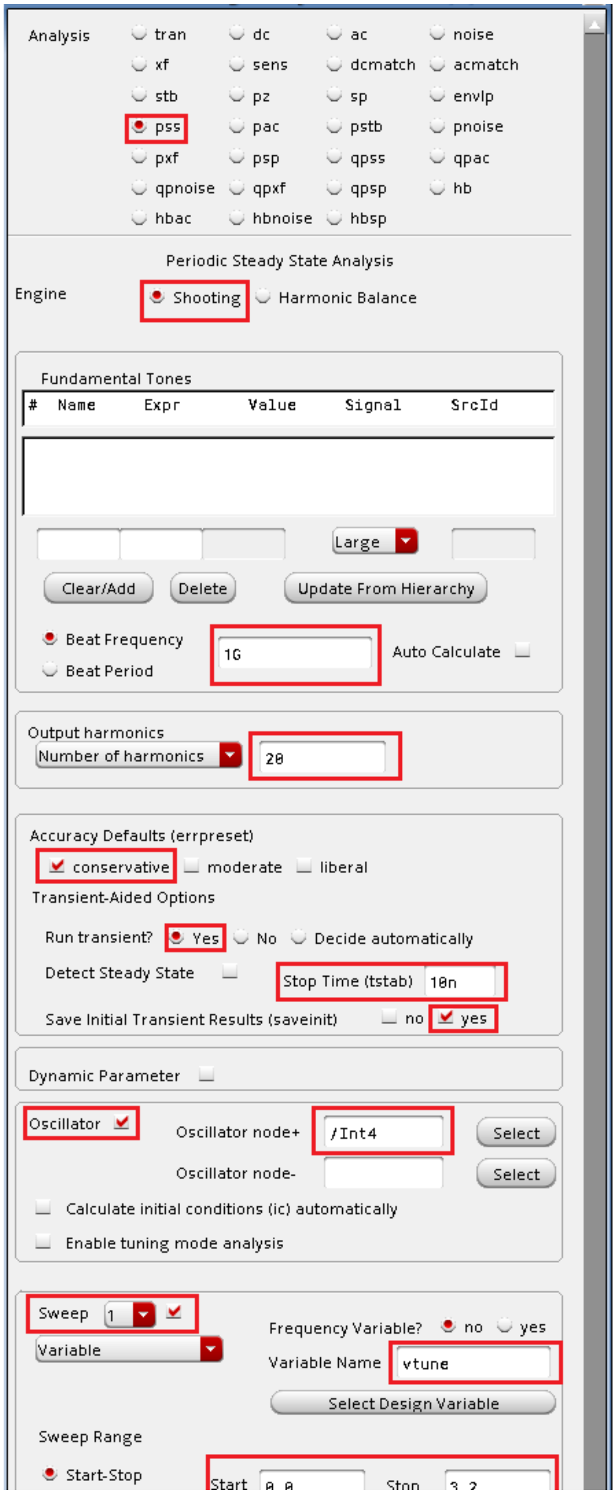
g. Type `0.2` in the *Step Size* field.

The sweep range for *vtune* is set based on for what tuning range your oscillator is designed for. This would be based on your oscillator design specifications.

31. Click *Apply* in *Choosing Analyses* Form. It checks the entries in the form for legality.

The completed *Choosing Analyses* form should like the figure below.

Figure 3-206 Choosing Analyses Form - PSS-Shooting Method Setup - Part1

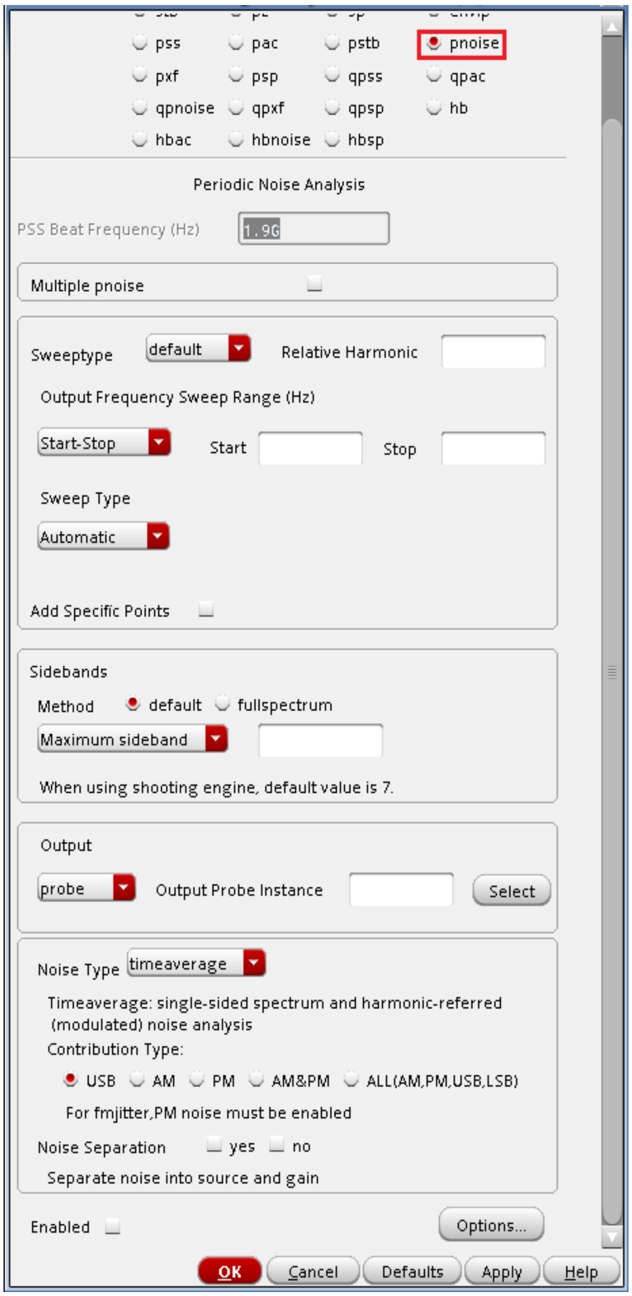


Setting up the Pnoise analysis

This analysis is set to do the phase noise measurement. It is run after *pss* analysis.

1. In the *Choosing Analyses* form, select *pnoise*. The form expands, as shown below.

Figure 3-207 The Choosing Analyses Form - *pnoise* Analysis Setup



2. Leave the *Sweep Type* to *relative*.

For oscillators, the *hbnoise/pnoise* frequency range defaults to *relative*. Specify the harmonic number as appropriate for the system you are simulating. If you are simulating an oscillator by itself, then the harmonic number is likely to be 1. If you have an oscillator and a diode frequency doubler, then the harmonic number is likely to be 2. If you have an oscillator with a frequency divider, in the hb/pss form, you should specify the approximate frequency of oscillation for the frequency-divided signal. In the *hbnoise/pnoise* form, if the noise is desired on the frequency divided output, then the relative harmonic is 1. If the noise is desired at the output of the oscillator, the relative harmonic number is the divide ratio. The meaning of *relative* is to take the frequency of the harmonic number specified and add to it the frequencies specified in the *Choosing Analyses* form. If the oscillator had a 1GHz output, and the pnoise had 1M relative to the first harmonic specified, the actual output frequency is 1G + 1M, or 1001M.

a. Type `1` in the *Relative Harmonic* field as you are simulating an oscillator by itself.

b. Select *Single Point* for *Output Frequency Sweep Range.*

c. Type `1M` in the *Freq* Field. In general, the pnoise frequency would be set as appropriate for your application.

3. Select the *Method* as *fullspectrum* in the *Sidebands* section.

Full-Spectrum pnoise is useful for circuits like switched-capacitor filters or sampling circuits where aliasing occurs through very high harmonics of the clock. The runtime advantages are large with no loss in accuracy of the result. Full-Spectrum pnoise is available when *Shooting* is selected for the pss *Engine* and *APS* is selected in the *Setup - High Performance Simulation* menu in ADE Explorer. Selecting *fullspectrum* in the pnoise form forces *APS* to be selected in ADE Explorer. If you are running from the command line without using *+aps*, *fullspectrum* will not be used. In normal pnoise when *Shooting* is selected as *Engine*, you need to set the maximum sideband term. In full-spectrum pnoise you do not do this except in cases where the pss beat frequency is 100KHz or less. In this case, set maximum sidebands to the 1/f noise corner frequency divided by the pss beat frequency. Pnoise calculates all the noise translations it can, based on the maximum timestep in the pss analysis. The easiest way to change the maximum timestep in pss is to increase the number of harmonics above 10 in the pss *Choosing Analyses* form. Full-spectrum pnoise runs faster than normal pnoise when the maximum number of sidebands in pnoise is about 50 or greater. The larger the number of sidebands, the larger the speedup for *fullspectrum* pnoise analysis.

Fullspectrum pnoise is available for all available noisetypes. Noise separation is not supported when fullspectrum is selected.

Note: When the default method is selected, specify the number of desired frequency translations in the *Maximum sideband* field. This will always be specific to the design.

For a rough estimate, plot the frequency domain content, and find the harmonic number where the amplitude has dropped about 40dB. Try this number in maximum sidebands. To verify that you have enough sidebands, raise the number by about 50% and run again. If the noise result did not change, then you had enough sidebands to begin with. Up to 40 sidebands can be specified by default. If you need more than 40 sidebands, set the number of pss harmonics to the number of sidebands divided by two to four or set the pss option maxacfreq to the beat frequency times the number of pnoise sidebands.

4. Leave *Maximum sideband* field blank as explained above since the *Method* selected is *fullspectrum*.

5. Set the *Output* to *voltage*.

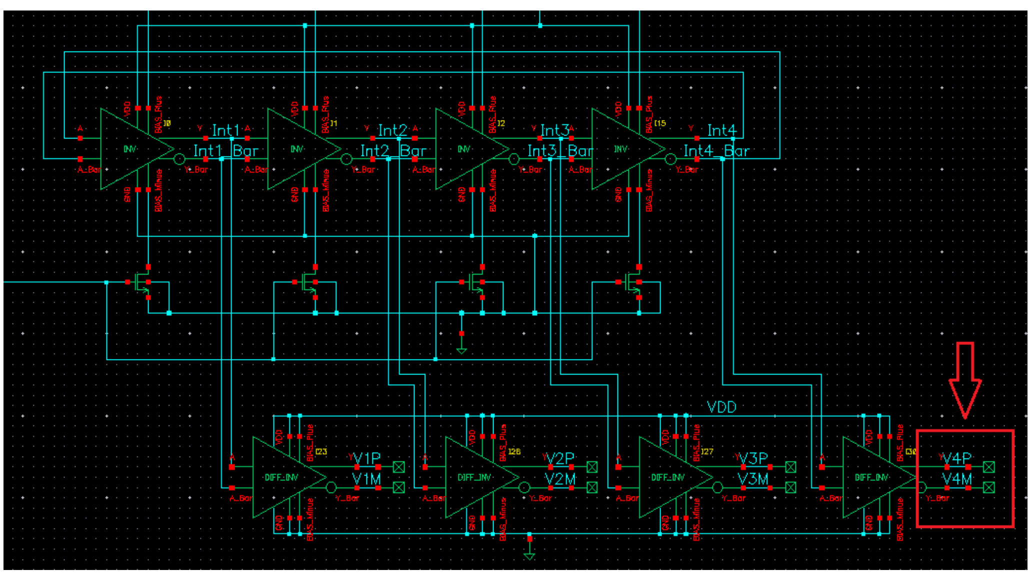
  a. Type `/V4P` in the *Positive Output Node* field. You can also select *V4P* net from schematic by clicking *Select* button on the right of the *Positive Output Node* field and then selecting the net just below the *out* label in schematic.

  b. Type `/V4M` in the *Negative Output Node* field. If the second node, that is, the *Negative Output Node* is left blank, it will be connected to the global ground node automatically. However, if you have differential oscillator, you need to specify both the nodes.

6. Choose noise type as *timeaverage* from the drop-down list.

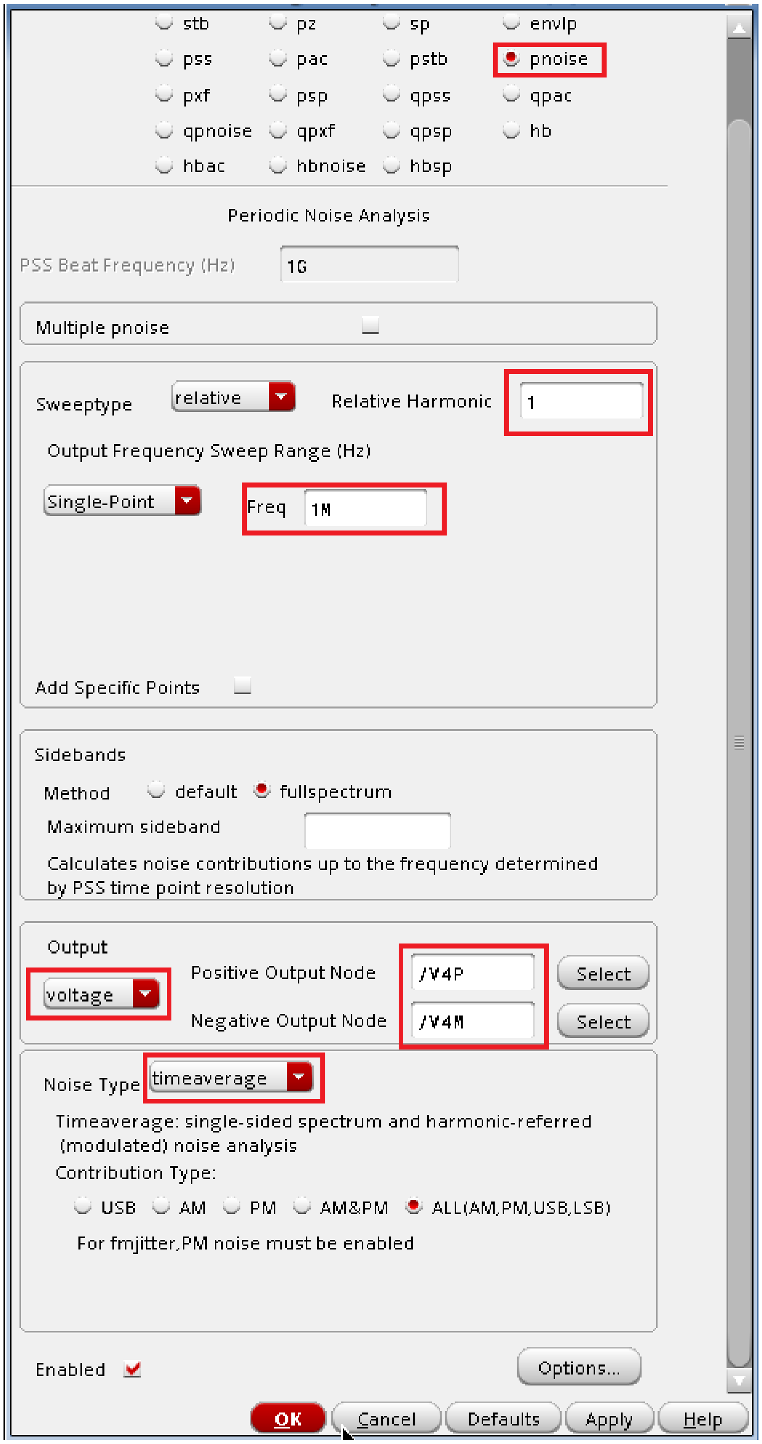
7. Select the *ALL(AM,PM,USB,LSB)* option.

Figure 3-208 Selecting *V4P and V4M* net from the schematic



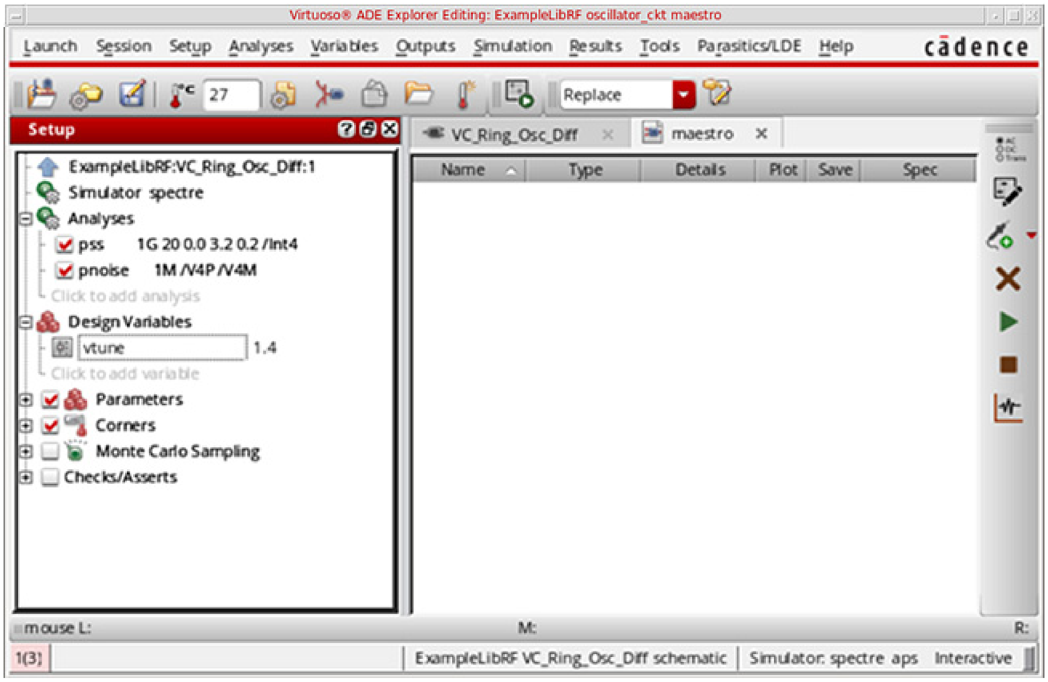
The *Choosing Analyses* form should look like the following:

Figure 3-209 Choosing Analysis Form - *pnoise* Analysis Setup

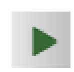


8. Click *OK* to close the *Choosing Analyses* form. This will add the *pnoise* analysis along with *pss* analysis in the *Analyses* section of ADE window, as shown below.

Figure 3-210 ADE Explorer Simulation Window - *pss* and *pnoise-jitter* analysis setup



Running the PSS and Pnoise analysis

Once finished setting up the PSS and Pnoise Analyses click the green icon  on the right of ADE Explorer or on the Schematic window to run the simulation.

This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` logfile). When the analysis has completed, you may iconify the status window.

Next, you will plot the results.

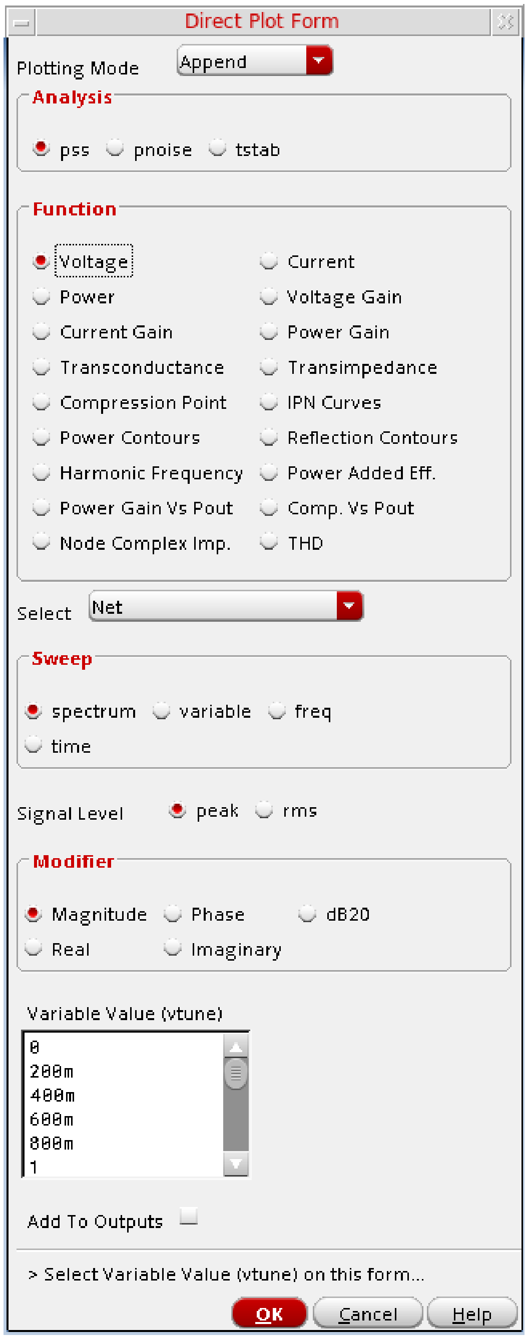
Plotting the results

First Plot the oscillator output frequency, as follows:

1. In ADE Explorer, select *Results - Direct Plot - Main Form*.

   The *Direct Plot Form* is displayed, as shown below.

Figure 3-211 Swept *pss* and *pnoise* Analysis Direct Plot Form

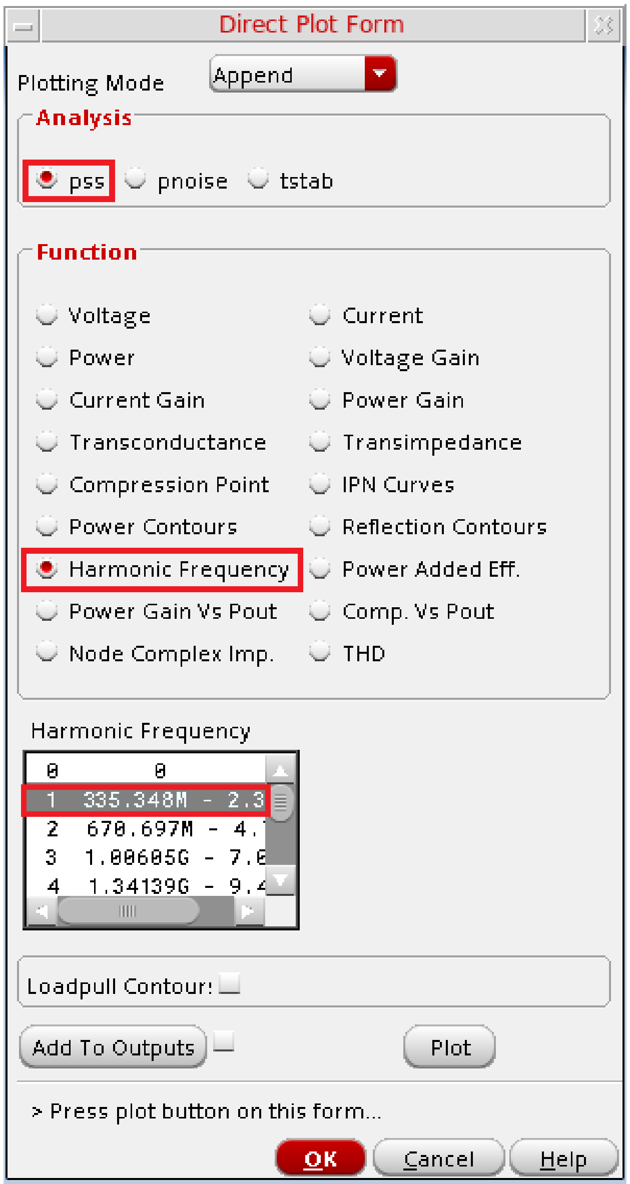


2. In the *Direct Plot Form*, select *pss* in the *Analysis* section. This is selected by default.
3. Select *Harmonic Frequency* in the *Function* section.

4. In the *Harmonic Frequency* section, select the first harmonic. This will plot only the change in first harmonic of the oscillation frequency vs. change in oscillator's tuning voltage.

   The *Direct Plot Form* should look like the following:

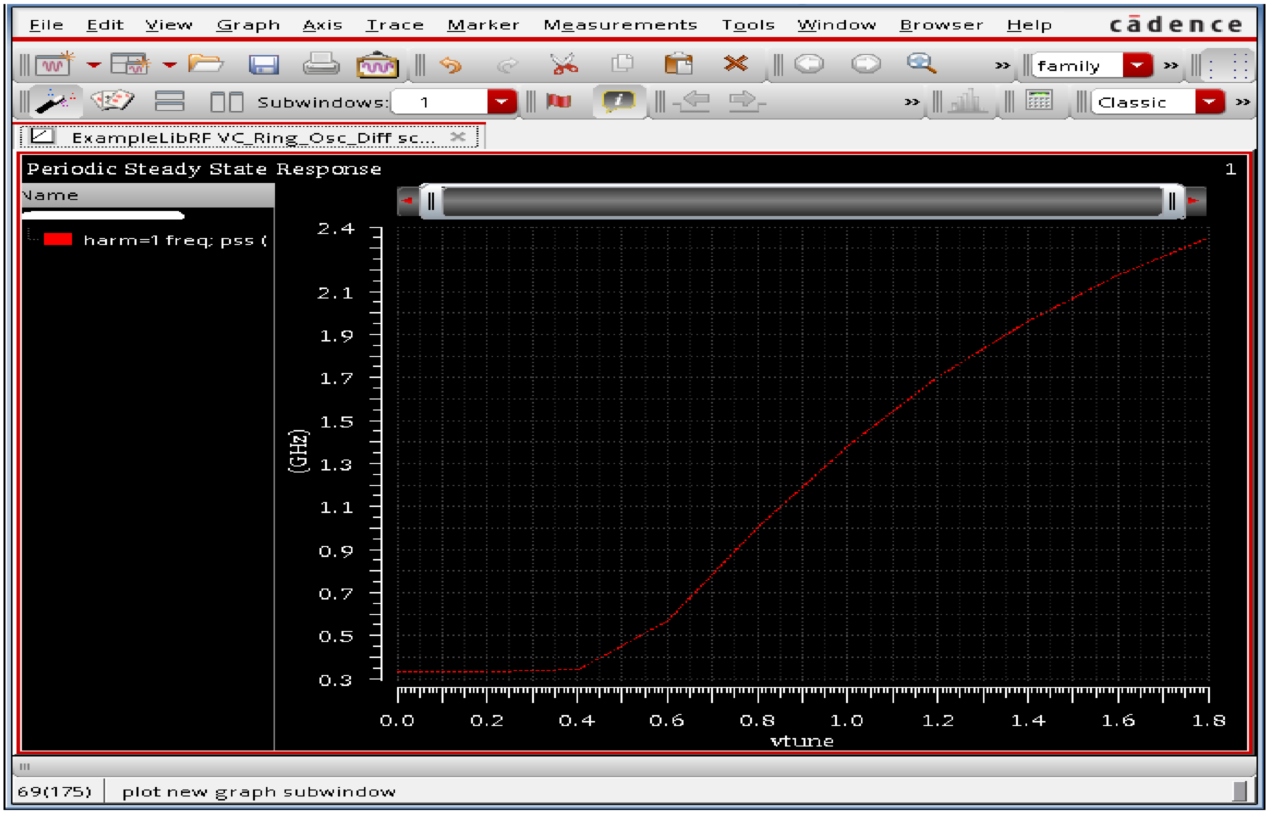
Figure 3-212 Swept *pss* Analysis Direct Plot Form Setup



5. Click *Plot*.

   The oscillator tuning range is plotted. From the plot you can observe that as you increase the tuning voltage, the oscillator frequency increases.

Figure 3-213 Swept PSS Measurement Plot - Harmonic frequency variation plot



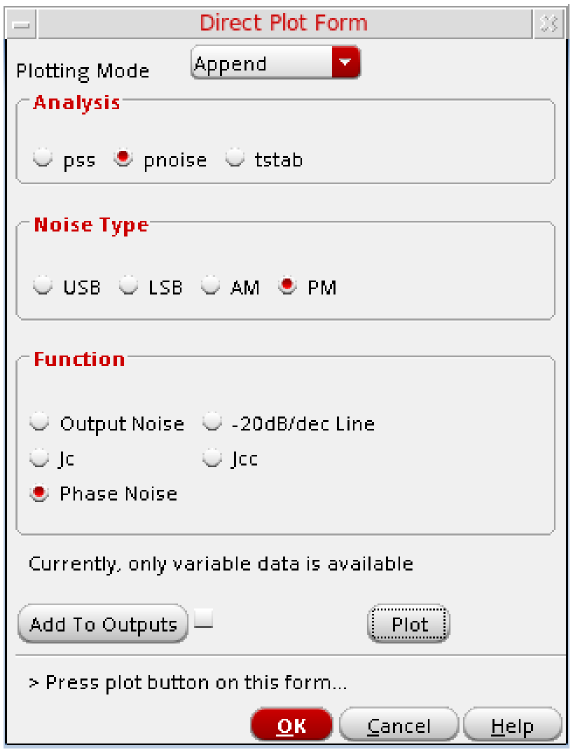
6. Select *File - Close All Windows* to close the waveform window.

You will now plot the phase noise.

1. In the *Direct Plot Form*, select *pnoise* in the *Analysis* section.
2. Select *PM* for *Noise Type*.
3. Select *Phase Noise* in the *Function* section.

   The *Direct Plot Form* should look like the following:

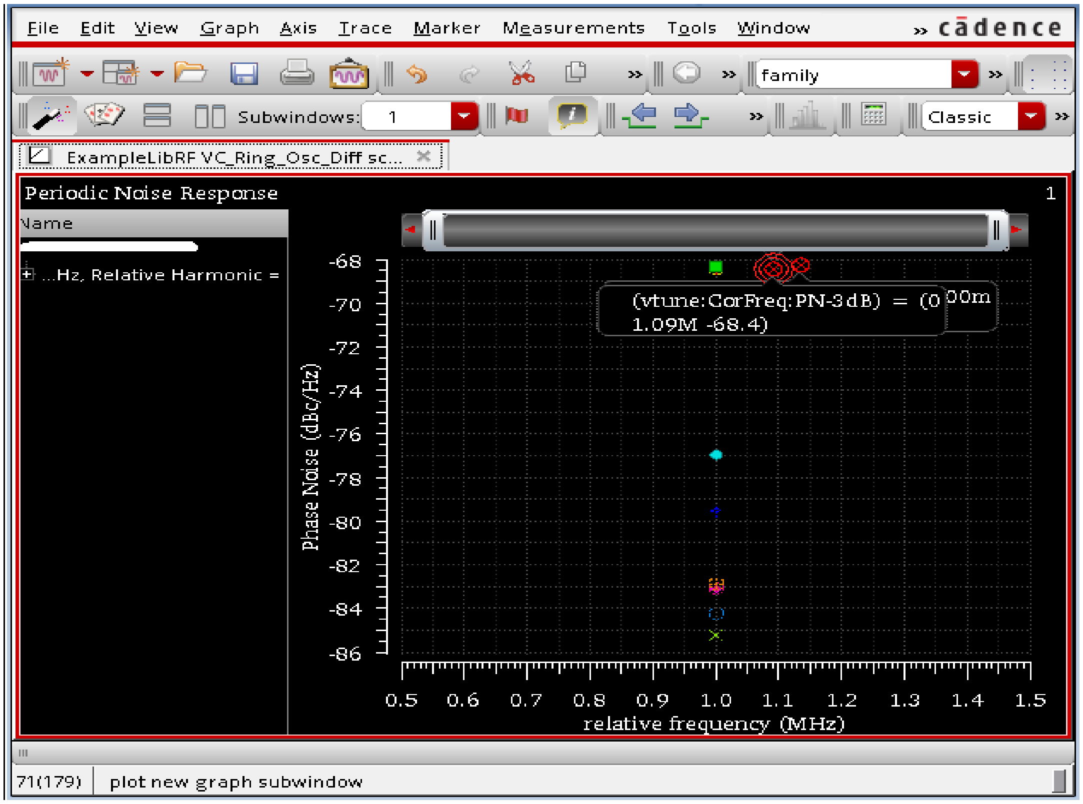
Figure 3-214 Direct Plot Phase Noise Setup



4. Click *Plot*.

This will plot the Single Sideband Phase Noise vs. Relative Frequency graph, as shown below:

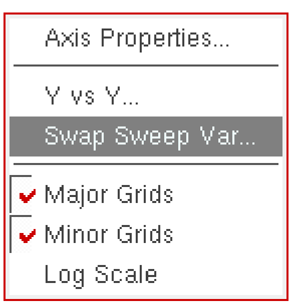
Figure 3-215 Phase Noise vs. Relative Frequency Plot



In the above plot, you have a graph of phase noise vs. relative frequency for each vtune sweep value. Next, you will modify the phase noise plot to provide more useful information.

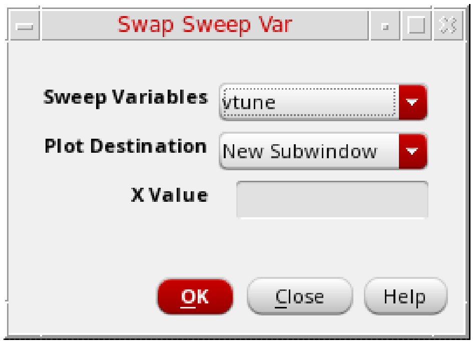
1. In the plot window, right-click on one of the numbers on the X Axis and select *Swap Sweep Var* from the context menu.

Figure 3-216 X-Axis - Right Mouse Button (RMB) - Object Menu



2. In *Select Sweep Var* dialog box which opens, select *vtune* from the *Sweep Variables* drop down list. Keep *Plot Destination* as *New SubWindow*.

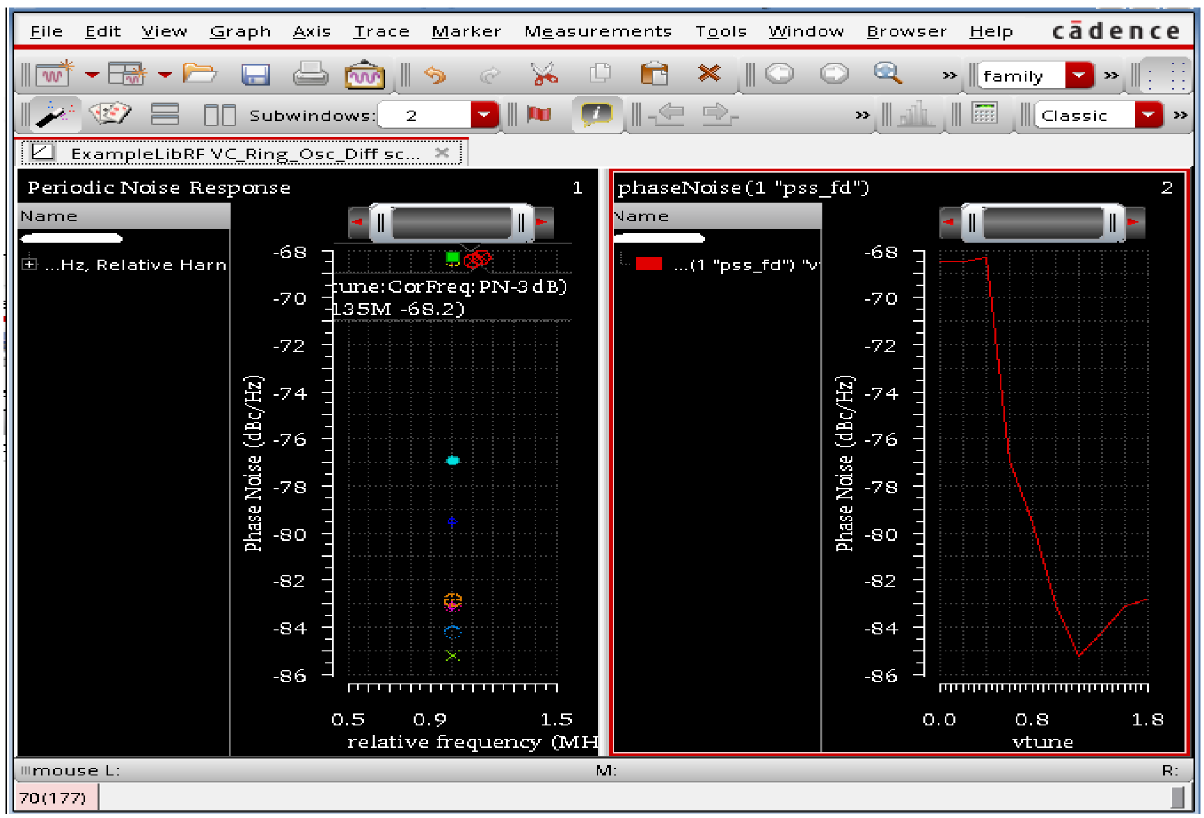
Figure 3-217 Swap Sweep Var Dialog Box Window



3. Click *OK*.

   The waveform tool draws a new subwindow.

Figure 3-218 Tuning voltage vs phase noise plot



If you look at the graph on the right, you will see that the phase noise is worse at the lower end of the tuning range.

4. Select *File - Close All Windows* to close the waveform window.

5. Clean up the screen for the next set of measurements.

   a. Close ADE Explorer by selecting *Session* - *Quit*.

   b. Exit the Virtuoso session by selecting *File - Exit in CIW*.

Summary

The Simulating Oscillator section shows how to simulate and make typical measurements on an oscillator.

In this Oscillator section, the following simulation setups/measurements were shown:

- Starting and Stabilizing of Feedback Oscillators
- Loop Gain Measurement of Feedback Oscillator
- Swept Tuning Range and Phase Noise Measurement
- Starting and Stabilization of Ring Oscillators
- FM Jitter Measurement of Ring Oscillator
- Swept Tuning Range and Phase Noise for the Ring Oscillator

For more information on simulating oscillators, please refer to the chapters in the *Virtuoso Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide* and the *Spectre Circuit Simulator RF Analysis Theory* guide.

4

Simulating Mixers

The SpectreRF simulator can simulate circuits, such as mixers, that show frequency conversion effects.

This section uses two double balanced mixer circuits, `db_mixer` and `db_mixer_xmit`, to illustrate how the SpectreRF simulator can determine the characteristics of a mixer design.

In the mixer examples that follow, you will plot the following nonlinear characteristics of the `db_mixer` and `db_mixer_xmit mixer` circuits.

Receive Mixer Measurements (db_mixer)	Analyses
Mixer Conversion Gain and RF to IF Isolation	HB and HBAC
LO Leakage (LO to IF Leakage)	HB
Noise Figure Measurement, Noise Summary Table	HB and HBnoise
1dB Compression Point, Desensitization and Blocking	HB, HBAC, and HBnoise
Third-Order Intercept measurement with HB	HB
Rapid IP2, Rapid IP3	Specialized HBAC
Compression Distortion Summary	HB, HBAC
Transmit Mixer Measurements (db_mixer_xmit)	**Analyses**
Image Rejection	HB
Three tone Swept IP3 (large signal)	HB
Noise Figure	HB and HBnoise

To use the examples in this section, you must be familiar with the SpectreRF simulator analyses as well as know about mixer design. For more information about the SpectreRF simulator analyses, see *SpectreRF Simulation Option Theory*.

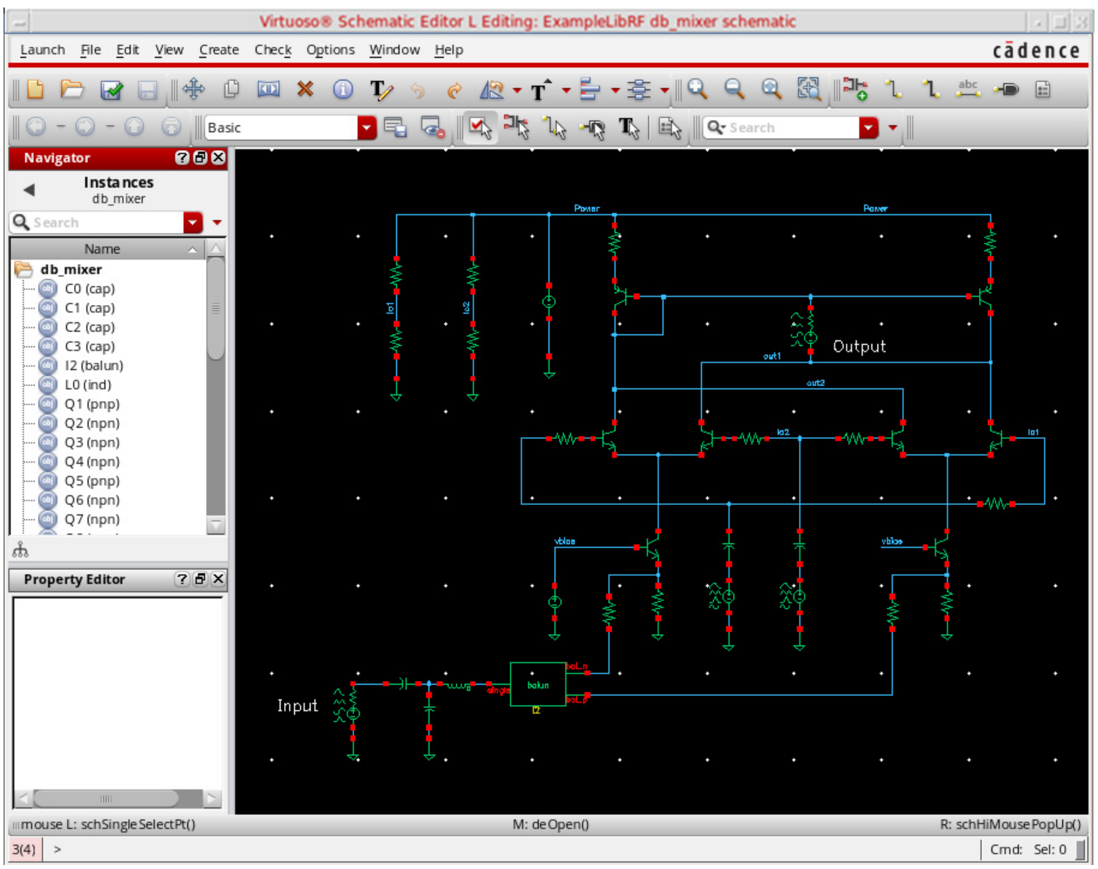
The db_mixer and db_mixer_xmit Mixer Circuits

The `db_mixer` and `db_mixer_xmit` circuits can be found in the *ExampleLibRF* library. See the Introduction chapter for the instructions on accessing the *ExampleLibRF* library.

The `db_mixer` integrated circuit is a Gilbert cell (down-converting double balanced) mixer.

The schematic for the `db_mixer` circuit is shown below.

Figure 4-1 Schematic for the db_mixer Mixer Circuit



On the left side of the schematic there is a port labeled *Input* which generates the input signal. To the right of that is a matching network and a behavioral balun from rfLib. This feeds the input to a double-balanced mixer. There are two LO sources in the circuit (in the middle of the schematic) between the bottom devices. The LO operates at 1.9GHz. Next to the label *Output*, is the output port of the mixer.

The `db_mixer_xmit` circuit is a similar circuit, but an up-converting double-balanced mixer with image rejection.

The following tables lists some measured values for different aspects of the `db_mixer` down-converting mixer.

Measurement	**Measured**
LO frequency (Hz)	1.9 GHz
RF frequencies (Hz)	1.904 GHz, 1.905 GHz
IF frequency (Hz)	4 MHz, 5 MHz
LO voltage	200mV peak
RF power	-30 dBm
Conversion gain	*measurement needed*
Double Sideband Noise figure	*measurement needed*
Input 1dB compression point	*measurement needed*
Input IP3 (from swept power)	*measurement needed*
Input IP3 (from HBAC analysis)	*measurement needed*

Design Variable	**Default Value**
prf (RF power)	-30 dBm
vlo (LO magnitude)	200 m
frf1 and frf2 (RF frequencies)	1.904 GHz, 1.905 GHz
flo (LO frequency)	1.9 GHz

Setting Up to Simulate the db_mixer Mixer

In a Unix window, type `virtuoso &` to start the Cadence software. (For more information, see the <u>Introduction</u> chapter.)

Opening the db_mixer Mixer Circuit in the Schematic Window

1. In the CIW (Command Interpreter Window), select *File – Open*.

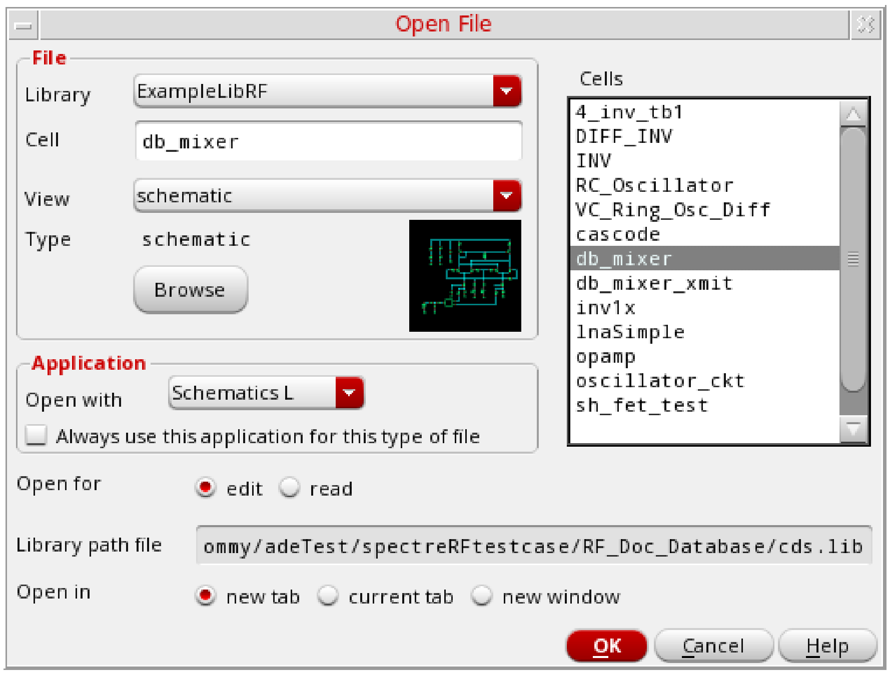
The Open File form is displayed.

2. In the Open File form, choose *ExampleLibRF* from the Library drop-down list.

3. Type `db_mixer` in the Cell field or select the cell from the *Cells* list box.

   The completed *Open File* form appears like the one below.

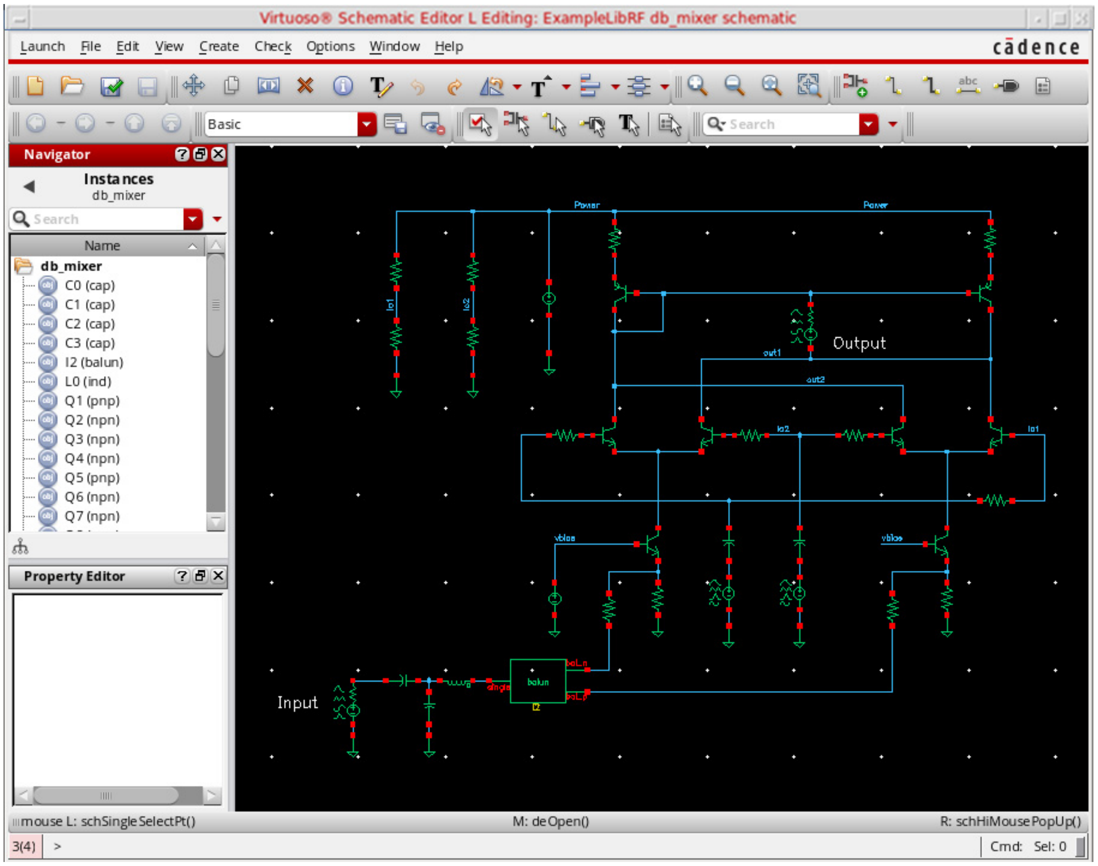
Figure 4-2 Open File Form



4. Click *OK*.

   The Schematic window for *db_mixer* mixer is displayed.

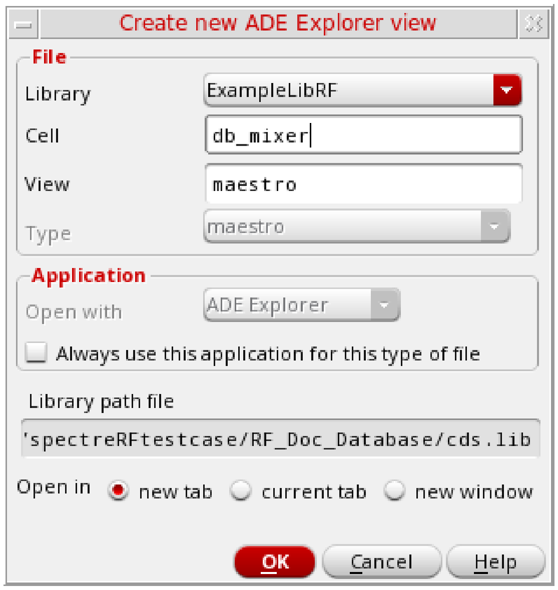
Figure 4-3 db_mixer Schematic



5. In the Schematic window, choose *Launch – ADE Explorer*.
6. In the *Launch ADE Explorer* dialog, select *Create New View*.

   The *Create new ADE Explorer view* form is displayed.

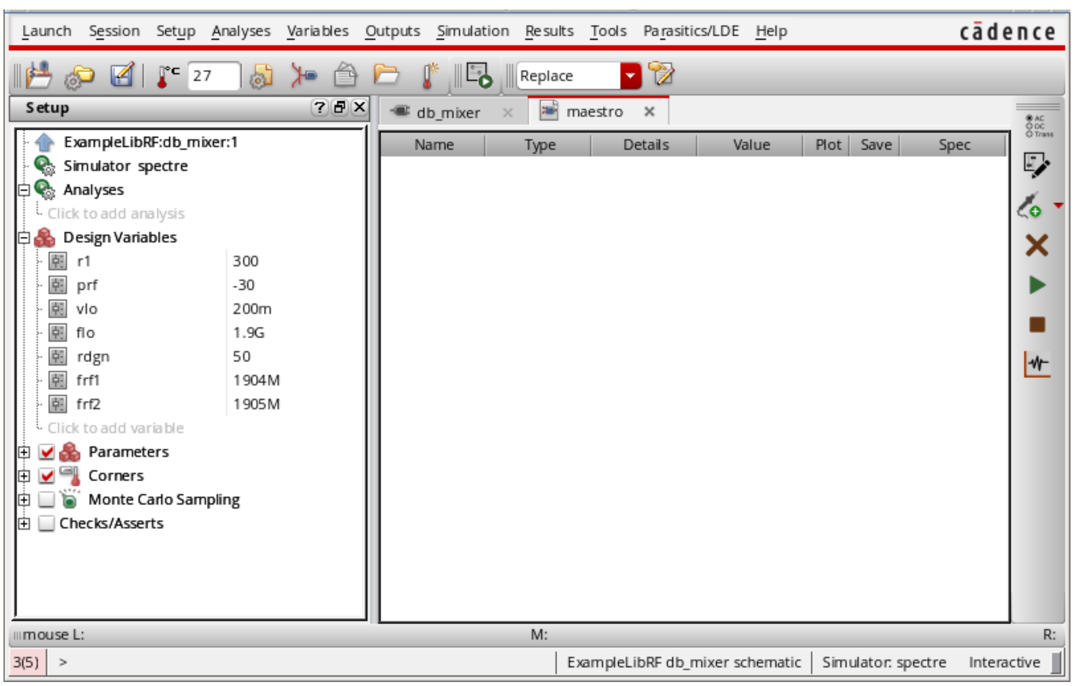
Figure 4-4 Create new ADE Explorer view



7. Leave each option to the default selections and click *OK*.

   The ADE Explorer window is displayed.

Figure 4-5 ADE Explorer Window



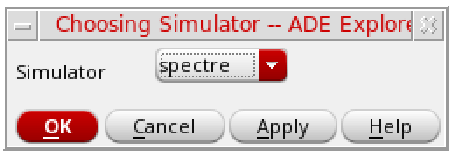
Choosing Simulator Options

1. In ADE Explorer, select *Setup – Simulator*.

   The *Choosing Simulator* form is displayed.

2. Select *spectre* from the *Simulator* drop-down list.

Figure 4-6 Choosing Simulator Form - Completed



3. Click *OK to close the Choosing Simulator* form.

4. Set up the High Performance Simulation Options, as follows:

   In the ADE window, choose *Setup - High Performance Simulation*. The *High Performance Simulation Options* window is displayed, as shown below.

Figure 4-7 High Performance Simulation Options

Select *APS* as the simulation performance mode. Note that *Auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 64) and then multi-thread on all the available cores.

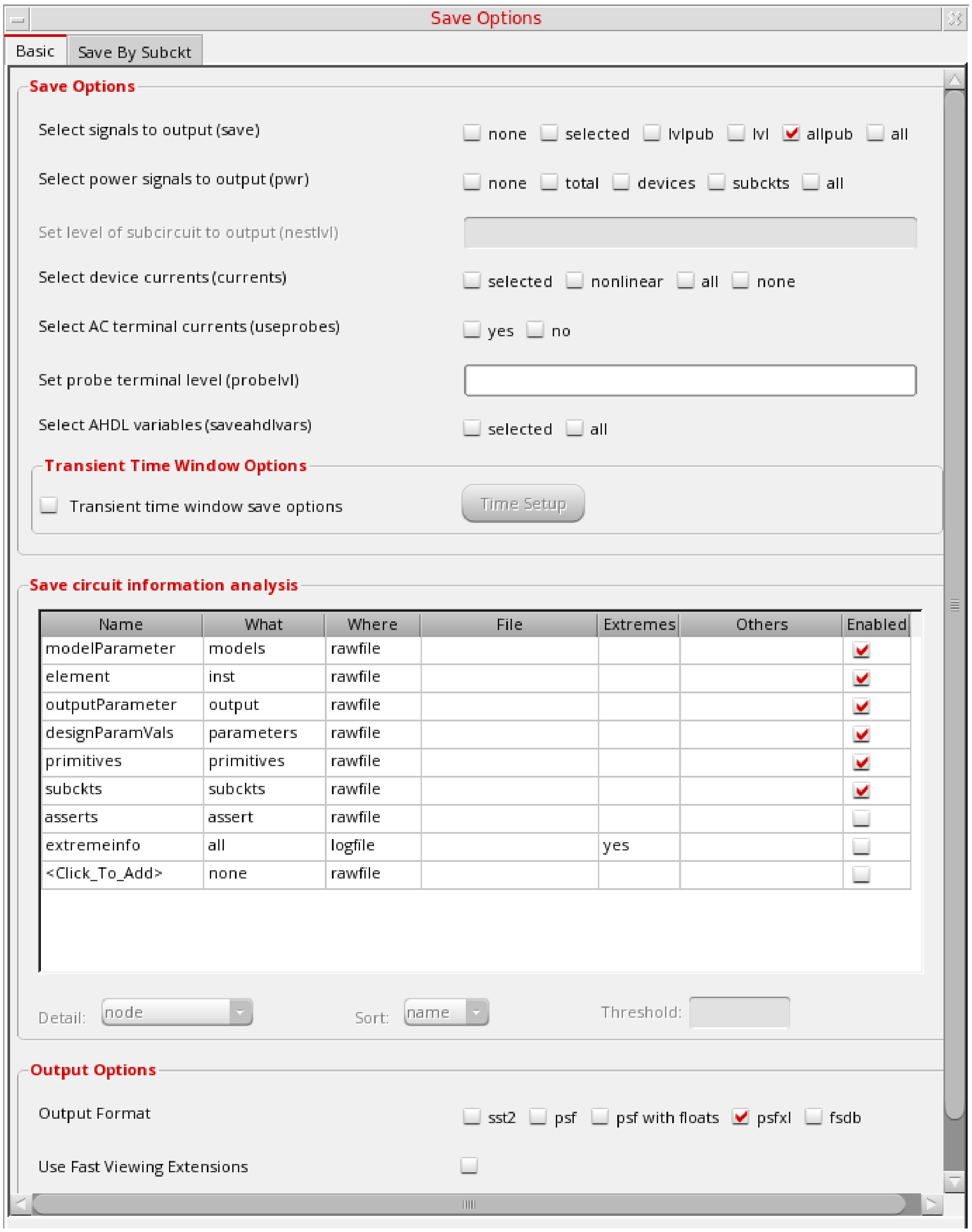
Note: The bigger the circuit, the more threads you should use. For a small circuit such as this, you may want to set the number of threads to 2. Using 16 threads on a small circuit might actually slow things down because of the overhead associated with multithreading. For more information, see the Virtuoso Spectre User Guide.

a. Click *OK*.

5. Select *Outputs – Save All*.

The *Save Options* form is displayed, as shown below.

Figure 4-8 Save Options Form



6. In the *Select signals to output* section, make sure that *allpub* is selected.

This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

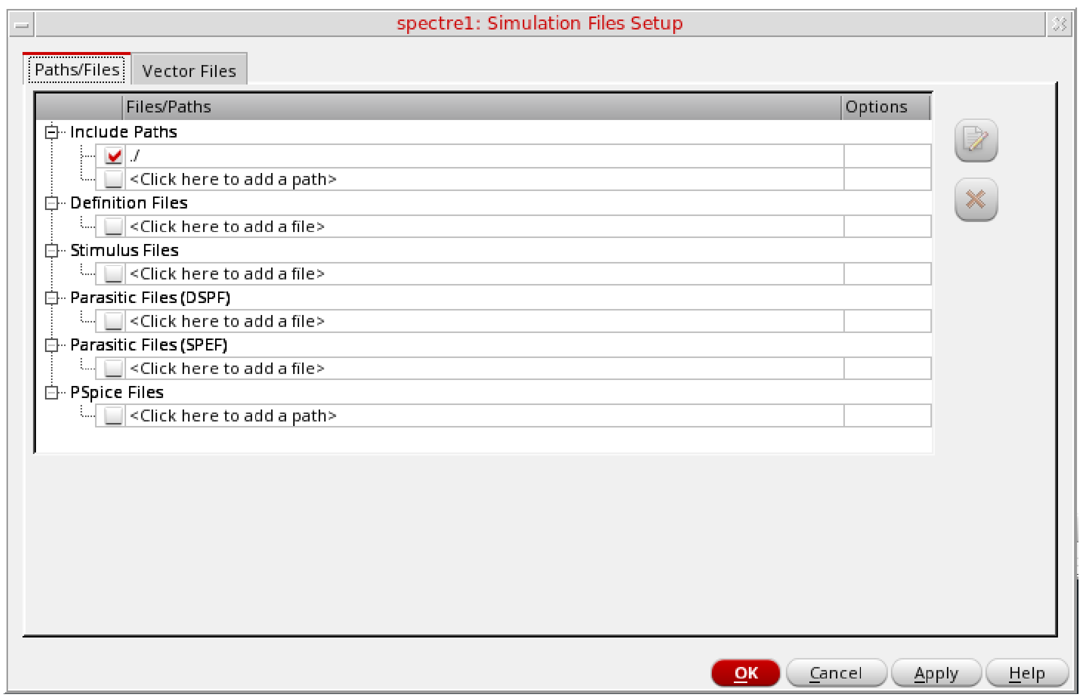
To save the currents, use the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or *all* if you want to save all the currents in the circuit.

7. Click *OK*.

Setting Up Model Libraries

1. In ADE Explorer, select *Setup - Simulation Files*. The *Simulation Files Setup* form is displayed, as shown below.

Figure 4-9 Simulation Files Setup Form



2. Verify that the *Include Path* is set as shown above.

3. Select *Setup – Model Libraries.*

   The *Model Library Setup* form is displayed.

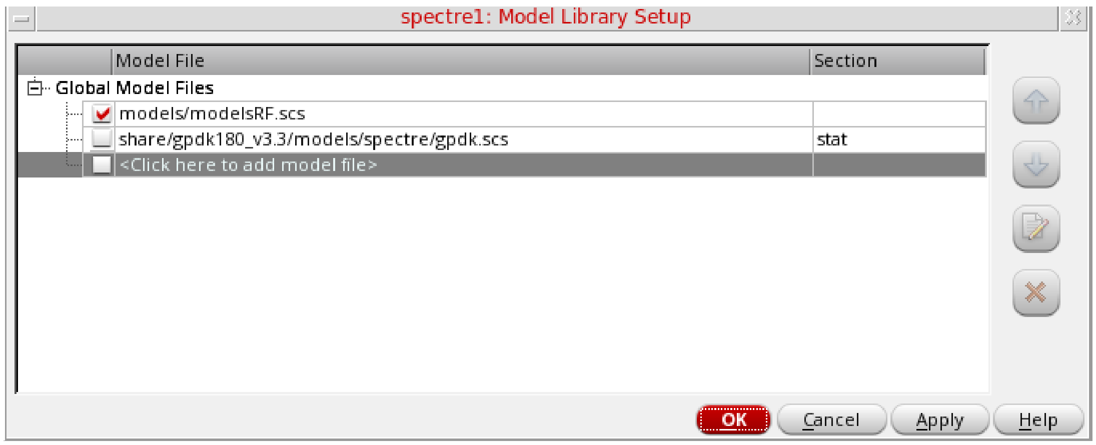
4. In the *Model Library File* field, type the following for the name of the model file:

```
models/modelsRF.scs
```

5. Click *Add*.

   The *Model Library Setup* form looks like the following:

   Figure 4-10 Model Library Setup



6. Click *OK*.

Mixer Conversion Gain, RF to IF Isolation, LO to IF Leakage, and Noise Figure

You can measure the conversion gain using hb (Harmonic Balance) by applying the signal which causes the frequency conversion (the LO) in a hb analysis and follow this with hbac (Harmonic Balance AC) to measure the small-signal conversion gain. This will also be used to measure RF to IF isolation. Because hbac is based on the hb result with all the harmonics that hb solved for, the mixing products produced by mixing the input frequency with any or all of those harmonics can be calculated with hbac. In addition, hbnoise (Harmonic Balance noise) will be used to calculate the output noise and noise figure with all the frequency translations inherent in the mixer.

Hb solves for the steady-state solution produced by the LO and captures the nonlinearity of the mixer. Hb calculates the nonlinearity in the frequency domain as a series of harmonics.

The hbac analysis calculates the conversion gain based on the nonlinearity created by the LO. Hbac is also used to calculate the RF to IF isolation. This measurement is not a simple AC frequency response. It is the frequency response from RF to IF with the LO applied to the mixer.

Hbnoise also uses the nonlinearity from the hb large-signal analysis in order to calculate how noise is folded to the desired output frequency. Both of these analyses are quite fast and accurate.

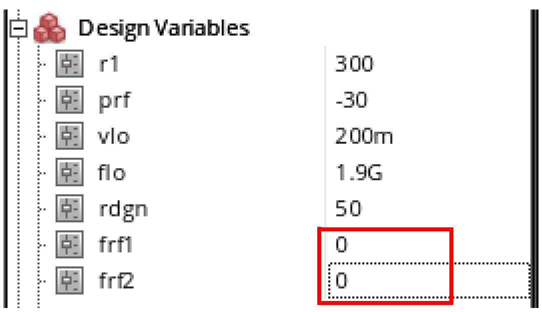
Hbac and hbnoise are both small-signal analyses. Both are nonlinear in the sense of being able to take into account the frequency conversions created by the LO. Noise is almost certainly a small-signal problem. In practice, hbac calculates the correct conversion gain for the cases where the RF input causes little compression (that is, the input is 10dB or more below the 1dB compression point.)

Setting Up the Simulation - Setting Design Variables

To set the design variables to the values required for each simulation, perform the following steps:

1. In the Design Variables section of ADE Explorer, change the design variables *frf1* and *frf2* to *0*. *flo* should already be set to *1.9G*. To edit the values, simply click on the value to the right of the variable name, and type in a value. Then, press *Enter*. Setting the input frequencies to 0 disables the production of waveforms for the large-signal analyses like tran, pss, and hb (harmonic balance).

Figure 4-11 Design Variables Section of ADE Explorer Window



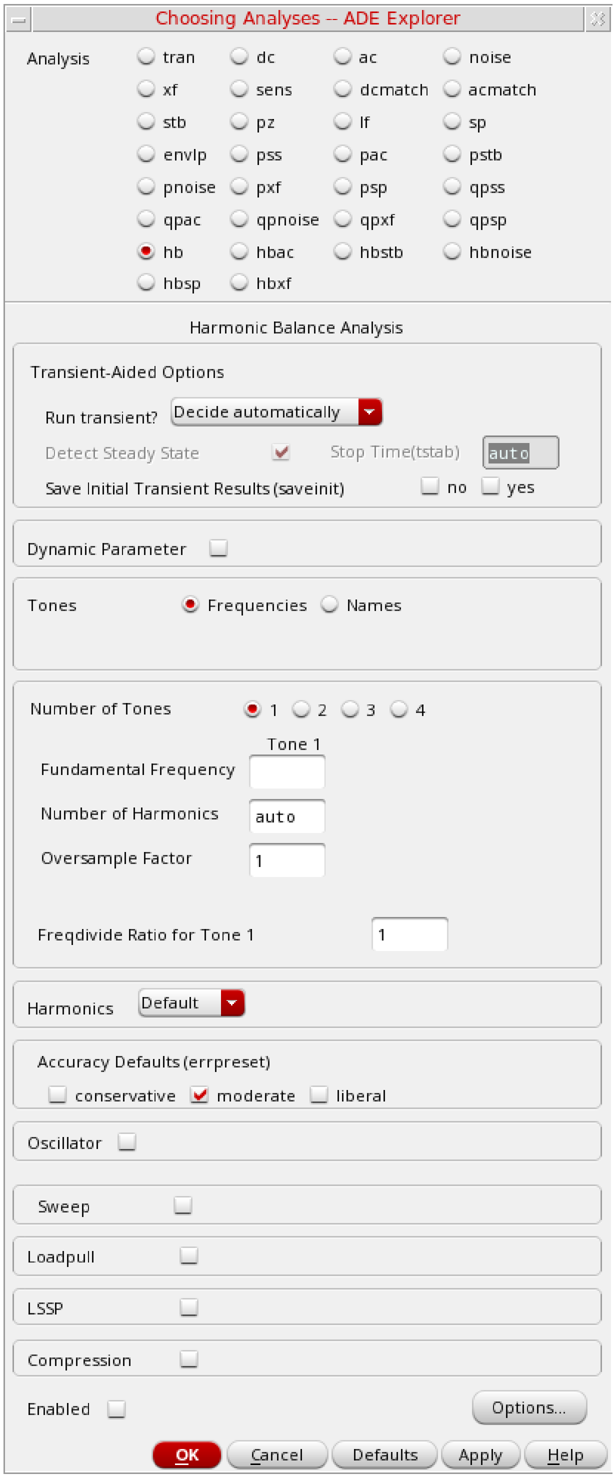
Setting Up the Harmonic Balance Analysis

1. In ADE Explorer, select *Analyses – Choose*.

   The *Choosing Analyses* form is displayed.

2. In the *Choosing Analyses* form, select *hb*. The form expands, as shown below.

Figure 4-12 The hb Choosing Analyses Form



3. In the *Transient-Aided Options section*, leave the settings at their default value:

a. Leave *Run Transient?* to the default value of *Decide Automatically.*

Run transient will run the LO signal using the transient (In SpectreRF, this is called the tstab interval) for a short period of time. At the end of tstab, an FFT is performed, and this is used as the starting point in the harmonic balance analysis. Doing this improves the convergence of hb by giving it a better starting point at the cost of a short transient analysis.*Stop Time (tstab) auto*.

When *auto* is selected for *Stop Time (tstab)*, a small number of periods of the LO is run using the transient analysis. During this time, the signal is checked for steady-state conditions. If steady-state is not reached in the initial number of periods, more periods can be added automatically by the simulator. Using this feature allows an accurate FFT for the starting point in the hb iterations.

b. When *Run transient* is set to *Decide automatically*, The *Detect Steady State* option is selected automatically. When this is set, when steady-state is detected in the tstab interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide automatically* simplifies the setup for harmonic balance, and produces correct answers from hb without needing to know how to set hb up manually.

c. Leave *Save Initial Transient Results (saveinit)* blank.

4. In the *Tones* section, ensure that *Frequencies* is selected. This is the default.

Harmonic balance can now set the harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. A transient analysis runs until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1* (when *Frequencies* is selected) or for the tone that has tstab enabled (when *Names* is selected). If you want to manually set transient-aided hb, select *Yes* from the *Run Transient?* drop-down list and set a time for the transient in the *Stop Time (tstab)* field. In this mode, the stop time of the transient analysis in the tstab interval cannot be automatically extended.

If you want to see the startup waveform, select *yes* for *Save Initial Transient Results (saveinit)*.

5. In the *Number of Tones* section, note that the number of tones is set to *1*. (This is the default in hb). Since the RF tones were disabled, only the LO tone remains.

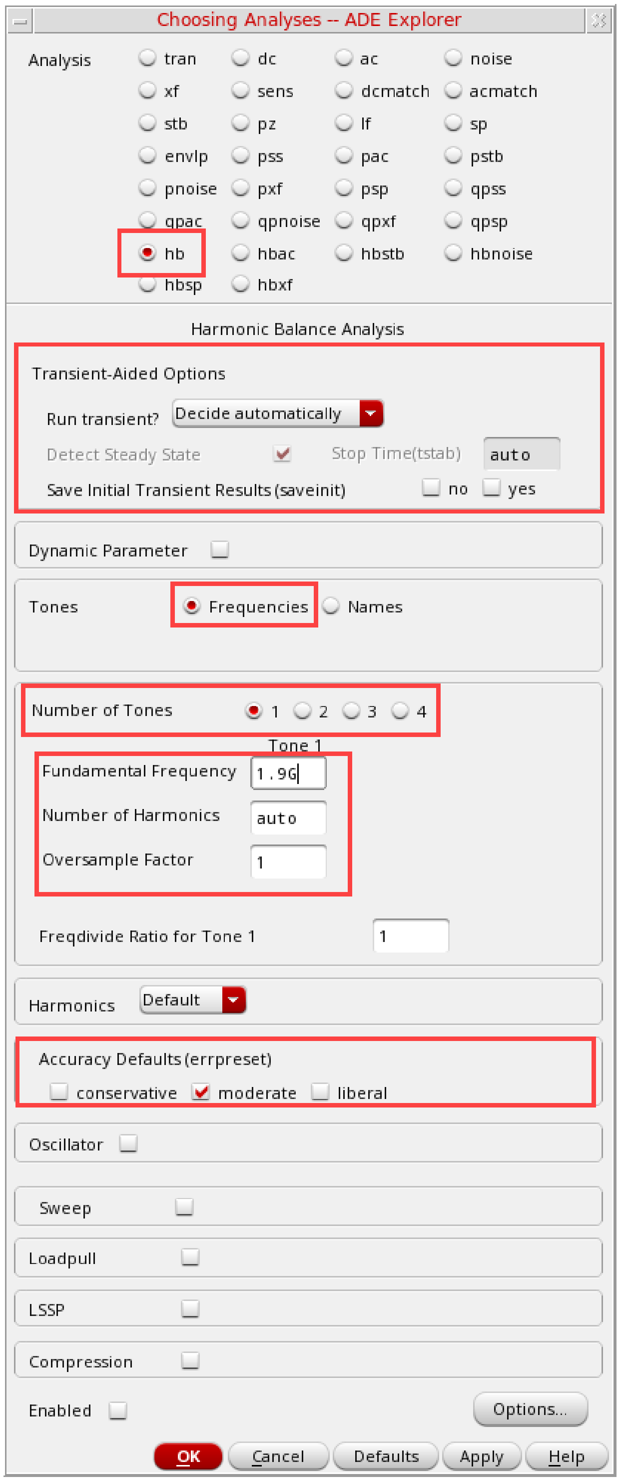
6. Enter `1.9G` as the *Fundamental Frequency.*

7. Leave *Number of Harmonics* set to *auto*. This is the default. Spectre will choose the appropriate number of harmonics for you. *auto* is allowed if *Decide automatically* or *Yes* are selected from the *Run transient?* drop-down list.

8. When you choose *auto* for the number of harmonics, leave *Oversample Factor* set to the default value of `1`. When all the signals in the system (including currents) are nearly sinusoidal, then, *Oversample Factor* should also be set to 1. Set *Accuracy Defaults (errpreset)* to *moderate*. Exceptional accuracy is not needed because only the high amplitude LO signal needs to be solved for, so *moderate* (the default) is selected.

9. Leave the rest of the form set to the default values. The hb *Choosing Analyses* form should look like the following figure:

Figure 4-13 hb Choosing Analyses Form



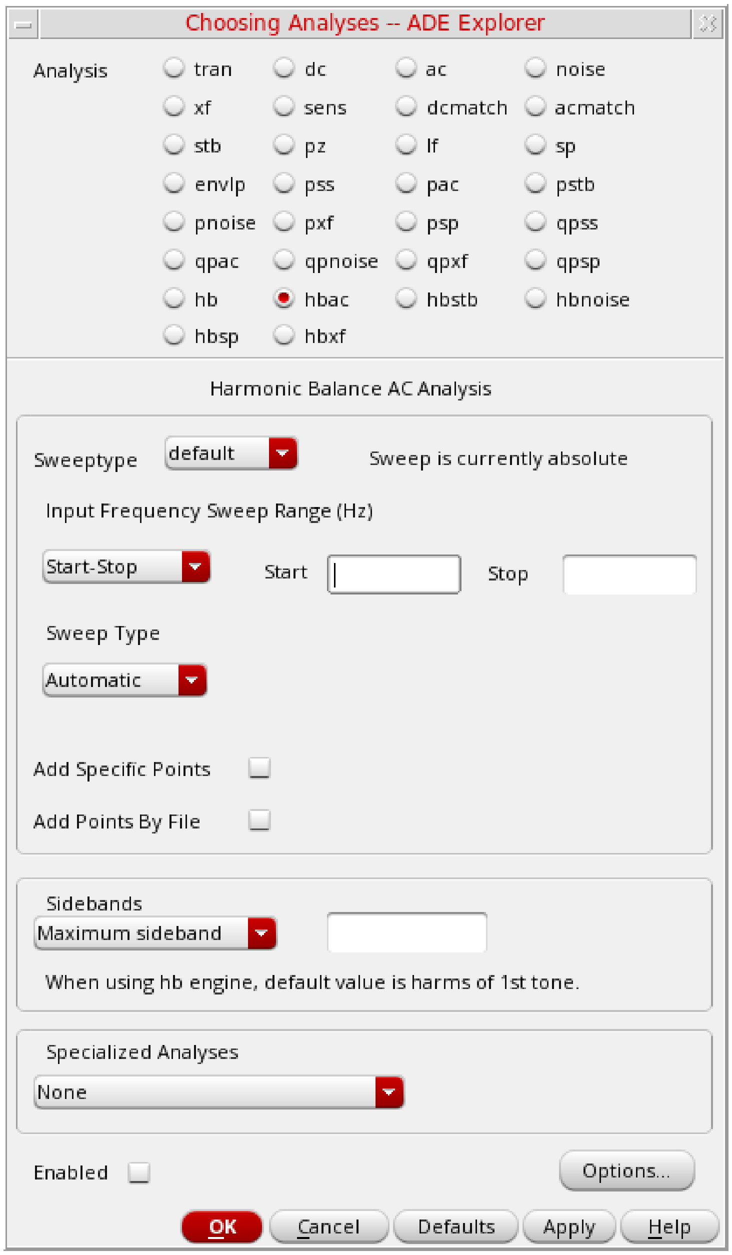
For more information on setting up the *Choosing Analyses* form, see Chapter 3: Frequency Domain Analyses: Harmonic Balance in the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide.

10. Click *Apply*.

Set up the HBAC Choosing Analysis form.

1. In the Choosing Analyses form, select *hbac*. The form changes, as shown below.

Figure 4-14 hbac Choosing Analysis Form



2. Leave the *Sweeptype* set to *default* (absolute).
3. Set the *Input Frequency Sweep Range*.

   Absolute takes the frequency range as specified with no frequency translation. Relative is also available where the input frequency can be shifted up or down in multiples of the PSS frequency. This is useful for having log sweeps above or below a PSS harmonic.

The frequency sweep range is always the input frequency range in hbac. Because hbac has the ability to calculate the outputs at different frequencies based on the nonlinearity caused by the LO, we choose which output frequencies to calculate in the Sidebands area of the Choosing Analyses form. Sideband is the name of the different output signals that are produced when mixing the input with harmonics of the LO signal.

a. Type `1.9001G` in the *Start* field.

b. Type `2G` in the *Stop* field.

c. Set the *Sweep Type* to *Linear*.

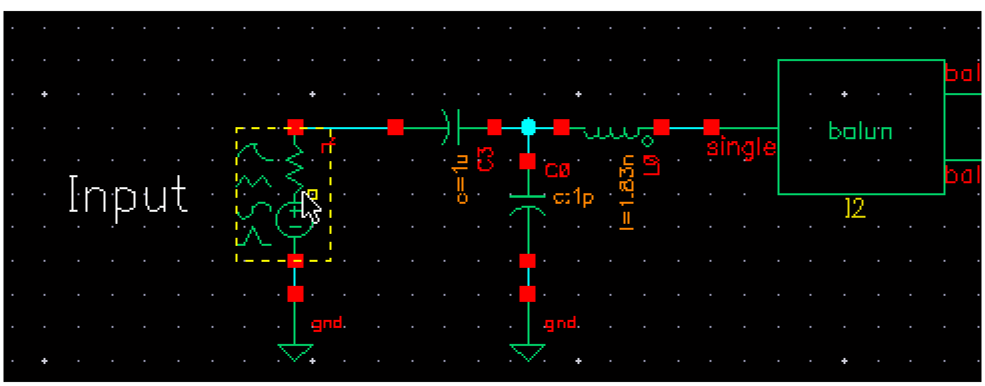
d. Set *Number of Steps* to `10`.

4. The amplitude for the hbac analysis is set in the input source (port *rf* with *PAC magnitude* =`1` in this case) in the schematic. 1 is convenient because it is 0 dBV. This allows direct conversion gain measurement by using the dB20 function when the output signal is plotted.

   Below are the steps showing how to set the *PAC Magnitude* in the schematic

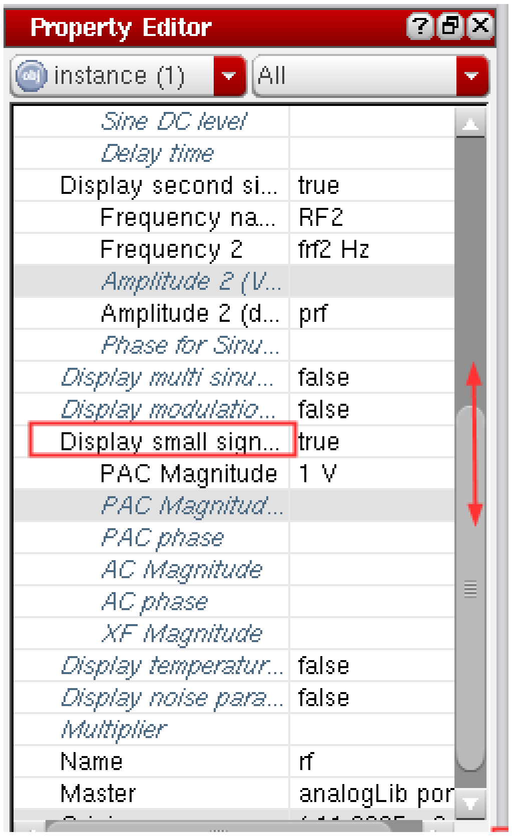
   a. To set the amplitude for hbac, select the *Input* port in the schematic.

Figure 4-15 Input Port on Schematic



   b. After you click on the port, refer to the *Property Editor* on the left side of the schematic. You can scroll through the *Property Editor* by moving the scroll bar on the right side of the *Property Editor*.

Figure 4-16 Property Editor



c. Select the *Display Small Signal Params* option near the bottom of the *Property Editor*, and set its value to *true*. This expands the *Property Editor* form.

   The value for *PAC Magnitude* should be set to `1 V`. If it is not, set it to `1 V`.

d. From the Schematic, select *Check - Current Cellview.* You do not have to save the design to simulate it. This allows "what if" analysis. If you like the results, you can then save the design. If you do not like the results, you do not have to save it.

   In the hbac *Choosing Analyses* form, you will be selecting the direct conversion IF sideband and the output with no frequency translation from the *Sidebands* section under *Select from Range.*

5. In the *Sidebands* section, select *Select from range*.

   Sidebands define the output frequencies to be calculated. In this setup, the direct conversion IF (1.9001G~2G-1*1.9G=100K~100M) is selected. In addition, hbac also calculates the outputs with no frequency translation (1.9001G~2G-0*1.9G=1.9001G~2G). You will be choosing these two sidebands.

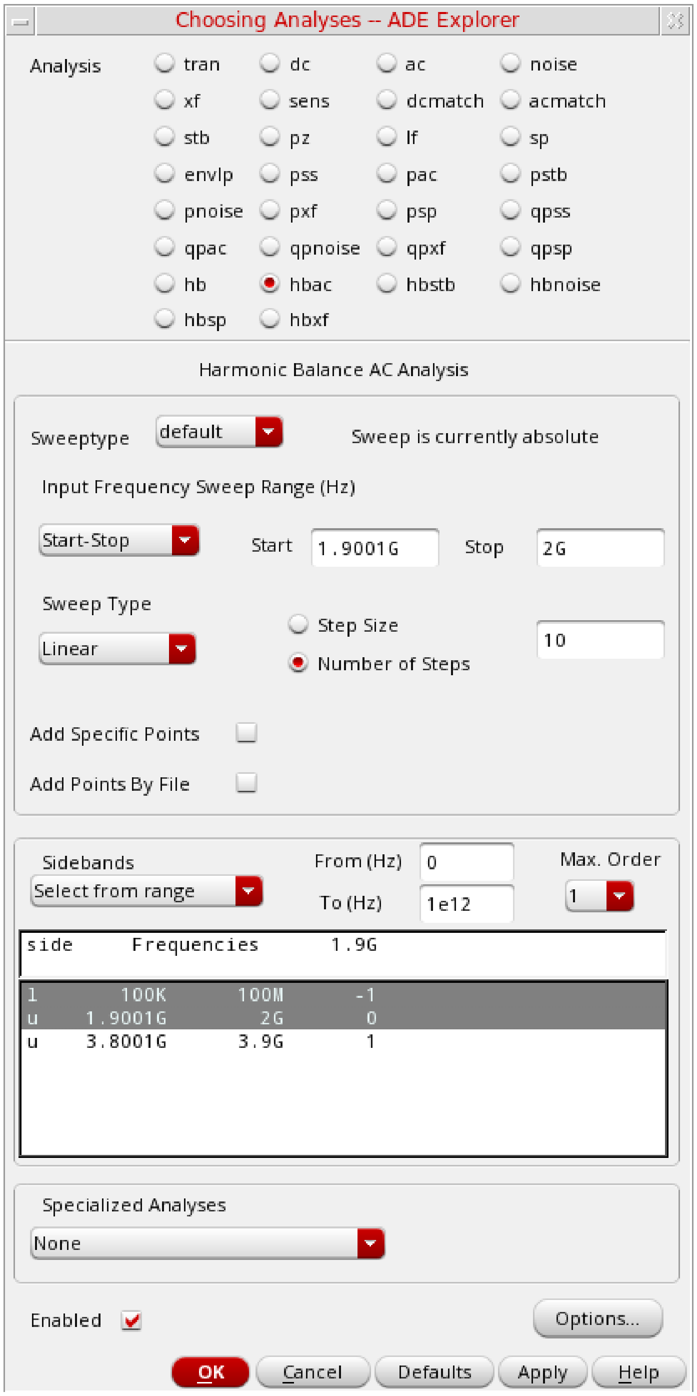
The levels that are produced at each node at all the selected frequencies are also calculated by the hbac analysis.

a. Highlight the *l 100K 100M -1* entry. This is the direct conversion IF sideband.

b. To select another sidebands, press and hold the *Ctrl* key, and select the *u 1.9001G 2G 0* entry. This is the output with no frequency translation.

The hbac *Choosing Analyses* form looks like the figure below:

Figure 4-17 hbac Choosing Analyses Form -Set up the hbnoise Choosing Analyses form

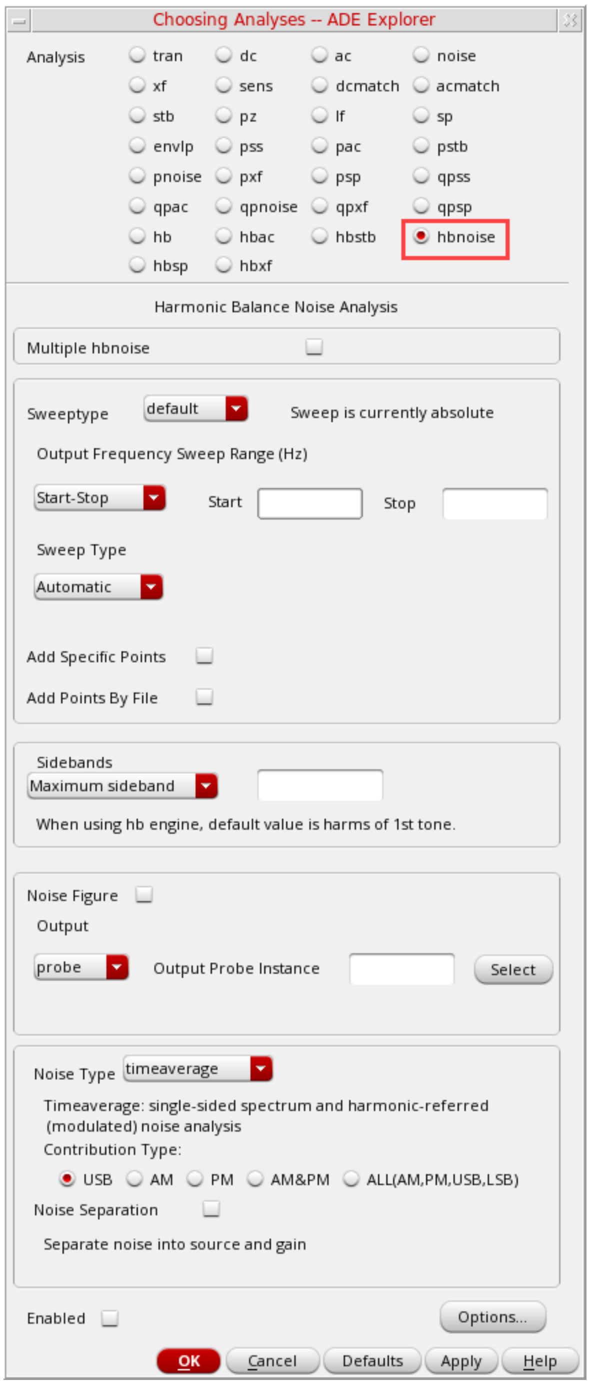


6. Click *Apply*.

Set up the hbnoise *Choosing Analyses* form, as follows:

1. In the *Choosing Analyses* form, select *hbnoise.* The form changes, as shown below.

Figure 4-18 hbnoise Choosing Analyses Form



2. Leave the *Sweeptype* set to *default*.
3. Set the *Output Frequency Sweep Range*.

   The *Frequency Sweep Range* fields are always the output frequency range in hbnoise. *Maximum sideband* specifies the highest harmonic of the LO we want to calculate the down converted noise from. In this case, noise through the 10th harmonic of the LO will be calculated.
4. Type `1K` in the *Start* field.
5. Type `100M` in the *Stop* field.
6. Select *Logarithmic* sweep.
7. Type `3` in the *Points Per Decade* field.
8. Leave the *Maximum sideband* field blank.

   Note: When left blank, the form uses the *number of harmonics* from the hb analysis.
9. Select the *Noise Figure* option.
10. In the *Output* section of the form, by default, the *Output* is set to *probe*. Click *Select* to the right of the *Output Probe Instance* field and select the source to the right of the label *Output* in the schematic window.

    When a port or a resistor is selected in this way, the noise of this component is excluded for the noise figure calculation. A port is a voltage source in series with a resistor as a single component and it is located in analogLib.
11. In the *Output* section, by default, *Input Source* is set to type *port*. If you want a noise figure calculation, you must select a port as the input. Click *Select* to the right of the *Input Port Source* field and select the source to the right of the label *Input* in the schematic window.

    In hbnoise, only output-referred and input-referred noise measurements are made, so both the input and output frequencies are declared in the Choosing Analyses form. In linear noise, the input-referred noise is the output-referred noise divided by the transfer function from input to output.

    In hbnoise, because we have the ability to calculate frequency conversion, the design input frequency range is chosen in the reference sideband section in order to get the correct input-referred noise and noise figure at the design input frequency. This can also be referred to as the passband frequency.

    In the *select from range* field, you will specify the input frequency range that goes with the output frequency range specified at the top of the form.

12. In the *Reference side-band label* section, choose *Select from list* from the drop-down list.

13. Select the *u 1.9G 2G 1* line.

Here is an example of how the reference sideband field is calculated:

suppose the reference sideband=k,

`|fin|=|fout|+k*fundamental frequency`.

Here, `fout=1K~100M` and the fundamental frequency is 1.9G.

`fin` should be `(1K~100M)+1.9G`,

thus the reference sideband `k` is `1`.

14. Select the *Do Noise* check box (this is the default).

15. Select *timeaverage* from the *Noise Type* drop down list.

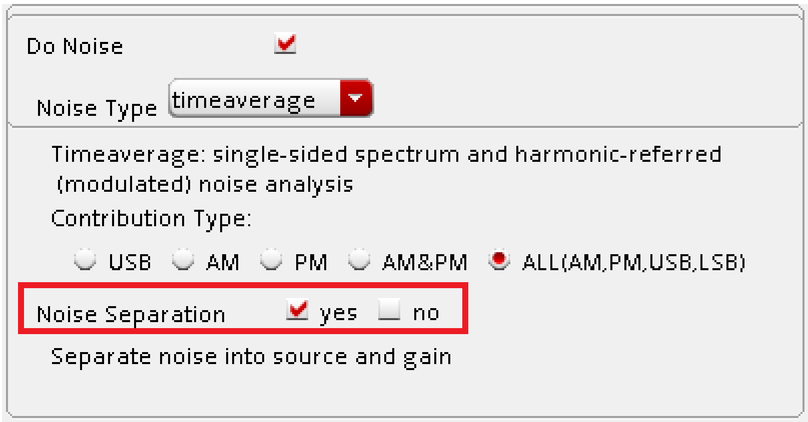
16. Select the *Contribution Type* to *All (AM,PM,USB,LSB)* and set *Relative Harmonic* to *0*.

When *Noise Type* is set to *timeaverage*, USB, LSB, AM and PM noise are directly available. You can choose, for example, PM noise alone, or you can select the ALL(AM,PM,USB,LSB) option, if you need all four types of noise.

When you select *AM*, *PM*, *AM&PM*, or *All(AM,PM,USB,LSB)*, a pop-up window appears and *Sweeptype* is set as `relative`. For db_mixer, the *Output Frequency Sweep Range* is set as `fout=1K~100M`, and the *Relative Harmonic* is set to `0`. Note that *AM* and *PM* are not available for relative harmonic *0*.

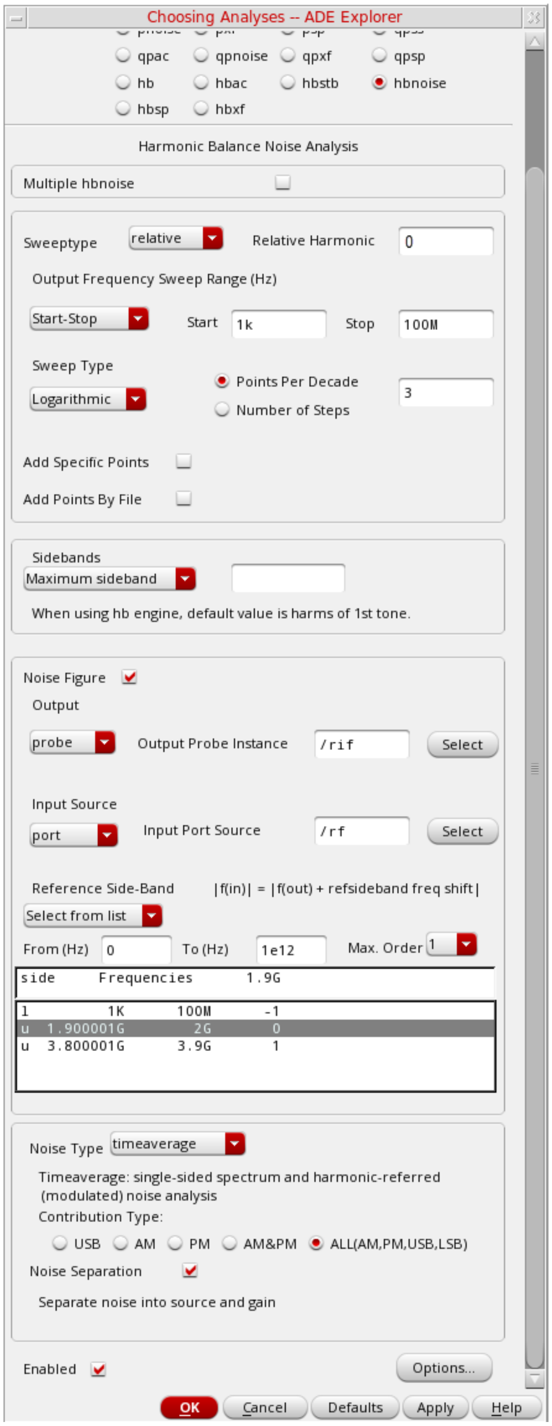
17. Select the *Noise Separation* check box.In addition to the total output noise, the individual noise contributions can be plotted when noise separation is selected. For more information on Noise Separation, see Chapter 3: Frequency Domain Analyses: Harmonic Balance in the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide.

Figure 4-19 hbnoise Noise Separation Check Box.



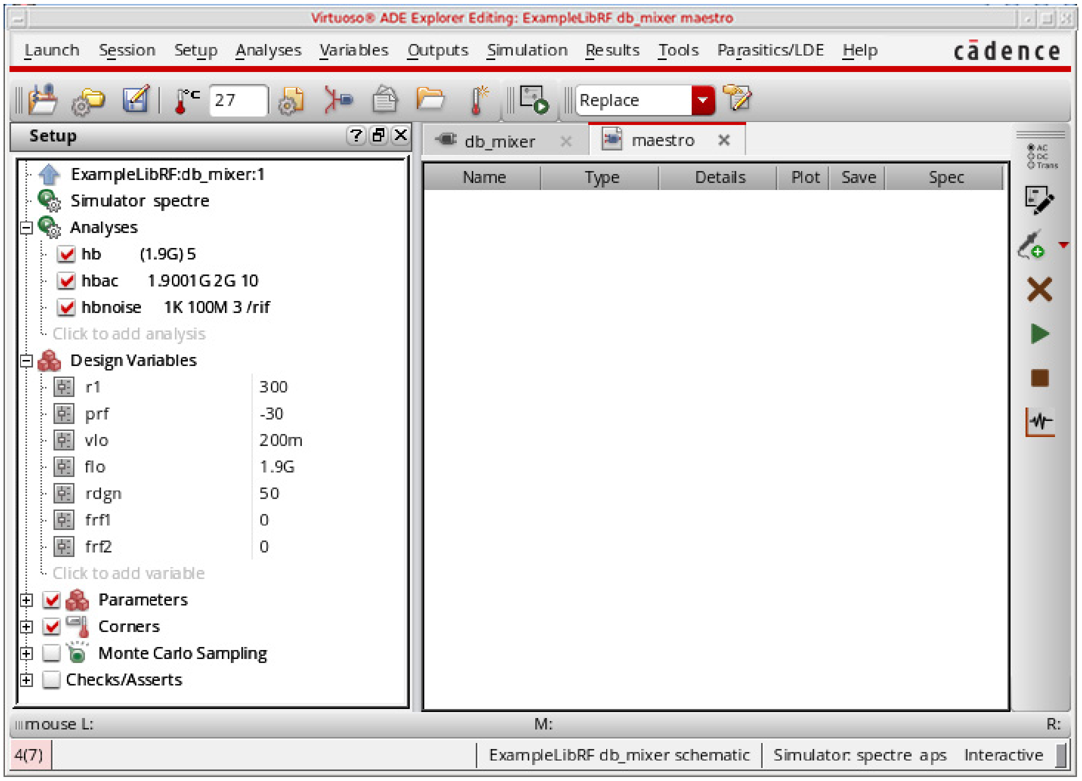
Your completed hbnoise *Choosing Analyses* form should look like the following figure.

Figure 4-20 hbnoise Choosing Analyses Form



18. Click *OK* at the bottom of the form.

Figure 4-21 ADE Explorer Simulation Window



Running the Simulation and Plotting the Results

Start the analyses by clicking the green arrow icon.  in ADE Explorer or in the Schematic Editor.

This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` logfile). When the analysis has completed, you may iconify the status window.

Next, you will plot the following:

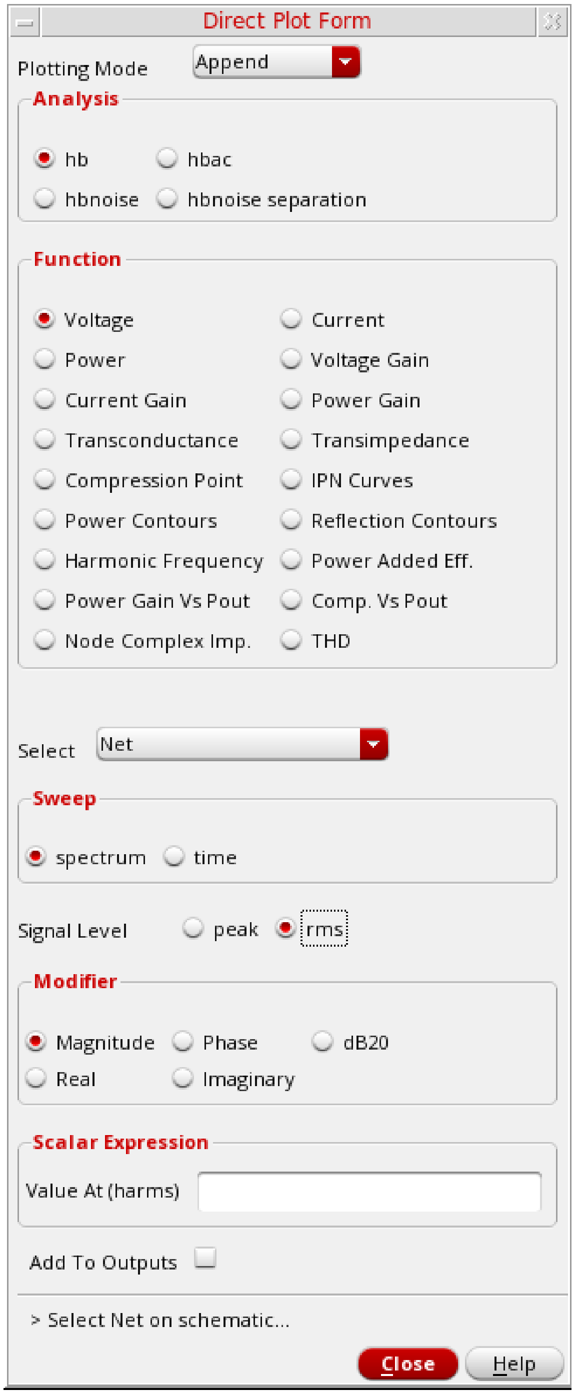
- Mixer Conversion Gain,
- RF to IF Isolation, LO to IF Leakage
- Noise Figure

Measuring Mixer LO to IF Leakage

1. In ADE Explorer, select *Results - Direct Plot -Main Form*.

The *Direct Plot Form* is displayed.

Figure 4-22 hb Direct Plot Form



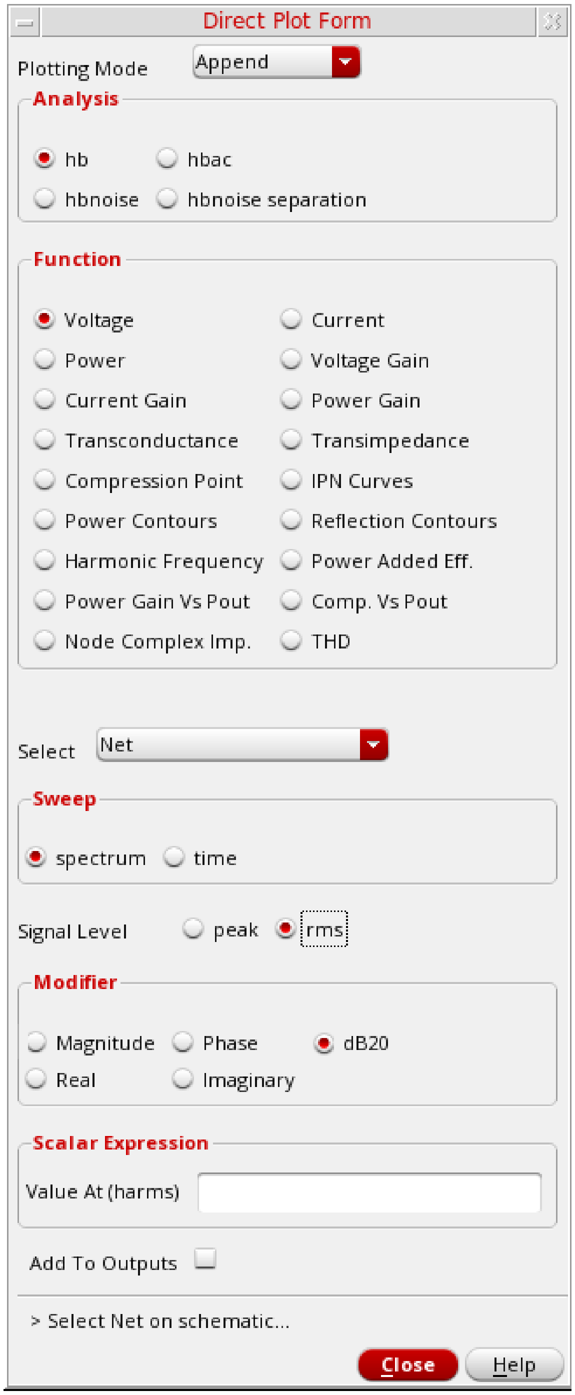
Note that there is an entry for hb noise separation. This is because you have selected *Noise Separation* option on the hb Choosing Analyses form.

First, plot the output spectrum

1. In the *Analysis* section, select *hb*.
2. In the *Function s*ection, select *Voltage*.
3. *Select* `Net` in the center of the form. (This is the default. You can also select *differential nets*).
4. In the *Sweep* section, select *spectrum* (this is the default).
5. In the *Signal Level* section*,* select *rms* (the default is *peak*).
6. In the *Modifier* section, select *dB20*.

   The *Direct Plot Form* should look like the following:

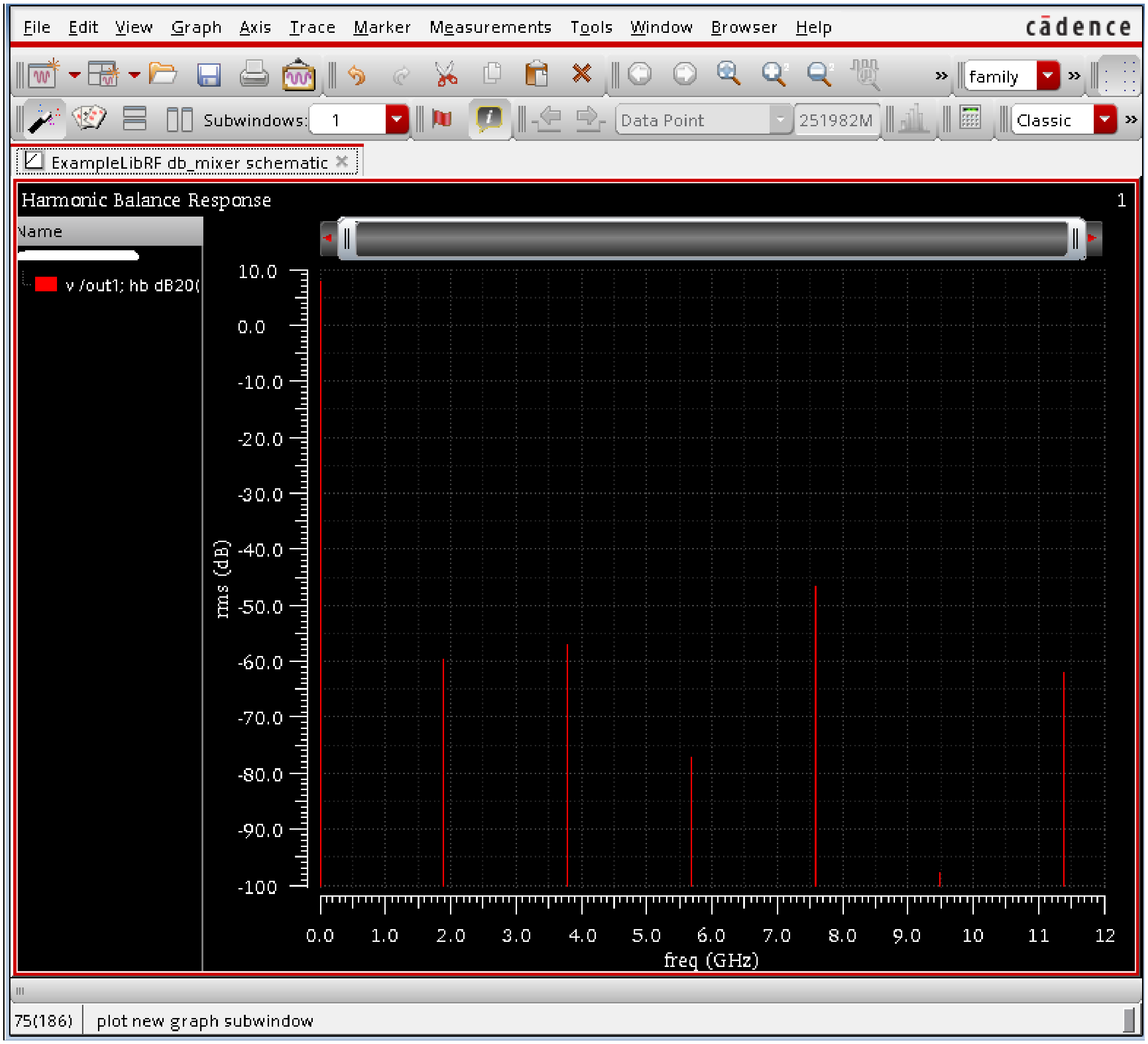
Figure 4-23 Completed Direct Plot Form - Measuring Mixer LO Leakage



7. Select the *out1* net in the schematic. It is located just below the *Output* label.

   The waveform window is displayed.

Figure 4-24 Harmonic Balance Response of /out1 net

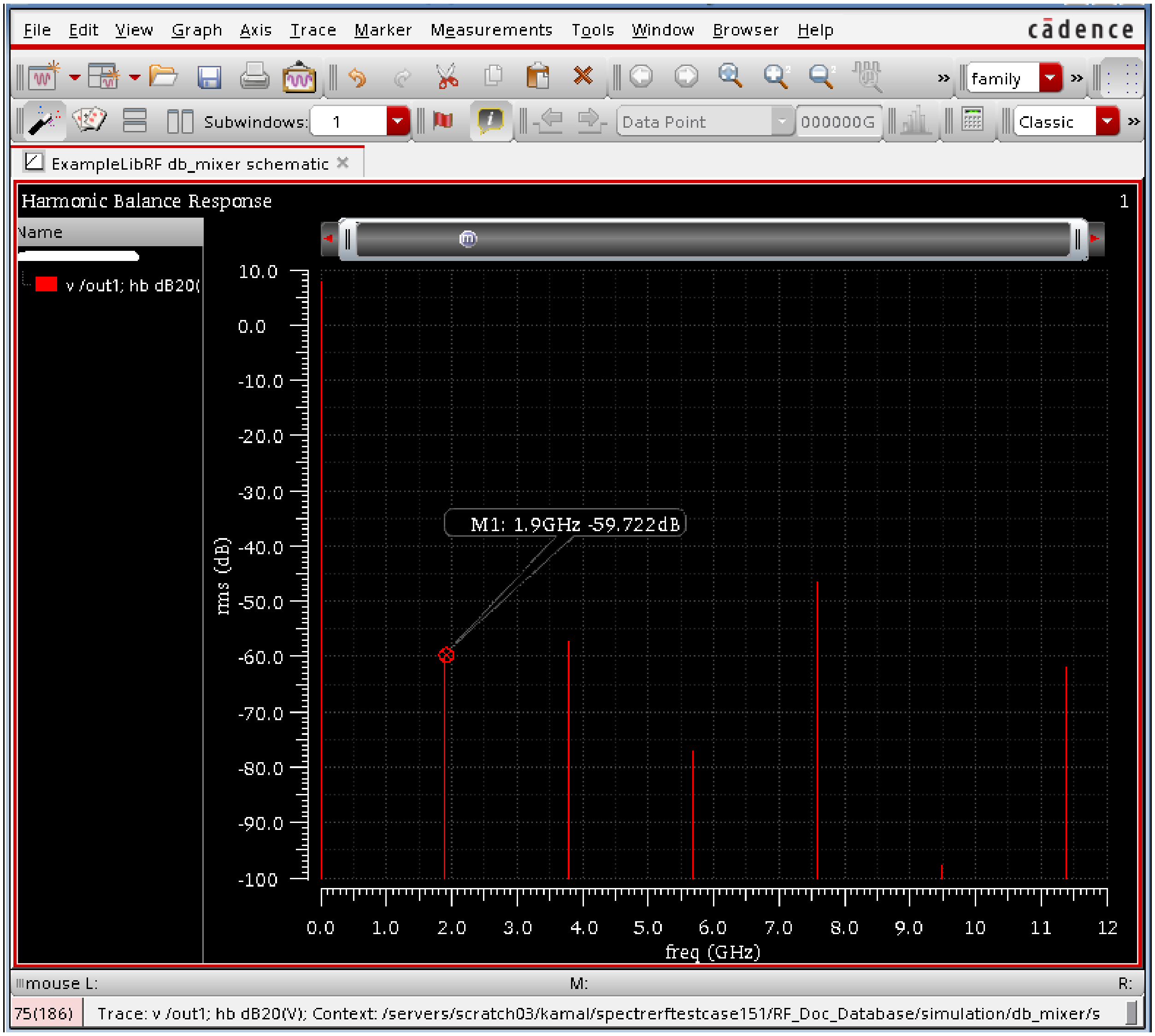


Measure LO leakage on the IF Output

8. Move the mouse cursor to the tip of the spike at *1.9GHz* and type `m`.

A marker appears on the plot. Click and hold on the marker readout and move it to a place where you can see all the harmonics, then release the mouse button. The LO amplitude is about *-59.72 dB*.

Figure 4-25 Measuring LO leakage on the IF Output



Plot the conversion gain and RF to IF isolation

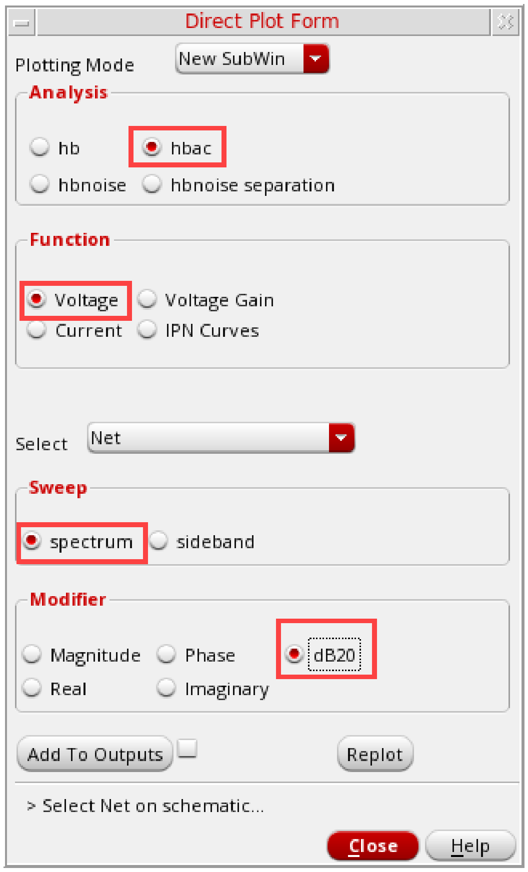
In the *Direct Plot Form*:

1. Change *Plotting Mode* to *New SubWin*.
2. In the *Analysis* section, select *hbac*.
3. In the *Function* section, select *Voltage* (this is the default).
4. In the *Sweep* section, select *spectrum* (this is the default).

5. In the *Modifier* section, select *dB20*.

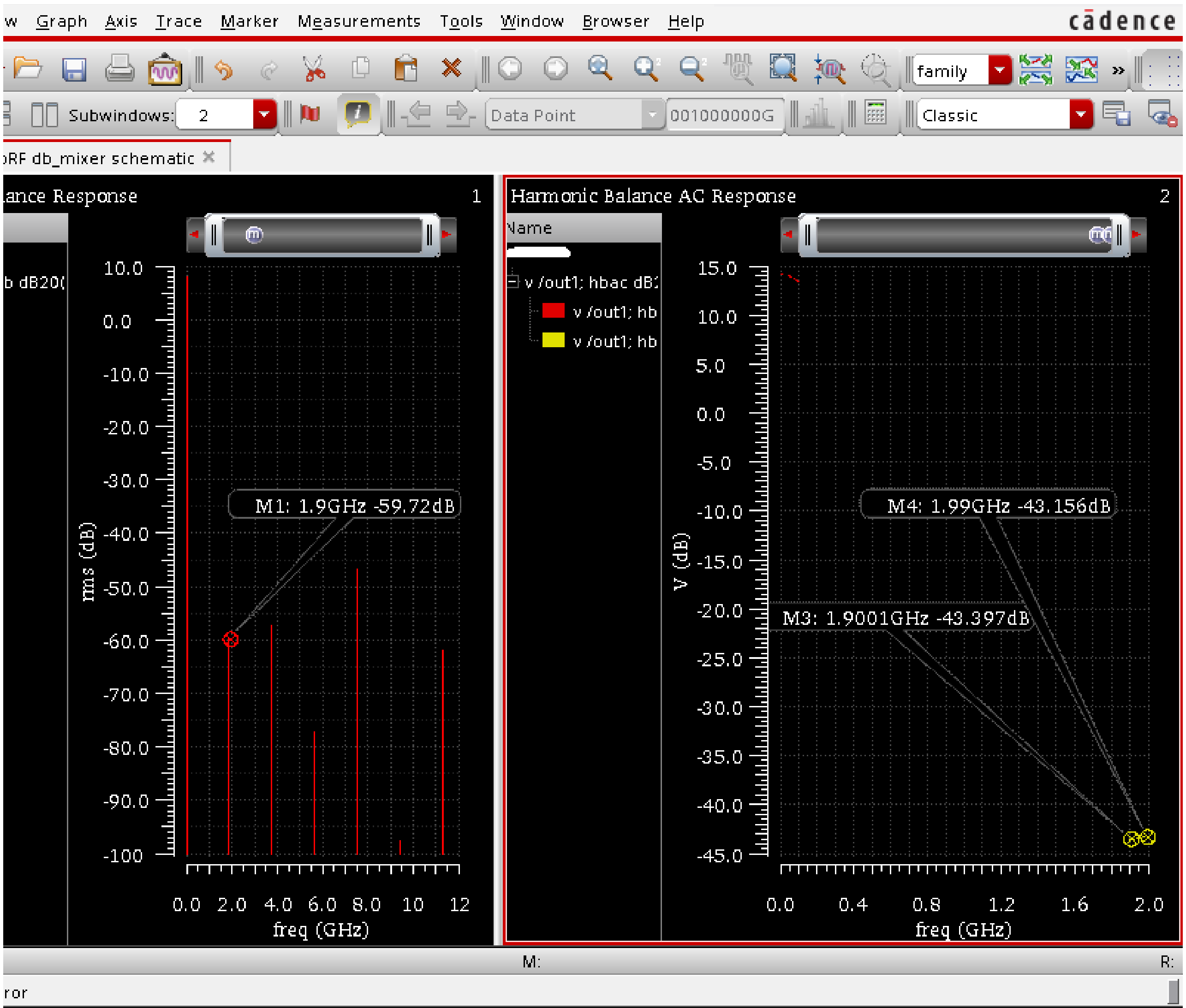
   The *Direct Plot Form* looks like this.

Figure 4-26 Direct Plot Form - Plotting Conversion Gain and RF to IF isolation



6. Click *Replot*. The waveform window updates, as shown below.

Figure 4-27 Conversion Gain and RF to IF Leakage



Look at the bottom subwindow for the HBAC results. The mixer circuit is deliberately very slightly unbalanced. In the HBAC *Choosing Analyses* form, you defined an input frequency range to be swept. HBAC calculates the amplitudes of the output signals at the frequencies you chose in the sidebands section of the *Choosing Analyses* form.

7. Note that the trace near 2GHz is about *-43.4dB*, and measures the gain from input to output with no frequency conversion. This is due to RF to IF leakage.

   If you want, place a marker at that location by moving the mouse cursor to the tip of the spike at 2GHz and typing `m`. Click and hold on the marker readout and move it to a place where you can see all the sidebands, then release the mouse button.

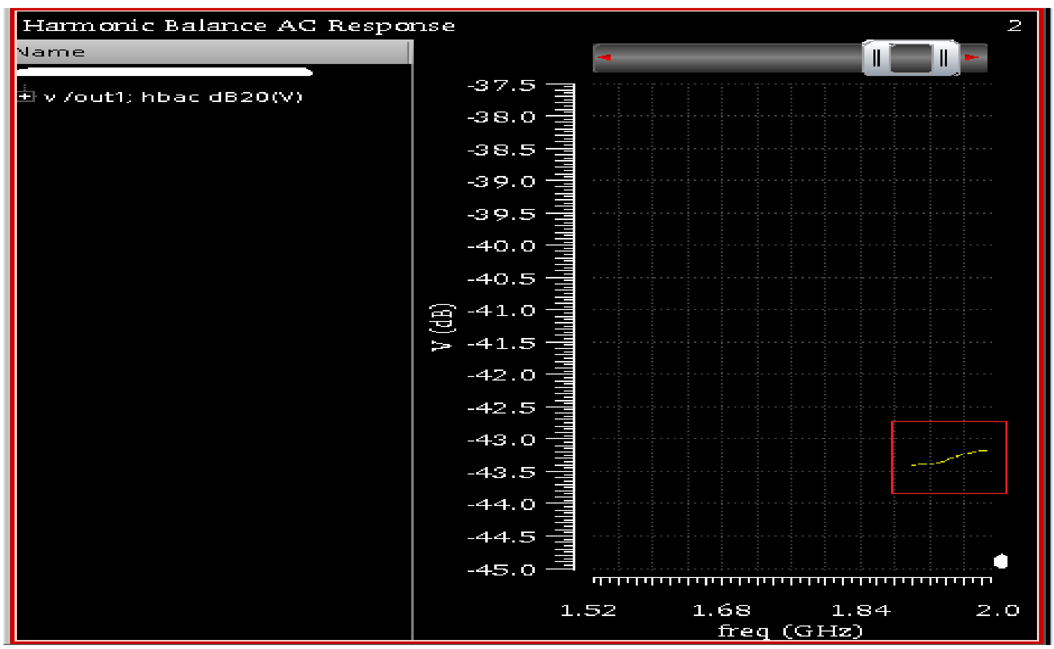
Next, move the cursor to 1.9001GHz and just below -40dB, read the value of the cursor marker. In this case, the RF to IF isolation is about *43.4 dB*. If you want, place a marker at that location as well.

Measure Voltage Gain vs Output Frequency

1. **Optional**: Remove the markers from the Waveform window by choosing *Marker - Delete All* or by pressing the *Ctrl+E* bindKey.

   You may need to zoom in close to the waveform. To do that, click and hold the right mouse button and drag a square outline around the waveform, as shown in the next figure.

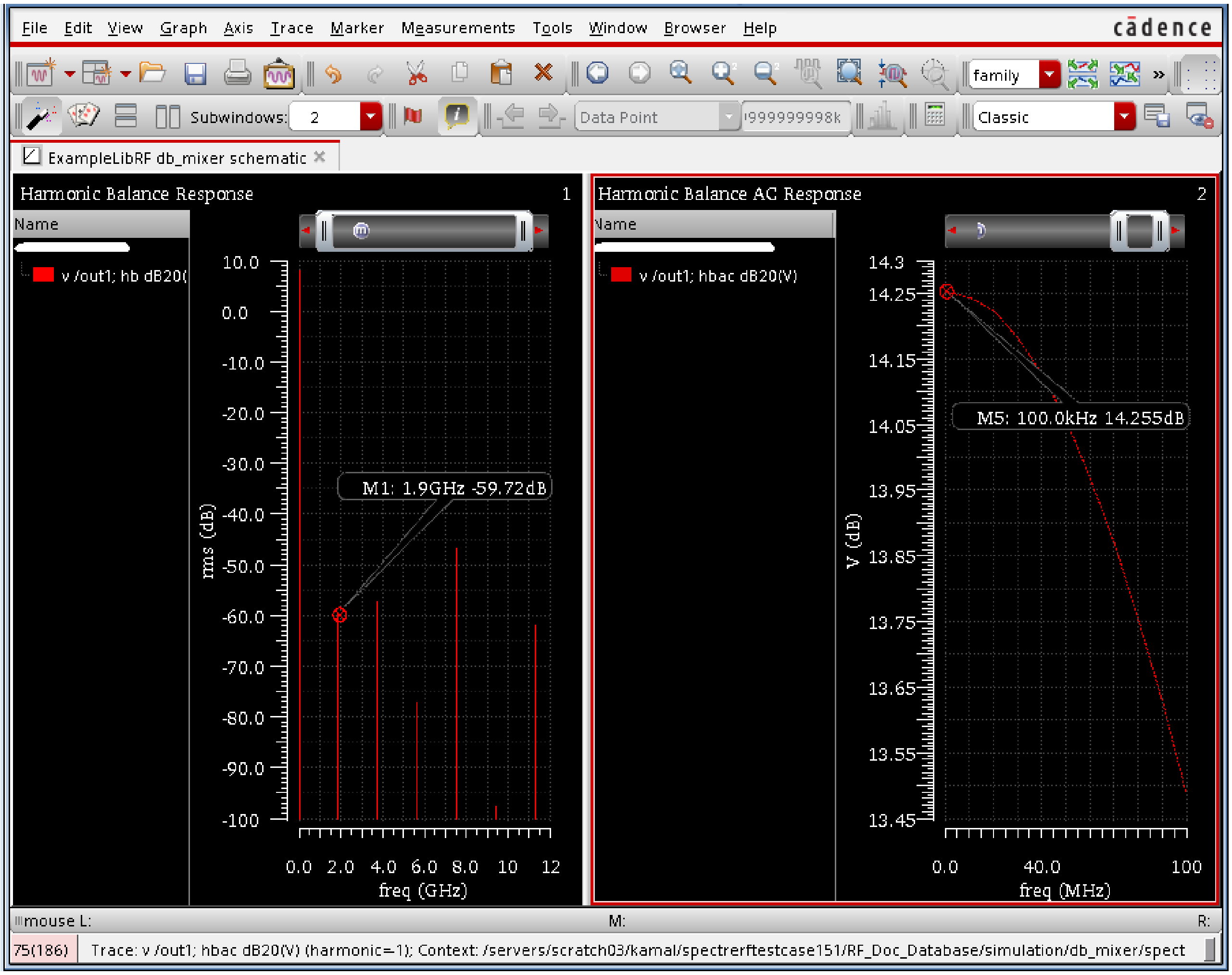
   Figure 4-28 : Choosing Sideband of Interest



2. Delete the result near 2GHz. To do this, in the waveform window, click the trace near 2GHz and just below -40 dB. Press the *Delete* key. To see the remaining trace, you may need use the bindkey *f* to fit (resize) the window so that the entire plot is shown.

   The waveform window redraws, as shown below.

Figure 4-29 Voltage Conversion Gain vs. Output Frequency



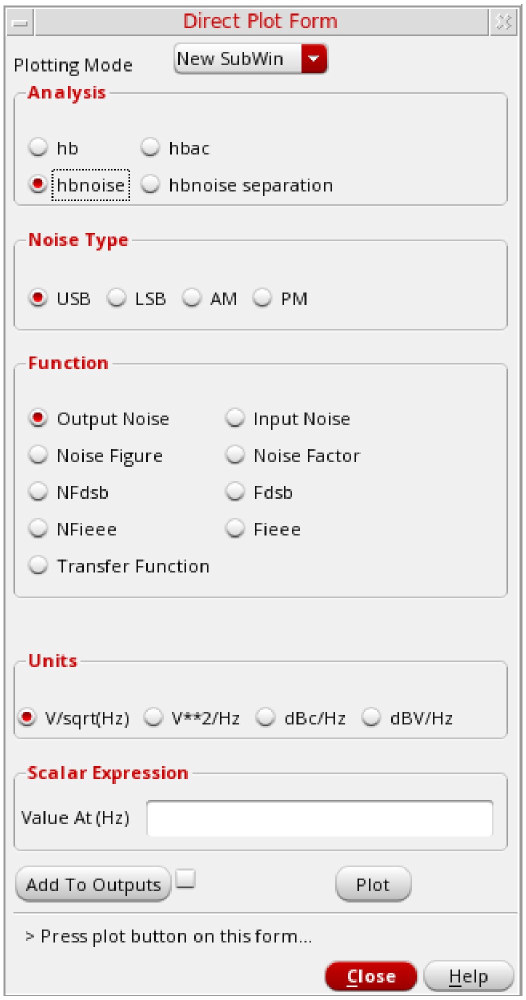
The lower subwindow displays the plot of the Voltage Conversion Gain versus the Output frequency. It rolls off at higher frequencies due to the degraded input match as the input moves above 1.9GHz. Since in the HBAC *Choosing Analyses* form, you set the *Sweep Type* to be *linear*, the plot has linear spacing on X-axis.

If you hold the cursor over the curve at 100KHz, you can read the low frequency conversion gain off the plot as shown in the above figure. It is *14.25dB*.

Plotting Mixer Noise Figure

1. In the *Direct Plot Form*, select *hbnoise*.

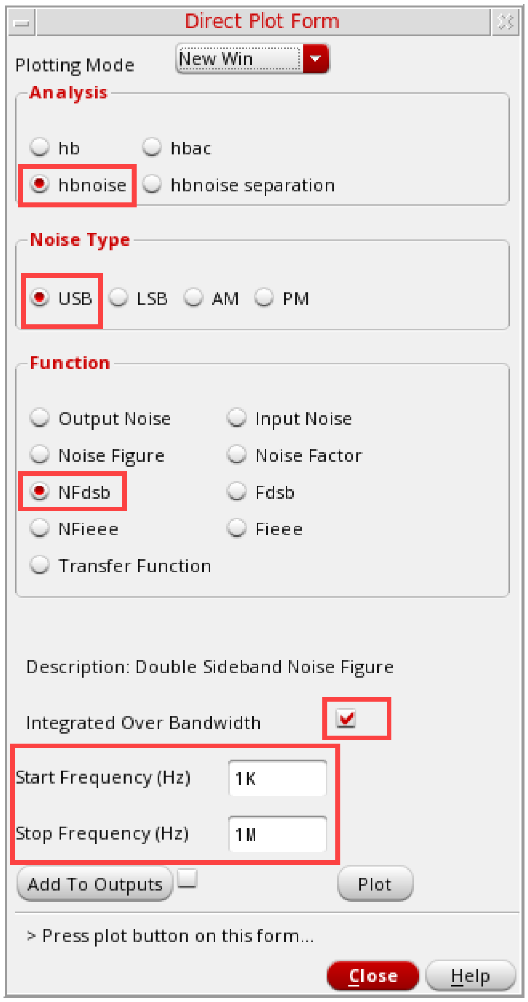
Figure 4-30 Direct Plot Form - Plotting Mixer Noise Figure



2. Change the *Plotting Mode* to *New Win.*
3. Select *USB* from the *Noise Type* section (this is the default).
4. Select *NFdsb,* in the *Function* section. NFdsb is double sideband noise figure. The form will change, as shown in the figure below
5. Select *Integrated Over Bandwidth*.
6. Type `1K` in the *Start Frequency (Hz)* field and `1M` in the *Stop Frequency (Hz)* field.

   The Direct Plot Form should look like below.

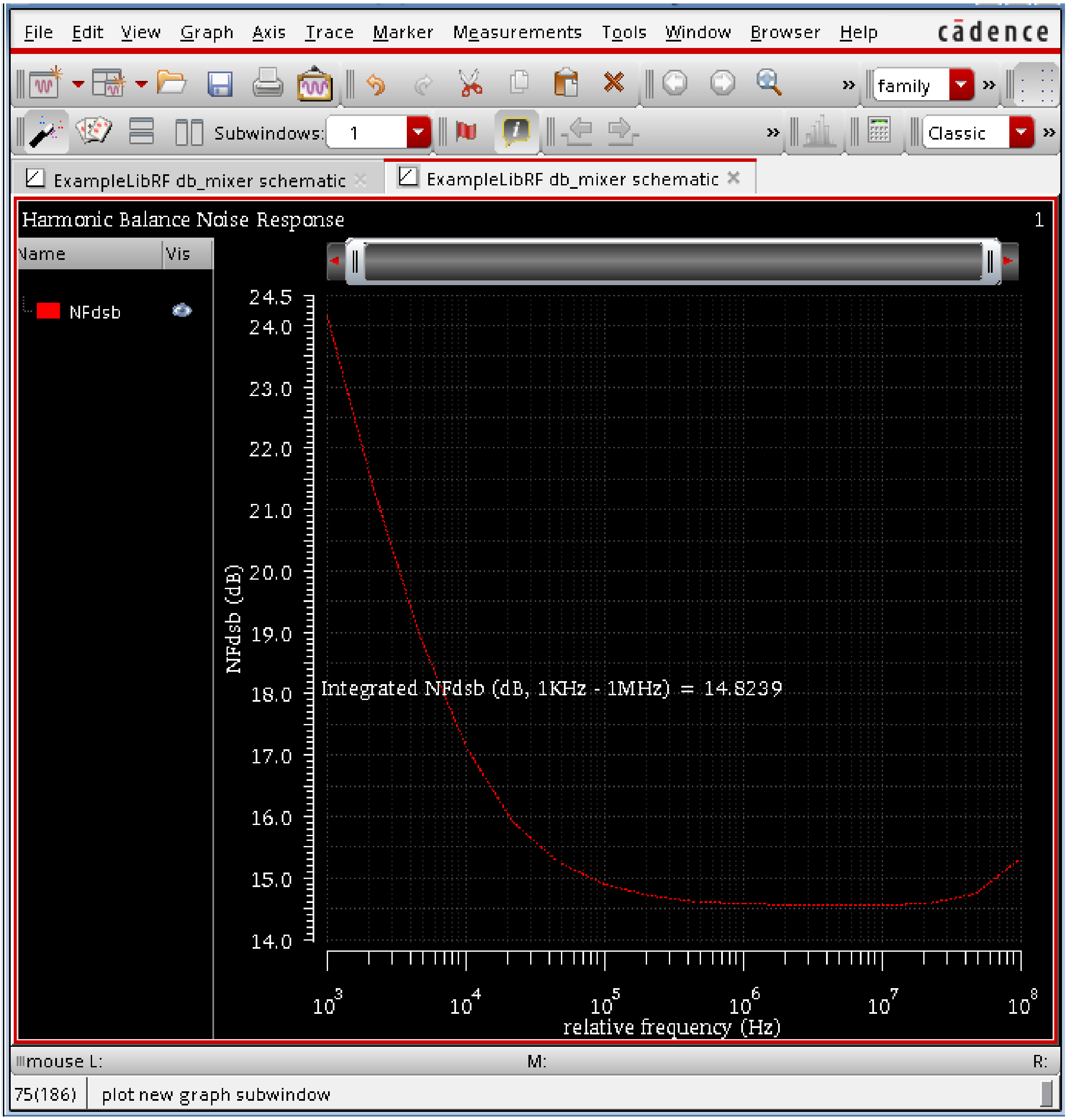
Figure 4-31 Direct Plot Form DSB Noise Figure Setup



7. Click *Plot*.

   The Integrated Noise Figure Plot is shown below:

Figure 4-32 Integrated Noise Figure Plot



To determine the noise figure at different frequencies, move the cursor along the noise figure curve in the waveform window.

Next, you will plot hbnoise separation. When you want to understand the specifics about your noise problem, you can use hbnoise separation. The Noise Summary is not able to differentiate how much noise is coming from the noise source and how much from the transfer function. With noise separation, you can attempt to decrease the output noise by decreasing the noise sources with different device dimensions or by decreasing the transfer function by choosing alternative circuit architectures. When *Noise separation* is set to *yes* in the *Choos*ing *Analyses* form, the hbnoise separation feature is included during the simulation and the corresponding results are saved.

8. In the *Direct Plot Form*, select *hbnoise separation* in the *Analysis* section.

9. In the *Function* section, select *Sideband Output.*

Sideband Output plots the noise contribution of selected sidebands to the output. Different sideband numbers measure the noise from different noise frequencies when mixed with different harmonics of the hb analysis. In this example, noise frequencies near zero Hz and the noise that mixes from just below the first and second harmonics of the LO are selected.

10. In the *Signal Level* section, select *V/sqrt(Hz).*

11. Choose *dB20* in the *Modifier* section.

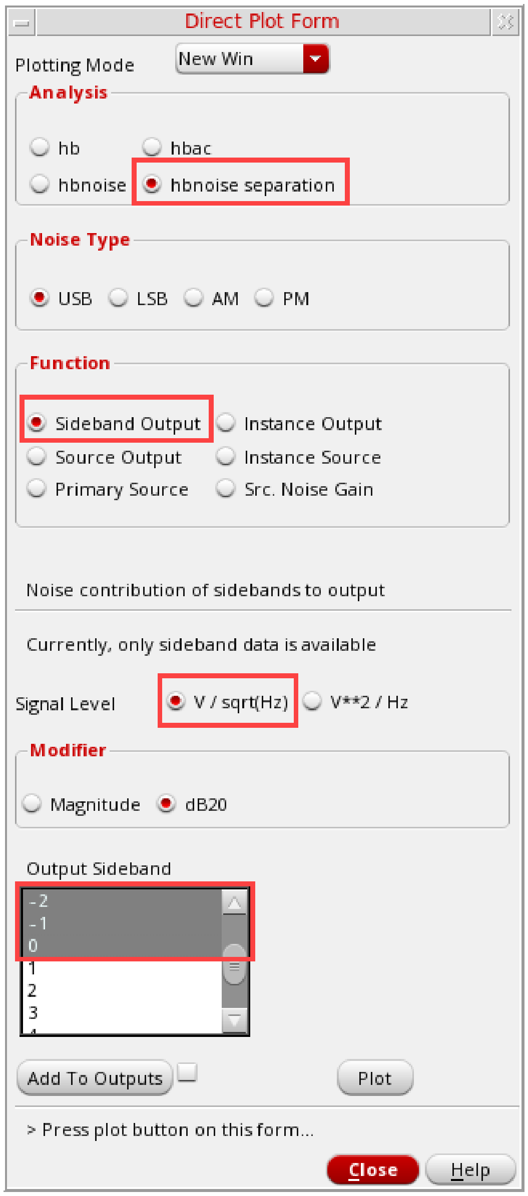
12. Select the *0, -1, and -2* in the *Output Sideband* section. This will plot the noise contribution of the *0, -1, and -2* sidebands to the output. This allows the identification of which noise frequencies are causing the largest noise contribution at he output of the circuit.

Tip

You can hold cursor over the *0* output sideband and slide the cursor to include the *-2* output sideband.

The Direct Plot Form should look like below.

Figure 4-33 Direct Plot Form for hbnoise Separation



13. Click *Plot*. The hbnoise separation is plotted, as shown in the figure below. Note that the harmonic numbers plotted are in the legend on the left side of the screen. HBnoise Separation, 0, -1, -2 Sideband.

Figure 4-34 HBnoise Separation, 0, -1, -2 Sideband

14. Click the + sign to the left of *output noise* in the legend on the left side of the waveform window. The legend expands. You can see that the *0 Output sideband* corresponds to the green trace. The *0 Output sideband* contributes the most noise and is the worst noise offender. Next, plot *Instance Output* for the largest noise contributor above. (The zero sideband) This will show which components contribute the most noise at the output.

a. In the *Direct Plot Form*, select *Instance Output* in the *Function* section.

b. Set the *Signal Level* to *V/sqrt(Hz)* and the *Modifier* to *Magnitude* .

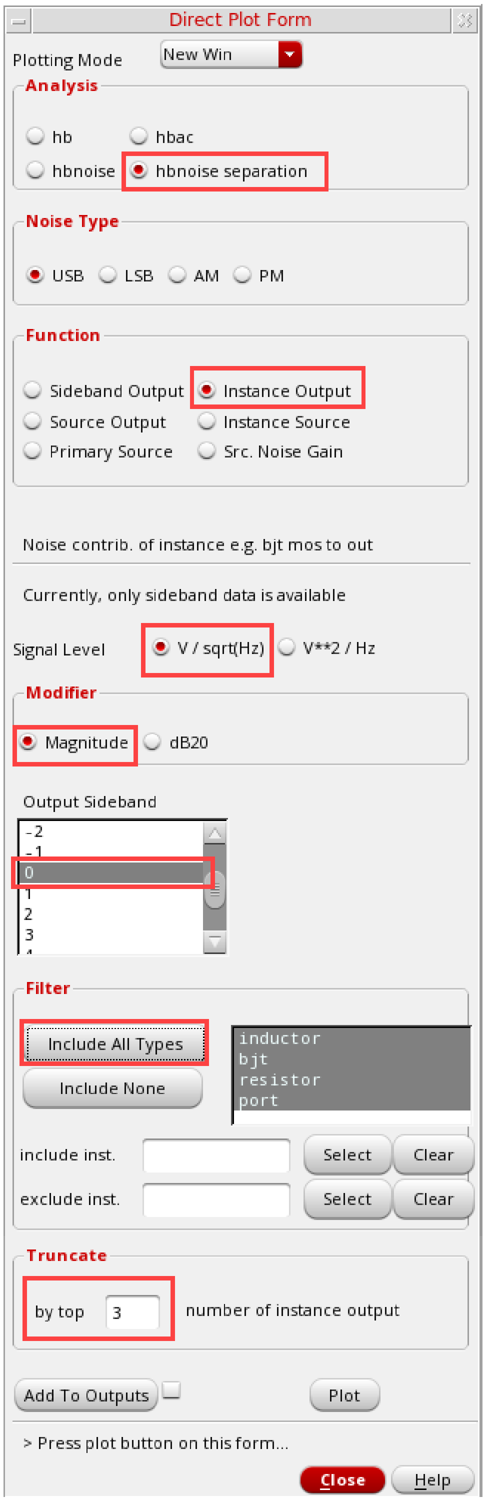
c. Select the *0 Output Sideband*.

d. In the *Filter* section, select *Include All Types.*

e. In the *Truncate* section, *type* `3` in the by top text field.

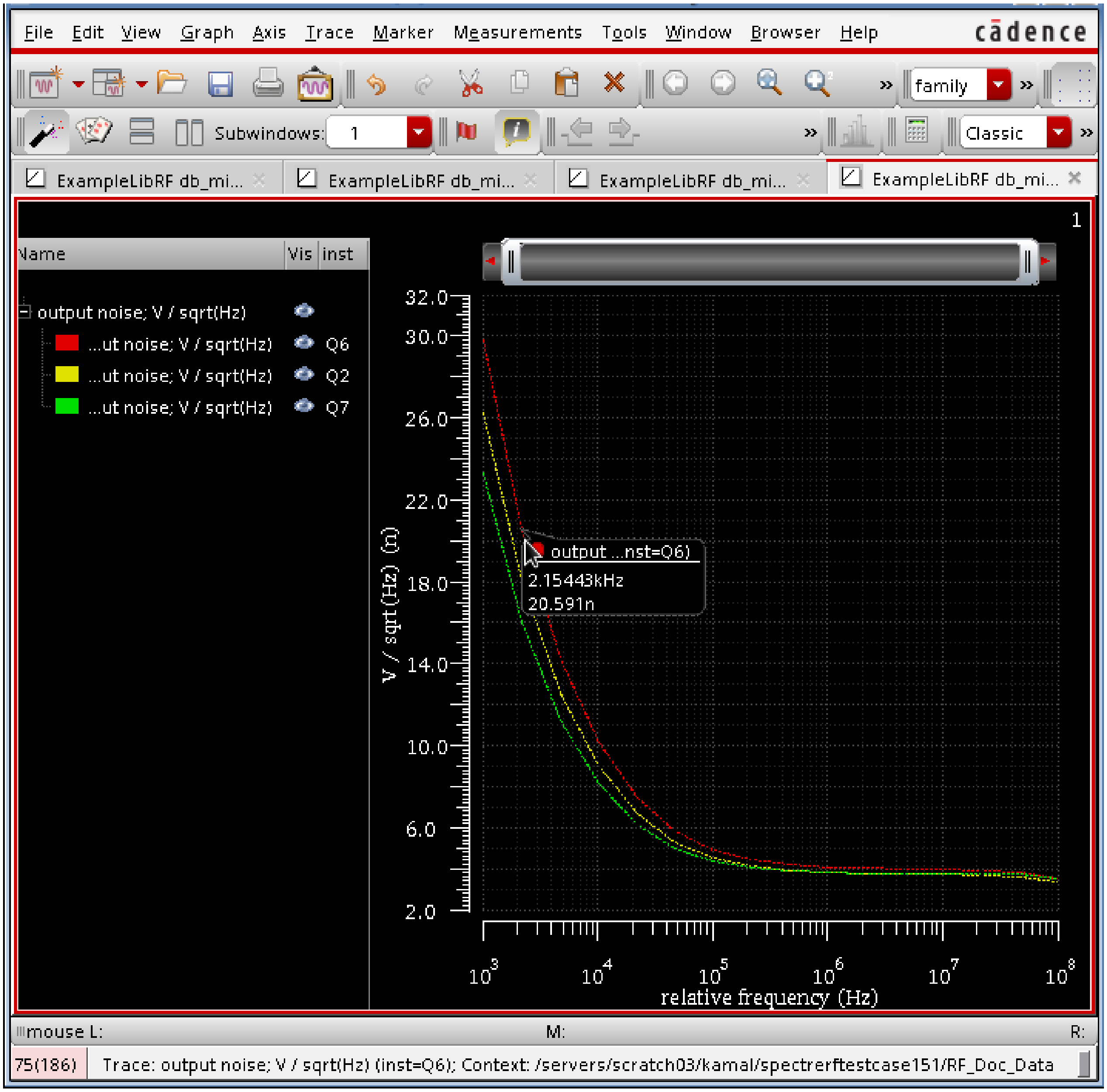
The *Direct Plot Form* should look like the one below.

Figure 4-35 Direct Plot Form for Plotting Instance Output for Noise Contribution of instances



15. Click *Plot*. The plot showing the top three noise contributors is displayed.

Figure 4-36 Output Noise Top Instance Contributors



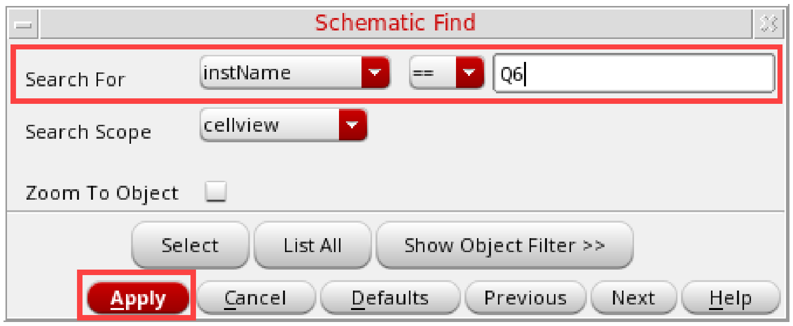
Click the + sign to the left of *output noise* in the legend on the left side of the waveform window. The legend expands to show the top three instance noise contributors.

1. If you move your mouse cursor over one of the traces, you can read the instance from the cursor readout and identify which instance contributes the most amount of noise.

Above, the red line represents the noise contribution from instance Q6 (legend shown), the yellow line below represents the noise contribution from instance Q2, and the bottom green line represents the noise contribution from instance Q7.

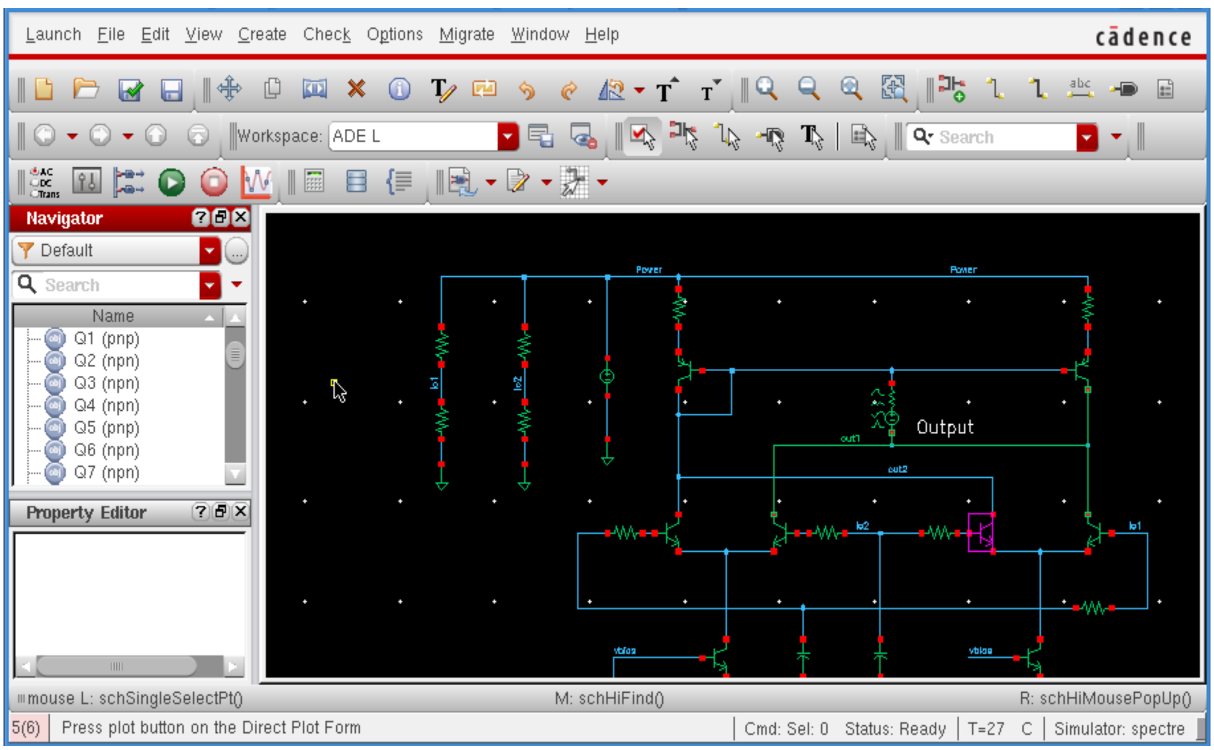
2. To find Q6 in the schematic, choose *Edit - Find* in the schematic. The *Schematic Find* form is displayed. Search for *instanceName == Q6* in the *cellview* and click *Apply*.

Figure 4-37 Schematic Find Form



Q6 is highlighted in the db_mixer schematic below in pink, as shown below.

Figure 4-38 Q6 Highlighted in dbmixer Schematic



3. Next, plot the Source Output so you can see which individual noise sources within the circuit are contributing the most noise. In the *Direct Plot Form*:

   a. In the *Function* section, select *Source Output.*

   b. In the *Signal Level* section, select *V/sqrt(Hz).*

   c. In the *Modifier* section, select *Magnitude*.

   d. Leave the *Output Sideband* at `0`.

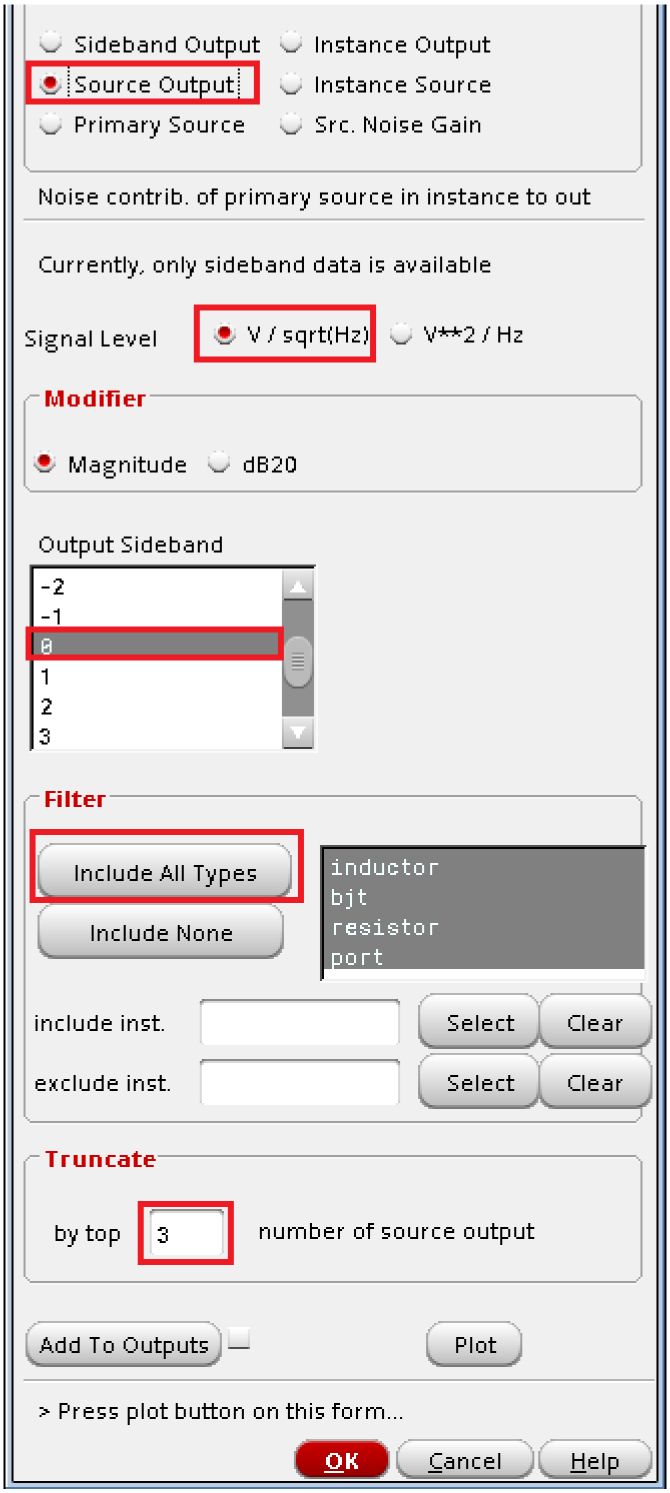
   e. In the *Filter* section, select *Include All Types*.

   f. In the *Truncate* section, *type* `3` in the *by top number of source output* text field

   g. The *Plotting Mode* should still be set to *New Window*.

   The *Direct Plot Form* should look like the one below.

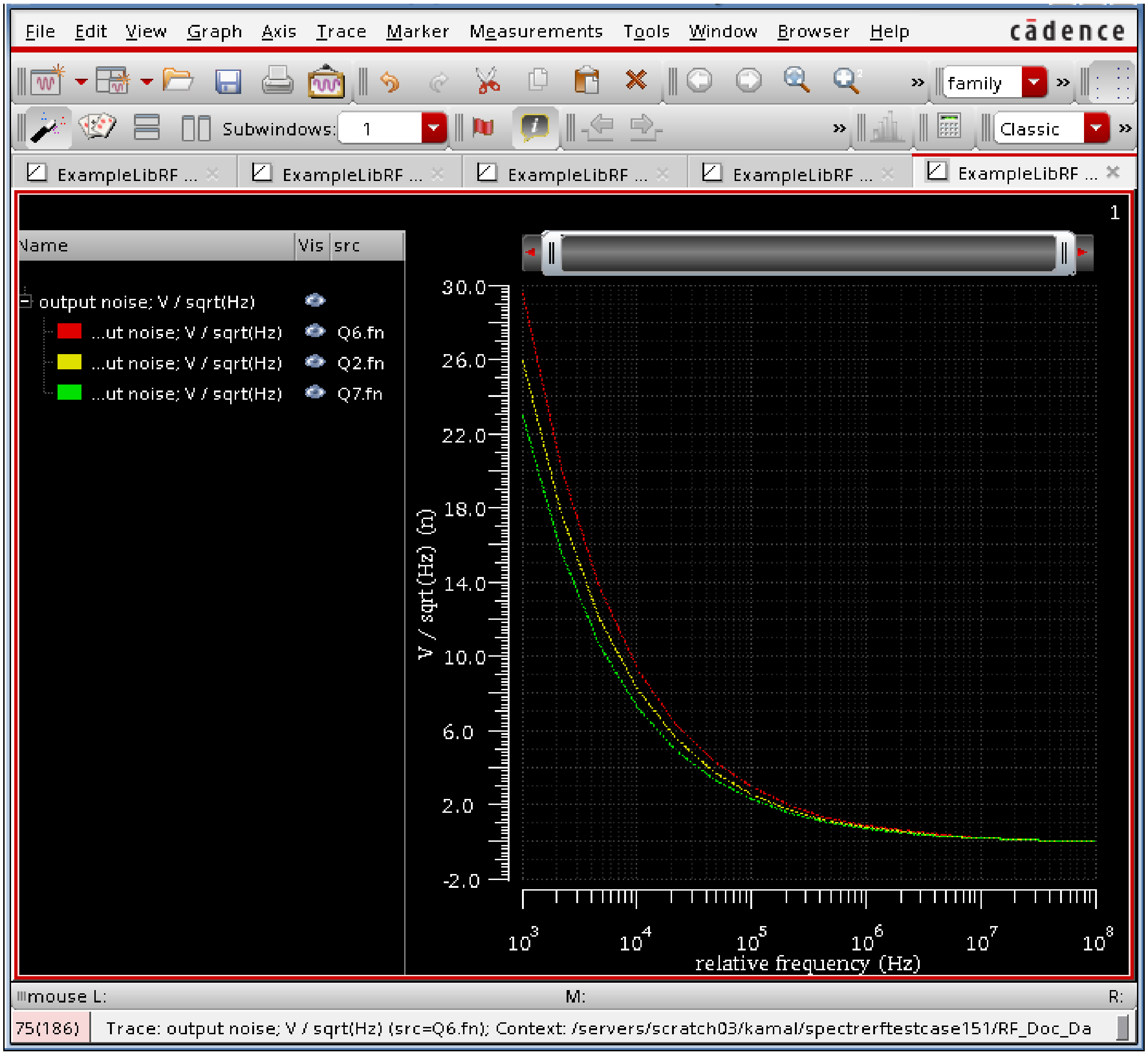
Figure 4-39 Direct Plot Form for Plotting by Top 3 Source Noise Contributors



4. Click *Plot* to see which individual sources within the circuit are making the most noise.

5. In the waveform window, click the + sign to the left of *output noise* in the legend on the left side of the waveform window. The legend expands to show the top three source noise contributors.

Figure 4-40 Top 3 Source Noise Contributors



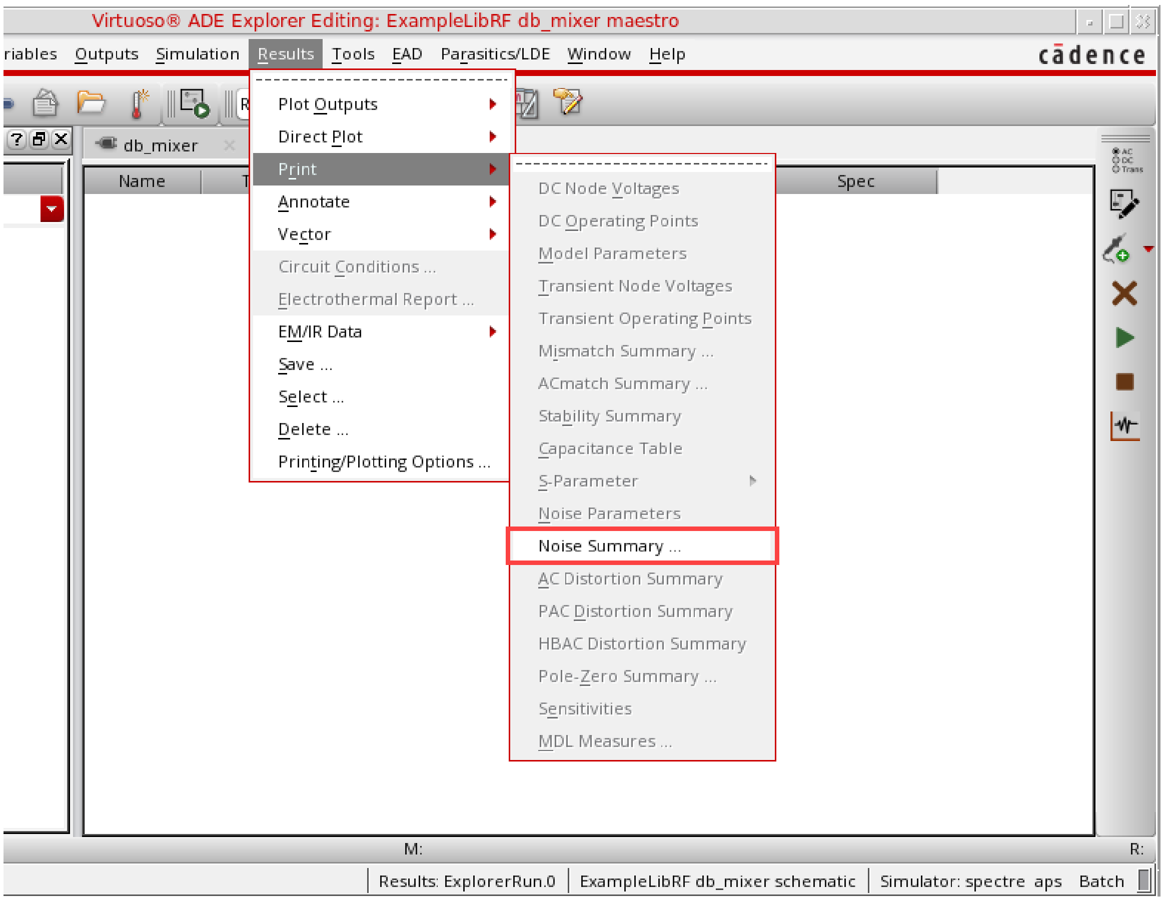
Note that the top three Source Output contributors are displayed in the legend on the left side of the graph: *Q6: fn*, *Q2: fn*, and *Q7:fn*. *fn* refers to flicker noise. Refer to the Noise Summary section in Chapter 3: Frequency Domain Analyses: Harmonic Balance of the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide to get a description of the parameter names for the different device types.

6. In the *Direct Plot Form*, click *Cancel*.

7. In the waveform window, choose *File - Close All Windows.*

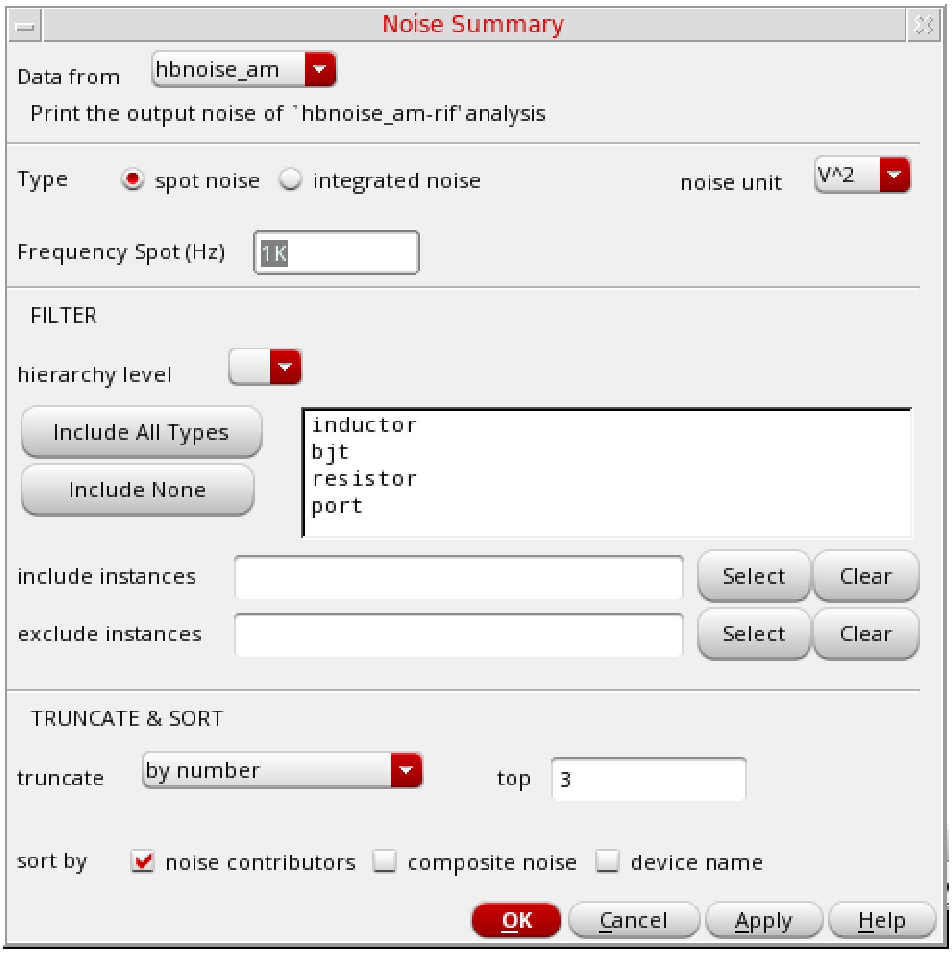
8. Next, look at the top noise contributors using the *Noise Summary Form*. In ADE Explorer, select *Results - Print - Noise Summary.*

Figure 4-41 Invoking Noise Summary Form



The Noise Summary window is displayed, as shown below.

Figure 4-42 Noise Summary Window



9. In the *Noise Summary* window, select *hbnoise_src* from the *Data from section*. This prints the output noise of the hbnoise-rif analysis. Note that the selection *hbnoise_src* lists the noise at the source, not at the output of the circuit.
10. Select *integrated noise* from the *Type* section
11. *Select flat* from the *weighting* section (this is the default).

Integrated noise produces a noise summary integrated over a frequency range using the specified weighting. If you choose integrated noise, you have the option of using a weighting factor. The flat weighting factor specifies that the integration be performed on the original unweighted waveform. The *from weight file selection* option specifies that, before the integration is performed, the noise contributions of particular frequencies in the original waveform be weighted by factors supplied from an input file.

12. Select *A* from the the *noise unit drop-down list*. This will be the noise units used in the hbnoise_src summary.

13. Type `1K` in the *From (Hz)* text field.

14. Type `1M` in the *To (Hz)* text field.

15. You can choose filtering details to include or exclude particular instances in your hbnoise summary. In the *FILTER* section, click *Include All Types*.

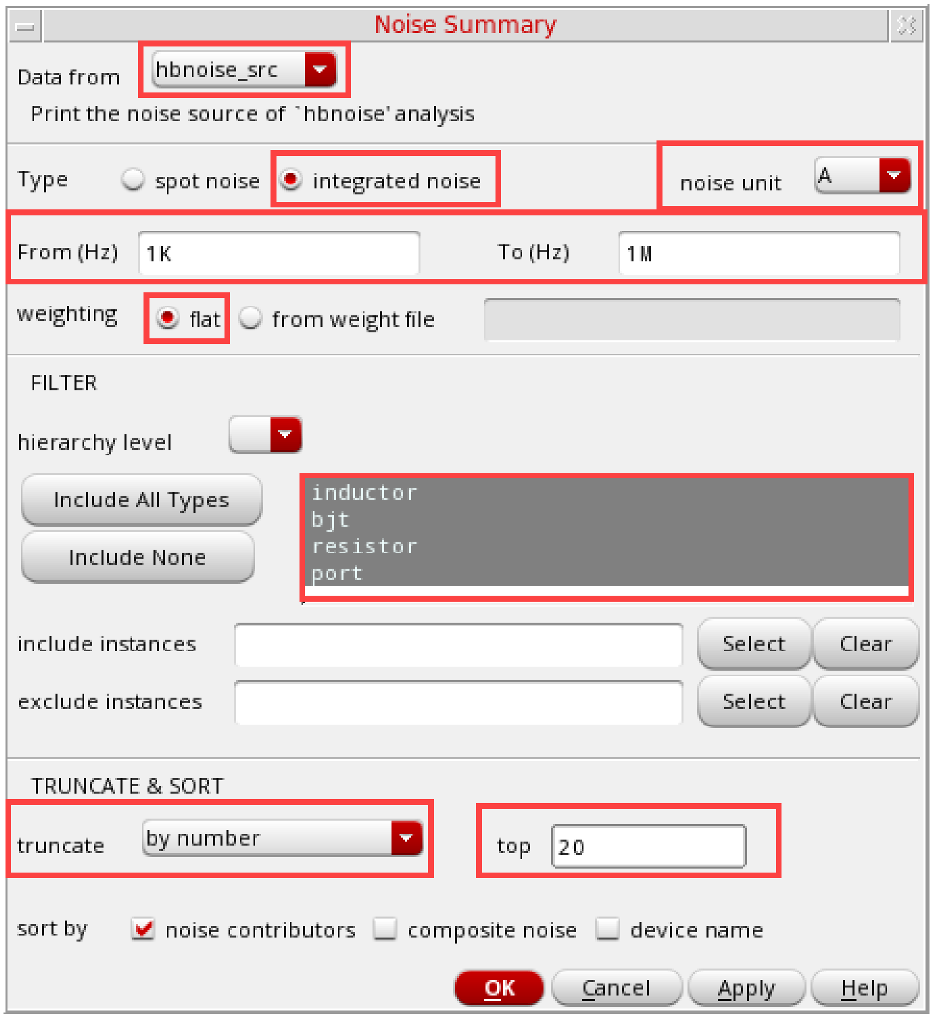
16. You can shorten your summary by specifying how many of the highest contributors to include in the summary, by specifying the percentage of noise a device must contribute to be included in the summary, or by specifying the level of noise a device must contribute to be included in the summary.

Select*by number* in the *truncate* section and type `20` in the *top* field. You will be printing the top 20 noise contributors.

17. Select *noise contributors* from the *sortby section*.

The *Noise Summary* form should look like the following:

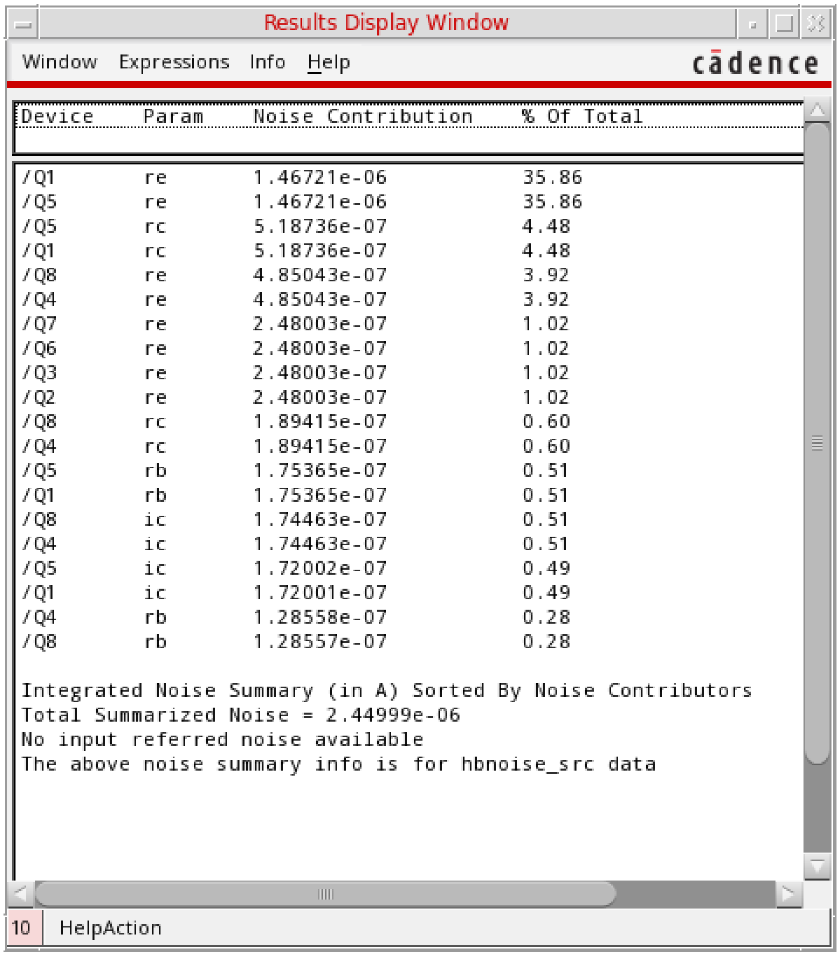
Figure 4-43 HBnoise Summary Form



18. Click *OK*.

The *Results Display Window* is displayed containing the noise summary results, as shown below.

Figure 4-44 Noise Summary Results



Device	Param	Noise Contribution	% Of Total
/Q1	re	1.46721e-06	35.86
/Q5	re	1.46721e-06	35.86
/Q5	rc	5.18736e-07	4.48
/Q1	rc	5.18736e-07	4.48
/Q8	re	4.85043e-07	3.92
/Q4	re	4.85043e-07	3.92
/Q7	re	2.48003e-07	1.02
/Q6	re	2.48003e-07	1.02
/Q3	re	2.48003e-07	1.02
/Q2	re	2.48003e-07	1.02
/Q8	rc	1.89415e-07	0.60
/Q4	rc	1.89415e-07	0.60
/Q5	rb	1.75365e-07	0.51
/Q1	rb	1.75365e-07	0.51
/Q8	ic	1.74463e-07	0.51
/Q4	ic	1.74463e-07	0.51
/Q5	ic	1.72002e-07	0.49
/Q1	ic	1.72001e-07	0.49
/Q4	rb	1.28558e-07	0.28
/Q8	rb	1.28557e-07	0.28

Integrated Noise Summary (in A) Sorted By Noise Contributors
Total Summarized Noise = 2.44999e-06
No input referred noise available
The above noise summary info is for hbnoise_src data

The *Results Display Window* lists the individual contributors, the specific noise mechanism within the semiconductors causing the noise, and the noise contribution. The total input-referred noise voltage and output-referred noise voltage is shown at the bottom of the *Results Display Window*. You may need to use the scroll bar at the right side of the *Results Display Window* to see the bottom part of the results.

Note that the output and input noise include the noise from all the noise contributors, and not just the contributors in the form. The *Param* column uses the same abbreviations as seen in the legend on the left side of the waveform window. For example, *fn* refers to flicker noise. Refer to the *Noise Summary* section in <u>Chapter 3: Frequency Domain Analyses: Harmonic Balance</u> of the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide to get a description of the parameter names for the different device types.

19. Clean up the screen for the next set of measurements.

a. Close the *Results Display Window* by choosing *Window - Close*.

b. Close the *Analog Design Environment* window by choosing *Session -Quit.*

c. In the Schematic window, choose *File - Close.*

Summary

Harmonic Balance (hb), hbac, and hbnoise analyses were setup and simulations were run to determine the Mixer Conversion Gain, RF to IF Isolation, LO to IF Leakage, and Noise Figure for a receive mixer. In addition, you identified different ways to view noise in the receiver using the Noise Separation feature and Noise Summary form

In the next section, you will measure 1dB Compression Point and Desensitization with an RF blocking signal present.

1dB Compression Point, Desensitization, and Blocking

All circuits reach a compression point. The large signal effects at this point cause the gain to stop increasing in a linear fashion. Since this is a large-signal effect, a large-signal analysis is required. When measuring 1dB Gain compression point, the idea is to sweep the input power over a range of power levels, and then plot the output power versus the input power. From the plot, the 1dB compression point can be determined (the 1dB compression point is the input signal level at with the receiver gain drops 1dB when compared to the ideal linear gain).

Receiver desensitization interference occurs when a strong off-channel signal overloads a receiver front end and thus reduces the sensitivity to weaker on-channel signals.

The setup for 1dB Compression Point is the same as that required by a desensitization measurement, except for the frequency of the RF input. For the compression point measurement, the RF signal is in the passband. For the desensitization measurement, the RF frequency is at the blocker frequency. Here, we show both the 1dB compression point and the desensitization measurement using the passband frequency. Usually, they are separate simulations with different frequencies for the RF signal. (This is done to save time in the workshop.)

For 1dB gain compression point measurement, the frequency component set in the RF input port is considered as the RF signal frequency.

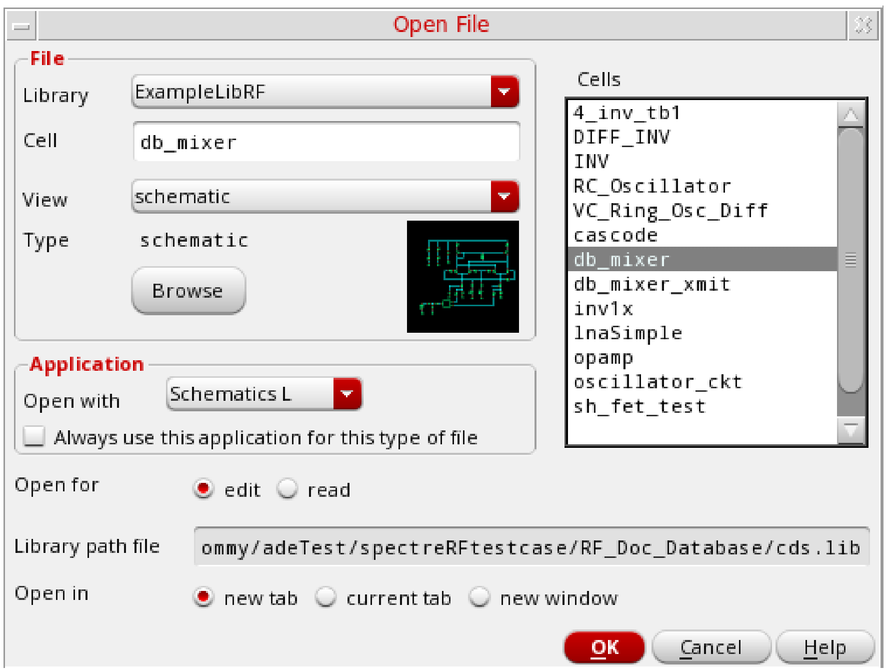
For desensitization measurement, the frequency component set in the RF input port is considered the RF blocker frequency.

You can set up the simulation to measure the conversion gain and noise figure as a function of RF blocker power.

Setting Up to Simulate 1dB CP and Desensitization

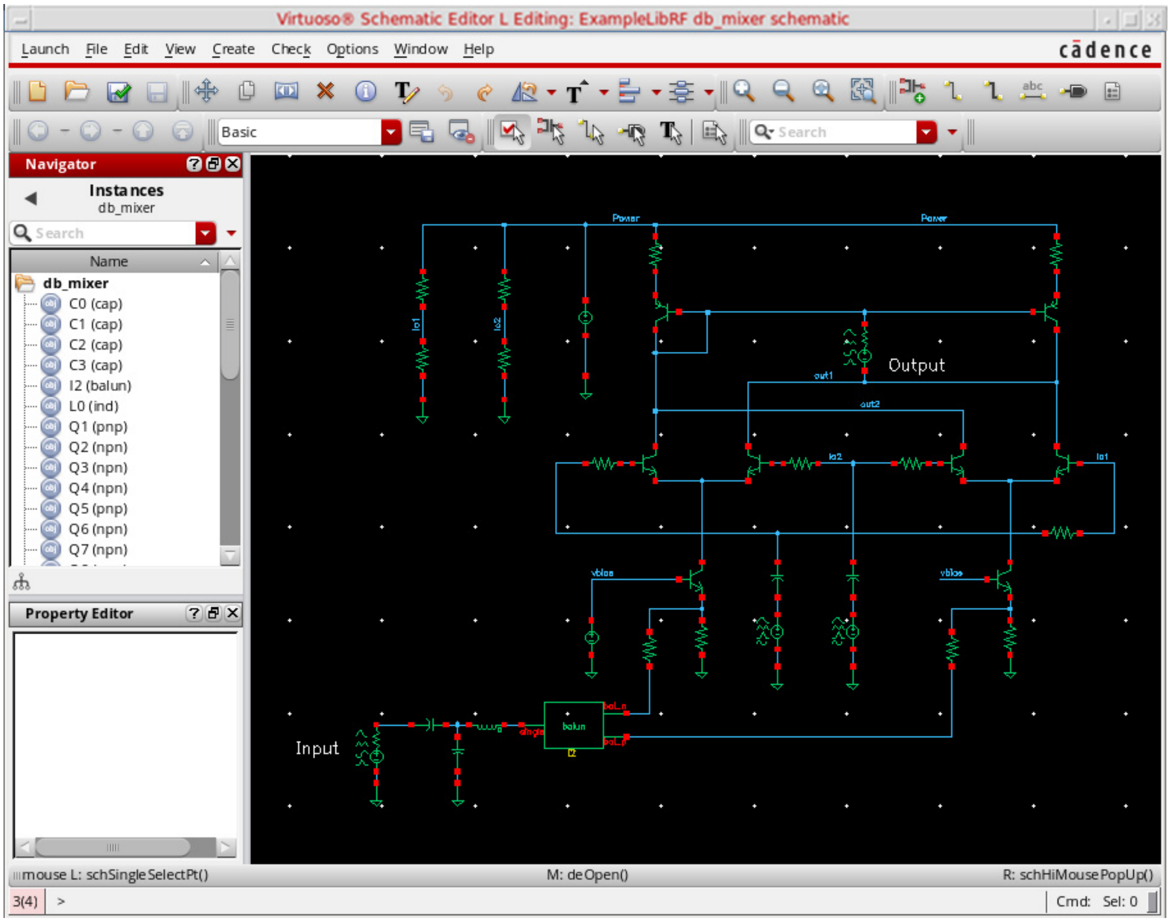
1. In the CIW, choose *File - Open - CellView*. The Open File form is displayed. Choose the *db_mixer schematic* from *ExampleLibRF*, as shown below.

Figure 4-45 Open File Form



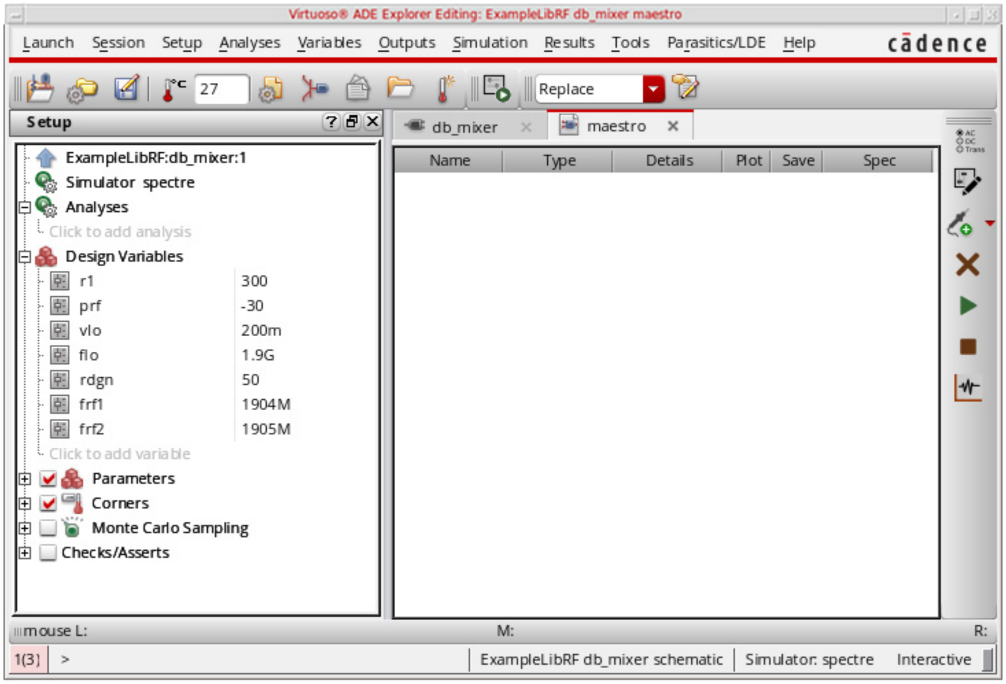
2. Click *OK*. The `db_mixer` schematic appears, as shown below.

Figure 4-46 db_mixer Schematic



3. Start the *Analog Design Environment* from the schematic by choosing *Launch - ADE Explorer*. The Simulation Window is displayed, as shown below.

Figure 4-47 Analog Design Environment Simulation Window



4. In ADE Explorer, select *Setup – Simulator*.

   The *Choosing Simulator* form is displayed.

5. Choose *spectre* as the *Simulator*.

Figure 4-48 Choosing Simulator/Director/Host Form

6. Click *OK* to close the *Choosing Simulator* form.

7. Set up the High Performance Simulation Options, as follows:

   In ADE Explorer, choose *Setup - High Performance Simulation*. The High Performance Simulation Options window is displayed, as shown below.

Figure 4-49 High Performance Simulation Options Form

a. In the High Performance Simulation window, select *APS*. Note that *auto* is selected for multithreading. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores.

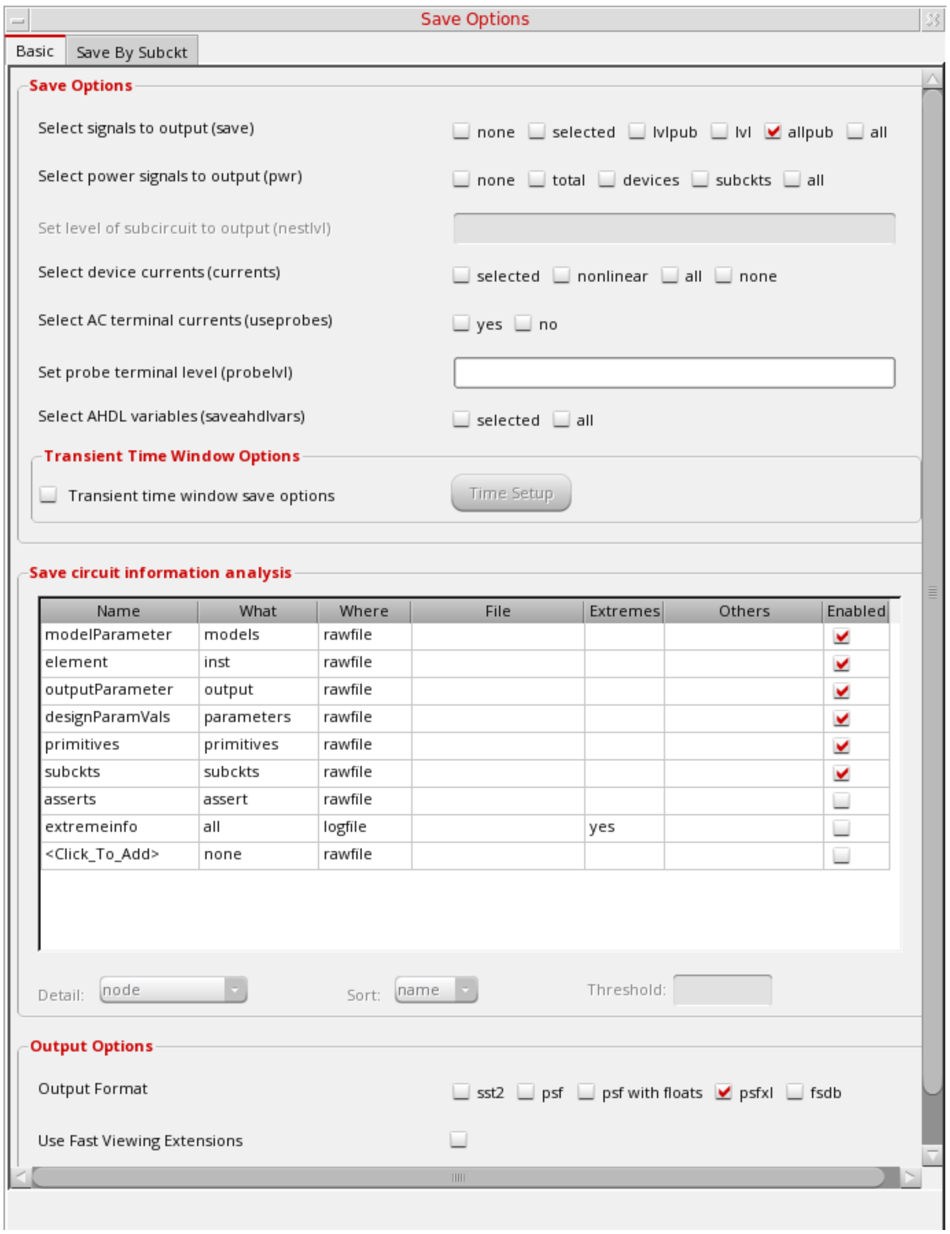
Note: The bigger the circuit, the more threads you should use. For a small circuit such as this, you may want to set the number of threads to 2. Using 16 threads on a small circuit might actually slow things down because of the overhead associated with multithreading. For more information, see the *Spectre Circuit Simulator and Accelerated Parallel Simulator User guide*.

b. Click *OK.*

8. Select *Outputs – Save All*.

   The *Save Options* form is displayed.

Figure 4-50 Save Options Form



9. In the *Select signals to output* section, make sure that *allpub* is selected. This is the default which saves all node voltages at all levels of hierarchy, but it does not include the node voltages inside the device models.

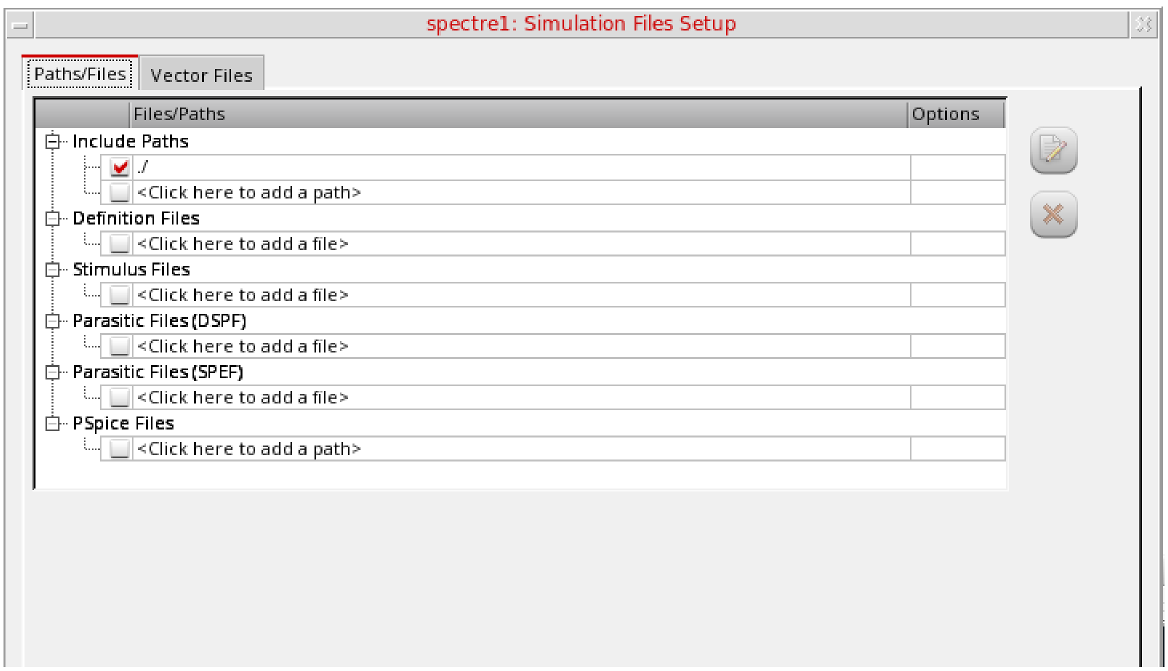
10. Click *OK* to close the *Save Options* form.

Setting Up Model Libraries

1. In ADE Explorer, choose *Setup - Simulation Files.*

   The *Simulation Files Setup* form is displayed, as shown below.

   Figure 4-51 Simulation Files Setup Form



2. Verify that *Include Paths* is set as shown above.
3. Select *Setup – Model Libraries*.

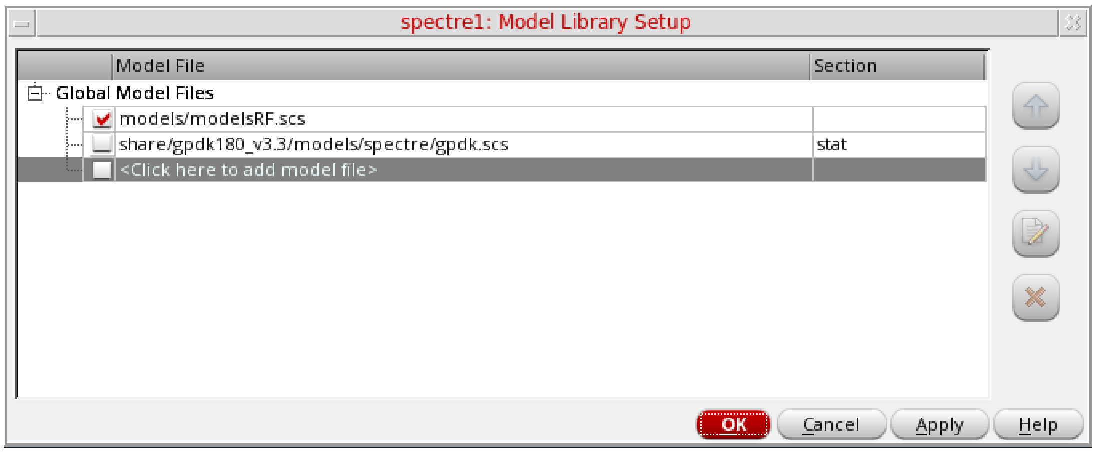
   The Model Library Setup form is displayed.
4. In the *Model Library File* field, type the following path to the model file including the file name:

   `models/modelsRF.scs`

   Alternately, you can browse to the location of your model files.

   The *Model Library Setup* form looks like the following:

Figure 4-52 Model Library Setup Form



5. When you are finished adding model files, click *OK* to close the *Model Library Setup* form.

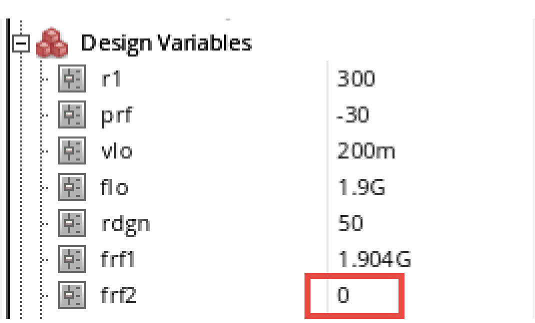
Setting Design Variables

In this simulation, the RF input frequency is set to 1.904G, and prf is the RF input power.

1. In ADE Explorer, change the value of the *frf2* to *0* in the *Design Variables* section.
2. Ensure that the *frf1* variable is set to `1.904G`. If not, change the value to 1.904G.
3. Press *Enter*.

   Your *Design Variables* section looks like the following figure:

Figure 4-53 Design Variables Section of ADE

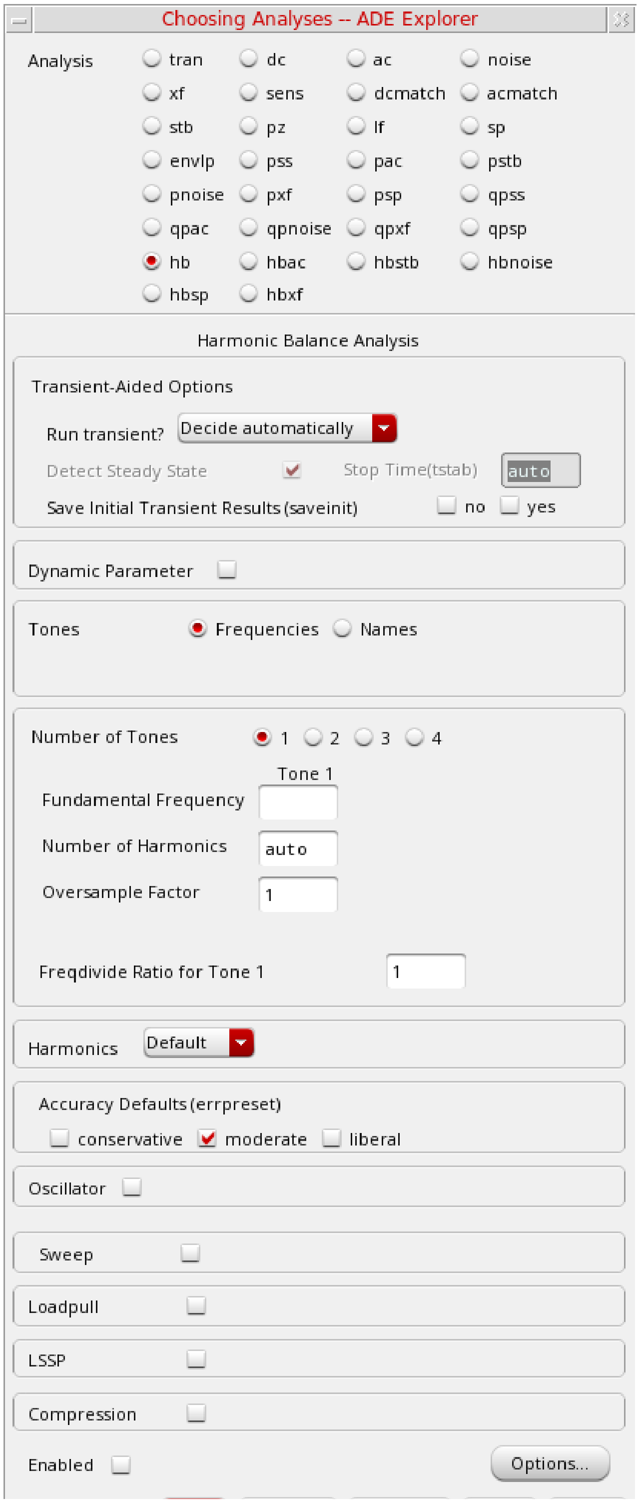


Set up the HB analysis

1. Click the *Choosing Analyses* icon on the right side of ADE Explorer. 

The *Choosing Analyses* form is displayed, as shown below.

Figure 4-54 hb Choosing Analyses Form



Fill in the form as follows:

2. Select *hb* analysis.
3. Harmonic balance can set the harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. To enable this, select *Decide automatically* or *Yes* for the *Run Transient* selection in the *Transient-Aided Options* section. This single action will cause a transient analysis to be run until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1* (when *Frequencies* is selected) or for the tone that has *tstab* enabled (when *Names* is selected).

    a. In the *Transient Aided Options* section, leave *Run transient?* at the default value of *Decide automatically*.

    Run transient? will run the LO signal using the transient (In SpectreRF, this is called the *tstab* interval) for a short period of time. At the end of *tstab*, an FFT is performed, and this is used as the starting point in the harmonic balance analysis. Doing this improves the convergence of hb by giving it a better starting point at the cost of a short transient analysis.

    b. Leave *Stop Time (tstab)* set to the default value of *auto*.

    When *auto* is selected for *Stop time*, a small number of periods of the LO is run using the transient analysis. During this time, the signal is checked for steady-state conditions. If steady-state is not reached in the initial number of periods, more periods can be added automatically by the simulator. Using this feature allows an accurate FFT for the starting point in the hb iterations.

    c. When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide automatically* simplifies the setup for harmonic balance, and produces correct answers from hb without needing to know how to set hb up manually.

    d. If you want to see the startup waveform, set *Save Initial Transient Results (saveinit)* to *yes*.

    e. In the *Tones* section, choose *Frequencies* (this is the default value).

    Harmonic balance can now set harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. A transient analysis runs until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1* (when *Frequencies* is selected) or for the tone that has *tstab* enabled (when *Names* is selected). If you want to manually set transient-aided hb, select *Yes* from the *Run Transient?* drop-down list and set a time for the transient in the *Stop Time (tstab)* field. In this mode, the stop time of the transient analysis in the *tstab* interval cannot be automatically extended.

f. Select *2* for the *Number of Tones*.

g. Type `1.9G` and `1.904G` in the *Fundamental Frequency* fields. *1.9 GHz* is the LO frequency and *1.904GHz* is the RF (frf1) frequency. *Tone 1* should be the LO or signal which causes the most nonlinearity.

The *Number of Harmonics* field for *Tone 1* is automatically set to *auto*. For *Tone 2*, type `5` harmonics. Your circuit is operating near the compression point, so a higher number of harmonics has been chosen.

h. Type `1` and `1` in the *Oversample Factor* fields. When all the signals in the system (including currents) are nearly sinusoidal, then an *Oversample* of *1* is appropriate.

i. Leave *Harmonics* set to *Default*.

j. In the *Accuracy Defaults* section, ensure that *moderate* is selected. For most normal measurements *errpreset* should be set to *moderate*. When you need to measure really small distortions, then use *conservative*.

k. To set up a sweep analysis, click *Sweep* and set *Sweep* to *1* (this is the default value).

l. For *Frequency Variable?* select *no*. You will be sweeping input power rather than frequency.

m. Type `prf` in the *Variable Name* field.

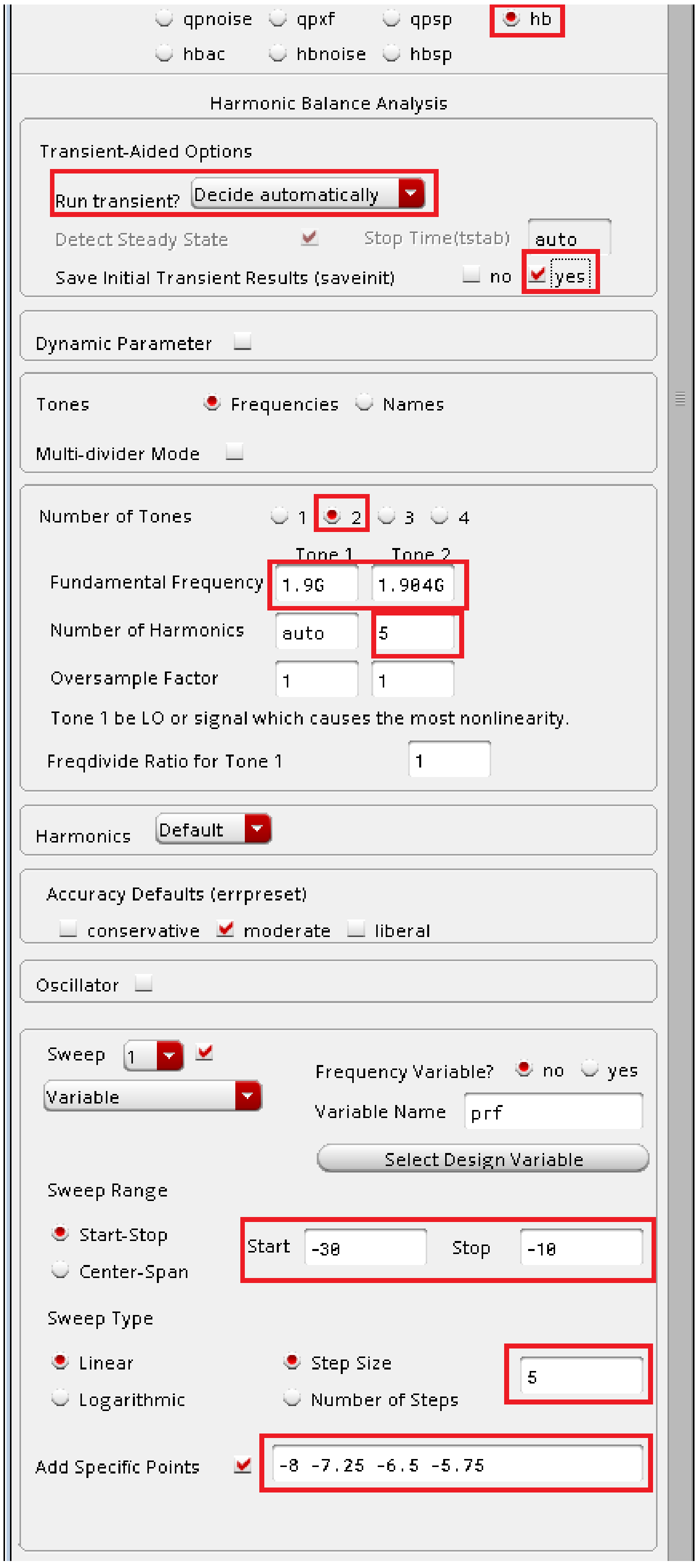
n. In the *Sweep Range* section, type `-30` in the *Start* field and `-10` in the *Stop* field.

o. Se tthe *Sweep Typeto Linear* and type `5` in the *Step Size* field.

p. Click *Add Specific Points*.

Type `-8 -7.25 -6.5 -5.75` in the *Additional Points* field. Note the spaces between the entries. You may need to use smaller spacings in the power sweep as the input power gets large and you approach the compression point. Using smaller spacings in the power sweep near the compression point helps with convergence.The *Choosing Analyses* form should look like the following:

Figure 4-55 hb Choosing Analyses Form



For more information on setting up the Choosing Analyses form, see Chapter 3: Frequency Domain Analyses: Harmonic Balance in the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide.

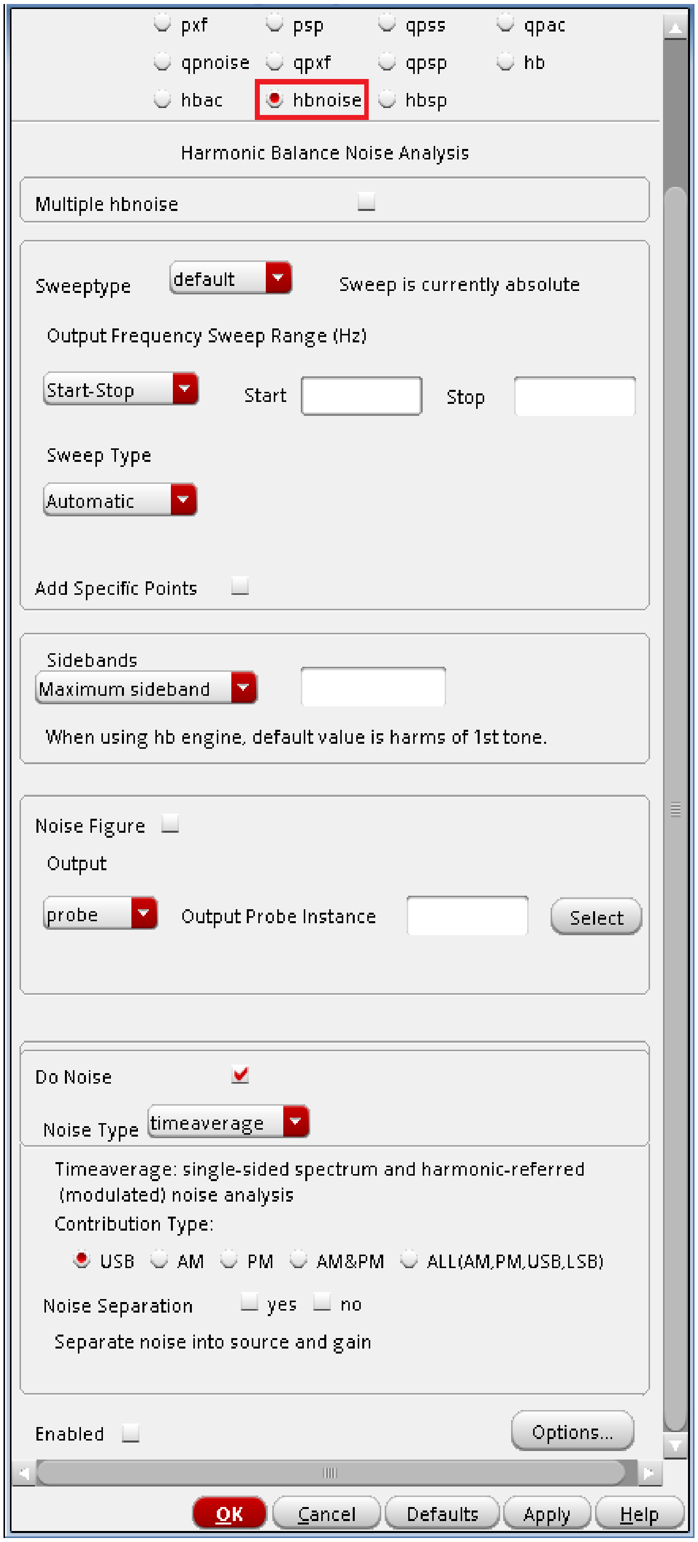
4. Click *Apply*.

Setting up hbnoise to measure noise figure

Hb is used to capture the large signal behavior. Hbnoise analysis follows, and is used to measure the noise performance. Because prf is being swept in hb, the noise figure can be measured as a function of input power (prf).

1. In the *Analysis* section of the Choosing Analyses form, select *hbnoise,* as shown below.

Figure 4-56 hbnoise Choosing Analyses Form



Discussion

In hbnoise, the output frequency is `fout=200K`. The LO is at `1.9GHz`. The RF signal frequency is `1.9002G`, and calculated as follows:

`fin=fout+1*flo +0*frf1=200K+1*1.9G+0*1.904G=1.9002G`

The LO and one RF signal are applied in HB. The second RF signal is set to zero. The blocker frequency is set to `1.904G` at the input port.

When the RF signal gets large, it introduces nonlinearity to the system, and that causes mixing of additional noise frequencies with the RF input, thus adding more noise at the output of the system. In this case, noise `200KHz` above and below the LO harmonics of `1.9GHz` will mix to `200KHz` at the output, and noise `200KHz` above and below the RF harmonics of 1.`904HGz` will also mix to `200KHz` at the output.

The noise figure can be measured as a function of the blocker power `prf`.

Note: The number of significant digits is set to `4`. This is the default netlisting resolution for all numerical values from the choosing analyses form. This is sufficient for this design.

However, in your particular design, you may need to increase it to a higher number (6 for example) to see all of the significant digits in the *Reference Sideband* field. If you do not set this, the frequency value will be truncated.

Type the following in CIW:

```
aelPushSignifDigits(6)
```

If you consistently need more than `4` significant digits in your forms, this can also be added to your `.cdsinitfile` so that it is set automatically every time you start ADE Explorer. In the workshop database, you can see in the `.cdsinit` file that this value has been set to `10`.

Fill the hbnoise form as follows:

1. Leave the *Sweeptype* set to *default* (*absolute*).
2. Select *Single-Point* from the *Output Frequency Sweep Range (Hz)* drop-down list.
3. Type `200K` in the *Freq* field. This the output frequency (*fout*) from the mixer.
4. In the *Sidebands* section, leave the *Maximum sideband* field blank. When the *Maximum sideband* field is left blank, (strongly recommended for noise analysis) all the noise folding from all the harmonics in the hb analysis are included in the noise result.
5. Select the *Noise Figure* option.

6. In the *Output* section, leave the *Output* set to the default value of *probe*. Type `/rif` in the field to the right *Output Probe Instance*. Alternately, you can click the *Select* button and select the output port in the schematic.

   Since *probe* was selected as the output measurement technique for the pnoise analysis, Spectre will subtract any noise contribution by the load resistance from the noise figure calculation. The load resistor noise is still present in the output noise.

7. Ensure that the *Input Source* is set to type *port* (this is the default). Type `/rf` for the *Input Port Source.* Alternately, click *Select* to the right of *Input Port Source* and select the input port in the schematic. If you want a noise figure calculation, you must select a port as the input source.

   In the *Reference Sideband* section, choose *Select From List*. The form will expand to show the available frequencies. The reference sideband specifies the RF passband frequency for the noise figure calculation.

8. Under *Select from list*, in the *From (Hz)* and *To (Hz)* fields, you should see *0* to *1e12* by default.

9. Select the *u 1.9002G 1 0* line. This corresponds to the input rf frequency.

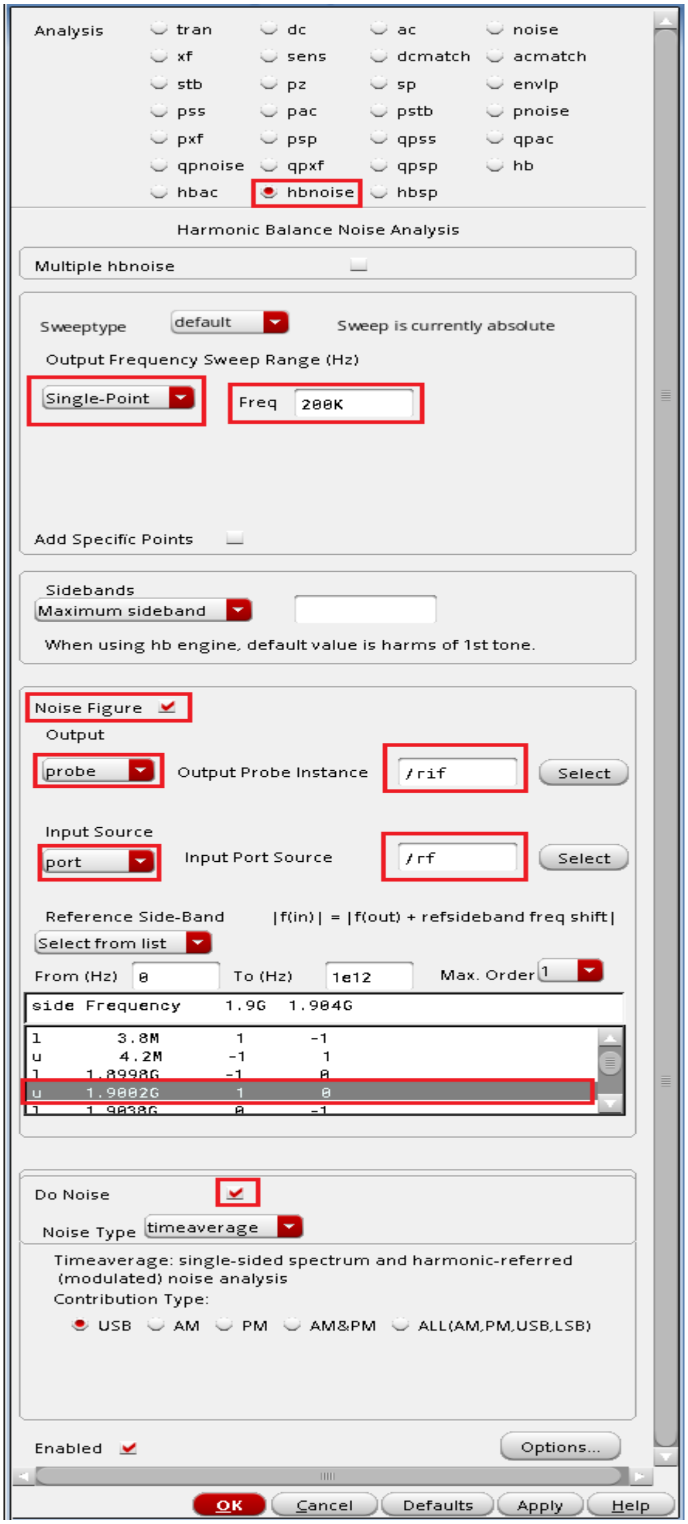
10. Ensure that *Do Noise* is selected and that *Noise Type* is set to *timeaverage*. When *Noise Type* is set to *timeaverage*, the output noise calculated by hbnoise is the average noise power that is present with both the LO and RF signals applied in the circuit. Average noise power is also called RMS noise power. It is the true heating power that would be present in the load resistor.

11. Select the *USB* option.

12. Leave the rest of the form set to the default values.

    The *Choosing Analyses* form should look like the following:

Figure 4-57 hbnoise Choosing Analyses form

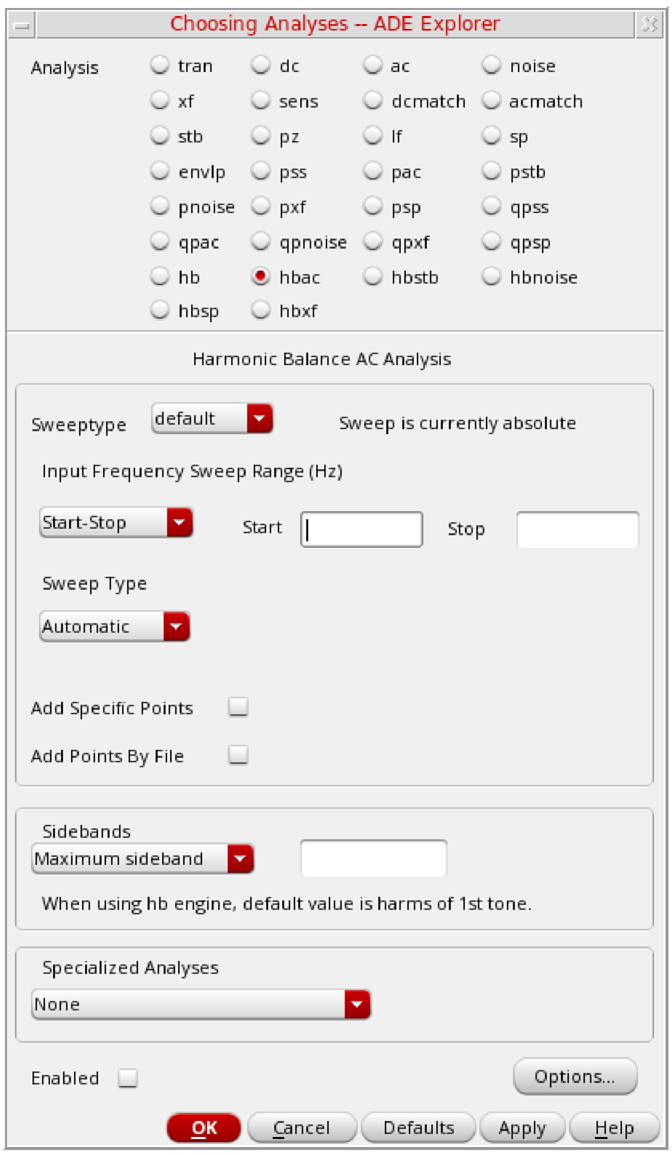


13. Click *Apply*.

Set up HBAC to measure conversion gain.

1. In the *Choose Analyses* form, select *hbac*. The hbac *Choosing Analyses* form is displayed, as shown below.

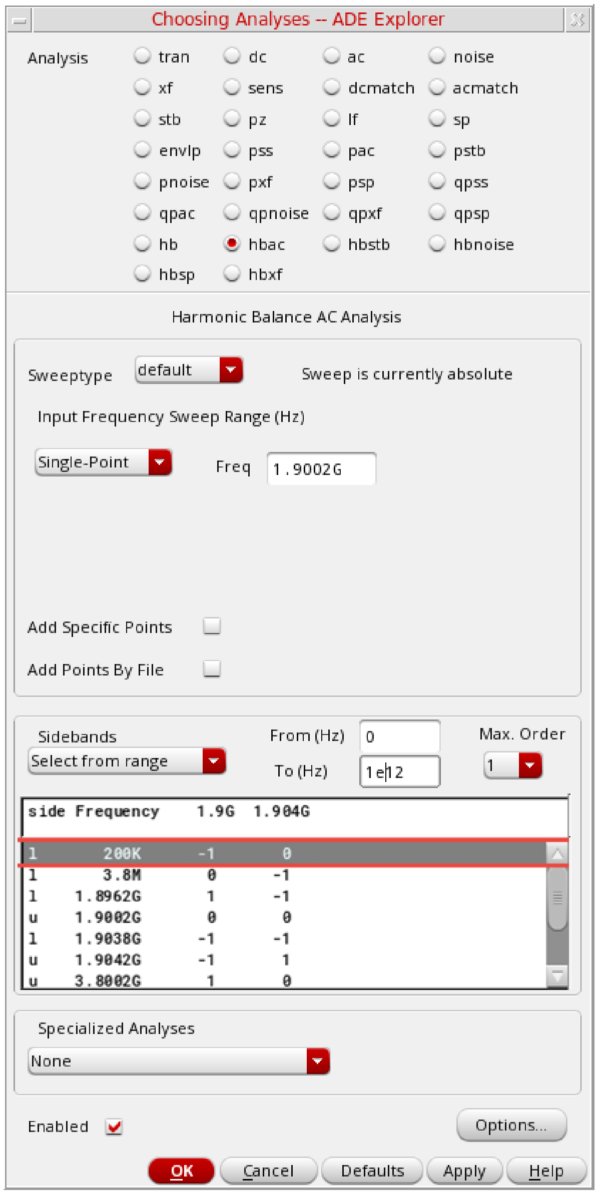
Figure 4-58 hbac Choosing Analyses Form



Fill in the hbac Choosing Analyses form as follows:

2. Set the *Input Frequency Sweep Range (Hz)* selection to *Single-Point.*
3. Type `1.9002G` in the *Freq* field. This is the input frequency.
4. In the *Sidebands* section, choose *Select from range*.
5. Under *Select from range*, in the *From (Hz)* and *To (Hz)* fields, you should see *0* to *1e12,* by default.
6. Select the *I 200K -1 0* line. *200K* is the output frequency from the mixer. This will measure the conversion gain.
7. The *Choosing Analyses* form looks like the following:

Figure 4-59 The hbac Choosing Analyses form



8. Click *OK*.

Run the simulation

1. Click the green arrow icon on the right side of ADE Explorer or in the Schematic window.  . The simulation runs.

   This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` logfile). When the analysis has completed, you may iconify the status window.

Plot the 1dB Gain Compression point

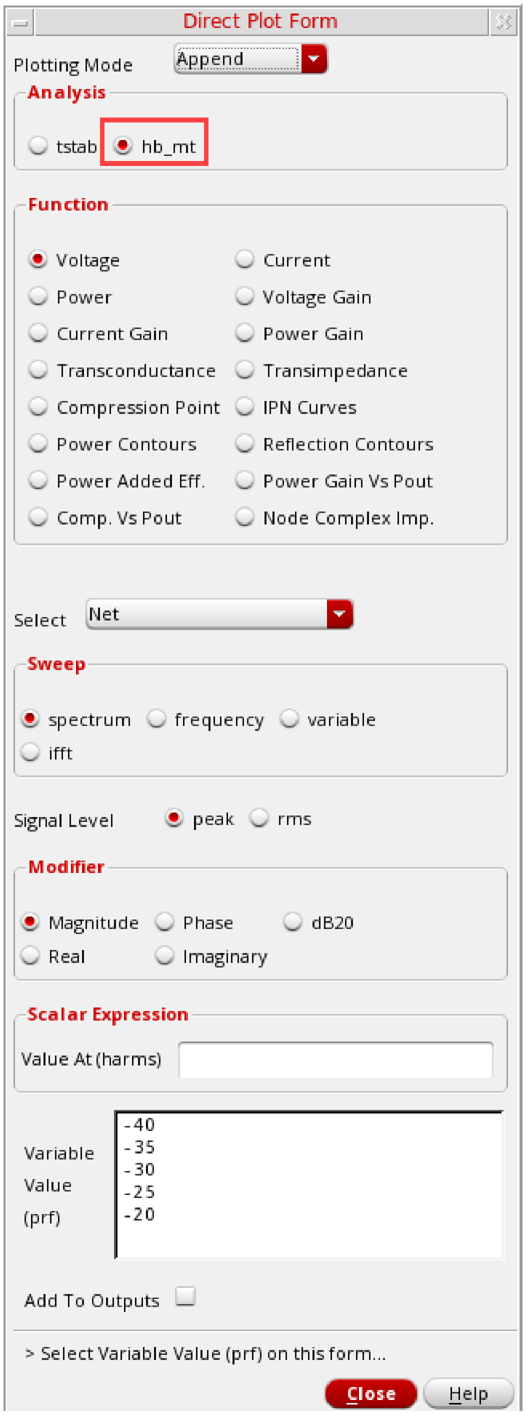
1. In ADE Explorer, choose *Results - Direct Plot - Main Form*.

   The *Direct Plot Form* is displayed. Alternately, you can press the *Direct Plot* icon in the schematic window, as shown below.

2. In the *Direct Plot Form*, select *hb_mt* .

   hb_mt refers to multitone harmonic balance.

Figure 4-60 Direct Plot Form

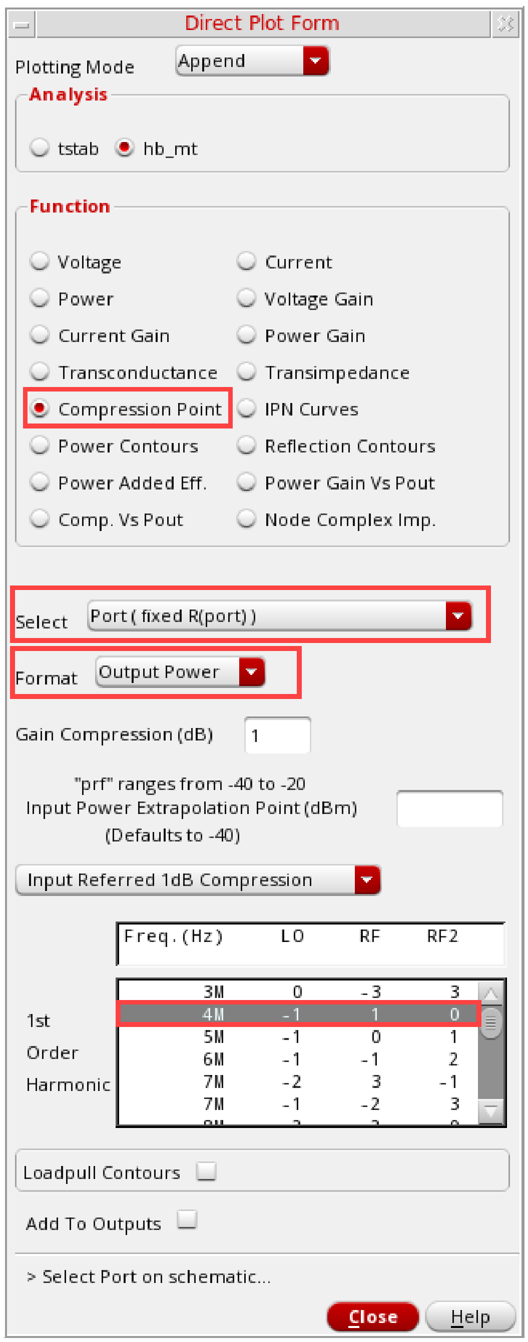


3. Select *Compression Point* in the *Function* section.
4. Ensure that *Port (fixed R(port))* is selected from the *Select* drop-down list.

5. Select *Output Power* is selected from the *Format* drop-down list.
6. Select the *4M -1 1* term in the *1st Order Harmonic* section.

   The *Direct Plot Form* should look like the following:

Figure 4-61 Direct Plot Form hb_mt



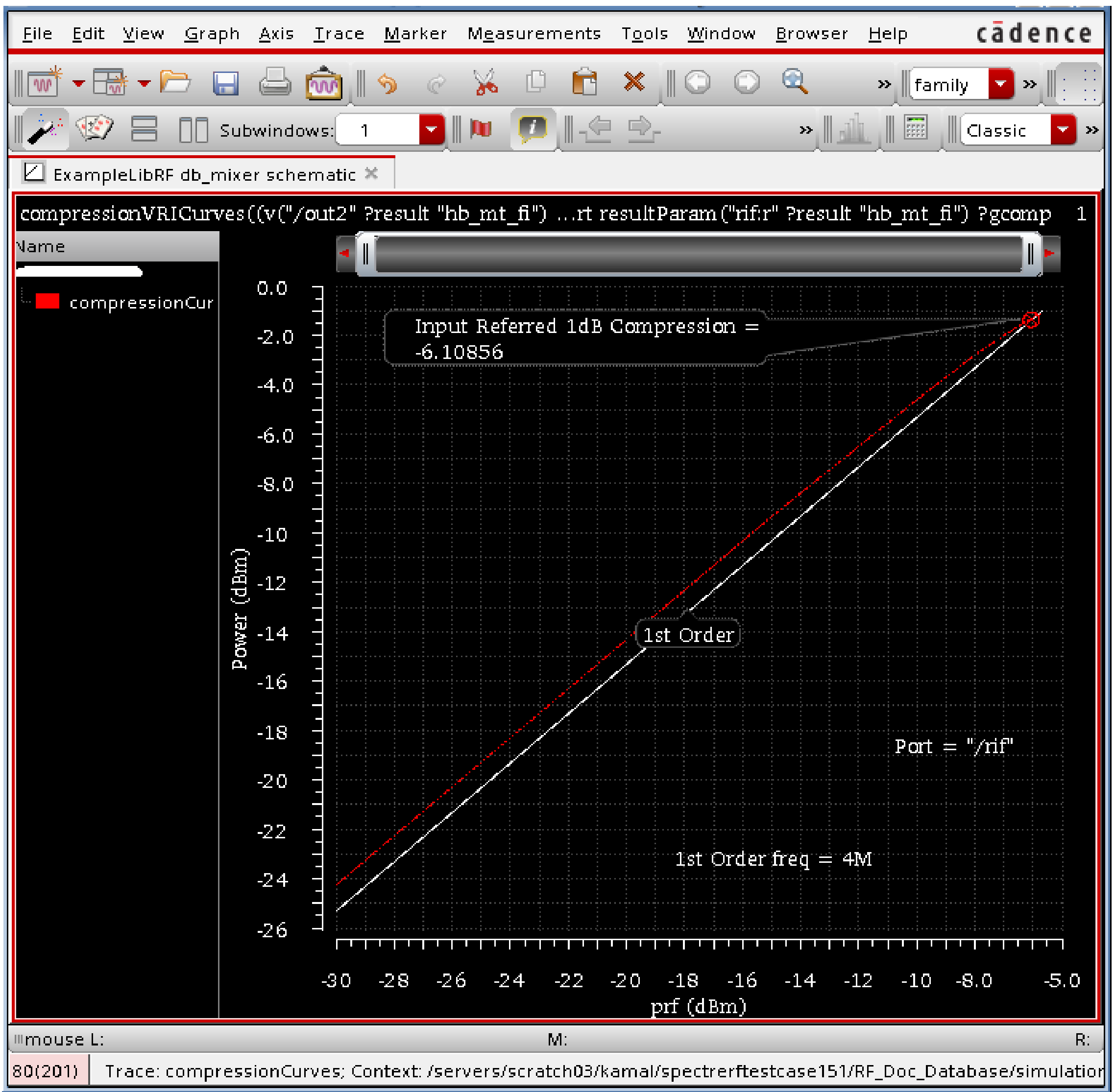
7. In the schematic, select the signal source just to the left of the *Output* label.

The plot of Output Power vs. prf is displayed.

To move the label, click to select the 1dB Compression point label, , and hold and drag the Compression Point label to the desired location. (First, select a random point in the graphics area to deselect the marker intercept point. Otherwise, the label will not move.)

Note the compression point. The input-referred compression point is about *-6.1dB.*

Figure 4-62 Input Referred 1dB Compression Point



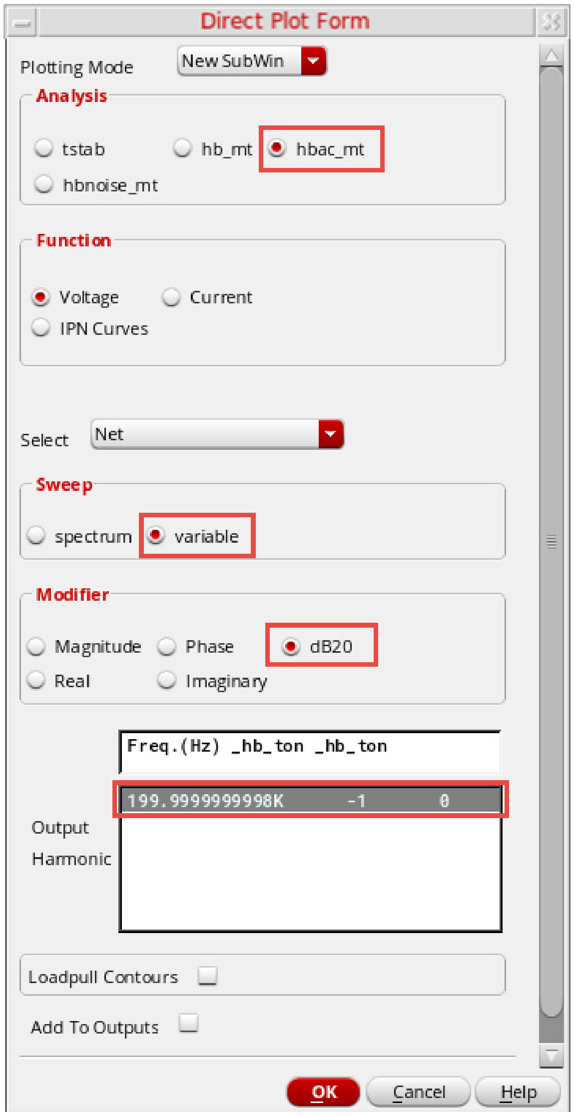
The expression used to plot the Compression curves is shown above in the waveform window.

Plot the RF-IF conversion gain as a function of RF (Blocker) power

8. In the *Direct Plot Form*, set *Plotting Mode* to *New SubWin*.
9. In the *Analysis* section, select *hbac_mt*.
10. In the *Function* section, select *Voltage.*
11. *Select Net* (this is the default).
12. In the *Sweep* section, select *variable*.
13. In the *Modifier* section select *dB20*.
14. In the *Output Harmonic* section, select the *199.9999999998K -1 0* line.

The *Direct Plot Form* should look like the following:

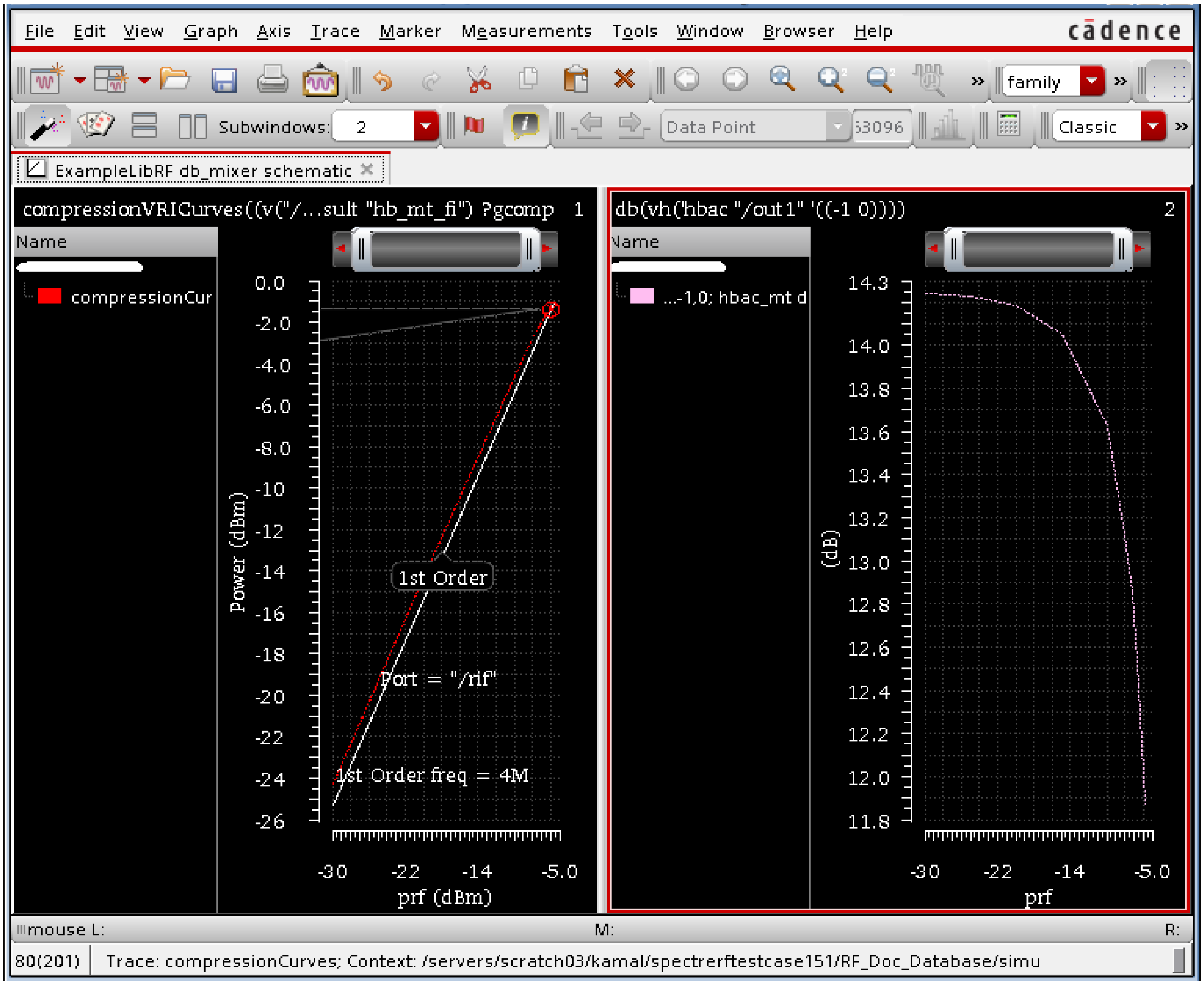
Figure 4-63 Direct Plot Form hbac_mt



15. In the schematic, select the *out1* net near the *Output* label.

The conversion gain is plotted. As the blocker power goes up, the conversion gain drops.

Figure 4-64 1dB Compression Point and Conversion Gain



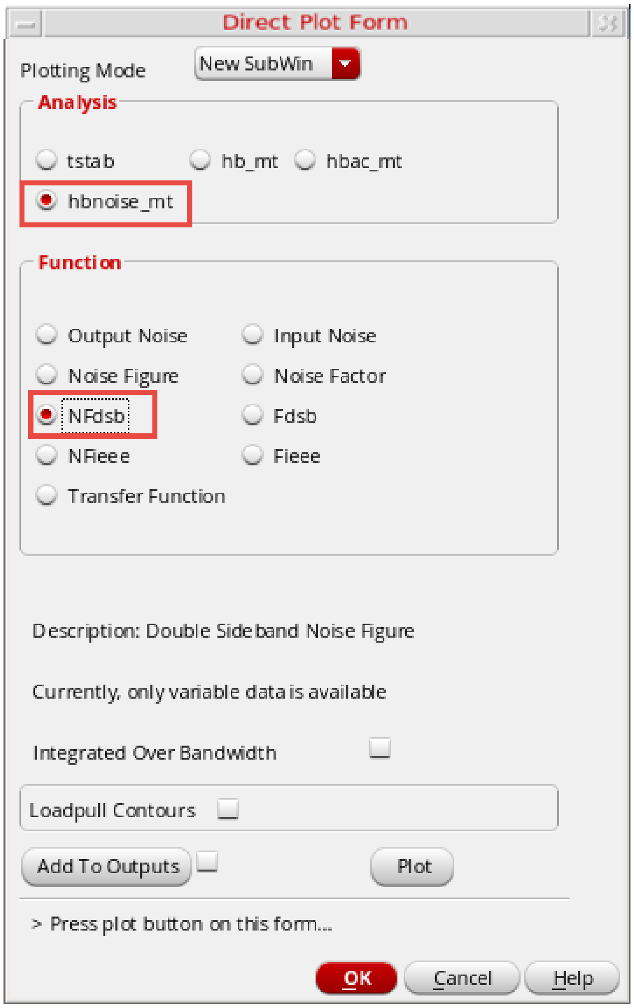
Plot the noise figure

16. In the *Direct Plot Form*, select *hbnoise_mt* in the *Analysis* section.

17. In the *Function* section, select *NFdsb*.

The *Direct Plot Form* should look like the following:

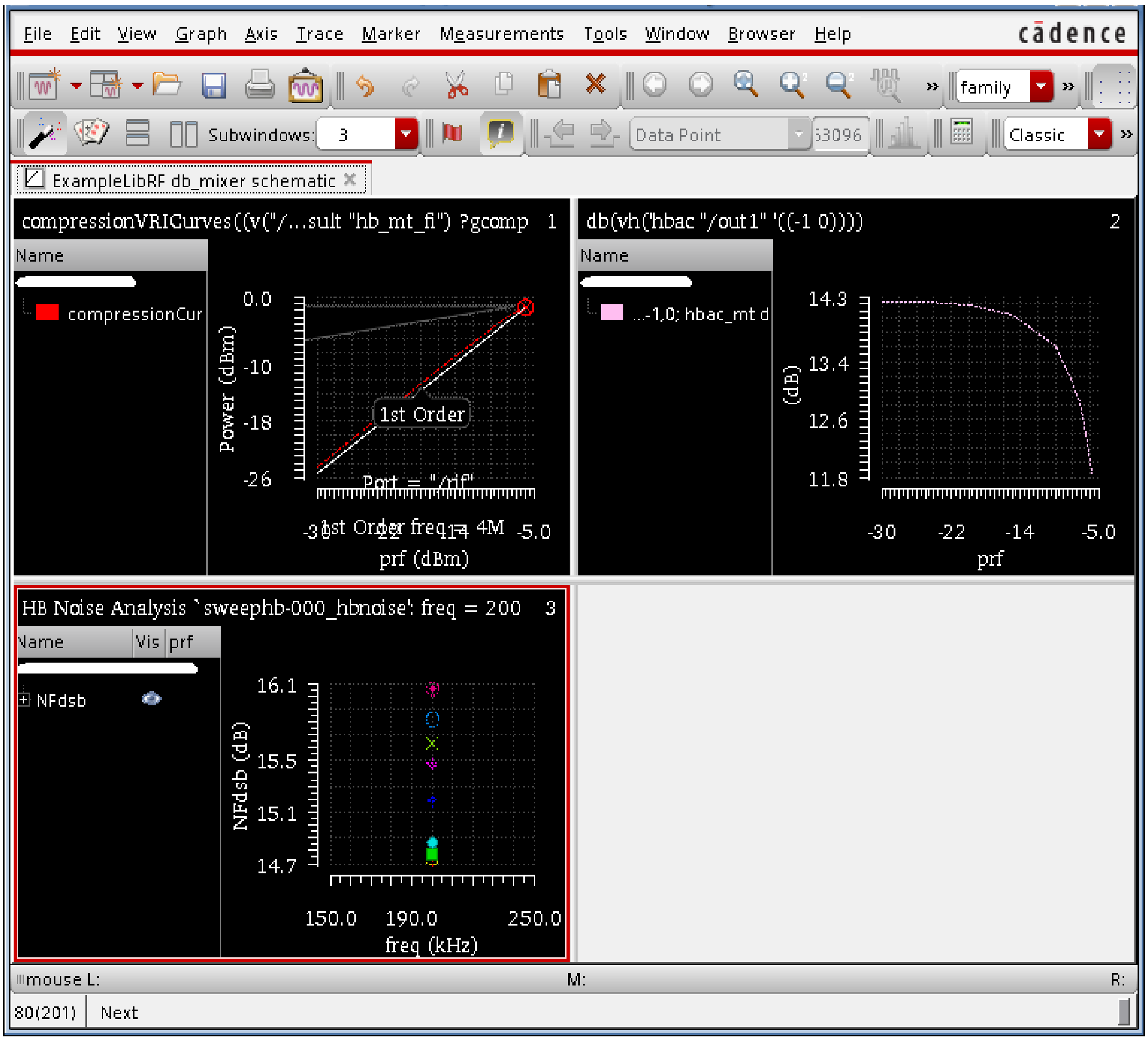
Figure 4-65 Direct Plot Form Plotting NFdsb



18. Click *Plot*.

The noise figure plot is added to the previous plot in a new sub window, as shown below.

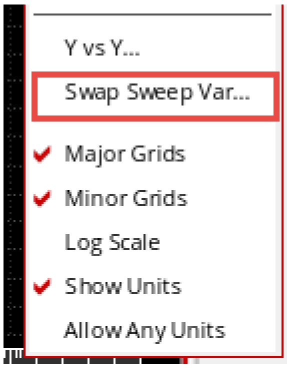
Figure 4-66 1dB Compression Point, Conversion Gain, and Double Sided Noise Figure



Make the noise figure result more readable.

19. In the most recent sub window, move your mouse cursor over one of the numbers on the X Axis, click the right mouse button, and select *Swap Sweep Var* from the context menu, as shown below.

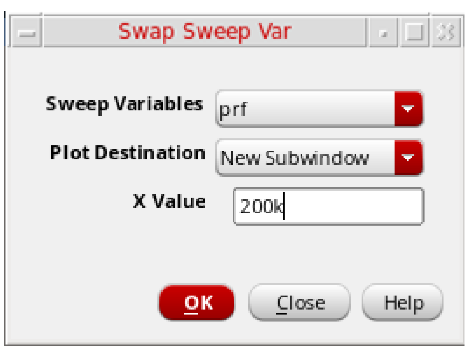
Figure 4-67 X-Axis Properties Form



The Swap Sweep Var form is displayed.

20. In the *Swap Sweep Var* form, select *prf* from the *Sweep Variable* drop-down list. Leave *Plot Destination* set to *New Subwindow.* Enter `200k` in the *X Value* field. Note that for this menu, you must enter `k`, not `K`.

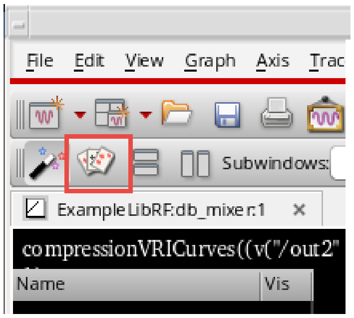
Figure 4-68 Swap Sweep Variable Form



21. Click *OK.*

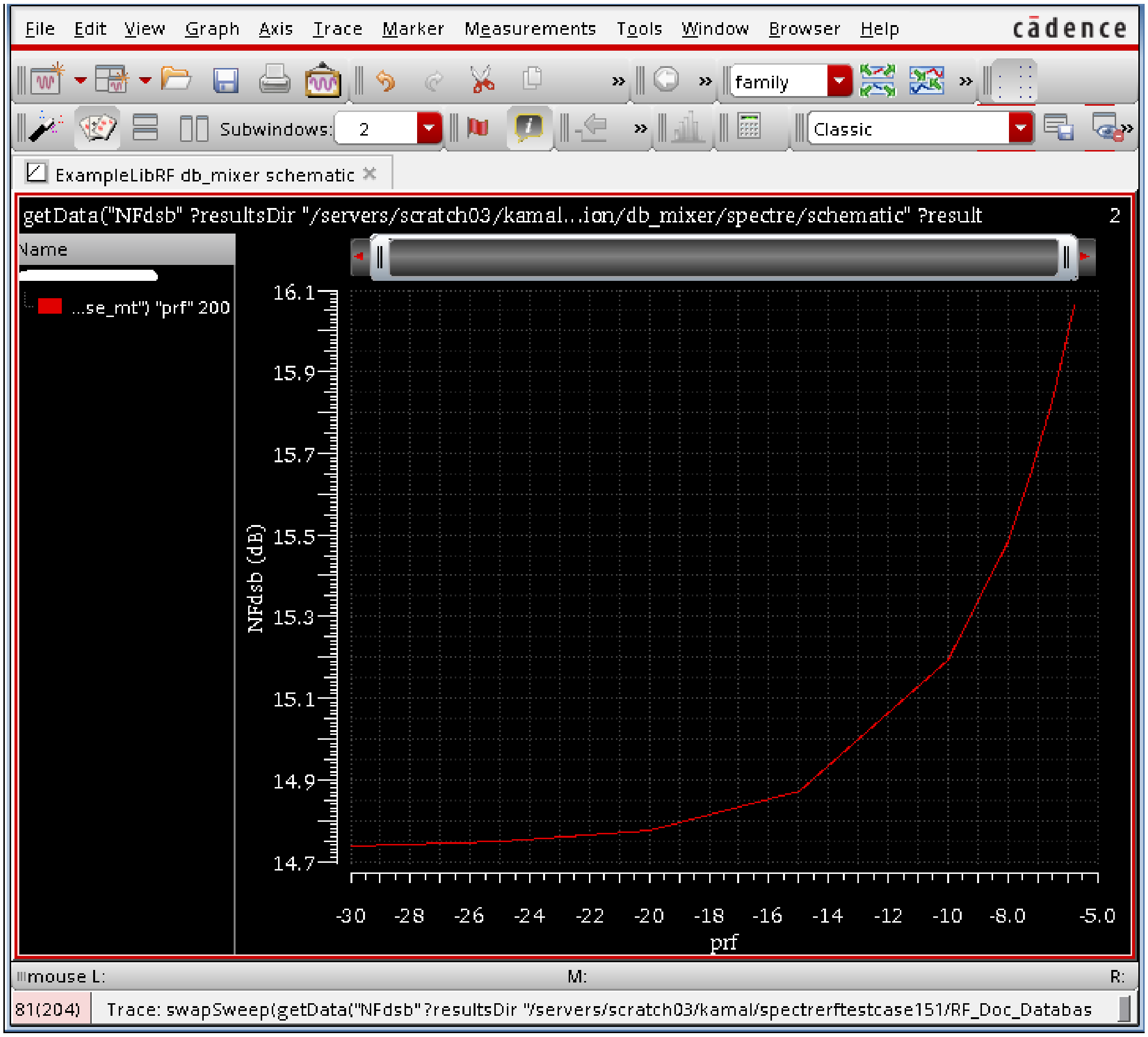
22. The waveform window is now fairly difficult to read. In the waveform window, select *Card* icon from the *Graph* toolbar. The *Card* mode displays one subwindow at a time so that the display area becomes larger. You can switch which subwindow is displayed at any time.

Figure 4-69 Selecting Card Mode Layout



The waveform window changes to the Card mode and the Noise Figure plot is much easier to read, as shown below.

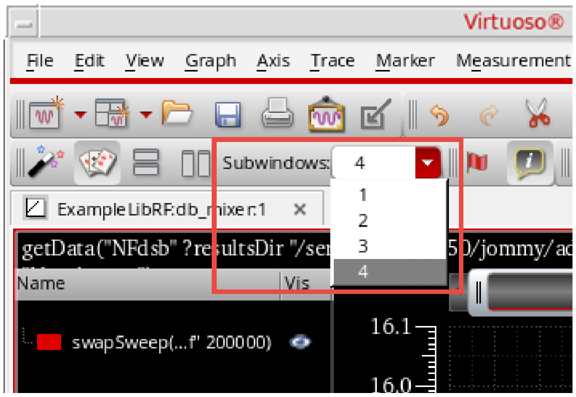
Figure 4-70 Noise Figure



Note the trend in conversion gain and noise figure. As expected, the noise figure goes up and the conversion gain goes down with increasing RF blocker power (prf).

23. To view other subwindows, select another plot to view from the *Subwindows* drop-down list.

Figure 4-71 Selecting Subwindow to View a Different Plot in Card Mode



Clean up the screen for the next set of measurements.

24. Close ADE Explorer by choosing *Session - Quit.*

25. In the Schematic window, choose *File - Close All.*

Summary:

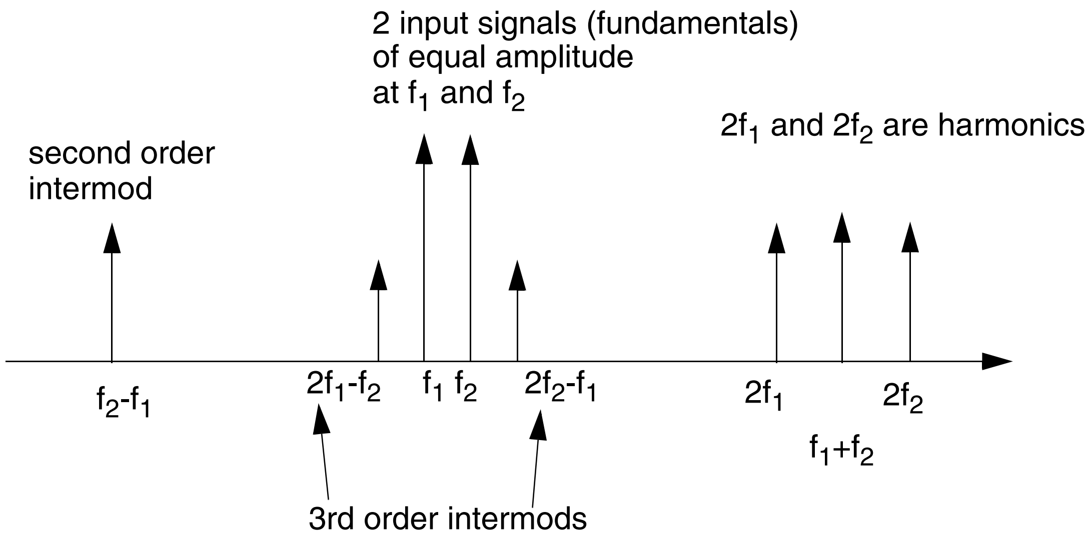
In this section, you measured 1dB compression point, conversion gain, and desensitization with an RF blocking signal present using hb, hbnoise, and hbac analyses.

In the next section, you will measure the Third-Order Intercept using Harmonic balance.

Third-Order Intercept measurement with HB

In the frequency domain, third-order products are the intermodulation distortion products between one of the fundamental signals and the second harmonic of the other signal.

Figure 4-72 Intermodulation Distortion Products



The presence of two or more tones in a nonlinear circuit generates intermodulation products. Many of the spurious tones are out-of-band and cause no problems. However, the third-order tones are nearest to the fundamental and are likely to fall in-band. They add distortion to the output signal. IP3 is a metric or figure of merit for linearity that is used to describe the intermodulation performance of a mixer.

To do the Third-Order Intercept measurement, in this section, both the RF and the LO signals are applied to the circuit. Using the transient analysis is impractical due to lengthy runtimes and spectral calculation times. The harmonic balance engine is used to overcome these disadvantages. *This setup has the advantage of being able to see all of the harmonics on the IF, but it takes longer to run than rapid IP3 for a small-signal measurement. If you want a small-signal IP3, a faster way to approach the problem is shown in the next section.* Using HB by itself for an IP3 measurement is typically only required for transmit mixers and power amplifiers. It is shown here so you can compare two different methods of obtaining IP3: Using three tone hb vs. rapid IP3.

Opening the db_mixer Mixer Circuit in the Schematic Window

1. In the CIW, choose *File – Open*.

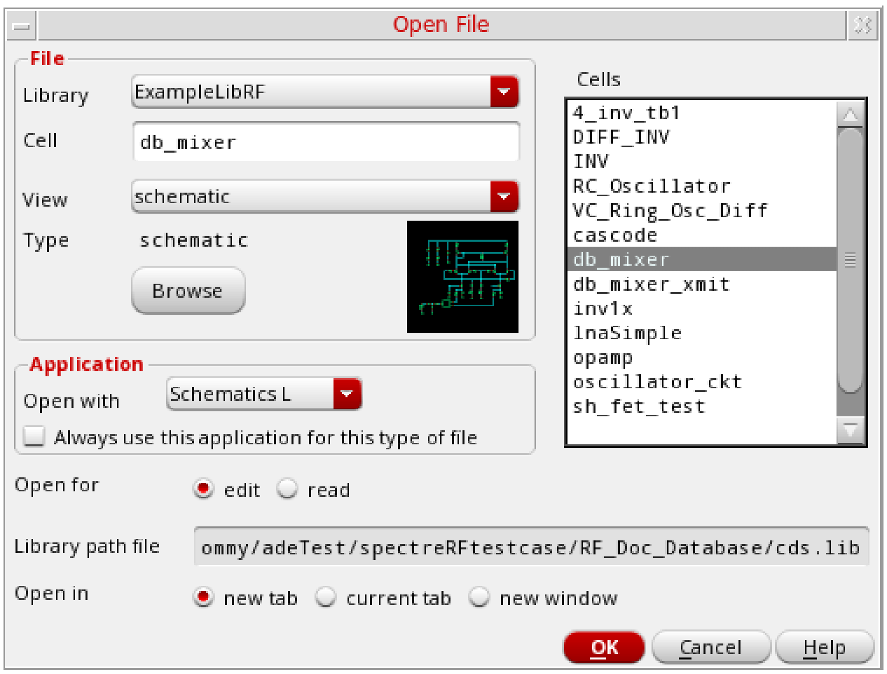
   The *Open File* form is displayed.

2. In the *Open File* form, choose *ExampleLibRF* from the *Library* drop-down list. Choose the editable copy of the *ExampleLibRF* that you created as described earlier in this manual.

3. Choose *db_mixer* in the *Cells* list box.

   The completed Open File form appears like the one below.

Figure 4-73 Open File Form

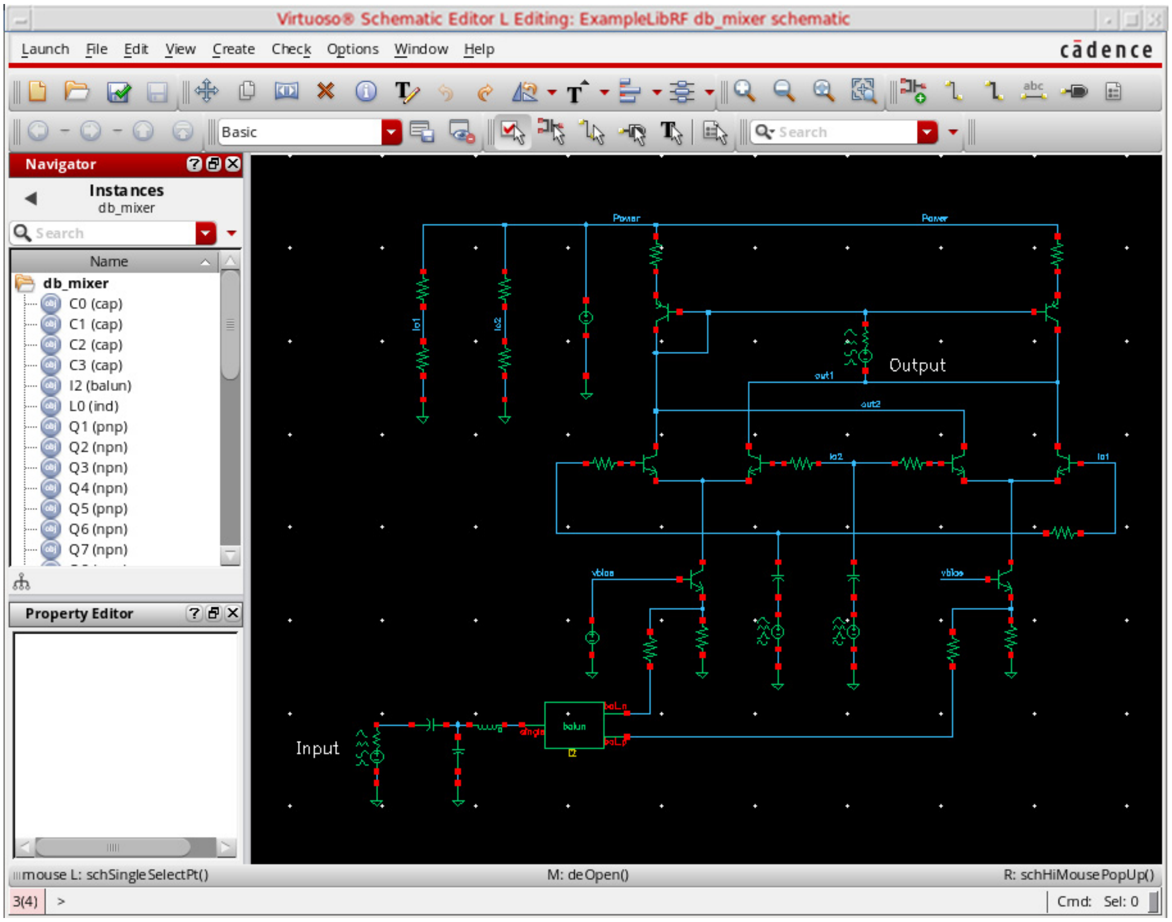


4. Click *OK*.

5. Click *OK* in the Create new ADE Explorer view form.

   The Schematic window for the *db_mixer* mixer is displayed.

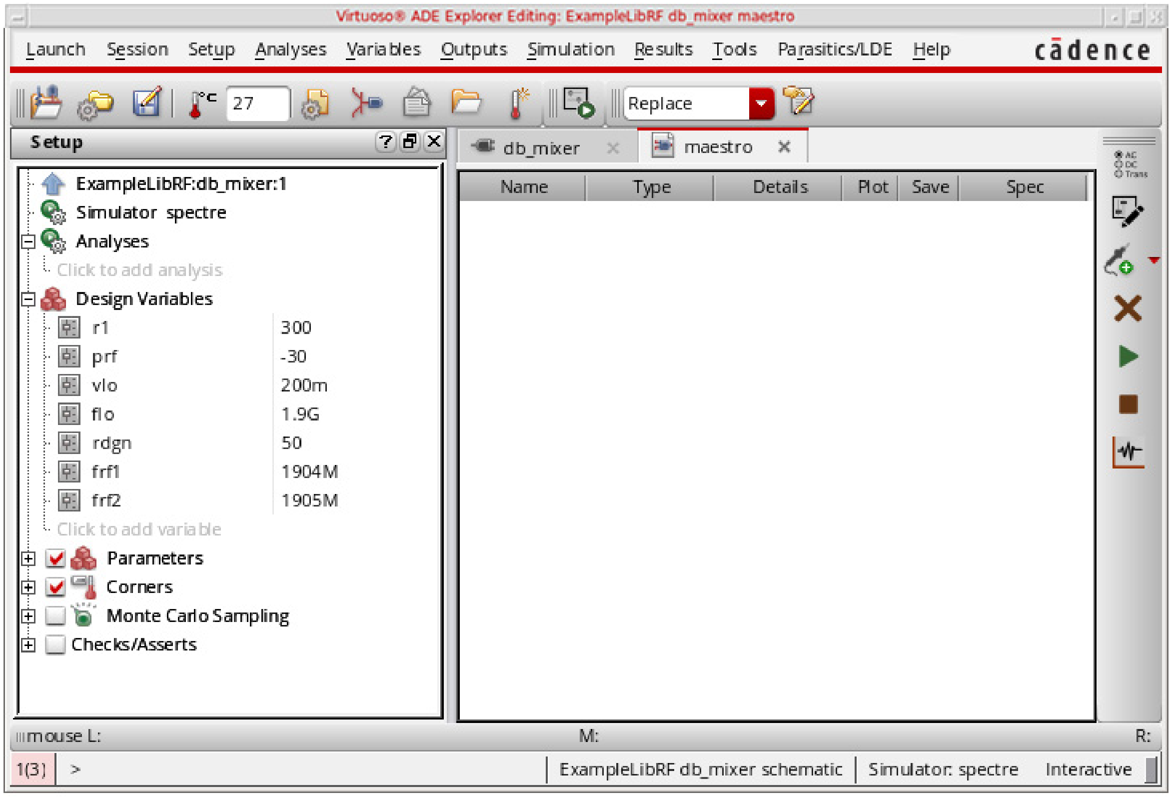
Figure 4-74 db_mixer Schematic



6. In the Schematic window, choose *Launch – ADE Explorer*.

   ADE Explorer is displayed, as shown below.

Figure 4-75 Virtuoso Analog Design Environment Window



Choosing Simulator Options

7. In ADE Explorer, select *Setup – Simulator*.

   The Choosing Simulator is displayed.

8. Select *spectre* from the *Simulator* drop-down list.

Figure 4-76 Choosing Simulator/Directory/Host Form

9. Click *OK* to close the *Choosing Simulator* form.

10. Set up the High Performance Simulation Options

In ADE Explorer, select *Setup - High Performance Simulation.* The *High Performance Simulation Options* window is displayed, as shown below.

Figure 4-77 High Performance Simulation Options Window

In the *High Performance Simulation Options* window, select *APS*. Note that *auto* is selected for *multithreading*. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores. The bigger the circuit, the more threads you should use. For a small circuit such as this, you may want to set the number of threads to 2. Using 16 threads on a small circuit might actually slow things down because of the overhead associated with multithreading. For more information, see the *Spectre Circuit Simulator and Accelerated Parallel Simulator User Guide*.

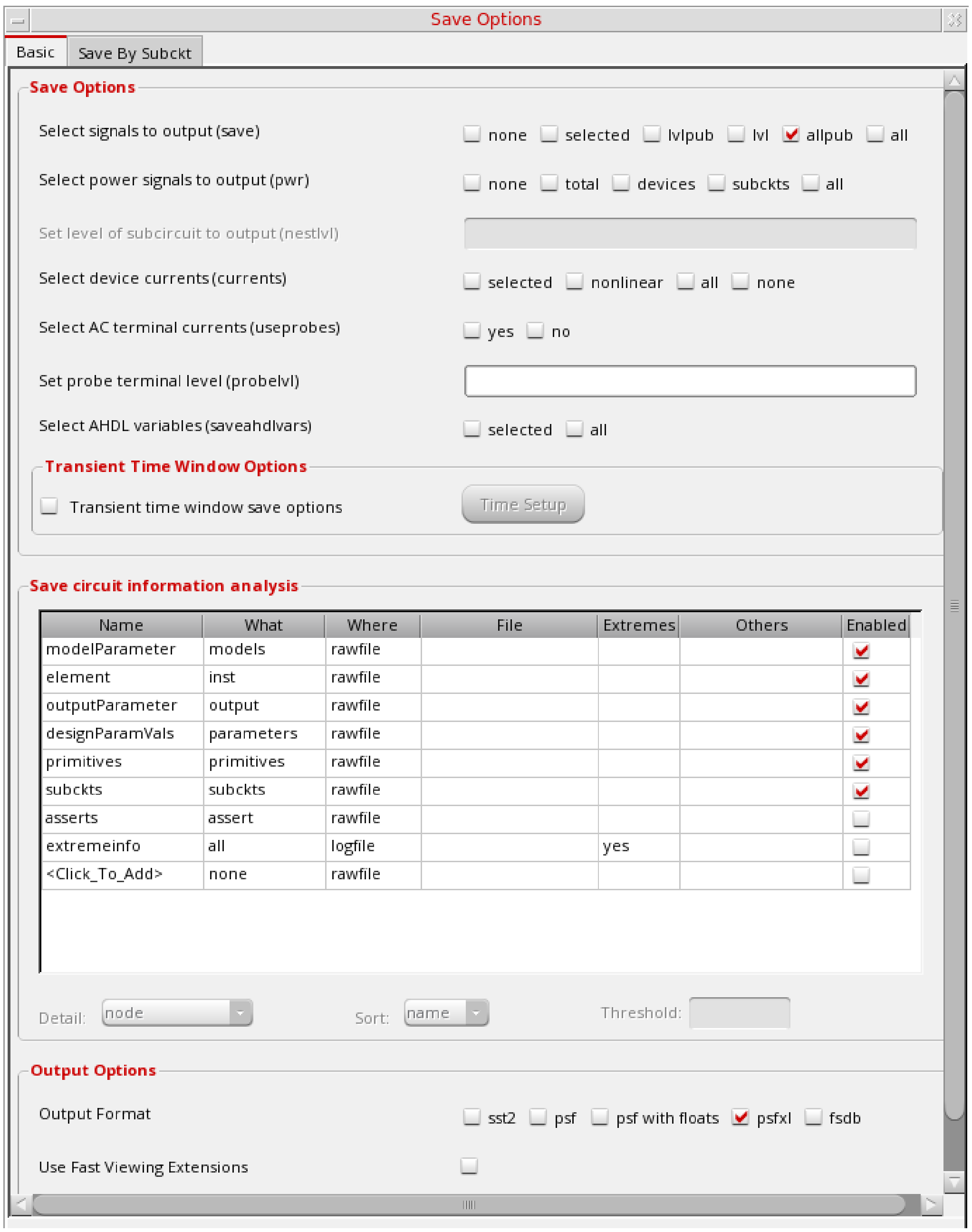
Click *OK*.

11. Select *Outputs – Save All*.

The *Save Options* form is displayed.

12. In the *Select signals to output* section, make sure that *allpub* is selected.

Figure 4-78 Save Options Form



This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

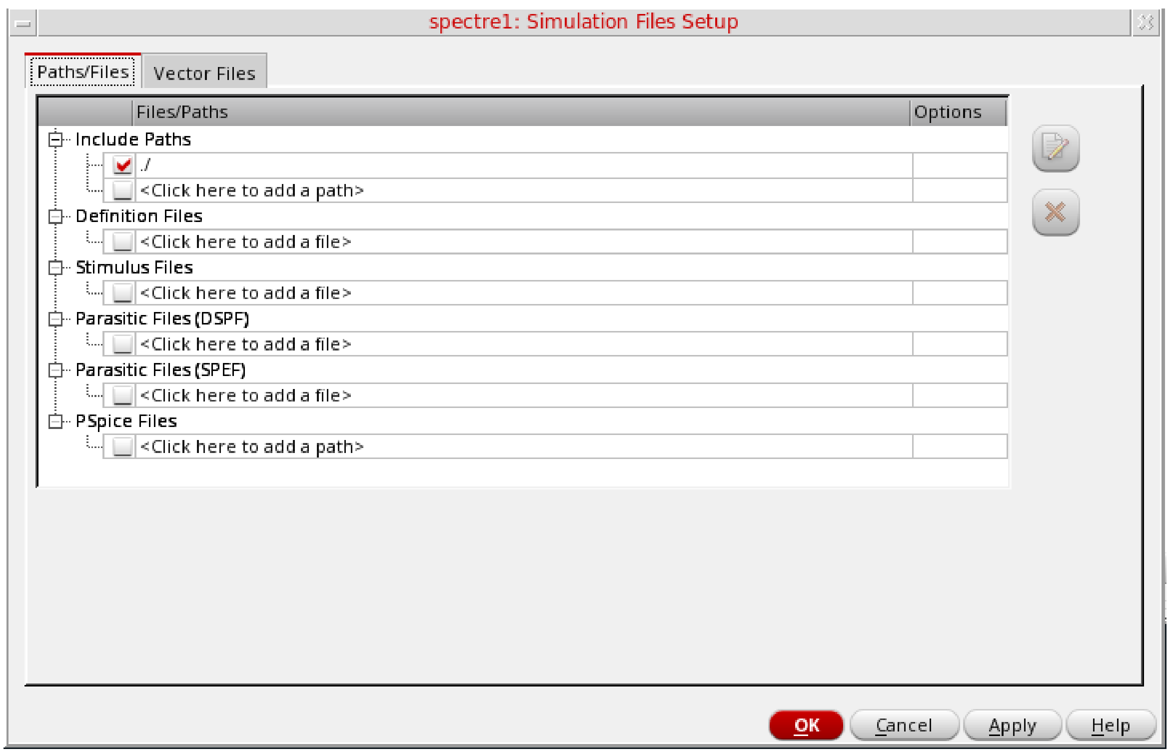
In the *Select signals to output* section, make sure that *allpub* is highlighted. This is the default which saves all node voltages at all levels of hierarchy, but it does not include the node voltages inside the device models.

13. Click *OK* to close the *Save Options* form.

Setting Up Model Libraries

1. In ADE Explorer, select *Setup - Simulation Files.* The Simulation Files Setup form is displayed, as shown below.

Figure 4-79 Simulation Files Setup Form



2. Ensure that the Include Path is set as shown above and click *OK* to close the form.

3. Select *Setup – Model Libraries*.

The *Model Library Setup* form is displayed.

4. In the *Model Library File* field, type the following in the name of the model file:

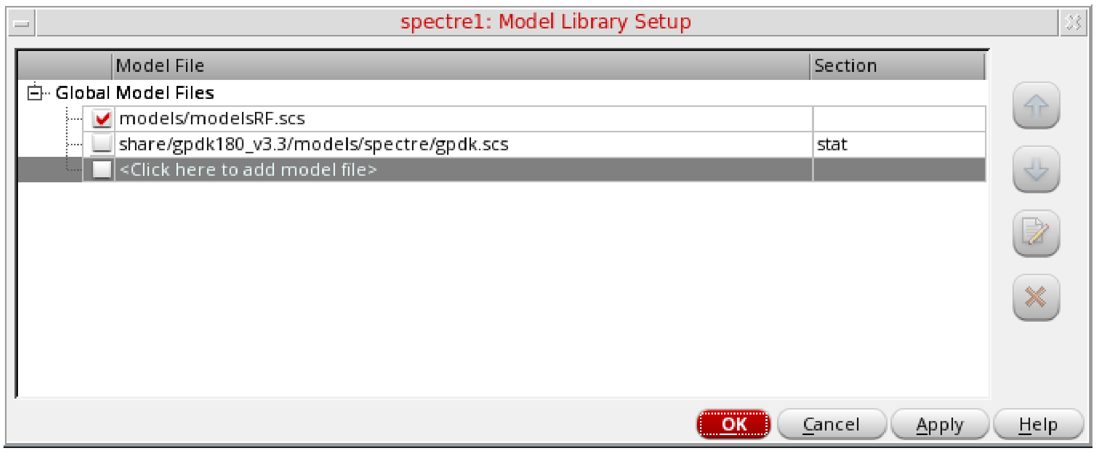
```
models/modelsRF.scs
```

Alternately, you can click the *Browse* button and browse to the `modelsRF.scs` model file.

5. Ensure that the Model File name is selected.
6. Click *Apply*.

   The *Model Library Setup* form looks like the following:

Figure 4-80 Model Library Setup Form



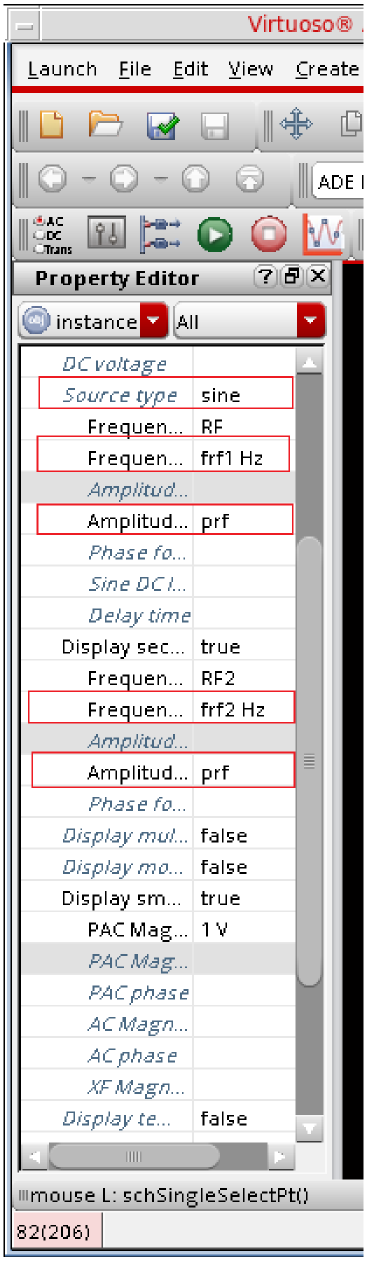
7. Click *OK* to close the *Model Library Setup* form.

Setting Up the Simulation

First view the input port settings.

1. In the schematic, select the RF source (port) just to the right of the *Input* label.
2. Note that the *Property Editor* (left side of schematic) displays the properties of the selected port. For instructions on how to re-size the *Property Editor* window, see the *Virtuoso Schematic Editor L User Guide*.

Figure 4-81 Property Editor Showing Properties of Selected Port



3. Use the scrollbar on the right of the form, if necessary, so that you can view all of the properties.

   Note that the *Source type* is set to *sine*. This enables the RF tones.

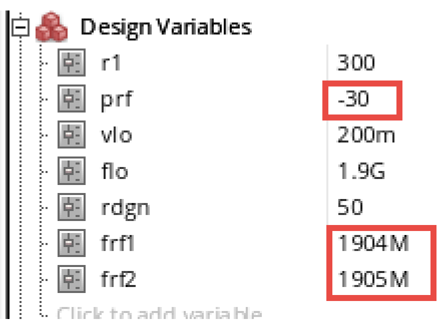
   Note that the first sinusoid's frequency is set to the variable *frf1* and the second sinusoid's frequency is set to the variable *frf2*.

Both sinusoids have the amplitude set to the variable *prf*.

4. In the *Design Variables* section of ADE Explorer, verify that variables are set as follows:

   - ❑ *frf1* to *1.904G*. This is the first RF frequency.
   - ❑ *frf2* to *1.905G*. This is the second RF frequency.
   - ❑ *prf* to *-30*. This is the input power level for both RF tones.

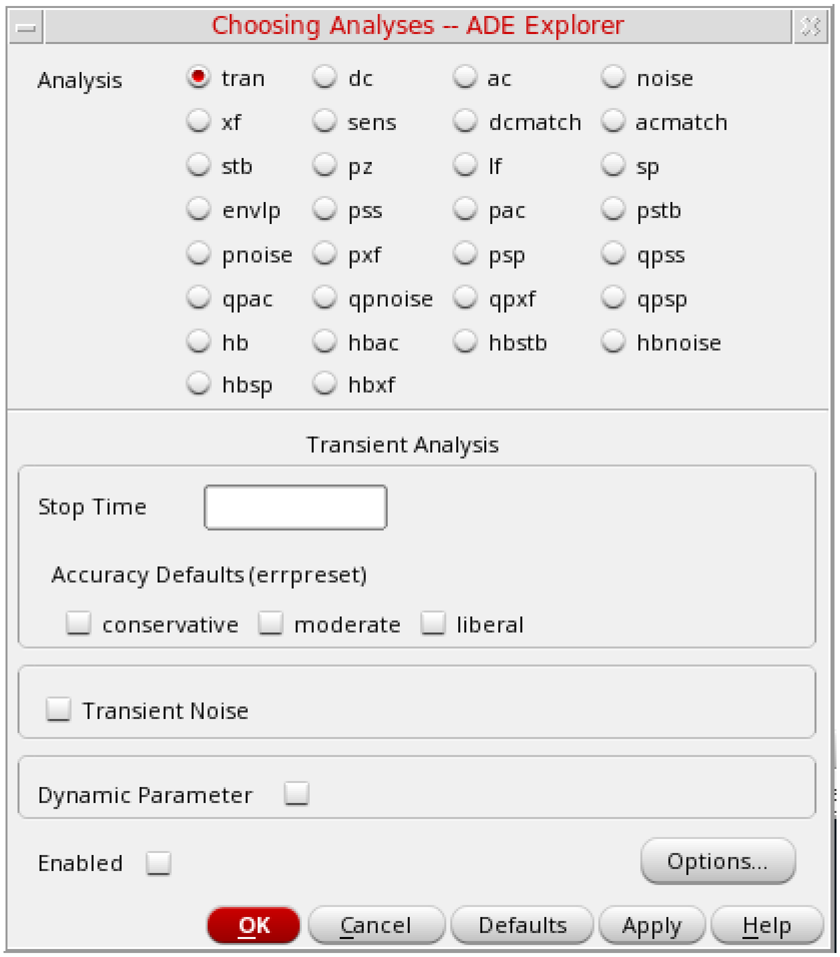
Figure 4-82 Design Variables section



5. Click the *Choosing Analyses* ()icon on the right side of the ADE Explorer window to open the *Choosing Analyses* form.

   The *Choosing Analyses* form is displayed, as shown below.

Figure 4-83 hb Choosing Analyses Form



6. In the *Analysis* section, choose *hb*.

   Harmonic balance can set harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. To enable this, select *Decide automatically* or *Yes* for the *Run Transient* selection in the *Transient-Aided Options* section. This single action will cause a transient analysis to be run until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1* (when *Frequencies* is selected) or for the tone that has *tstab* enabled (when *Names* is selected).

7. In the *Transient-Aided Options* section of the form, leave the settings at their default values, unless otherwise noted.

   a. For *Run transient?* select *Decide automatically.*

      Run transient? will run the LO signal using the transient (In SpectreRF, this is called the *tstab* interval) for a short period of time. At the end of *tstab*, an FFT is performed, and this is used as the starting point in the harmonic balance analysis. Doing this improves the convergence of hb by giving it a better starting point at the cost of a short transient analysis.

b. For *Stop time (tstab), auto* is automatically populated in the field.

When *auto* is selected for *Stop time*, a small number of periods of the LO is run using the transient analysis. During this time, the signal is checked for steady-state conditions. If steady-state is not reached in the initial number of periods, more periods can be added automatically by the simulator. Using this feature allows an accurate FFT for the starting point in the hb iterations.

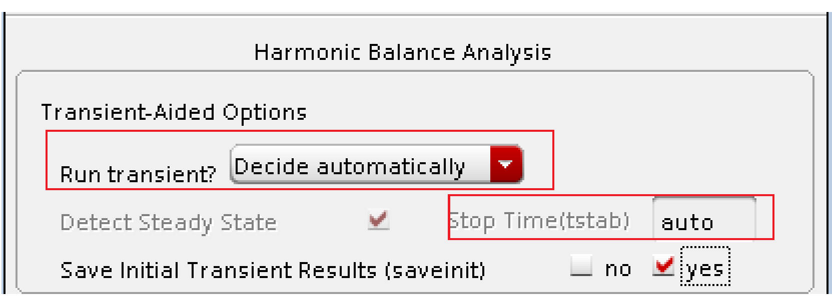
When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide automatically* simplifies the setup for harmonic balance, and produces correct answers from hb without needing to know how to set hb up manually.

c. For *Save Initial Transient Results (saveinit)*, select *yes*.

During the transient-assisted HB simulation, a transient simulation runs before the frequency domain iteration of harmonic balance. Only the signal in *Tone1* is enabled for this. Make sure you set the signal that causes the largest amount of distortion in the circuit as *Tone1*. This is usually the LO signal. At the end of the *tstab*, an FFT is run and its result is used as the starting point for the frequency domain iterations.

All the signals are applied and the simulation is done in the frequency domain. Only the signals, harmonics, and the mixing products are calculated by hb.

Figure 4-84 Transient Assisted Harmonic Balance



8. In the *Tones* section, select *Names*. When *Names* is selected, the *Tones* portion of the form expands. All the sources in the top-level schematic are read into the form automatically. Only one signal can have transient assist. When *Tones* is set to *Names*, the signal with *tstab* set to *yes* can have transient assist. Choose the signal that produces the largest amount of distortion for transient assist, in this case, it is LO.

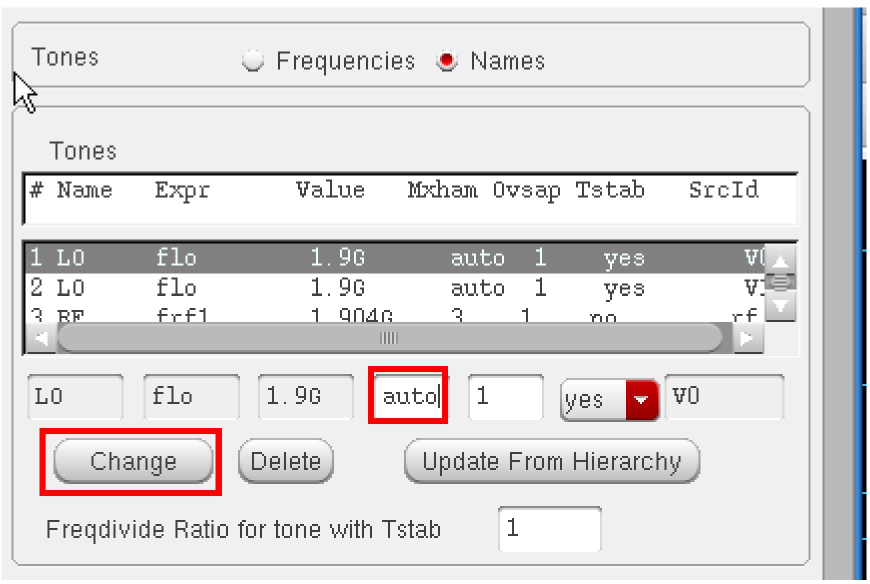
9. Select *LO* in the *Tones* section.

You can use hb with up to four signals present in the circuit. In this circuit, there are three tones, the LO and two RF tones. The two LO signals have the same name, the same frequency, and are considered a single tone. Whenever you have two signals at the *same* frequency, make sure you set the *Frequency Name 1* or *Frequency Name 2* property on the source to the *same* name as it was done in this example. When *Tones* is set to *Names*, the simulator considers both of the sources as a single input signal.

You viewed the names (*frf1* and *frf2*) in the input port sources in an earlier step.

Figure 4-85 Tones Section of the Choosing Analyses Form with Tones Set to Names

.



10. Ensure that *Tstab* is set to *yes* in the *Tones* section for the LO tone. Set *tstab* to *yes* on the signal that causes the largest amount of distortion in the system. (In this example, that is the *LO* tone).

11. On the LO tone, ensure that *Mxham* is set to *auto* (type *auto* in the field, if necessary). Since the circuit is mildly nonlinear (that is, the signals have no sharp edges), leave *Ovsap* (oversample) set to the default value of *1*.

12. Click *Change*. The form updates. Both LO tones now have *Mxham=auto* and *Tstab=yes*.

Note: If not using *Mxham* as *auto* on the LO tone, change the *Maximum harmonics* on the LO tone from *3* to *10*. Place the cursor in the field under *Mxham* and type *10*. Since the LO signal is well above compression, a higher number of harmonics is advised. When you set harmonics manually, if a signal is below compression, start with 3 harmonics. If a signal is at compression, start with 5 harmonics. If a signal is above compression, start with 7 harmonics. Run the simulation. Now increase the number of harmonics on each tone separately by about a factor of 1.5, and run again. If the result changes, increase the harmonics again. If the result does not change, the original number of harmonics was enough, and you might be able to decrease the number of harmonics. For fastest runtime with full accuracy, use the smallest number of harmonics for each tone that is applied to the circuit.

13. Select the RF tone in the *Tones* box. Ensure that *Mxham* is set to `3`. Do the same for *RF2*.

14. Click *Change* to update the form.

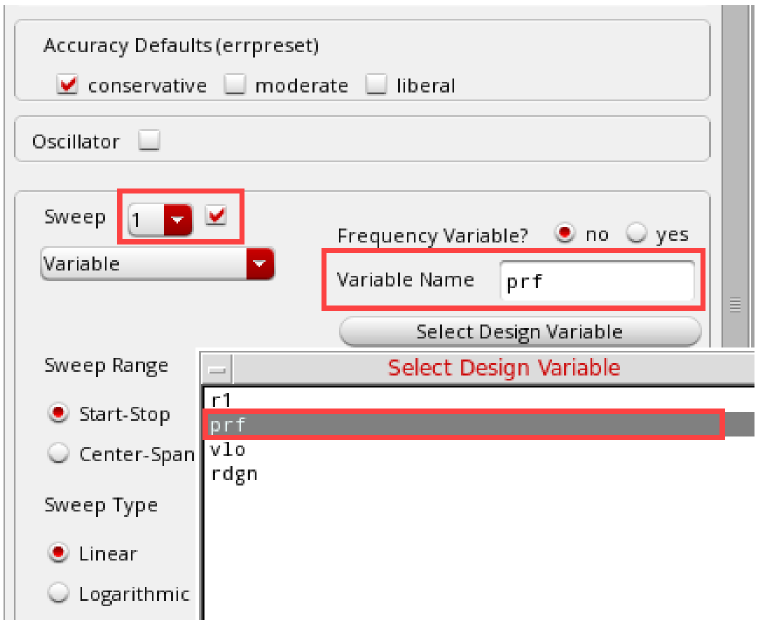
15. For the Harmonics selection (just below the list of inputs), leave it at the default value of *Default*. If you click *Select*, you would have the option of setting the method to *diamond*, *funnel*, or *axis* cut. (For more information on diamond, funnel, and axis cut, see Chapter 3: Frequency Domain Analyses: Harmonic Balance in the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide)

16. Set *Accuracy (errpreset)* to *conservative*. The third and fifth order intermodulation distortion is calculated with this setup.These amplitudes are small, and thus high accuracy is needed.

17. In the *Sweep* section, click *Sweep*. Leave it set it to the default value of *1.*

18. For *Frequency Variable?* select *no*. You will be sweeping input power rather than frequency.

Figure 4-86 Sweep Section in hb Choosing Analyses Form



19. Click *Select Design Variable* and select the *prf* variable for the power sweep in the *Select Design Variable* form.

20. *Click OK* in the *Select Design Variable* form.

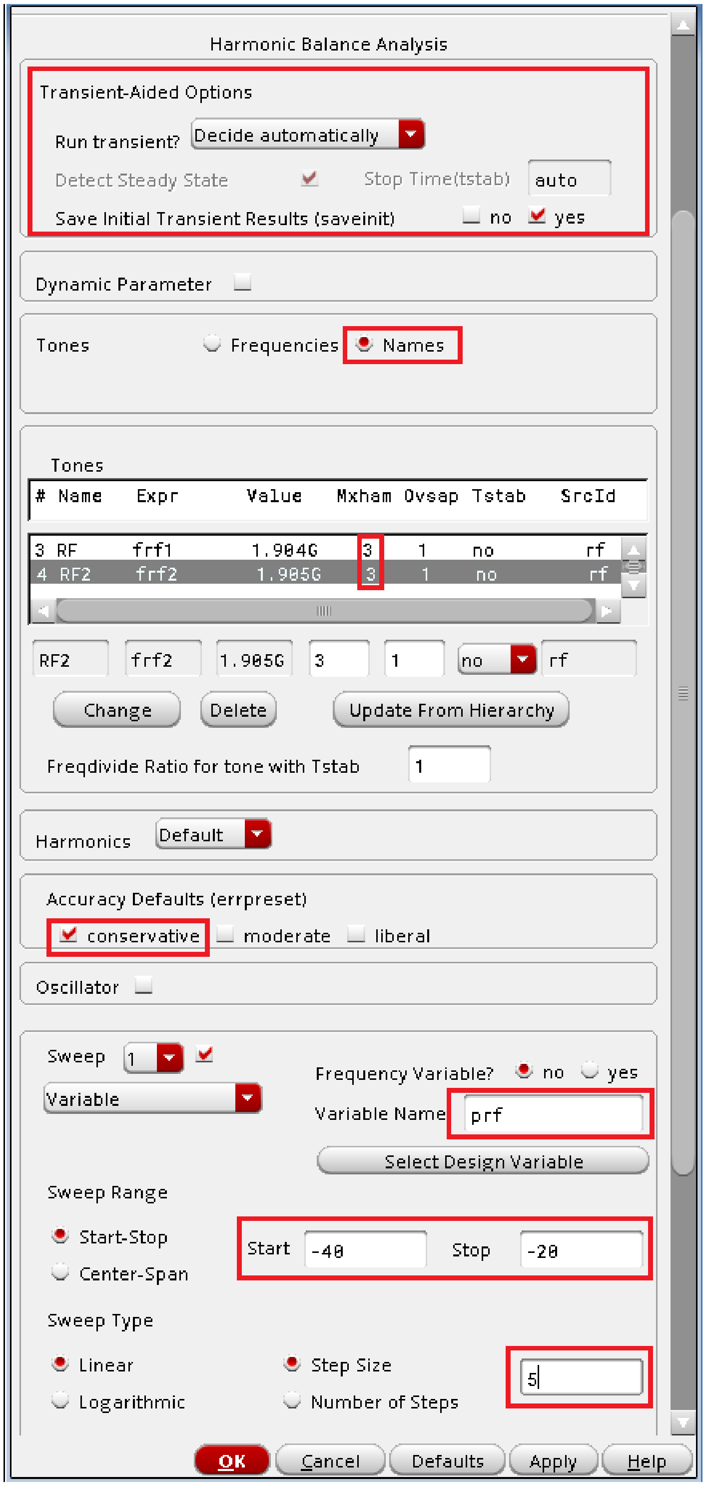
21. In the *Sweep Range* section, type `-40` in the *Start* field and `-20` in the *Stop* field.

22. Select *Sweep Type* as *Linear* and type `5` in the *Step Size* field.

Note: Because this is a small-signal measurement, you want to make sure your power sweep range does not go too high (stay operating at least 10dB below the 1dB compression point).

The completed *Choosing Analyses* form should look like the following:

Figure 4-87 Choosing Analyses Form Setup for IP3

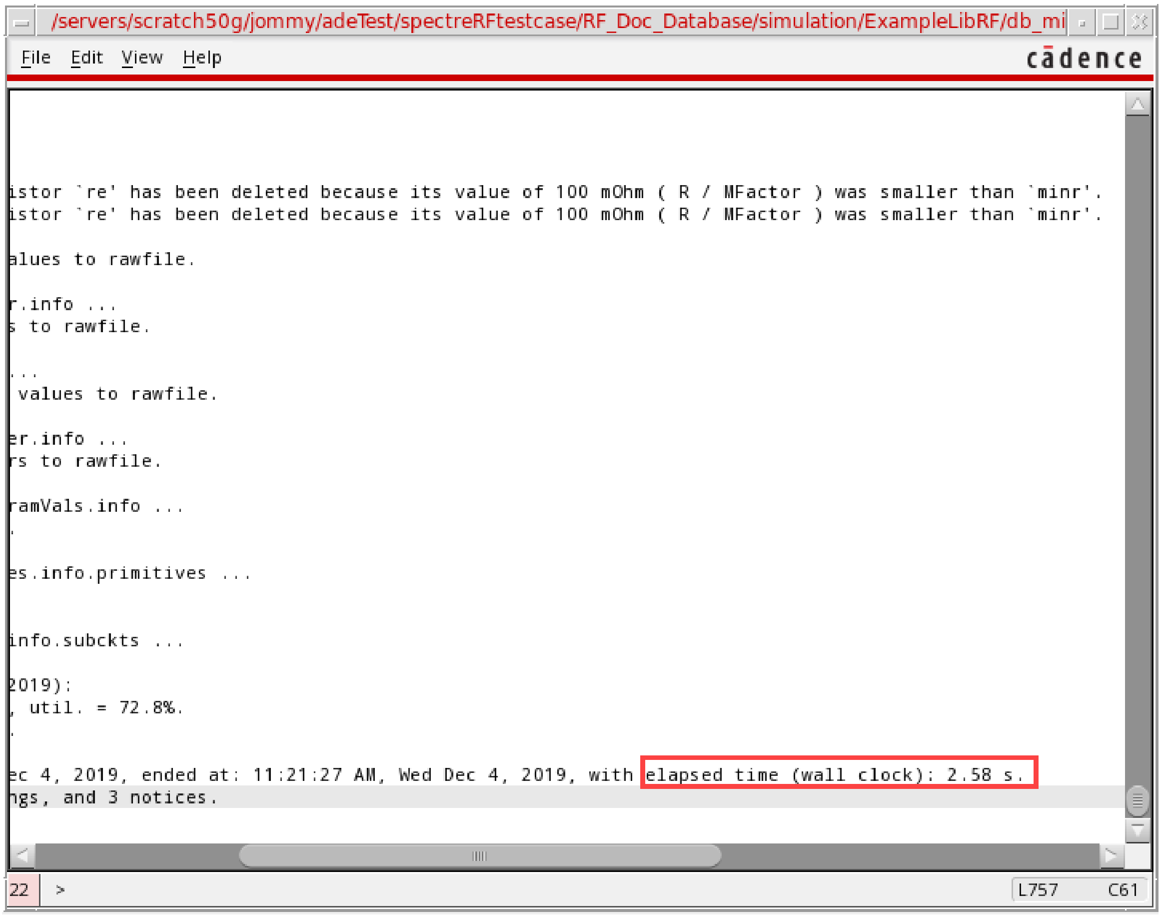


23. Click *OK*.

24. Start the simulation. From the ADE Explorer or the Schematic, click the green arrow icon ( )on the right side of the window.

The Spectre output window is displayed with the simulator status information. Note the time it takes to run the simulation.

Figure 4-88 Spectre.out File Showing Elapsed Time



25. After the simulation finishes, plot the output spectrum.

In ADE Explorer, choose *Results - Direct Plot - Main Form.* Alternately, you can click the *Direct Plot* icon ( )in the Schematic window.

26. In the *Analysis* Section, select *hb_mt*.

27. In the *Function* Section, select *IPN Curves.*

28. From the *Select* drop-sown, choose *Net (Specify R)*.

29. Leave the *Resistance* set to the default value of *50*.

30. Select *Variable Sweep ("prf")* for *Circuit Input Power*.

31. Leave *Input Power Extrapolation Point (dBm)* set to the default (first point in the sweep).

32. Choose *Input Referred IP3, 3rd Order.*

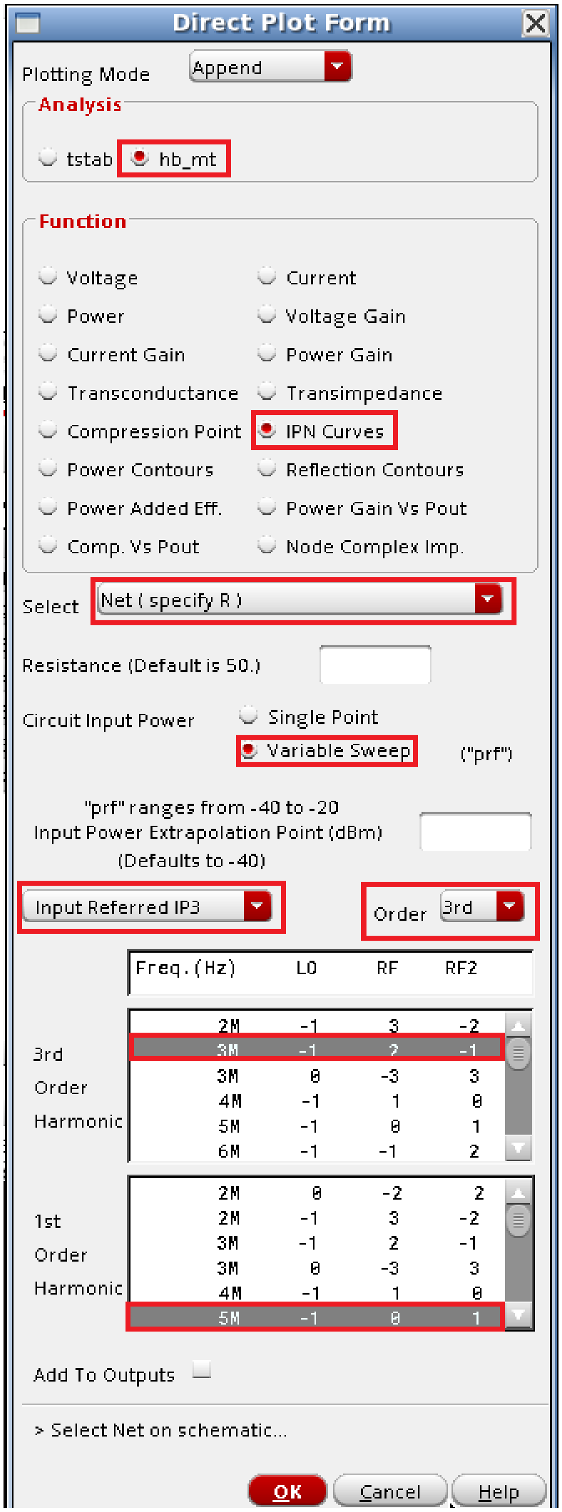
33. For the *3rd Order Harmonic*, choose the *3M -1 2 -1* entry.

Depending on the input frequencies chosen, hb may calculate multiple outputs at the same frequency. To determine which entry to select, add up the absolute value of all of the terms (for example, `|-1| + 2 + |-1| = 4` and `0 + |-3| + 3 = 6`) and choose the one with the lowest value. This is the lowest order term, which is usually the term that is desired.

34. For the *1st Order Harmonic,* choose the *5M -1 0 1* entry.

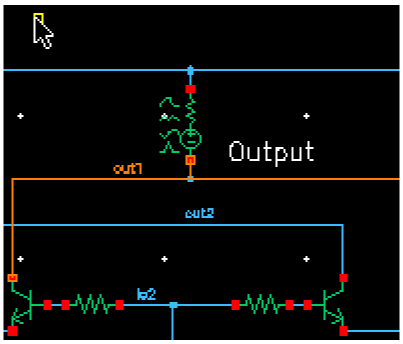
The completed *Direct Plot Form* should look like this:

Figure 4-89 Direct Plot Form - IP3 from HB simulation



35. Click the *out1* net on the schematic just below the *Output* label, as shown below. Then press the *Esc* key, with the mouse cursor in the schematic window. This closes the *Direct Plot Form*.

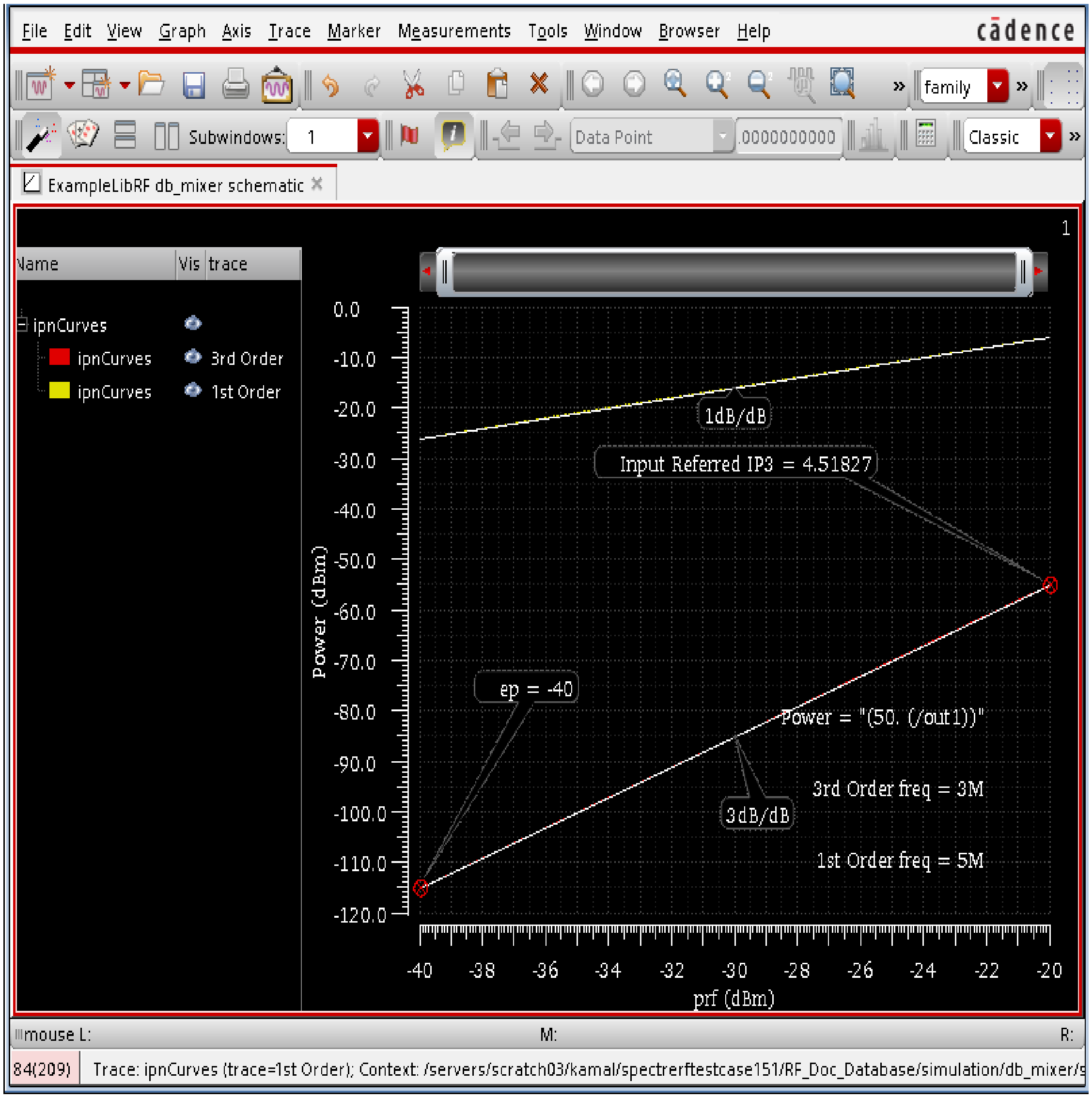
Figure 4-90 Select Output Net on Schematic



The IP3 Plot is displayed in the waveform window.

Figure 4-91 IP3 Plot from 3-tone Harmonic Balance simulation

.



Intermodulation products increase at rates that are multiples of the fundamentals. In the small-signal region, third-order terms increase 3dB per dB and the second-order terms increase 2dB per dB.

Note: If you do not see the IP3 readout, you may need to click in the graphics area to deselect the marker, then select and move the Input Referred IP3 readout so that it is positioned in the visible area of the graph.

In the previous plot, you can see that the circuit is operating within the small signal region. The third-order curve is following a 3dB/dB slope. Note the IP3 measurement. You will compare this to the IP3 measurement using the Rapid IP3 methodology.

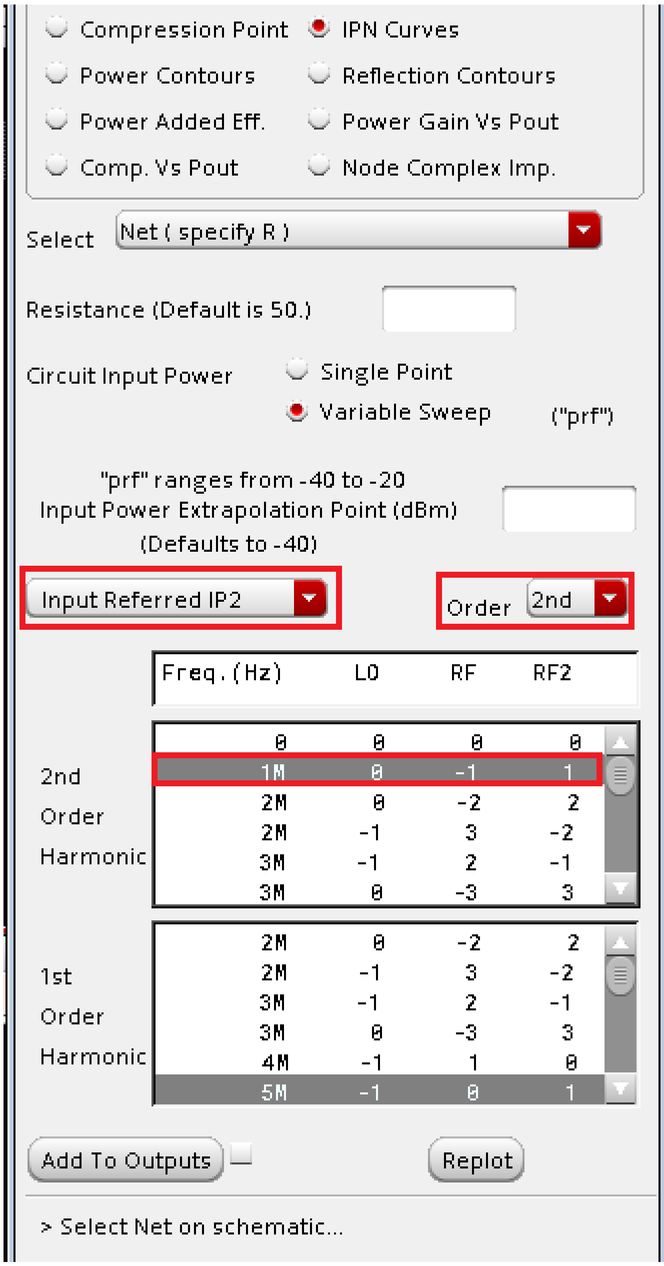
In the next section, you will plot the IP2 curves.

Plotting the IP2 Curves

1. In the *Direct Plot Form*, change the *Plotting Mode* to *Replace*.
2. Change the *Order* to *2nd*.
3. Choose *Input Referred IP2.*
4. For the *2nd Order Harmonic*, choose the *1M 0 -1 1* entry.
5. For the *1st Order Harmonic*, choose the *5M -1 0 1* entry.

   The *Direct Plot Form* should look like the following:

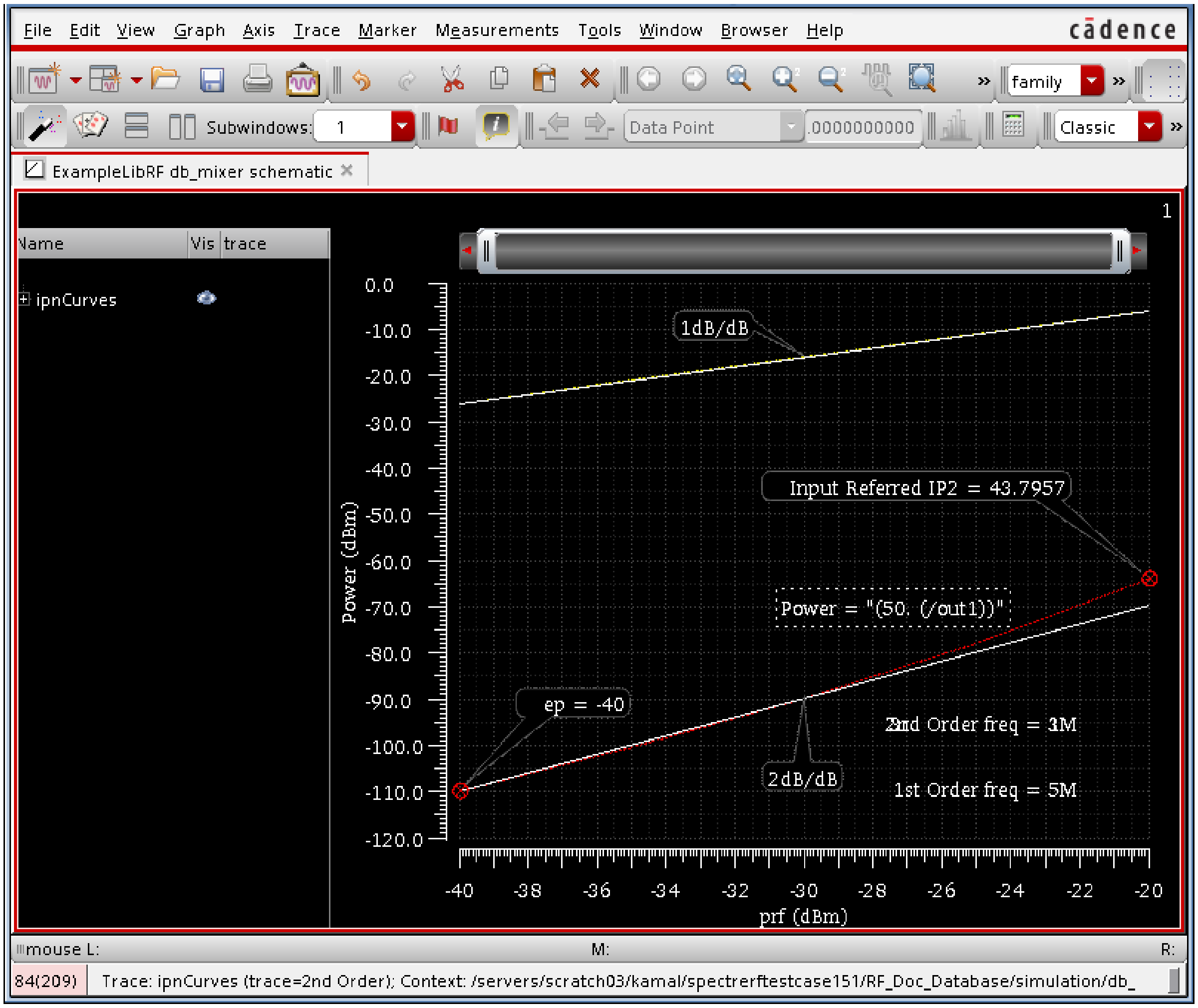
Figure 4-92 Direct Plot Form for Input Referred IP2 Plot



6. Click *Replot*.

The IP2 Plot is shown in the following figure. Although it appears that the IP2 point is not at the intercept point of the first and second order curves, it does actually calculate that point. It is attached to the last result of the second order curve so the X axis can display just the simulation result. Also, note that above -30 dBm, the input power is high enough to cause deviations from the small-signal 2dB/dB curve. Since the extrapolation point at the first value of the sweep is used to do the IP2 calculation, the IP2 result is correct because the data follows the 2dB/dB curve below -30 dBm input power.

Figure 4-93 IP2 Plot from 3-tone HB Analysis



Note: If you do not see the IP2 readout, you may need to click in the graphics area to deselect the marker, then select and move the Input Referred IP2 readout so that it is positioned in the visible area of the graph.

Note the IP2 measurement for comparison later when measuring IP2 using the Rapid IP2 methodology.

Leave the db_ mixer schematic and ADE Explorer opened. You will use them in the next section.

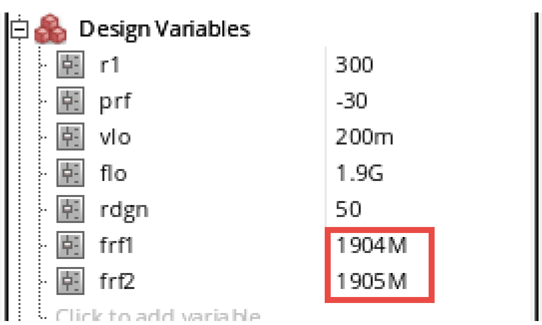
Rapid IP2 and IP3 Measurement

Another way to calculate IP3 and IP2 is to apply the LO only in the hb analysis and select Rapid IP2/IP3 from the hbac analysis. This is the fastest approach but it is limited to a small-signal IP3 measurement. It is also important to note that Rapid IP3 is not recommended for passive FET mixers where bsim3 or bsim4 models are used. In a small-signal application, this technique and the 3 tone hb method produce answers typically within 0.1dB of each other for IP3. For IP2, if extreme accuracy measures like very small reltol and vabstol, and a large number of harmonics are set in the 3-tone HB analysis, then closer agreement can be expected. These measures cause the 3-Tone HB analysis runtime to increase dramatically.

IP2/3 is calculated from a first and second/third order term. Because of the small-signal projection, hbac is used to measure IP2 and IP3. hb analysis applied to the LO tone captures the nonlinearity of the circuit created by the LO and the resultant frequency translation. The hbac analysis is used to calculate the amplitude of the first, second, and third order terms that are downshifted by the LO. The *Analog Design Environment* has a *Direct Plot* function to automate the IP2 and IP3 calculations.

1. In ADE Explorer, set the *frf1* and *frf2* design variables to `0`.

Figure 4-94 Design Variables Section of ADE Simulation Window

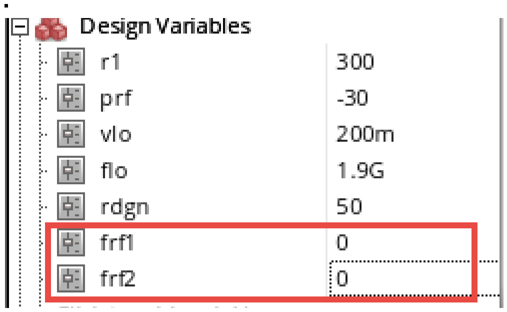


2. In the *Design Variables* section, click *1.904G* to the right of the variable *frf1* and type `0` and press *Enter*.
3. Select *1.095G* to the right of the variable *frf2* and type `0` and press *Enter*.

   Setting the input frequencies to `0` disables the production of waveforms for the large-signal analyses like tran, pss, and hb (harmonic balance). The updated *Design Variables* form looks like the following:

Figure 4-95 Design Variables Section of ADE Simulation Window with frf1 and frf2 set to zero

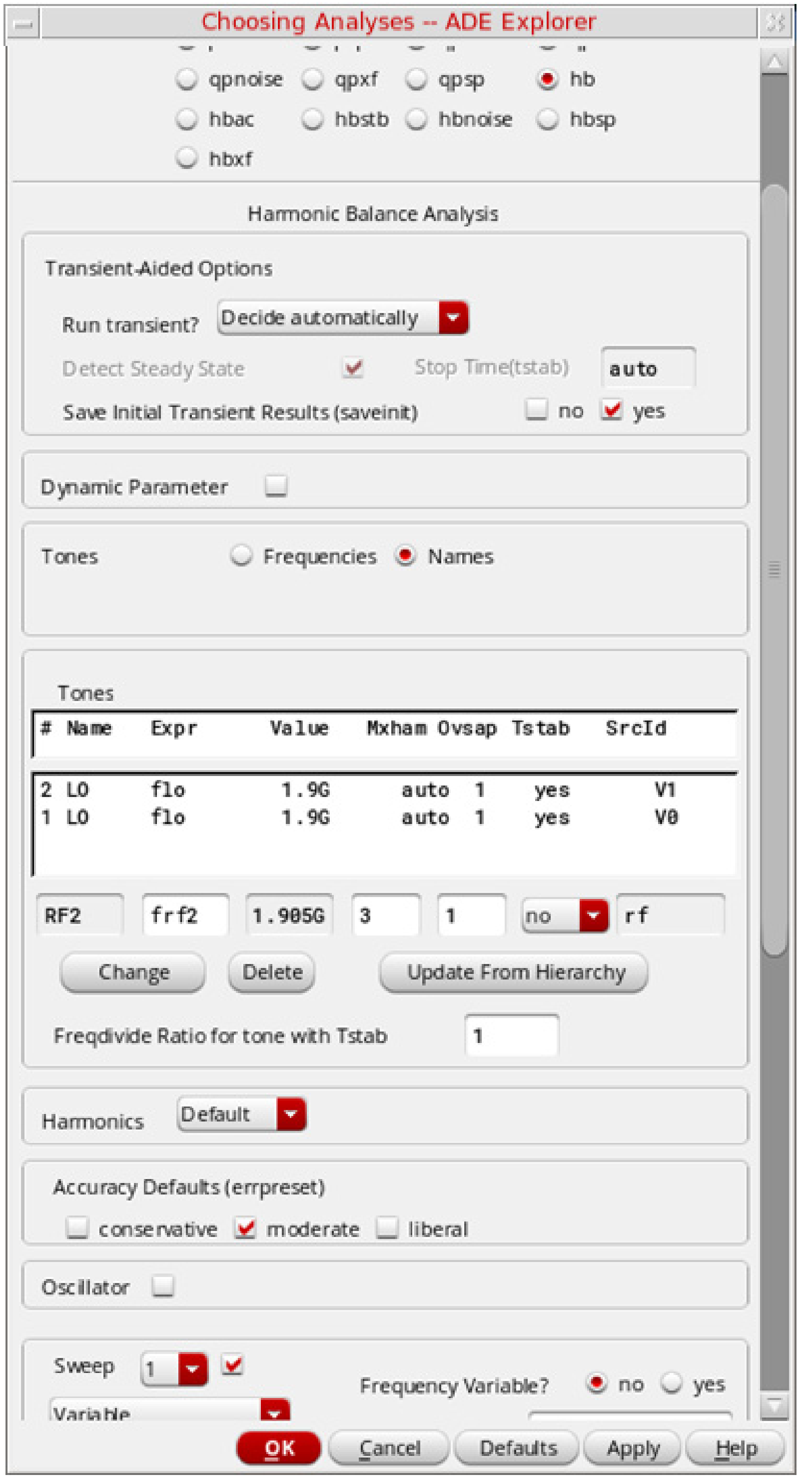
.



4. In ADE Explorer, select *Analyses - Choose.*

The *Choosing Analyses* form is displayed. Select *hb* for the *Analysis* type. The form expands, as shown in the following figure:

Figure 4-96 Choosing Analyses Form for hb Analysis



5. . Harmonic balance can set harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. To enable this, select *Decide automatically* for the *Run Transient s*election in the *Transient-Aided Options* section.

This single action will cause a transient analysis to be run until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1* (when *Frequencies* is selected) or for the tone that has *tstab* enabled (when *Names* is selected).

In the *Transient-Aided Options* section of the form, leave the settings at their default values unless otherwise noted.

a. For *Run transient?* select *Decide automatically*. (this is the default)

Run transient? will run the LO signal using the transient (In SpectreRF, this is called the *tstab* interval) for a short period of time. At the end of *tstab*, an FFT is performed, and this is used as the starting point in the harmonic balance analysis. Doing this improves the convergence of hb by giving it a better starting point at the cost of a short transient analysis.

b. For *Stop time (tstab), auto* is automatically populated in the field.

When *auto* is selected for *Stop time*, a small number of periods of the LO is run using the transient analysis. During this time, the signal is checked for steady-state conditions. If steady-state is not reached in the initial number of periods, more periods can be added automatically by the simulator. Using this feature allows an accurate FFT for the starting point in the hb iterations.

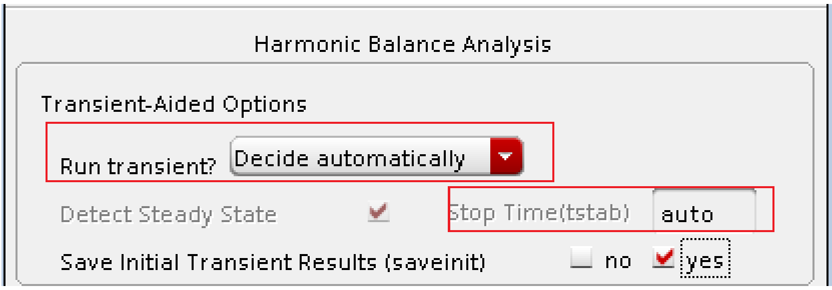
When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide automatically* simplifies the setup for harmonic balance, and produces correct answers from hb without needing to know how to set hb up manually.

c. For *Save Initial Transient Results (saveinit),* select *yes*.

During the transient-assisted HB simulation, a transient simulation runs before the frequency domain iteration of harmonic balance. Only the LO signal in *Tone1* is enabled for this measurement. At the end of the *tstab*, an FFT is run and its result is used as the starting point for the frequency domain iterations

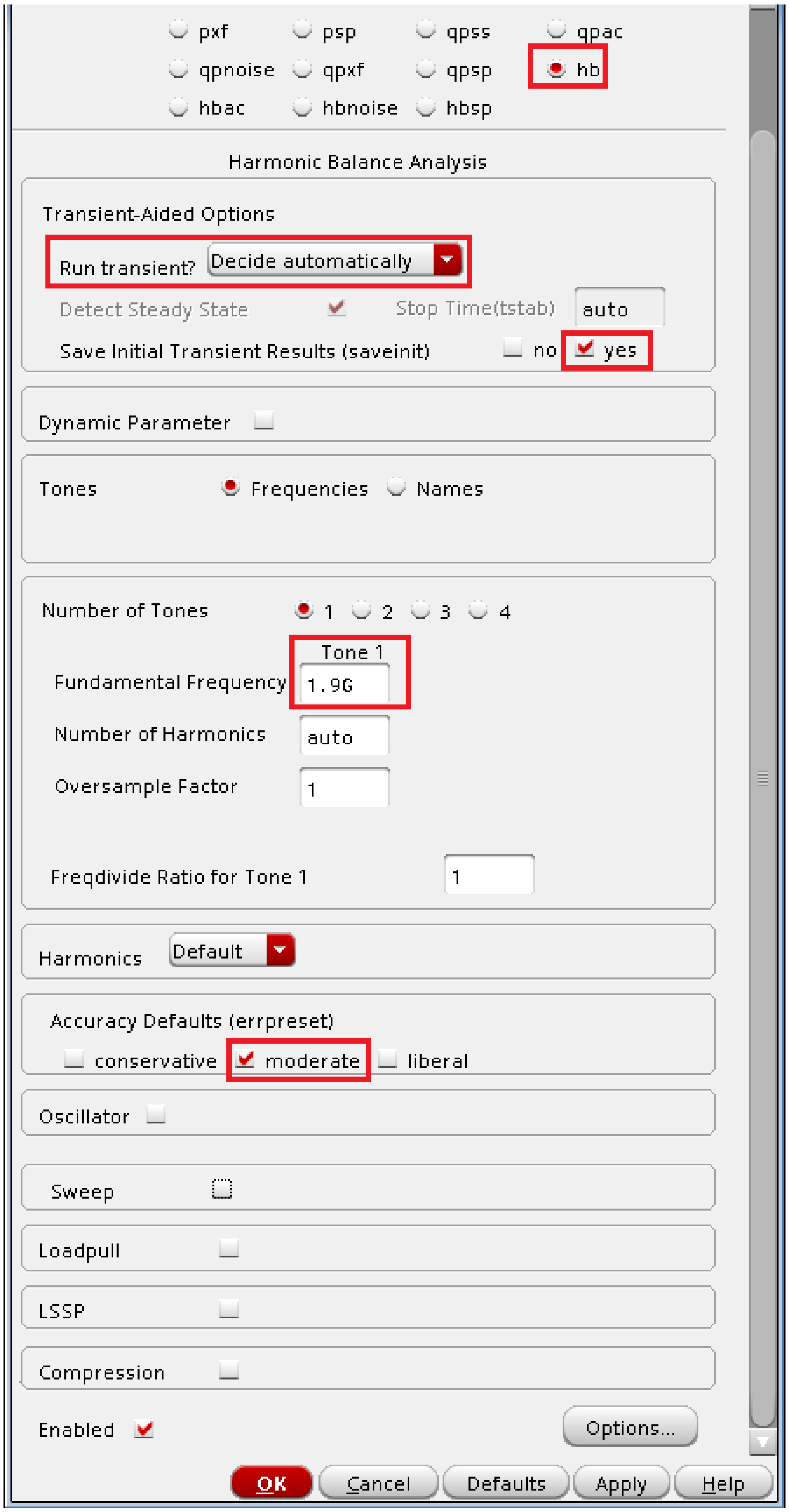
All the signals are applied and the simulation is done in the frequency domain. Only the harmonics of the LO are calculated by hb.

Figure 4-97 Transient Assisted Harmonic Balance



6. Set *Tones* to *Frequencies*.
7. The *Number of Tones* defaults to *1*.
8. Enter `1.9G` for the *Fundamental Frequency.*
9. Note that *Number of Harmonics* is set to *auto* by default.
10. Leave *Oversample factor* set to the default value of `1`. With auto harmonics, you do not need to set *oversample*. This circuit has sinusoidal-like waveforms (voltage and current), and an *oversample* of *1* is appropriate.
11. Because the first and third order mixing terms are calculated by hbac, high accuracy is not necessary in the hb analysis, so *moderate* is selected for *errpreset* and provides reasonable accuracy. The *Choosing Analyses* form should look like the following:

Figure 4-98 Choosing Analyses Form - hb Setup for Rapid IP3

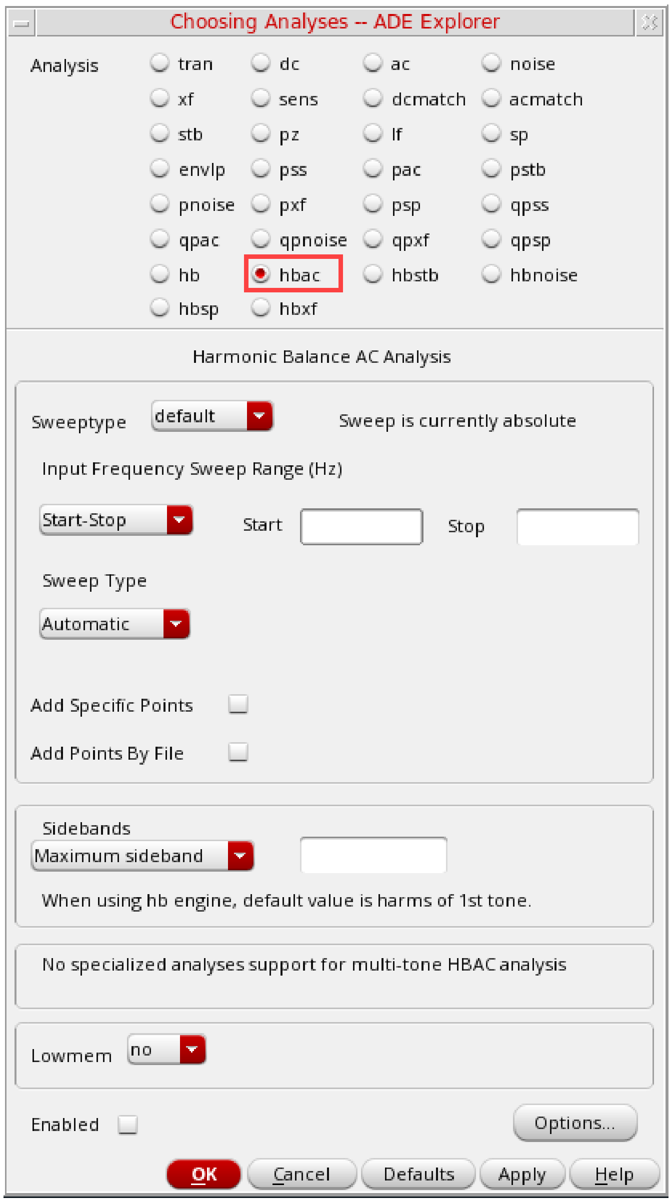


12. Click *Apply*.

Set-up HBAC Choosing Analyses Form

1. In the *Choosing Analyses* form, select *hbac* in the *Analysis* section. The *Choosing Analyses* form changes, as shown below.

Figure 4-99 Choosing Analyses Form for hbac



2. At the bottom of the form, in the *Specialized Analyses* section, select *RapidIP3*. The form expands. Just below the *Rapid IP3* selection, for *Source Type*, leave it at the default setting of *port*.
3. Type `/rf` for *Input Sources 1* and *2.* Alternately, you can click *Select* in the form and choose the *Input* rf port in the schematic.
4. Type *1.904G* in the *Frequency* Field to the right of *Input Sources 1*.
5. Type *1.905G* in the *Frequency* Field to the right of *Input Sources 2*.

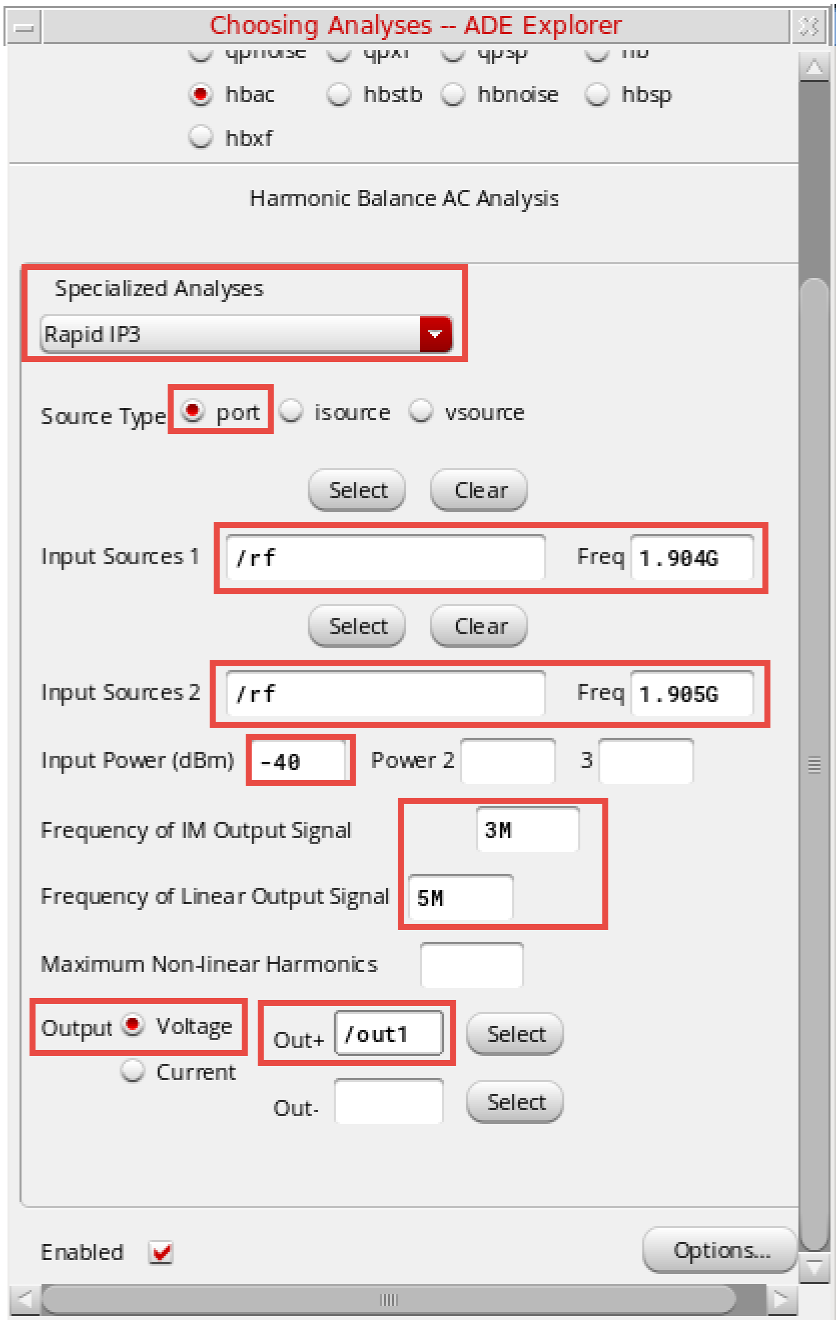
   1.904G and *1.905G* are the RF input frequencies.
6. Type `-40` for *Input Power (dBm).*
7. In the *Frequency of IM Output Signal* field, type `3M`.
8. In the *Frequency of Linear Output Signal* field, type `5M`.

   These are the linear and third-order output frequencies
9. Leave the *Maximum Non-linear harmonics* field blank. The default is fine for IP2 and IP3 measurments.
10. Select *Output Voltage* and in the *Out+* field, type `/out1`.

    The*Choosing Analyses* form should look like the following

Figure 4-100 HBAC Rapid IP3 Choosing Analyses Form

:



The selection of input power is very important for Rapid IP3. It should be set so that the system is in the small-signal range or linear region (more than 10dB below the 1dB compression point). If it is set too large, the system is in the large-signal range.

Three input power levels can be set in the Rapid IP3 *Choosing Analyses* form. To confirm if the IP3 result is accurate or not, you can set two different power levels around the chosen input power. If you get the same IP3 result, then the IP3 result is accurate, and the chosen input power level is in the suitable region.

The frequencies to be calculated depend on the choice of frequencies at the input and the LO frequency. Individual choices are provided for the frequencies which allows the selection of the smallest first order and largest second/third order product for your application.

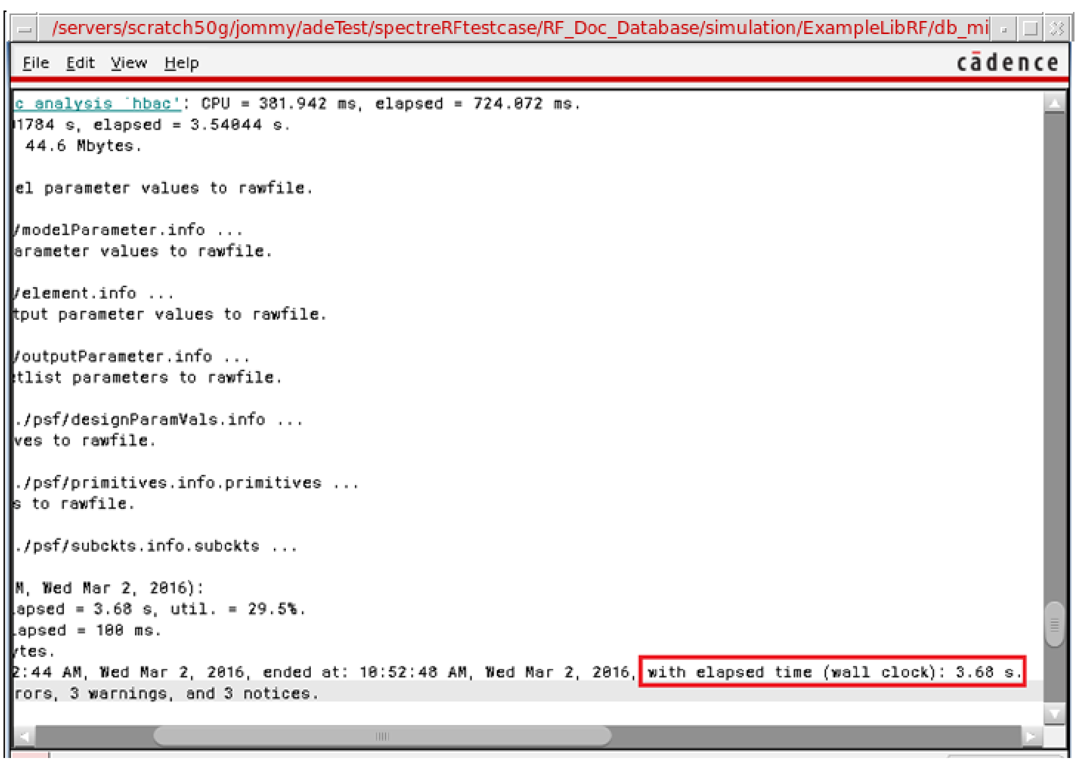
11. Click *OK* at the bottom of the *Choosing Analyses* form.

12. Start the simulation.

From ADE Explorer or the Schematic, click the green arrow icon( )on the right side of the window.

13. The Spectre output window will appear with the simulator status information. Details of the IP3 measurement are also printed in the Spectre simulation log file. Note how much quicker the simulation finishes, compared to the three-tone hb simulation used previously to calculate IP3.

Figure 4-101 Rapid IP3 spectre.out logfile

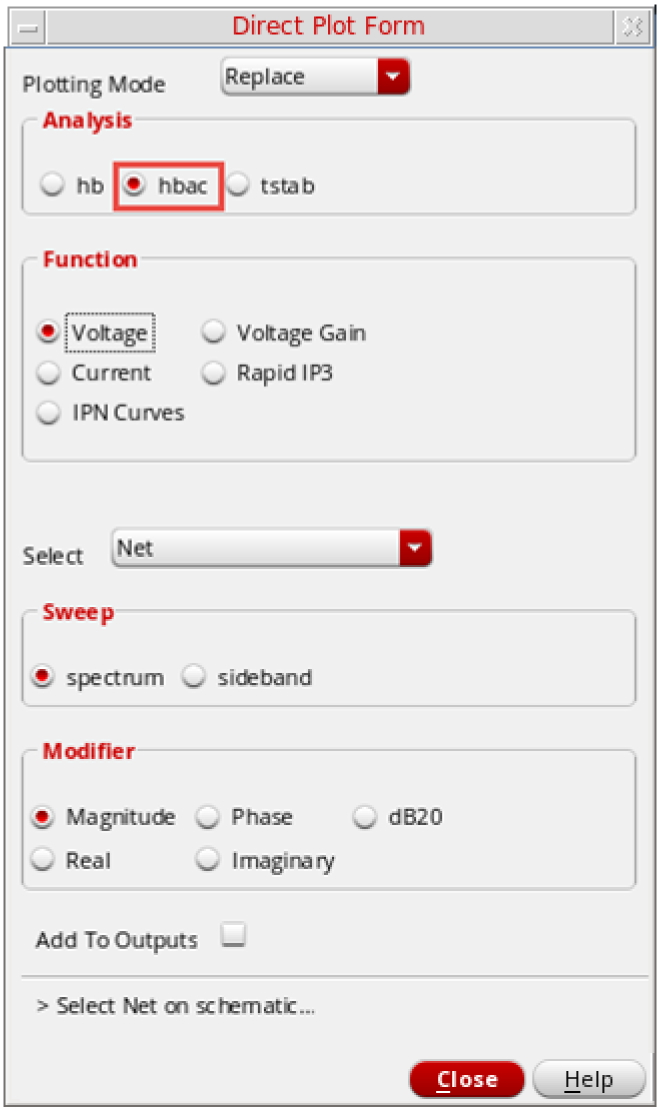


After the simulation finishes, plot the Rapid IP3.

14. In the *Analog Design Environment* window, select *Results - Direct Plot - Main Form.* You may also invoke the *Direct Plot Form* by clicking the *Direct Plot* icon ()in the schematic.

15. In the *Direct Plot Form*, select *hbac*. The form changes, as shown below.

Figure 4-102 Direct Plot Form after Rapid IP3 Simulation

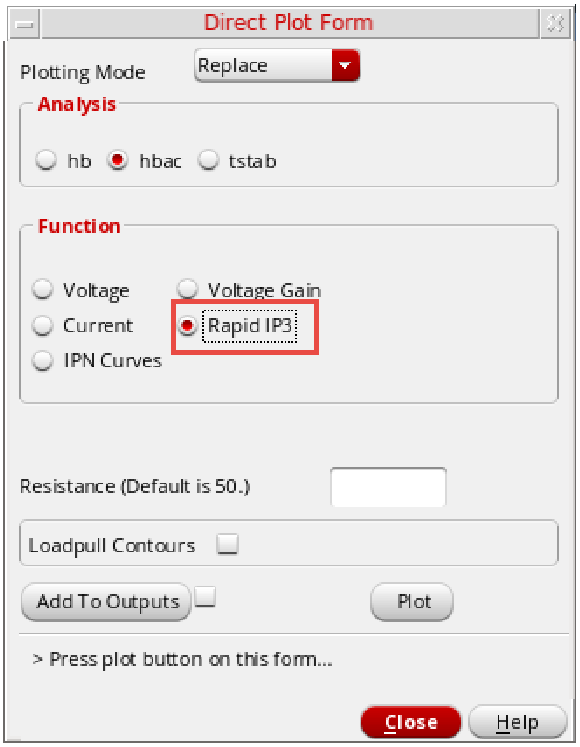


16. In the *Function* section, select *Rapid IP3.*

Leave the *Resistance* set to the default.

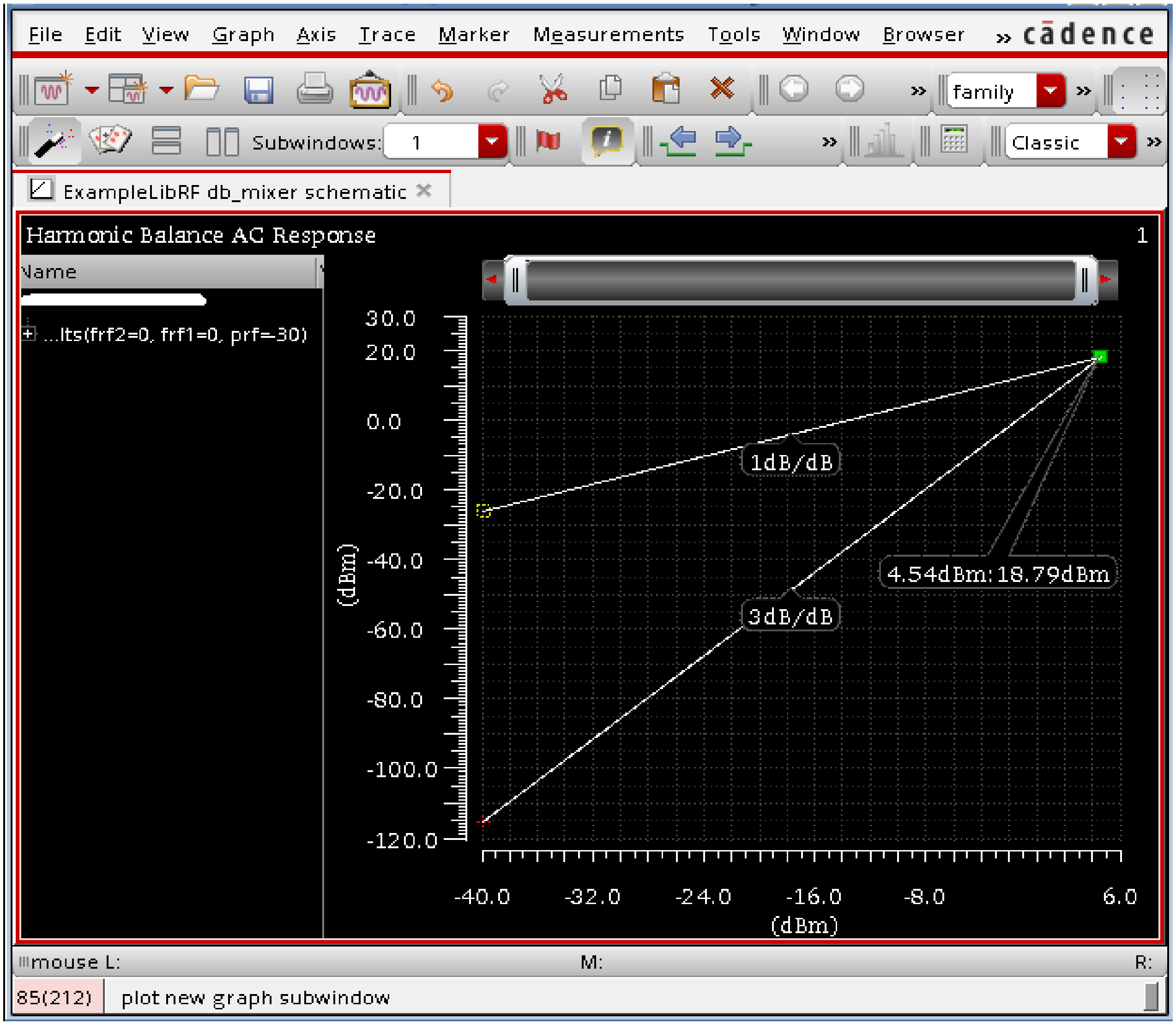
The *Direct Plot Form* should look like the following:

Figure 4-103 hbac Direct Plot Form for Rapid IP3



17. Click *Plot*. The Rapid IP3 response is plotted, as shown below.

Figure 4-104 Rapid IP3 Plot for db_mixer circuit



The input and output referred IP3 are shown in the waveform tool at the intercept point. The X axis of the plot is input power and the Y axis of the plot is output power, thus, the X-Y readout at the intercept point are the input-referred and output-referred IP3.

Note that the IP3 value from the 3 tone hb simulation (4.52) and rapid IP3 (4.54) are very close.

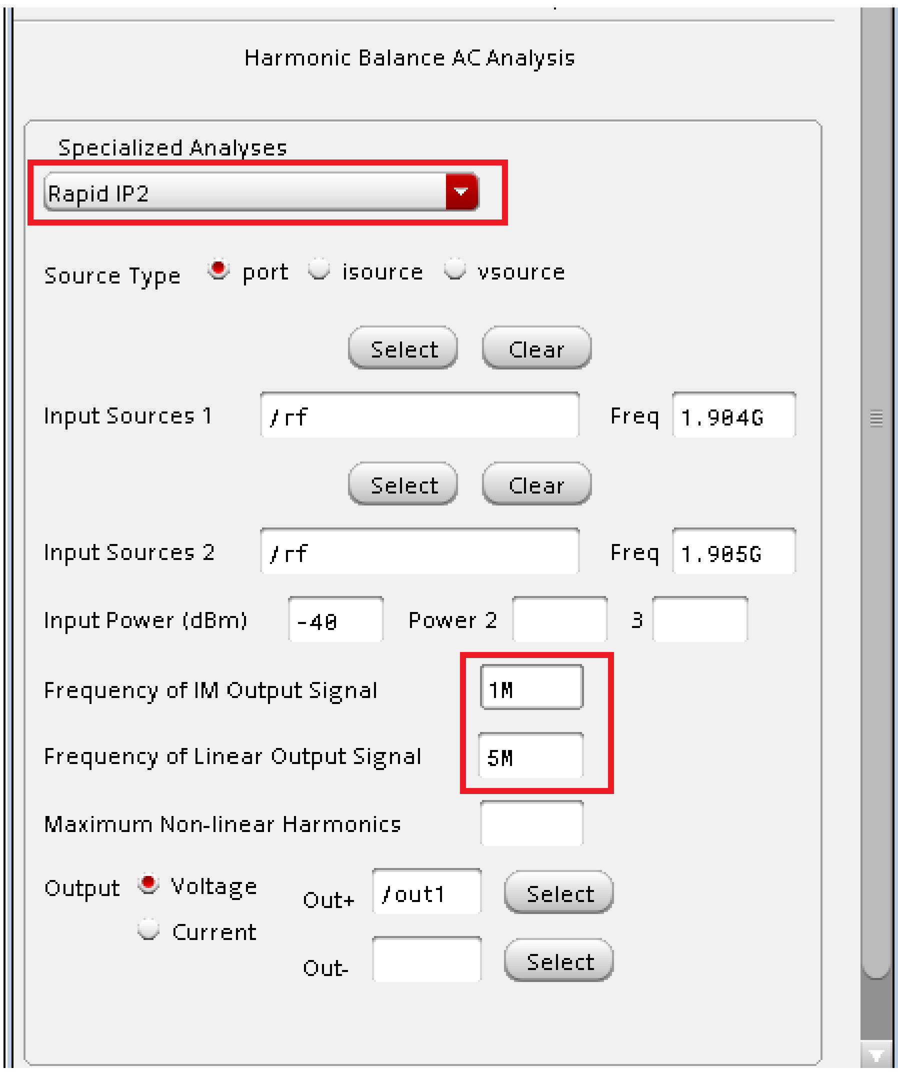
In the *Direct Plot Form*, click *Cancel*. In the next step, you will simulate and plot rapid IP2.

Simulate and Plot Rapid IP2

Next, set up the HBAC *Choosing Analyses* form, as shown below.

1. In the *Choosing Analyses* form, in the *Analysis* section, select *hbac*. Alternatively, in ADE Explorer, double click the hbac in the *Analyses* section of the *Setup* pane.
2. At the bottom of the form, in the *Specialized Analyses* section, select *RapidIP2.* The form changes, as shown in the following figure.

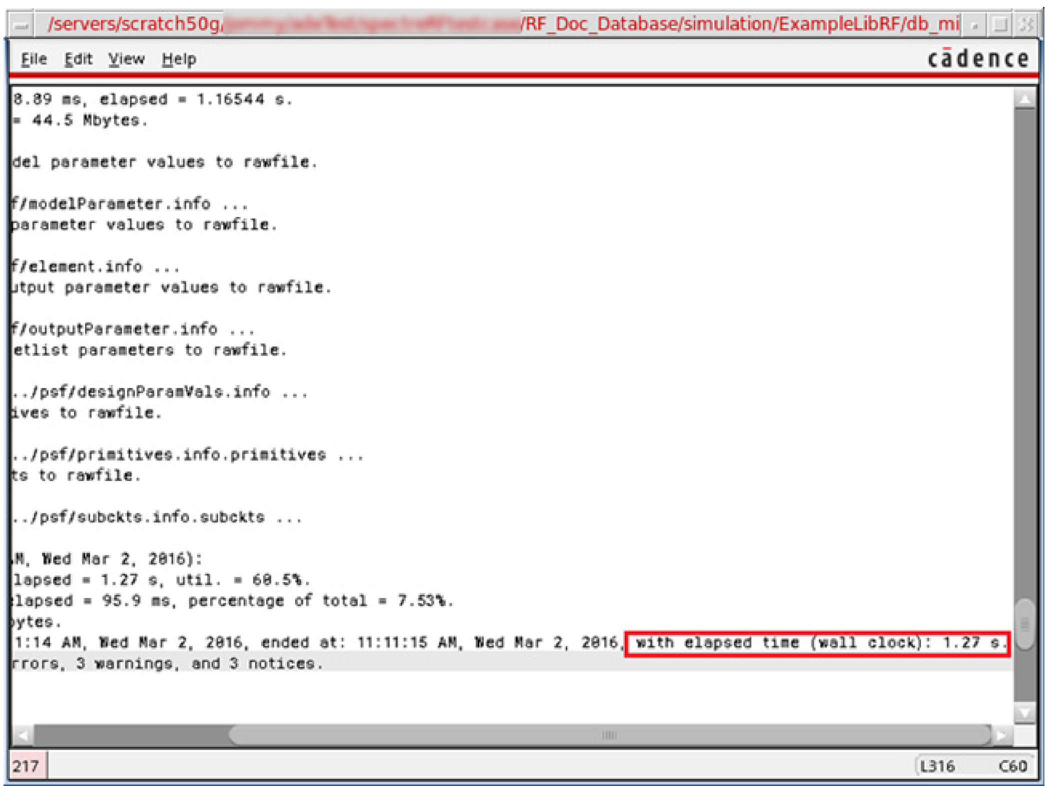
Figure 4-105 Rapid IP2 Choosing Analyses Form



The form retains all the settings that were set for the rapid IP3. The only thing that needs to be changed is the frequency of the intermodulation product. The main down conversion frequencies are at 4MHz and 5MHz. The second order intermodulation product is at 1MHz. The frequencies to be calculated depend on the choice of frequencies at the input and the LO frequency. Individual choices are provided for the frequencies which allows the selection of the smallest first order and largest second/third order product for your application.

3. In the *Frequency of IM Output Signal* field, type `1M`.
4. In the *Frequency of Linear Output signal* field, leave it set to `5M`.
5. Click *OK* at the bottom of the *Choosing Analyses* form.
6. Start the simulation. In ADE Explorer or the schematic, click the green arrow icon ( )on the right side of the window.
7. The Spectre output window isdisplayed with the simulator status information. Details of the IP2 measurement are also printed in the Spectre simulation log file. Note how much quicker the simulation finishes, compared to three-tone hb simulation used previously to calculate IP2.

Figure 4-106 Rapid IP2 spectre.out Logfile



8. After the simulation finishes, plot the output spectrum.

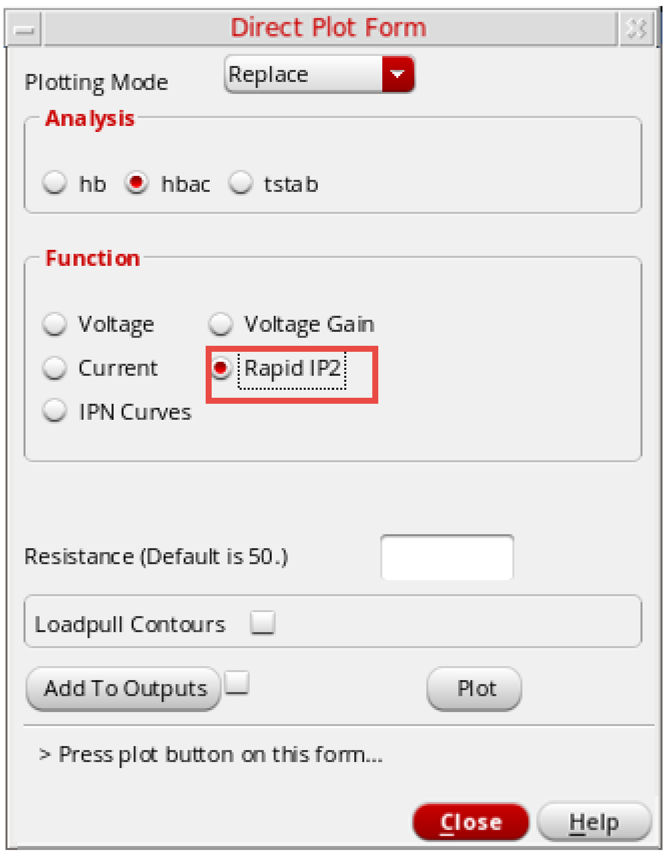
   In ADE Explorer, choose *Results - Direct Plot - Main Form*.

You may also invoke the Direct Plot Form by clicking the *Direct Plot* icon () in the schematic.

9. In the *Direct Plot Form*, select *hbac* in the *Analysis* section and *Rapid IP2* in the *Function* section. Leave the *Resistance* set to the default value *(50)*.

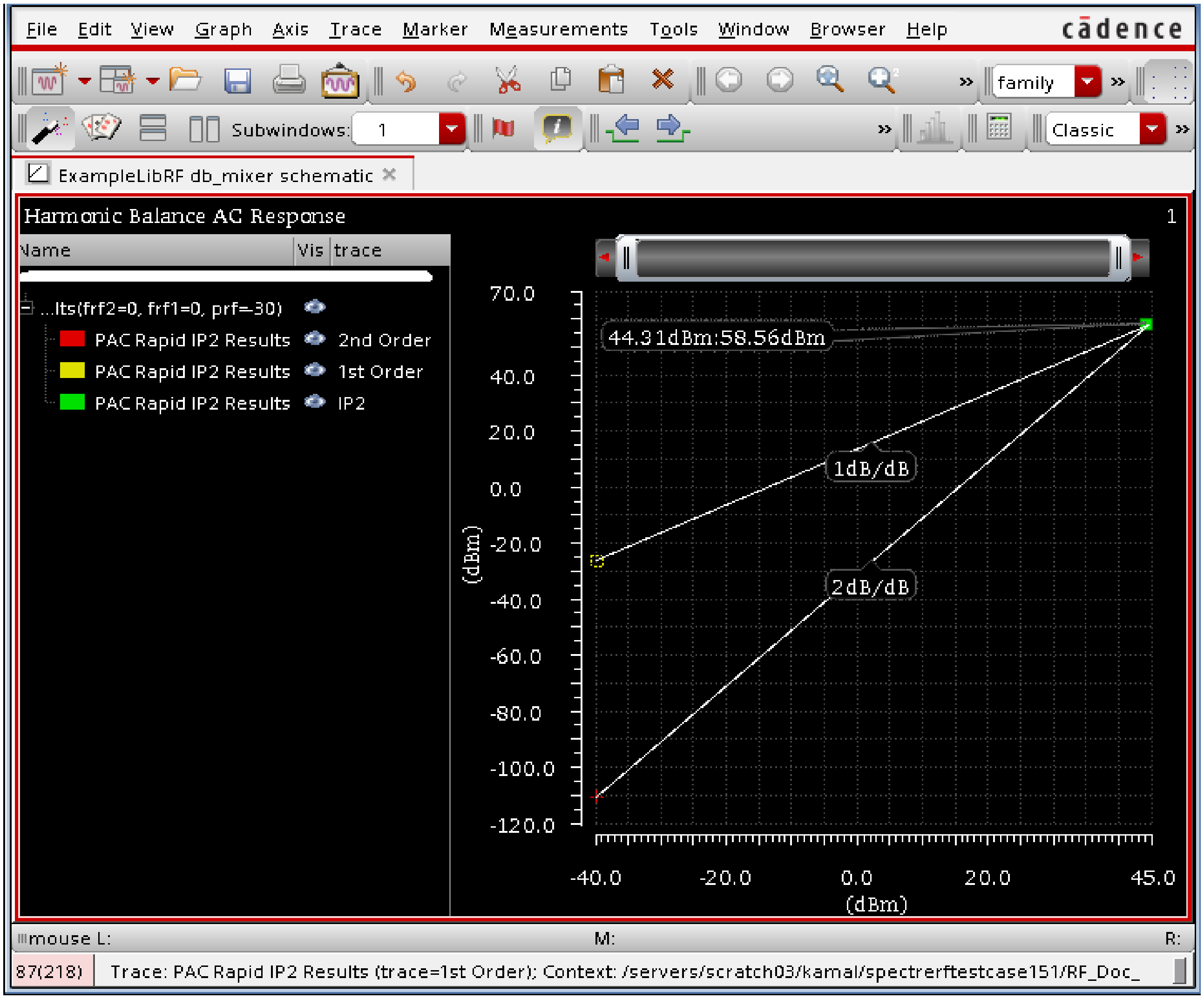
The *Direct Plot Form* is shown below.

Figure 4-107 Rapid IP2 Direct Plot Form



10. Click *Plot*. The Rapid IP2 is plotted in the waveform window.

Figure 4-108 Rapid IP2 Plot Results



The input and output referred IP2 are shown in the waveform tool at the intercept point. The X axis of the plot is input power and the Y axis of the plot is output power, thus the Xoutput-Y readout at the intercept point are the input referred IP2.

Note that the IP2 value from the three-tone hb simulation and rapid IP2 are within 1dB.

11. In the *Direct Plot Form*, click *Cancel*.

12. Close all waveform windows by choosing *File - Close all windows*.

13. In ADE Explorer, choose *Session - Quit.*

14. In the Schematic Window, choose *File - Close All.*

Summary

In this section, you have measured the third-order intercept (IP3) using three-tone hb analysis and Rapid IP3 (hb and hbac specialized analysis). Rapid IP3 is a much faster way to measure IP3 and is just as accurate. In addition, you measured IP2 using the Rapid IP2 methodology.

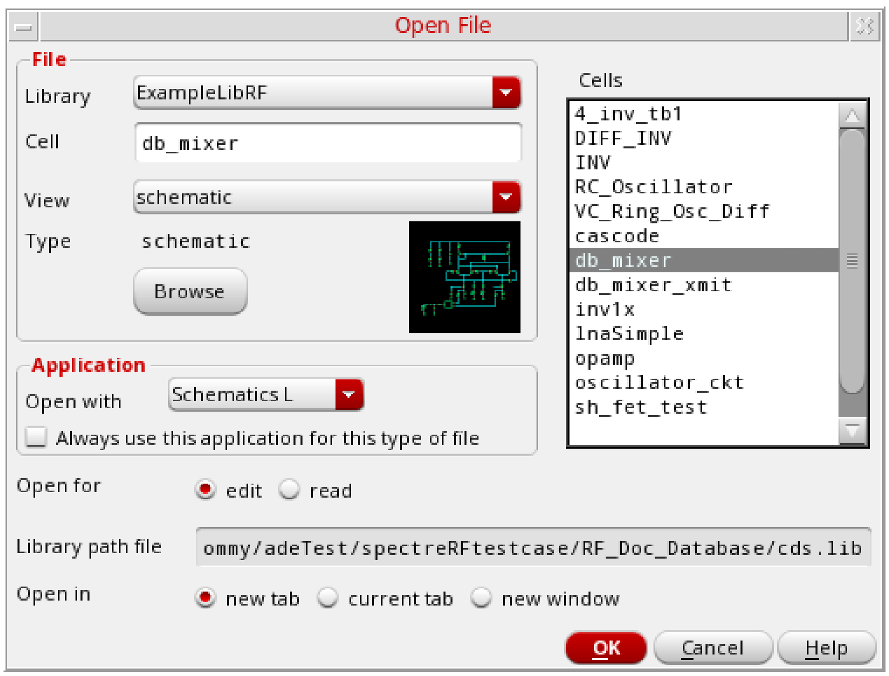
In the next section, you will make distortion measurements for the receive mixer.

Mixer Distortion Measurement

In the previous section, you measured IP3 and IP2. If you need to improve IP3 or IP2, it is helpful to know where the distortion is coming from, so you can fix the problem. Compression and IP3 are numerically related, so identifying components that cause compression also identify components that cause IP3 problems. In this next section, you will identify which components are causing compression in the circuit. For circuits that convert frequencies, hb and hbac compression distortion analyses are used. Similar capability for IP2 is provided in the IP2 distortion summary.

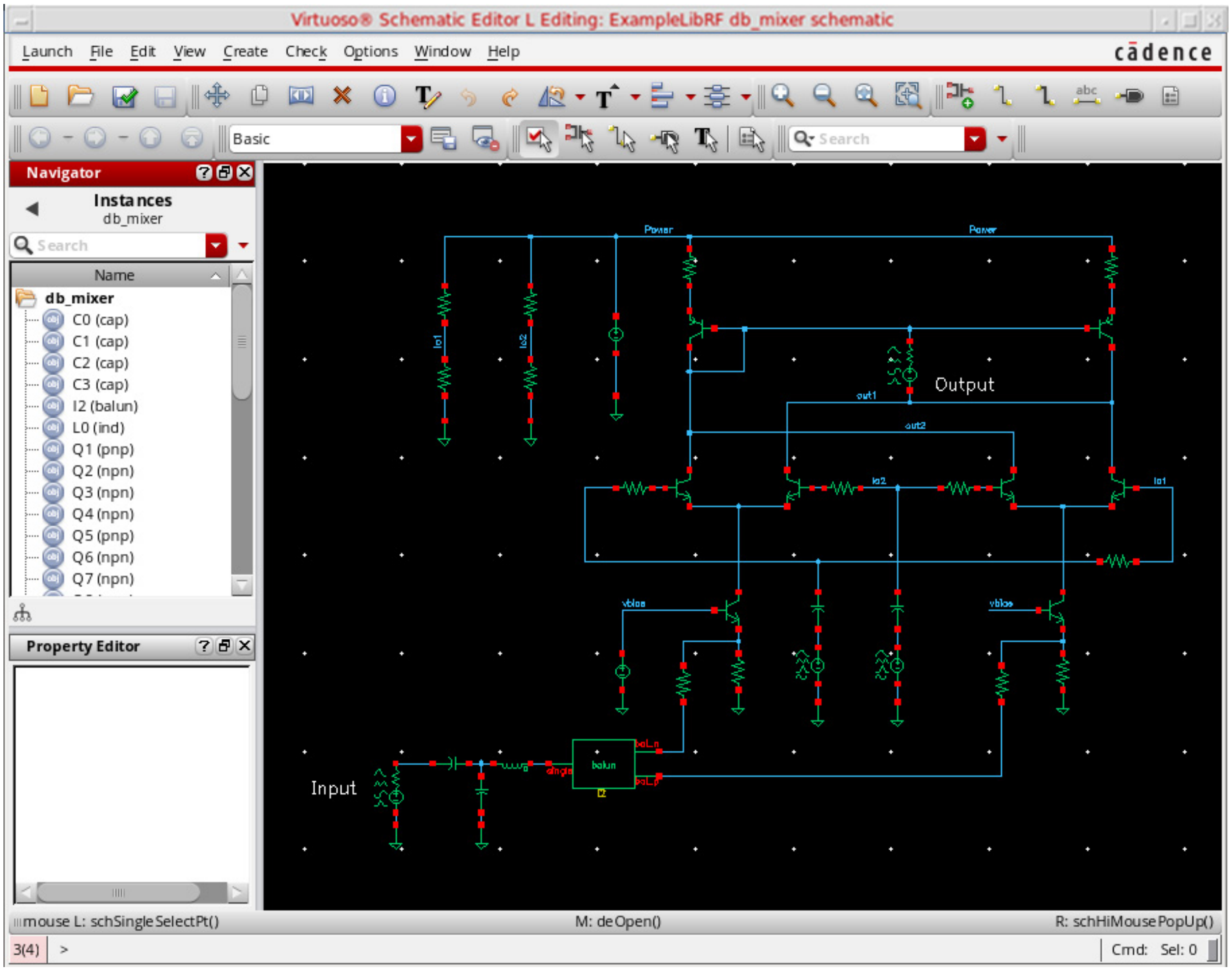
1. . In the CIW, choose *File - Open - CellView.* The *Open File* form is displayed. Choose the *db_mixer* schematic from *ExampleLibRF*.

Figure 4-109 Open File Form.



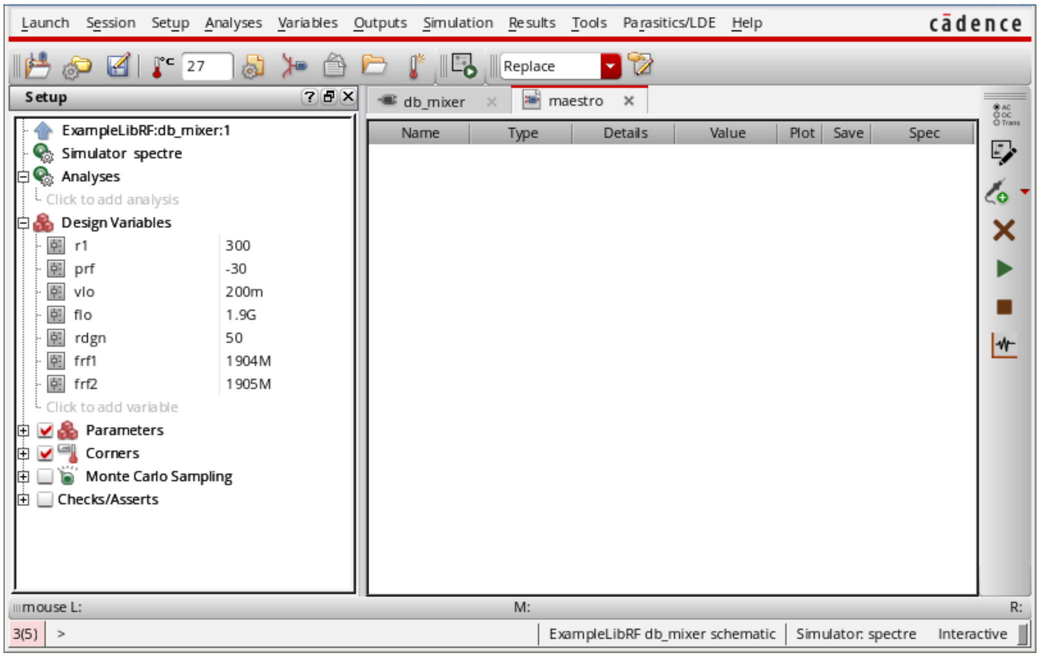
2. Click *OK*. The *db_mixer* schematic is displayed, as shown below.

Figure 4-110 db_mixer Schematic.



3. Open the Analog Design Environment from the schematic by selecting *Launch - ADE Explorer*. The simulation window is displayed, as shown below.

Figure 4-111 Analog Design Environment Simulation Window



4. In ADE Explorer, select *Setup – Simulator.*

   The *Choosing Simulator* form is displayed.

5. Select *spectre* from the *Simulator* drop-down list.

Figure 4-112 Choosing Simulator/Director/Host Form

6. Click *OK* to close the *Choosing Simulator* form.

7. Set up the High Performance Simulation Options, as follows:

   a. In ADE Explorer, choose *Setup - High Performance Simulation.* The *High Performance Simulation Options* window is displayed, as shown below.

Figure 4-113 High Performance Simulation Options Form

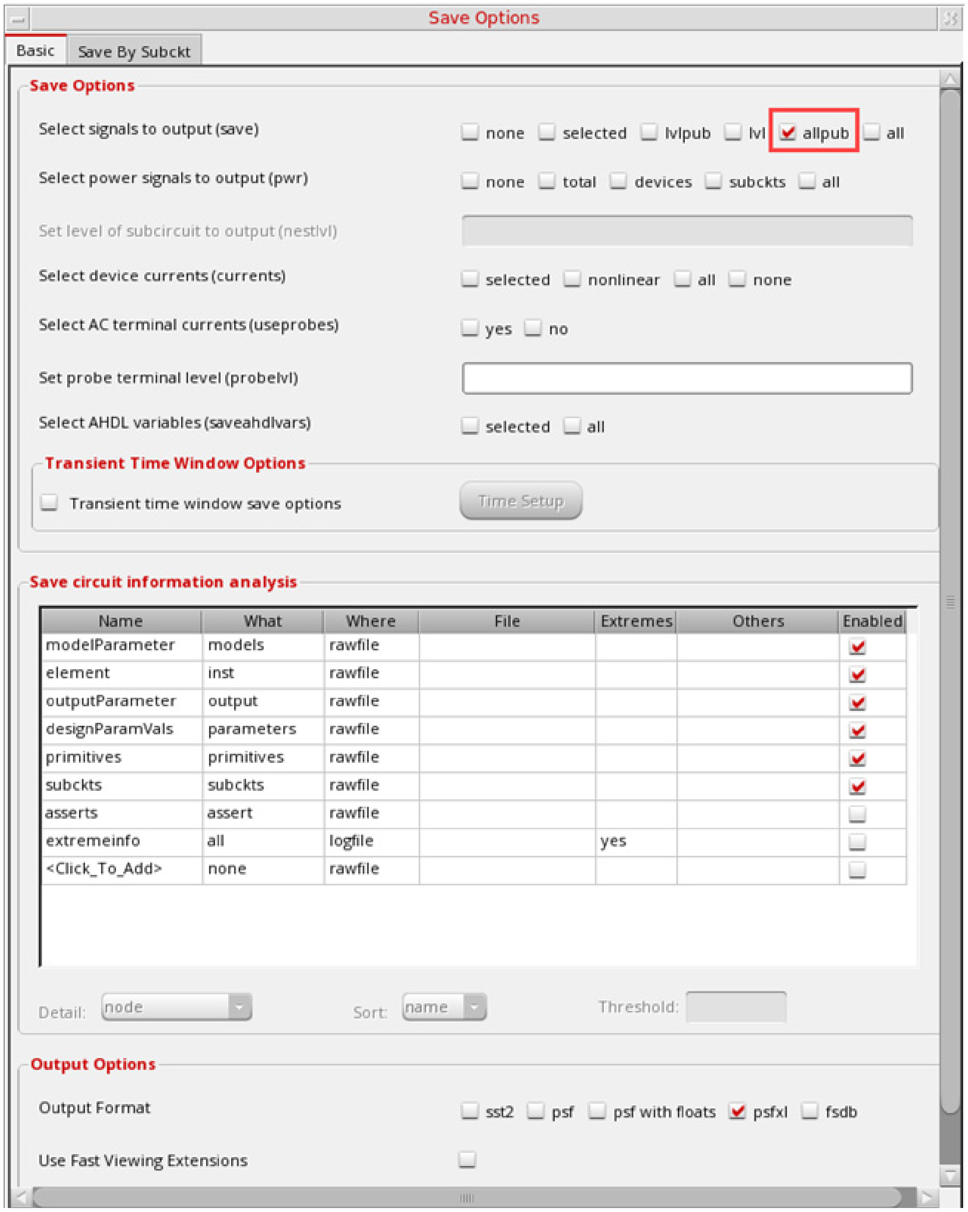
b. In the High Performance Simulation window, select *APS*. Note that *auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores. The bigger the circuit, the more threads you should use. For a small circuit such as this, you may want to set the number of threads to 2. Using 16 threads on a small circuit might actually slow things down because of the overhead associated with multithreading. For more information, refer to the *Spectre Circuit Simulator and Accelerated Parallel Simulator User Guide*.

c. Click *OK*.

8. Select *Outputs – Save All.*

The *Save Options* form is displayed, as shown below.

Figure 4-114 Save Options Form



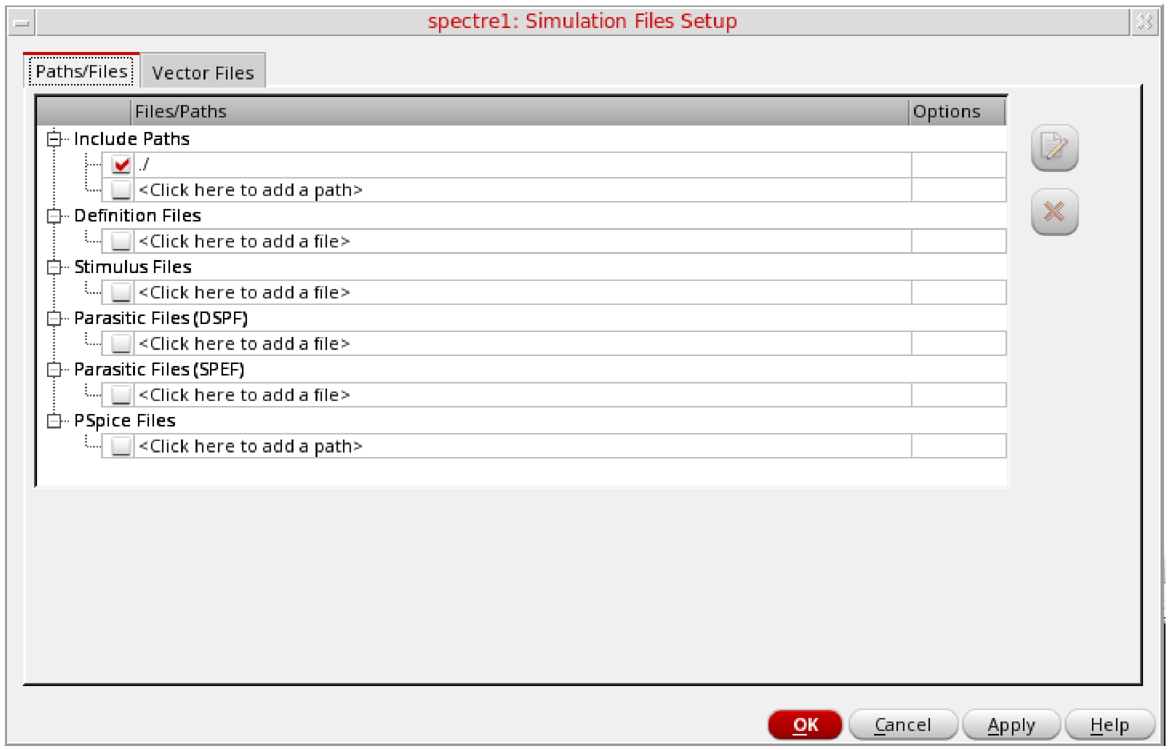
9. In the *Select signals to output* section, make sure that *allpub* is selected. This is the default which saves all node voltages at all levels of hierarchy, but it does not include the node voltages inside the device models.

10. Click *OK* to close the *Save Options* form.

Setting Up Model Libraries

1. In ADE Explorer, select *Setup - Simulation Files*. The *Simulation Files Setup* form is displayed, as shown below.

Figure 4-115 Simulation Files Setup Form



2. Ensure that the Include Path is set as shown above and click *OK* to close the form.
3. Select *Setup – Model Libraries.*

   The *Model Library Setup* form is displayed.

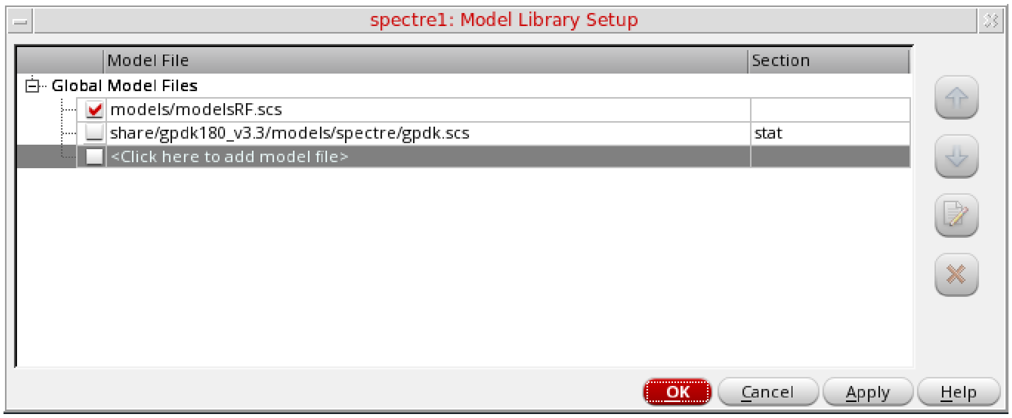
   In the *Model Library File* field, type the following path to the model file including the file name:

   `models/modelsRF.scs`

   Alternately, you can click *Browse* button and browse to the `modelsRF.scs` model file.
4. Ensure that the Model File name is selected.
5. Click *Apply*.

   The *Model Library Setup* form looks like the following:

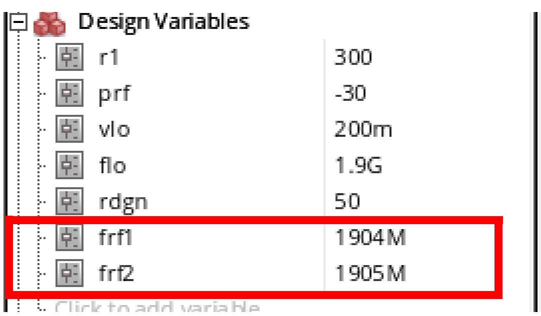
Figure 4-116 Model Library Setup Form



6. Click *OK* to close the Model Library Setup form.

Setting the Design Variables

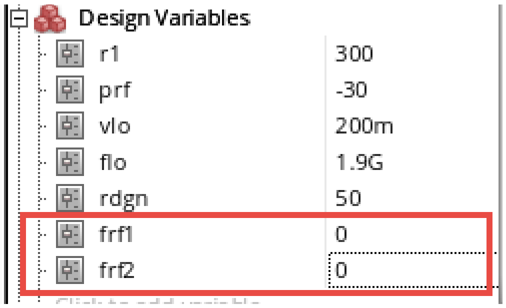
Figure 4-117 Design Variables Section



1. In the *Design Variables* section in ADE Explorer, click *frf1*.
2. Select the value in the box to the right of the variable *frf1*. Type *0* (zero) and press *Enter*. Do the same for the variable *frf2*. Setting the input frequencies to 0 disables the production of waveforms for the large-signal analyses like tran, pss, and hb (harmonic balance).

   The *Design Variables* section looks like the following:

Figure 4-118 Design Variables Section of ADE



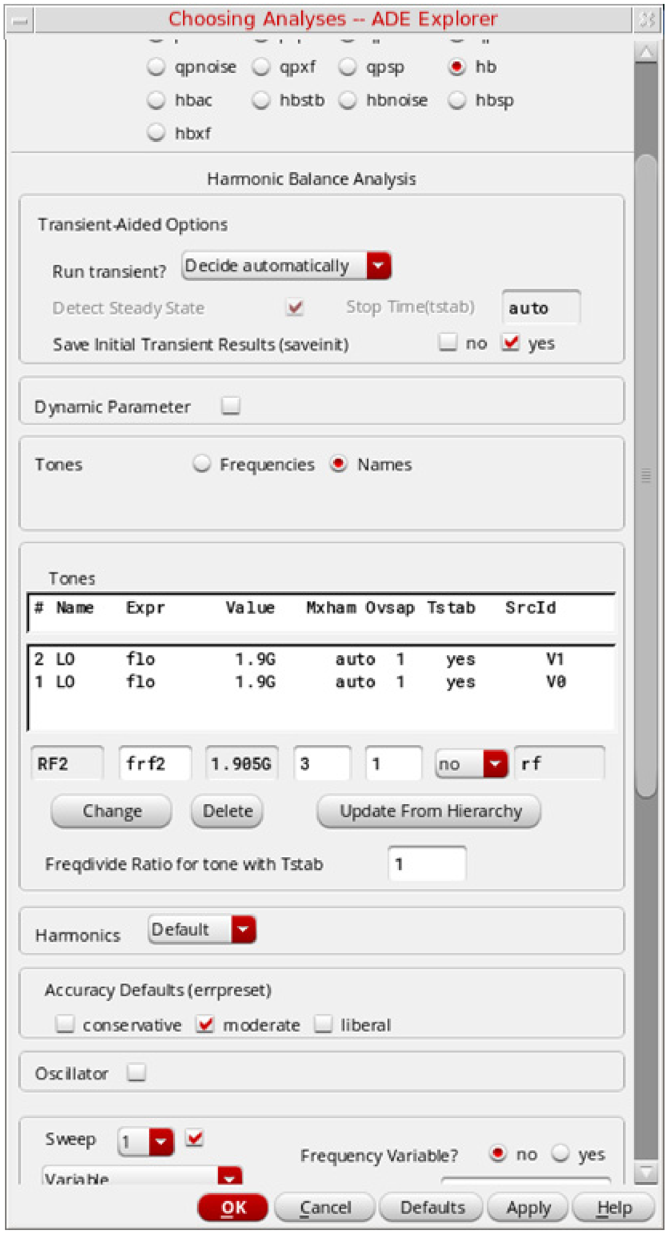
Setting up the HB analysis

1. Click the *Choosing Analyses* icon ( )on the right side of ADE Explorer.

   The Choosing Analyses form is displayed.

2. Select *hb* in the *Analysis* section. The form expands, as shown in the following figure:

Figure 4-119 hb Choosing Analyses Form



3. Fill in the form as follows:

   Harmonic balance can set harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. To enable this, select *Decide automatically* for the *Run Transient* selection in the *Transient-Aided Options* section. This single action will cause a transient analysis to be run until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1* (when *Frequencies* is selected) or for the tone that has *tstab* enabled (when *Names* is selected).

In the Transient-Aided Options section of the form, select the following:

a. For *Run transient?* select *Decide automatically*. (this is the default)

Run transient? will run the LO signal using the transient (In SpectreRF, this is called the *tstab* interval) for a short period of time. At the end of *tstab*, an FFT is performed, and this is used as the starting point in the harmonic balance analysis. Doing this improves the convergence of hb by giving it a better starting point at the cost of a short transient analysis.

b. For *Stop time (tstab), auto* is automatically populated in the field.

When *auto* is selected for *Stop time*, a small number of periods of the LO is run using the transient analysis. During this time, the signal is checked for steady-state conditions. If steady-state is not reached in the initial number of periods, more periods can be added automatically by the simulator. Using this feature allows an accurate FFT for the starting point in the hb iterations.

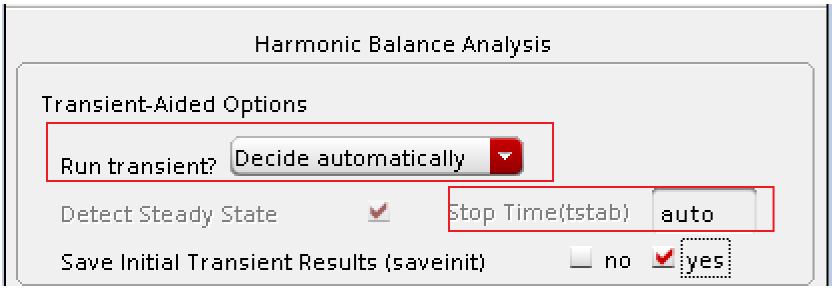
When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide automatically* simplifies the setup for harmonic balance, and produces correct answers from hb without needing to know how to set hb up manually.

c. For *Save Initial Transient Results (saveinit),* select *yes*.

During the transient-assisted HB simulation, a transient simulation runs before the frequency domain iteration of harmonic balance. The LO signal in *Tone1* is enabled for this measurement. At the end of the *tstab*, an FFT is run and its result is used as the starting point for the frequency domain iterations

All the signals are applied and the simulation is done in the frequency domain.Only the signal and its harmonics are calculated.

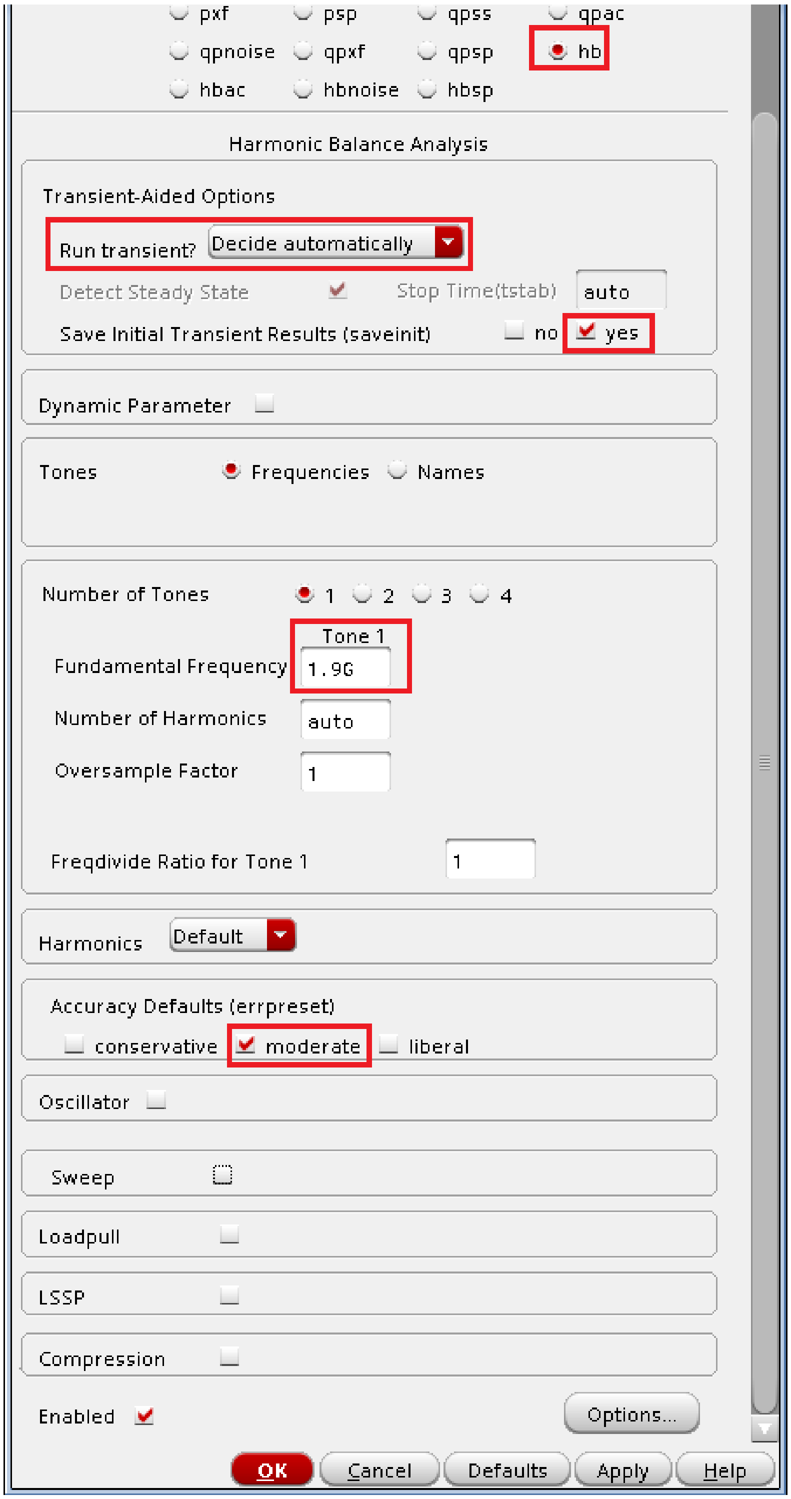
Figure 4-120 Transient Assisted Harmonic Balance



4. In the *Tones* section, choose *Frequencies* (this is the default).
5. *Number of Tones* is set to *1* by default. The only large signal tone in this simulation is the LO tone at 1.9GHz.
6. Set the *Fundamental Frequency* to `1.9G`.
7. Leave the *Number of Harmonics* set to the default value of `auto`.
8. Leave *Oversample Factor* set to the default value of `1`. When using the autoharmonics feature, you do not have to set *Oversample Factor.*
9. In the *Accuracy Defaults (errpreset)* section, select *moderate*. For most typical measurements *errpreset* should be set to *moderate*. When you need to measure really small distortions, then *conservative* would be used.

   The *Choosing Analyses* form should look like the following:

Figure 4-121 hb Choosing Analyses Form Set-up for Distortion Measurement

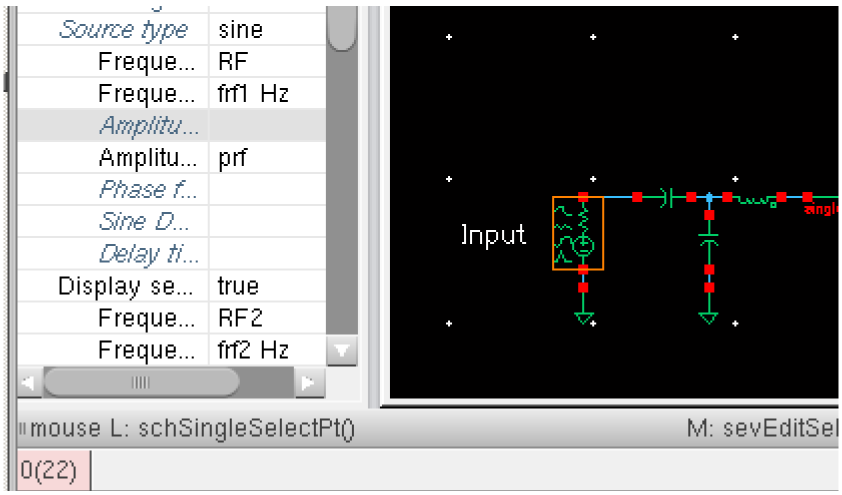


10. Click *Apply*.

11. In the Schematic window, set up the input port to perform a compression measurement.

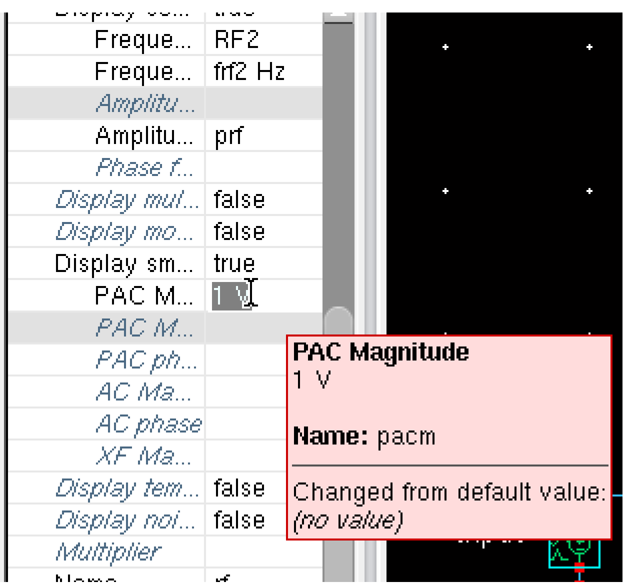
In the schematic, select the *Input* port. Once the input port is selected, the *Property Editor* on the left side of the schematic is populated with the properties of the selected instance, as shown below.

Figure 4-122 Setting Up Input Port for Compression Distortion Measurement



12. Search the *Property Editor* (scroll down) for the *PAC Magnitude* entry.

Figure 4-123 Editing Properties on Input Port for Compression Distortion Measurement

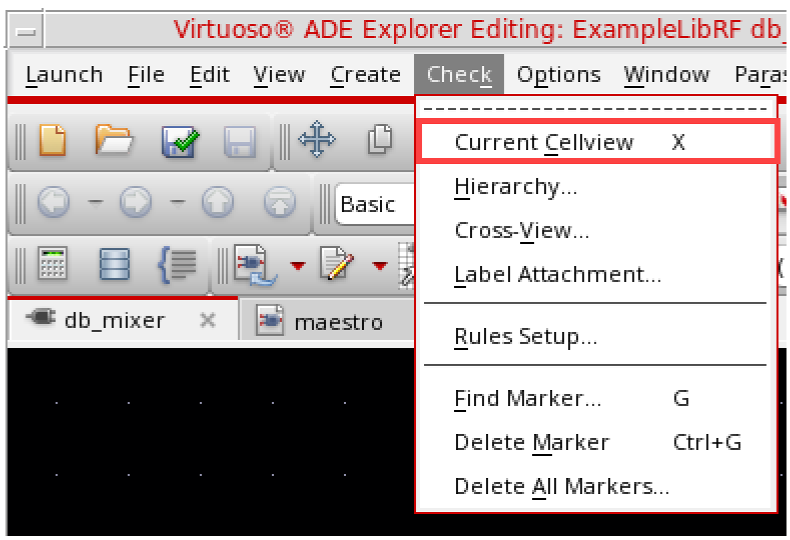


13. View the entry for *PAC Magnitude*. If set to *prf*, do not change anything. If set to *1* volt, select the entry, press the *Tab* key, and then set *PAC Magnitude (dBm)* to *prf*. The port needs to be set up to supply a signal in the small-signal region because the HBAC analysis which is a small-signal analysis will be used to calculate the distortion.

    The amplitude of the input signal is specified on the input port. For the distortion summary, the *PAC Magnitude (dBm)* property is typically used on the input port. This is the amplitude that is used as the input for the distortion measurement. The *prf* value is set in the variables section of the Analog Design Environment and is *-30* in this example.

14. Check the schematic without saving it. In the schematic window, choose *Check - Current Cellview.*

Figure 4-124 Check Current Cellview Menu

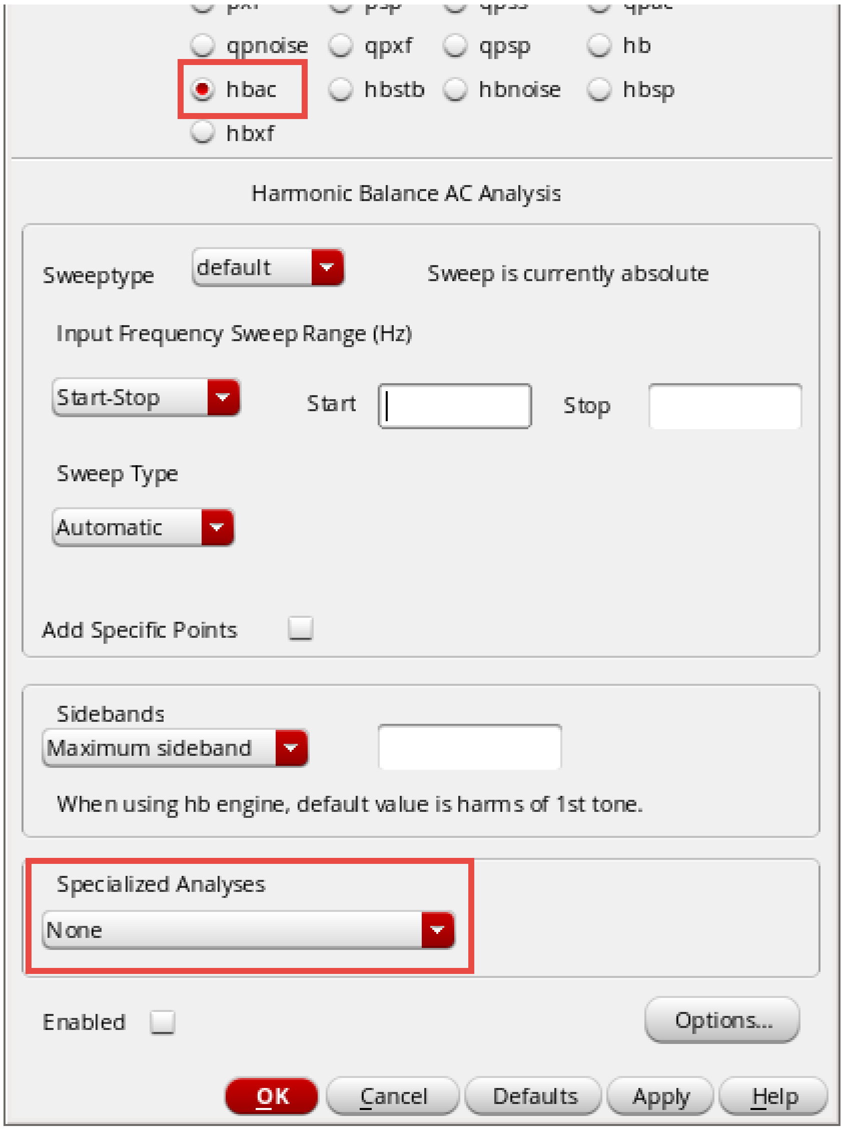


This allows ADE Explorer to netlist from the schematic without saving a copy of the newly modified circuit to the disk. This allows “what-if analysis” without changing the original schematic. When the design gets to the desired performance, then a *Check and Save* can be done. If you want to revert back to the original schematic, quit the schematic without saving it, and then recall it.

Setting up the HBAC form

1. In the *Choosing Analyses* form, select *hbac*. The form expands, as shown below.

Figure 4-125 hbac Choosing Analyses Form



2. At the bottom of the form in the *Specialized Analysis* section, select *Compression Distortion Summary*.

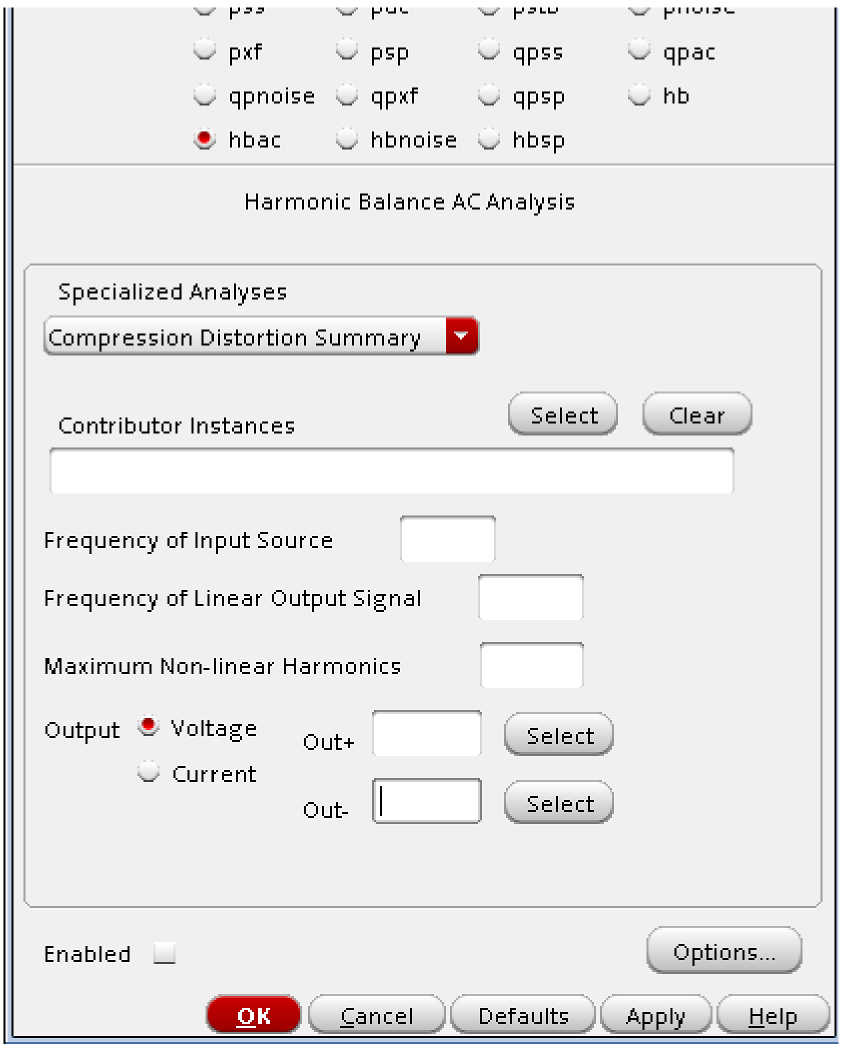
   A dialog box is displayed that reminds you to set the *pacmag* parameter on the RF source.

Figure 4-126 Compression Distortion Summary Dialog Box

Since you have already set this parameter in the preceding step, click *Close* to close the dialog box.

3. Set up the *hbac Choosing Analyses* form.

Figure 4-127 Compression Distortion Summary Choosing Analyses Form



4. You provide a list of the devices you want the distortion calculation for. If you leave this field blank, all of the nonlinear elements will be run. Since hbac will be run for each device specified, the computation can be time-consuming if the device list is large. You should select only those devices that are important to output. It is not recommended to run distortion summary for all nonlinear devices in the circuit if your circuit is large.

   In the *Contributor Instances* field, type `/Q1 /Q2 /Q3 /Q4 /Q5 /Q6 /Q7 /Q8`. Note the spaces between the entries. Alternately, you may click *Select* in the and select the components in the schematic.

5. Type `1.905G` in the *Frequency of Input Source* field. This is the RF input frequency.

6. Type `5M` in the *Frequency of Linear Output Signal* field. This is the mixer IF output frequency.

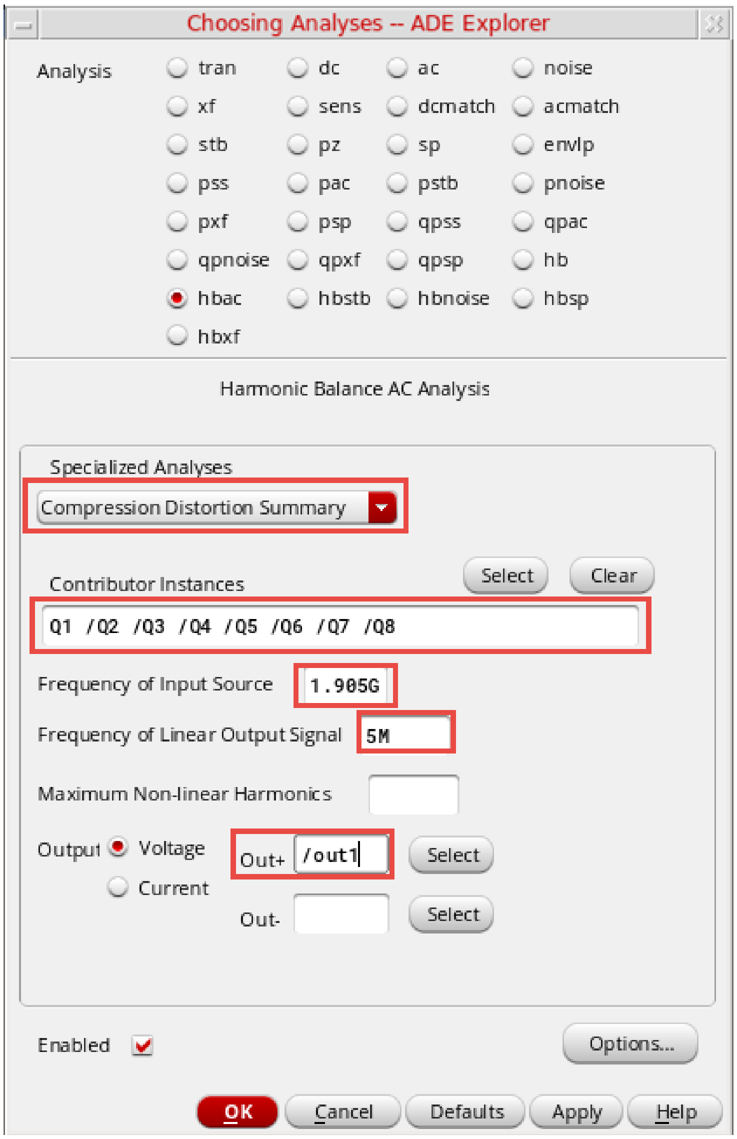
   Input and output frequencies are specified as usual. Only a single input frequency is allowed at a time.

7. Leave the *Maximum Non-linear harmonics* field blank. The default value is optimum for the compression distortion measurement.

8. In the *Out+* field, type `/out1`. You can also click *Select* to the right of the *Out+* field and choose a net in the schematic. When the *Out-* field is left blank, ADE Explorer automatically assigns this to the global ground node.

   The *Choosing Analyses* form should look like the following:

Figure 4-128 Choosing Analyses Compression Distortion Summary



9. Click *OK* at the bottom of the *Choosing Analyses* form.

The distortion measurement is performed along the entire signal path, which includes frequency translation and requires a declaration of the output node in the *Choosing Analyses* form.

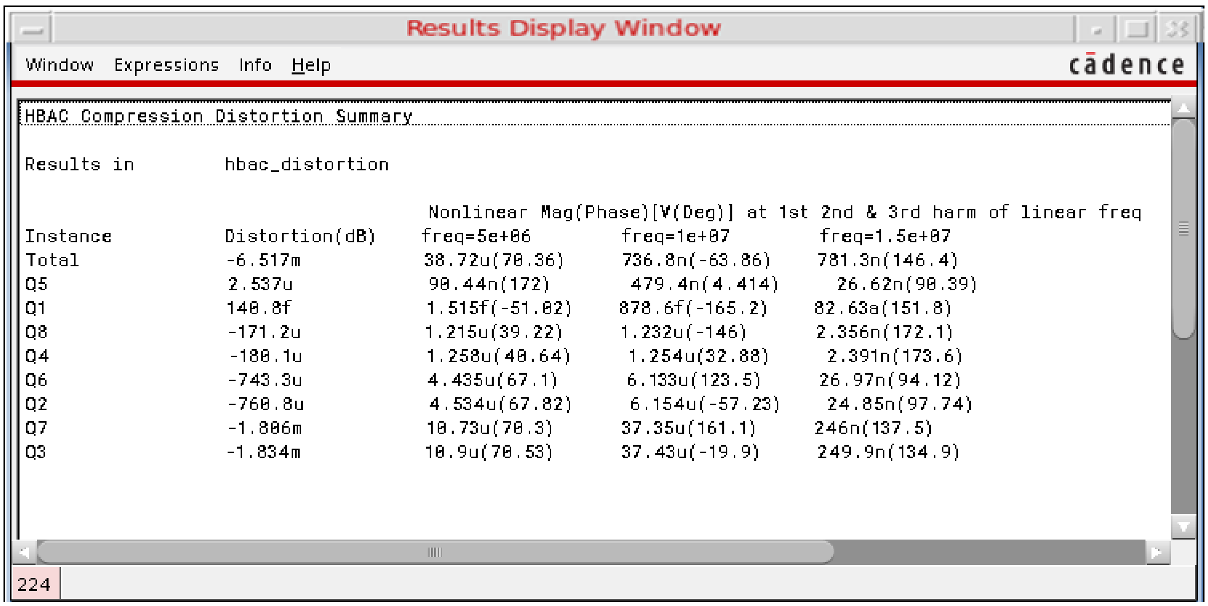
The compression distortion summary provides a relative measure of which components in a gain path contribute more or less compression. Because compression and IP3 are related mathematically, the compression distortion provides information about which components contribute to IP3.Start the simulation. In the ADE Explorer main window or the Schematic, click the green arrow icon ( ) on the right side of the window.

10. The Spectre output window will appear with the simulator status information.

11. After the simulation finishes, choose *Results - Print-HBAC Distortion Summary.*

The *Results Display Window* appears with the distortion contributions from the selected devices in the circuit. This summary gives the list of the various distortion contributors and how much distortion they contribute to the output. (If no devices are selected, all the nonlinear devices will be calculated). It helps identify the distortion sources. You can quickly see what your top distortion contributors are, and you can look up the distortion contribution for a particular device. The measurement in the output is the output level with the distortion included divided by the output level of the ideal system.

Figure 4-129 Compression Distortion Summary Results Display Window



HBAC Compression Distortion Summary

Results in hbac_distortion

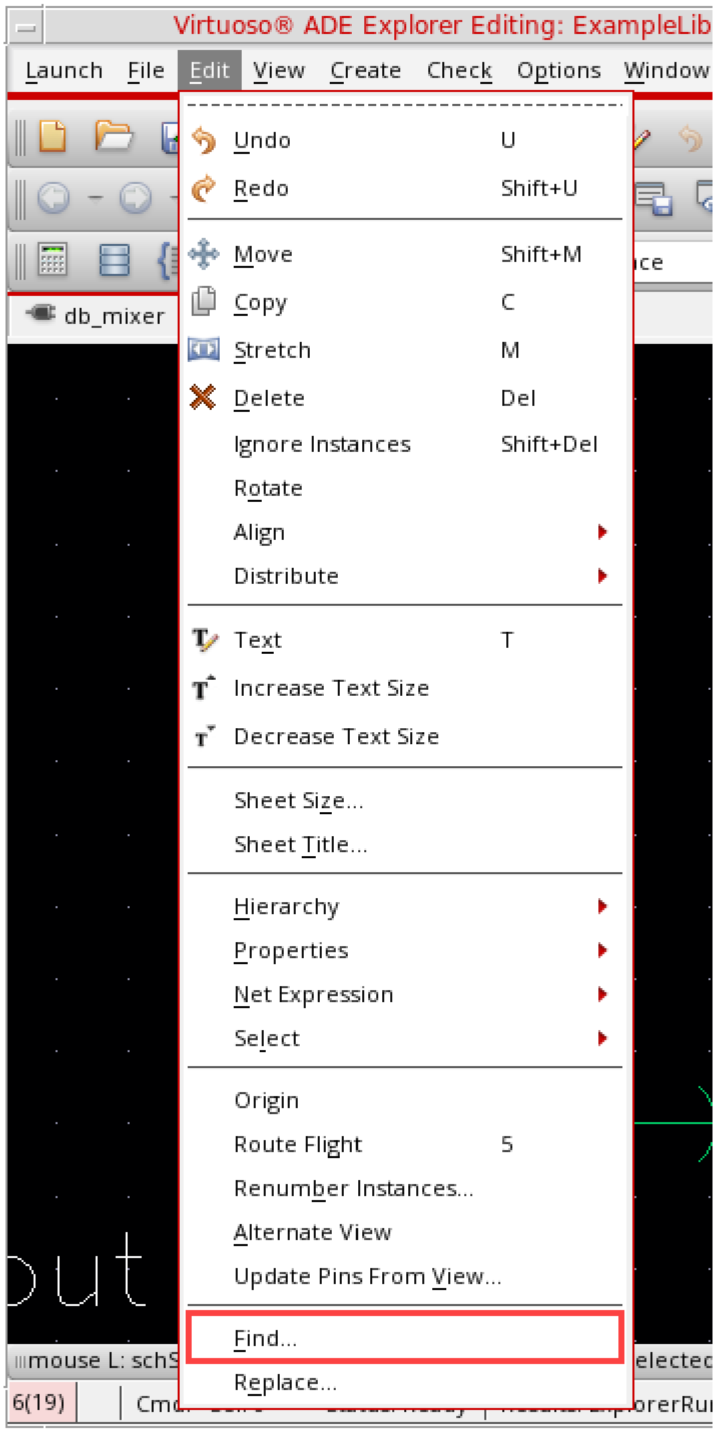
		Nonlinear Mag(Phase)[V(Deg)] at 1st 2nd & 3rd harm of linear freq		
Instance	Distortion(dB)	freq=5e+06	freq=1e+07	freq=1.5e+07
Total	-6.517m	38.72u(70.36)	736.8n(-63.86)	781.3n(146.4)
Q5	2.537u	90.44n(172)	479.4n(4.414)	26.62n(90.39)
Q1	140.8f	1.515f(-51.02)	878.6f(-165.2)	82.63a(151.8)
Q8	-171.2u	1.215u(39.22)	1.232u(-146)	2.356n(172.1)
Q4	-180.1u	1.258u(40.64)	1.254u(32.88)	2.391n(173.6)
Q6	-743.3u	4.435u(67.1)	6.133u(123.5)	26.97n(94.12)
Q2	-760.8u	4.534u(67.82)	6.154u(-57.23)	24.85n(97.74)
Q7	-1.806m	10.73u(70.3)	37.35u(161.1)	246n(137.5)
Q3	-1.834m	10.9u(70.53)	37.43u(-19.9)	249.9n(134.9)

12. Note that *Q3*, *Q7*, Q2, and Q6 have the largest distortion contributors.

13. Find Q3 in the schematic.

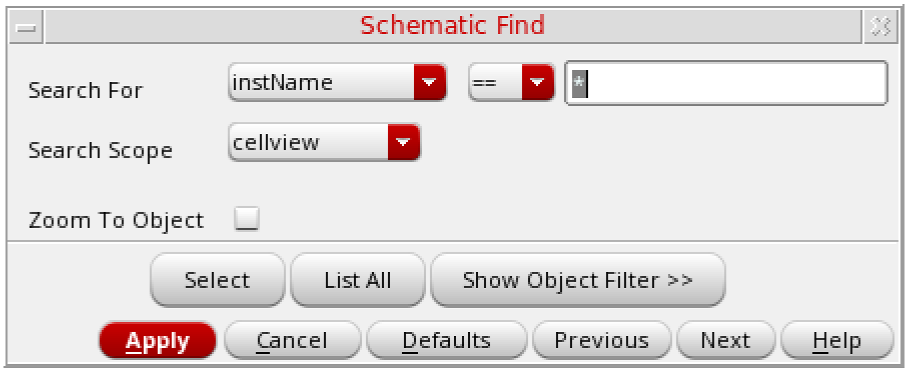
In the schematic window, choose *Edit - Find.*

Figure 4-130 Edit-Find to bring up Schematic Find Form



The *Schematic Find* form is displayed, as shown below.

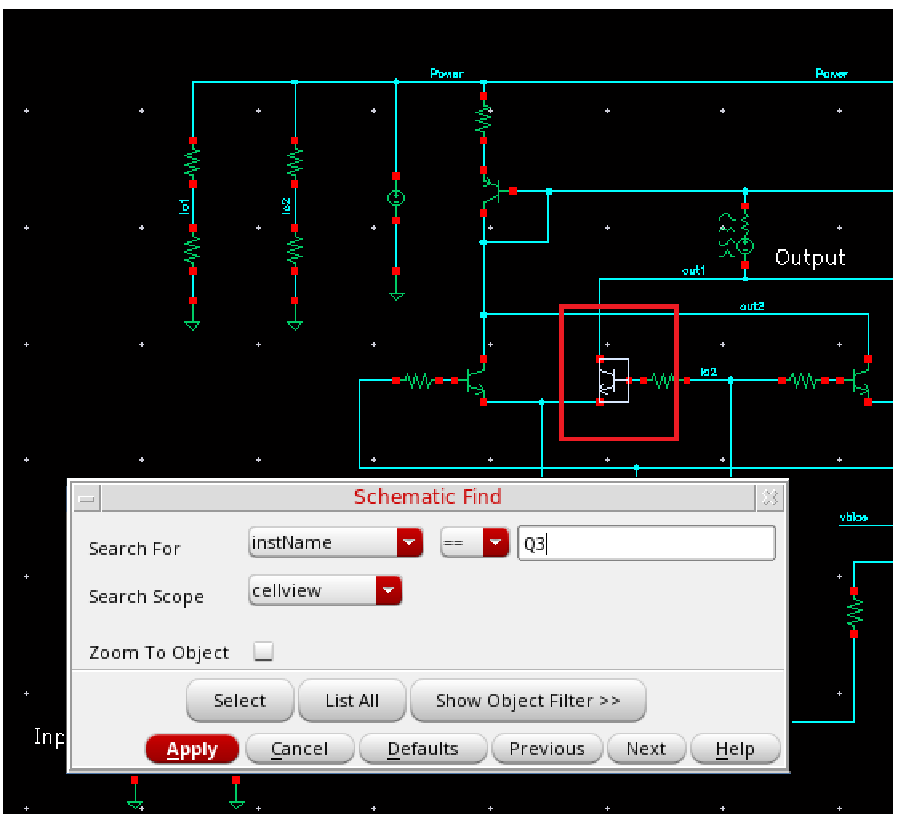
Figure 4-131 Schematic Find Form



14. Delete the asterisk, type `Q3`, and click *Apply*.

Q3 is highlighted in the Schematic window, as shown below.

Figure 4-132 Finding Sources of Distortion, in the Schematic Design.

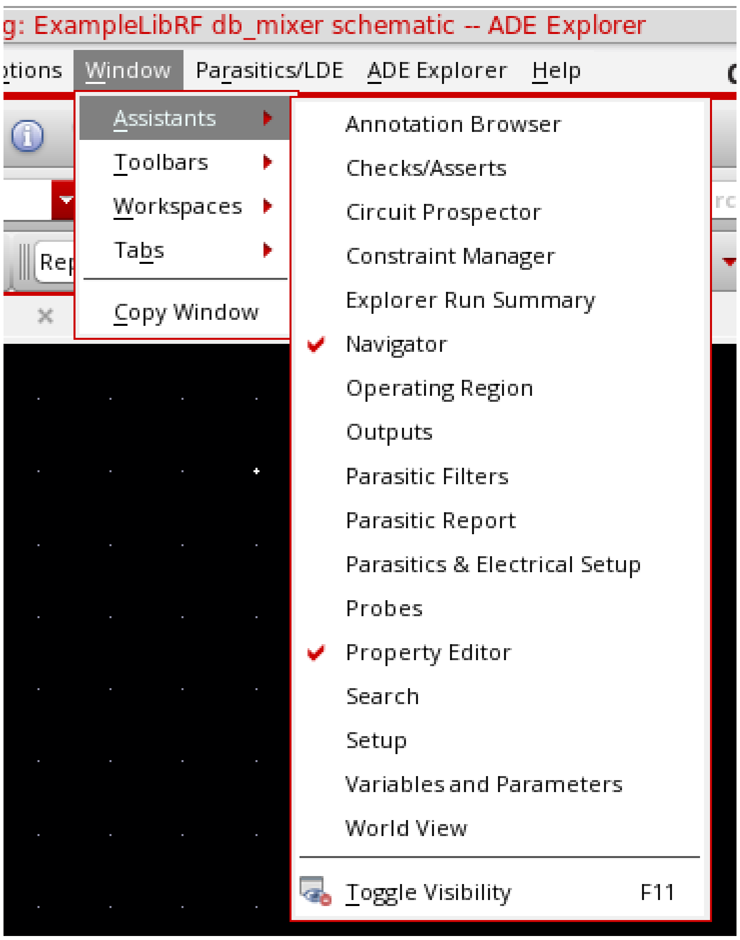


15. In a similar manner, find *Q2*, *Q6*, and *Q7*.

In the *Schematic Find* window, click *Cancel*.

16. Click in the graphics area away from any component so the input port is deselected.

17. If you deleted the Navigator in an earlier step, recall the Window Navigator by choosing *Window - Assistants - Navigator*.



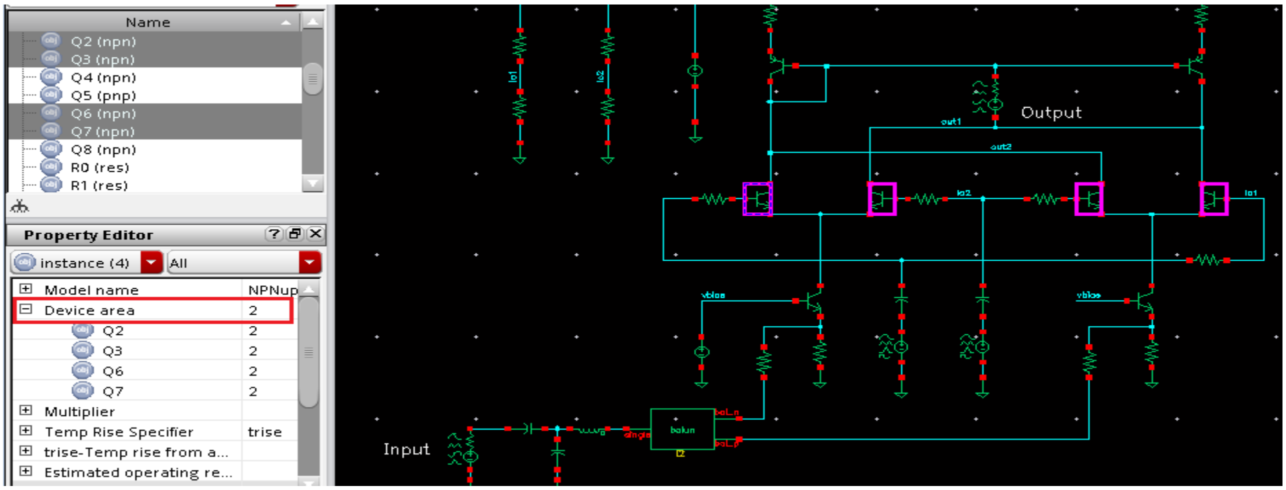
18. While holding the *Shift* key, select the four instances you found in the search. You should see *Q3*, *Q2*, *Q6*, and *Q7* highlighted in the *Navigator Assistant* and in the schematic.

19. In the *Property Editor* assistant, type `2` in the *Device area* field.

20. Press *Enter.*

The *Navigator Assistant* and the *Property Editor* on the right side of the schematic should look like the following:

Figure 4-133 Navigator Assistant Showing Four Selected Transistors



21. In the schematic window, choose *Check - Current Cellview.* (This is to do a “what-if analysis” and not overwrite the Schematic)

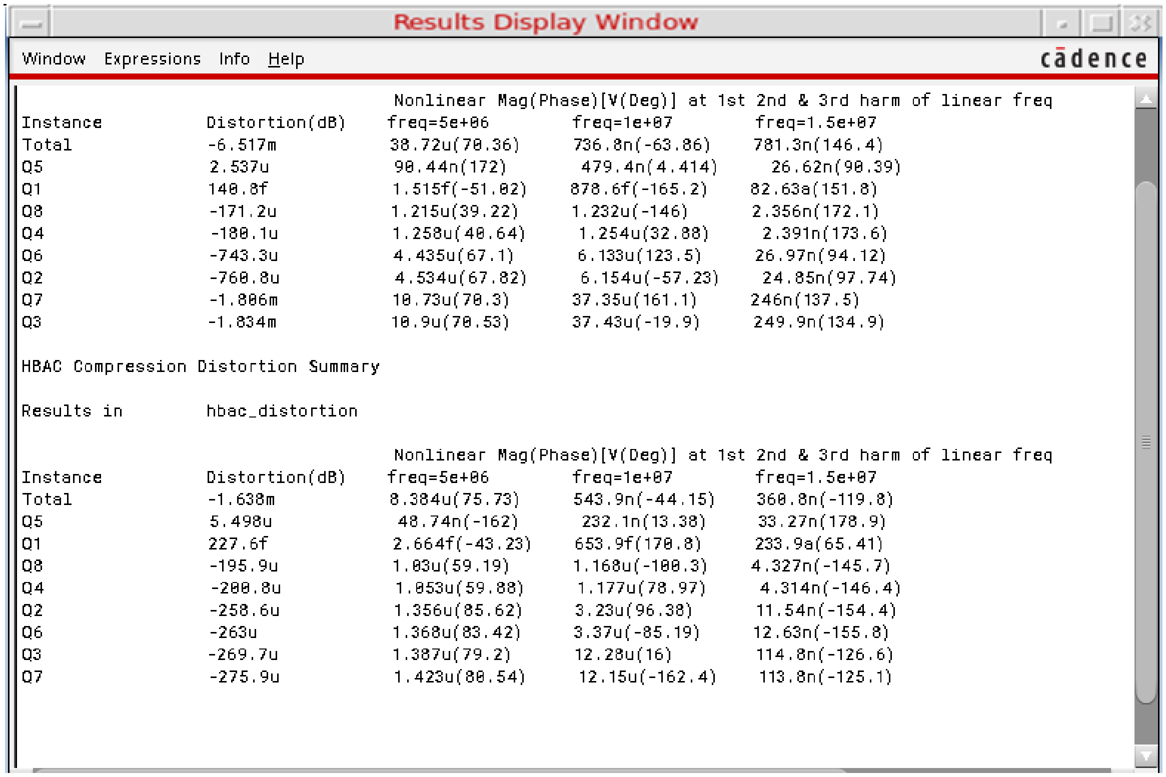
22. Start the simulation. From the ADE Explorer main window or the Schematic, click the green arrow icon () on the right side of the window.

23. The Spectre output window will appear with the simulator status information.

24. After the simulation finishes, choose *Results- Print- HBAC Distortion Summary.*

25. The Results are appended in the *Results Display Window* at the bottom of the list.

Figure 4-134 Results Display Window -Two HBAC Compression Distortion Summary Simulations



Results Display Window

Window Expressions Info Help

Nonlinear Mag(Phase)[V(Deg)] at 1st 2nd & 3rd harm of linear freq

Instance	Distortion(dB)	freq=5e+06	freq=1e+07	freq=1.5e+07
Total	-6.517m	30.72u(70.36)	736.8n(-63.86)	781.3n(146.4)
Q5	2.537u	90.44n(172)	479.4n(4.414)	26.62n(90.39)
Q1	140.8f	1.515f(-51.02)	878.6f(-165.2)	82.63a(151.8)
Q8	-171.2u	1.215u(39.22)	1.232u(-146)	2.356n(172.1)
Q4	-180.1u	1.258u(40.64)	1.254u(32.88)	2.391n(173.6)
Q6	-743.3u	4.435u(67.1)	6.133u(123.5)	26.97n(94.12)
Q2	-760.8u	4.534u(67.82)	6.154u(-57.23)	24.85n(97.74)
Q7	-1.806m	10.73u(70.3)	37.35u(161.1)	246n(137.5)
Q3	-1.834m	10.9u(70.53)	37.43u(-19.9)	249.9n(134.9)

HBAC Compression Distortion Summary

Results in hbac_distortion

Nonlinear Mag(Phase)[V(Deg)] at 1st 2nd & 3rd harm of linear freq

Instance	Distortion(dB)	freq=5e+06	freq=1e+07	freq=1.5e+07
Total	-1.638m	8.384u(75.73)	543.9n(-44.15)	360.8n(-119.8)
Q5	5.498u	40.74n(-162)	232.1n(13.38)	33.27n(178.9)
Q1	227.6f	2.664f(-43.23)	653.9f(170.8)	233.9a(65.41)
Q8	-195.9u	1.03u(59.19)	1.168u(-100.3)	4.327n(-145.7)
Q4	-200.8u	1.053u(59.88)	1.177u(78.97)	4.314n(-146.4)
Q2	-250.6u	1.356u(85.62)	3.23u(96.38)	11.54n(-154.4)
Q6	-263u	1.368u(83.42)	3.37u(-85.19)	12.63n(-155.8)
Q3	-269.7u	1.387u(79.2)	12.28u(16)	114.8n(-126.6)
Q7	-275.9u	1.423u(80.54)	12.15u(-162.4)	113.8n(-125.1)

Note that *Q3*, *Q7*, *Q6*, and *Q2* are still the largest distortion contributors. The distortion is lower with the Device area = `2`, as is the Total Distortion. This demonstrates the ability to do a “what-if” analysis changing the size of the NPN differential pairs.

Close the Analog Design Environment and the Schematic Window

1. In ADE Explorer, choose *Session - Quit.*
2. In the schematic window, choose *File - Close All.*

Summary

You have used hb and hbac analyses to perform Mixer Distortion Measurements. You used Compression Distortion Summary to locate the distortion contributions from various devices in the design.

This concludes the section on Receive Mixers. For more information on simulating receive mixers, please refer to the chapters in this user guide. In addition, see *Spectre Circuit Simulator RF Analysis Theory Guide*.

The remaining sections focus on simulating Transmit Mixers.

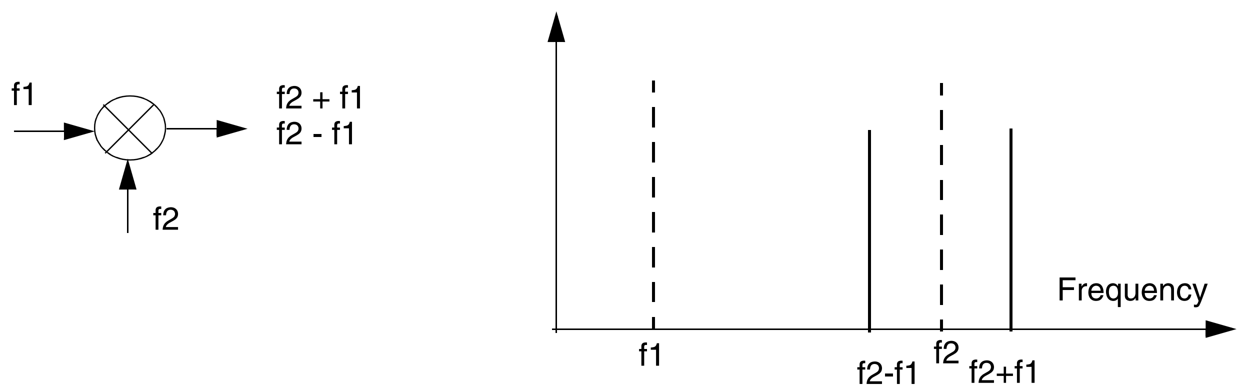
Setting Up to Simulate the db_mixer_xmit Mixer

In this workshop, you will make common measurements for an up-converting mixer.

The `db_mixer_xmit` circuit is found in the *ExampleLibRF* library. The `db_mixer_xmit` integrated circuit consists of two Gilbert cell (up-converting double-balanced) mixers. Looking at the very left side of the schematic are two ports labeled *Input_top* and *Input_bottom* which generate the input signals. Both inputs feed the input to two double-balanced mixers through two voltage controlled voltage sources. There are four LO sources in the circuit between the bottom devices of the respective mixers. The LO operates at 1.9GHz. Next to the label *Output*, is the output port of the mixer.

Image reject mixers use phase canceling techniques and remove one of the two major mixer products from the output of the mixing or multiplication process.

When two RF signals are mixed (*f1* and *f2*), the sum and difference signals are produced, as shown below.



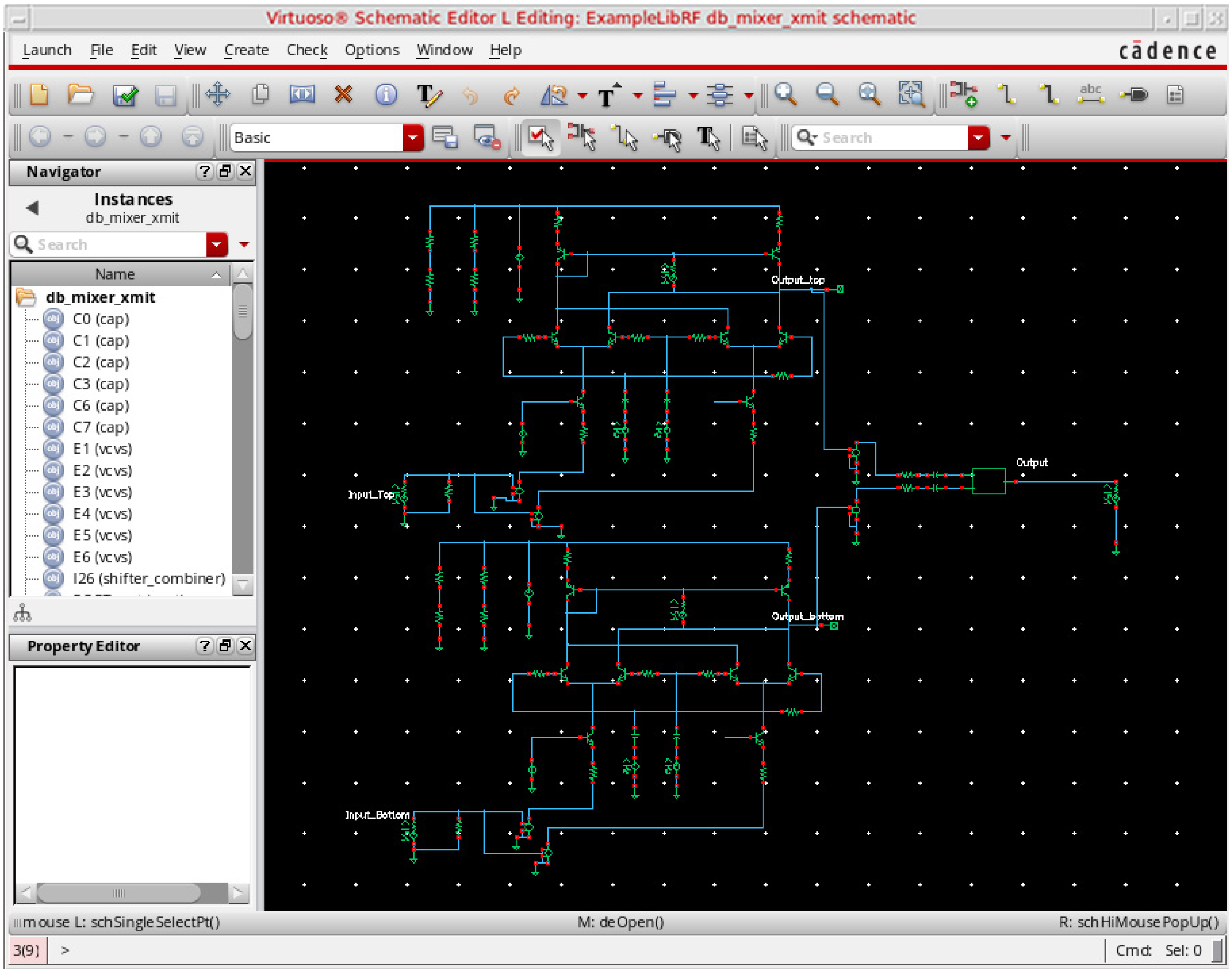
The output signals are (`f1 + f2`) and (`f1-f2`).

Of the two products from a mixer, normally only one is required. Often, the unwanted one will fall outside the required bandwidths and can be removed very easily. If the unwanted or image product is close to the desired signal, it can require complicated filtering to remove it sufficiently.

Image rejection mixers utilize phasing techniques to cancel out the unwanted mix products.

In the figure below, is the schematic of the upconverting mixer `db_mixer_xmit`. Following that is the list of measurements you will make in this module.

Figure 4-135 db_mixer_xmit Schematic



Transmit Mixer Measurements (db_mixer_xmit)	**Analyses**
Three Tone Spectral Content and Image Rejection	HB
Three tone Swept IP3 (large signal)	HB
Noise Figure/Signal to Noise Ratio	HB and HBnoise

Measurement	**Measured**
LO frequency (Hz)	1.9 GHz
Desired RF frequencies(Hz)	1.904 GHz, 1.905 GHz
Image frequencies	1.895 GHz, 1.896 GHz
IF frequencies (Hz)	4 MHz, 5 MHz
LO voltage	200mV peak
IF power	-9 dBm
Image rejection	*measurement needed*
Input IP3 (from swept power)	*measurement needed*
Noise figure/SNR	*measurement needed*

Three Tone Spectral Content and Image Rejection

Transmit mixers usually operate at power levels high enough that small-signal analyses should not be used. Therefore, HB is used to capture the actual, large-signal behavior. When measuring three-tone spectral content, all signals (LO, IF1, and IF2) are applied to the circuit. Harmonic balance is used, rather than transient analysis, because it is quicker.

A conventional transmit mixer has two output responses at points above and below the LO frequency at `f1+f2` and `|f1-f2|`. The unused response, known as the image frequency, can be suppressed by an image-reject mixer. For the first measurement, you will check the output spectrum and view the frequency response to determine the image rejection. You will also look at IP3 measurements, which is a common figure of merit for mixers. For this circuit, the desired response is at 1.904G and 1.905G.

Setting Up the Simulation

Open the *db_mixer_xmit* Mixer Circuit in the Schematic Window, as follows:

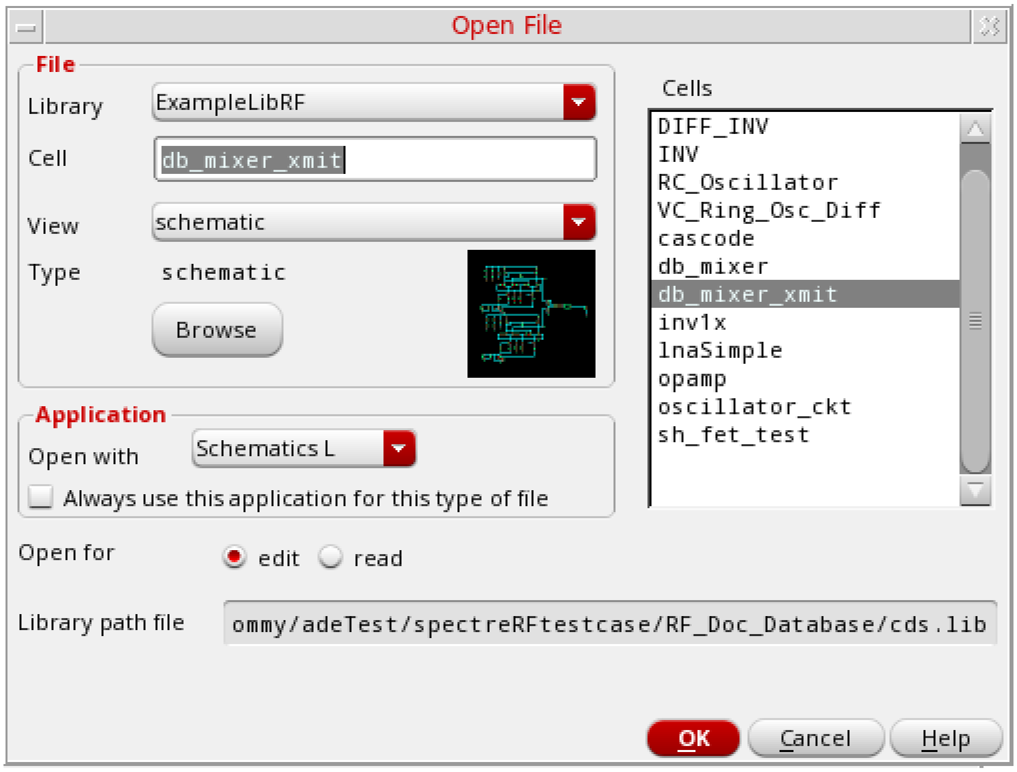
1. In the CIW, choose *File – Open.*

   The Open File form is displayed.

2. In the *Open File* form, select *ExampleLibRF* from the *Library* drop-down list.
3. Select *db_mixer_xmit* from the *Cells* list box.

   The completed *Open File* form looks like the following:

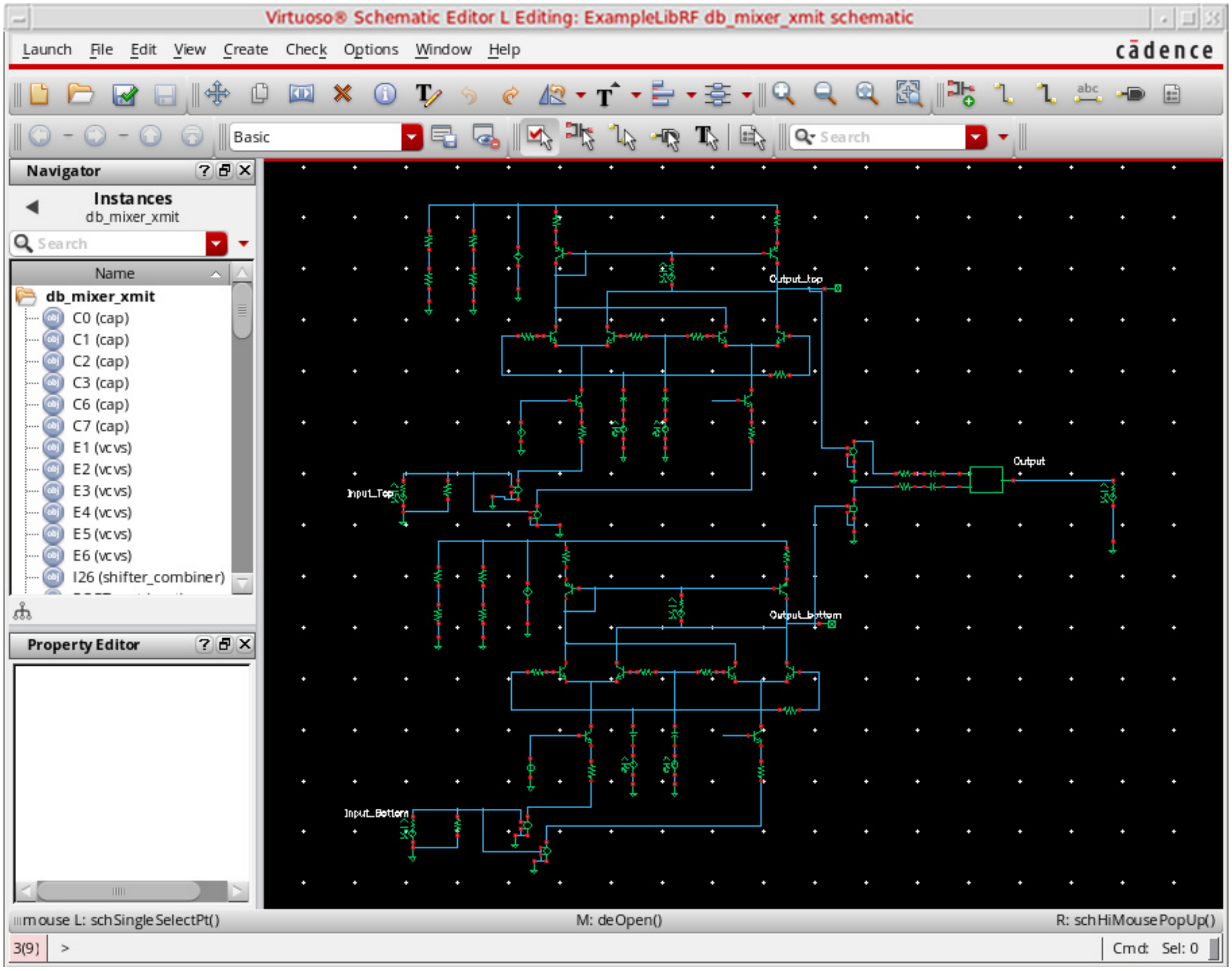
Figure 4-136 Open File Form



4. Click *OK*.

The Schematic window for the `db_mixer_xmit` mixer is displayed, as shown below.

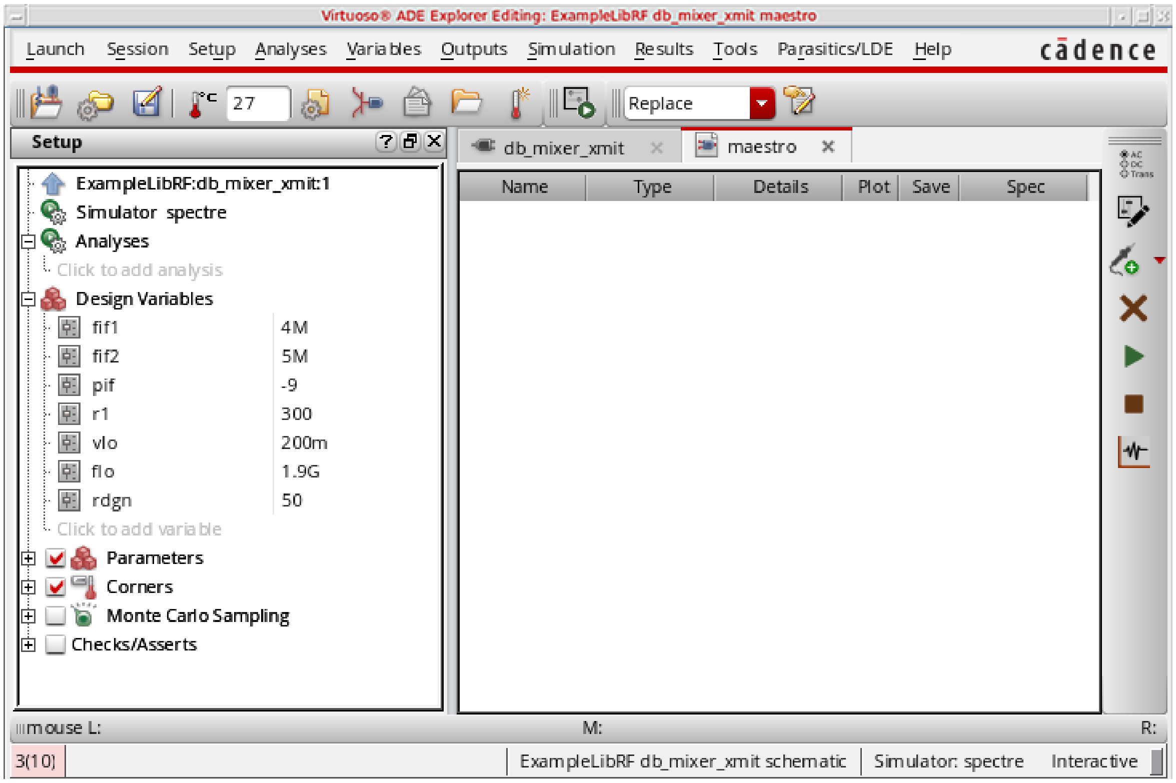
Figure 4-137 db_mixer_xmit Schematic



5. In the Schematic window, choose *Launch – ADE Explorer.*

   The Virtuoso Analog Design Environment Explorer window is displayed, as shown below.

Figure 4-138 ADE Explorer



Choose the simulator options, as follows:

6. In ADE Explorer, select *Setup – Simulator*.

   The *Choosing Simulator/Directory/Host* form is displayed.

7. Select *spectre* from the *Simulator* drop-down list.

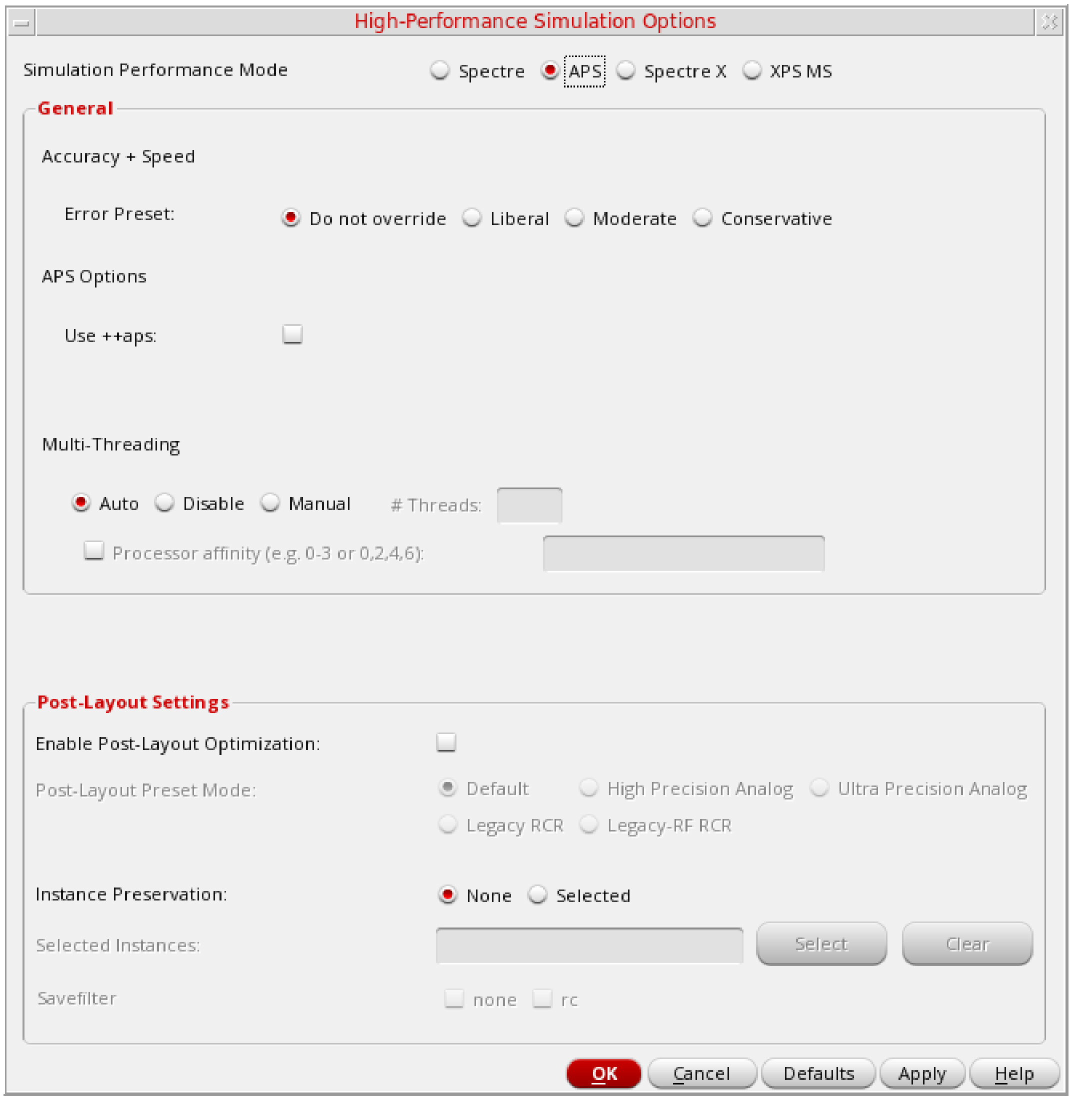
Figure 4-139 Choosing Simulator/Directory/Host Form

8. Click *OK* to close the *Choosing Simulator*.

9. Set up the High Performance Simulation Options, as follows:

   a. In the ADE window, select *Setup – High Performance Simulation*. The *High Performance Simulation Options* window is displayed, as shown below.

Figure 4-140 High Performance Simulation Options



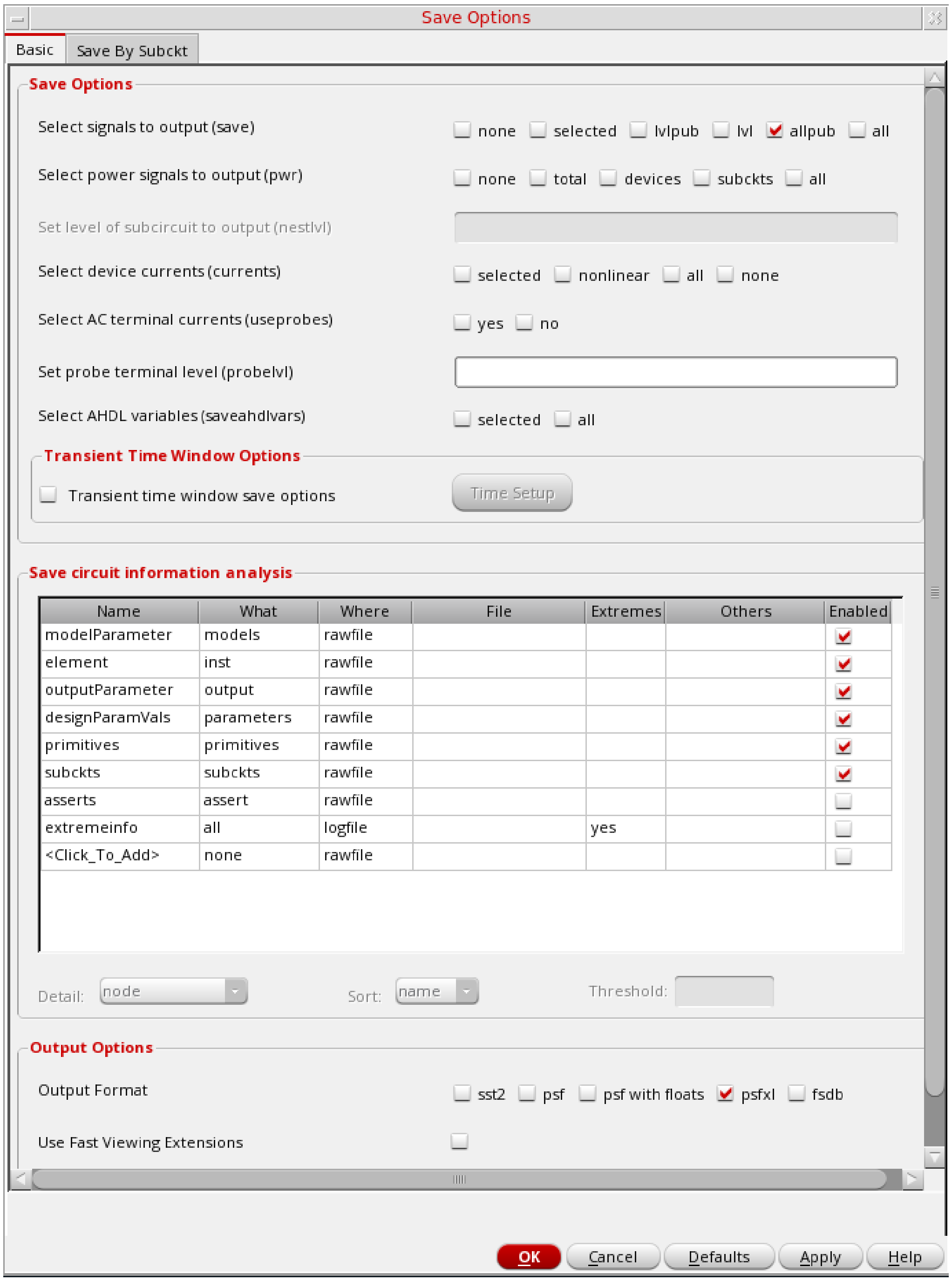
b. In the *High Performance Simulation* window, select *APS* as the *Simulation Performance Mode.* Note that *Auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores.

c. Click *OK*.

10. Select *Outputs – Save All*.

The Save Options form is displayed, as shown below.

Figure 4-141 Save Options Form



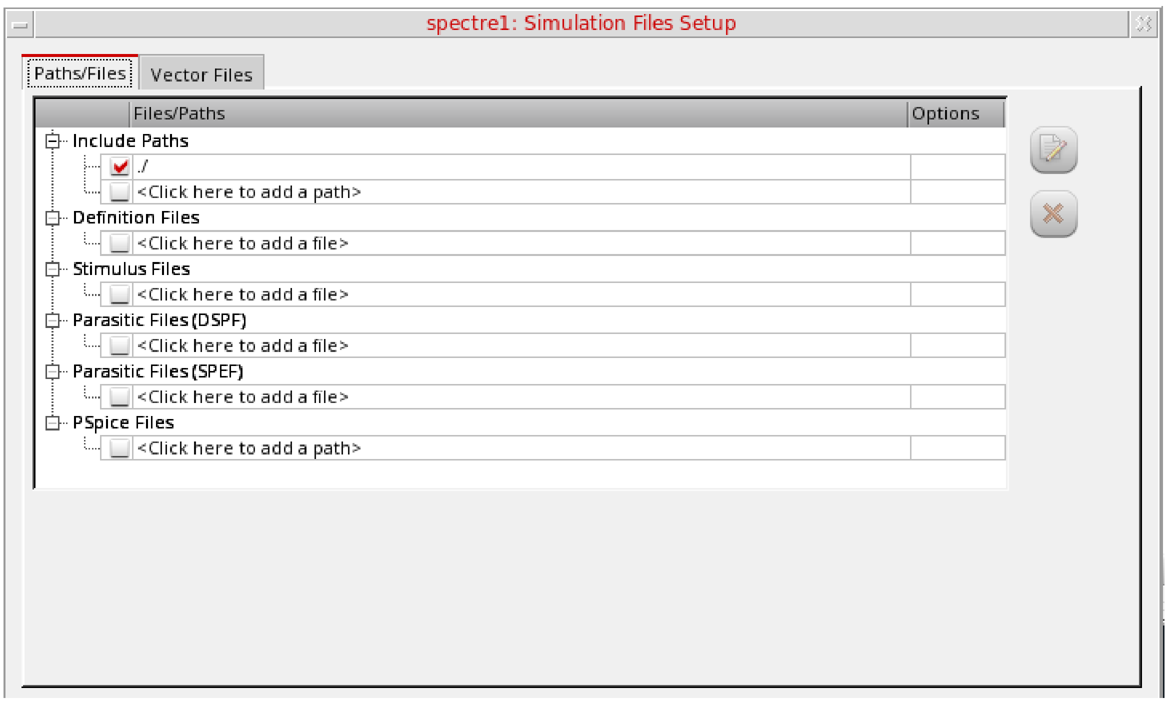
a. In the *Select signals to output(save)* section, make sure that *allpub* is selected. This is the default which saves all node voltages at all levels of hierarchy, but it does not include the node voltages inside the device models.

b. Click *OK* to close the *Save Options* form.

Setup the model libraries, as follows:

11. In the Virtuoso Analog Design Environment window, choose *Setup - Simulation Files*. The *Simulation Files Setup* form is displayed, as shown below.

Figure 4-142 Simulation Files Setup Form



12. Ensure that the *Include Path* is set, as shown above.

13. Select *Setup – Model Libraries.*

The *Model Library Setup* form is displayed.

14. In the *Model Library File* field, type the following as the name of the model file:

`models/modelsRF.scs`

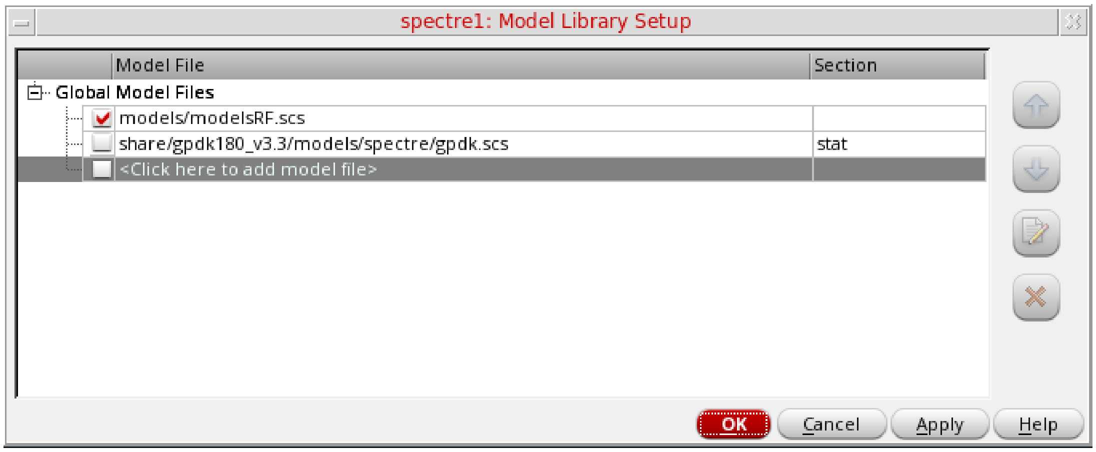
Alternately, you can click *Browse* and browse to the `modelsRF.scs` model file.

15. Make sure that the Model File name is selected.

16. Click *Apply*.

The *Model Library Setup* form looks like the following.

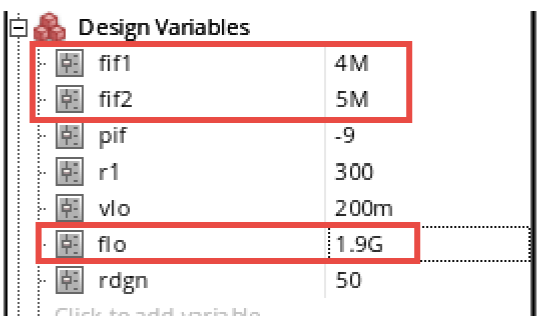
Figure 4-143 Model Library Setup



17. Click *OK* to close the *Model Library Setup* form.

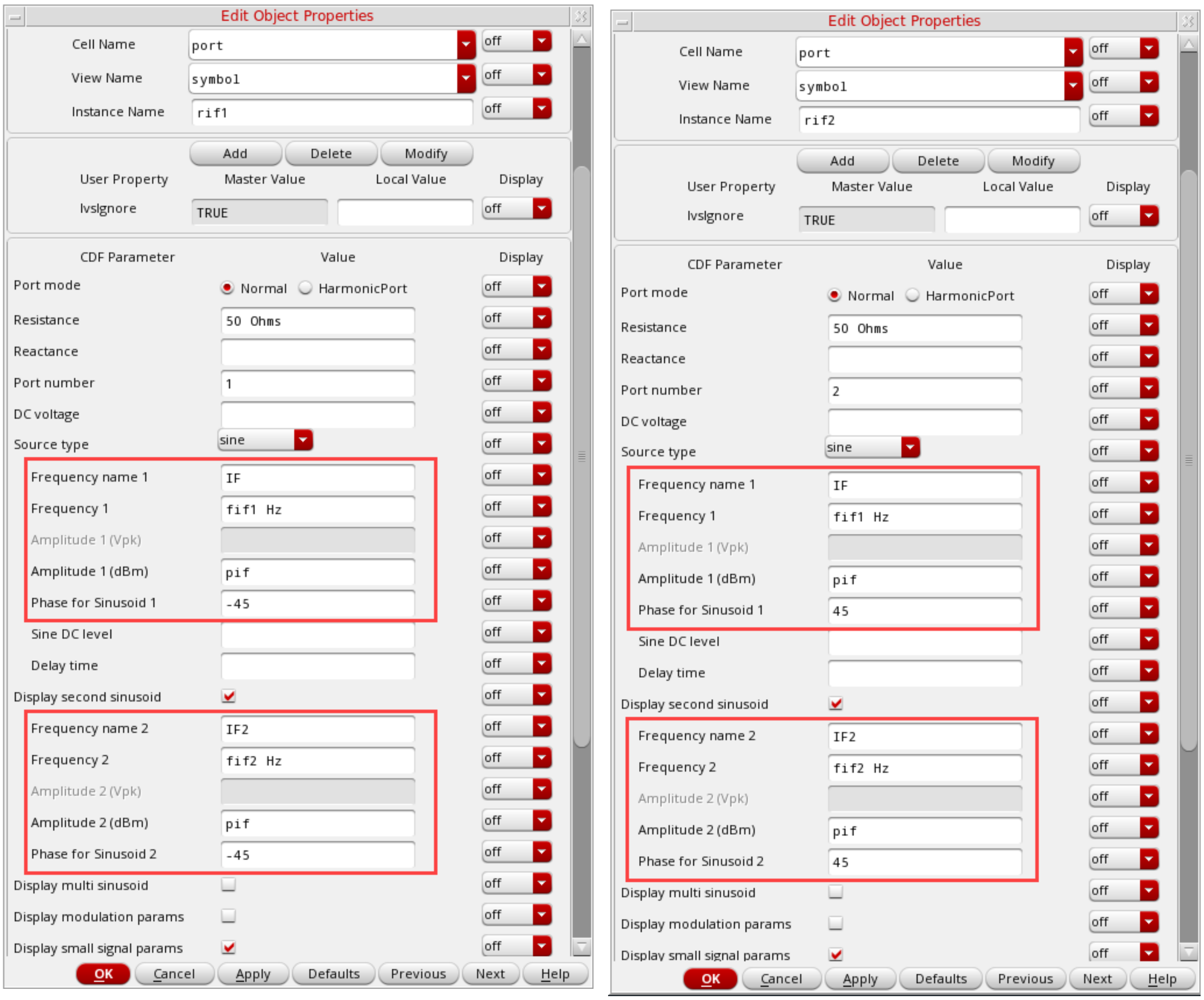
Set the design variables, as follows:

Figure 4-144 Design Variables Section of ADE Explorer



1. In the Design Variables section in the ADE Explorer, verify that the design variables *fif1* and *fif2* are *4M* and *5M*, and *flo* is *1.9G*. If you view the *Edit Object Properties* form for the Input ports (select either the *Input_Top* or *Input_Bottom* port on the left side of the schematic and select the bindkey *q*), you will see how the *rif* tones are specified on the two *rif* ports, as shown below.

Figure 4-145 Edit Properties Forms for IF ports



Set up the HB analysis, as follows:

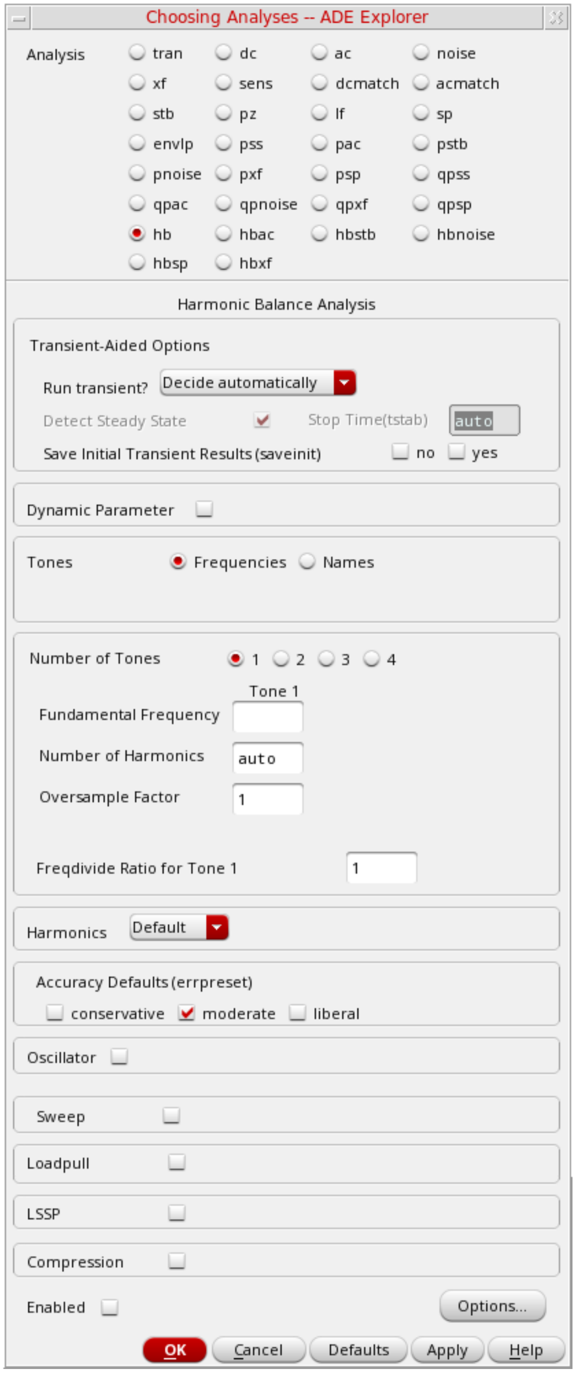
1. In ADE Explorer, select *Analyses – Choose*.

   The *Choosing Analyses* form is displayed.

2. In the *Choosing Analyses* form, select *hb.* The form expands. Since the circuit is mostly sinusoidal (not strongly nonlinear) Harmonic Balance is the appropriate analysis to choose.

Figure 4-146 The HB Choosing Analyses Form

.



Harmonic balance can set harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. To enable this, select *Decide automatically* in the *Transient-Aided Options* section. This single action will cause a transient analysis to be run until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1* (when *Frequencies* is selected) or for the tone that has *tstab* enabled (when *Names* is selected).

3. In the Transient-Aided Options section of the form, select the following

a. For *Run transient?* select *Decide automatically* (this is the default).

Run transient? will run the LO signal using the transient (In SpectreRF, this is called the *tstab* interval) for a short period of time. At the end of *tstab*, an FFT is performed, and this is used as the starting point in the harmonic balance analysis. Doing this improves the convergence of hb by giving it a better starting point at the cost of a short transient analysis.

b. For *Stop time (tstab), auto* is automatically populated in the field.

When *auto* is selected for *Stop time*, a small number of periods of the LO is run using the transient analysis. During this time, the signal is checked for steady-state conditions. If steady-state is not reached in the initial number of periods, more periods can be added automatically by the simulator. Using this feature allows an accurate FFT for the starting point in the hb iterations.

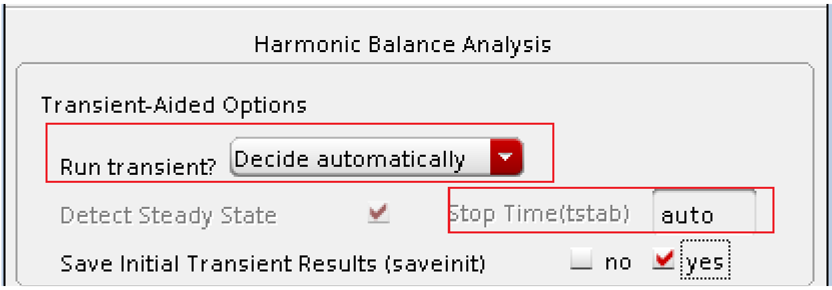
When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide automatically* simplifies the setup for harmonic balance, and produces correct answers from hb without needing to know how to set hb up manually.

c. For *Save Initial Transient Results (saveinit),* select *yes*.

During the transient-assisted HB simulation, a transient simulation runs before the frequency domain iteration of harmonic balance. The LO signal in *Tone1* is enabled for this measurement. At the end of the *tstab*, an FFT is run and its result is used as the starting point for the frequency domain iterations

All the signals are applied and the simulation is done in the frequency domain. Only the signal and its harmonics are calculated.Transient Assisted Harmonic Balance.

Figure 4-147 Transient-Assisted Harmonic Balance



4. In the *Tones* section, select *Names*. When *Names* is selected, the *Tones* portion of the form expands. All the sources in the top-level schematic are read into the form automatically.

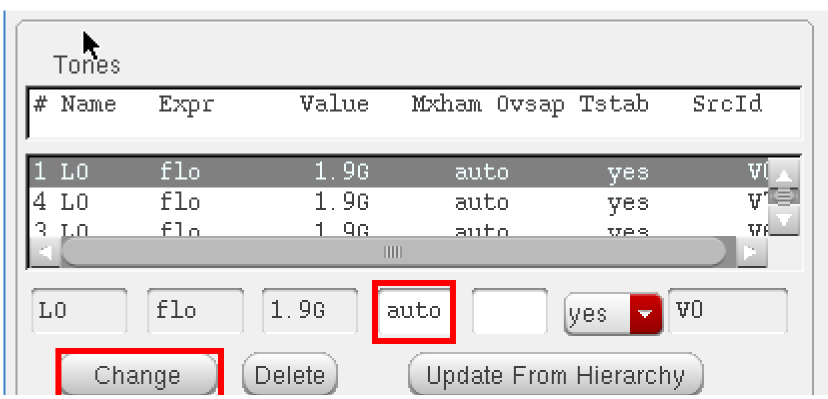
5. Select one of the LO sources in the *Tones* section. In the *Mxham* field, type `auto`.

   Spectre will automatically detect the number of harmonics required. Note that all four LO sources are now updated with a *Mxham* of *auto*.

   You can use hb with up to four signals present in the circuit. In this circuit, there are three tones, the LO and two RF tones. The LO signals have the same name, the same frequency, and are considered a single tone. Whenever you have two signals at the same frequency, make sure you set the *Frequency Name 1* or *Frequency Name 2* property on the source to the same name as it was done in this example. When *Tones* is set to *Names*, the simulator considers both of the sources as a single frequency.

   You viewed the names (*frf1* and *frf2*) in the input port sources in an earlier step.

Figure 4-148 Tones Section of hb Choosing Analyses Form



When you set the number of harmonics to *auto*, this identifies the signal on which to run *tstab*. This tone should be the LO tone. Only one signal can have transient assist, that being the signal with *tstab* set to *yes*; the signal with auto harmonics set.

Note: If you set the harmonics manually for all of the tones, set *tstab* to *yes* for the LO tone. Because you are using the auto-harmonics feature, you do not need to set the number of harmonics on the LO tone, nor do you need to set *Ovsap (*oversample*).* By default, *Ovsap* is set to *1.*If for some reason, you are not using the auto-harmonics feature, follow these guidelines:

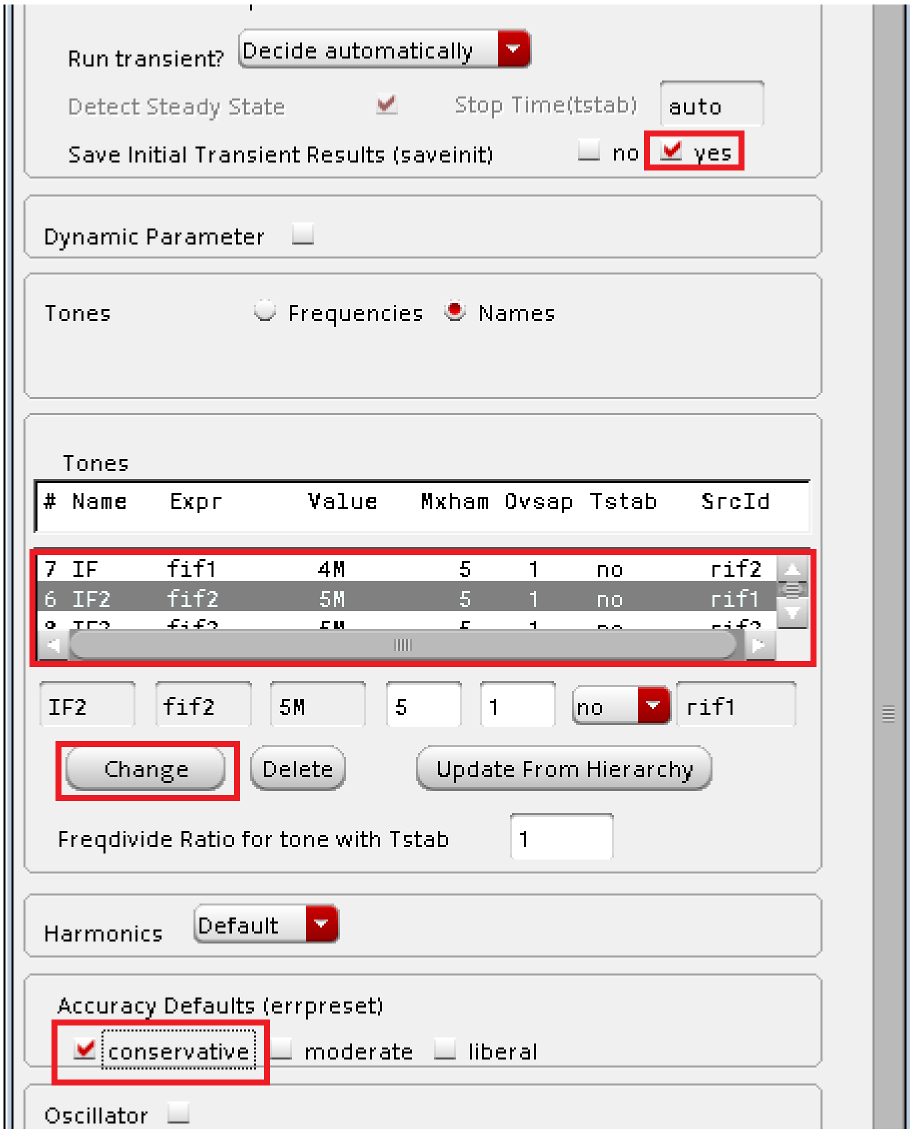
> If not using *Mxham* set to *auto* on the LO tone, change the *Maximum harmonics* on the LO tone from *3* to *10*. Place the cursor in the field under *Mxham* and type *10*.
>
> Since the circuit is operating well above the compression point, a higher number of harmonics is advised.
>
> If the circuit is mostly sinusoidal (voltages *and currents*), leave *oversample* set to *1*. For strongly nonlinear circuits, you may need to increase oversample.

In general, when you set harmonics manually, if a signal is below compression, start with three harmonics. If a signal is at compression, start with five harmonics. If a signal is above compression, start with seven harmonics. Run the simulation. Now increase the number of harmonics on each tone separately by about a factor of 1.5, and run the simulation again. If the result changes, increase harmonics again. If the result did not change, the original number of harmonics was enough, and you might be able to decrease the number of harmonics. For fastest runtime with full accuracy, use the smallest number of harmonics for each tone that is applied to the circuit

6. Once finished, click *Change*. The form updates. Make sure that all of the LO tones have *Mxham=auto* and *Tstab=yes*.
7. Select the IF tone in the *Tones* box. For this measurement, three harmonics on the IF tones are not enough. You need to increase the number of IF tones for greater accuracy. Set *Mxham* to *5*. Do the same for IF2. Five (5) was found to be the optimum number of harmonics for the two RF tones, which are deliberately matched in amplitude.
8. Set *Accuracy Defaults (errpreset)* to *conservative*. *conservative* is used because the third and fifth order intermodulation distortion will be calculated with this setup. Since these amplitudes are small, high accuracy is needed.
9. Leave the rest of the form set to the default values.

   The *Choosing Analyses* form should look like the following:

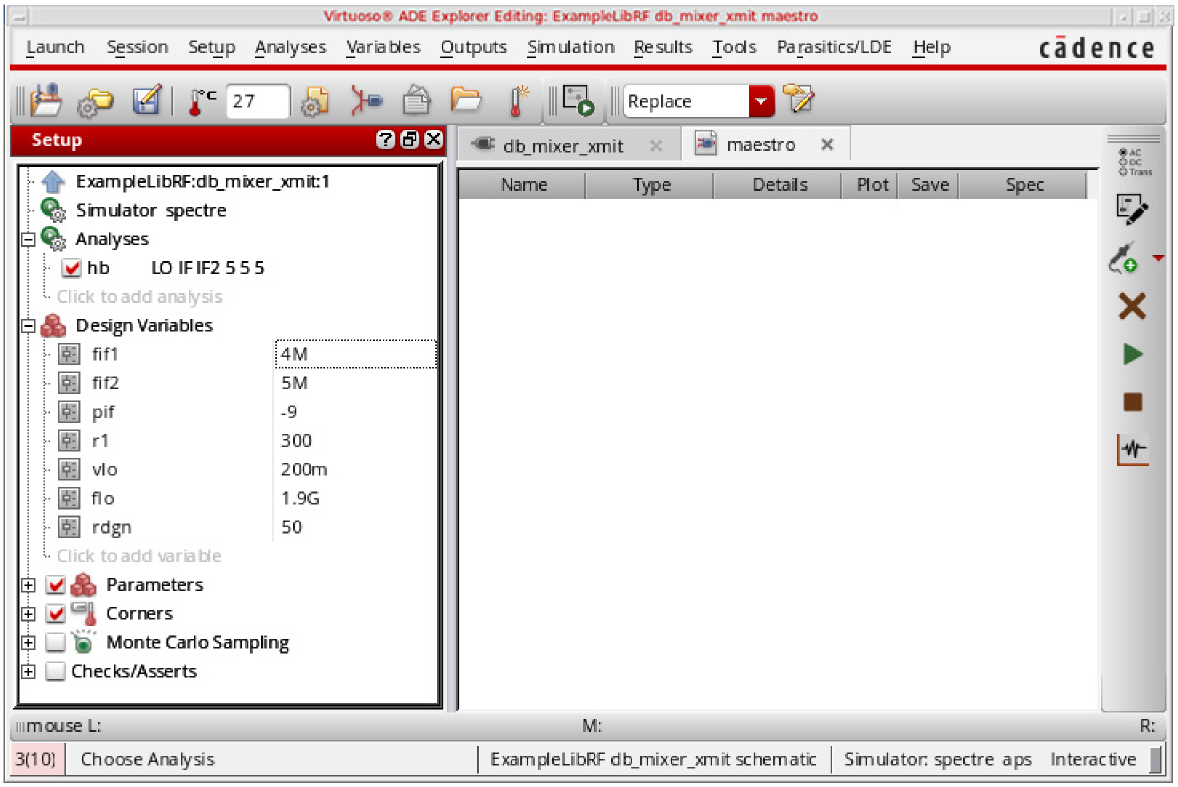
Figure 4-149 HB Choosing Analyses Form



10. Click *OK* at the bottom of the *Choosing Analyses* form.

ADE Explorer will look like the following:

Figure 4-150 ADE Simulation Window



Run the simulation and plot the results, as follows:

1. Start the simulation by clicking the green arrow icon. in ADE or in the Schematic Editor.

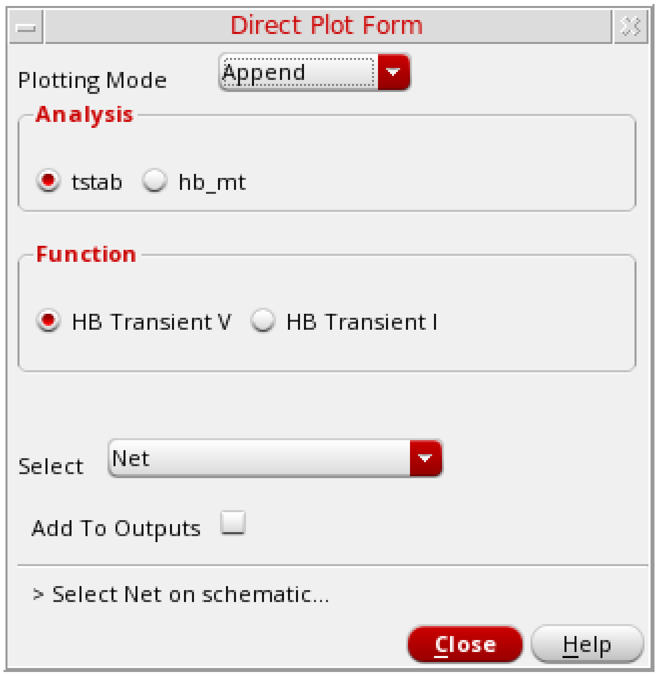
   This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` logfile). When the analysis has completed, you may iconify the status window.

2. Next, you will plot the Voltage Spectrum.

   In ADE Explorer, select *Results - Direct Plot - Main Form*. Alternately, you can click the *Direct Plot* icon ( )in the schematic window.

   The Direct Plot Form is displayed.

Figure 4-151 Direct Plot Form



3. In the *Direct Plot Form*, you will notice that there are two available analyses, *tstab* and *hb_mt*. The tstab results appear because you chose *Save Initial Transient Results" (saveinit)* in the *Choosing Analyses* form. When there is more than one tone in the circuit, you will see *hb_mt*. A single tone simulation will show *hb*.

   a. Select *hb_mt* in the Analysis section.

   b. Select *Voltage* in the *Function* section.

   c. Select *Net* in the *Select* drop-down list.

   d. Select *spectrum* in the *Sweep* section.

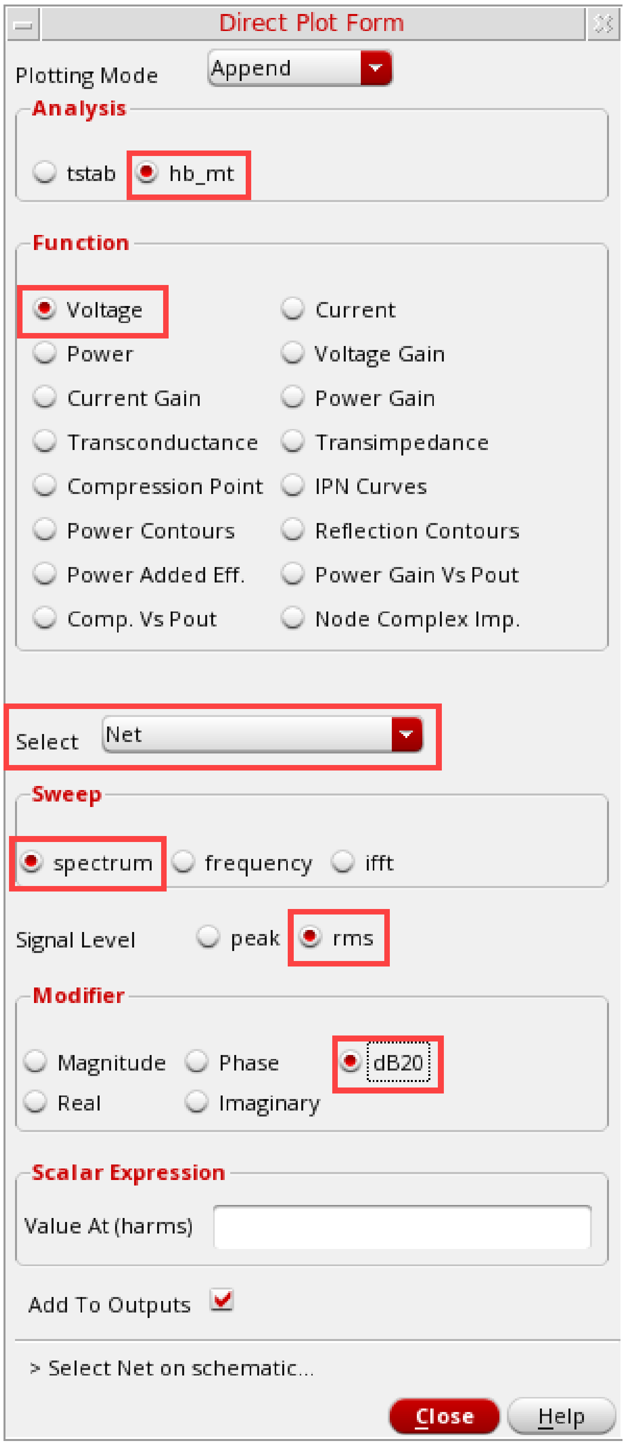
   e. Select *rms* in the *Signal level* section.

   f. Select *dB20* in the *Modifier* section.

   g. If you want to plot this setup in subsequent simulation, Select the *Add to Outputs* option.

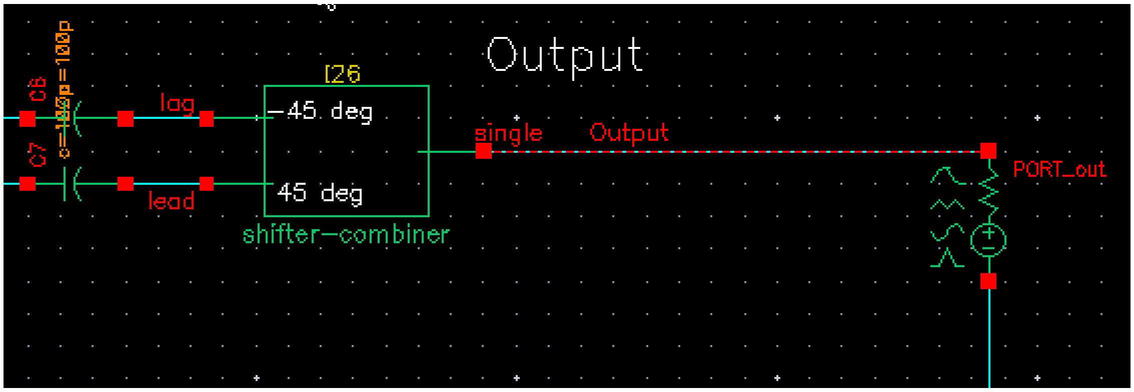
   The *Direct Plot Form* is shown below.

Figure 4-152 Harmonic Balance Direct Plot Form



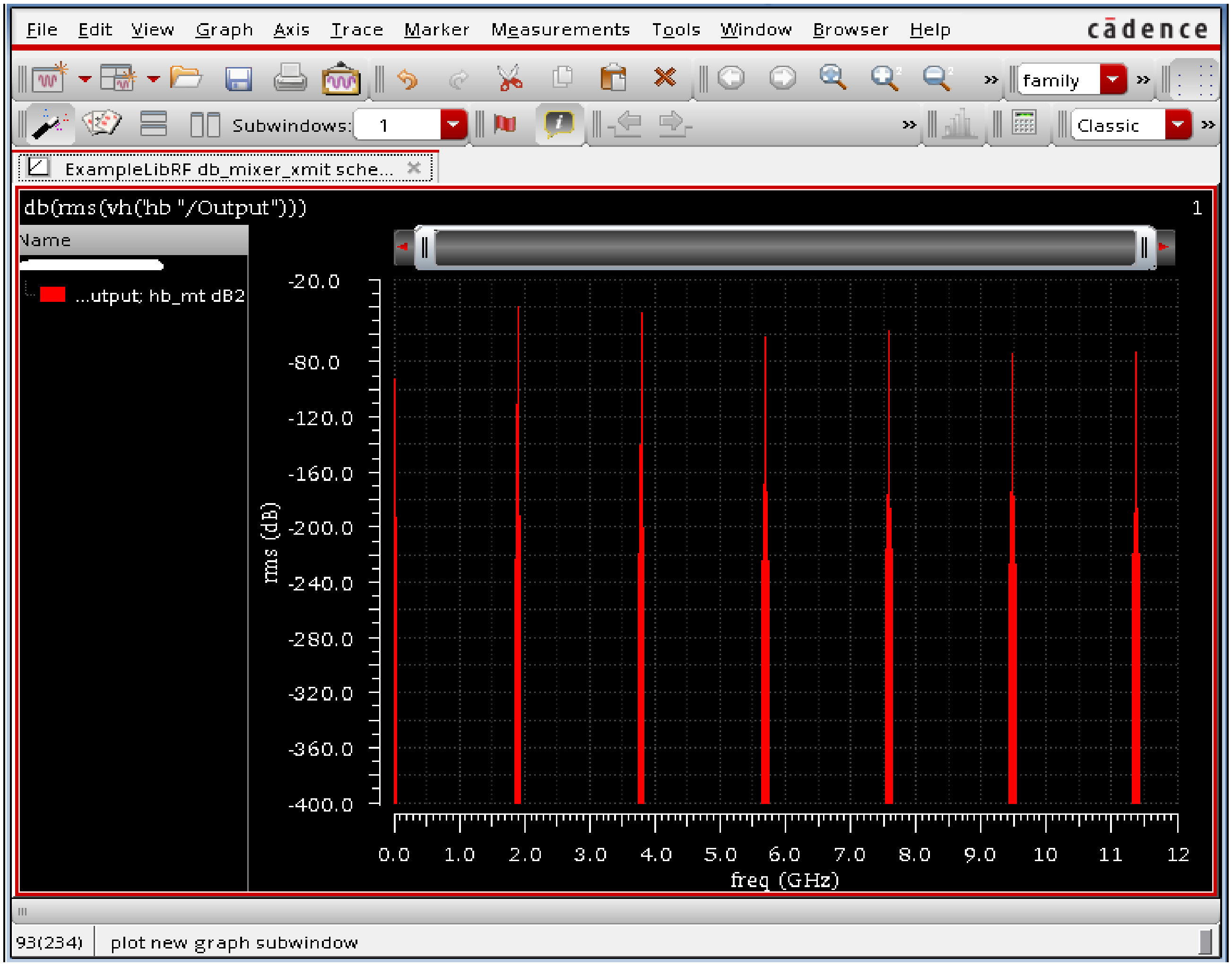
4. Click on the net labeled *Output* in the schematic.

Figure 4-153 Output Net in Schematic



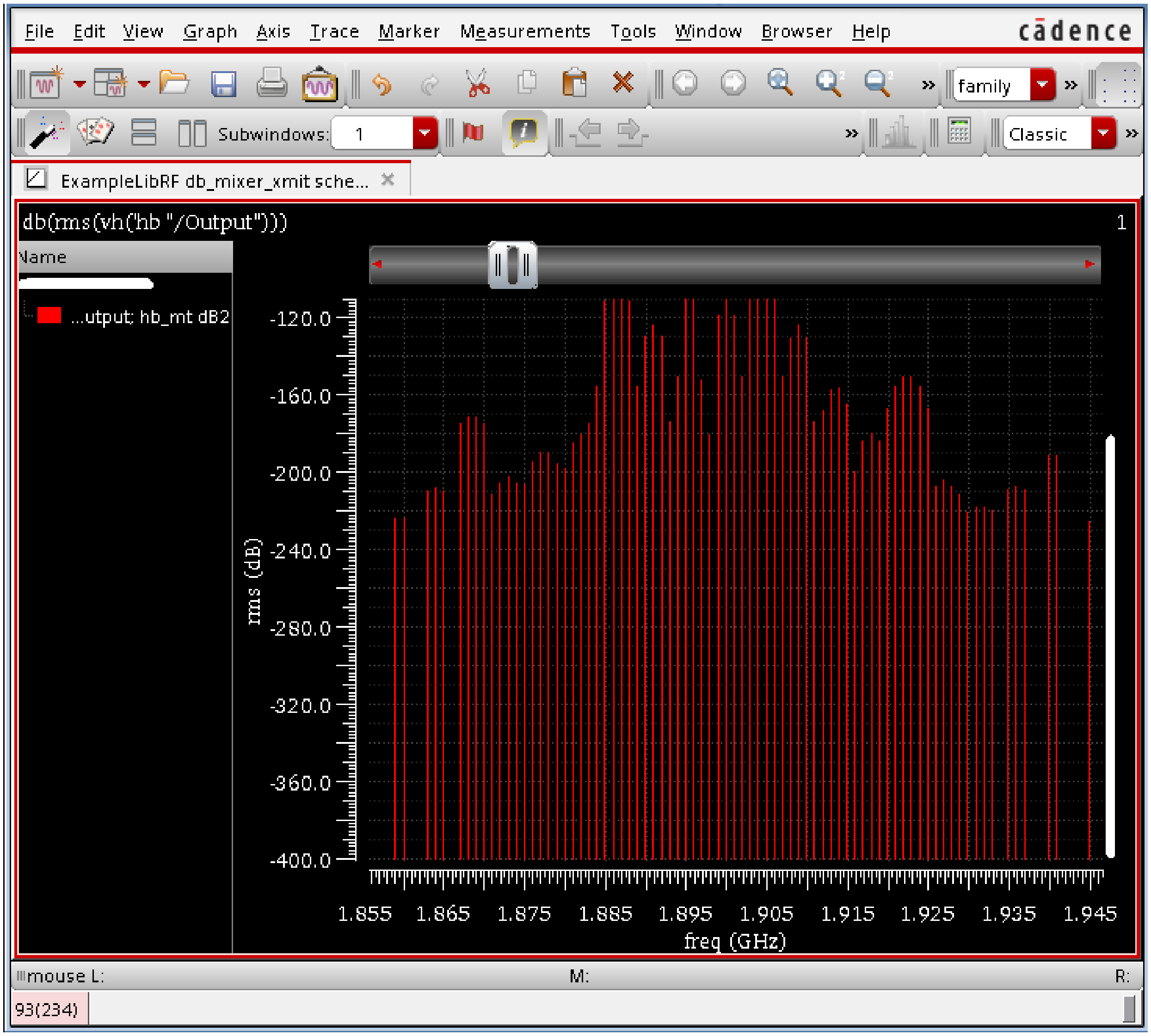
The plot is displayed.

Figure 4-154 Voltage Spectrum of Transmit Mixer



5. Zoom in on the area around 1.9GHz. Place the cursor in the waveform window, click the right mouse button and draw a box around the signals surrounding 1.9GHz. You may need to do this several times to zoom in specifically to the area around 1.9GHz.
6. The waveform window should look similar to the following:

Figure 4-155 Zoomed in Voltage Spectrum

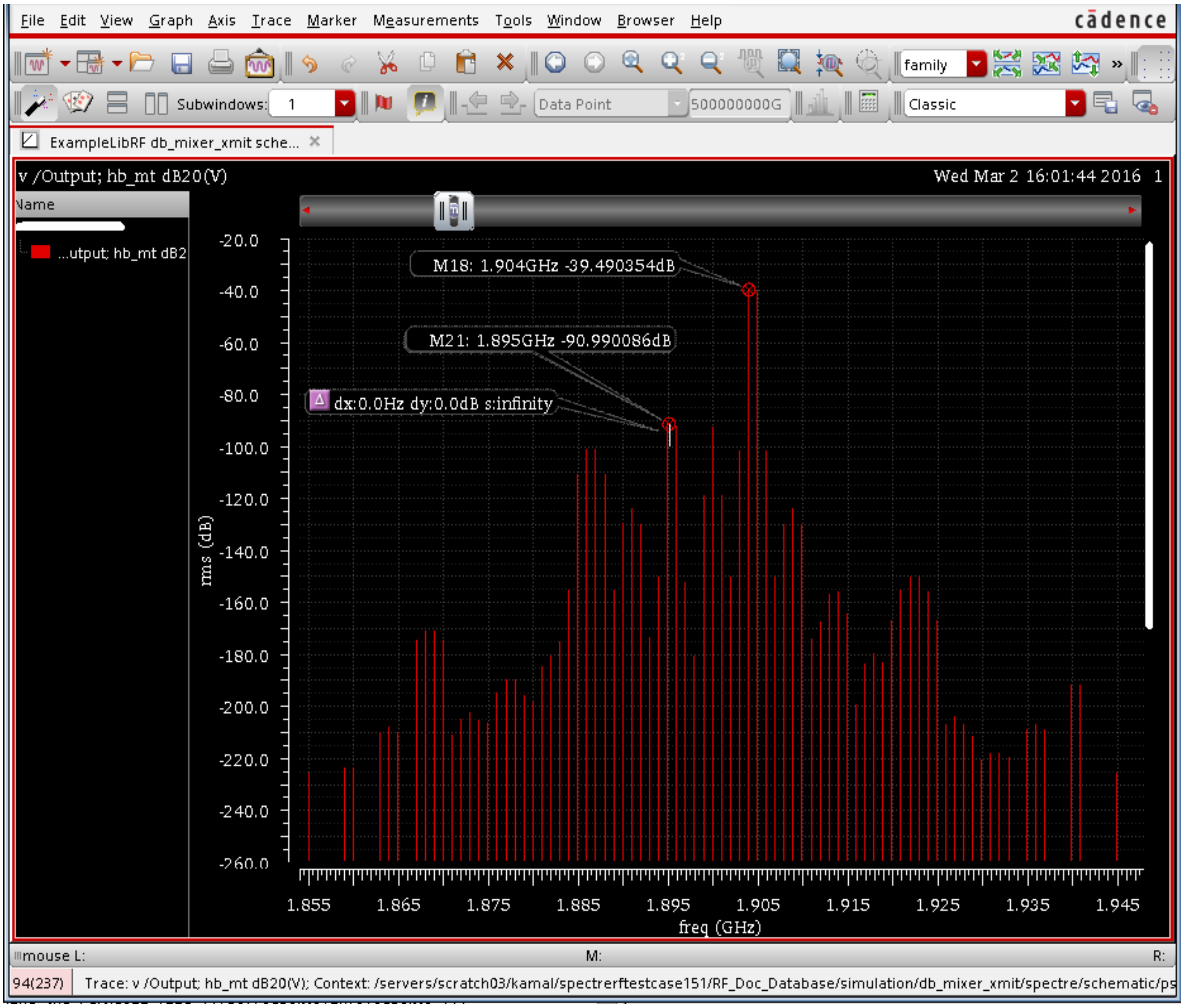


7. Move the mouse cursor to the tip of the harmonic at frequencies *1.895G*, *1.896G*, *1.9G*, *1.904G*, and *1.905G*. The tracking cursor values read the following:

	Frequency (GHz)	Amplitude (peak, dB)
Low side RF freq 2	1.8995	-90.99
Low side RF freq 1	1.896	-91.90
LO Frequency	1.9	-92.43
Hi side RF Freq 1	1.904	-39.49
Hi side RF Freq 2	1.905	-39.49

Image rejection is the ratio of the power in the passband divided by the power in the image. If you have dB, this is dB at the passband frequency minus dB at the image frequency. Place a marker on the RF1 frequency (1.904G) by placing the cursor on the trace and pressing the *m* bindkey Place a delta marker on the Image frequency (1.895GHz) by placing the cursor on the trace and pressing *d*.

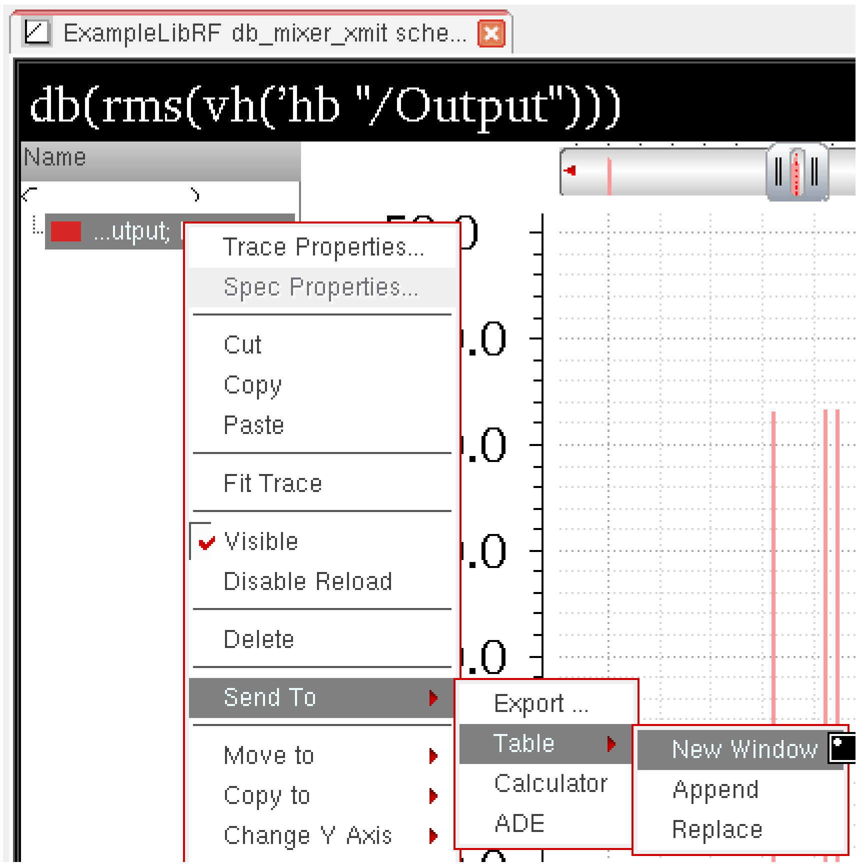
Figure 4-156 Image Rejection



The image rejection is 51.5dB.

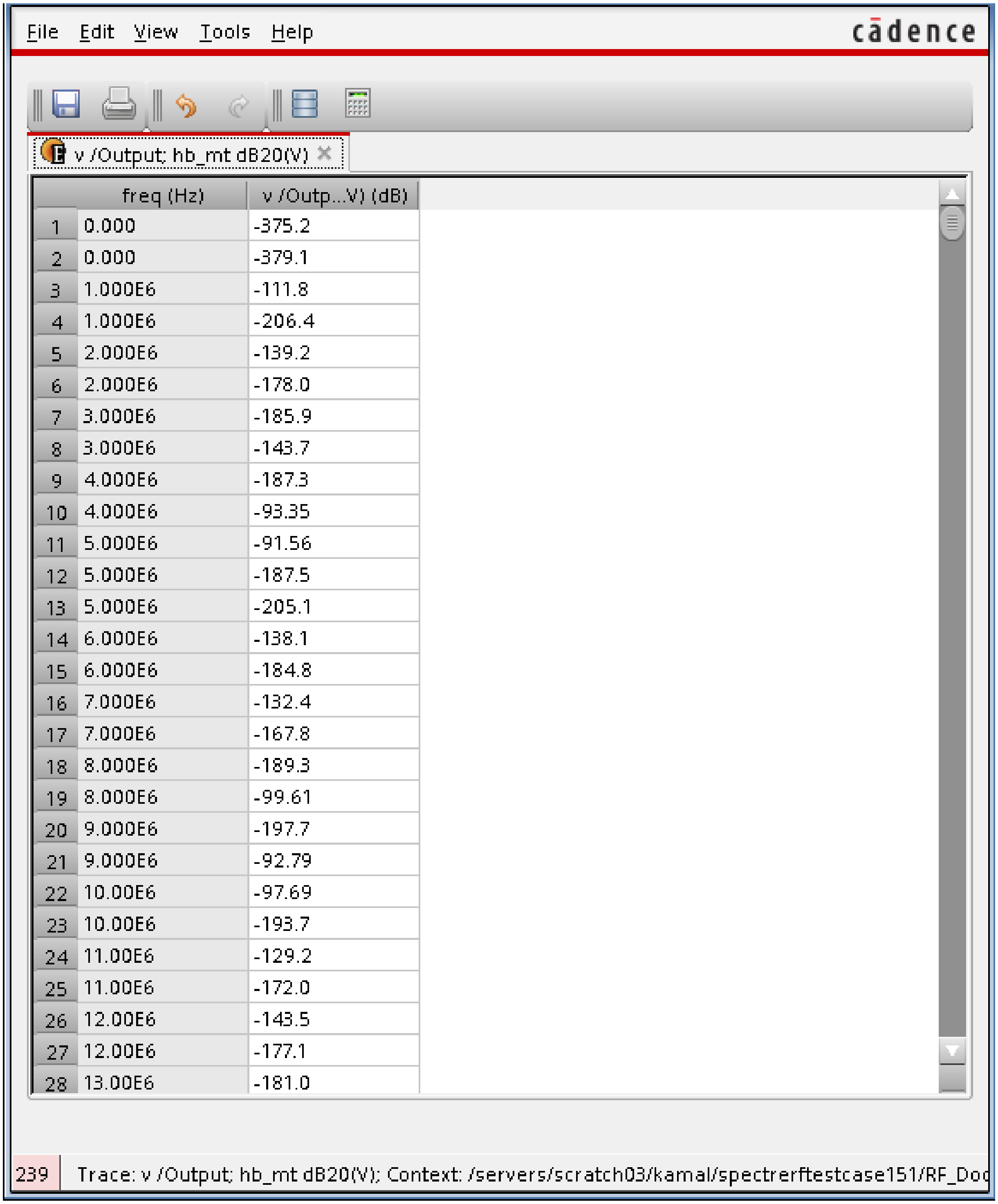
8. You can create tabular data from the results. In the waveform window, select the trace by clicking the legend on the left side of the graph area, clicking the right mouse button, and selecting *Send to - Table - New Window.*

Figure 4-157 Sending Waveform Data to Table



A tabular table of data is displayed in a Virtuoso Visualization and Analysis XL Table window. You can scroll down to the frequencies of interest using the scroll bar on the right side of the table.

Figure 4-158 Tabular Data



	freq (Hz)	v /Outp...V) (dB)
1	0.000	-375.2
2	0.000	-379.1
3	1.000E6	-111.8
4	1.000E6	-206.4
5	2.000E6	-139.2
6	2.000E6	-178.0
7	3.000E6	-185.9
8	3.000E6	-143.7
9	4.000E6	-187.3
10	4.000E6	-93.35
11	5.000E6	-91.56
12	5.000E6	-187.5
13	5.000E6	-205.1
14	6.000E6	-138.1
15	6.000E6	-184.8
16	7.000E6	-132.4
17	7.000E6	-167.8
18	8.000E6	-189.3
19	8.000E6	-99.61
20	9.000E6	-197.7
21	9.000E6	-92.79
22	10.00E6	-97.69
23	10.00E6	-193.7
24	11.00E6	-129.2
25	11.00E6	-172.0
26	12.00E6	-143.5
27	12.00E6	-177.1
28	13.00E6	-181.0

Close the *Direct Plot Form* and the waveform window.

Summary

In this section, you measured the voltage spectrum and image rejection on an up-converting double-balanced (transmit) mixer using hb analysis. In the next section, you will measure the third-order intercept.

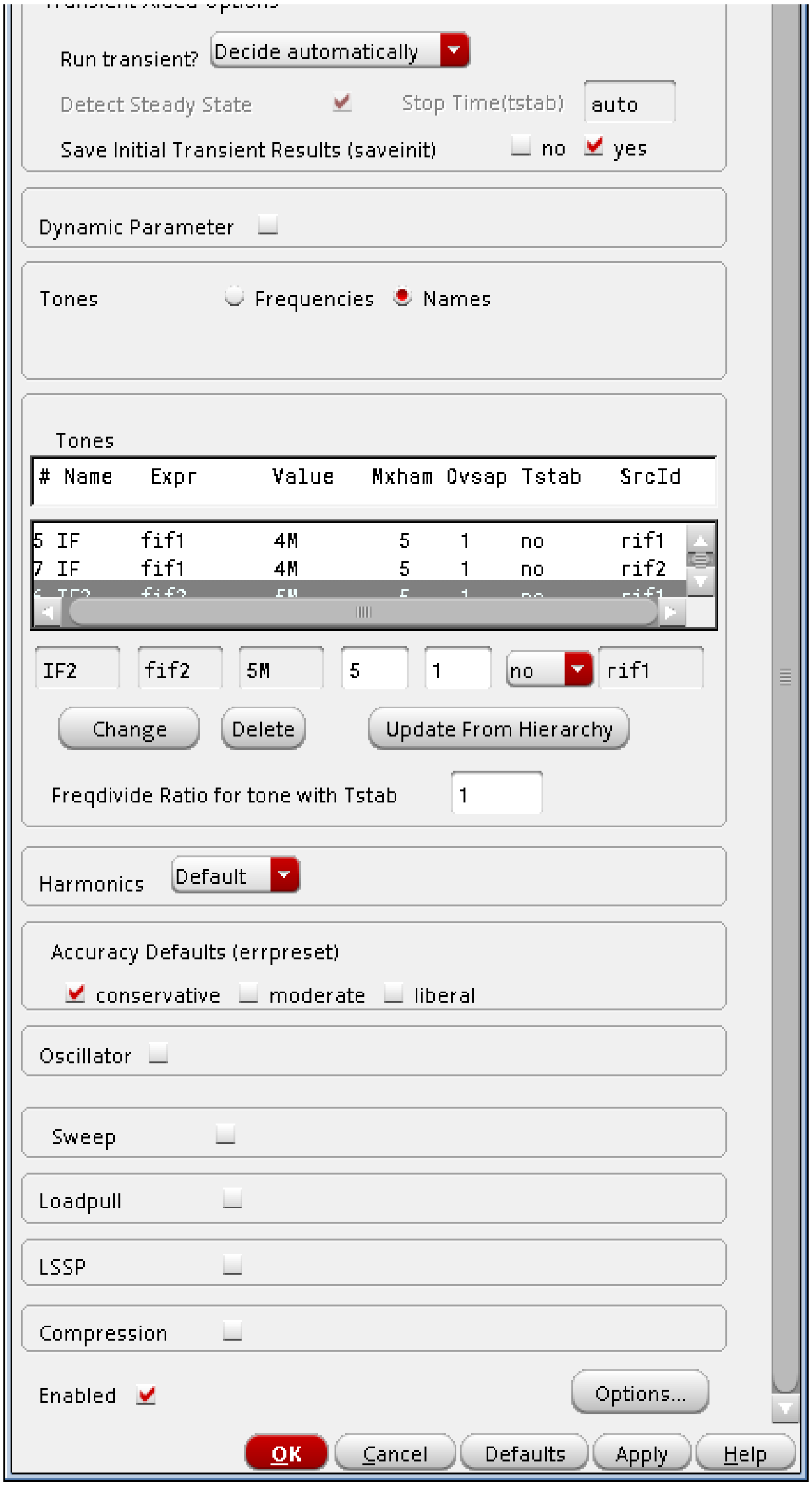
Three Tone IP3

For transmitters, there might be an IP3 spec at a signal in the large-signal range. This will not be a small-signal projection, so the traditional HB/HBAC IP3 or rapid IP3 cannot be used for this measurement. Instead, multi tone HB or QPSS-HB would be appropriate.

The simulation setup is nearly the same as the previous example. You only need to make a couple of modifications.

1. Open the hb *Choosing Analyses* form by double-clicking the *hb* line in ADE Explorer. The Choosing Analyses form is displayed, as shown below.

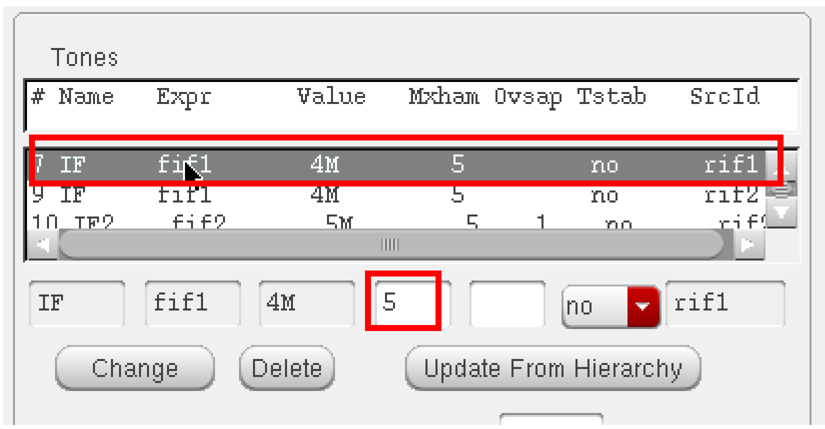
Figure 4-159 hb Choosing Analyses Form



2. Ensure that *Accuracy Defaults (errpreset)* is set to *conservative*. *conservative* is set because third and fifth-order intermodulation distortion will be calculated with this setup. Because these amplitudes are small, high accuracy is needed.

3. In the *Tones* section, select the line containing one of the IF tones. For this measurement, you will again use *5* harmonics for the IF and IF2 tones.

Figure 4-160 Choosing Analyses Tones Section - Setting Mxham for IF Tones



4. When making an IP3 measurement using three large tones, you will need to do a power sweep. Click the *Sweep* button in the *Choosing Analyses* form. The form expands.

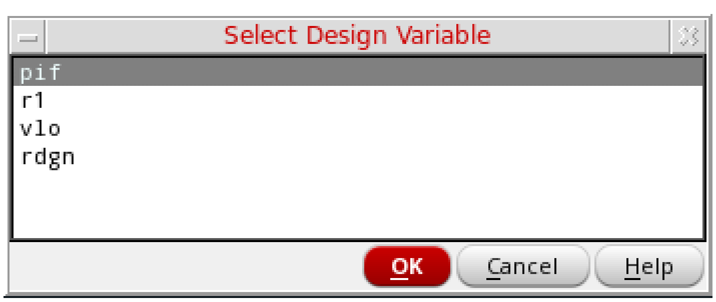
5. In the *Sweep* section:

   a. Set *Sweep* to *Variable*.

   b. For *Frequency Variable?* select *no*. You will be sweeping input power rather than frequency.

   c. Click the *Select Design Variable* button and choose *pif*. You will be sweeping the input power on the IF ports.

Figure 4-161 Select Design Variable Form



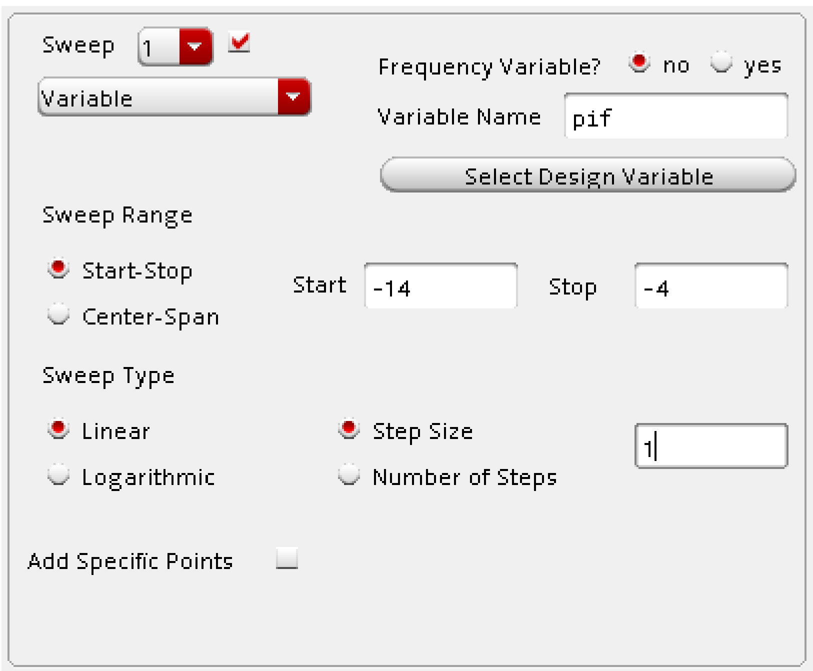
d. *Click OK*.

e. In the *Sweep Range* section, choose *Start-Stop* and sweep from `-14` to `-4`.

f. Set the *Sweep Type* to *Linear* and *Step Size* to `1`.

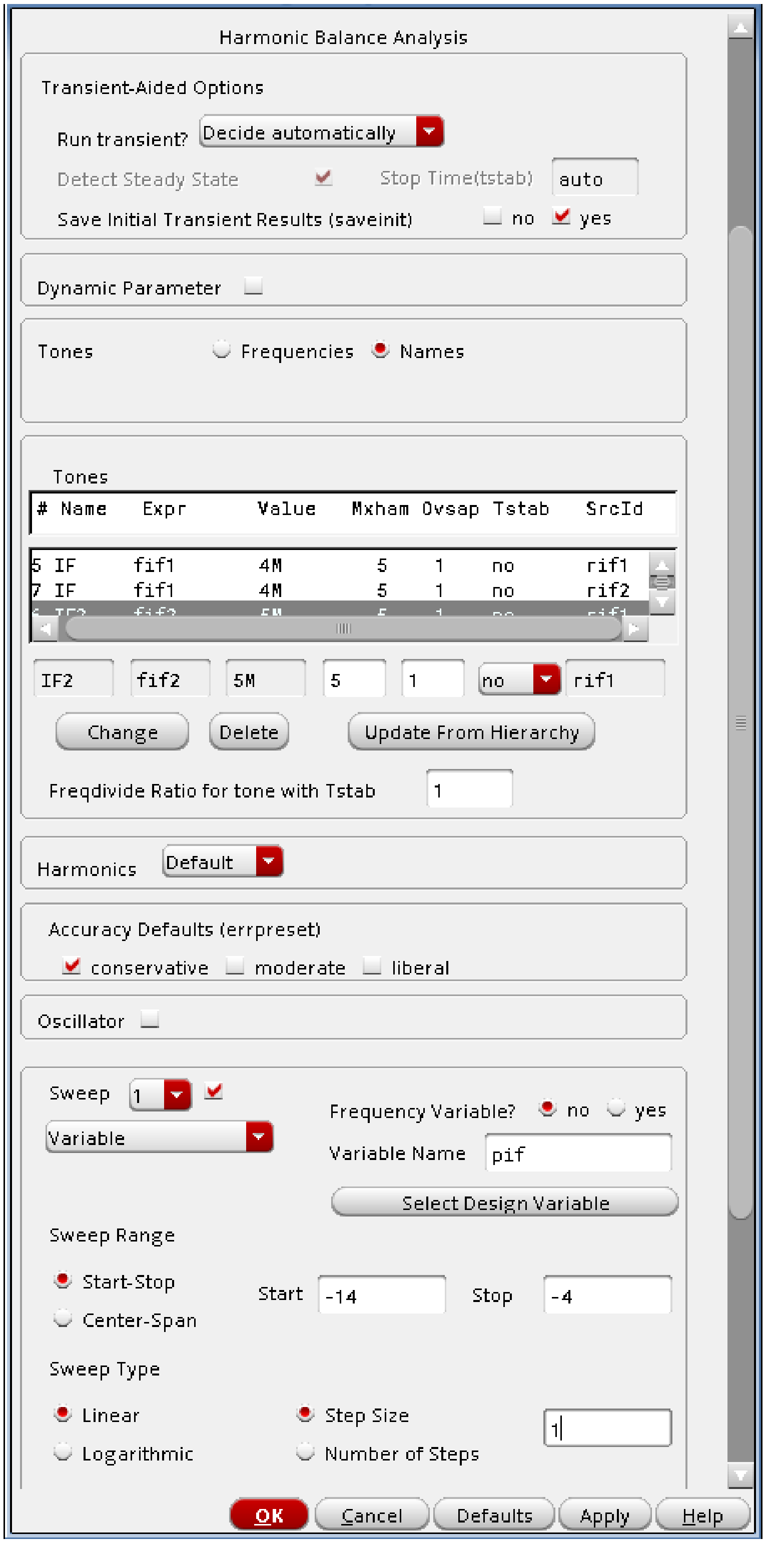
The Sweep section of the form should look like the following:

Figure 4-162 hb Choosing Analyses Form Sweep Section for 3 Tone IP3 Measurement.



6. The complete hb *Choosing Analyses* form should look like the following:

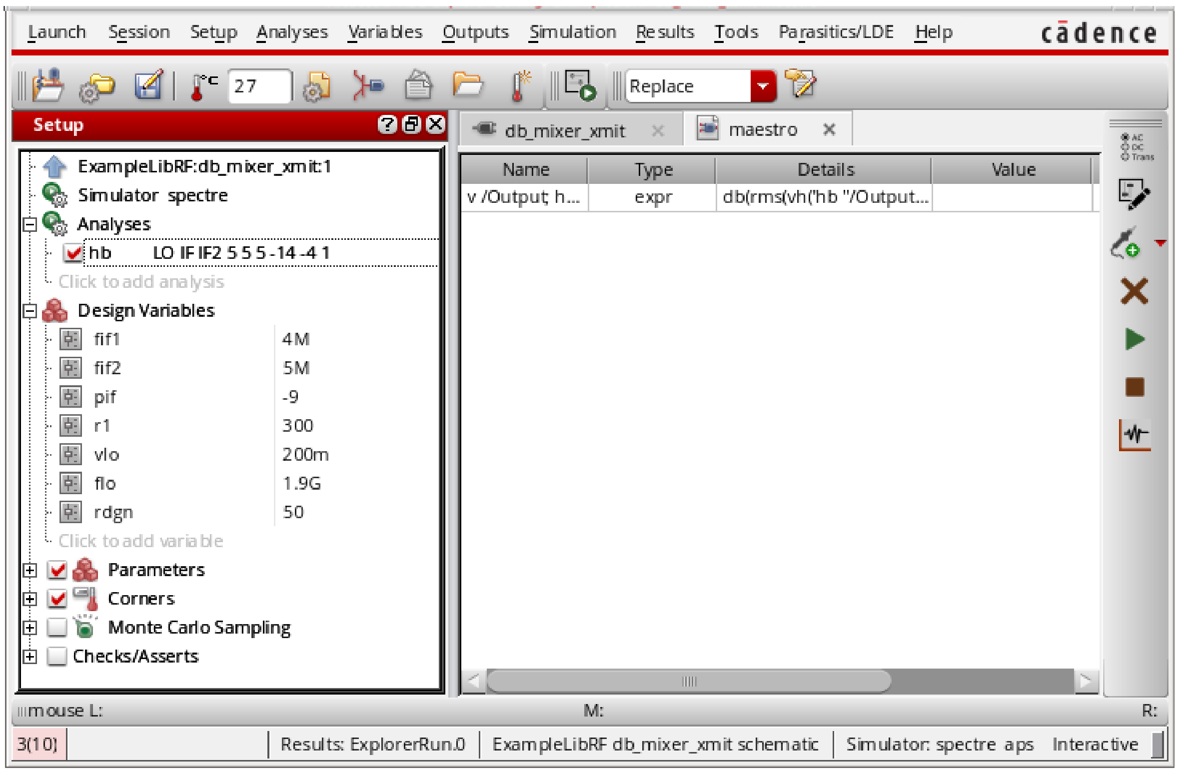
Figure 4-163 hb Choosing Analyses Form for 3 Tone IP3



7. Click *OK* at the bottom of the *Choosing Analyses* form.

   The *Analyses* section of ADE Explorer must be as shown below.

Figure 4-164 ADE Simulation Window



Note: If you selected the *Add to Outputs* button in the *Direct Plot Form* in one of the earlier simulations, you will see the *Outputs* in the *Outputs* pane of the simulation window. *Deselect* the plot expressions in the *Outputs* pane. If you do not, a family of spectral plots will be displayed when you plot the results.

Run the simulation and plot the results, as follows:

1. Start the analyses by clicking the green arrow icon.  in ADE Explorer or in the Schematic Editor.

   This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` logfile). When the analysis has completed, you may iconify the status window.

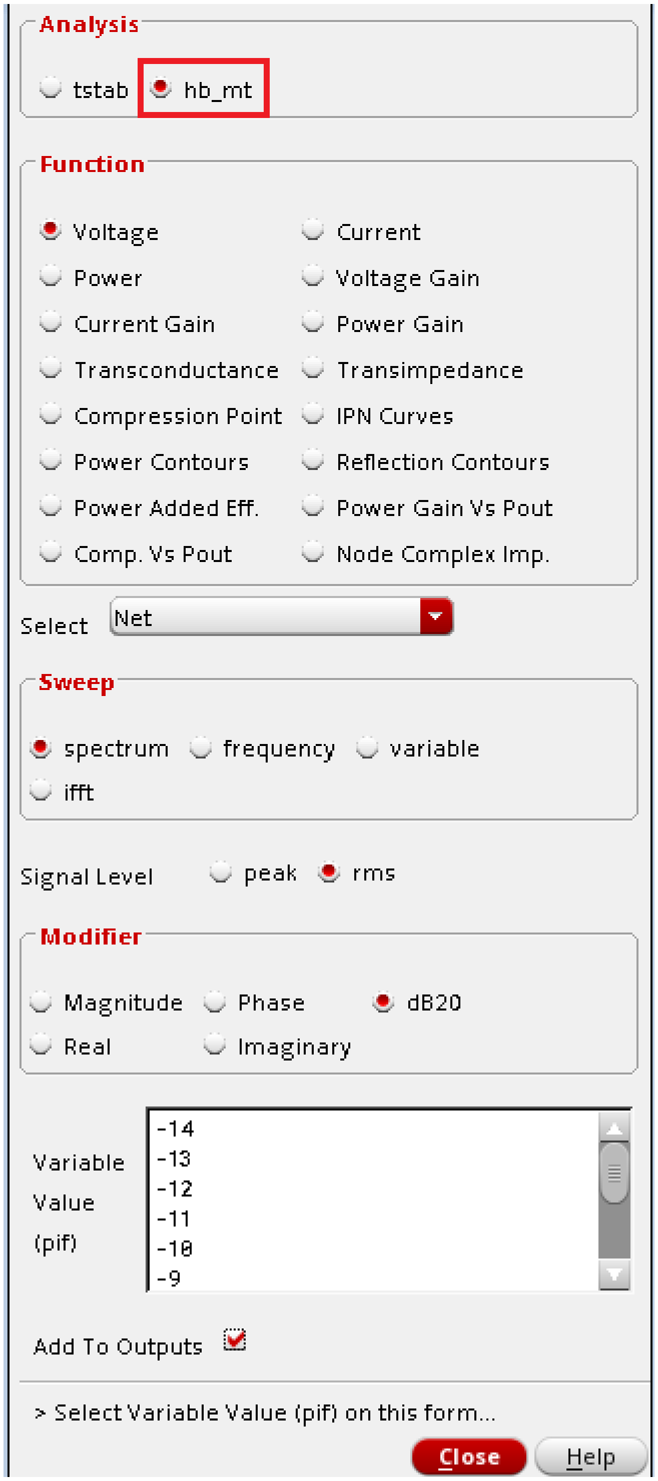
   Next, you will plot the IP3 results.

2. When the simulation has finished, in the ADE Explorer, select *Results - Direct Plot - Main Form.*

The *Direct Plot Form* is displayed, as shown below.

Figure 4-165 Harmonic Balance Direct Plot Form

.



3. In the *Direct Plot Form*, you will notice that there are two available analyses, *tstab* and *hb_mt*. The transient-assisted results are accessible by selecting the *tstab* analysis. The hb results are accessible by selecting the *hb_mt* analysis. Note that when there is more than one tone, you will see *hb_mt* (signifying harmonic balance multi-tone). A single tone simulation will show *hb*. In the *Analysis* Section, make sure that `hb_mt` is selected.

   a. In the *Function* section select *IPN Curves*.

      Note: You will be plotting Input IP3. Note that the N in IPN Curves is selectable an can be 2,3, 4, 5, 6, or 7.

   b. Select *Port(Fixed R(port))*.

   c. Set *Circuit Input Power* to *Variable Sweep ("pif")*. You will be plotting output power vs. swept input power. *pif* is the swept power specified on the port

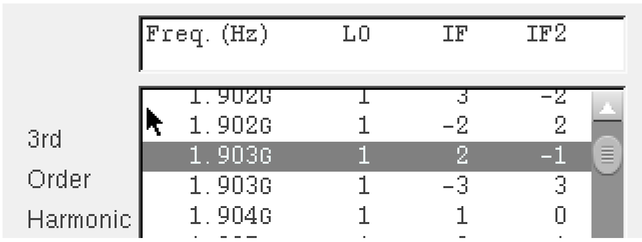
   d. Set the *Input Power Extrapolation Point* to *-9.* This is the power at which the circuit operates.

   e. Select *Output Referred IP3* and *Order 3rd* to plot input referred IP3.

   f. The main upconversion frequencies are at 1.904G and 1.905G. The third order intermodulation products are at 1.903G and 1.906G. Select the lowest amplitude first order term and the largest amplitude of the third order term.In this case, since both first order terms and both third order terms are almost identical in amplitude, it does not matter which first and third order terms are selected. Select *1.906G* for the third order term and *1.904G* for the first order term. Alternately, you can select *1.905G* for the first order term and *1.903G* for the third order term.

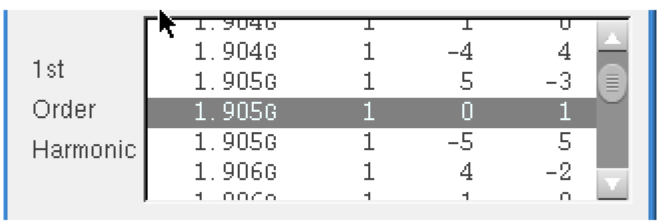
   g. Note that if you have more than one selection for any of the frequency terms (for example, the third-order term), you will want to select the correct one. To select the correct frequency, add the absolute value of the indices displayed for the desired frequency and choose the frequency with the lowest value. For example, for the two 1.903G, frequencies, you have `|1|+|2|+|-1|=4` and `|1|+|-3|+|3|=7`. You want to choose the first *1.903G* entry because adding the absolute value of the indices produces the lowest number.

Figure 4-166 Selecting the Correct 3rd Order Term When Plotting IP3



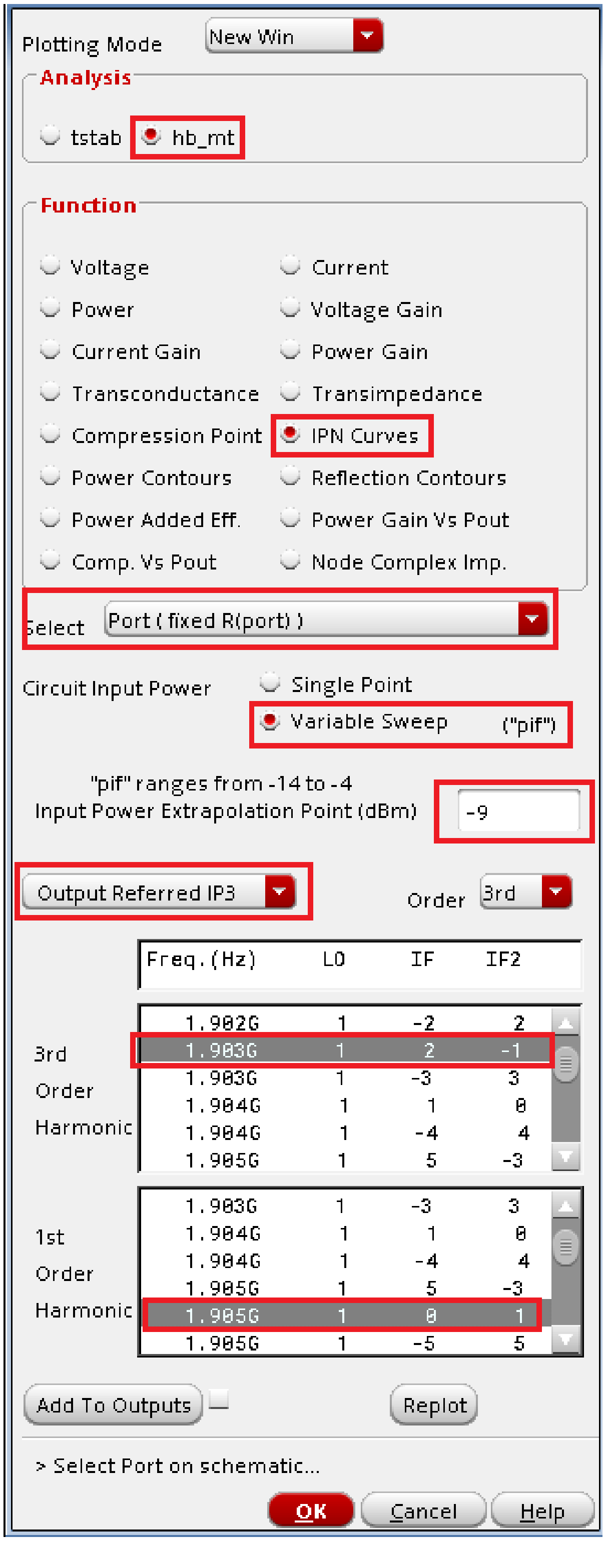
h. Similarly, you will need to choose the appropriate first order term. Note the selected *1.905G* entry. When adding the absolute value of the indices, it produces the lowest number.

Figure 4-167 Setting the Correct 1st Order Term when Plotting IP3



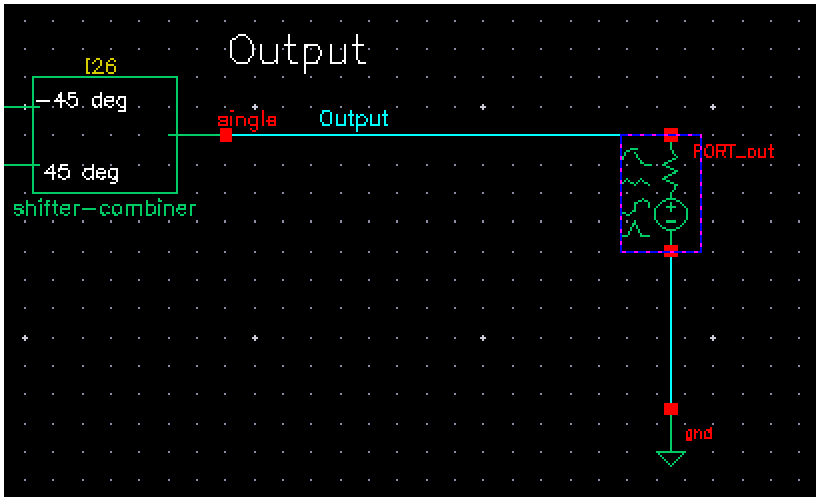
The *Direct Plot Form* should look like the following:

Figure 4-168 Direct Plot Form for Measuring IP3



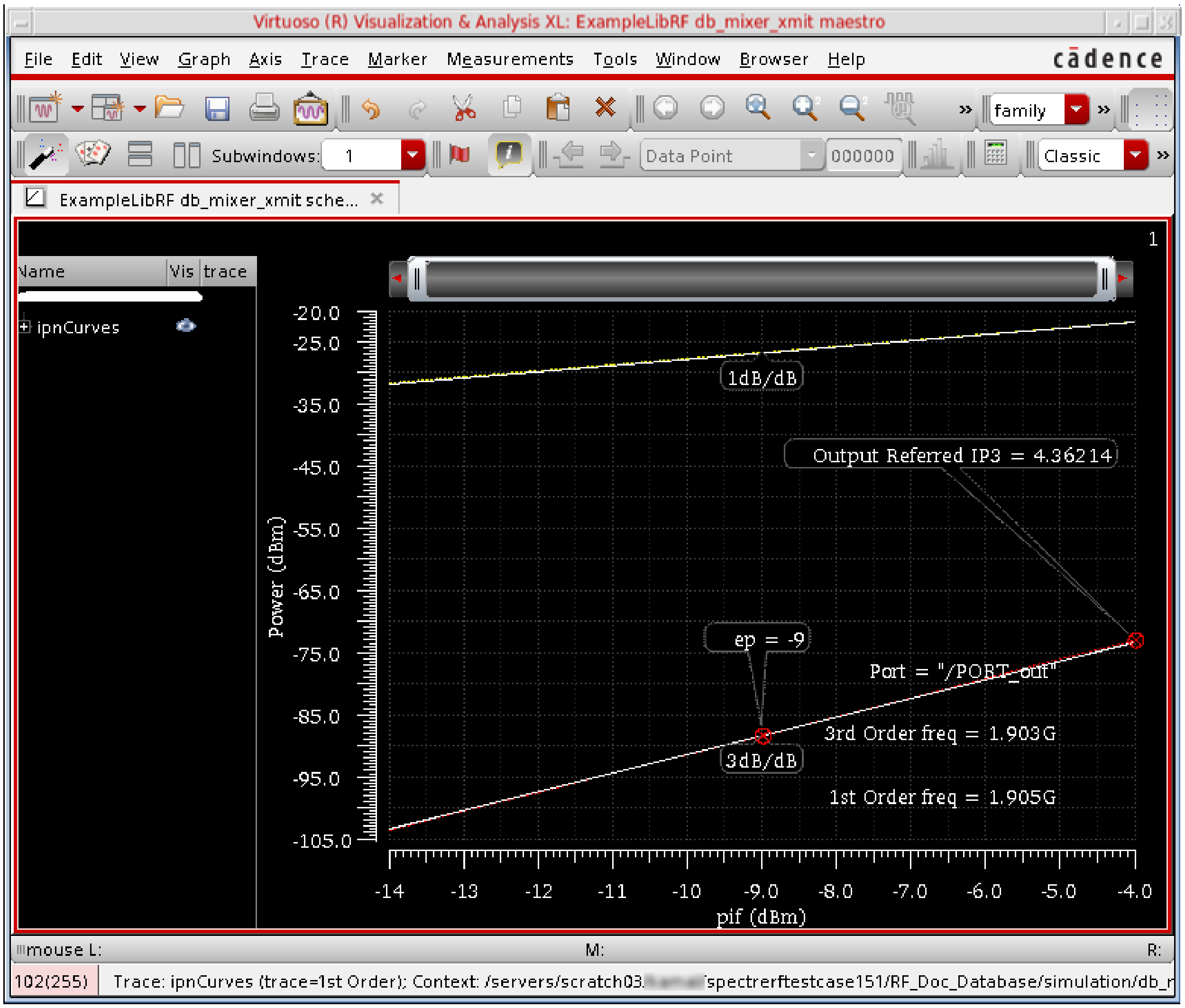
4. Select the *Output* port on the schematic.

Figure 4-169 output port



The IP3 plot appears. Note that you can move the Input Referred IP3 label by clicking and holding the right mouse button and dragging the label to another part of the screen. You may need to first click in the graphics to deselect the IP3 marker.

Figure 4-170 Output Referred IP3



The output referred IP3 is shown on the plot as *4.36dBm*. You can also plot input referred IP3 from the *Direct Plot Form* or read it directly off the x-axis.

Close ADE Explorer and exit the Schematic Window, as follows**:**

1. In ADE Explorer, choose *Session - Quit.*
2. In the schematic window, choose *File - Close All.*

Summary

In this section, you measured the three tone spectral content, image rejection, and output IP3 of a Transmit Mixer. In the next section, you will measure the Signal-to-Noise ratio of a Transmit Mixer.

Signal-to-Noise Ratio

The image reject mixer makes it possible to transmit one of the two sidebands (for example, f2+f1 or f2-f1). The only way to achieve image rejection in both legs is to have a +/- 45 degree phase difference on the inputs. Typically, the DSP (digital signal processor) produces the 45 degree phase difference. Since we are not able to measure the noise figure directly on this circuit, we calculate the signal to noise ratio (SNR) instead. The inputs are set in order to mimic a balun. We cannot use a shifter combiner on the input (it is an RC circuit) because it would be too narrowband.

Note: That there will be no image rejection for noise because noise is random and has uncorrelated phase. The only image rejection is on the signals 4MHz and 5MHz. The bandwidth of the signal that will be transmitted is approx 10MHz.

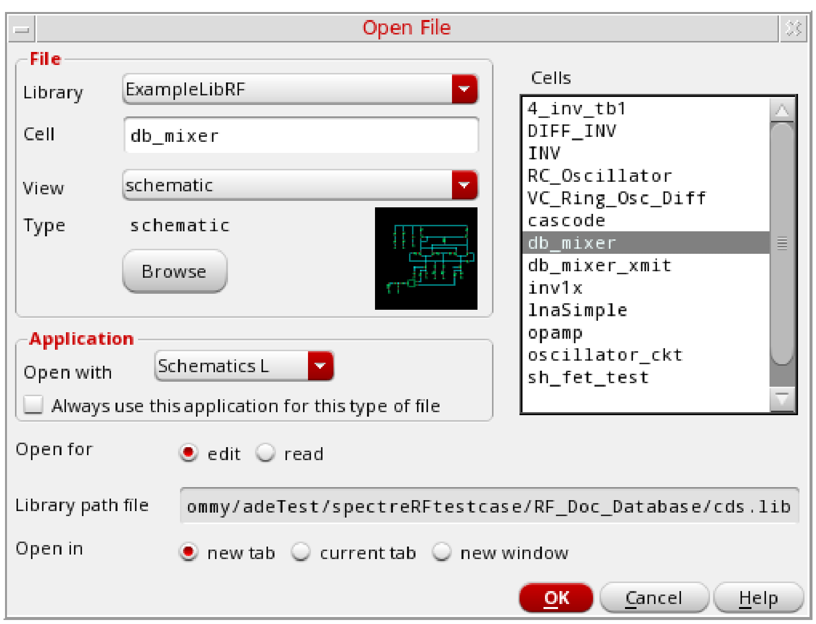
Open the db_mixer_xmit mixer circuit in the Schematic Window, as follows:

1. In the CIW, choose *File – Open*.

   The *Open File* form is displayed.
2. In the Open File form, choose *ExampleLibRF* from the *Library* drop-down list.
3. Choose *db_mixer_xmit* from the *Cells* list box.

   The completed *Open File* form looks like the following:

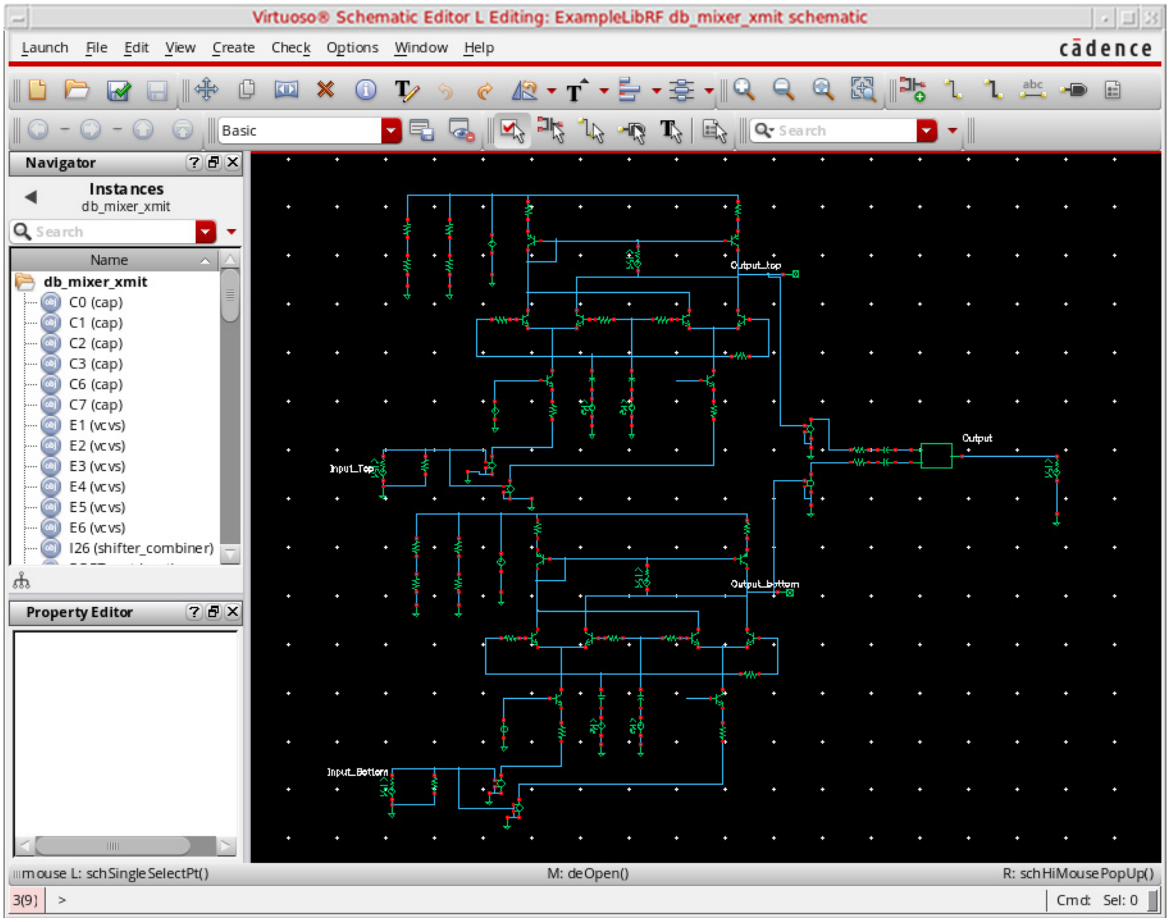
Figure 4-171 Open File Form



4. Click *OK*.

   The Schematic window for the *db_mixer_xmit* mixer is displayed.

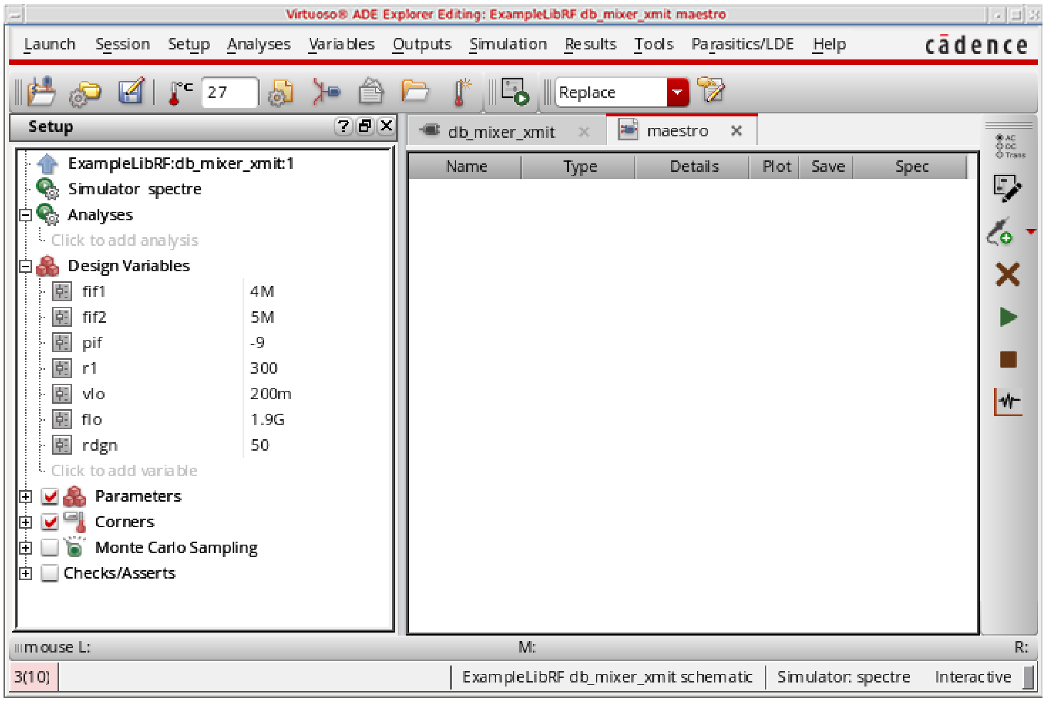
Figure 4-172 db_mixer_xmit Schematic



5. In the Schematic window, choose *Launch – ADE Explorer.*

   The Virtuoso Analog Design Environment Explorer window is displayed, as shown below.

Figure 4-173 Analog Design Environment Window



Choosing the simulator options, as follows:

1. In ADE Explorer, select *Setup – Simulator*.

   The *Choosing Simulator* is displayed.

2. Select *spectre* as the *Simulator*.

Figure 4-174 Choosing Simulator/Director/Host Form

   a. Click *OK*.

3. Set up the *High Performance Simulation Options*, as follows:

   In ADE Explorer, select *Setup - High Performance Simulation.* The *High Performance Simulation Options* window is displayed, as shown below.

Figure 4-175 High Performance Simulation Options

In the *High Performance Simulation* window, select *APS*. Note that *auto* is selected for *Multithreading options*. The effect of this is to detect the number of cores on the system (up to 16) and then multi-thread on all the available cores. The bigger the circuit, the more threads you should use. For a small circuit such as this, you may want to set the number of threads to 2. Using 16 threads on a small circuit might actually slow things down because of the overhead associated with multithreading.

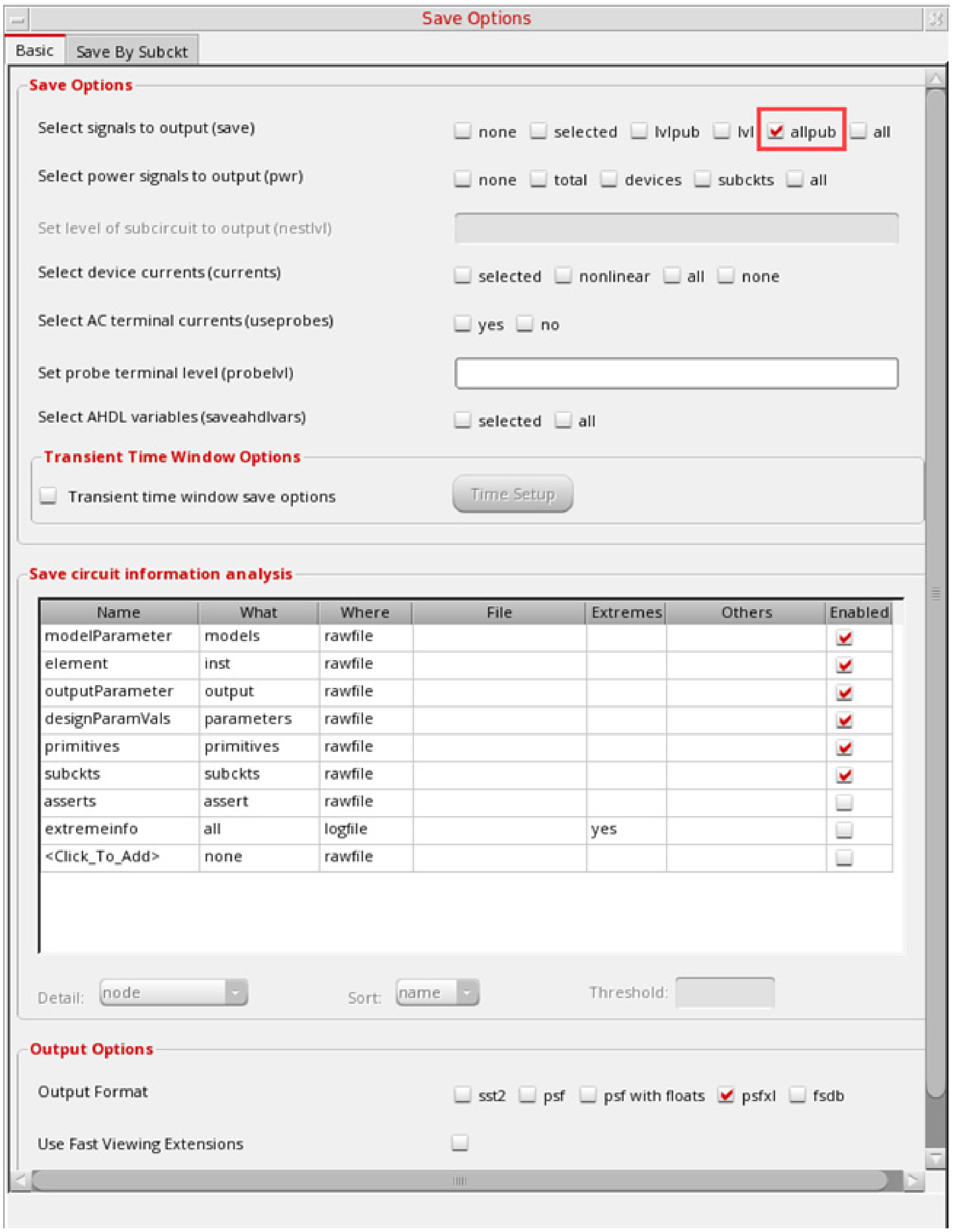
Click *OK*.

4. Select *Outputs – Save All.*

   The *Save Options* form is displayed.

5. In the *Select signals to output* section, ensure that *allpub* is selected.

Figure 4-176 Save Options Form



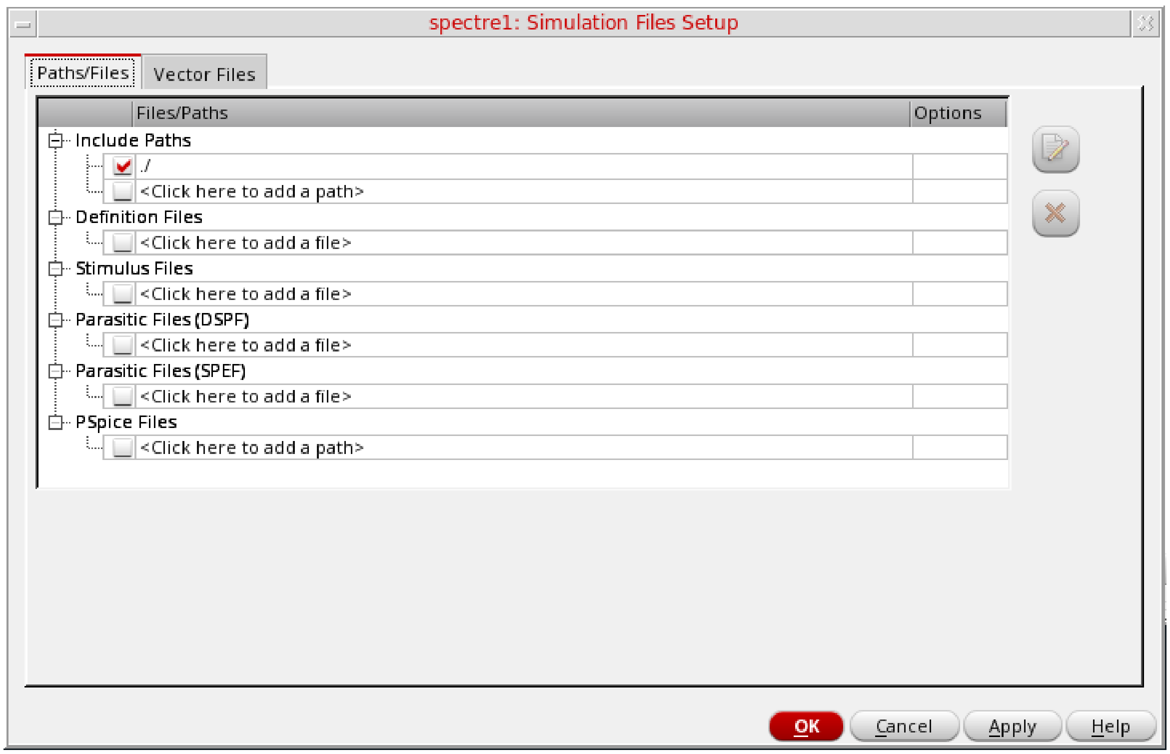
This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

6. Click *OK*.

Set up the model libraries, as follows:

1. Select *Setup - Simulation Files.* The Simulation Files Setup form is displayed, as shown below.

Figure 4-177 Simulation Files Setup Form



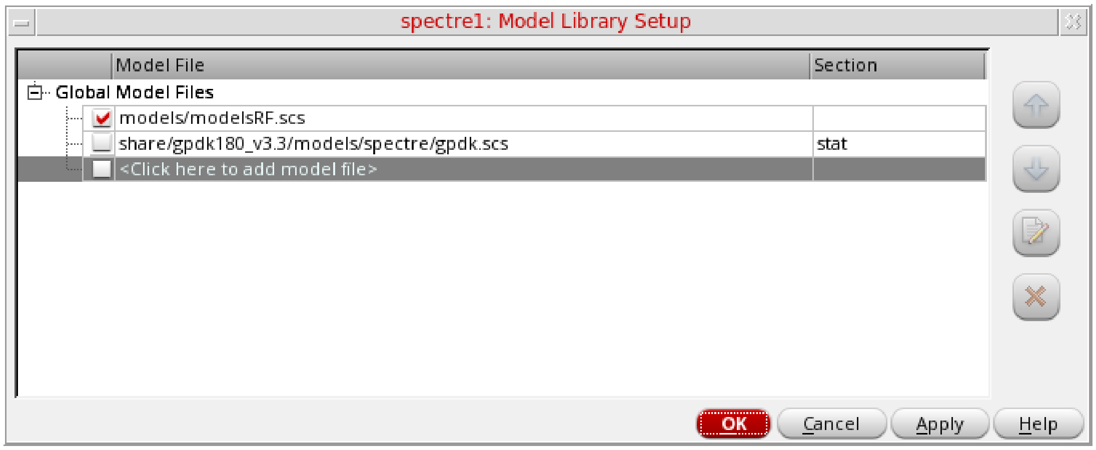
2. Ensure that the *Include Path* is set as shown above.
3. Select *Setup – Model Libraries.*

   The Model Library Setup form is displayed.
4. In the *Model Library File* field, type the following for the name of the model file:

   `models/modelsRF.scs`
5. Click *OK*.

   The *Model Library Setup* form looks like the following.

Figure 4-178 Model Library Setup



Alternately, you can click on the *Browse* button and browse to the `modelsRF.scs` model file.

6. Make sure that the *Model File* name is selected.
7. Click *OK*.

Setting Up For Calculating Signal to Noise Ratio

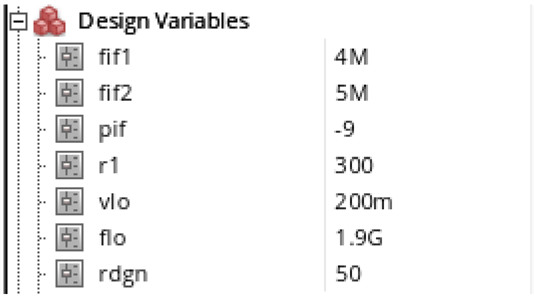
To calculate signal to noise ratio, you first need to calculate the signal using a three-tone hb analysis. You will take the signal value at 1.904G (alternately, you could look at 1.905G) and note that value for later. Then, you will set up a pnoise analysis.

Three Tone HB Analysis Setup

Setting the Design Variables

1. Ensure that your settings look like the following, in the Design Variables section of ADE Explorer:

Figure 4-179 Design Variables Section of ADE Explorer Window



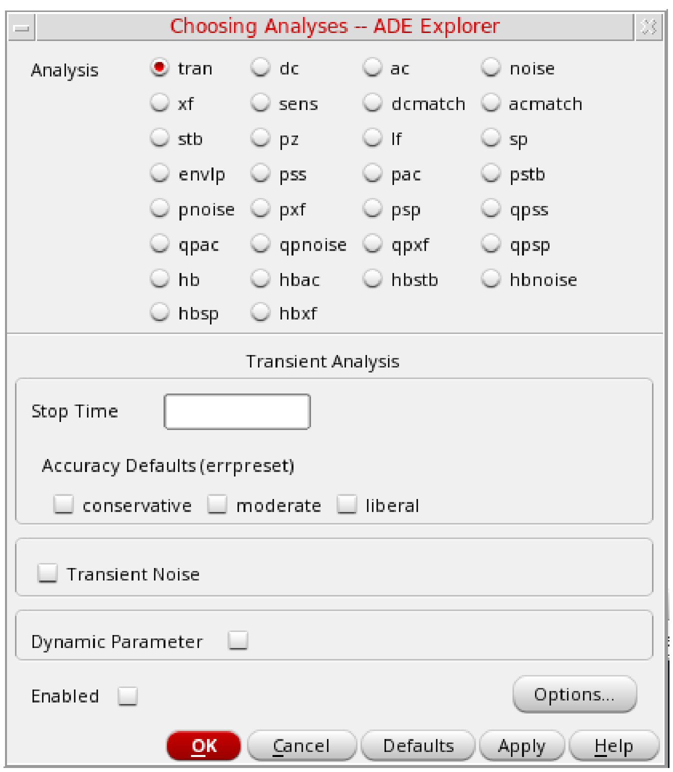
Setting Up the HB Analysis

1. In ADE Explorer, select *Analyses – Choose*. You can also select the *Choosing Analyses* icon ( ) at the right of the ADE Explorer window.

   The *Choosing Analyses* form is displayed, as shown blow.

Figure 4-180 The Choosing Analyses Form

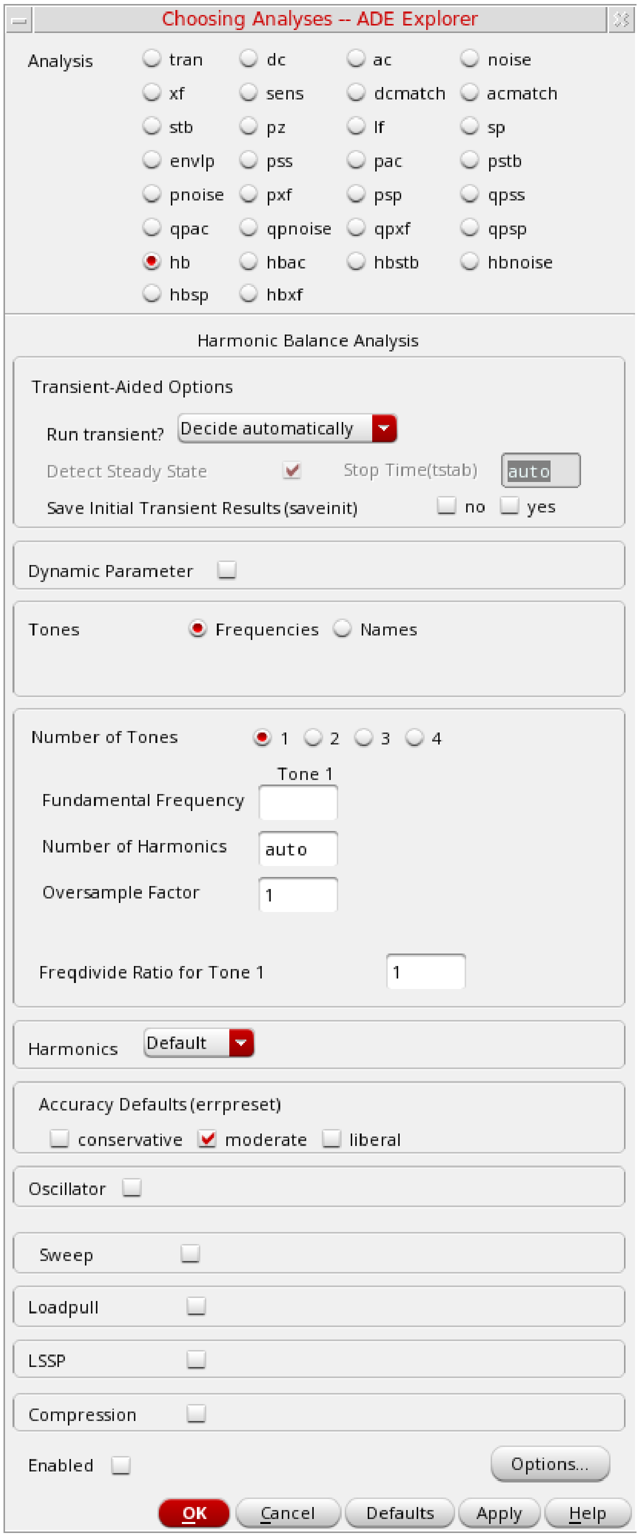
.



2. In the *Choosing Analyses form*, select *hb*. The form expands. Since the circuit is mostly sinusoidal (not strongly nonlinear) Harmonic Balance is the appropriate analysis to choose.

Figure 4-181 The HB Choosing Analyses Form

.



Harmonic balance can set the harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. To enable this, select *Decide automatically* or *Yes* for the *Run Transient* selection in the *Transient-Aided Options* section. This single action will cause a transient analysis to be run until steady-state is detected, and then from the transient analysis, the number of harmonics for *Tone1 (*when *Frequencies* is selected) or for the tone that has *tstab* enabled (when *Names* is selected).

3. In the Transient-Aided Options section of the form, select the following:

   a. For *Run transient?* select *Decide automatically*.

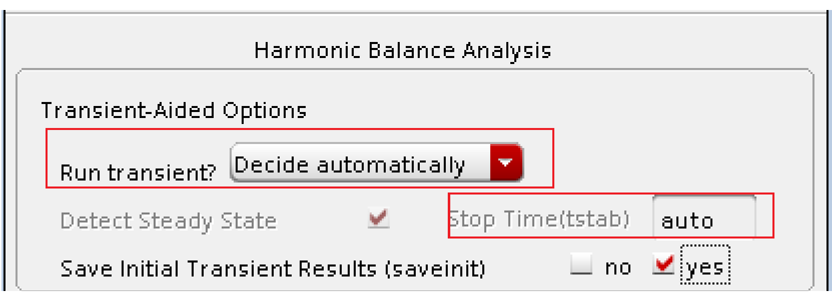
   b. For *Stop time (tstab)*, *auto* is automatically populated in the field.

   c. For *Save Initial Transient Results (saveinit),* select *yes*.

   During the transient-assisted HB simulation, a transient simulation runs before the frequency domain iteration of harmonic balance. Only the LO signal is “on” during the *tstab* transient. At the end of the *tstab*, an FFT is run and its result is used as the starting point for the frequency domain iterations

   All the signals are applied and the simulation is done in the frequency domain. Only the signals and the mixing products are calculated by hb.

Figure 4-182 Transient Assisted Harmonic Balance



4. In the *Tones* section, select *Names*. When *Names* is selected, the *Tones* portion of the form expands. All the sources in the top-level schematic are read into the form automatically.

5. Select one of the LO sources in the Tones section. You can use hb with up to four signals present in the circuit. In this circuit, there are three tones, the LO and two RF tones. The four LO signals have the same name, the same frequency, and are considered a single tone. Whenever you have two signals at the same frequency, make sure you set the *Frequency Name 1* or *Frequency Name 2* property on the source to the same name as it was done in this example. When *Tones* is set to *Names*, the simulator considers both of the sources as a single frequency.

Because you are using auto-tstab, *Tstab* is set to *yes* by default in the *Tones* section for the LO tone. The signal with `tstab=yes` is the signal that is used for transient-assisted harmonic balance. Only one signal can have transient assist, that being the signal with *tstab* set to *yes*.

Note: *tstab* is set to *yes* on the signal that causes the largest amount of distortion in the system. (In this example, that is the LO tone).

During the tstab interval, a transient analysis is run before the frequency domain iteration of harmonic balance. At the end of the tstab, an FFT is run and its result is used as the starting point for the frequency domain iterations.

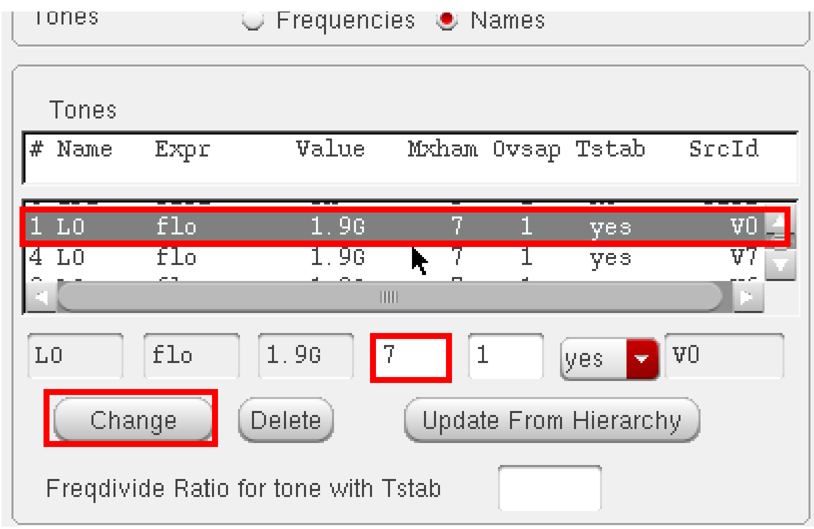
Then all the signals are applied and the simulation is done in the frequency domain. Just the signals and the mixing products are calculated by hb.

6. Because you are not using the auto-harmonics feature, you need to change the *Number of harmonics* on the LO tone to *7*. Place the cursor in the field under *Mxham* and type *7*. Verify that *tstab=yes* on the LO tone. Click the *Change* button. Note that all four LO sources are now updated with a *`Mxham`* of *7*.

   Typically, you want to use the auto-harmonic feature (type *auto* into this field).You will be using the manual method in this example to show how this is done. Leave *Ovsap* (oversample) set to the default value of *1*.

7. Click the *Change* button when finished. The form updates. Note that the LO tones now have *Mxham=7* and *Tstab=yes*.

Figure 4-183 Tones Section of hb Choosing Analyses Form



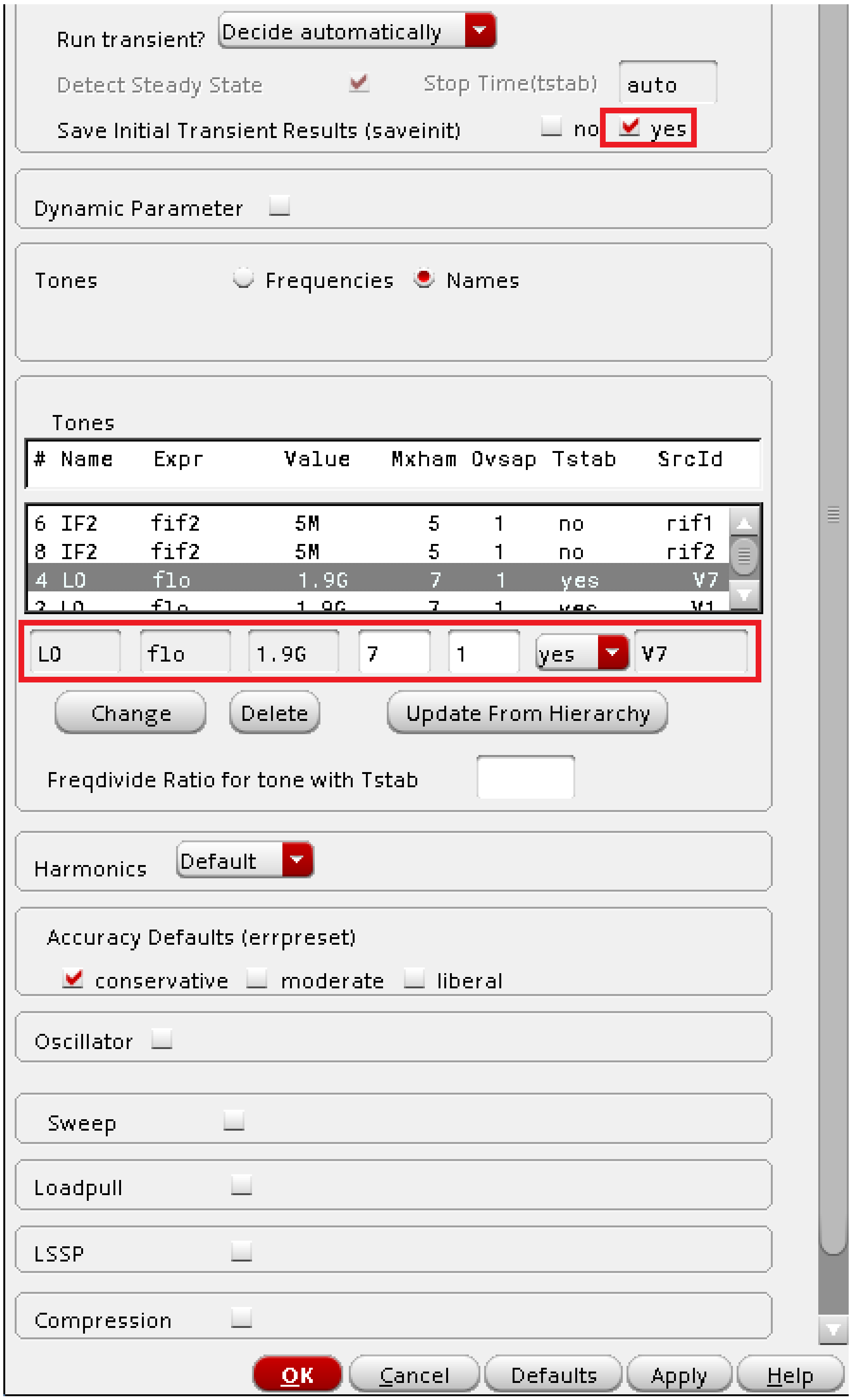
8. Select the IF tone in the *Tones* list box. Change the *Mxham* setting on both *IF1* and *IF2* tones from the default value of *3* to *5*. In the *Tones* section, select the line containing one

of the IF tones. For this measurement three IF harmonics is not enough. You will need to increase the number of IF tones for greater accuracy.

9. Set *Accuracy Defaults (errpreset)* to *conservative*.
10. Leave the rest of the form set to the default values.

    The *Choosing Analyses* form should look like the following:

Figure 4-184 HB Choosing Analyses Form



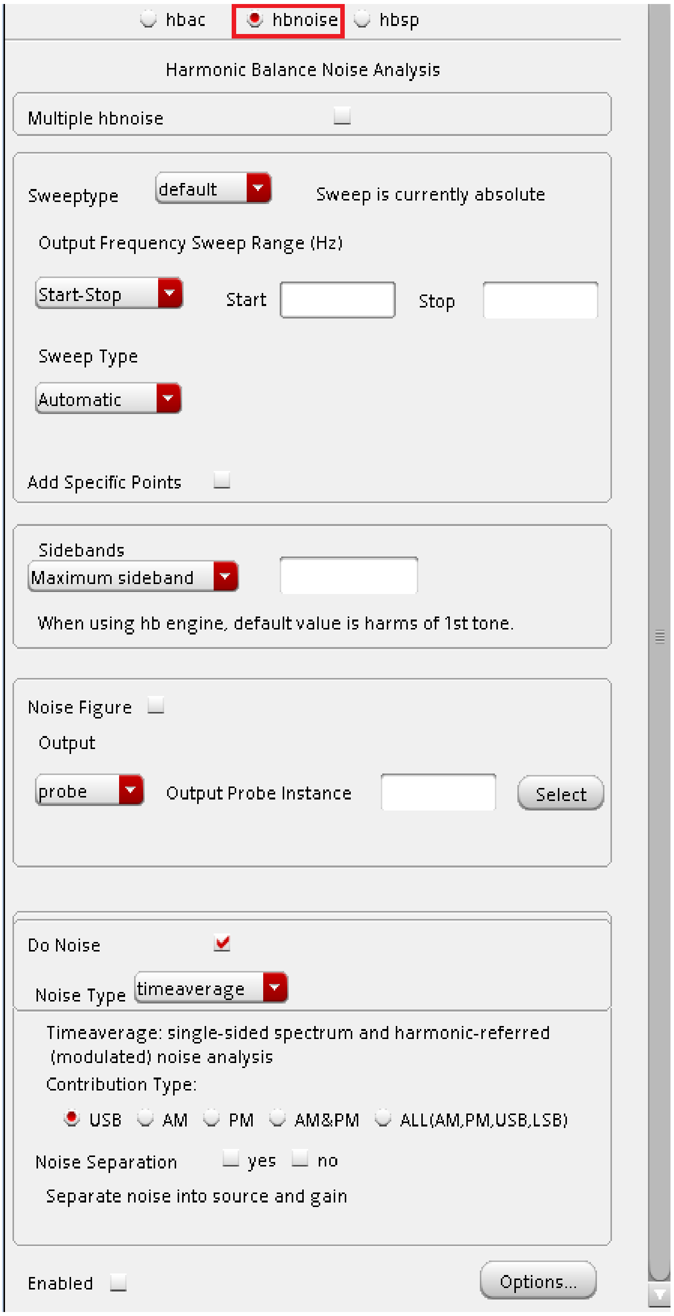
11. Click *Apply* at the bottom of the *Choosing Analyses* form.

Setting Up the HBnoise Analysis

Set up the HB Noise simulation. You will be running an hbnoise simulation with the rf tones turned on.

12. In the *Analysis* section select *hbnoise*. The form changes, as shown below.

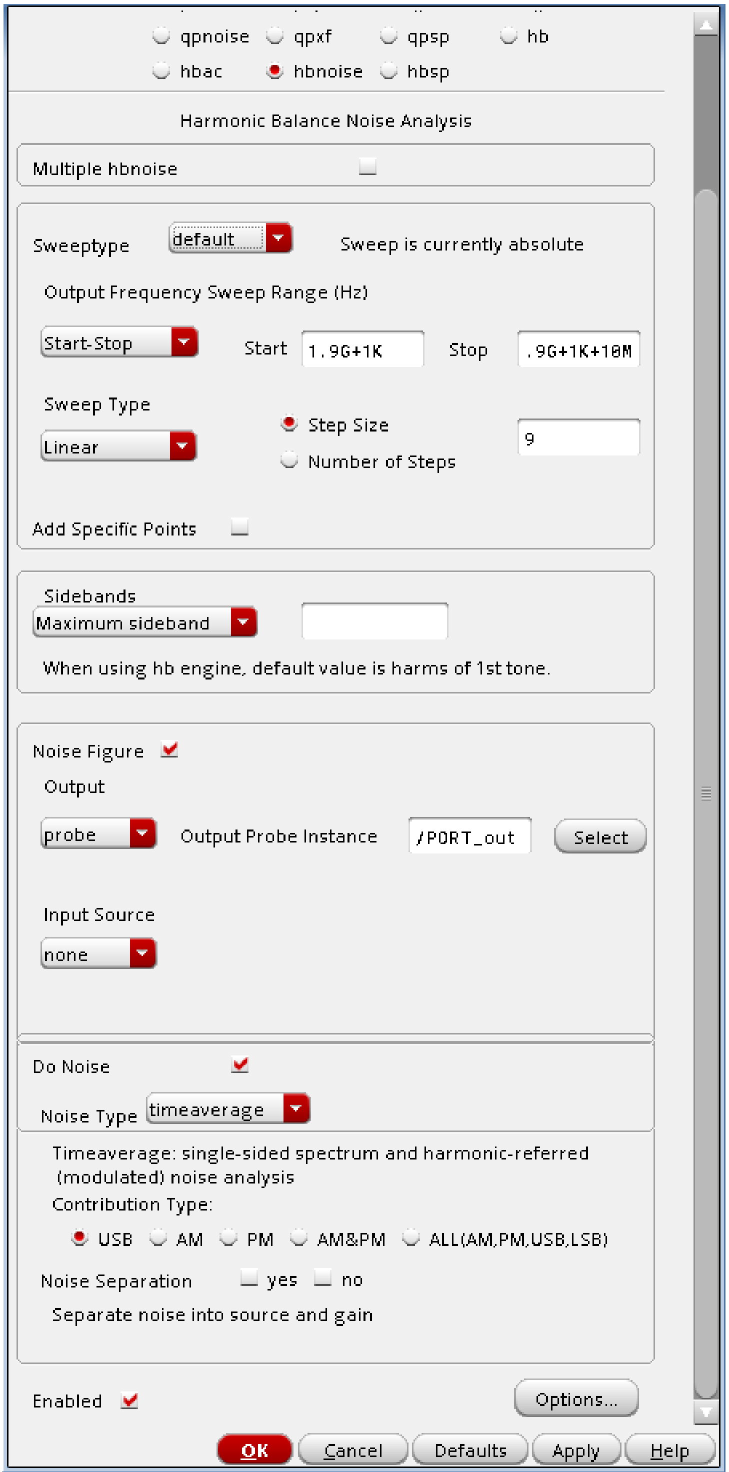
Figure 4-185 hbnoise Choosing Analyses Form



13. The default *Sweeptype* should be left at *default*.
14. In the *Output Frequency Sweep Range (Hz)* section, type `1.9G+1K` in the *Start* field and type `1.9G+1K+10M` in the *Stop* field.
15. In the *Sweep Type* section, select *Linear*. Type `9` in the *Number of Steps* field. Choose a number which does not allow the simulator to take a step at any of hb harmonics to avoid getting *infinite flicker noise* warnings.
16. Leave the *Maximum sideband* field blank. In HBnoise, this value should either match the number of *harmonics* in HB analysis, or be left blank, in which case it uses the number of *harmonics* in HB.
17. Select the *Noise Figure* option.
18. In the *Output* section, select *probe*. Click *Select* to the right of the *Output Probe Instance* field In the schematic, click the source to the left of the *Output* label. Alternately, you can type `/PORT_out` in the *Output Probe Instance* field.
19. For *Input Source*, select *none*. You only need to measure output noise and in this setup, you cannot measure noise figure.

    Hbnoise analysis fundamentally calculates the output-referred noise, and it is the output frequency range that is specified in the hbnoise *Choosing Analyses* form. Mixing can occur with harmonics of all the input frequencies, and to get this, leave the *Maximum sideband* field blank. The *Choosing Analyses* form should look like the following:

Figure 4-186 hbnoise Choosing Analyses form for Output Noise Measurement



Running the Simulation and Plotting the Results

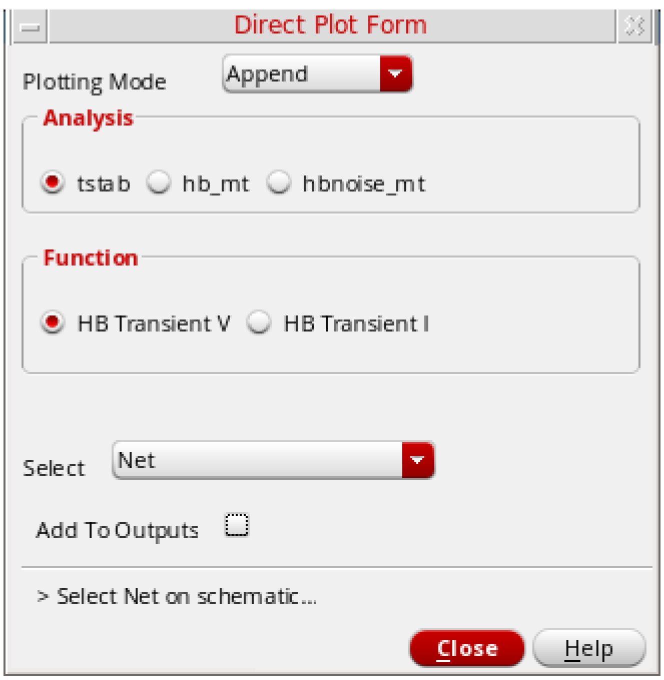
1. Start the analyses by clicking the green arrow icon. in ADE Explorer or in the Schematic Editor.

   This netlists the design and runs the simulation. A SpectreRF status window appears (`spectre.out` log file). When the analysis has completed, you may iconify the status window.

2. When the simulation has finished, select *Results - Direct Plot - Main Form* in ADE Explorer. The *Direct Plot Form* is displayed, as shown below.

   Figure 4-187 Harmonic Balance Direct Plot Form

   .



   In the *Direct Plot Form*, you will notice that there are three available analyses: *tstab*, *hb_mt*, and *hbnoise*. The transient-assisted results are accessible by selecting the *tstab* analysis. The hb results are accessible by selecting the *hb_mt* analysis. Note that when there is more than one tone, you will see *hb_mt* (signifying harmonic balance multi-tone). A single tone simulation will show *hb*. The hb results are accessed by selecting the *hbnoise* analysis.

3. In the *Analysis* section, select *hb_mt*. The *Direct Plot Form* expands.

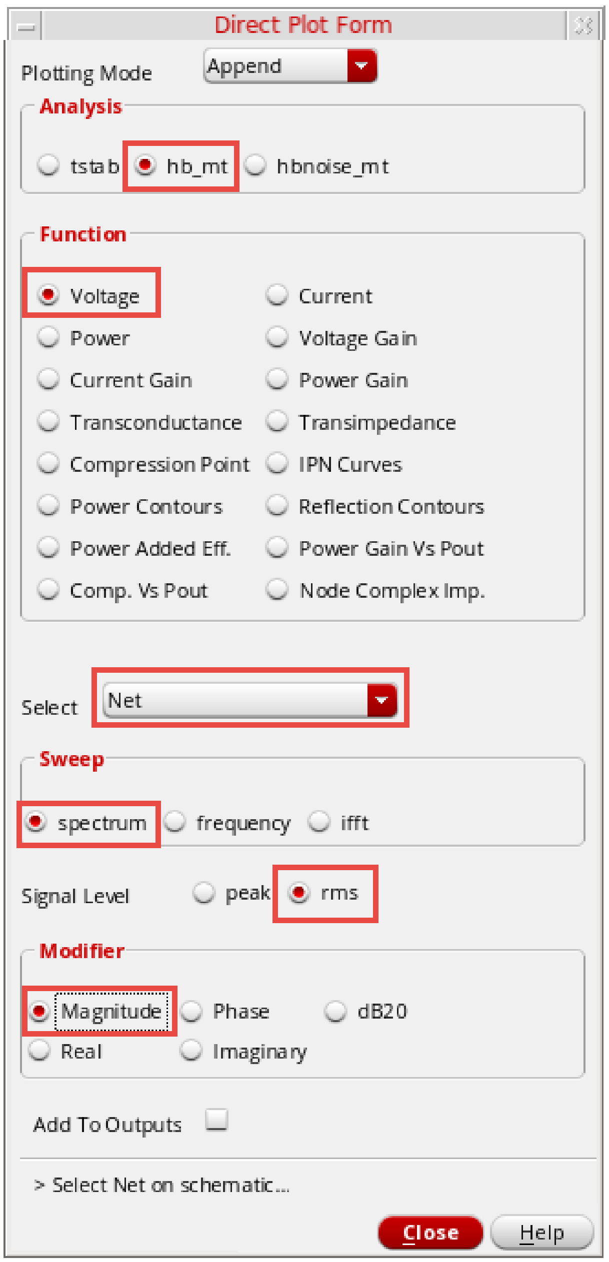
4. In the *Function* section, verify that *Voltage* is selected.

5. Select *Net.*

6. In the *Sweep* Section, select *spectrum*.
7. In the *Signal Level* section, select *rms* .
8. In the *Modifier* section, select *Magnitude*.

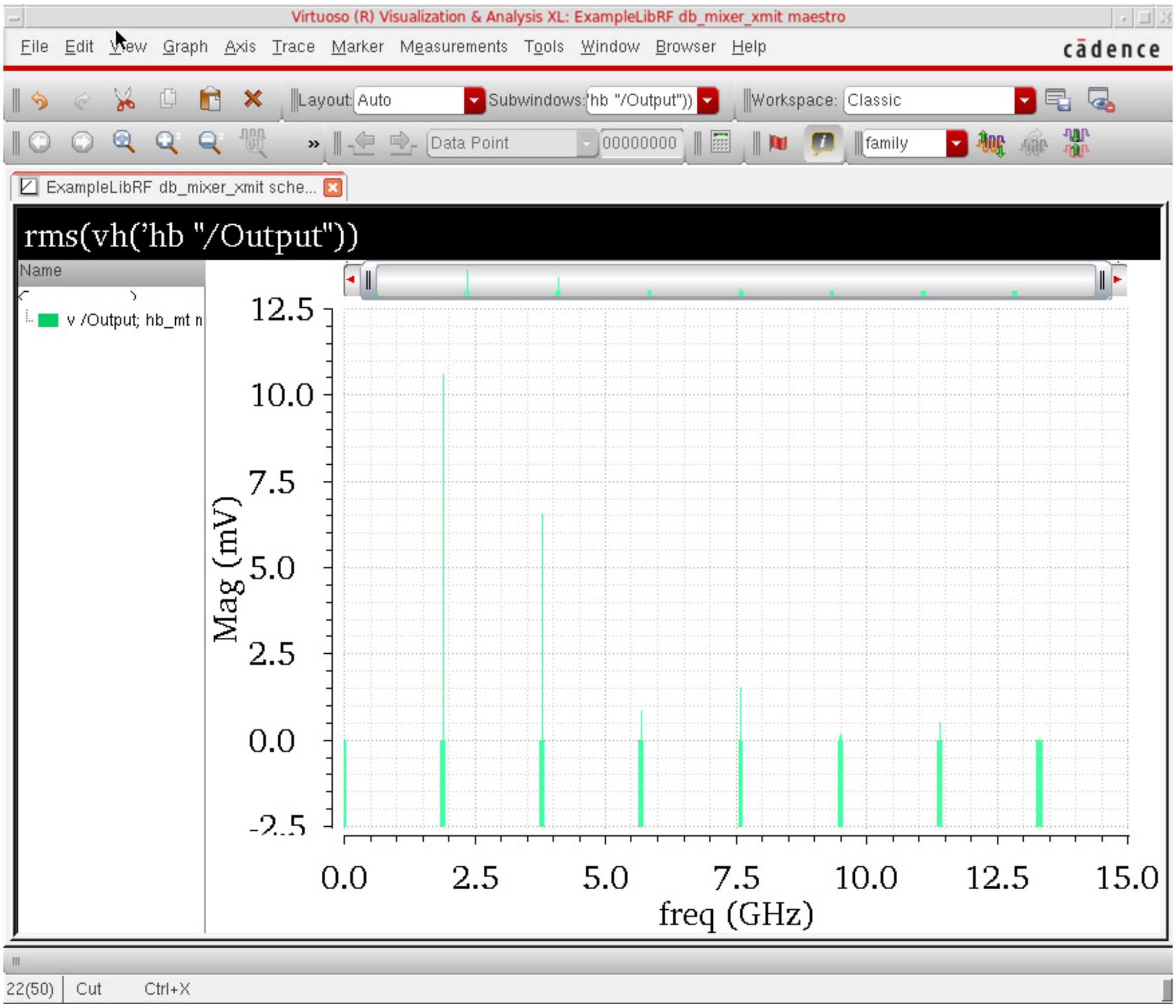
   The *Direct Plot Form* should look like the following:

Figure 4-188 Direct Plot Form



9. Click on the net labeled *Output* on the right side of the schematic.
10. The plot of Voltage Spectrum vs Frequency is displayed in the waveform window, as shown below.

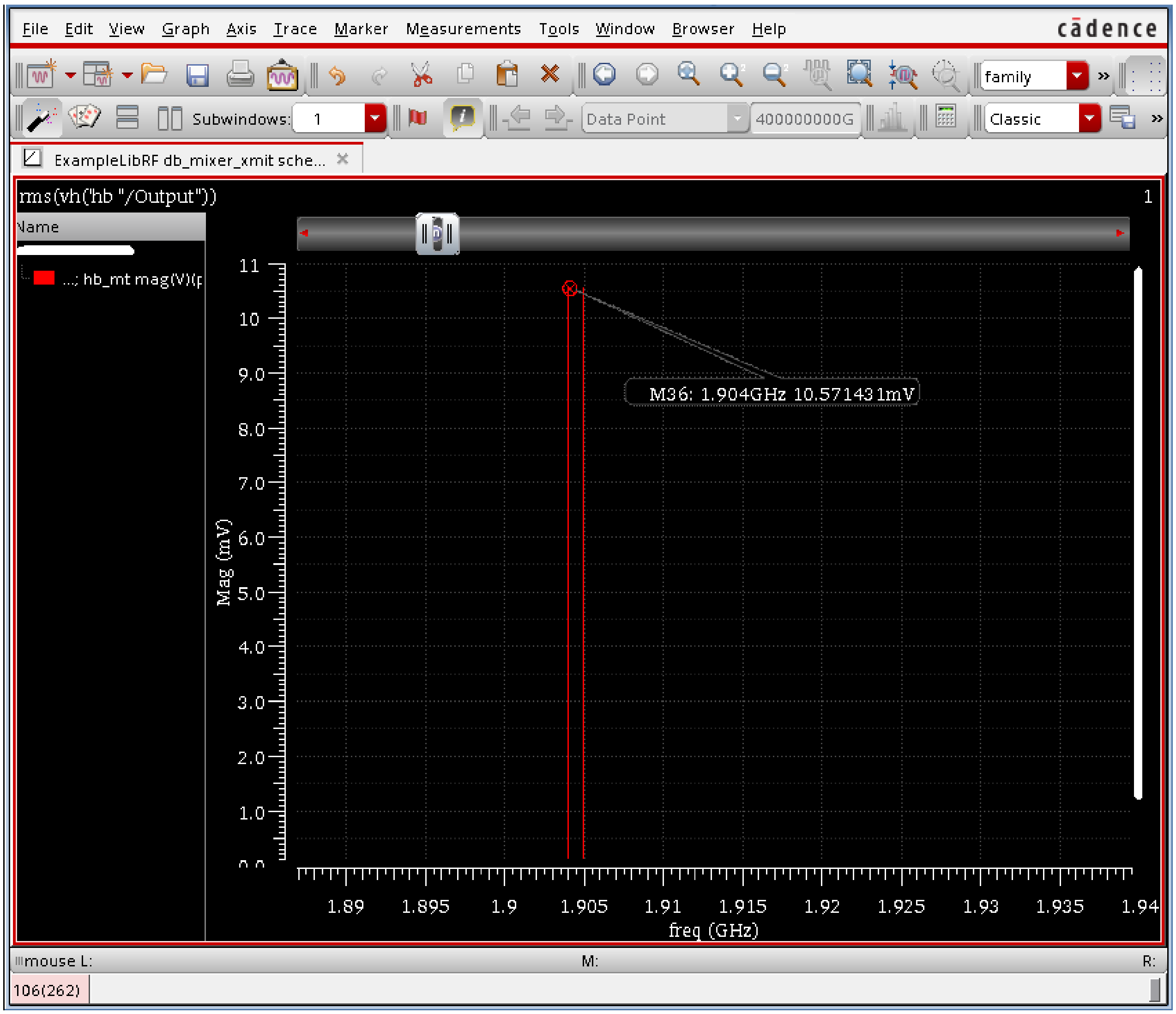
Figure 4-189 RMS Voltage vs Frequency



11. Zoom into the region of the graph near 1.9GHz. Press and hold the right mouse button while moving the cursor to draw a box around the signals near 1.9GHz. When you release the mouse, you are zoomed in. You may need to do this several times to get zoomed in to just the area around 1.9GHz.
12. Place a marker at 1.904GHz (or 1.905GHz). Place the mouse over the 1.904G trace at the top and select the bindKey `m` to place the marker. The rms voltage at 9.104G is

10.571431mV, as shown below. This is the *signal* part of the Signal-to-Noise ratio you will be calculating in the next step.

Figure 4-190 Zoomed in RMS Voltage vs Frequency

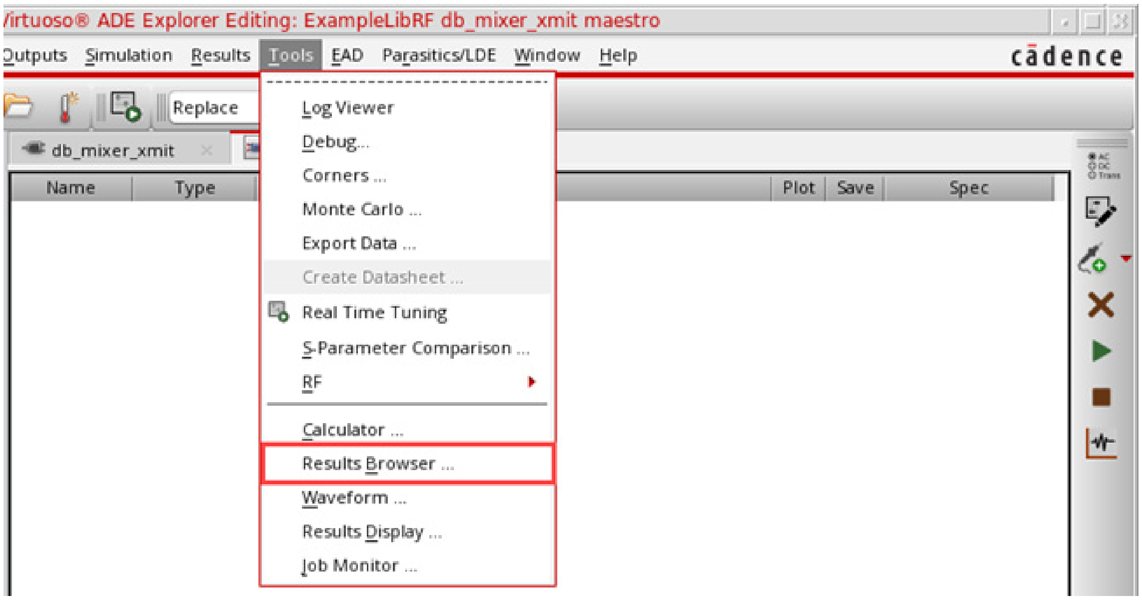


Plotting the Noise Results

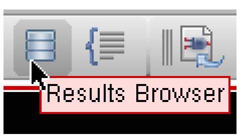
1. When the simulation has finished, plot the *Voltage Spectrum* in *V/sqrt(Hz)* from the Results Browser.

   ADE Explorer, select *Tools - Results Browser.*

Figure 4-191 Invoking the Results Browser After Simulation

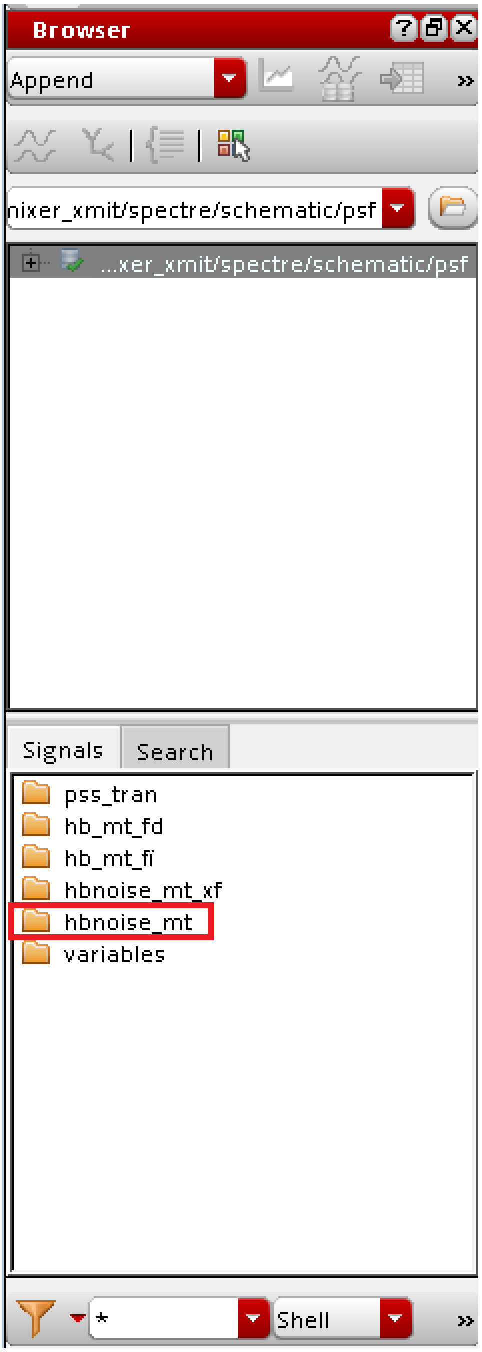


Alternately, you can click the Results Browser icon in the Schematic, as shown below.



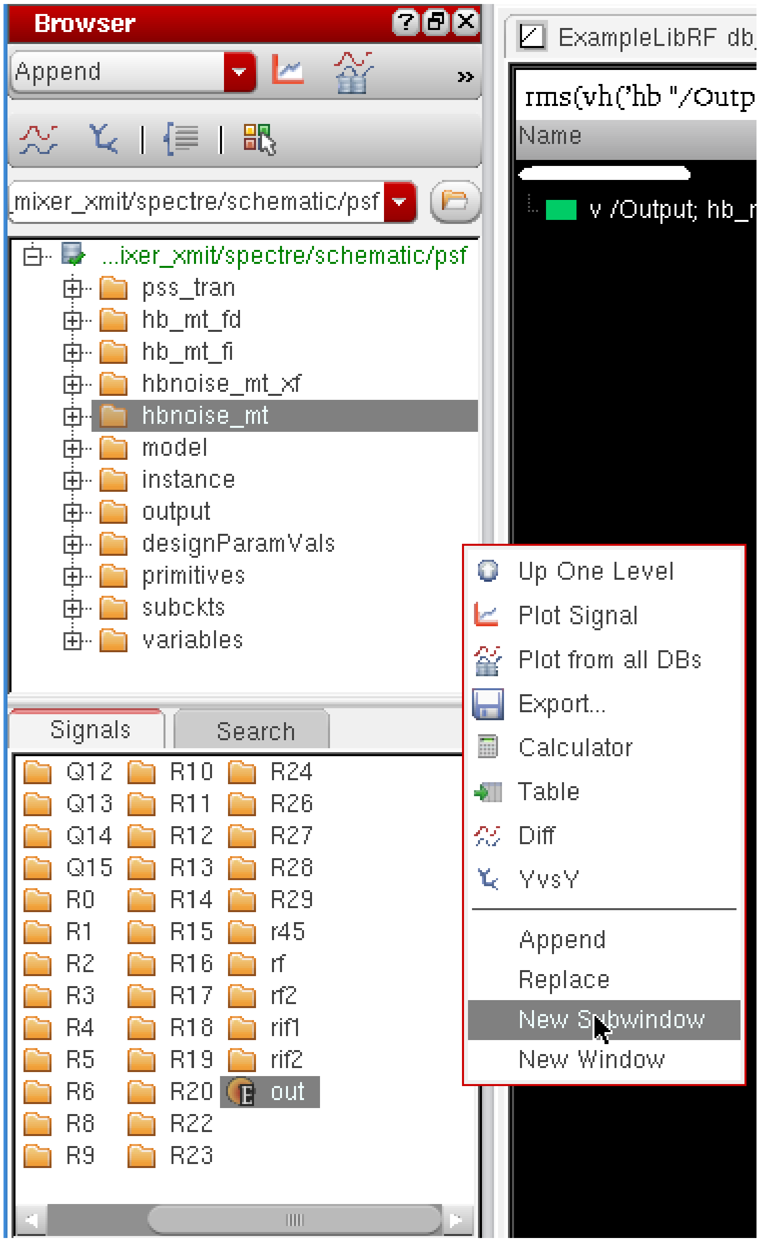
2. In the Results Browser, select *hbnoise_mt*.

Figure 4-192 Selecting hbnoise results using the Results Browser



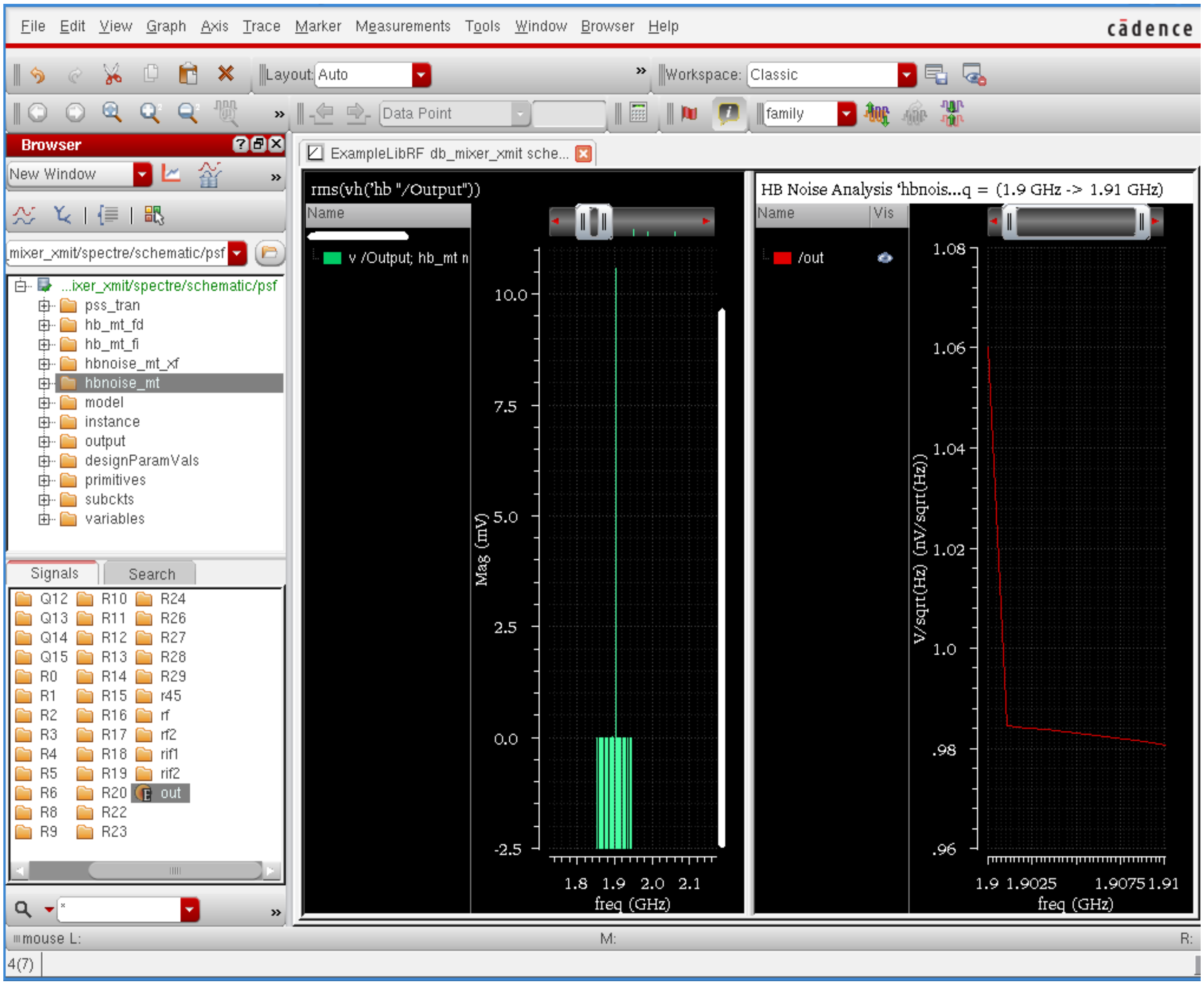
3. Double-click on the *hbnoise_mt* folder shown highlighted in the above figure. The folder expands. Click the right mouse button on the *out* signal, and select *Plot Signal* from the context menu, as shown below.

Figure 4-193 Selecting out Net in the Results Browser



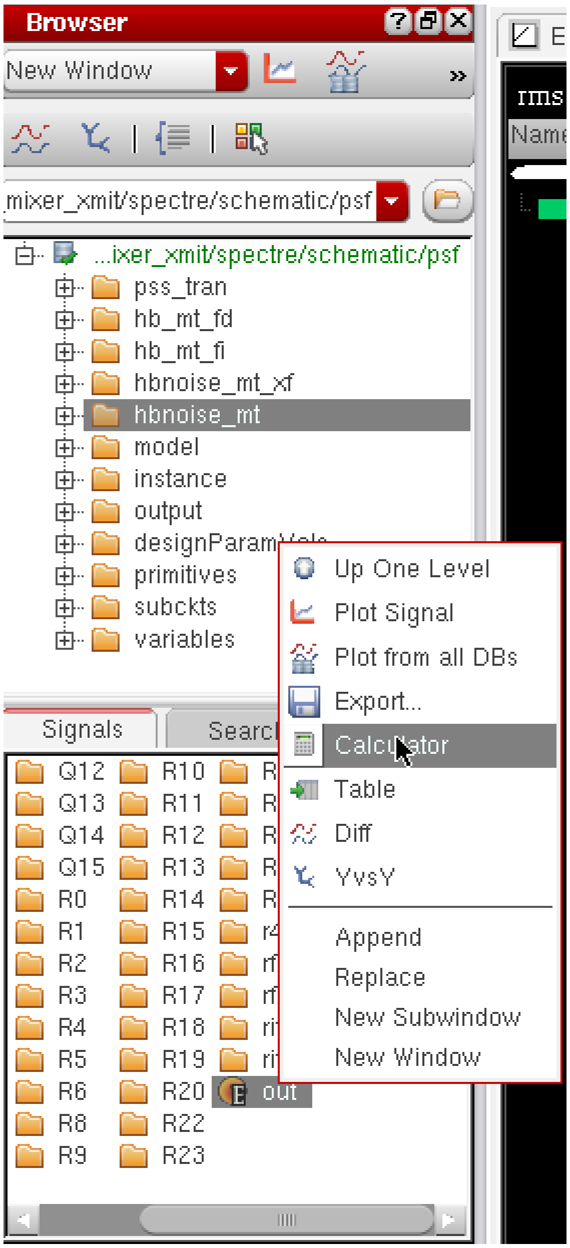
4. Select and release the mouse over the *New Subwindow*. The Noise is plotted in *V/ sqrt(Hz),* as shown below.

Figure 4-194 Noise Results



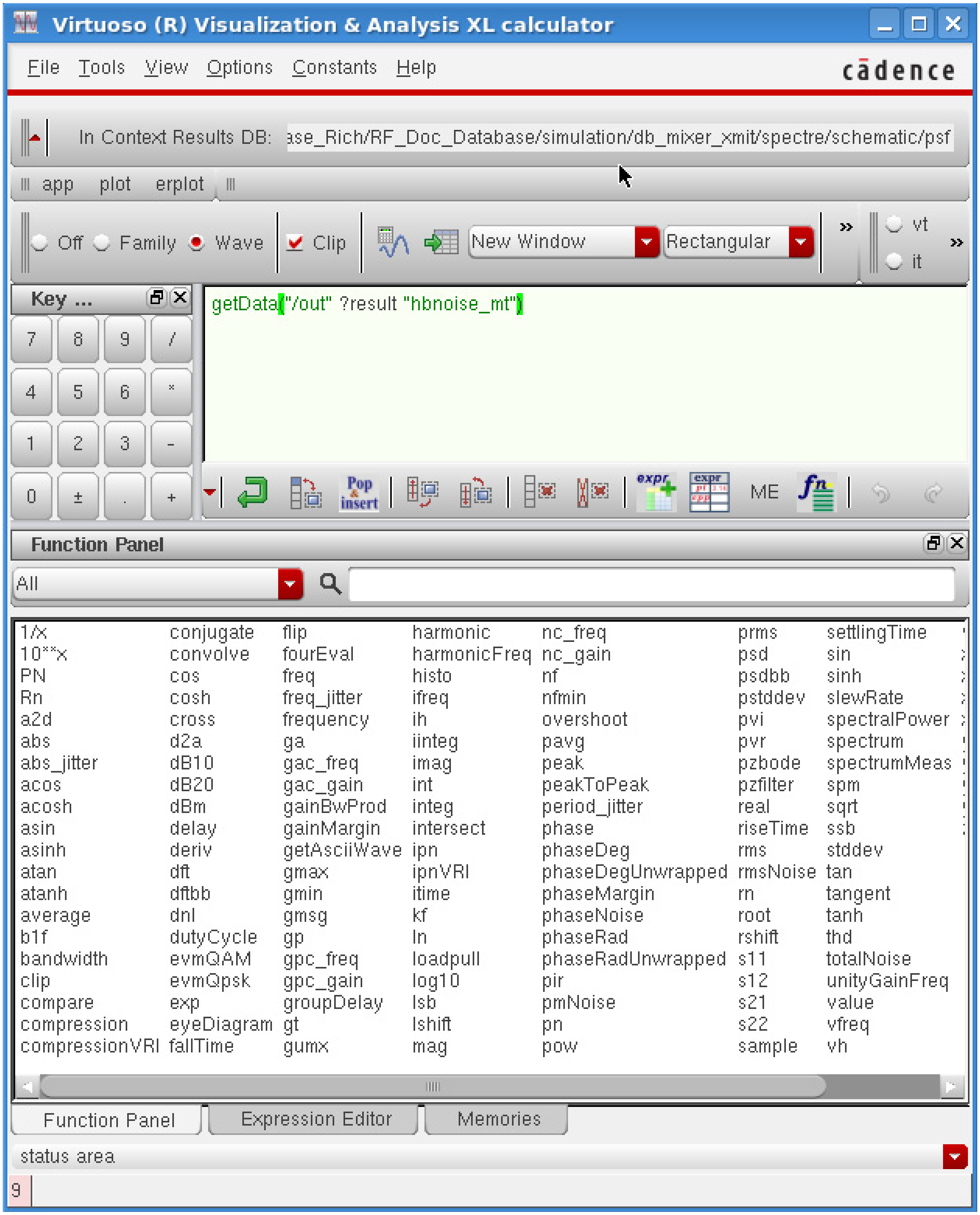
5. Note that the units are in *V/sqrt(Hz)*. You want to calculate the noise voltage in a 1Hz chunk of the spectrum. To do this, open the Calculator by clicking the right mouse button on the *out* signal in the Results Browser and selecting *Calculator*.

Figure 4-195 Sending Output Waveform to Calculator



The information is added in the Calculator Buffer, as shown below:

Figure 4-196 Output Noise Data Entered into Calculator Buffer

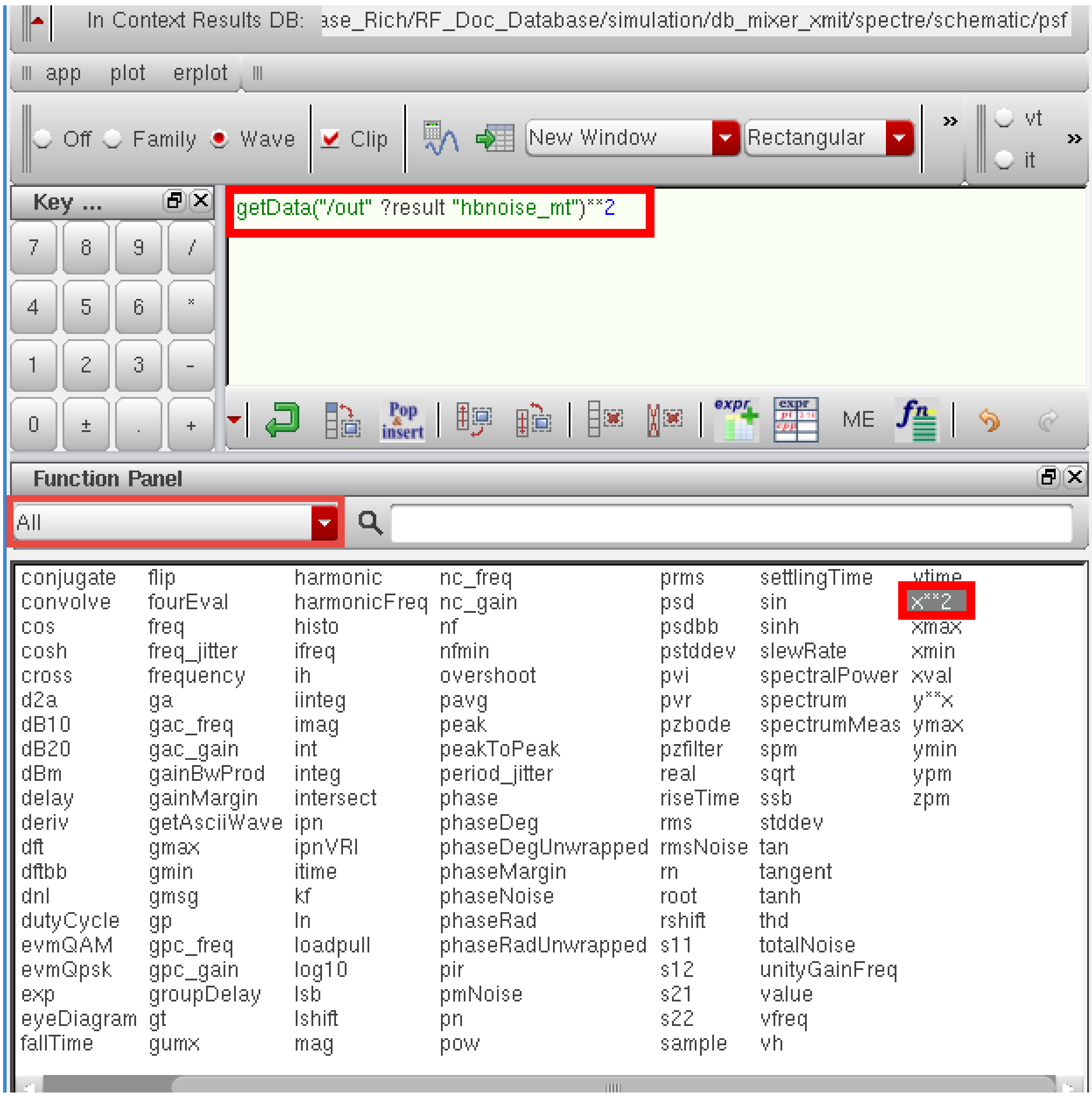


6. You want to look at the Noise Voltage over a 1 Hz bandwidth.

In the Calculator you need to square the voltage to get it in units of power. `P=V`^{2}`/R`, where R is some arbitrary value.

In the lower part of the Calculator, in the *Function Panel* section, ensure that *All* is selected from the drop-down and select *x**2*. The Calculator Buffer updates, as shown below.

Figure 4-197 Squaring the Noise Voltage

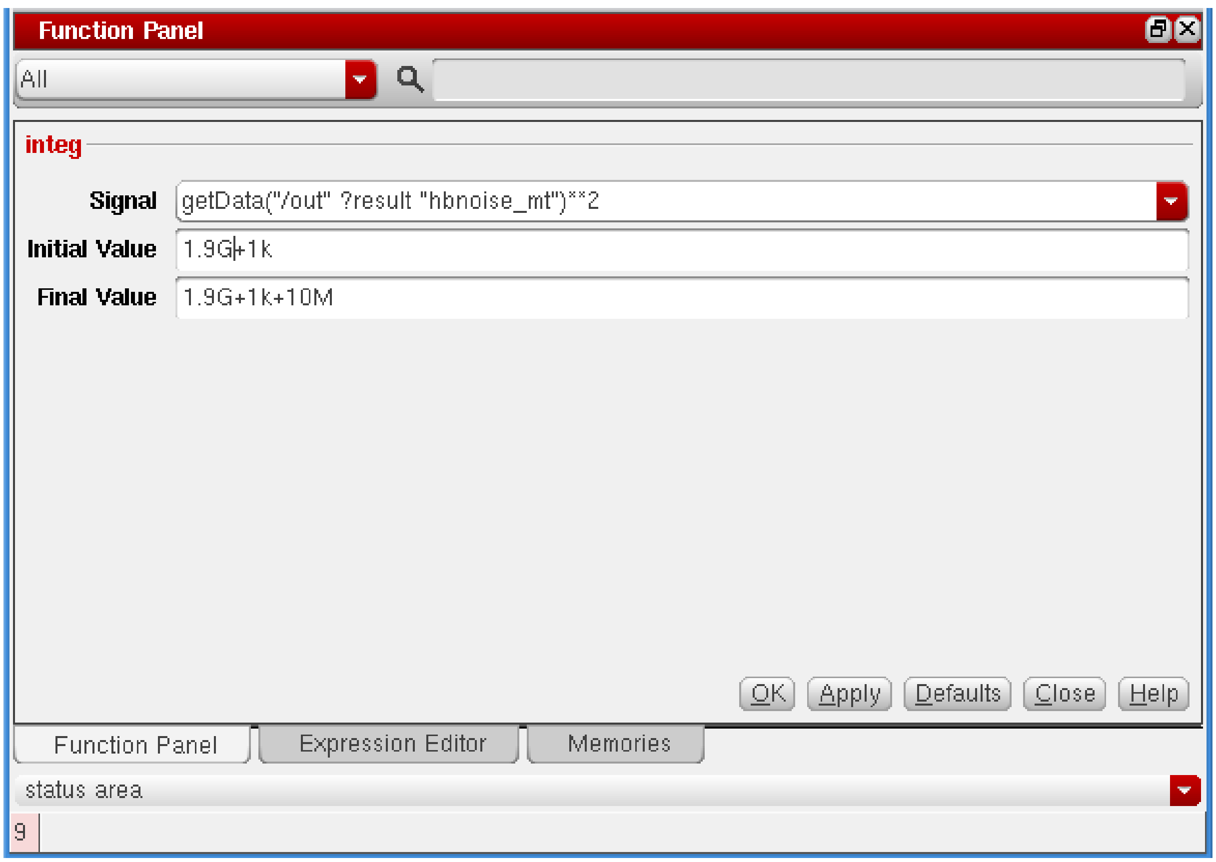


7. Next, you need to integrate the power over bandwidth. Select the *integ* function from the *Function Panel*. The form changes. You are integrating the power from `1K` to `10M` (these are the same values that were in the hbnoise *Choosing Analyses* form).

8. Verify that the signal is automatically populated in the *Signal* field*.* If it is not auto-populated, enter `1.9G+1K` as the *Initial Value* and `1.9G+1K+10M` as the *Final Value*.

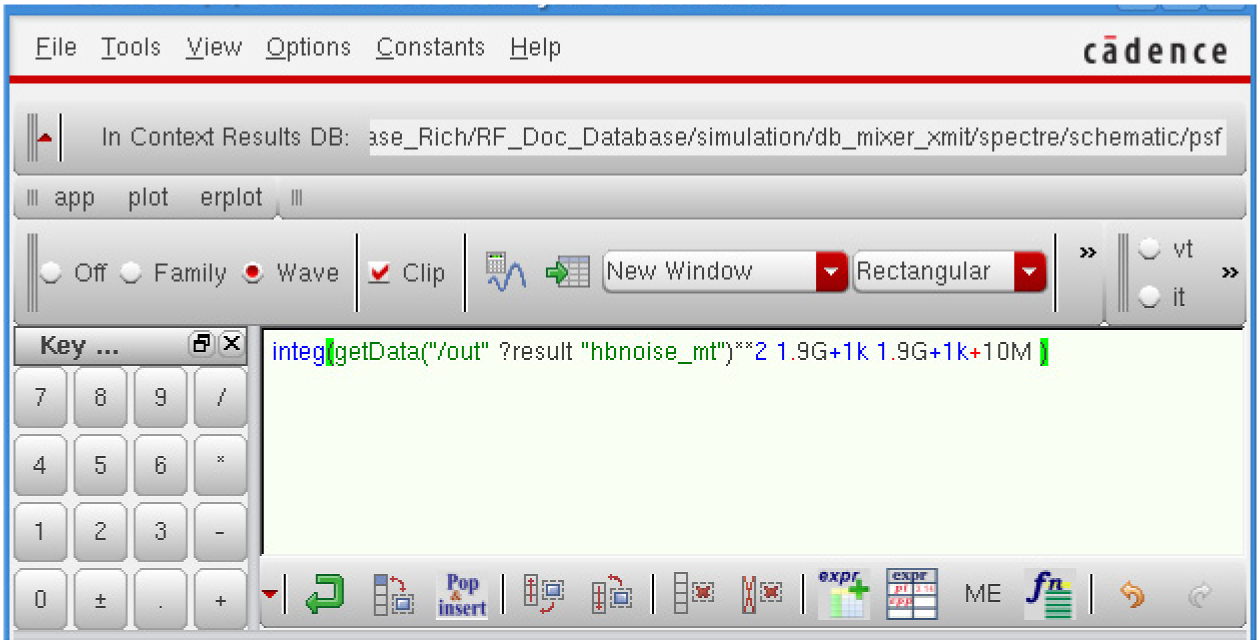
   Note that this is the same range used in the hbnoise *Choosing Analyses* form.

Figure 4-198 Integrating the Squared Noise Voltage over a 1Hz Bandwidth.



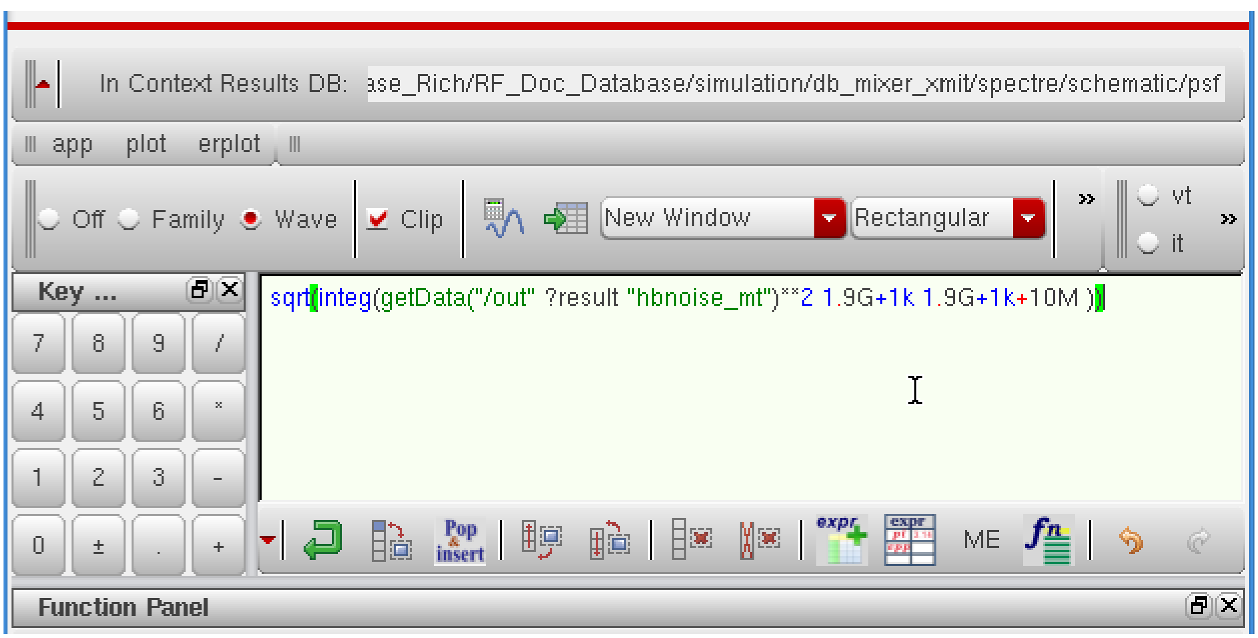
9. Click *OK*. The Calculator buffer is updated, as shown in the figure below.

Figure 4-199 Integrating the Noise Power over a 1Hz Bandwidth



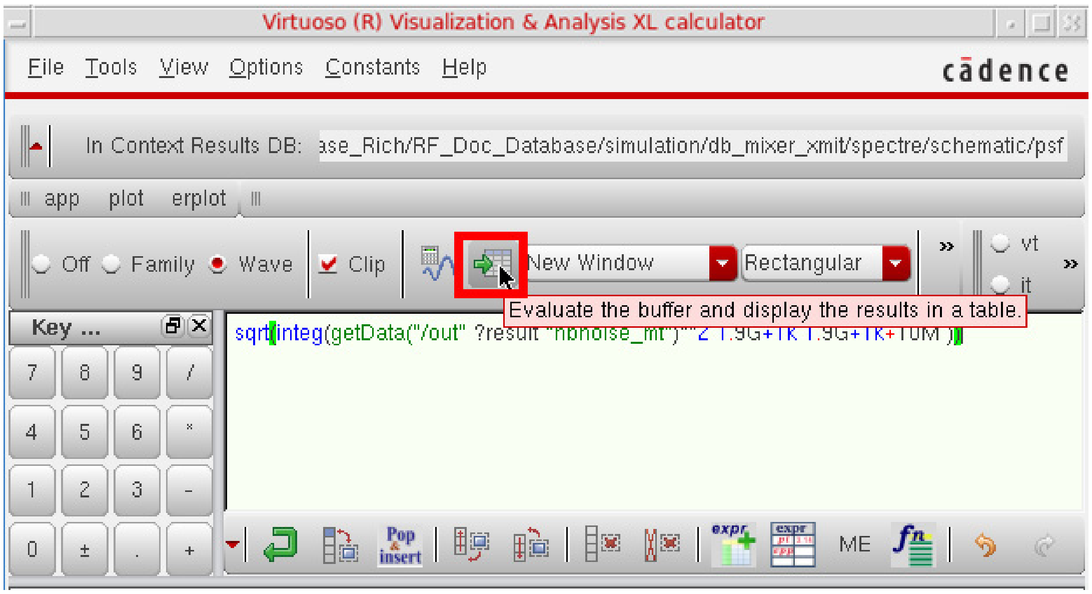
10. Finally, you need to take the square root of the expression in the buffer to put it back into terms of voltage. Click on the *sqrt* function in the *Function Panel.* The calculator buffer looks like the figure below:

Figure 4-200 Integrated Noise Voltage



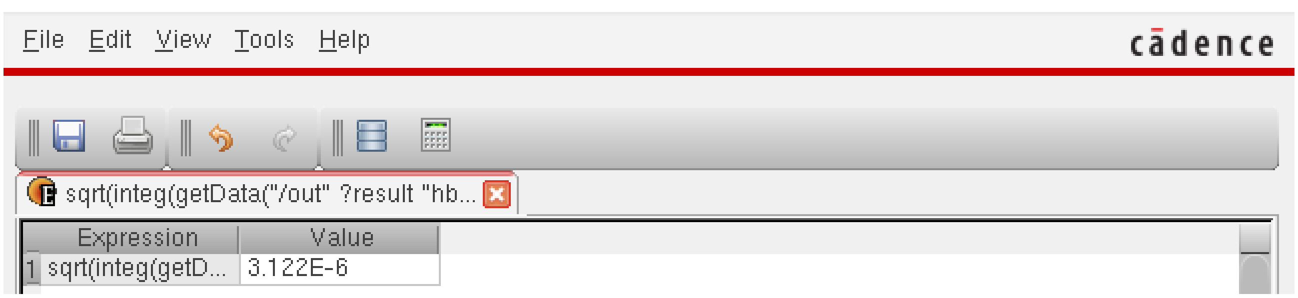
11. Click the Table icon ( )to evaluate the Calculator buffer contents and display the results in a table, as shown below:

Figure 4-201 Evaluating the Calculator Buffer



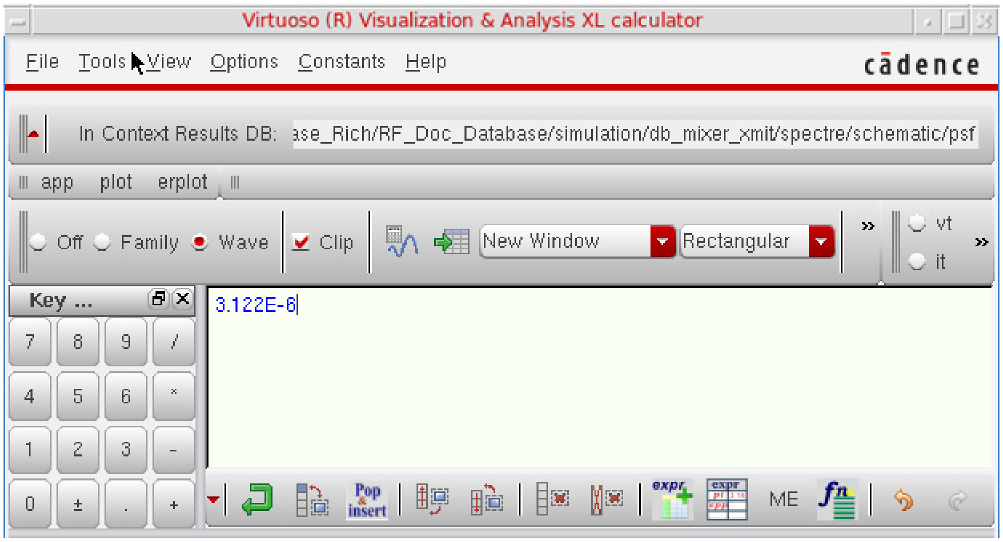
12. The result is sent to the Table, as shown below.

Figure 4-202 Calculating Integrated Noise Voltage Using the Table Icon.



Alternately, if you click the *Plot* icon, the value for noise voltage appears in the Calculator buffer, as shown below.

Figure 4-203 Calculating Integrated Noise Voltage Using the Plot icon



The Noise voltage is 3.122E-6 Volts.

Now you are ready to calculate the Signal to Noise ratio (SNR). You first plotted the rms Voltage (from a three-tone harmonic balance simulation) at 1.904G to be 10.57134mV. Next, you divide the Signal by the Noise to get the SNR.

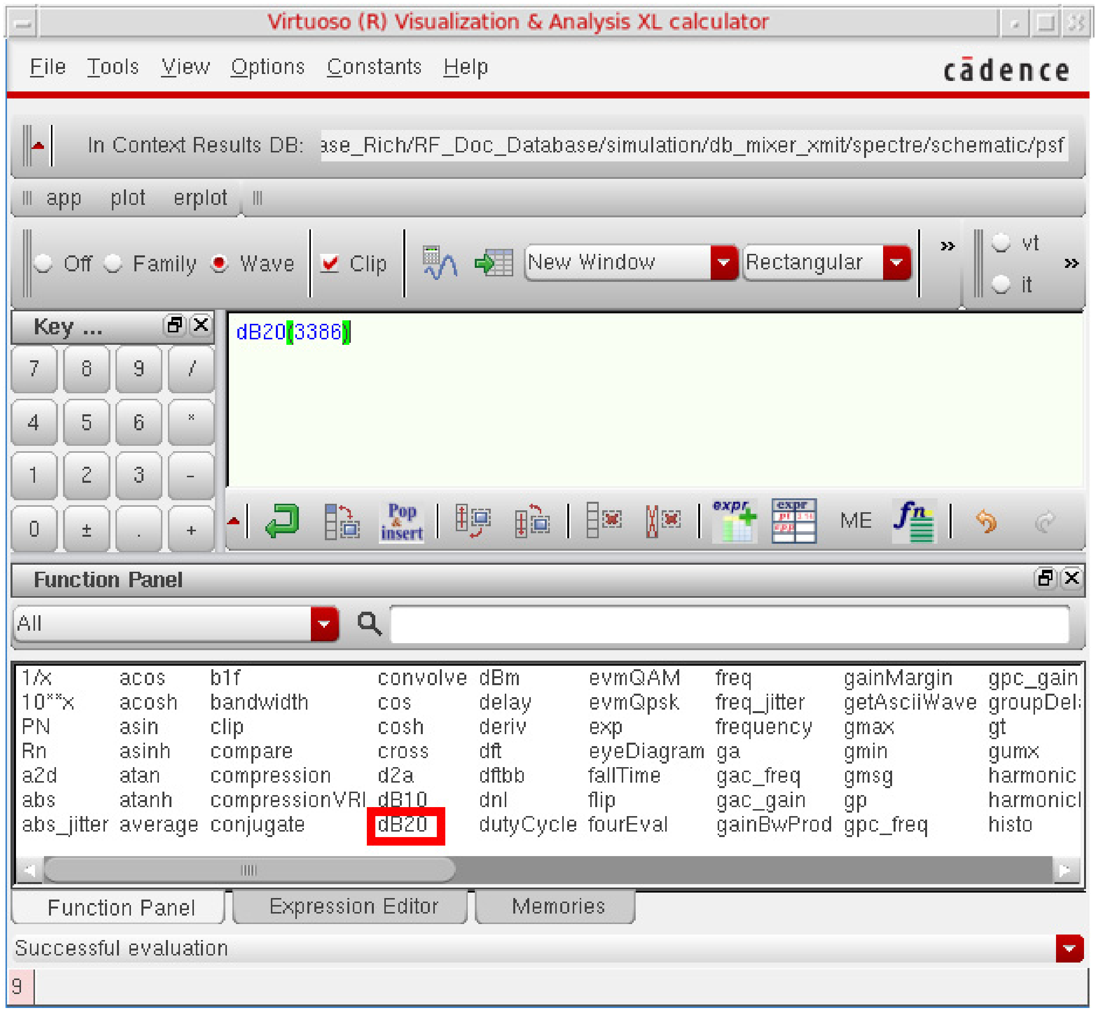
$$SNR = \frac{Signal}{Noise} = \frac{10.57134e^{-3}}{3.122e^{-6}} = 3386$$

To get the SNR in dB at 1.904GHz, take the log of the SNR value and multiple by 20.

$$SNR = 20 \bullet \log(3386) = 70.59dB$$

You can easily do this in the Calculator using the *dB20* Math function, as shown below.

Figure 4-204 Using the dB20 Math Function



Click the *Plot* button (). This gives 70.59dB.

Note that the SNR is fairly high.

Summary

The Mixer section showed how to simulate and make typical measurements on a receiver mixer and a transmit mixer.

In the Receive Mixer Measurement section, the following measurements were shown:

- Mixer Conversion Gain using hb and hbac analyses
- LO to IF leakage using hb analysis

- Noise Figure measurements using hb and hbnoise analyses
- 1dB compression point, desensitization, and blocking using hb, hbac, and hbnoise
- Third-Order Intercept measurement using three-tone harmonic balance
- Rapid IP2 and Rapid IP3 using specialized hbac analysis
- Compression Distortion Summary using hb and specialized hbac analysis.

In the Transmit Mixer Measurement section, the following measurements were shown:

- Image rejection using hb analysis.
- Three-tone swept IP3 (large signal) using multi-tone hb analysis.
- Signal-to-Noise Ratio using the hb and hbnoise analyses.

For more information on simulating Mixers, please refer to the chapters in this user guide and the *Spectre Circuit Simulator RF Analysis Theory Guide*.