

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

**Product Version 23.1  
September 2023**

© 1990–2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation. Used by permission.

Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission. Analog Design Environment XL contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2007, Apache Software Foundation.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522. All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor

# **Contents**

---

|                                                                                  |            |
|----------------------------------------------------------------------------------|------------|
| <b>Introduction</b>                                                              | <b>7</b>   |
| <a href="#">Using SpectreRF from the SPECTRE Hierarchy</a>                       | 7          |
| <a href="#">Accessing the Most Current SpectreRF Documentation</a>               | 7          |
| <a href="#">Creating a Local Editable Copy of the ExampleLibRF Library</a>       | 8          |
| <a href="#">Downloading and Using GPDK180</a>                                    | 8          |
| <a href="#">Starting Virtuoso</a>                                                | 14         |
| <b>Simulating Low Noise Amplifiers</b>                                           | <b>17</b>  |
| <a href="#">The InaSimple Low Noise Amplifier Circuit</a>                        | 18         |
| <a href="#">Setting Up to Simulate the InaSimple Low Noise Amplifier</a>         | 19         |
| <a href="#">Opening the InaSimple Circuit in the Schematic Window</a>            | 19         |
| <a href="#">Choosing Simulator Options</a>                                       | 23         |
| <a href="#">SP Analysis and Small Signal Gain</a>                                | 28         |
| <a href="#">Stability</a>                                                        | 53         |
| <a href="#">Linear 2-port noise analysis (NF, NFmin) and Noise circles</a>       | 63         |
| <a href="#">Summary</a>                                                          | 71         |
| <a href="#">Third-Order Intercept measurement with HB (2 tone HB)</a>            | 72         |
| <a href="#">Choosing Simulator Options</a>                                       | 80         |
| <a href="#">Measuring IP3 with Rapid IP3</a>                                     | 96         |
| <a href="#">Summary</a>                                                          | 104        |
| <b>Simulating Oscillators</b>                                                    | <b>105</b> |
| <a href="#">Simulation Methods</a>                                               | 107        |
| <a href="#">Phases of Autonomous PSS/HB Analysis</a>                             | 107        |
| <a href="#">Phase Noise and Oscillators</a>                                      | 108        |
| <a href="#">Starting and Stabilizing Feedback Oscillators</a>                    | 108        |
| <a href="#">The Oscillator Circuit</a>                                           | 109        |
| <a href="#">Setting Up to Simulate the Oscillator Circuit</a>                    | 111        |
| <a href="#">Setting up ADE Explorer for Oscillator Simulation</a>                | 113        |
| <a href="#">Calculating the Steady-State Solution using PSS Harmonic Balance</a> | 120        |
| <a href="#">Setting up the PSS Analysis</a>                                      | 120        |
| <a href="#">Running the PSS analysis</a>                                         | 126        |

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

---

|                                                                                   |     |
|-----------------------------------------------------------------------------------|-----|
| <u>Oscillator Loop Gain Measurement</u>                                           | 133 |
| <u>Setting up the stb Analysis</u>                                                | 135 |
| <u>Setting up the HB Analysis</u>                                                 | 147 |
| <u>Setting up the HBSTB Analysis</u>                                              | 152 |
| <u>Running the HB, stb and HBSTB analysis</u>                                     | 156 |
| <u>Plotting the results</u>                                                       | 156 |
| <u>Phase Noise Measurement and Noise Summary Table</u>                            | 176 |
| <u>Setting up the HB and HBnoise Analysis</u>                                     | 178 |
| <u>Oscillator Swept Tuning Range and Phase Noise Measurement</u>                  | 217 |
| <u>Setting up the HB and HBnoise Analysis</u>                                     | 219 |
| <u>Ring Oscillator Measurements</u>                                               | 246 |
| <u>Starting and Stabilization of Ring Oscillators</u>                             | 246 |
| <u>The Oscillator Circuit</u>                                                     | 246 |
| <u>Simulating the Oscillator Circuit</u>                                          | 247 |
| <u>Calculating the Steady-State Solution using PSS Shooting Analysis</u>          | 249 |
| <u>Setting up the PSS Analysis</u>                                                | 249 |
| <u>Running the PSS analysis</u>                                                   | 263 |
| <u>FM Jitter Measurement using PSS Shooting and Pnoise Jitter Analyses</u>        | 272 |
| <u>Determining FM Jitter</u>                                                      | 272 |
| <u>Setting up the PSS Analysis</u>                                                | 272 |
| <u>Running the PSS and Pnoise analysis</u>                                        | 291 |
| <u>Calculating the Swept Tuning Range and Phase Noise for the Ring Oscillator</u> | 298 |
| <u>Setting up the PSS Analysis</u>                                                | 298 |
| <u>Running the PSS and Pnoise analysis</u>                                        | 315 |
| <u>Summary</u>                                                                    | 322 |
| <u>Simulating Mixers</u>                                                          | 323 |
| <u>The db_mixer and db_mixer_xmit Mixer Circuits</u>                              | 324 |
| <u>Setting Up to Simulate the db_mixer Mixer</u>                                  | 325 |
| <u>Opening the db_mixer Mixer Circuit in the Schematic Window</u>                 | 325 |
| <u>Choosing Simulator Options</u>                                                 | 329 |
| <u>Setting Up the Simulation - Setting Design Variables</u>                       | 334 |
| <u>Setting Up Model Libraries</u>                                                 | 384 |
| <u>Setting Design Variables</u>                                                   | 385 |
| <u>Setting up hbnoise to measure noise figure</u>                                 | 390 |
| <u>Third-Order Intercept measurement with HB</u>                                  | 411 |
| <u>Mixer Distortion Measurement</u>                                               | 453 |



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

---

|                                                              |     |
|--------------------------------------------------------------|-----|
| <u>Setting Up to Simulate the db_mixer_xmit Mixer</u> .....  | 478 |
| <u>Three Tone Spectral Content and Image Rejection</u> ..... | 481 |
| <u>Three Tone IP3</u> .....                                  | 506 |
| <u>Signal-to-Noise Ratio</u> .....                           | 519 |
| <u>Three Tone HB Analysis Setup</u> .....                    | 525 |
| <u>Summary</u> .....                                         | 550 |

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

---

---

# **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.



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 xzf 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/`

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

---

- ❑ `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.

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

Figure 1-1 Cadence Online Support Website

The screenshot shows a web browser window with the address bar displaying "https://secure1.cadence.com/loginapp/CSSOLogin/login.action?TAM\_C". The page is titled "Cadence Online Support" and "Cadence Log In". On the left, there is a sidebar with sections: "Overview" (describing support resources), "Supported Browsers:" (listing Internet Explorer 9.x and above, Mozilla Firefox 36.x and above, Mozilla Firefox 15.x and above (linux), Google Chrome 40.x and above, Safari, and iPad and Android Tablets 7" and above), "Important – Please Read" (advising a page refresh on first login), "Related Links" (with links to "Getting Started with Cadence Online Support (Video)" and "Contact Customer Support"), and "Other Sites" (with links to "Cadence.com", "Cadence User Community", "Education", "Software Downloads", and "PSpice User Community"). The main content area on the right is titled "Cadence Log In" and includes a message: "Cadence uses a single Login ID for all applications. You will use the same Login ID and password for each application you are registered with." Below this is a login form with fields for "Email" and "Password", a "Remember my email" checkbox, and a "LOG IN" button. There is also a "Forgot your password?" link with the instruction: "Just enter your Email and click on the 'Forgot your password?' link." At the bottom of the main content area, there is a "New User?" section with a "Register Now" link and a "Registration Help" section with a "Help" link.

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.

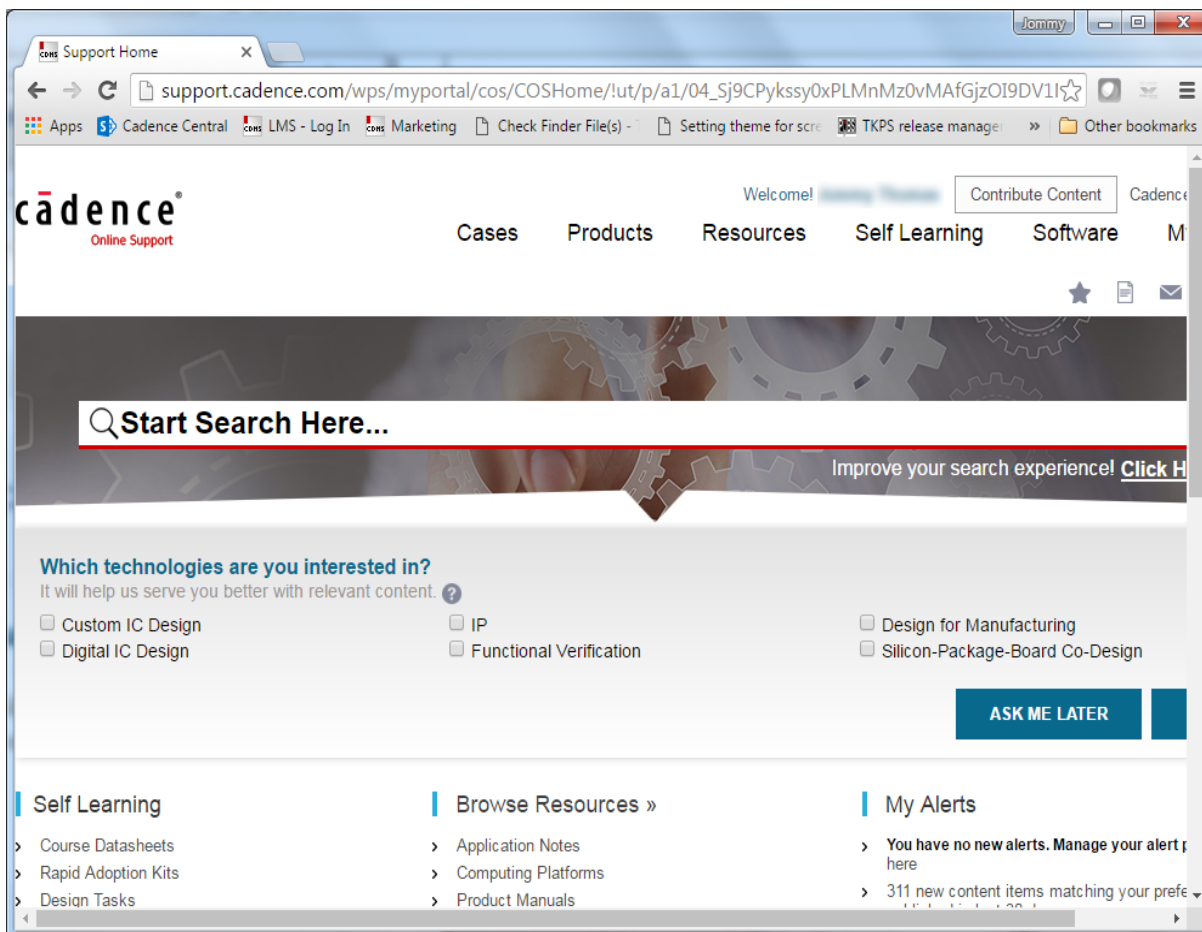
For the example designs in Appendix A, you need to use Cadence's GPDK180 which can be downloaded as follows.

1. Enter your email and password.

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

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

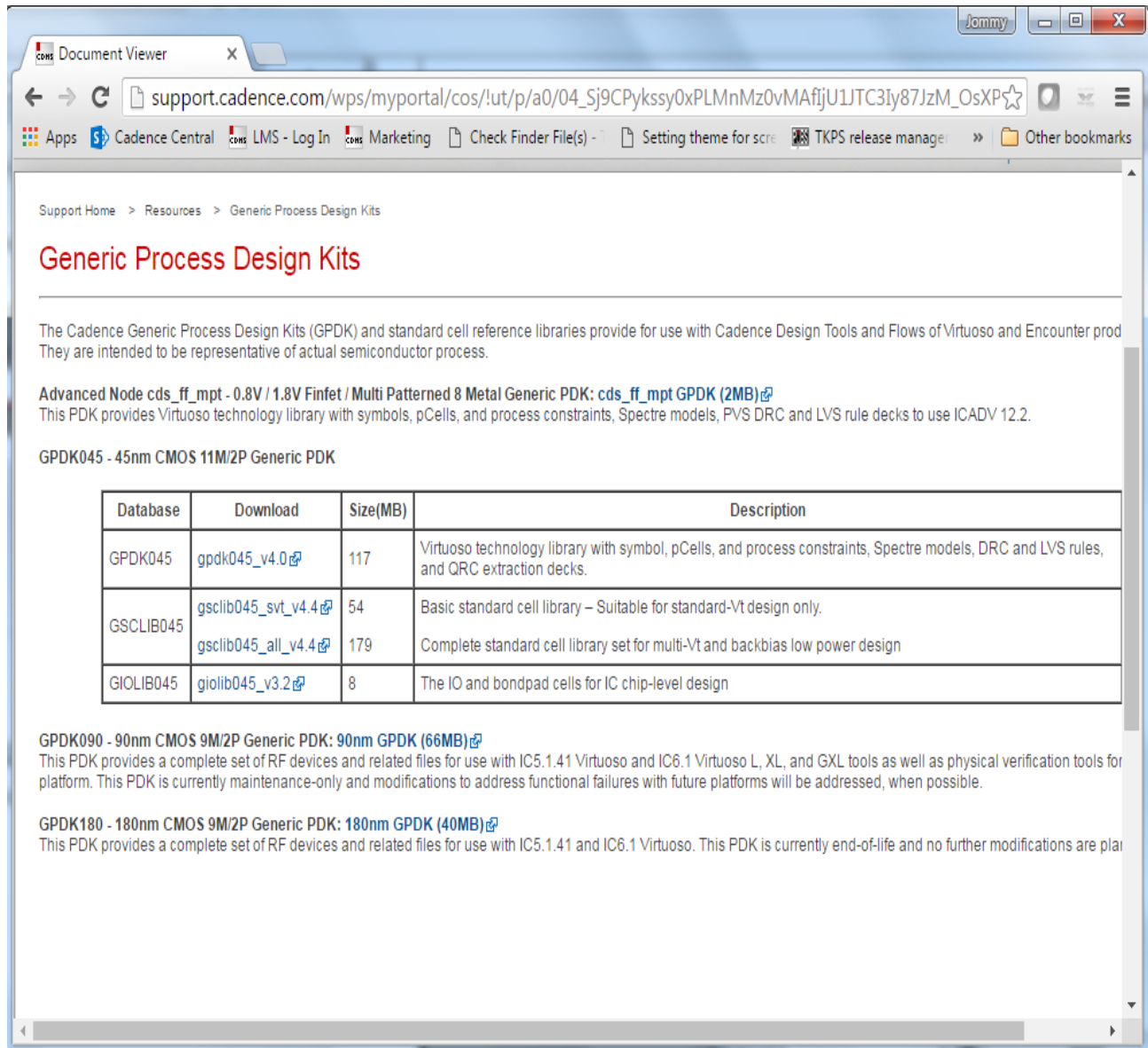
Figure 1-2 Cadence Support Home Page



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

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

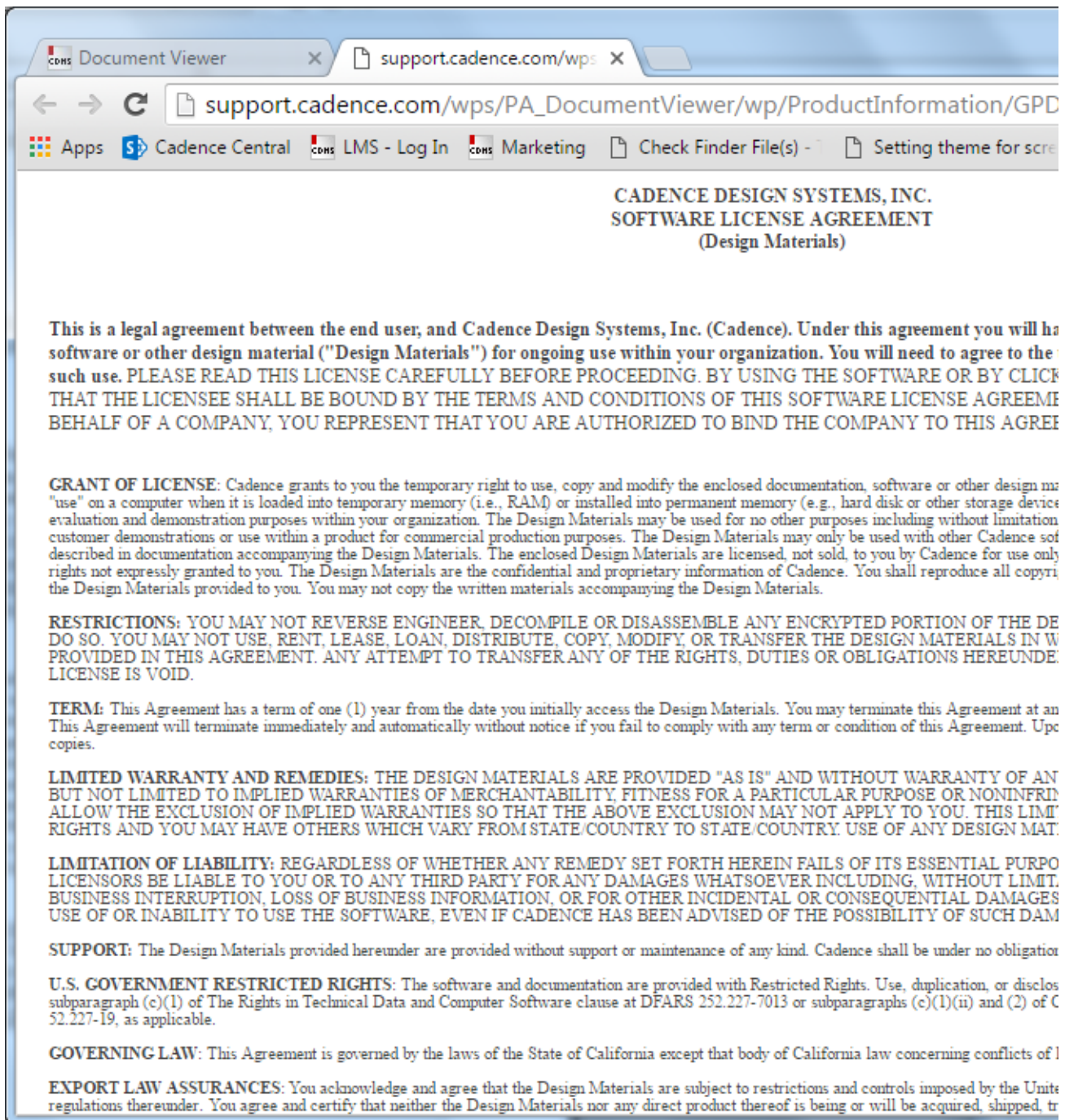
Figure 1-3 Generic Process Design Kits Web Page





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

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

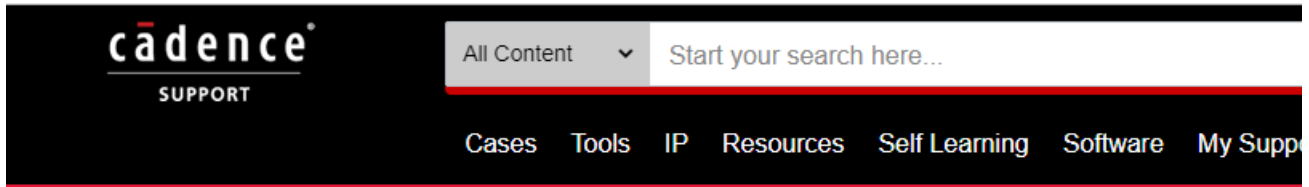


4. Scroll down the page and click *I ACCEPT*.

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

---

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:

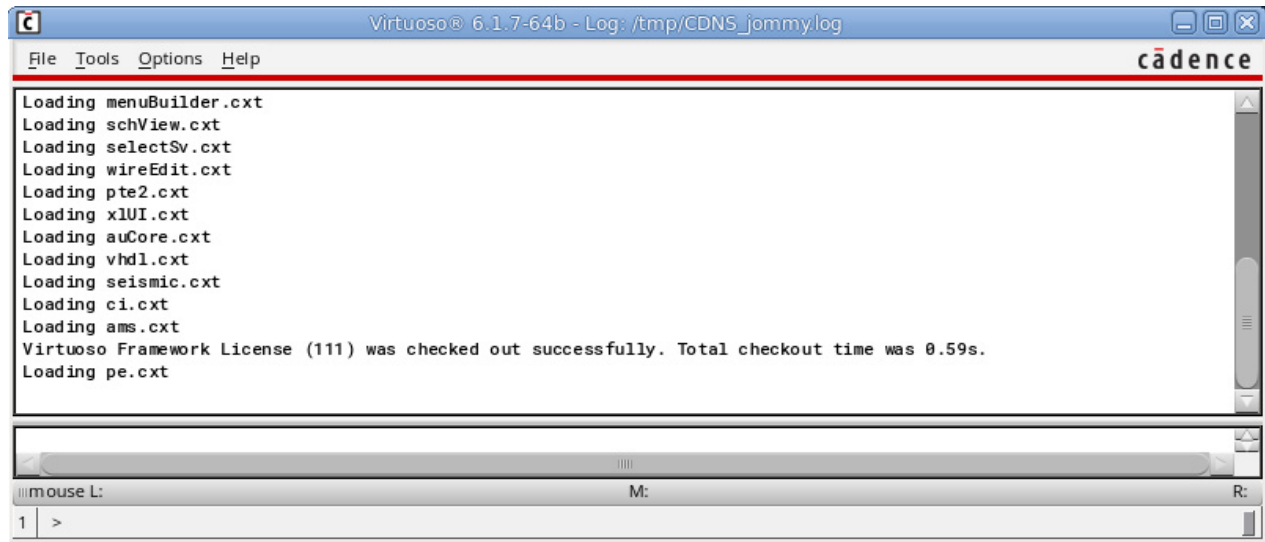
1. In a UNIX window, type `virtuoso &` to start the Cadence software.

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

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

---

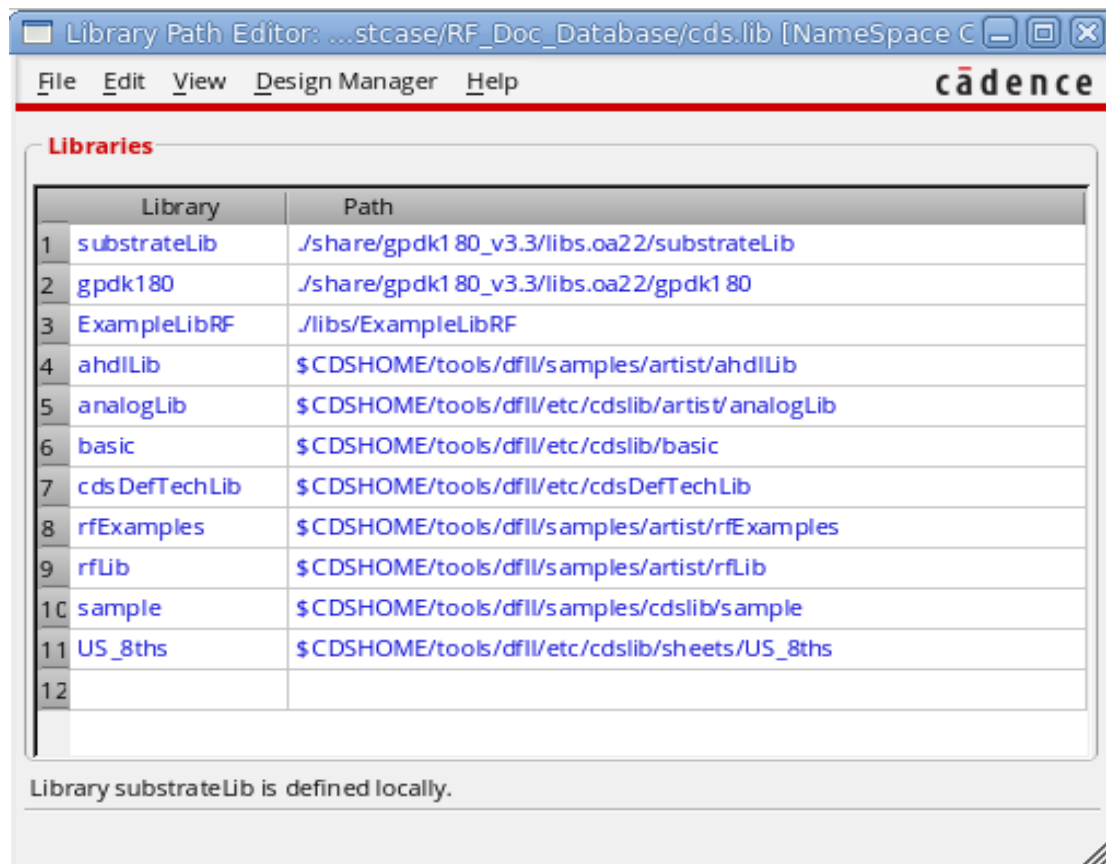
**Figure 1-4 Command Interpreter Window**



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*.

---

## Simulating Low Noise Amplifiers

---

The SpectreRF simulator can simulate very linear circuits, such as Low Noise Amplifiers (LNA). This section uses the Ina 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.

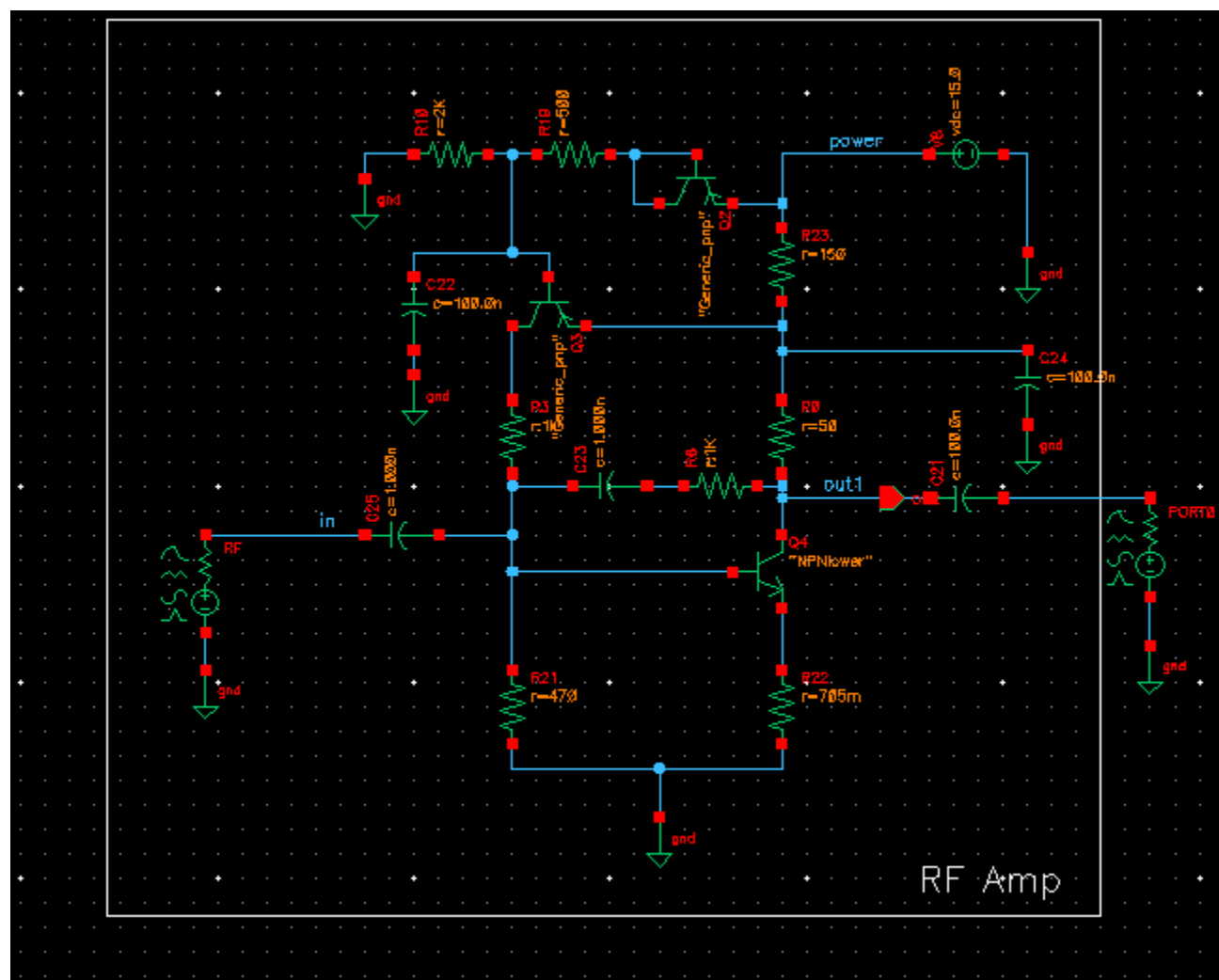
| LNA Measurements (InaSimple                                 | Analyses |
|-------------------------------------------------------------|----------|
| S-Parameter Analysis, Small Signal Gain                     | sp       |
| Stability, Stability Circles                                | sp       |
| 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 InaSimple Low Noise Amplifier Circuit

The InaSimple 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 InaSimple circuit is shown below. It is a differential low noise amplifier.

Figure 2-1 Schematic for the InaSimple Low Noise Amplifier



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

---

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

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

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

## Setting Up to Simulate the InaSimple Low Noise Amplifier

### Opening the InaSimple Circuit in the Schematic Window

1. In CIW, choose *File – Open*.

The *Open File* form is displayed.

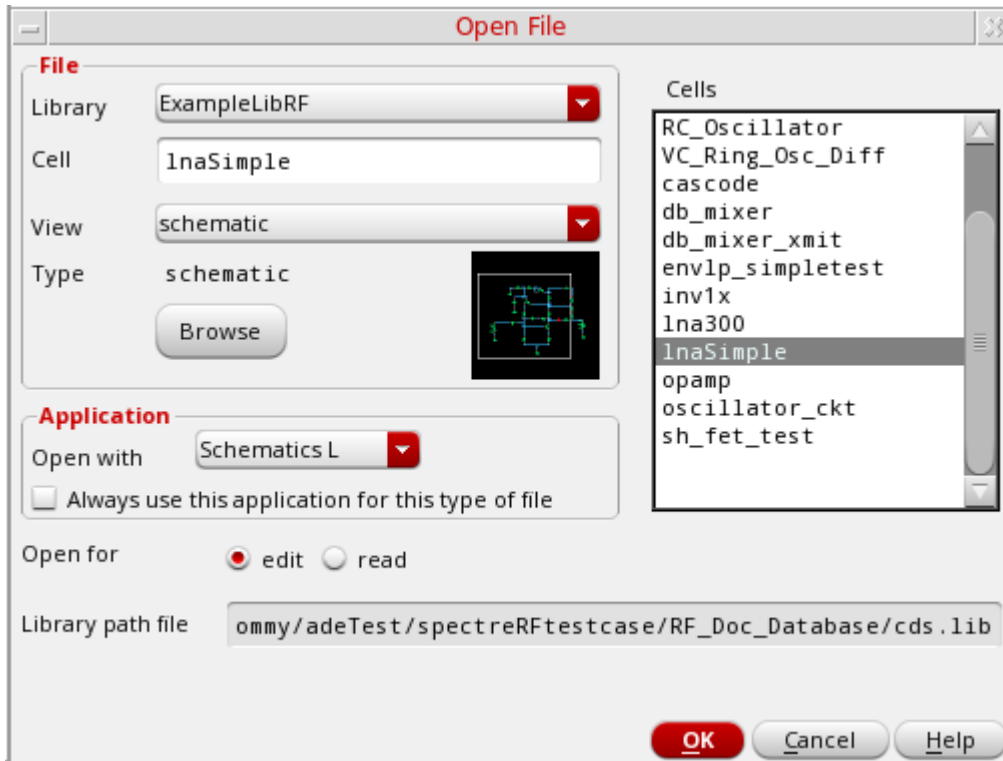
2. Choose *ExampleLibRF* from the *Library* drop-down list.
3. Choose *InaSimple* from the *Cells* list box.
4. Choose *schematic* from the *View* drop-down list.

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

---

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

**Figure 2-2 Open File Form**



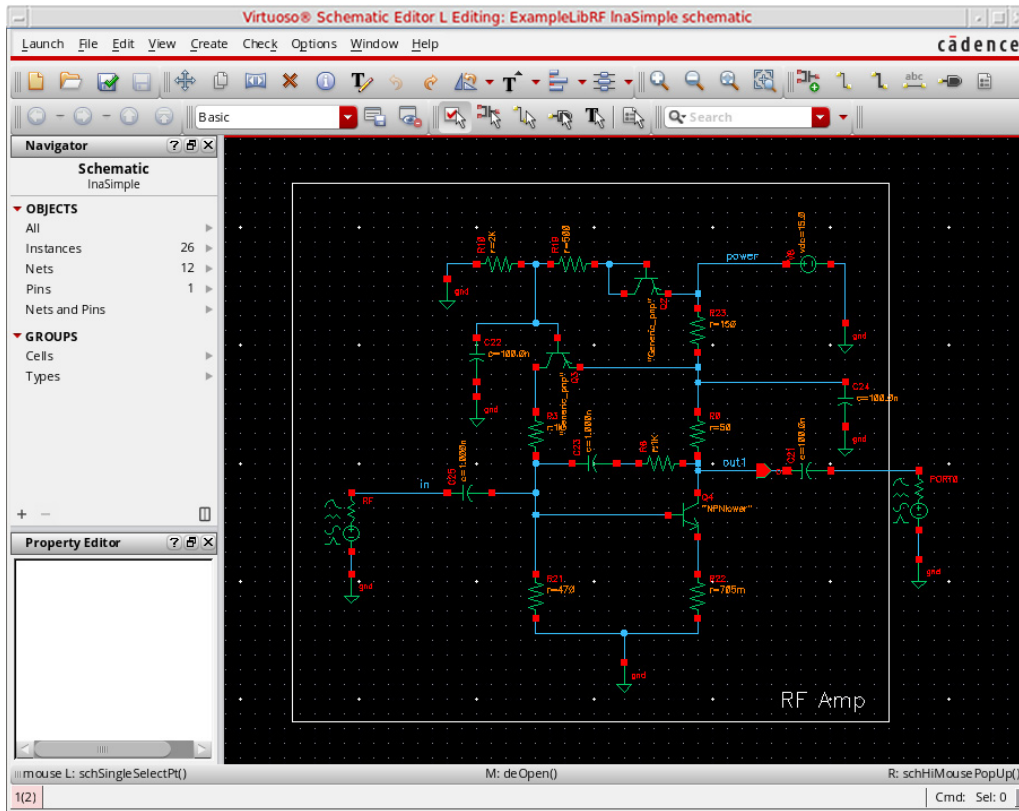
5. Click *OK*.

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

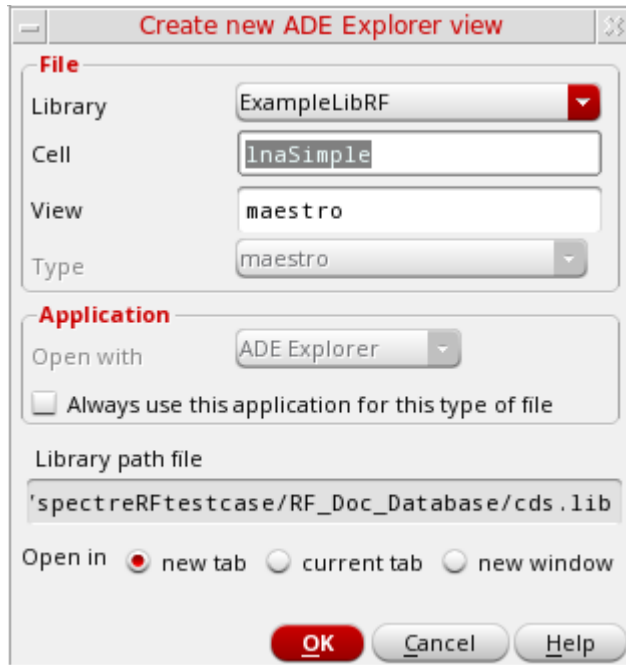


## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

Figure 2-3 InaSimple Schematic



**Figure 2-4 Create new ADE Explorer view**

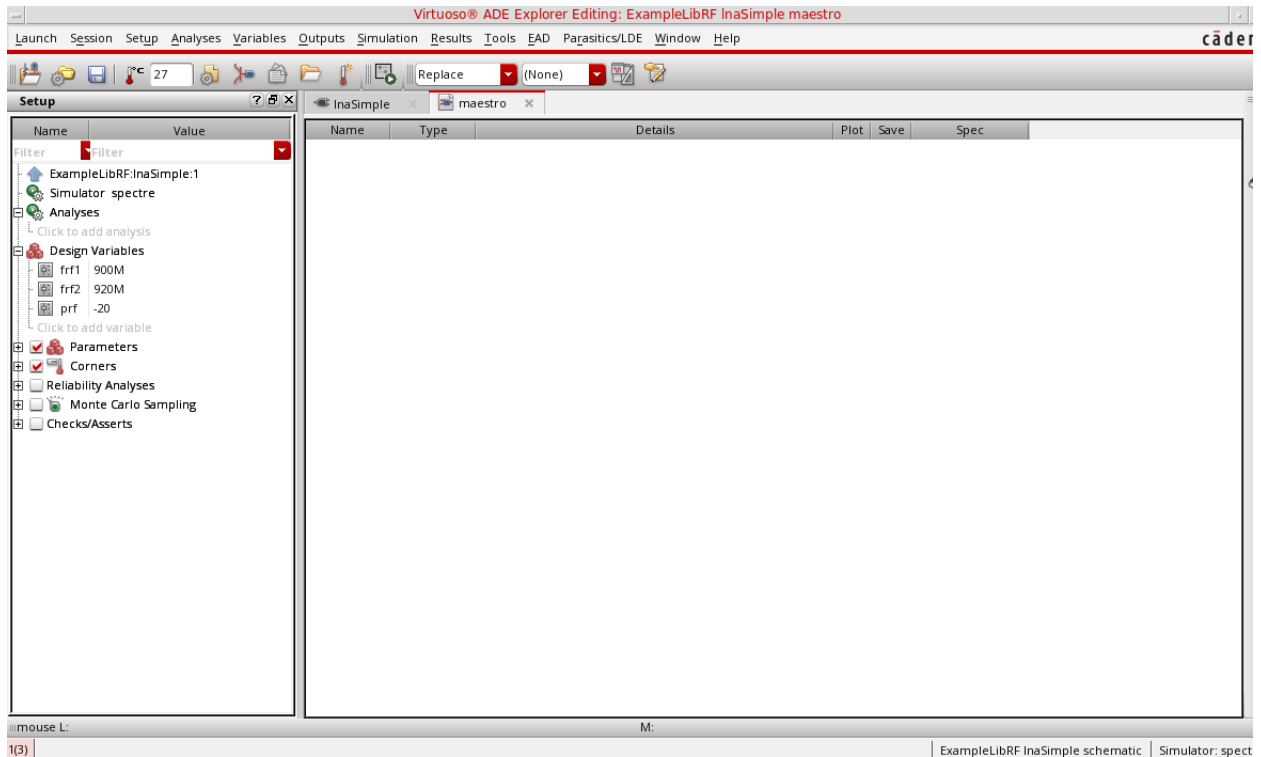


8. Leave each option to the default selections and click *OK*.

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

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

**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:

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

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.

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

---

**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.

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

Figure 2-8 Save Options Form

Save Options

BasicSave By Subckt

Save Options

Select signals to output (save)

☐ none☐ selected☐ lvlpub☐ lvl☒ allpub☐ all

Select power signals to output (pwr)

☐ none☐ total☐ devices☐ subckts☐ all

Select level of subcircuit to output (nestlvl)

Select device currents (currents)

☐ selected☐ nonlinear☐ all☐ none

Select AC terminal currents (useprobes)

☐ yes☐ no

Select probe terminal level (probelvl)

Select AHDL variables (saveahdlvars)

☐ selected☐ all

Transient Time Window Options

☐ Transient time window save options

Time Setup

Save circuit information analysis

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail: nodeSort: nameThreshold:

Output Options

Output Format

☐ sst2☐ psf☐ psf with floats☒ psfxl☐ fsdb

Use Fast Viewing Extensions

☐

September 2023  
© 1990-2023

26

Product Version 23.1  
All Rights Reserved.

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

---

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.

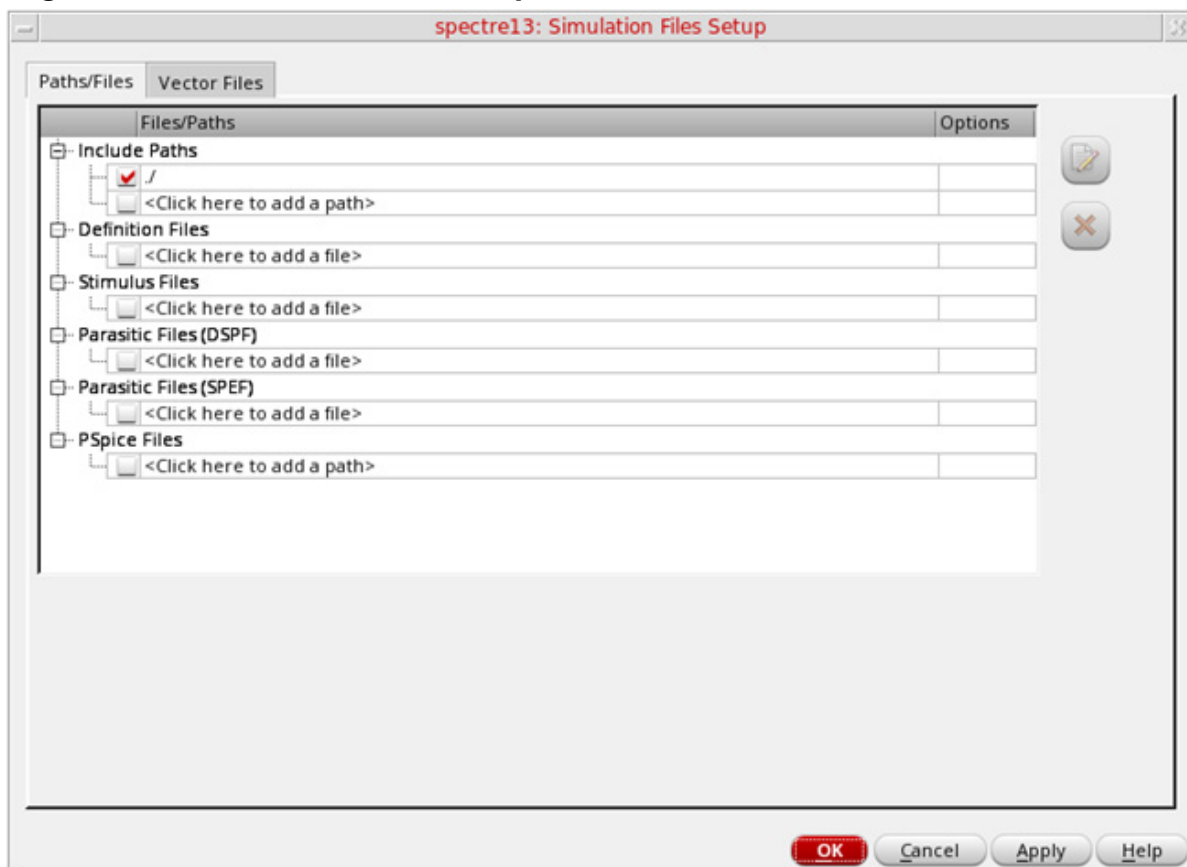
To save the currents, select *nonlinear* in the *Select device currents (currents)* section if you just want to save the device currents, or select *all* if you want to save all the currents in the circuit.

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.

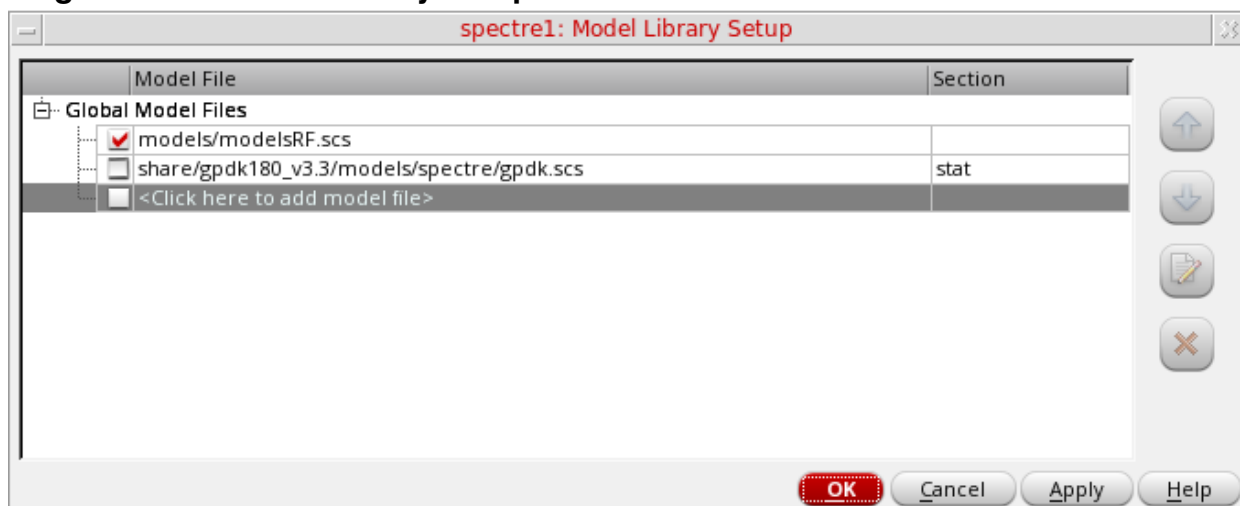
4. Type in the name of the model file, as follows:

`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:



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

---

- *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.

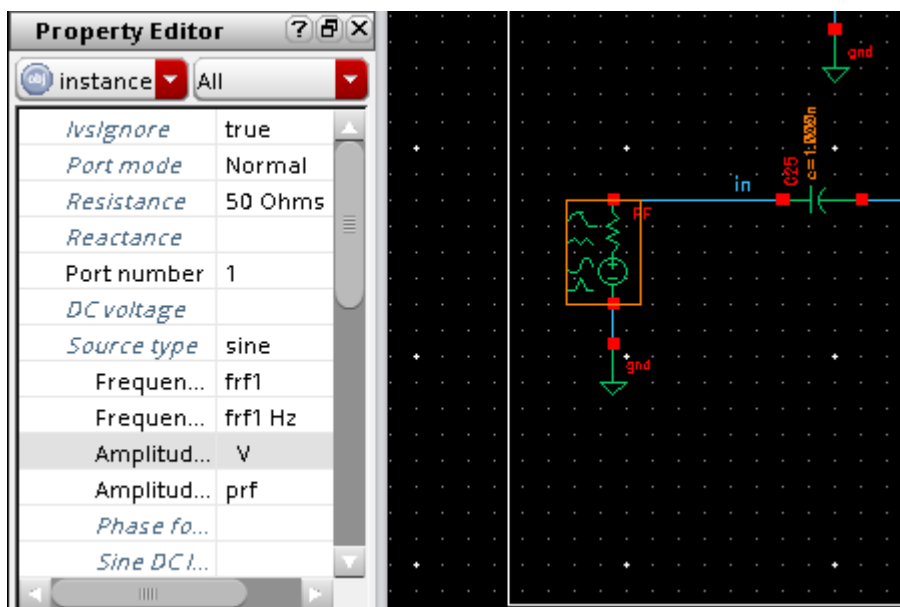
- *GPC*: power gain circle
- *GAC*: available gain circle

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 InaSimple.

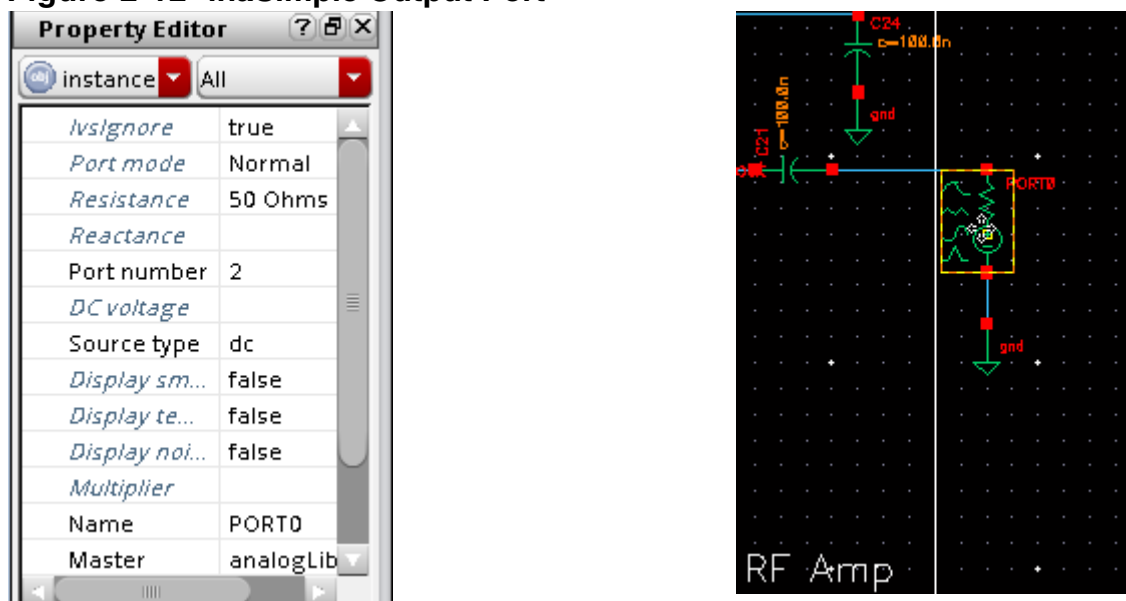
Figure 2-11 RF Voltage Source in InaSimple



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

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

Figure 2-12 InaSimple Output Port



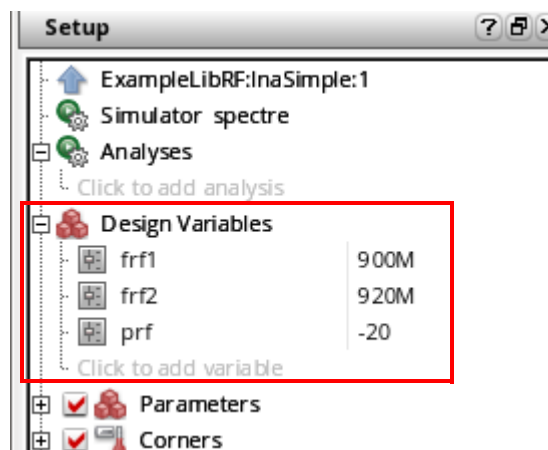
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

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.

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

---

The *Choosing Analyses* form is displayed.

**Figure 2-15 Choosing Analyses Form**

**Choosing Analyses -- ADE Explorer**

Analysis

☒ tran   ☐ dc   ☐ ac   ☐ noise  
☐ xf   ☐ sens   ☐ dcmatch   ☐ acmatch  
☐ stb   ☐ pz   ☐ lf   ☐ sp  
☐ envlp   ☐ pss   ☐ pac   ☐ pstb  
☐ pnoise   ☐ pxf   ☐ psp   ☐ qpss  
☐ qpac   ☐ qpnoise   ☐ qpxf   ☐ qpsp  
☐ hb   ☐ hbac   ☐ hbstb   ☐ hbnoise  
☐ hbsp   ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)  
☐ conservative   ☐ moderate   ☐ liberal

☐ Transient Noise

Dynamic Parameter

Enabled ☐

Options...

**OK**   Cancel   Defaults   Apply   Help

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

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

---

**Figure 2-16 sp Choosing Analyses Form**

**Choosing Analyses -- ADE Explorer**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☒ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☐ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbisp ☐ hbxf

S-Parameter Analysis

Ports

Probes

Sweep Variable

☒ Frequency

☐ Design Variable

☐ Temperature

☐ Component Parameter

☐ Model Parameter

☐ None

Sweep Range

☒ Start-Stop Start  Stop

☐ Center-Span

Sweep Type

Automatic

Add Specific Points ☐

Add Points By File ☐

Do Noise

☐ yes

☒ no

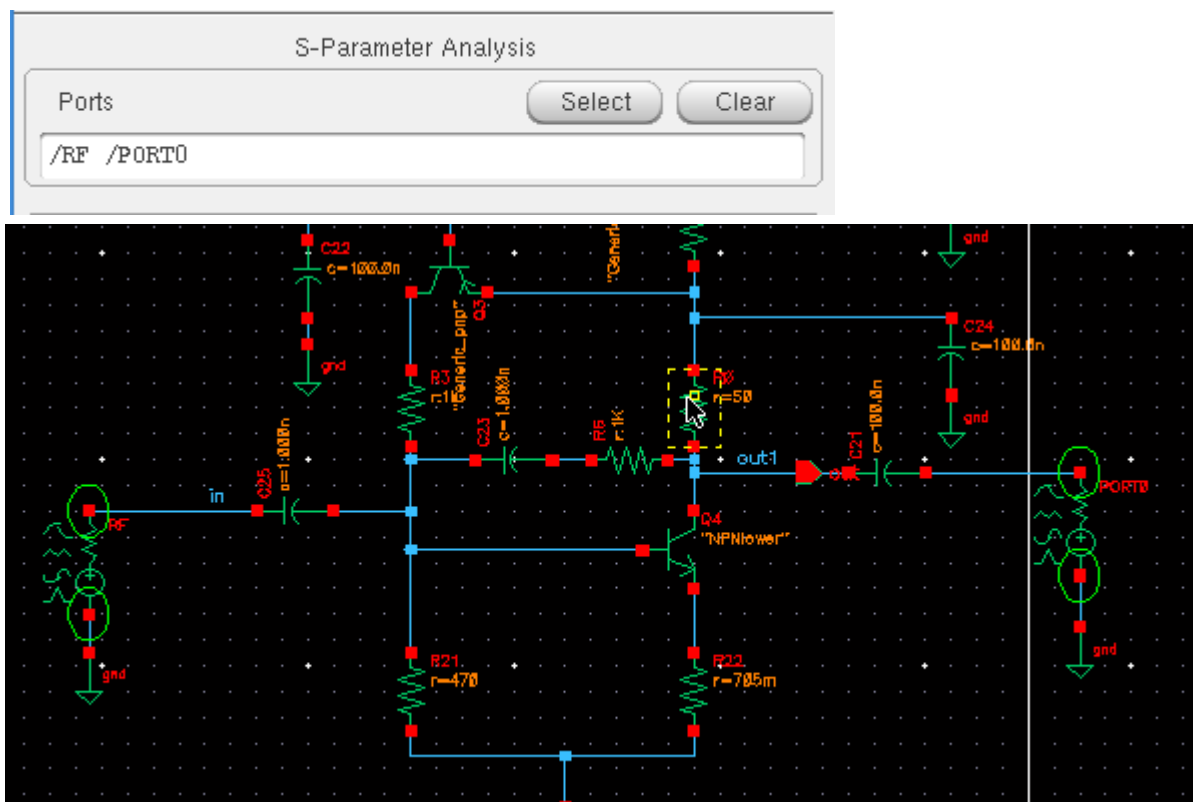
Mode

☒ Single-Ended ☐ Mixed In/Out ☐ Other

Enabled ☐

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 within 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.

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

---

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.

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

Figure 2-18 sp Choosing Analyses Form

**Choosing Analyses -- ADE Explorer**

**Analysis**

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☒ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpsp

☐ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

**S-Parameter Analysis**

**Ports**

/RF /PORT0

**Sprobes**

**Sweep Variable**

☒ Frequency

☐ Design Variable

☐ Temperature

☐ Component Parameter

☐ Model Parameter

☐ None

**Sweep Range**

☒ Start-Stop ☐ Center-Span

Start 500M Stop 4.0G

**Sweep Type**

Linear

☐ Step Size ☐ Number of Steps 50

Add Specific Points ☐

Add Points By File ☐

**Do Noise**

☒ yes ☐ no

Output port /PORT0

Input port /RF

**Mode**

☒ Single-Ended ☐ Mixed In/Out ☐ Other

Enabled ☒

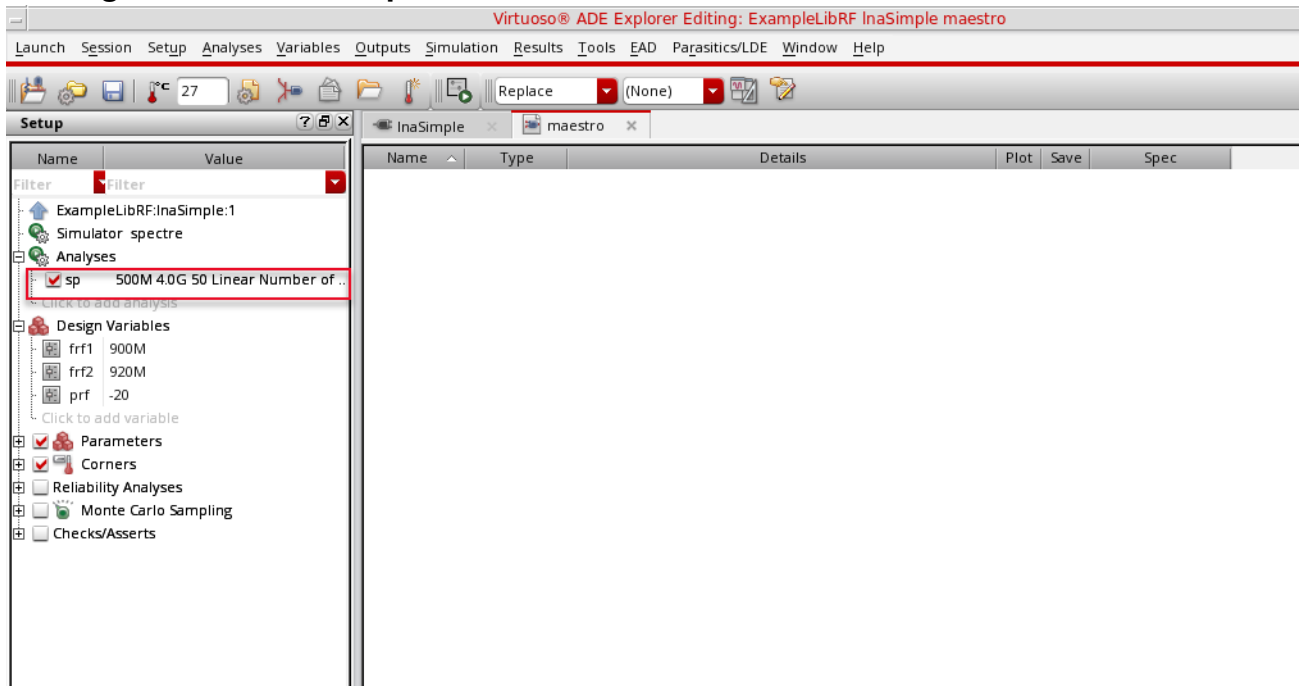


## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

### 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.

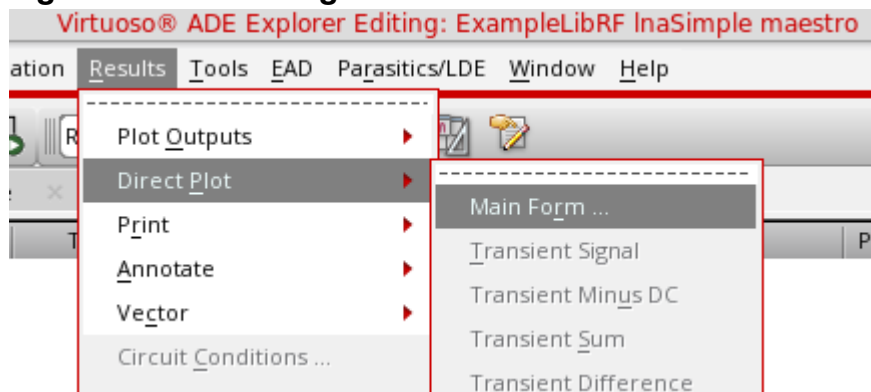
This netlists the design and runs the simulation. A SpectreRF status window is displayed (spectre.out log file). When the analysis has completed, you may iconify the status window.

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

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

---

**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

Direct Plot Form

Plotting Mode: Append

**Analysis**

☒ sp

**Function**

☒ SP ☐ ZP ☐ YP ☐ HP  
☐ GD ☐ VSWR ☐ NFmin ☐ Gmin  
☐ Rn ☐ rn ☐ NF ☐ Kf  
☐ B1f ☐ GT ☐ GA ☐ GP  
☐ Gmax ☐ Gmsg ☐ Gumx ☐ ZM  
☐ NC ☐ GAC ☐ GPC ☐ LSB  
☐ SSB

Description: S-Parameter

**Plot Type**

☒ Rectangular ☐ Z-Smith ☐ Y-Smith  
☐ Polar

**Modifier**

☒ Magnitude ☐ Phase ☐ dB20  
☐ Real ☐ Imaginary

S11 S12  
S21 S22

Add To Outputs ☐

> To plot, press Sij-button on this form...

Close Help

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 ( $\Gamma_S$ ) and load ( $\Gamma_L$ ) 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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☒ sp

**Function**

☐ SP ☐ ZP ☐ YP ☐ HP  
☐ GD ☐ VSWR ☐ NFmin ☐ Gmin  
☐ Rn ☐ rn ☐ NF ☐ Kf  
☐ B1f ☒ GT ☐ GA ☐ GP  
☐ Gmax ☐ Gmsg ☐ Gumx ☐ ZM  
☐ NC ☐ GAC ☐ GPC ☐ LSB  
☐ SSB

Description: Transducer Gain

**Modifier**

☐ Magnitude ☒ dB10

Add To Outputs ☐ Plot

> Press plot button on this form...

Close Help

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*,  $\Gamma_S$  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.

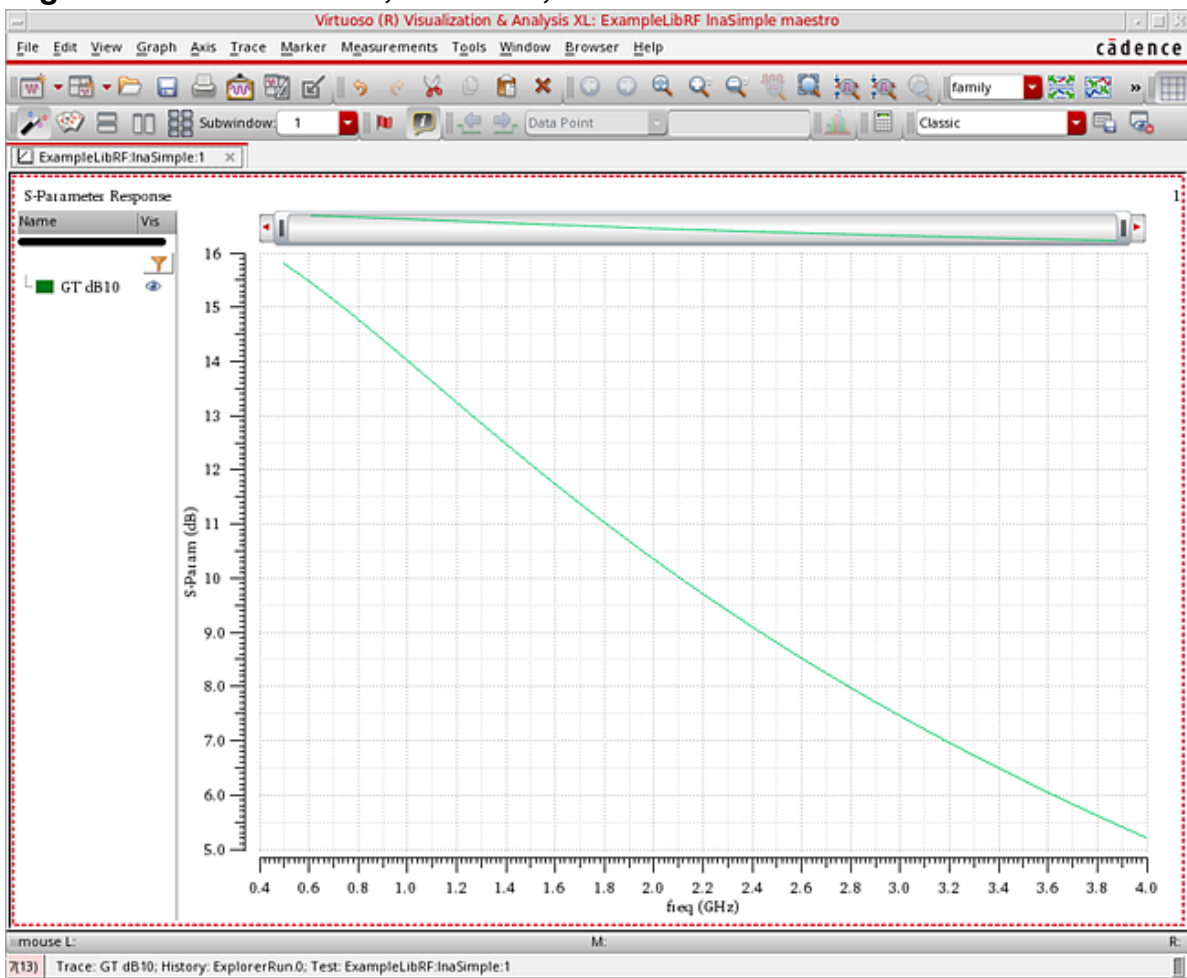
**Note:** The ADE Explorer environment assumes that  $\Gamma_{in}=0$  (the input is matched) so the equation for power gain ( $GP$ ) reduces to the equation below:

**(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,  $S_{11}$  is close to zero.

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

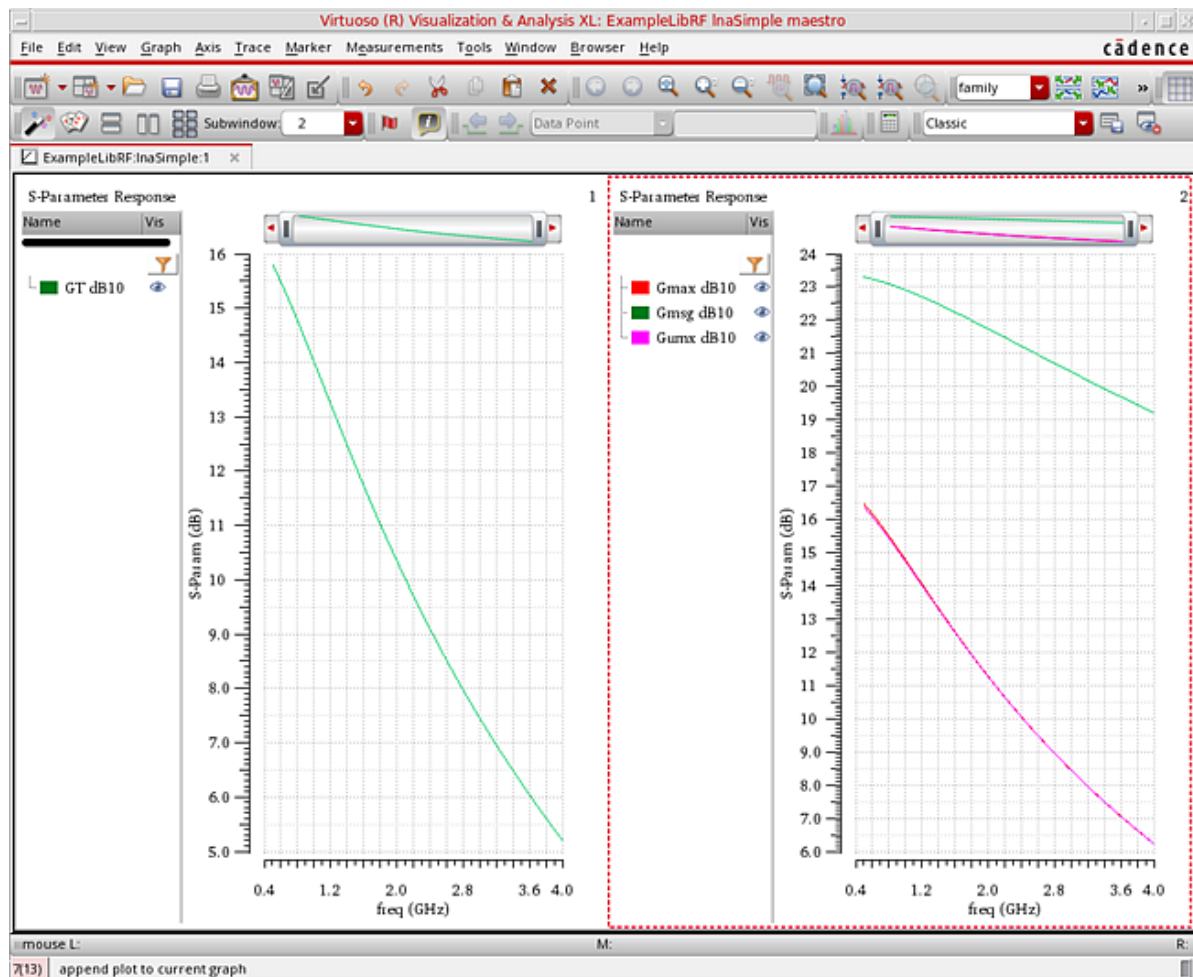
**Figure 2-24 Changing Plot Mode to New SubWindow**



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 ( $G_{umx}$ ) 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  $S_{11} = \Gamma_s$  and  $S_{22} = \Gamma_L$
- ❑ Maximum transducer power gain,  $G_{max}$ , is the simultaneous conjugate matching power gain when both the input and output are conjugately matched.
- ❑ Maximum stable gain,  $G_{msg}$ , is the maximum of  $G_{max}$  when the stability condition,  $K > 1$ , is satisfied.

**Note:** Equations for  $G_{umx}$ ,  $G_{max}$ , and  $G_{msg}$  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.



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

---

In the plots above, the maximum unilateral transducer power gain ( $G_{umx}$ ) is very close to the maximum transducer power gain ( $G_{max}$ ) which means the reverse coupling  $S_{12}$  is small. The maximum stable gain ( $G_{msg}$ ) 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**



The image shows a software dialog box titled "Direct Plot Form". It contains several sections for configuring a plot:

- Plotting Mode:** A dropdown menu set to "Append".
- Analysis:** A radio button labeled "sp" is selected.
- Function:** A grid of radio buttons for various functions. "GAC" (Available Gain Circle) is selected. Other options include SP, ZP, YP, HP, GD, VSWR, NFmin, Gmin, Rn, rn, NF, Kf, B1f, GT, GA, GP, Gmax, Gmsg, Gumx, ZM, NC, GPC, LSB, and SSB.
- Description:** The text "Available Gain Circles" is displayed.
- Plot Type:** Radio buttons for "Z-Smith" (selected) and "Y-Smith".
- Sweep:** Radio buttons for "frequency" (selected) and "Gain Level (dB)".
- Level (dB):** An empty text input field.
- Frequency Range (Hz):** Input fields for "Start", "Stop", and "Step".
- Add To Outputs:** An unchecked checkbox.
- Footer:** A message "> Provide a numeric value for 'Level (dB)' on this form..." and two buttons: "Close" and "Help".

### Available Gain Circle (GAC)

Available Gain (GA) is solely a function of the source reflection coefficient  $\Gamma_S$ . Thus, you can draw available gain contours on the Smith chart of  $\Gamma_S$ . The location for the peak of

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

---

the contour corresponds to  $\Gamma_S$  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  $\Gamma_S=0$ .

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 InaSimple 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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☒ sp

**Function**

☐ SP ☐ ZP ☐ YP ☐ HP  
☐ GD ☐ VSWR ☐ NFmin ☐ Gmin  
☐ Rn ☐ rn ☐ NF ☐ Kf  
☐ B1f ☐ GT ☐ GA ☐ GP  
☐ Gmax ☐ Gmsg ☐ Gumx ☐ ZM  
☐ NC ☒ GAC ☐ GPC ☐ LSB  
☐ SSB

Description: Available Gain Circles

**Plot Type**

☒ Z-Smith ☐ Y-Smith

Sweep: ☐ frequency ☒ Gain Level (dB)

Frequency (Hz): 900M

**Level Range (dB)**

Start: 4 Stop: 14  
Step: 2

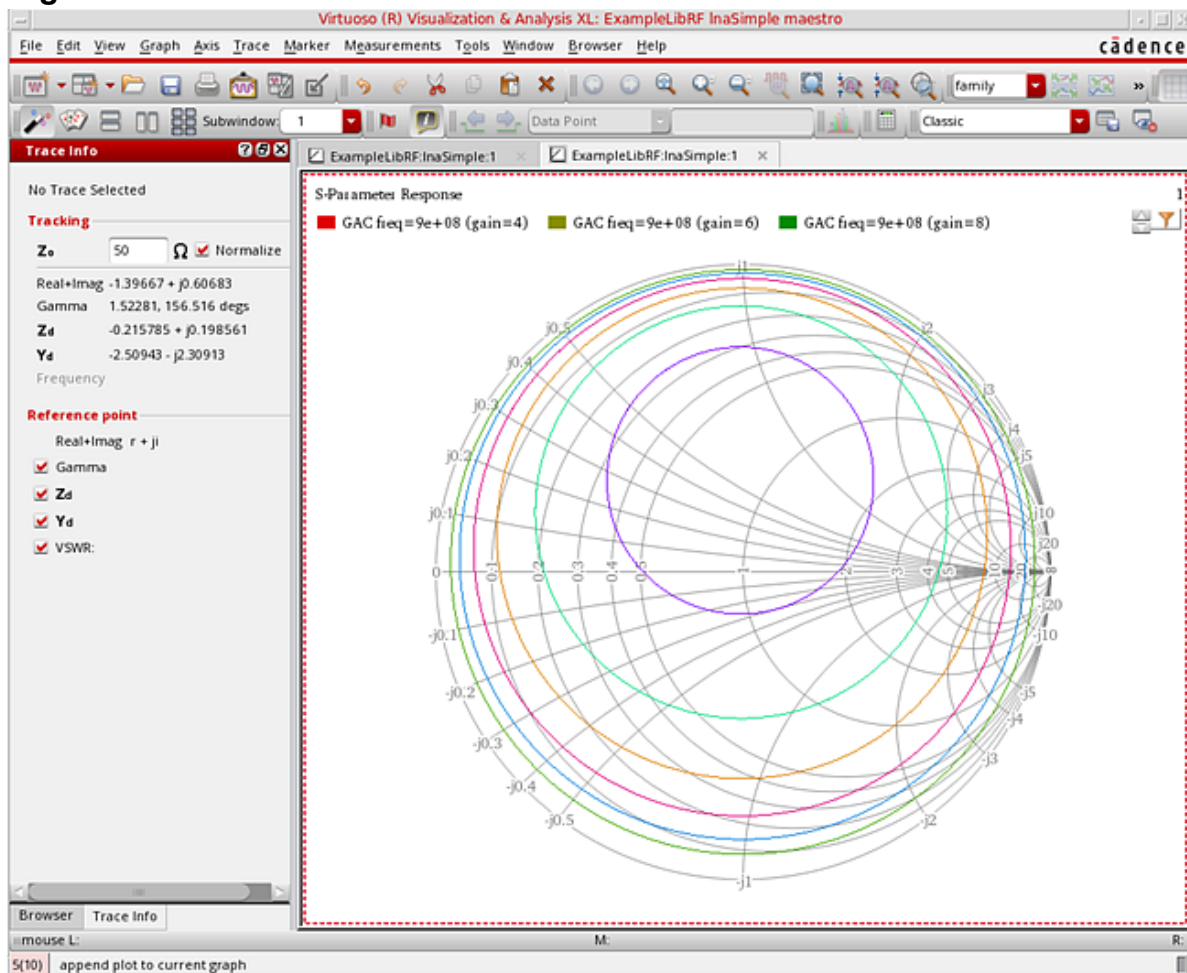
Add To Outputs ☐ Plot

> Press plot button on this form...

Close Help

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  $\Gamma_L$ . Thus, you can draw the power gain contours on the Smith chart of  $\Gamma_L$ . The location for the peak of the contour corresponds to  $\Gamma_L$  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  $\Gamma_L = 0$ .

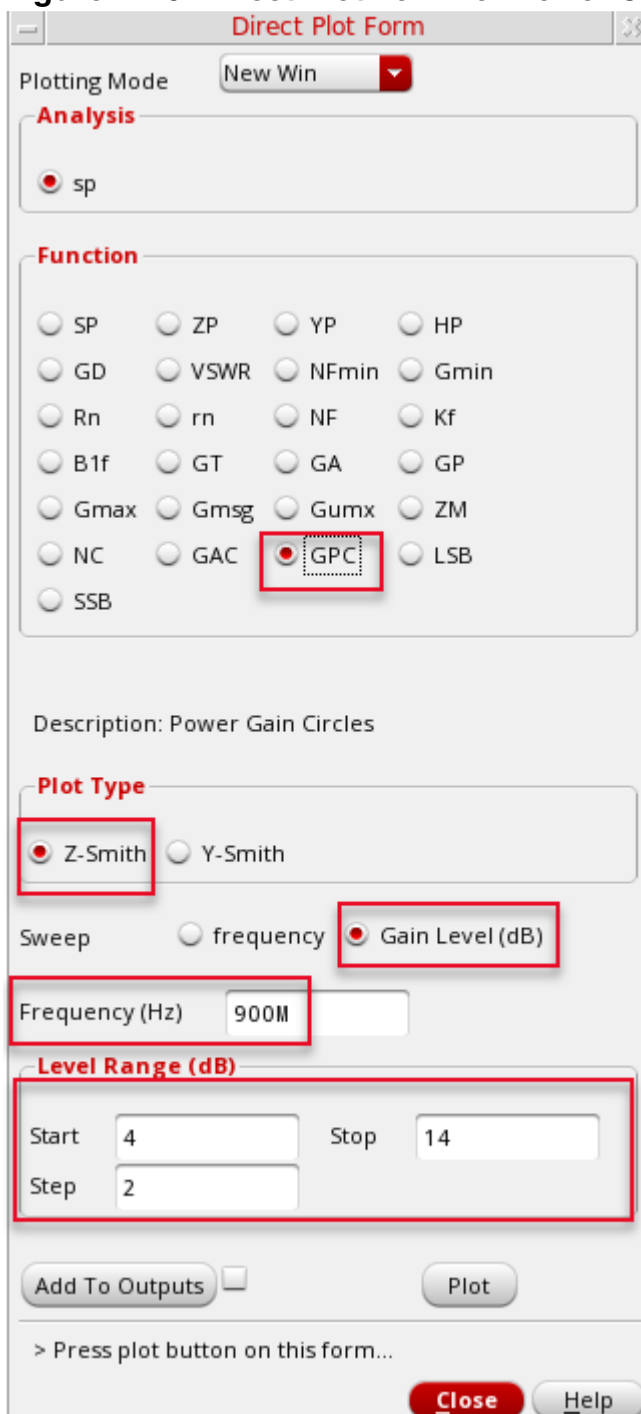
8. In the *sp Direct Plot Form*, select  $GPC$  in the *Function* section.

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

---

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 InaSimple 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**



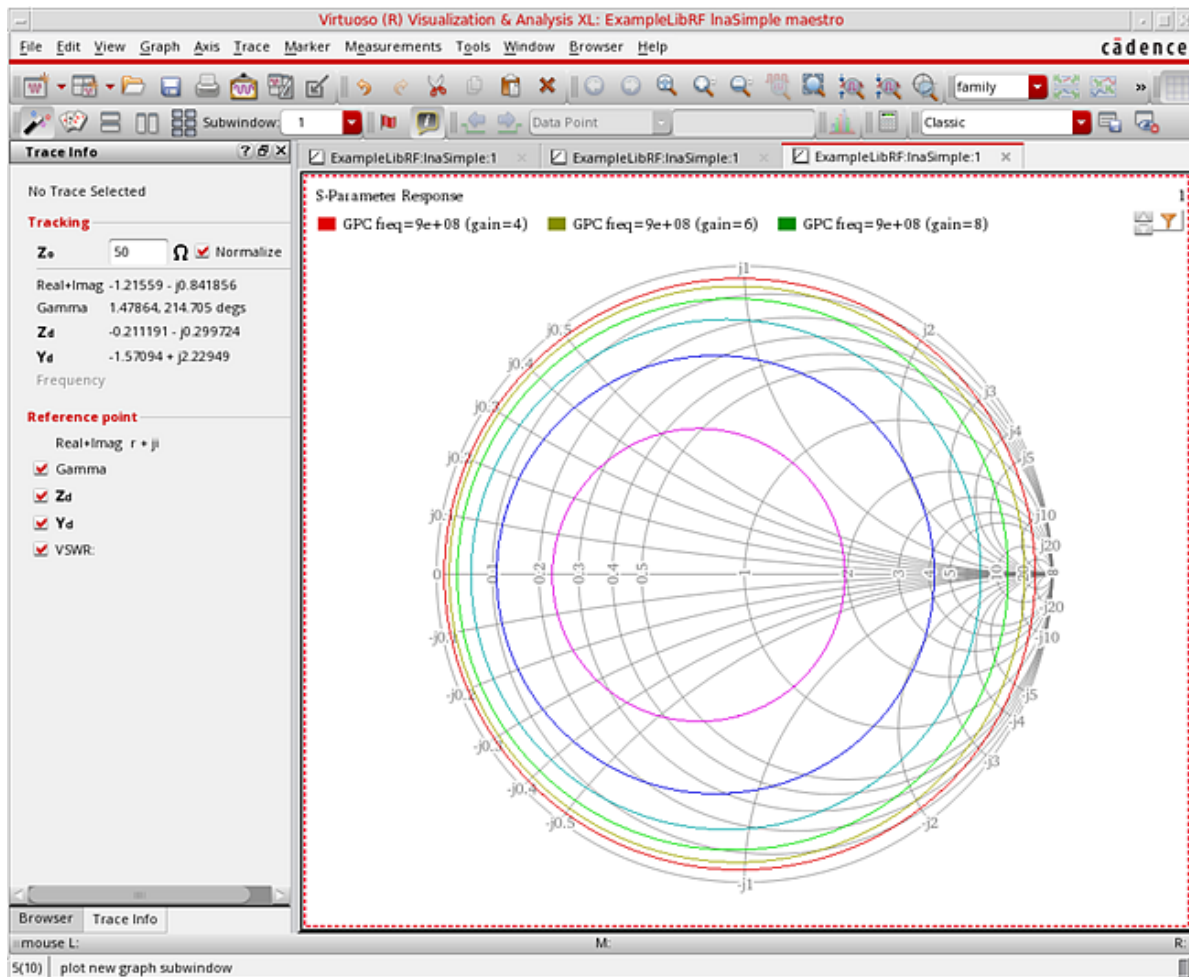
The image shows a software window titled "Direct Plot Form". It contains several sections for configuring a plot:

- Plotting Mode:** A dropdown menu set to "New Win".
- Analysis:** A radio button labeled "sp" is selected.
- Function:** A grid of radio buttons for various functions. "GPC" (Power Gain Circles) is selected and highlighted with a red box.
- Description:** The text "Power Gain Circles" is displayed.
- Plot Type:** Two radio buttons, "Z-Smith" and "Y-Smith". "Z-Smith" is selected and highlighted with a red box.
- Sweep:** Three radio buttons: "frequency", "Gain Level (dB)", and an unlabeled one. "Gain Level (dB)" is selected and highlighted with a red box.
- Frequency (Hz):** A text input field containing "900M", highlighted with a red box.
- Level Range (dB):** A section with three input fields: "Start" (4), "Stop" (14), and "Step" (2). This entire section is highlighted with a red box.
- Buttons:** "Add To Outputs" (disabled), "Plot", "Close", and "Help".
- Footer:** A message "> Press plot button on this form..." and a red "Close" button.

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

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

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.

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

**Figure 2-31 Equation for  $K_f$ , 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  $K_f$ .
2. Set the *Plotting Mode* to *New Window*.

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

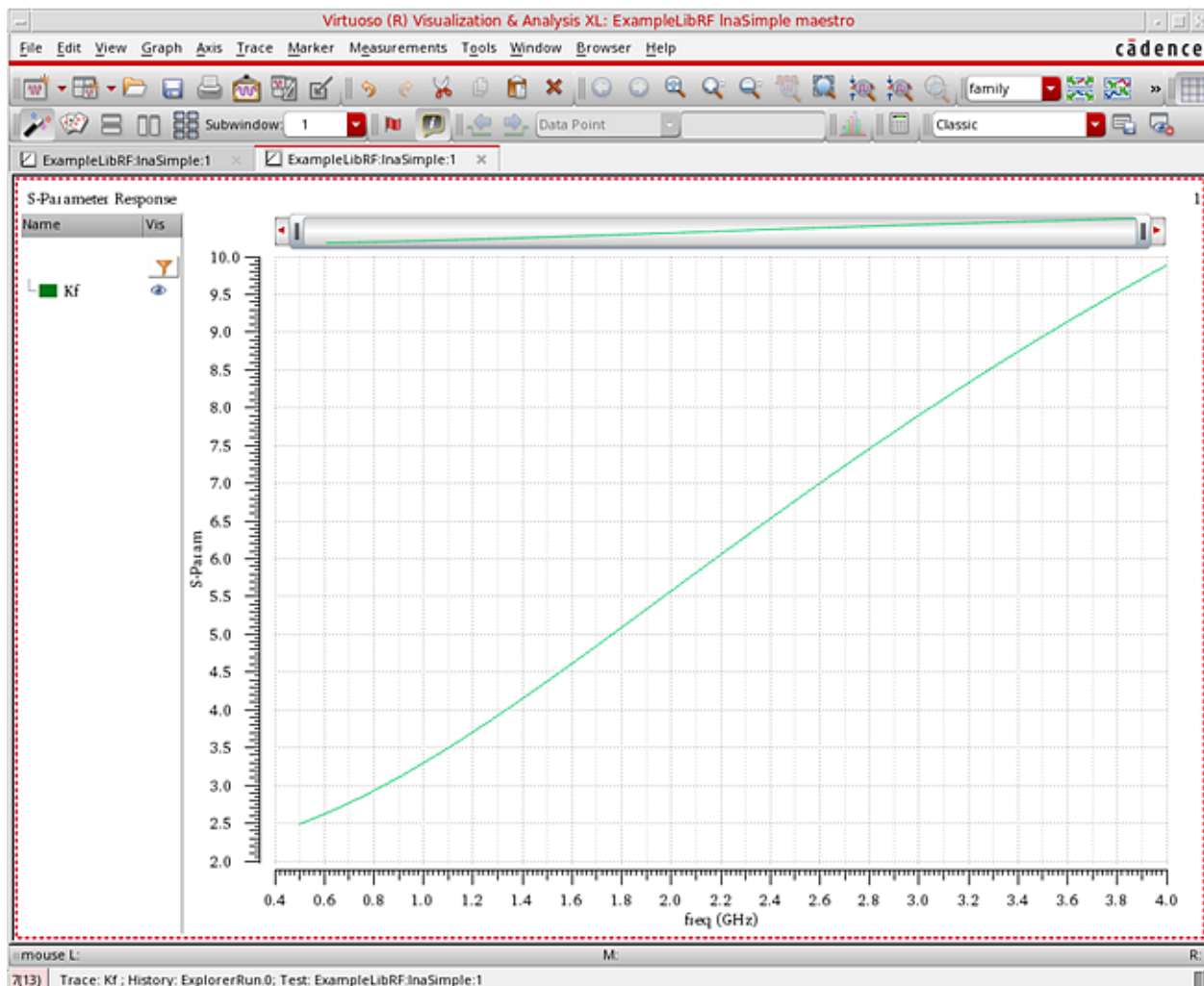
**Figure 2-32 Plotting Kf from the sp Direct Plot Form**

The image shows a software dialog box titled "Direct Plot Form". At the top, there is a "Plotting Mode" dropdown menu set to "New Win". Below this is a section labeled "Analysis" with a radio button selected for "sp". The main section is labeled "Function" and contains a grid of radio buttons for various parameters: SP, ZP, YP, HP, GD, VSWR, NFmin, Gmin, Rn, rn, NF, Kf, B1f, GT, GA, GP, Gmax, Gmsg, Gumx, ZM, NC, GAC, GPC, LSB, and SSB. The "Kf" radio button is selected and highlighted with a dashed border. Below the function grid, the text "Description: Stability Factor" is displayed. At the bottom left is an "Add To Outputs" button with an unchecked checkbox. At the bottom right is a "Plot" button. Below these buttons is a hint text: "> Press plot button on this form...". At the very bottom are "Close" and "Help" buttons.

**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  $K_f$  is greater than 1 for all frequencies viewed, indicating that the circuit is stable at these frequencies. As the coupling ( $S_{12}$ ) 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).

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

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

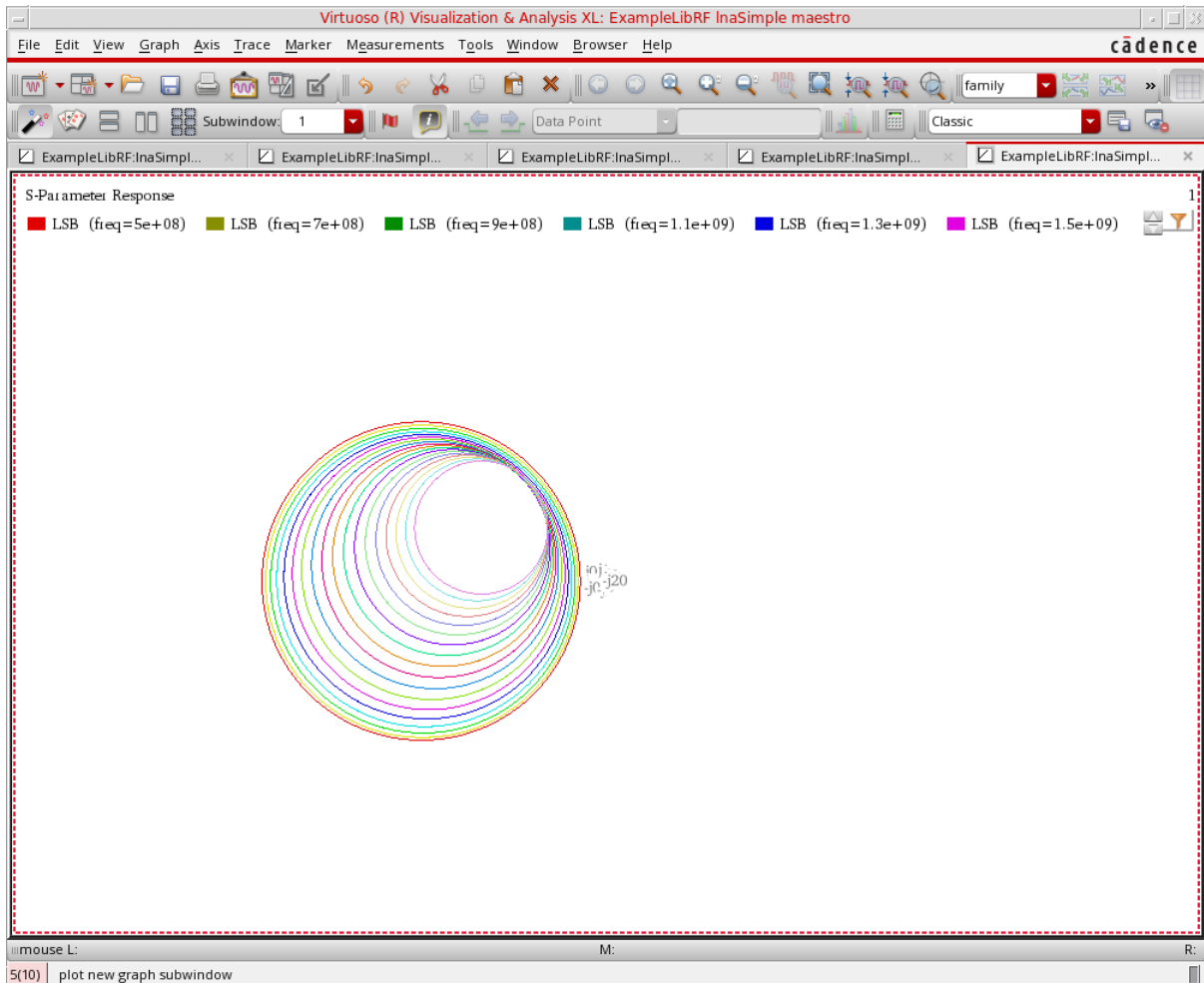
**Figure 2-34 Direct Plot Form for Plotting Load Stability Circles**

The screenshot shows the 'Direct Plot Form' window. At the top, the title bar says 'Direct Plot Form'. Below it, the 'Plotting Mode' is set to 'New Win'. The 'Analysis' section has a radio button selected for 'sp'. The 'Function' section contains a grid of radio buttons for various functions: SP, ZP, YP, HP, GD, VSWR, NFmin, Gmin, Rn, rn, NF, Kf, B1f, GT, GA, GP, Gmax, Gmsg, Gumx, ZM, NC, GAC, GPC, and LSB. The 'LSB' option is selected and highlighted with a red box. Below the function grid, the 'Description' is 'Load Stability Circles'. The 'Plot Type' section has two radio buttons: 'Z-Smith' (selected and highlighted with a red box) and 'Y-Smith'. The 'Frequency Range (Hz)' section has three input fields: 'Start' (500M), 'Stop' (4G), and 'Step' (200M). This entire section is highlighted with a red box. At the bottom, there are buttons for 'Add To Outputs' (disabled), 'Plot', and 'Close'. A red box highlights the 'Plot' button. Below the 'Plot' button, there is a text prompt: '> Press plot button on this form...'. At the very bottom, there are 'Close' and 'Help' buttons.

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

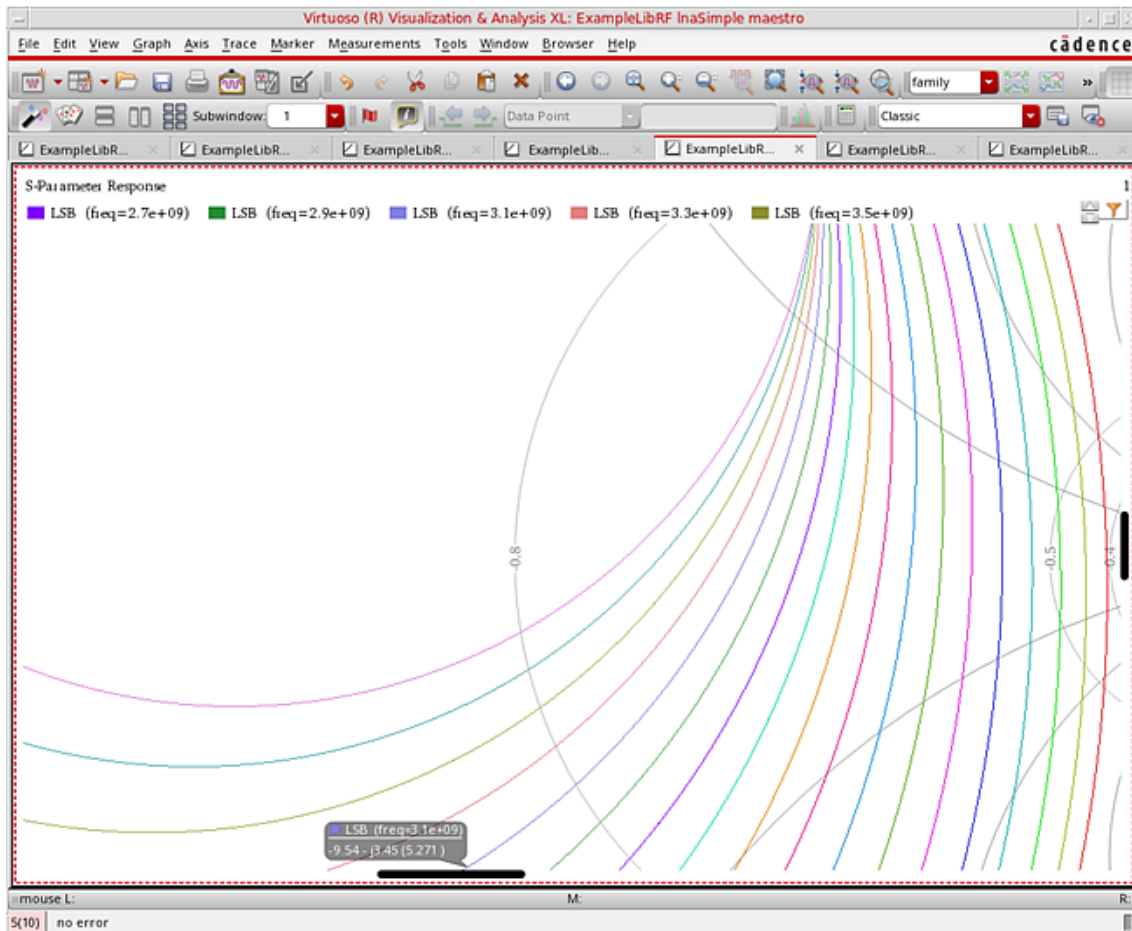
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

**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.

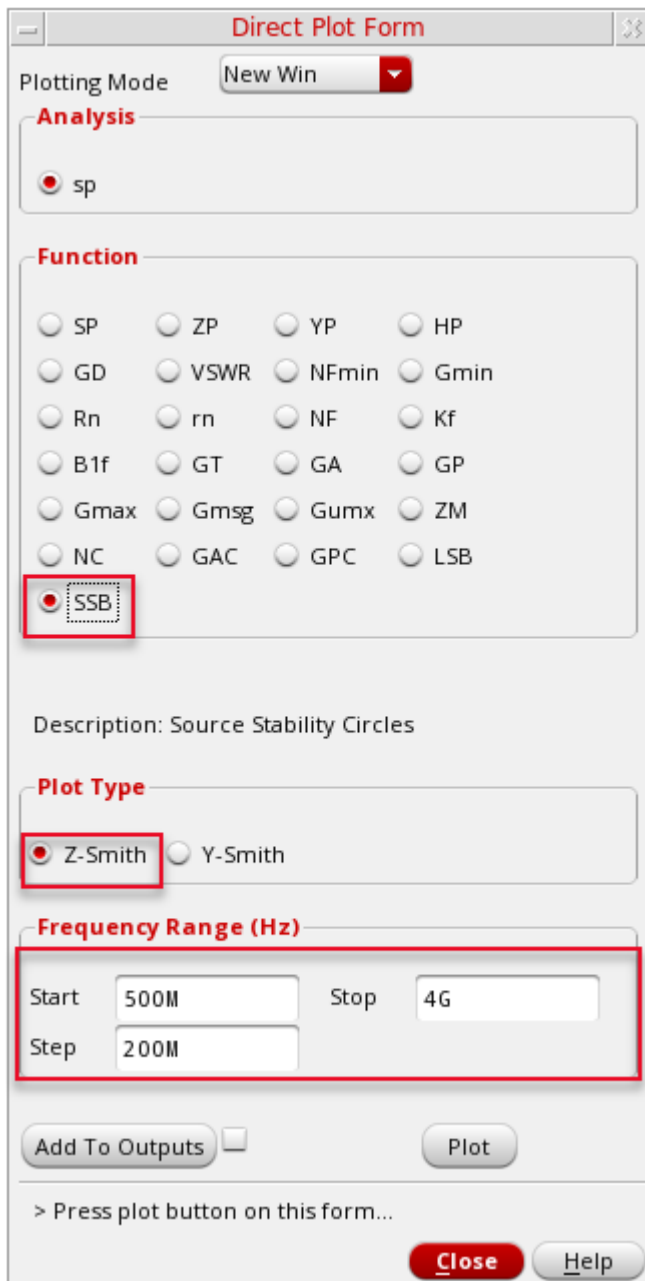
**Figure 2-36 Zoomed in Load Stability Circles**



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**

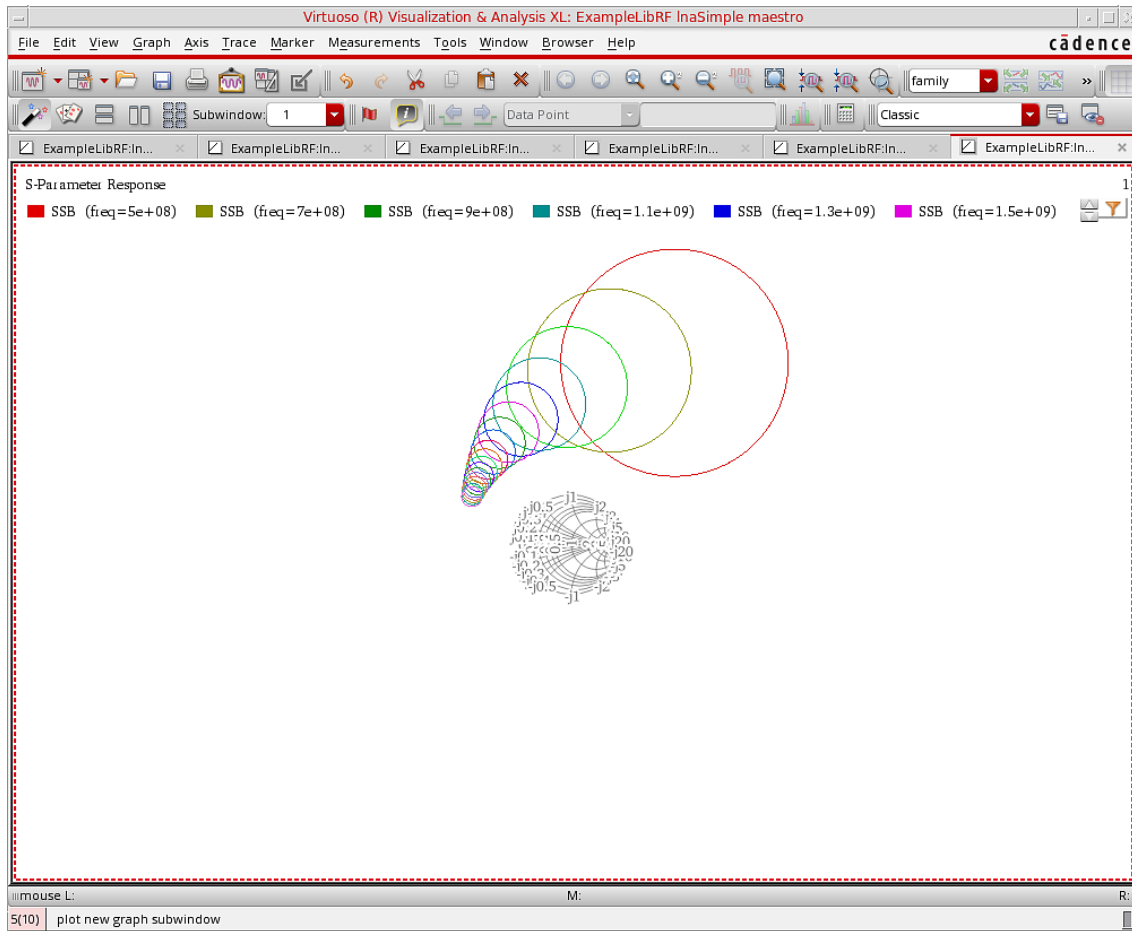


The image shows a software window titled "Direct Plot Form". At the top, there is a "Plotting Mode" dropdown menu set to "New Win". Below this is the "Analysis" section with a radio button selected for "sp". The "Function" section contains a grid of radio buttons for various parameters: SP, ZP, YP, HP, GD, VSWR, NFmin, Gmin, Rn, rn, NF, Kf, B1f, GT, GA, GP, Gmax, Gmsg, Gumx, ZM, NC, GAC, GPC, and LSB. The "SSB" option is highlighted with a red box. Below the function list is the text "Description: Source Stability Circles". The "Plot Type" section has two radio buttons: "Z-Smith" (selected and highlighted with a red box) and "Y-Smith". The "Frequency Range (Hz)" section is highlighted with a red box and contains three input fields: "Start" with the value "500M", "Stop" with the value "4G", and "Step" with the value "200M". At the bottom, there are two buttons: "Add To Outputs" (with a small square icon) and "Plot". Below these buttons is a text prompt "> Press plot button on this form...". At the very bottom are two buttons: "Close" (highlighted in red) and "Help".

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

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

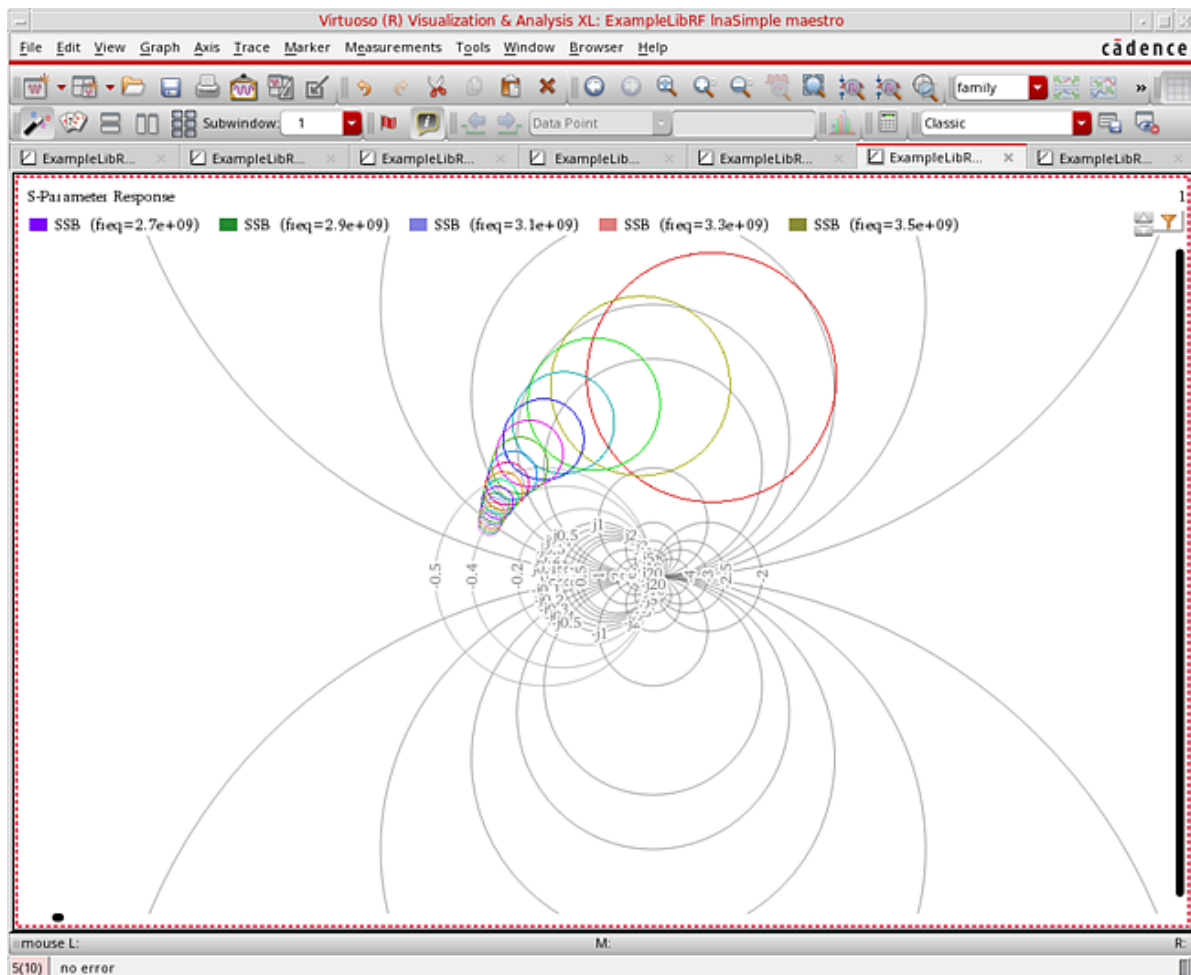
**Figure 2-38 Source Stability Circles for InaSimple**



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.

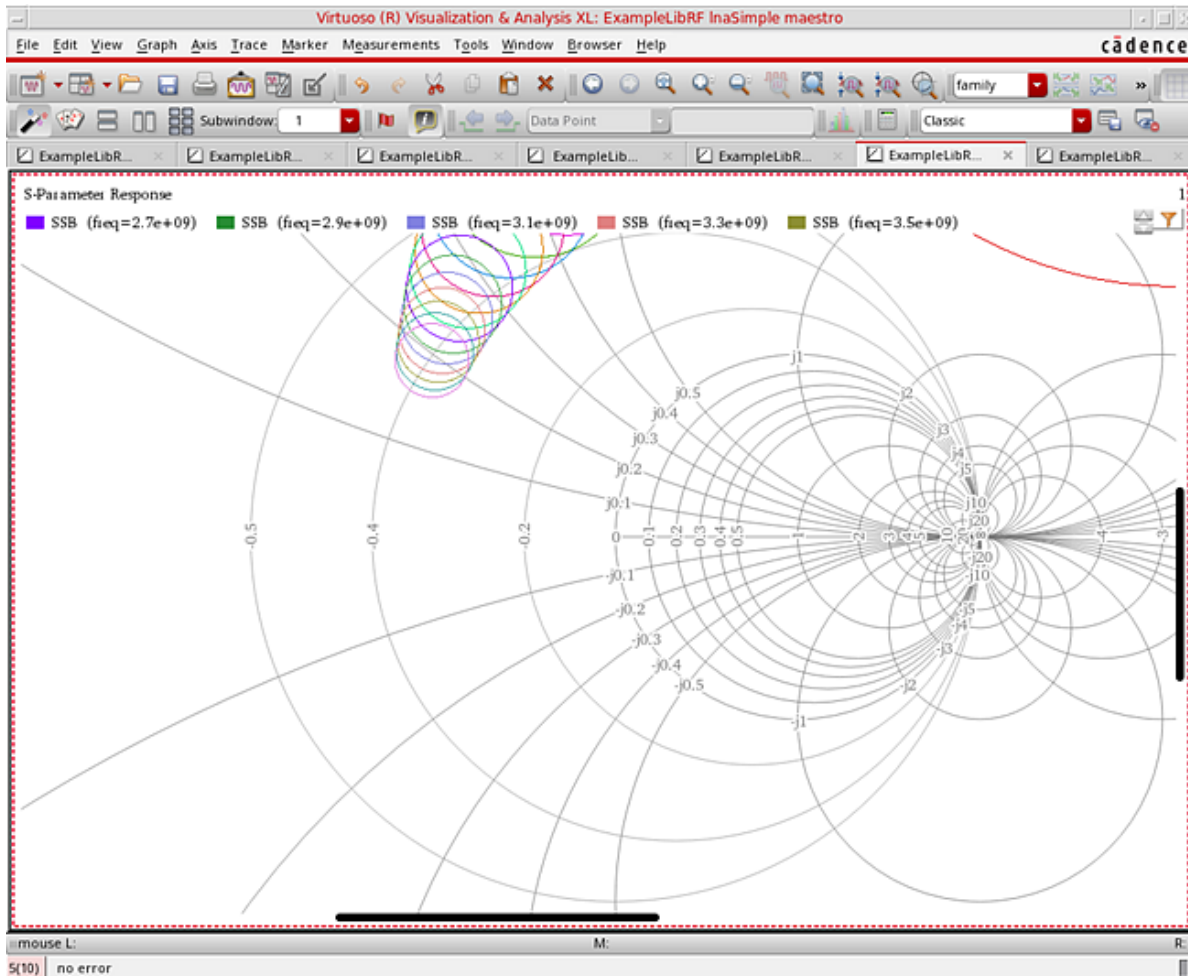


### Figure 2-39 Zoom in Source Stability Circles



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  $\Gamma_{\text{out}} = 1$  out on the Smith chart of  $\Gamma_S$ . The output stability circle draws the circle  $\Gamma_{\text{in}} = 1$  on the Smith chart of  $\Gamma_L$ .

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.

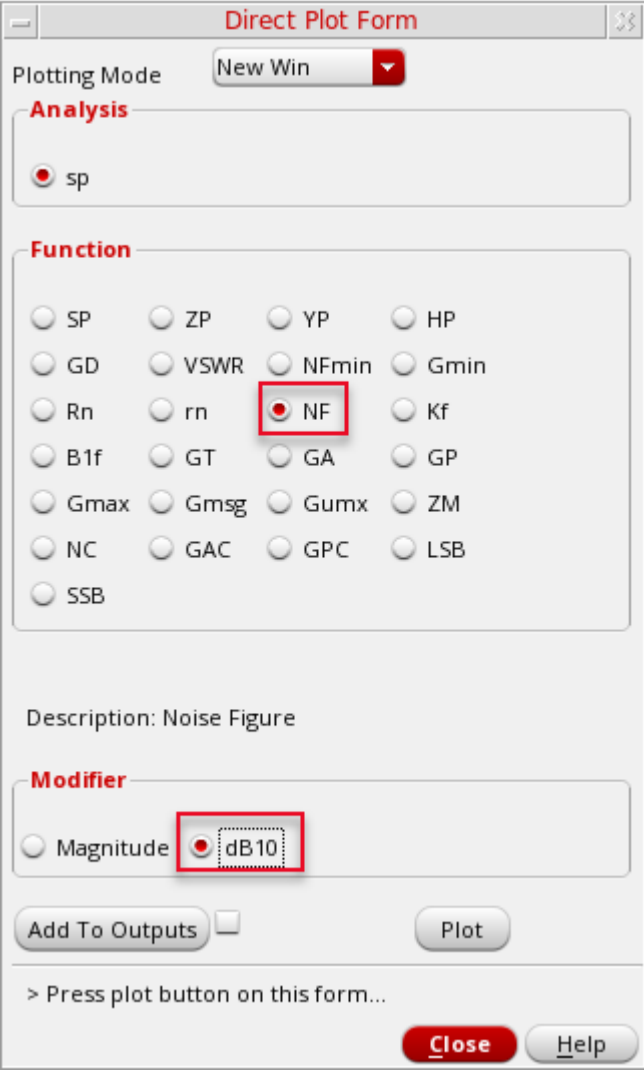
### Linear 2-port noise analysis (NF, NFmin) and Noise circles

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.

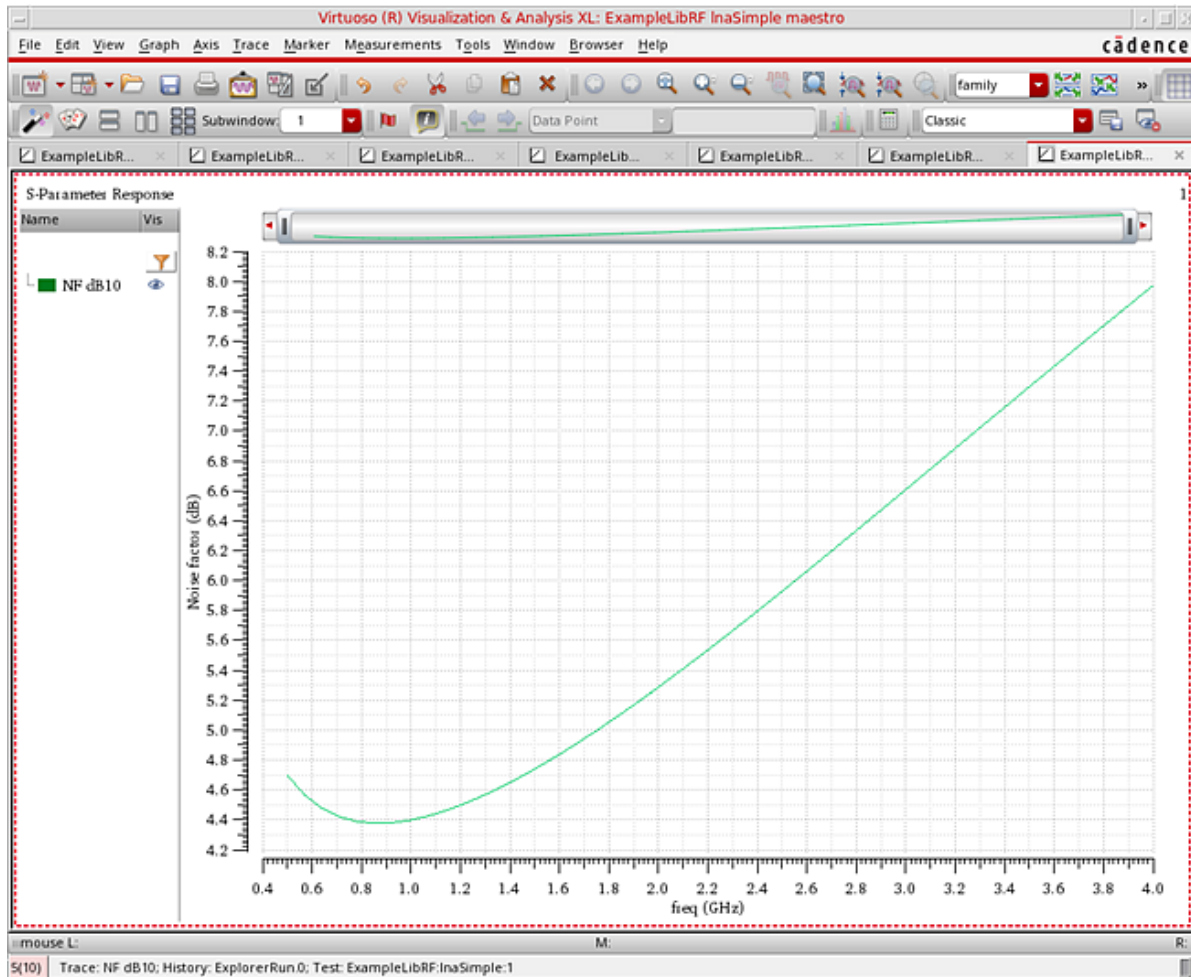
**Figure 2-41 sp Direct Plot Form for Plotting Noise Figure**



The image shows a software window titled "Direct Plot Form". At the top, there is a "Plotting Mode" dropdown menu set to "New Win". Below this is the "Analysis" section with a radio button labeled "sp" selected. The "Function" section contains a grid of radio buttons for various parameters: SP, ZP, YP, HP, GD, VSWR, NFmin, Gmin, Rn, rn, NF (which is selected and highlighted with a red box), Kf, B1f, GT, GA, GP, Gmax, Gmsg, Gumx, ZM, NC, GAC, GPC, LSB, and SSB. Below the function grid, the text "Description: Noise Figure" is displayed. The "Modifier" section has a radio button labeled "Magnitude" and a dropdown menu set to "dB10" (highlighted with a red box). At the bottom, there are two buttons: "Add To Outputs" (with a small square icon) and "Plot". Below these buttons is a text prompt "> Press plot button on this form...". At the very bottom are two buttons: "Close" (in red) and "Help".

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

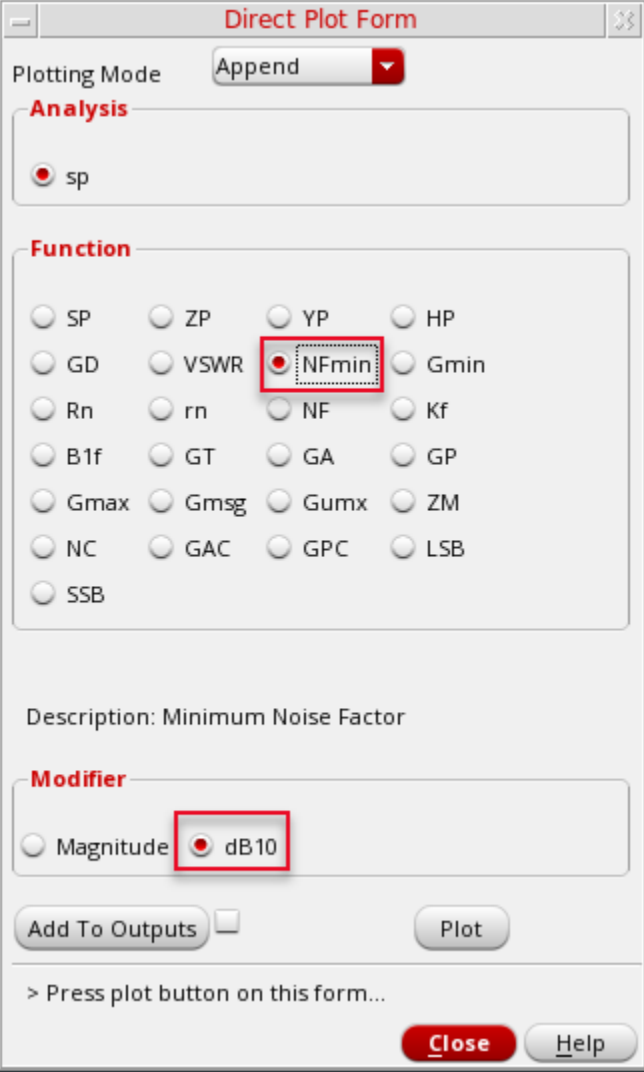
Figure 2-42 InaSimple Noise Figure Plot



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:

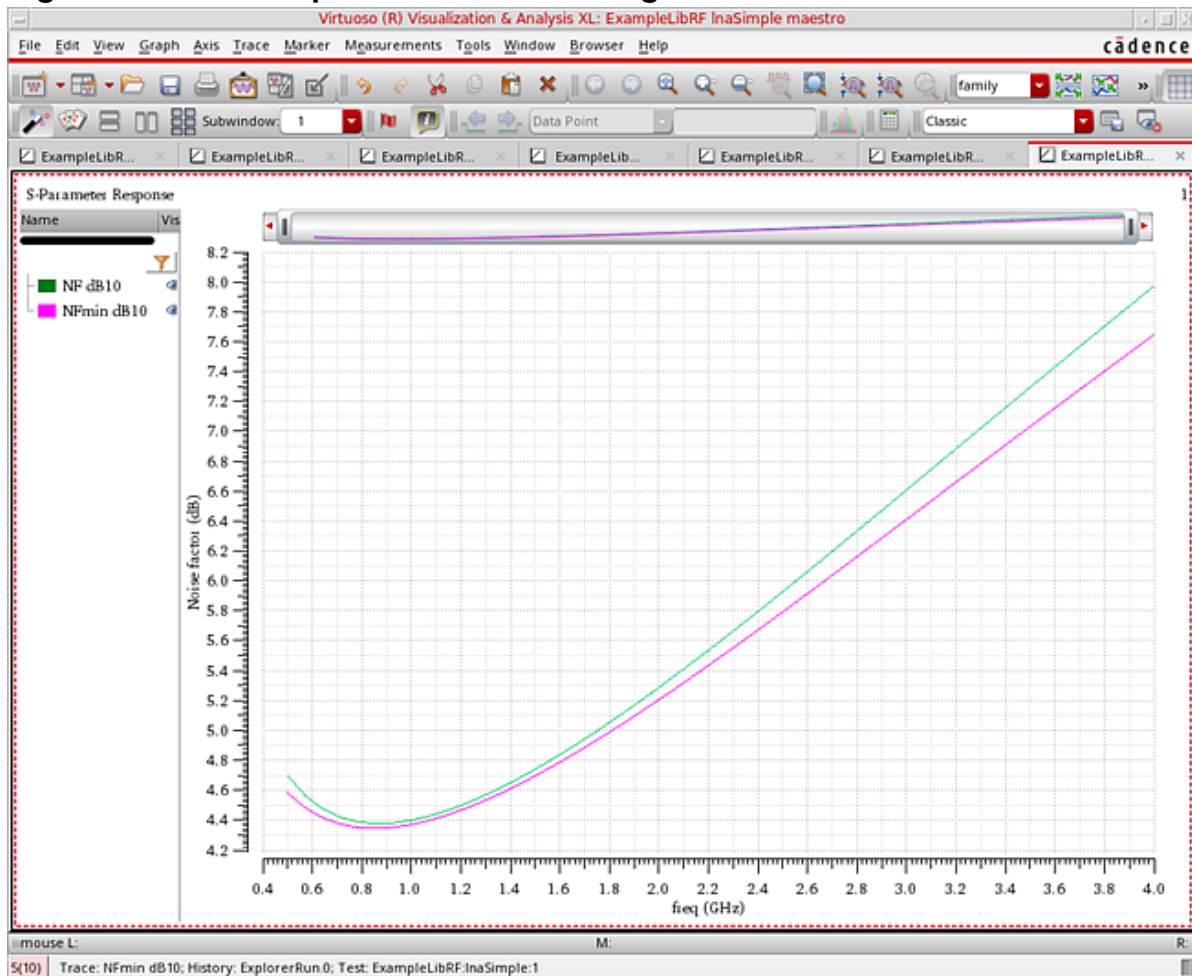
**Figure 2-43** sp Direct Plot Form for Minimum Noise Figure



The image shows a software dialog box titled "Direct Plot Form". At the top, there is a "Plotting Mode" dropdown menu set to "Append". Below this is a section labeled "Analysis" with a radio button selected for "sp". The next section is labeled "Function" and contains a grid of radio buttons for various parameters: SP, ZP, YP, HP, GD, VSWR, NFmin (which is selected and highlighted with a red box), Gmin, Rn, rn, NF, Kf, B1f, GT, GA, GP, Gmax, Gmsg, Gumx, ZM, NC, GAC, GPC, LSB, and SSB. Below the function section is a "Description" field containing the text "Minimum Noise Factor". The "Modifier" section has two radio buttons: "Magnitude" and "dB10" (which is selected and highlighted with a red box). At the bottom, there are two buttons: "Add To Outputs" (with a small square icon) and "Plot". Below these buttons is a text prompt "> Press plot button on this form...". At the very bottom are two buttons: "Close" (in red) and "Help".

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

**Figure 2-44 InaSimple 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.

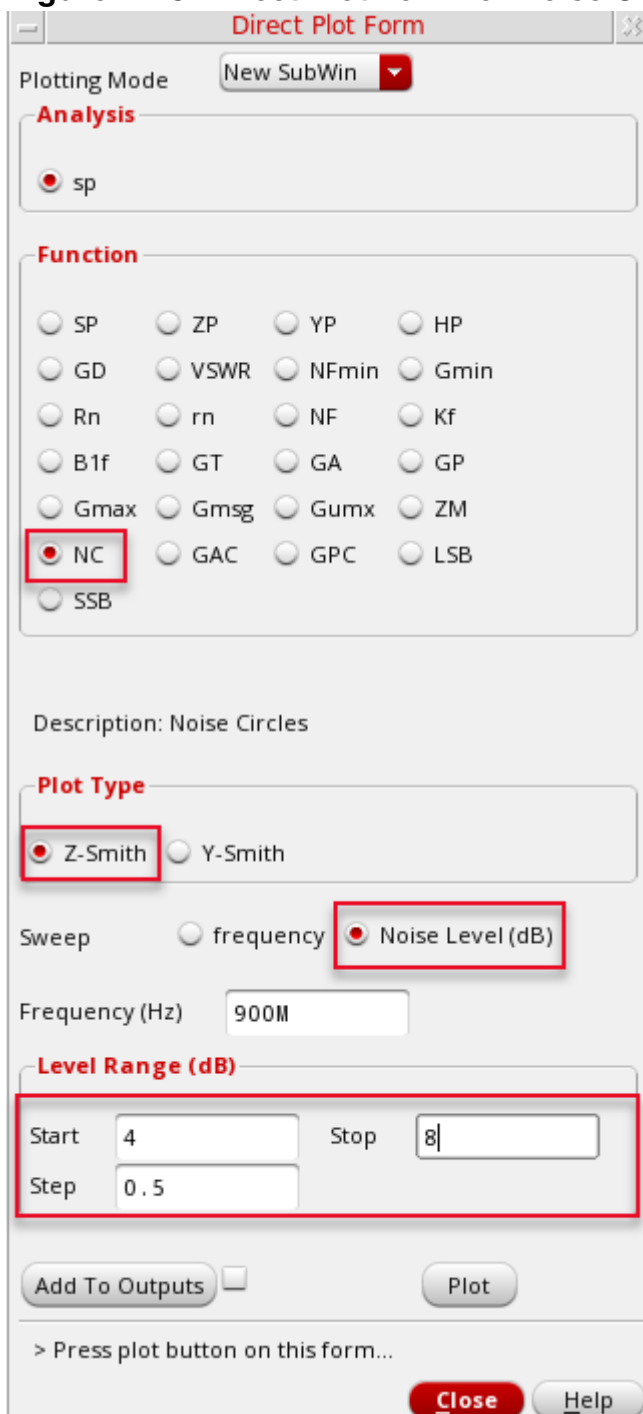
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

12. In the *Frequency(Hz)* field, type 900M. This is the operating frequency of the InaSimple 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**



The image shows the 'Direct Plot Form' dialog box for Noise Circles. The 'Plotting Mode' is set to 'New SubWin'. Under the 'Analysis' section, 'sp' is selected. In the 'Function' section, 'NC' is selected and highlighted with a red box. The 'Description' is 'Noise Circles'. In the 'Plot Type' section, 'Z-Smith' is selected and highlighted with a red box. The 'Sweep' section has 'Noise Level (dB)' selected and highlighted with a red box, while 'frequency' is unselected. The 'Frequency (Hz)' is set to '900M'. In the 'Level Range (dB)' section, 'Start' is 4, 'Stop' is 8, and 'Step' is 0.5, all highlighted with a red box. At the bottom, there is an 'Add To Outputs' checkbox, a 'Plot' button, and a 'Close' button. A note at the bottom says '> Press plot button on this form...'

Direct Plot Form

Plotting Mode: New SubWin

**Analysis**

☒ sp

**Function**

☐ SP ☐ ZP ☐ YP ☐ HP  
☐ GD ☐ VSWR ☐ NFmin ☐ Gmin  
☐ Rn ☐ rn ☐ NF ☐ Kf  
☐ B1f ☐ GT ☐ GA ☐ GP  
☐ Gmax ☐ Gmsg ☐ Gumx ☐ ZM  
☒ NC ☐ GAC ☐ GPC ☐ LSB  
☐ SSB

Description: Noise Circles

**Plot Type**

☒ Z-Smith ☐ Y-Smith

Sweep: ☐ frequency ☒ Noise Level (dB)

Frequency (Hz): 900M

**Level Range (dB)**

Start: 4 Stop: 8  
Step: 0.5

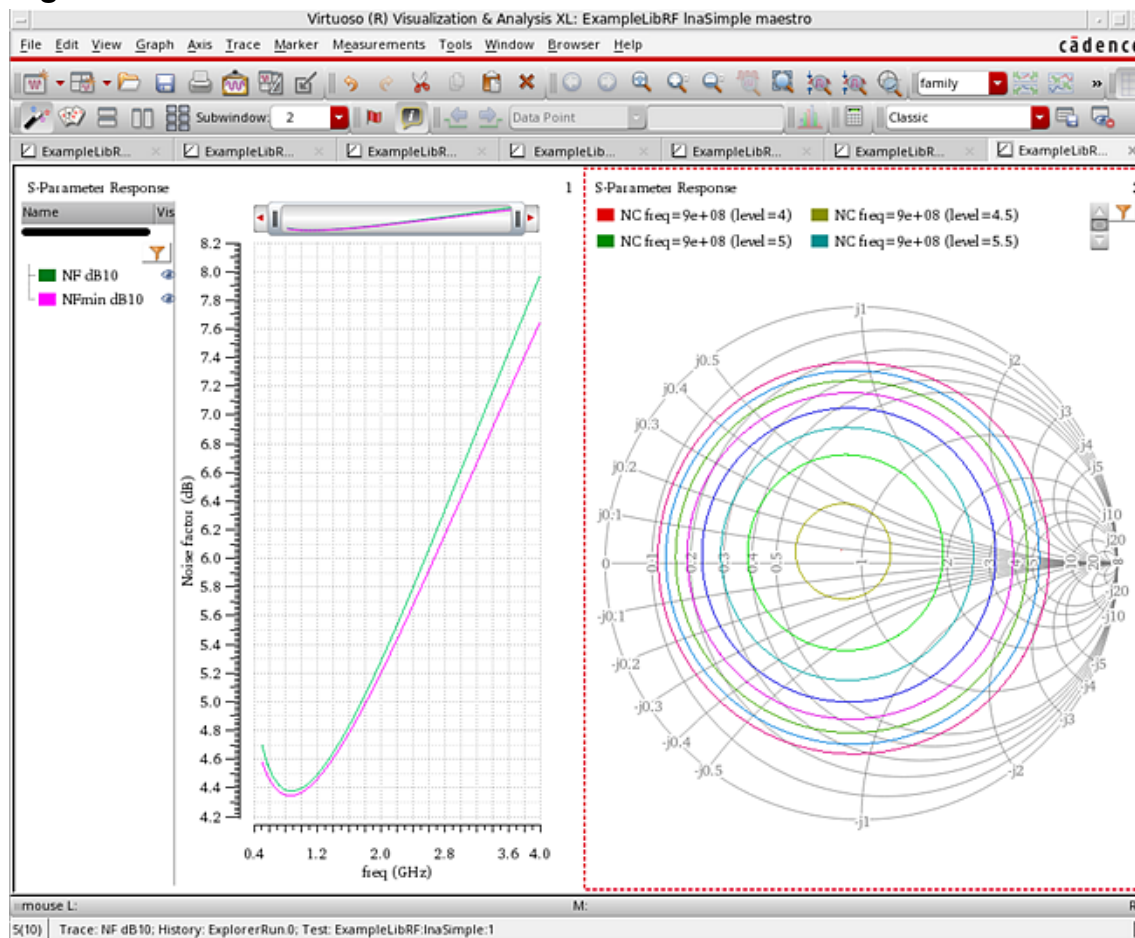
Add To Outputs ☐ Plot

> Press plot button on this form...

Close Help

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

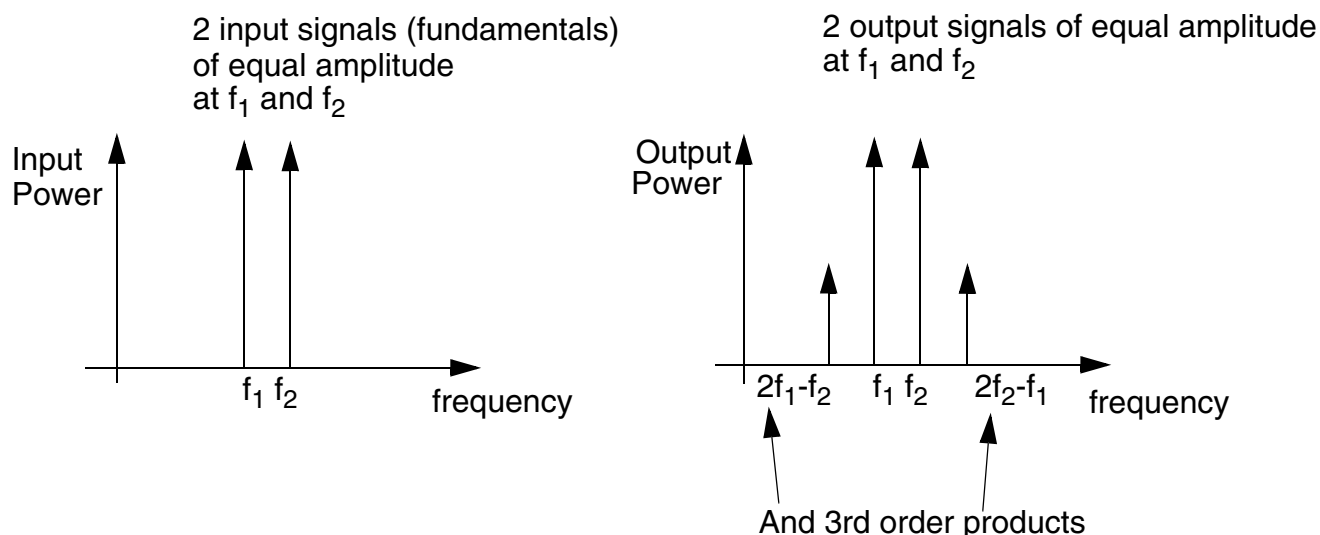
In this section, you have looked at common measurements made on Low Noise Amplifier circuits, specifically Gain, Stability, and Noise. For other examples of Measurements, see [Chapter 3: Frequency Domain Analyses: Harmonic Balance](#) in the Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel 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  $f_1$  and  $f_2$ ). 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.

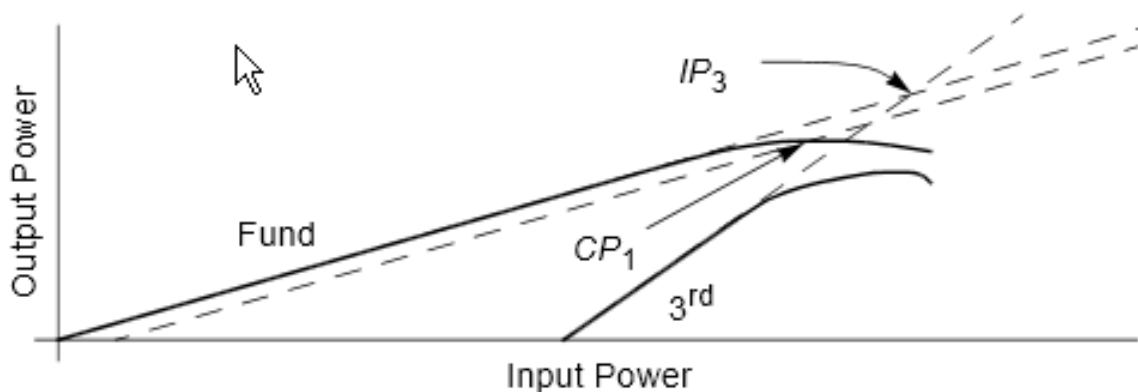


Assuming the input amplitudes are equal, the output first order terms will have the same amplitude and the output third order terms will also have the same amplitude.

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<sub>1</sub> is the 1dB compression point  
IP<sub>3</sub> 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 InaSimple LNACircuit in the Schematic Window

1. In the CIW, choose *File – Open*.

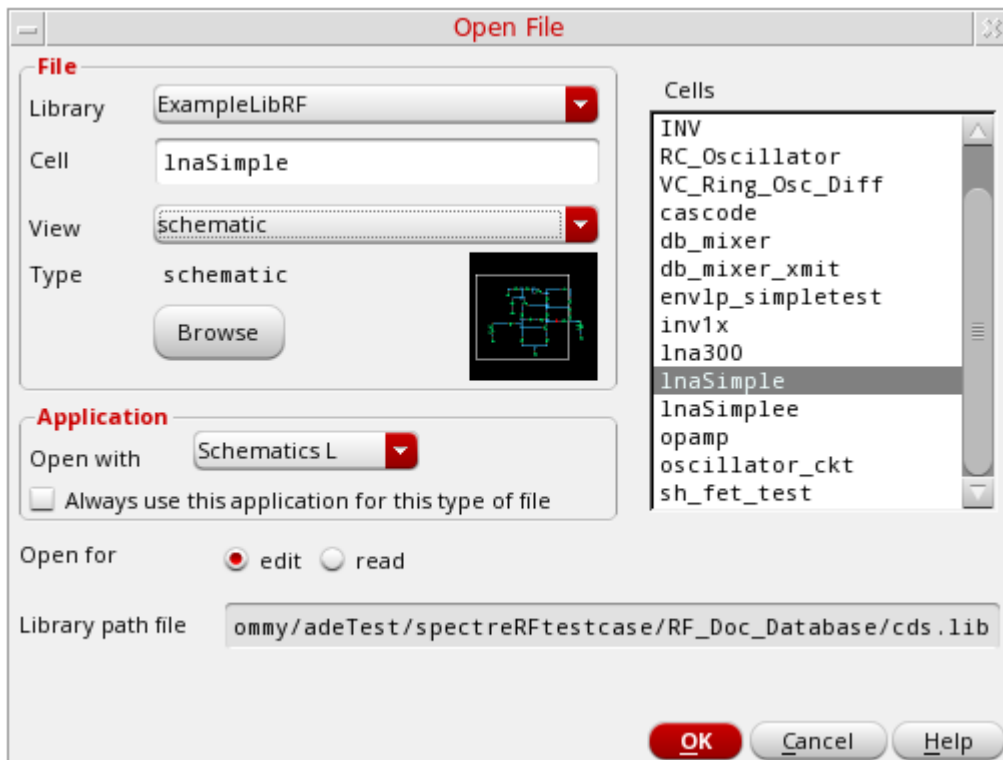
The *Open File* form is displayed.

2. Choose *ExampleLibRF* from the *Library* drop-down list.
3. Choose *InaSimple* 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.

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

---

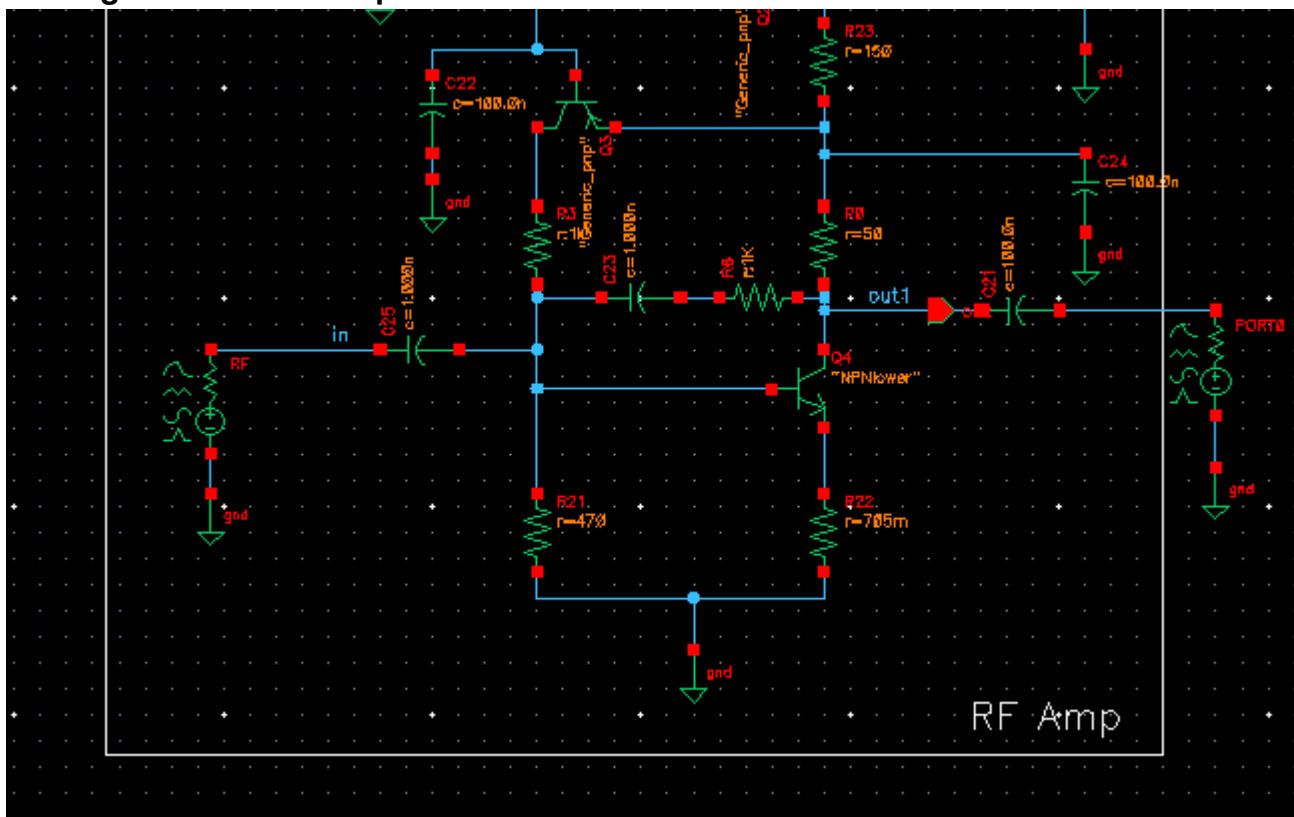
**Figure 2-48 Open File Form**



5. Click **OK**.

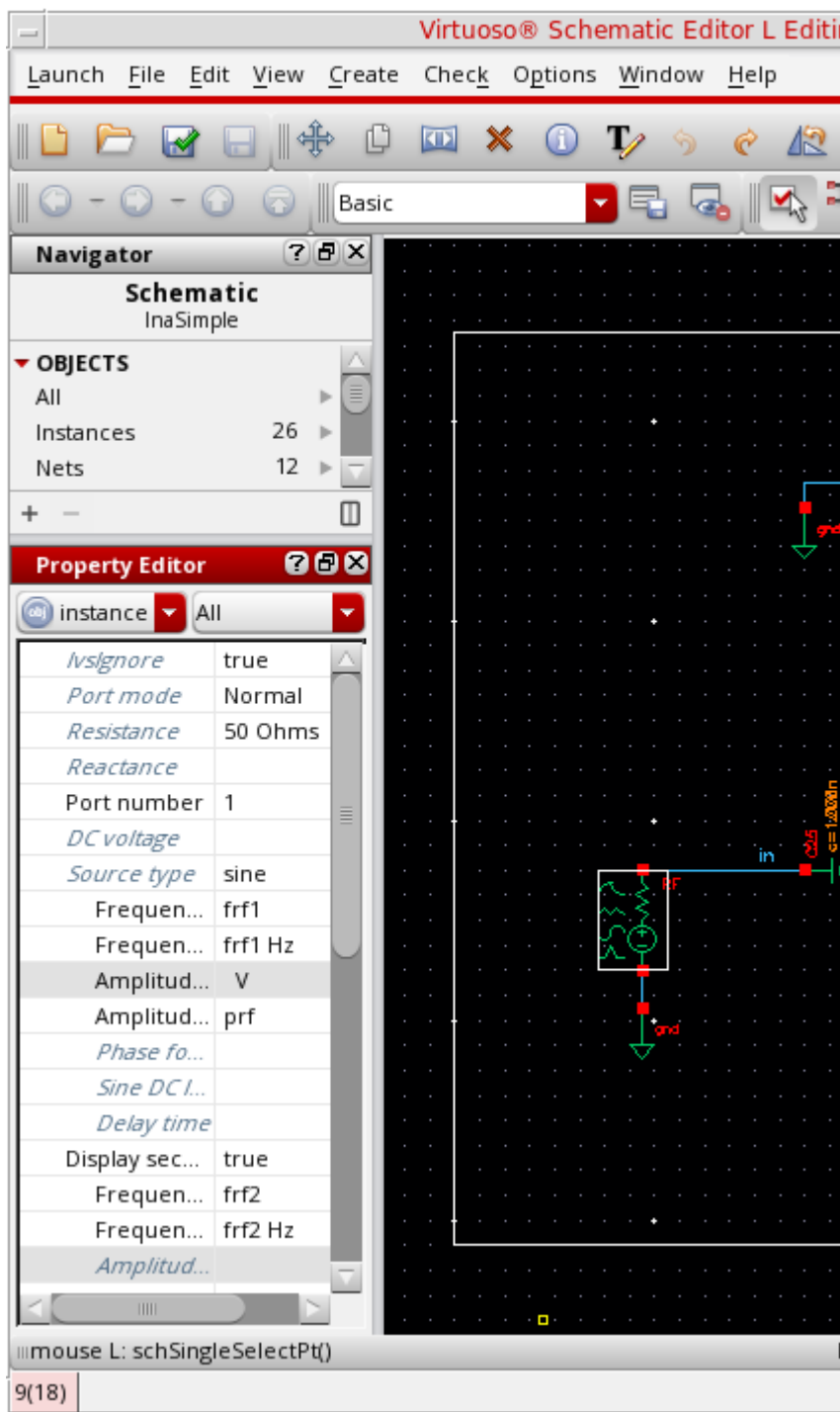
The Schematic window for InaSimple is displayed.

Figure 2-49 InaSimple 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.



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

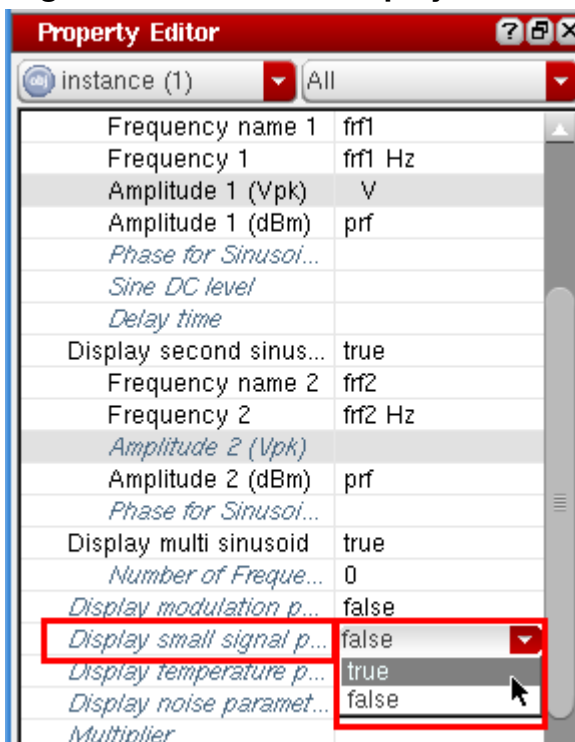
- a. *Source type* `sine` means generate large signals for the time domain (transient, pss-shooting, qpss-shooting, envlp-shooting, and so on) or frequency domain (HB or envlp-HB) analyses.

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 be 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.

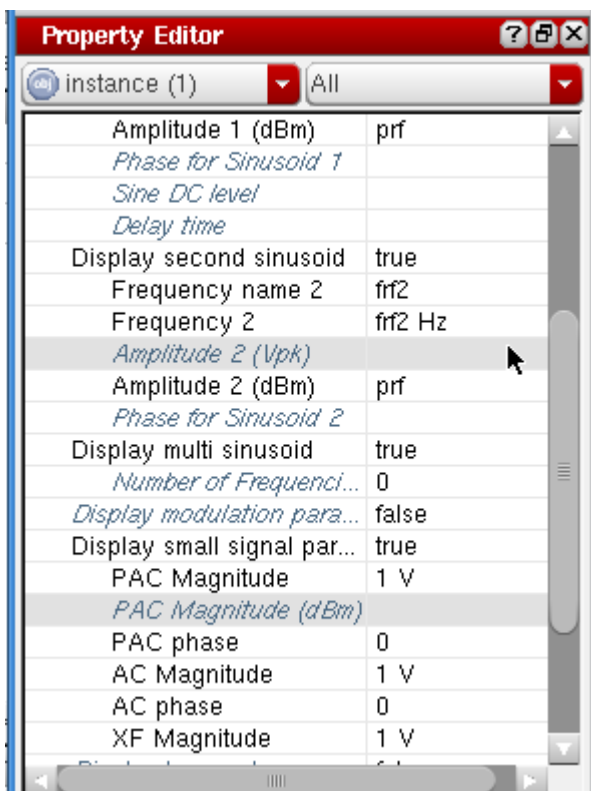
The amplitude for the small-signal analyses AC and PAC are set in the source. To view the small signal values, scroll down the form to the *Display small signal parameter* drop-down list and select *true*. The amplitude for AC and PAC can be either in Volts peak or dBm, but not both at the same time.

**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**

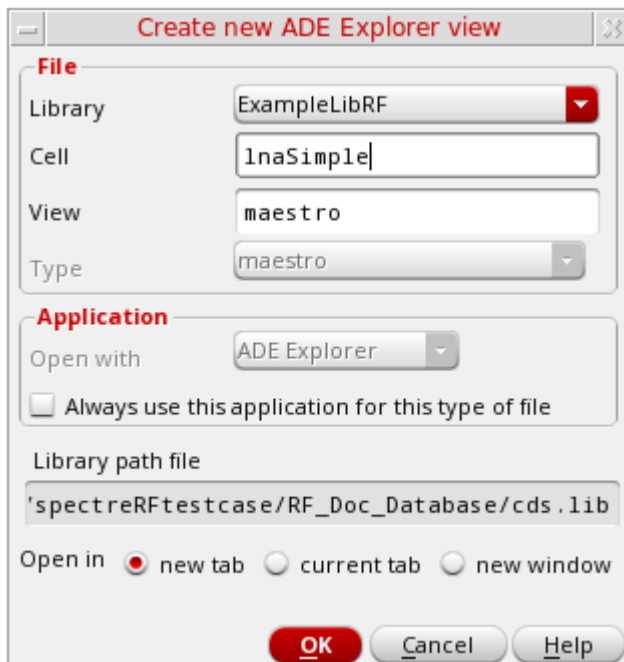


The frequency of these signals will be set in the small-signal setup form in ADE Explorer. The amplitude for PAC can be either in volts peak or in dBm, again with the form not allowing both to be set at the same time.

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**

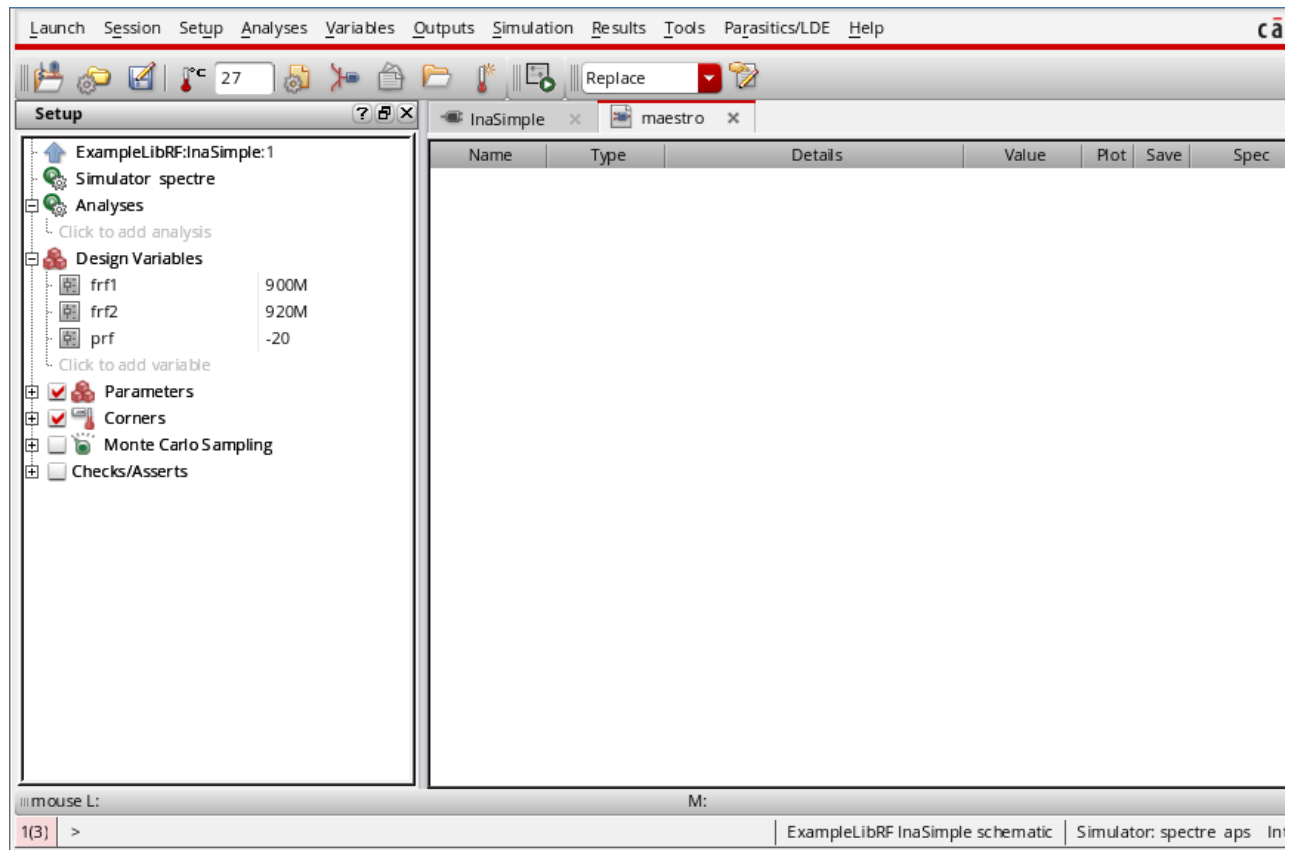


9. Leave each option to the default selections and click *OK*.

The Virtuoso ADE Explorer window opens.

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

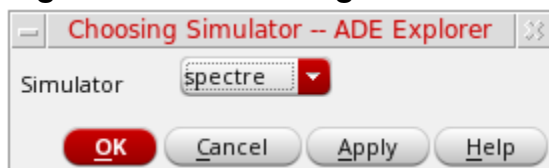
**Figure 2-54 Analog Design Environment Window**



### 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.

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

**Figure 2-56 High Performance Simulation Options**

High-Performance Simulation Options

Simulation Performance Mode: ☐ Spectre ☒ APS ☐ Spectre X ☐ XPS MS

**General**

Accuracy + Speed

Error Preset: ☒ Do not override ☐ Liberal ☐ Moderate ☐ Conservative

APS Options

Use ++aps: ☐

Multi-Threading

☒ Auto ☐ Disable ☐ Manual # Threads:

☐ Processor affinity (e.g. 0-3 or 0,2,4,6):

**Post-Layout Settings**

Enable Post-Layout Optimization: ☐

Post-Layout Preset Mode: ☒ Default ☐ High Precision Analog ☐ Ultra Precision Analog ☐ Legacy RCR ☐ Legacy-RF RCR

Instance Preservation: ☒ None ☐ Selected

Selected Instances:

Savefilter: ☐ none ☐ rc

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.

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

Figure 2-57 Save Options Form

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvpub ☐ lv ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

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

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

---

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.

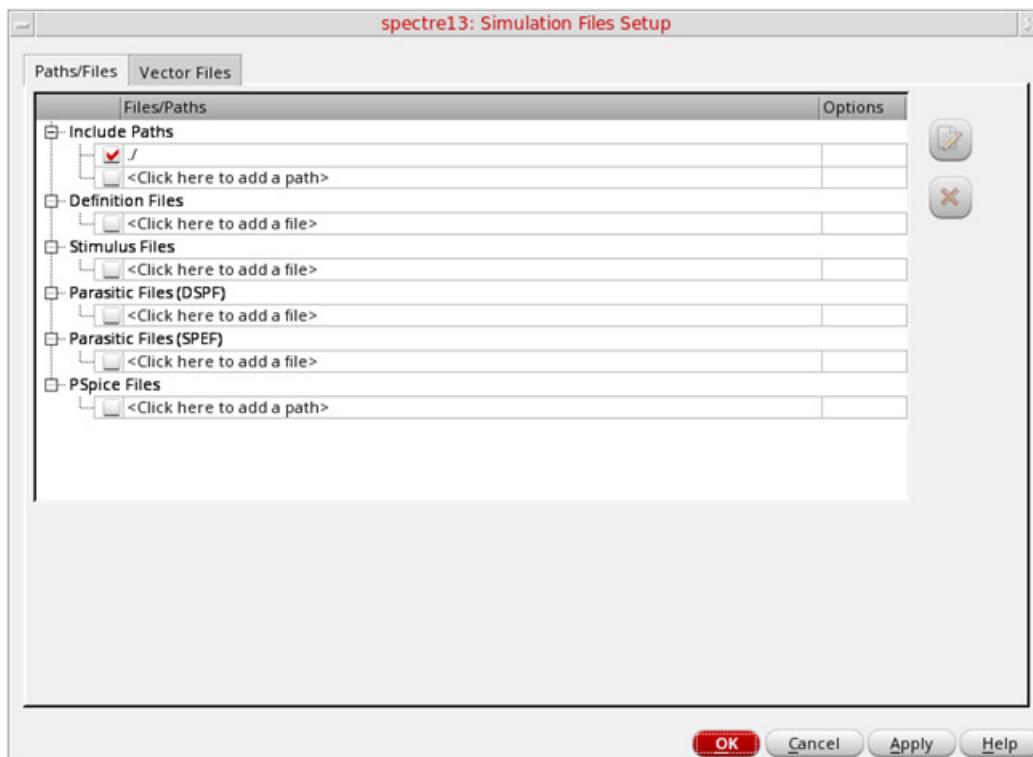
9. Click *OK*.

### 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.

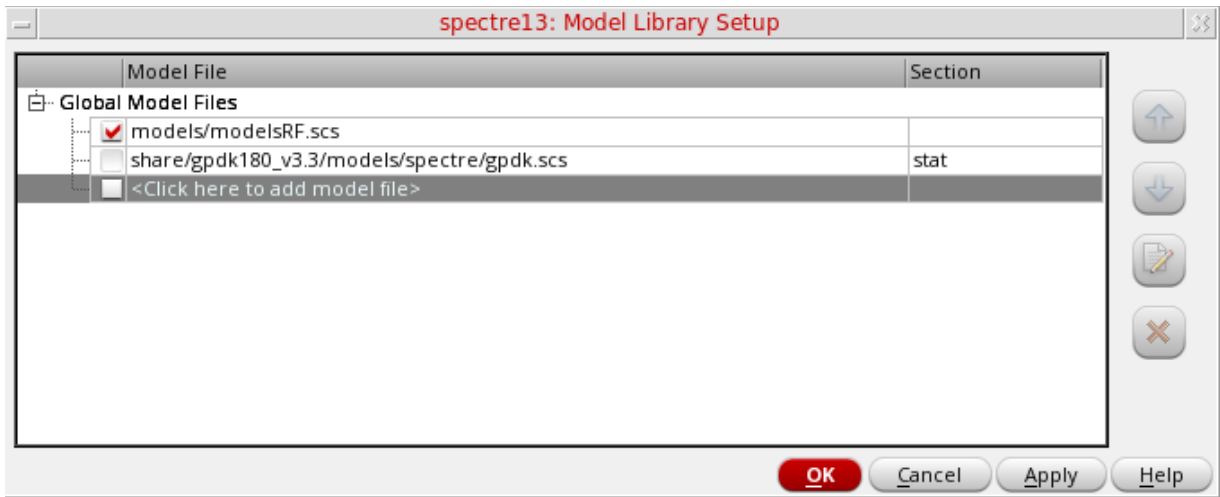
3. In the *Model Library File* field, type in the name of the model file, as shown below.

`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.



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

---

**Figure 2-60 Choosing Analyses Form**

Choosing Analyses -- ADE Explorer

Analysis

☒ tran   ☐ dc   ☐ ac   ☐ noise

☐ xf   ☐ sens   ☐ dcmatch   ☐ acmatch

☐ stb   ☐ pz   ☐ lf   ☐ sp

☐ envlp   ☐ pss   ☐ pac   ☐ pstb

☐ pnoise   ☐ pxf   ☐ psp   ☐ qpss

☐ qpac   ☐ qpnoise   ☐ qpxf   ☐ qpsp

☐ hb   ☐ hbac   ☐ hbstb   ☐ hbnoise

☐ hbsp   ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative   ☐ moderate   ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

**OK**   Cancel   Defaults   Apply   Help

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

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

Figure 2-61 hb Choosing Analyses Form

**Choosing Analyses -- ADE Explorer**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☒ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbisp ☐ hbxf

Harmonic Balance Analysis

Transient-Aided Options

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) **auto**

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

Tone 1

Fundamental Frequency

Number of Harmonics **auto**

Oversample Factor

Freqdivide Ratio for Tone 1

Harmonics **Default**

Accuracy Defaults (errpreset)

☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☐

**OK** Cancel Defaults Apply Help

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**

Harmonic Balance Analysis

Transient-Aided Options

Run transient? Decide automatically

Detect Steady State ☒

Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☒ no ☒ yes

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**

Tones ☐ Frequencies ☒ Names

Tones

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 1 | frf1 | frf1 | 900M  | auto  | 1     | yes   | RF    |
| 2 | frf2 | frf2 | 920M  | 3     | 1     | no    | RF    |

frf1 frf1 900M auto 1 yes RF

Change Delete Update From Hierarchy

Freqdivide Ratio for tone with Tstab 1

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

---

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:

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

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

qnoise qpxf qpsp hb  
hbac hbnoise hbasp

Harmonic Balance Analysis

Transient-Aided Options

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) **auto**

Save Initial Transient Results (saveinit) ☐ no ☒ **yes**

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ **Names**

Tones

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 1 | frf1 | frf1 | 900M  | auto  | 1     | yes   | RF    |
| 2 | frf2 | frf2 | 920M  | 3     | 1     | no    | RF    |

frf1 frf1 900M **auto** 1 yes **yes** RF

Change Delete Update From Hierarchy

Freqdivide Ratio for tone with Tstab 1

Harmonics **Default**

Accuracy Defaults (errpreset)  
☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

**Sweep** 1 ☒ ☒

Variable ☒

Frequency Variable? ☒ no ☐ yes

Variable Name **prf**

Select Design Variable

Sweep Range

☒ Start-Stop Start **-50** Stop **-20**

☐ Center-Span

Sweep Type

☒ **Linear** ☐ Logarithmic

☒ Step Size **5**

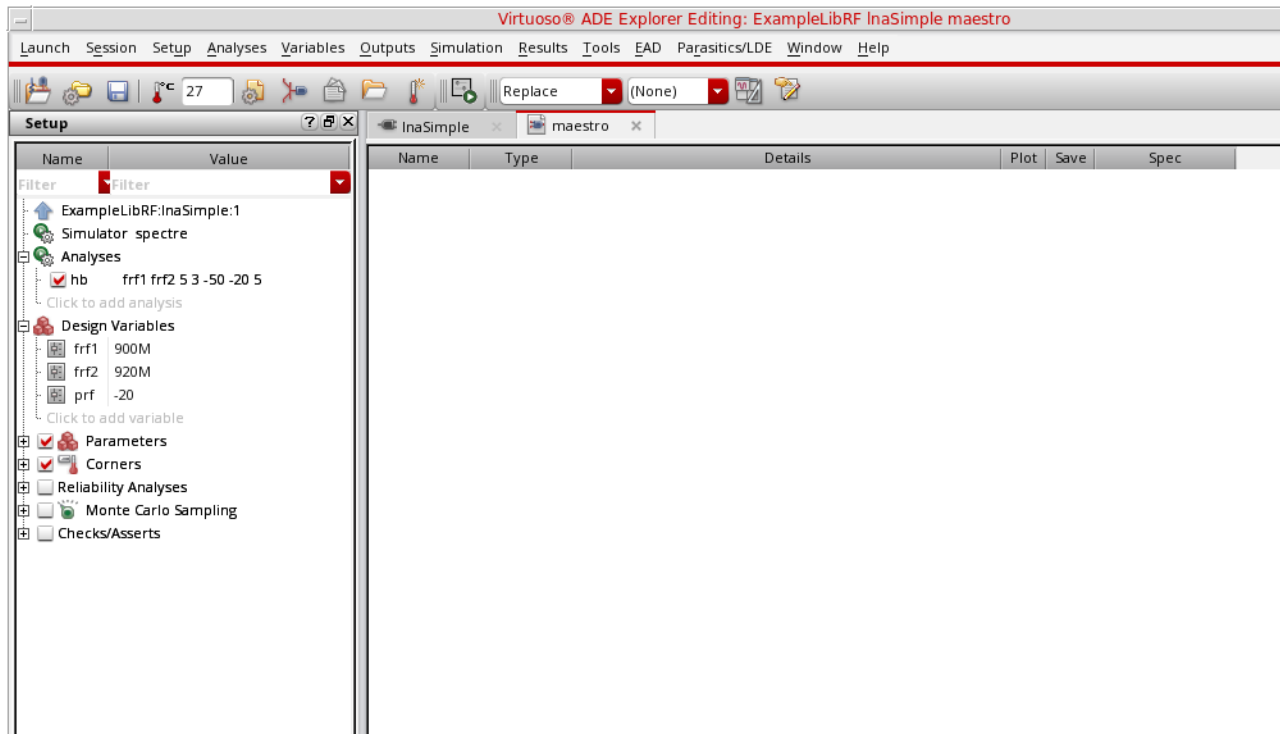
☐ Number of Steps

OK Cancel Defaults Apply Help

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

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.

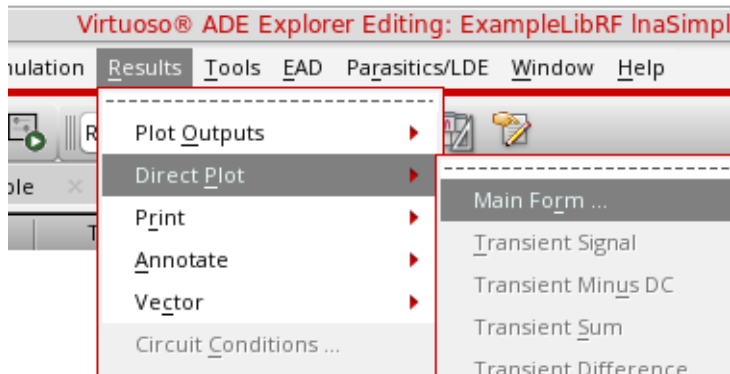
```
Total time required for hb analysis `sweep_hb-006_hb': CPU = 17.997 ms, elapsed = 49.8779 ms.  
Time accumulated: CPU = 232.963 ms, elapsed = 1.02819 s.  
Peak resident memory used = 30.8 Mbytes.  
  
Total time required for sweep analysis `sweep_hb': CPU = 126.98 ms, elapsed = 481.89 ms.  
Time 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.

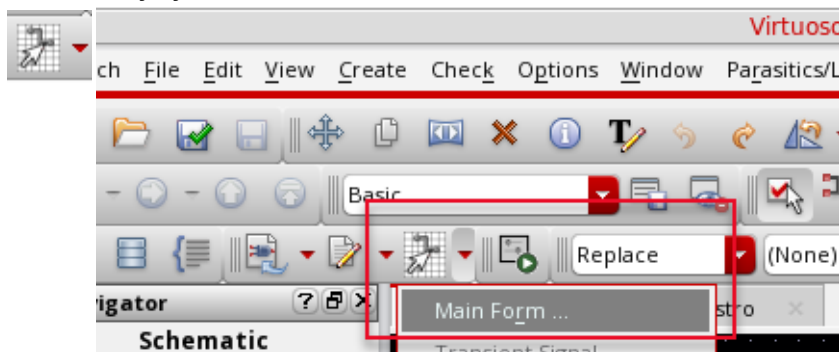
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

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



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

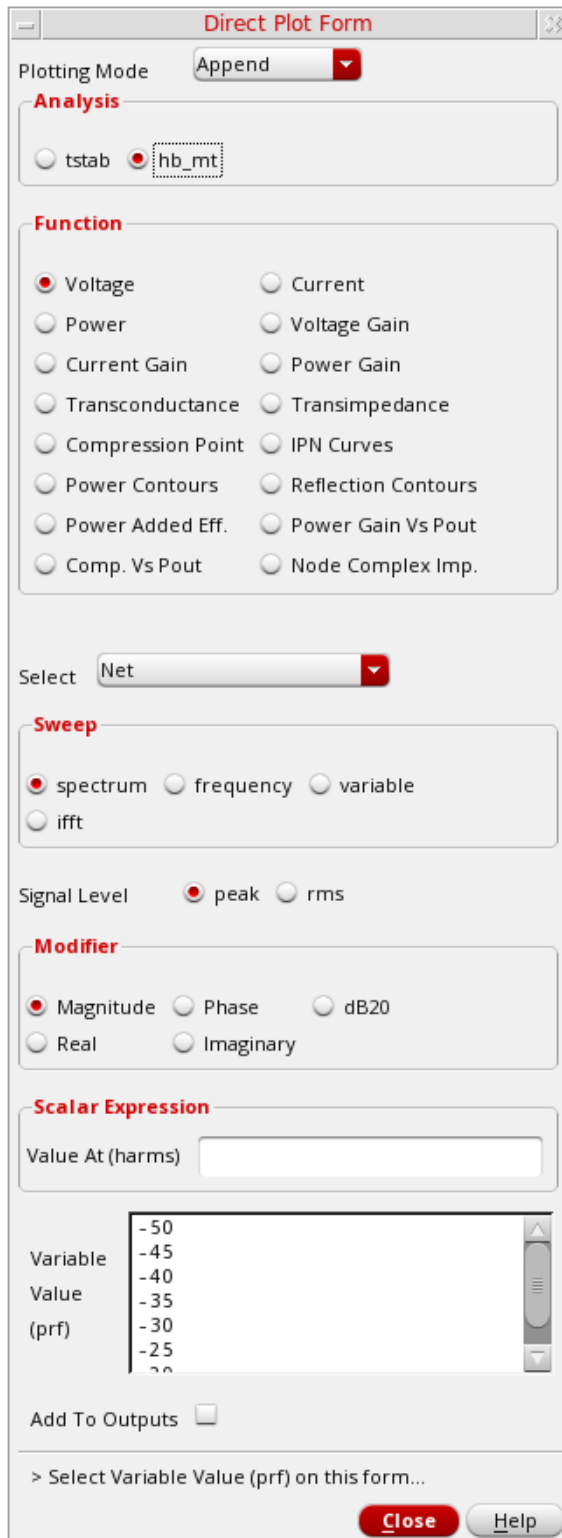




## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

27. The Direct Plot Form is displayed.



The Direct Plot Form dialog box is shown with the following settings:

- Plotting Mode:** Append
- Analysis:** ☒ hb\_mt
- Function:**
  - ☒ Voltage
  - ☐ Current
  - ☐ Power
  - ☐ Voltage Gain
  - ☐ Current Gain
  - ☐ Power Gain
  - ☐ Transconductance
  - ☐ Transimpedance
  - ☐ Compression Point
  - ☐ IPN Curves
  - ☐ Power Contours
  - ☐ Reflection Contours
  - ☐ Power Added Eff.
  - ☐ Power Gain Vs Pout
  - ☐ Comp. Vs Pout
  - ☐ Node Complex Imp.
- Select:** Net
- Sweep:**
  - ☒ spectrum
  - ☐ frequency
  - ☐ variable
  - ☐ ifft
- Signal Level:** ☒ peak ☐ rms
- Modifier:**
  - ☒ Magnitude
  - ☐ Phase
  - ☐ dB20
  - ☐ Real
  - ☐ Imaginary
- Scalar Expression:**
  - Value At (harms):
  - Variable Value (prf):
    - 50
    - 45
    - 40
    - 35
    - 30
    - 25
    - 20
- Add To Outputs:** ☐
- Footer:** > Select Variable Value (prf) on this form... **Close** **Help**

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

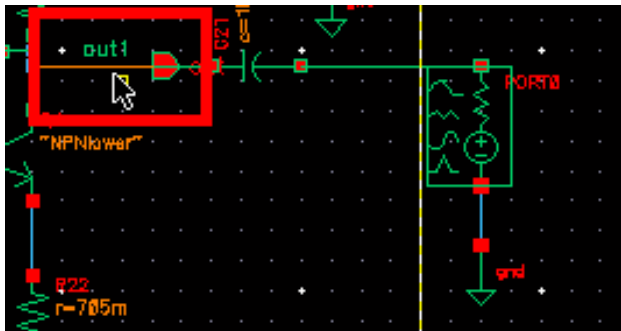
---

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.

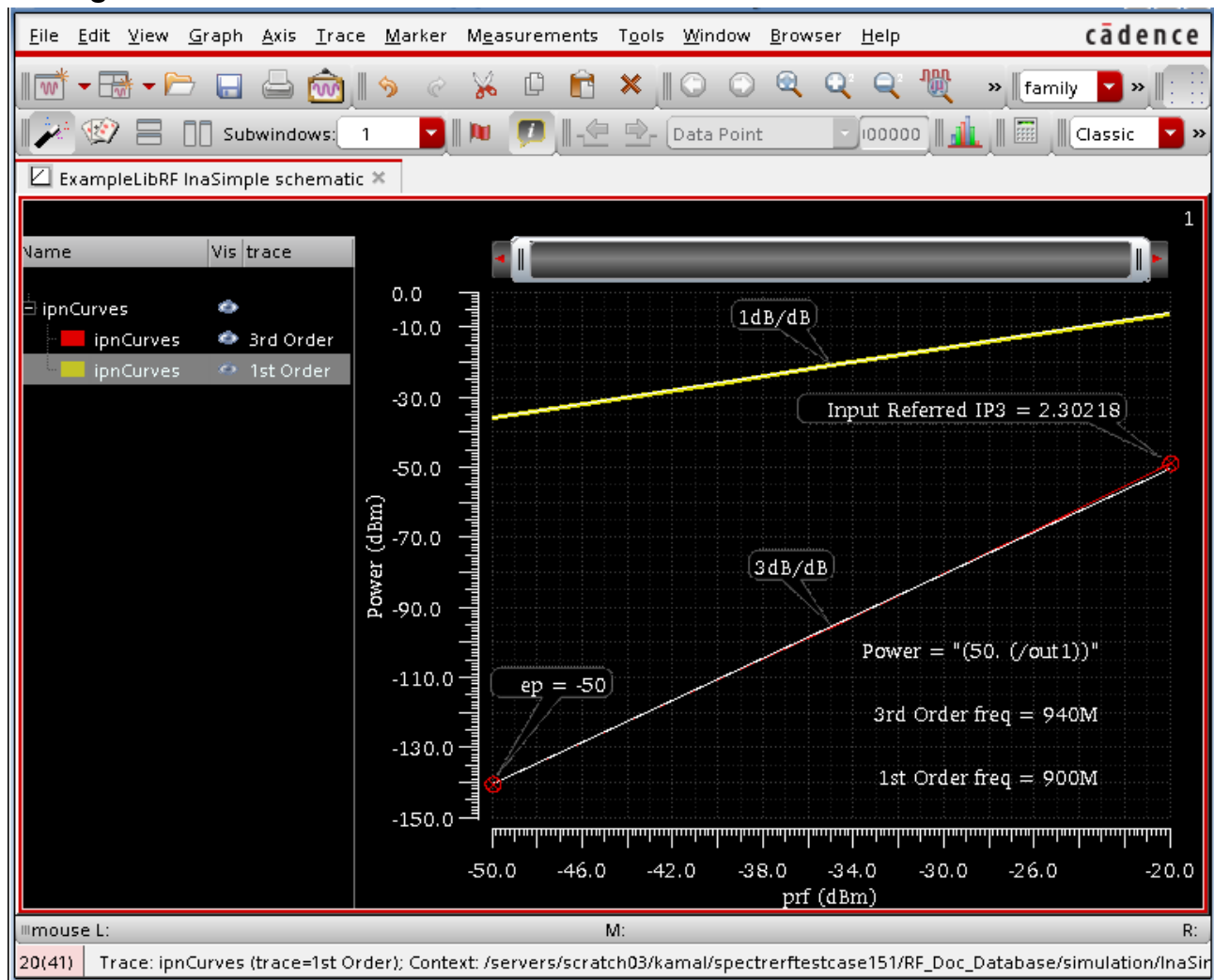
37. Select the *out1* node on the right side of the schematic

**Figure 2-66 Selecting the out1 Net on the InaSimple Schematic.**



The IP3 plot is displayed. Note the IP3 readout.

**Figure 2-67 The IP3 Plot**



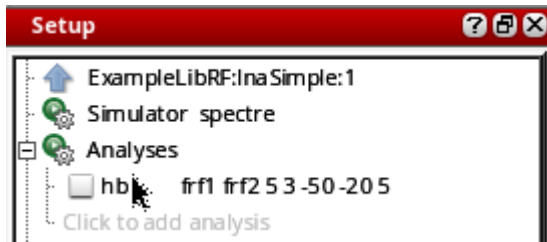
**Note:** If you do not see the IP3 readout, click in the graphics area to deselect the marker, then select and move the *Input Referred IP3* readout so that it is positioned in the visible area of the graph.

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.

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

---

**Figure 2-69 Choosing Analyses Form**

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☒ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☐ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

AC Analysis

Sweep Variable

☒ Frequency

☐ Design Variable

☐ Temperature

☐ Component Parameter

☐ Model Parameter

☐ None

Sweep Range

☒ Start-Stop Start  Stop

☐ Center-Span

Sweep Type

Automatic

Add Specific Points ☐

Add Points By File ☐

Specialized Analyses

None

Enabled ☐

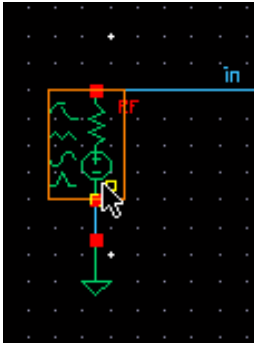
Options...

OK Cancel Defaults Apply Help

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 *Input 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**

The image shows the 'Rapid IP3 AC Analysis Form' in the Spectre Circuit Simulator. The form is titled 'AC Analysis' and contains several sections for configuring the analysis.

- Sweep Variable:** Radio buttons for Frequency (selected), Design Variable, Temperature, Component Parameter, Model Parameter, and None.
- Sweep Range:** Radio buttons for Start-Stop (selected) and Center-Span. Start frequency is 900M, Stop frequency is 920M.
- Sweep Type:** A dropdown menu set to 'Automatic'.
- Add Specific Points:** A checkbox that is currently unchecked.
- Specialized Analyses:** A dropdown menu set to 'Rapid IP3'.
- Source Type:** Radio buttons for port (selected), isource, and vsource.
- Input Sources 1:** A text field containing '/RF' and a frequency field set to 900M. There are 'Select' and 'Clear' buttons.
- Input Sources 2:** A text field containing '/RF' and a frequency field set to 920M. There are 'Select' and 'Clear' buttons.
- Input Power (dBm):** A text field containing '-50'. There are fields for 'Power 2' and '3'.
- Frequency of IM Output Signal:** A text field containing '940M'.
- Frequency of Linear Output Signal:** A text field containing '900M'.
- Maximum Non-linear Harmonics:** An empty text field.
- Output:** Radio buttons for Voltage (selected) and Current. There are 'Out+' and 'Out-' labels. The 'Out+' field contains '/out1' and has a 'Select' button. The 'Out-' field is empty and has a 'Select' button.
- Enabled:** A checkbox that is checked.
- Buttons:** 'Options...', 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'.

17. Click **OK**.

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

---

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**

The image shows the 'Direct Plot Form' dialog box. At the top, the title bar says 'Direct Plot Form'. Below it, the 'Plotting Mode' is set to 'Append'. The 'Analysis' section has a radio button selected for 'ac'. The 'Function' section has four radio buttons: 'Voltage' (selected), 'Current', 'Rapid IP3', and 'GD'. The 'Select' dropdown menu is set to 'Net'. The 'Modifier' section has four radio buttons: 'Magnitude' (selected), 'Phase', 'dB10', and 'dB20'. There is an 'Add To Outputs' checkbox which is unchecked. At the bottom, there is a text field with the prompt '> Select Net on schematic...'. The bottom right corner has three buttons: 'OK' (highlighted in red), 'Cancel', and 'Help'.

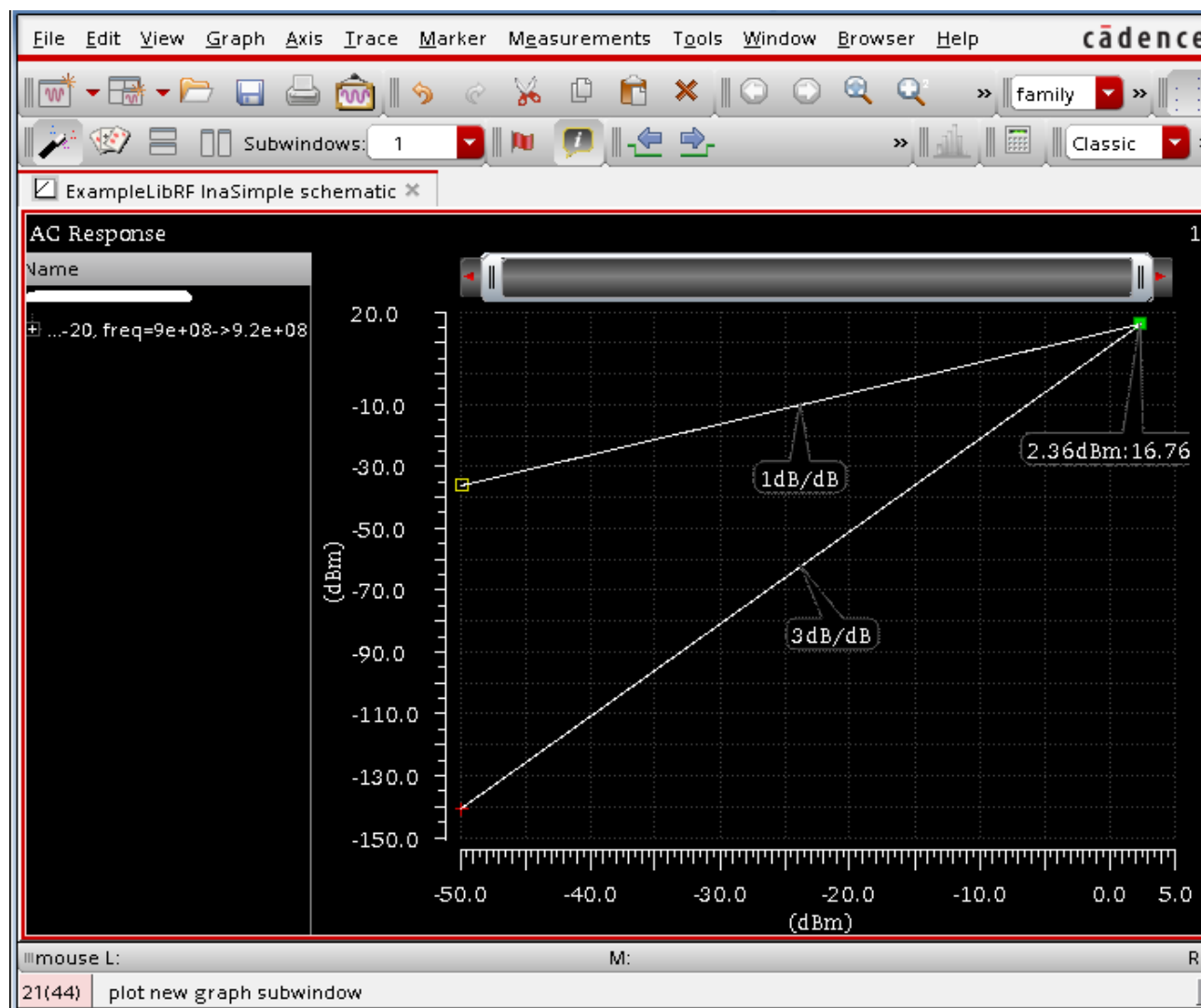
21. In the *Direct Plot Form*, select *Rapid IP3*. The form changes.

**Figure 2-73 AC Direct Plot Form - Rapid IP3**

The image shows a software dialog box titled "Direct Plot Form". At the top, there is a "Plotting Mode" dropdown menu set to "Append". Below this is a section labeled "Analysis" with a radio button selected for "ac". Another section labeled "Function" contains four radio buttons: "Voltage", "Current", "Rapid IP3" (which is selected and has a dashed border around it), and "GD". Below the "Function" section is a "Resistance (Default is 50.)" text label next to a text input field containing the value "50". There are two buttons: "Add To Outputs" with an unchecked checkbox and a "Plot" button. At the bottom, there is a line of text that says "> Press plot button on this form..." followed by three buttons: "OK", "Cancel", and "Help".

- 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.

---

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

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

---

| Measurements                                                                      | Analysis              |
|-----------------------------------------------------------------------------------|-----------------------|
| <u>Calculating the Swept Tuning Range and Phase Noise for the Ring Oscillator</u> | 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 running the transient analysis for the whole *tstab* value.

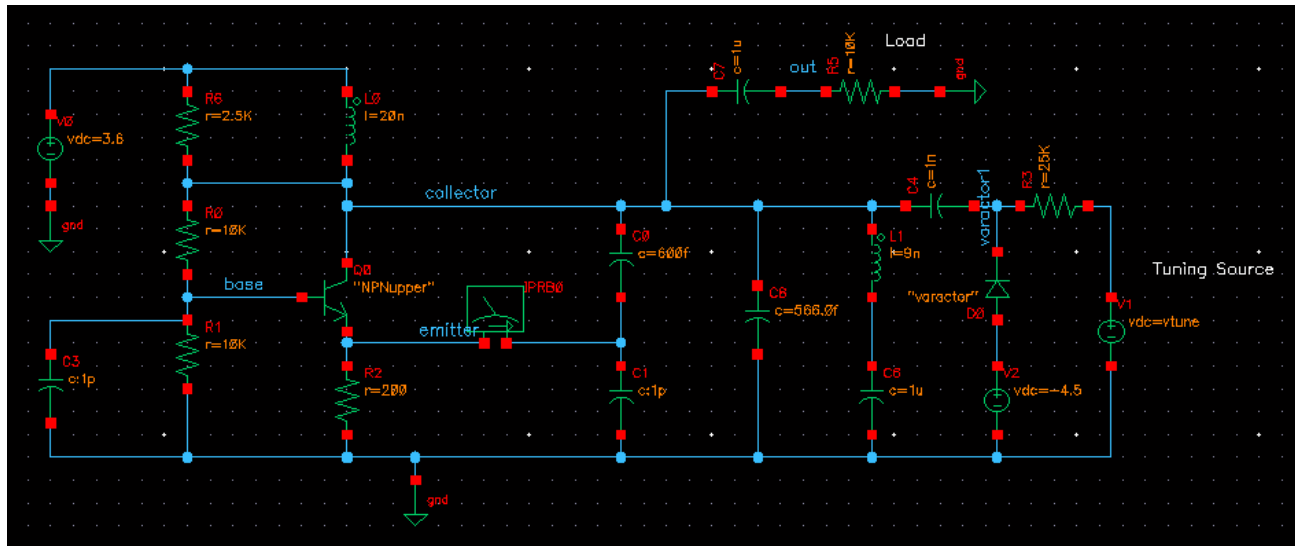
It is recommended to save the initial transient simulation results. This can be done by setting the *Save Initial Transient Results (saveinit)* option to *yes*. You can use this to verify that the oscillator has started up and is stable.

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

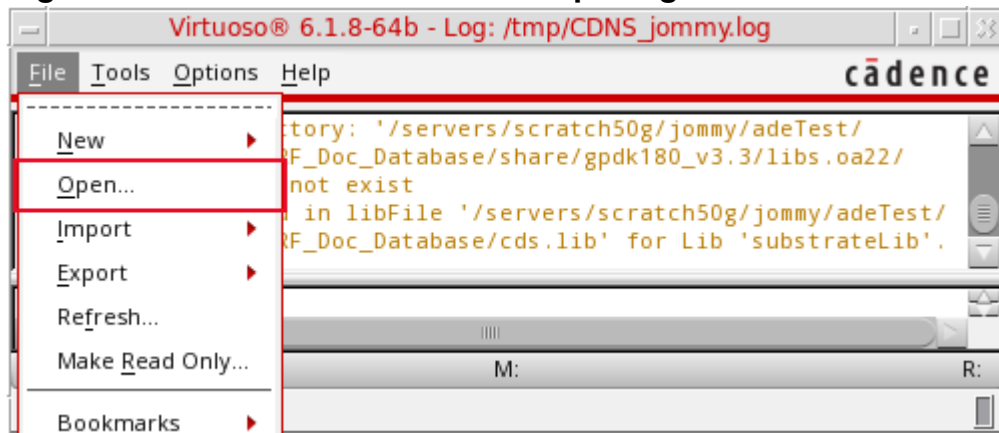


## Setting Up to Simulate the Oscillator Circuit

### Opening the Oscillator Circuit in the Schematic Window

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

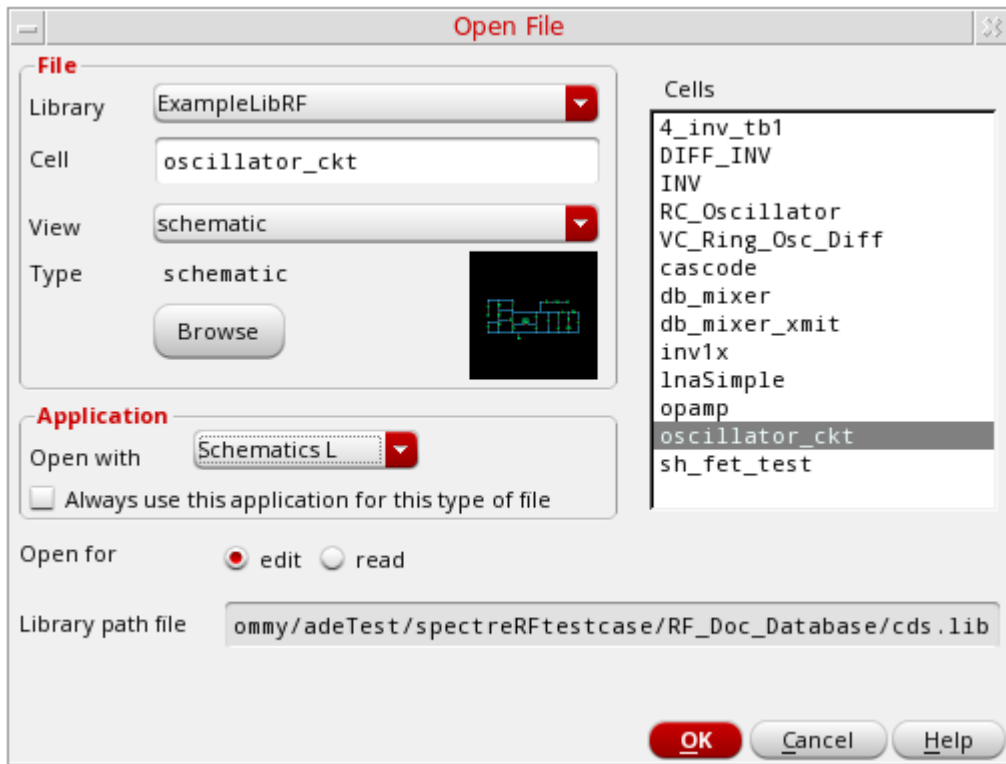
**Figure 3-2 Virtuoso CIW Window - Opening Cellview**



The *Open File* form is displayed.

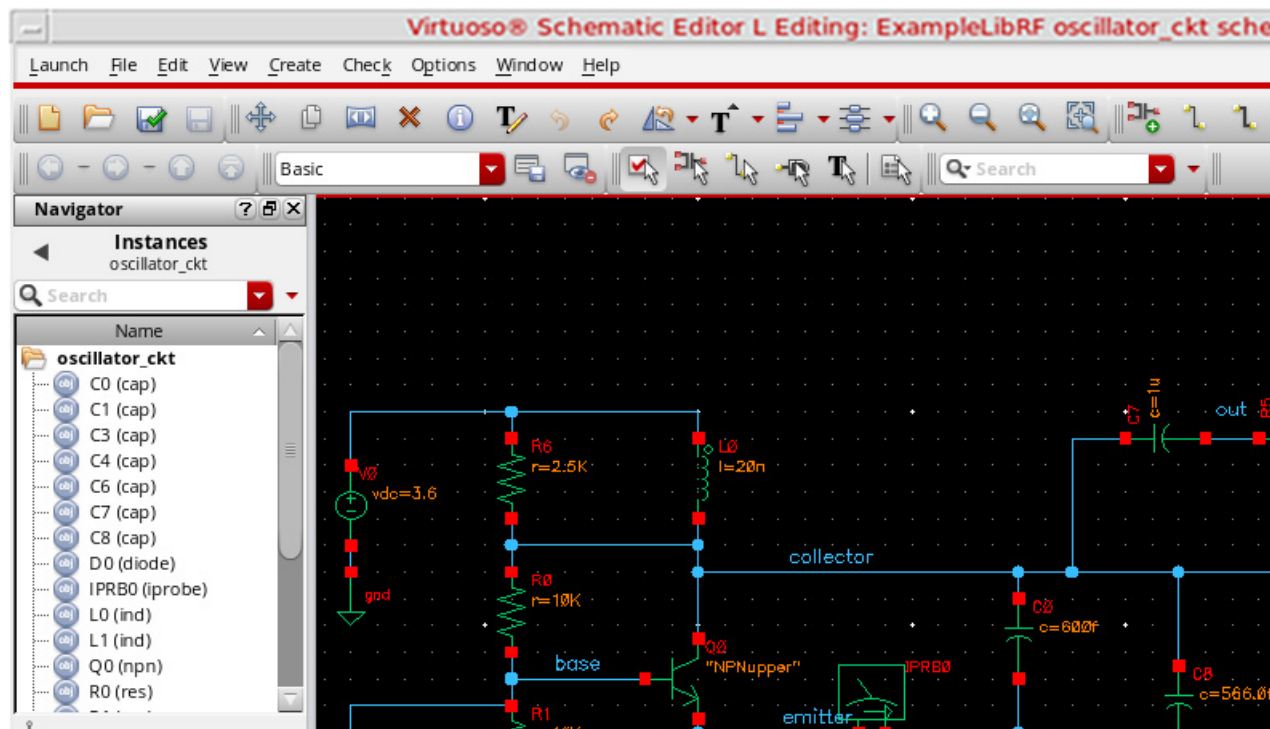
2. Select *ExampleLibRF* from the *Library* drop-down list.
3. In the *Cells* list box, choose *oscillator\_ckt*.
4. Select *schematic* from the *View* drop-down list.
5. In the *Application* section, select *Schematic-L* from the *Open with* drop-down list.
6. Leave *Open For* to *edit* (which is set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

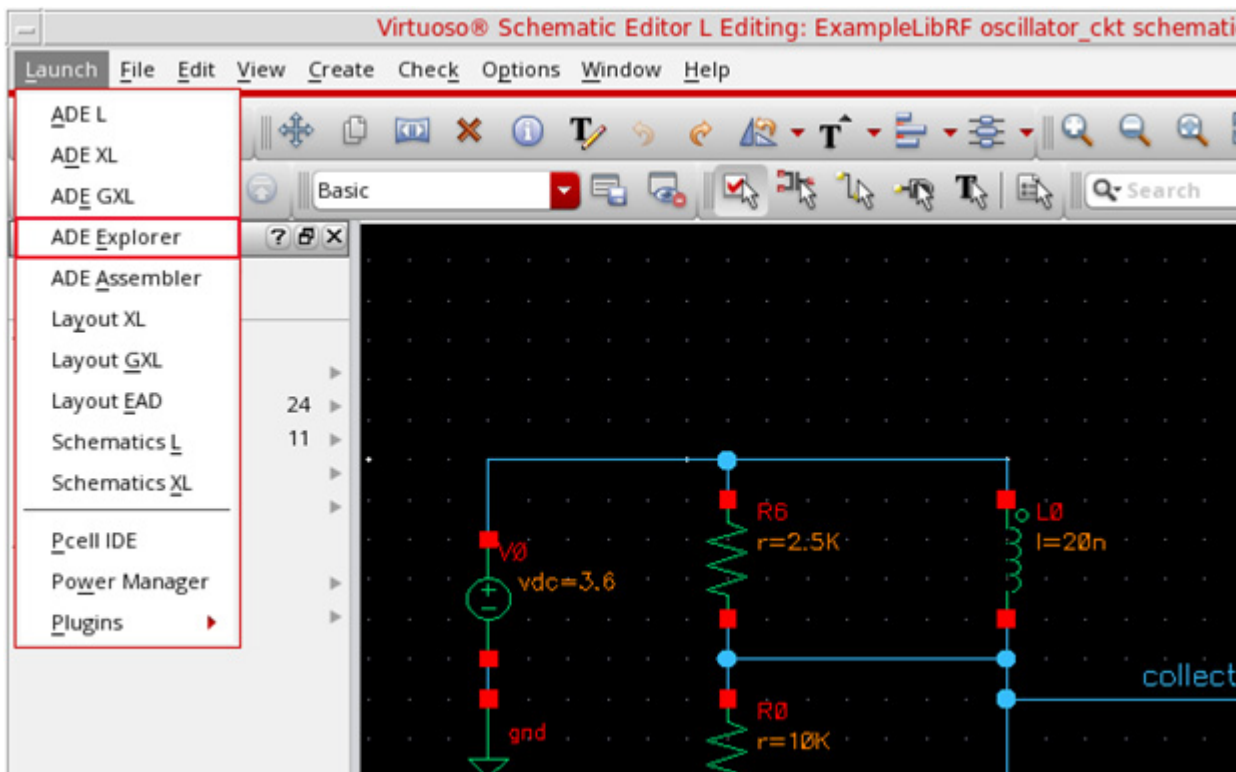


This example computes the periodic steady state solution for the *oscillator\_ckt* RF oscillator circuit. You perform a PSS-HB or HB analysis first because the periodic steady state solution must be determined before you can perform any other periodic small-signal analysis like *pstb* or *hbnoise* etc.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

### Setting up ADE Explorer for Oscillator Simulation

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

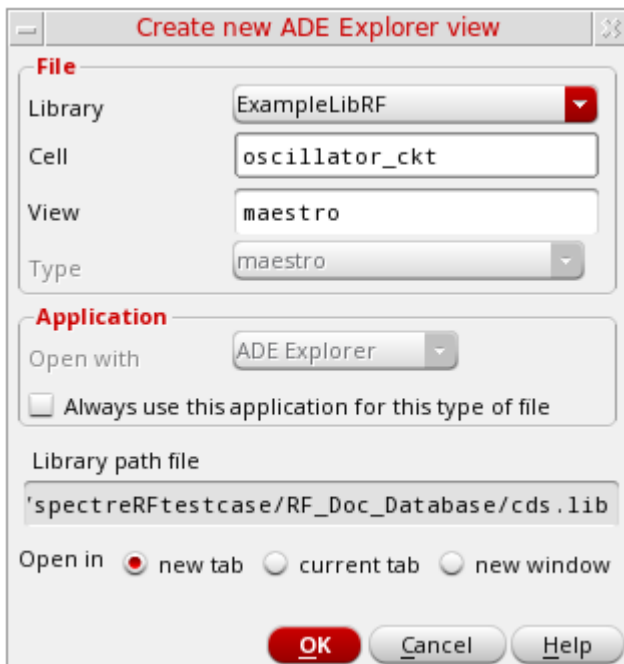
Figure 3-5 Opening ADE Explorer from VSE L window



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**

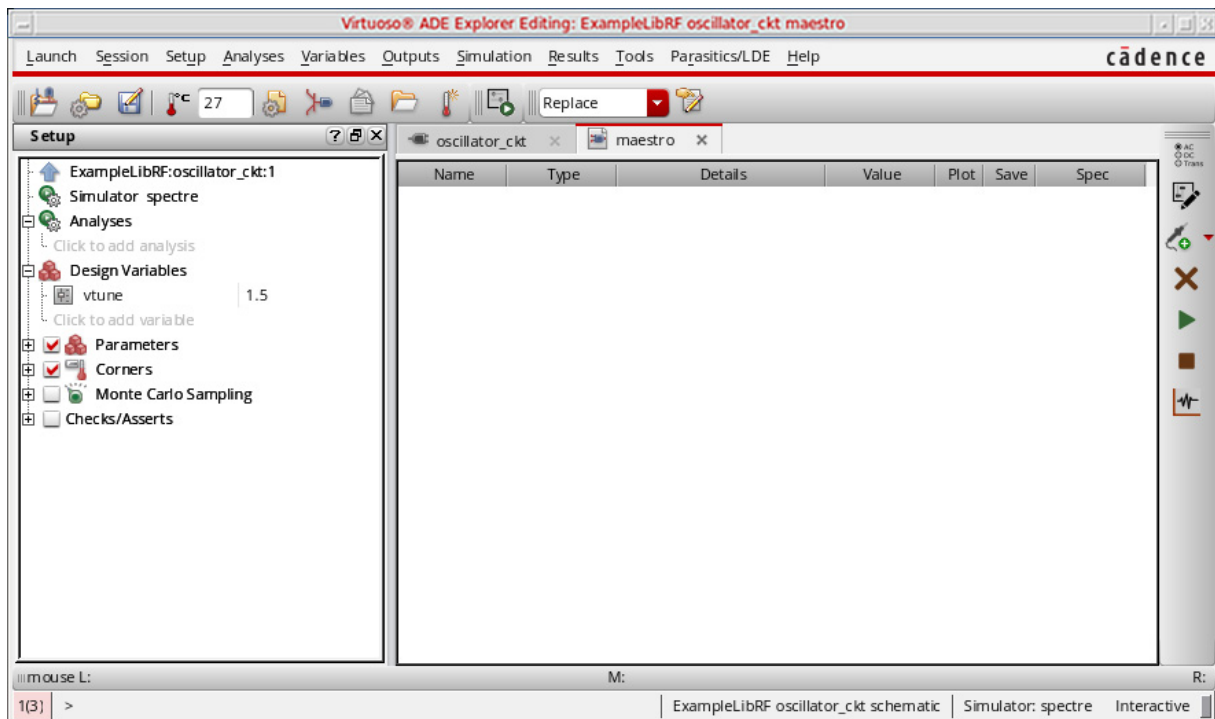


3. Leave each option to the default selections and click *OK*.

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

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

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

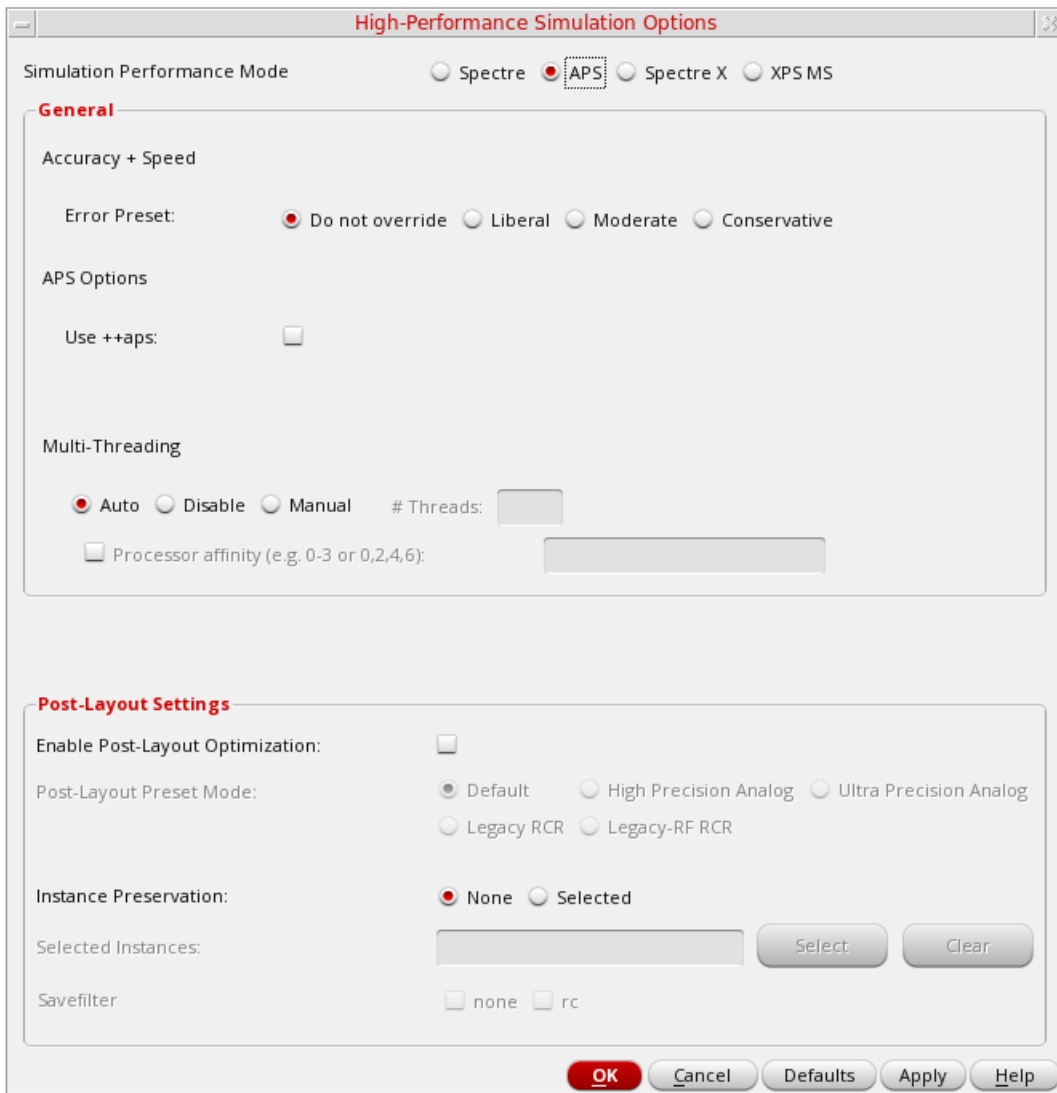


## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

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**



The image shows the 'High-Performance Simulation Options' dialog box. At the top, the title bar reads 'High-Performance Simulation Options'. Below the title bar, the 'Simulation Performance Mode' section has four radio buttons: 'Spectre', 'APS' (which is selected and highlighted with a dashed box), 'Spectre X', and 'XPS MS'. The 'General' section is expanded, showing 'Accuracy + Speed' with an 'Error Preset' of 'Do not override' (selected). Below this, 'APS Options' includes a 'Use ++aps' checkbox which is unchecked. The 'Multi-Threading' section has 'Auto' selected, and a '# Threads' input field. There is also a checkbox for 'Processor affinity (e.g. 0-3 or 0,2,4,6):' which is unchecked. The 'Post-Layout Settings' section is also expanded, showing 'Enable Post-Layout Optimization:' as unchecked. 'Post-Layout Preset Mode:' has 'Default' selected. 'Instance Preservation:' has 'None' selected. There is a 'Selected Instances:' text field with 'Select' and 'Clear' buttons. At the bottom, there is a 'Savefilter' section with 'none' and 'rc' checkboxes. The bottom of the dialog has 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help' buttons.

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

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

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

Figure 3-10 Save Options Form

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

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

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

---

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.

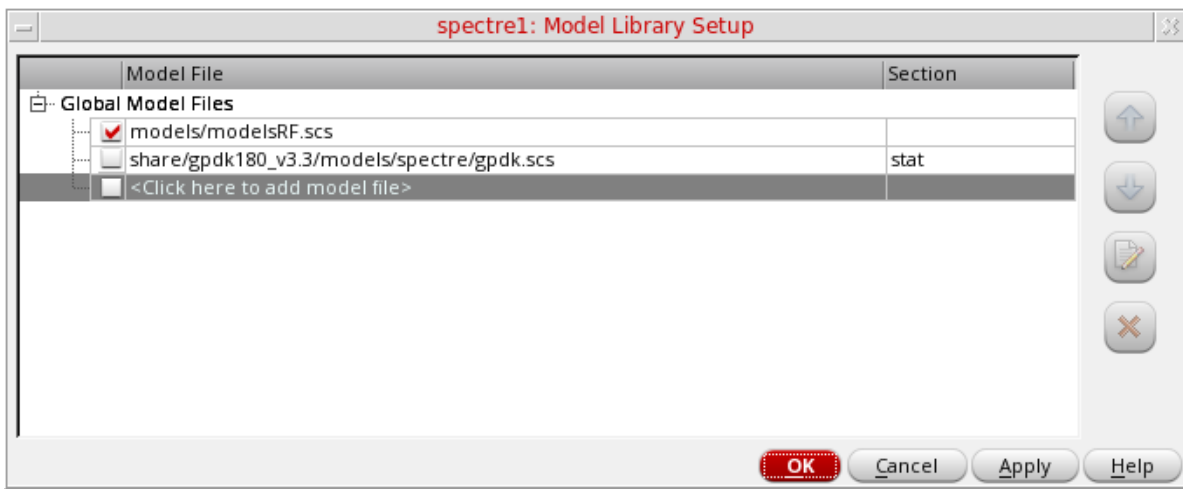
15. Click *OK* to close the *Simulation Files Setup* form.

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.

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

---

**Figure 3-13 The *Choosing Analyses* Form**

**Choosing Analyses -- ADE Explorer**

Analysis

☒ tran   ☐ dc   ☐ ac   ☐ noise

☐ xf   ☐ sens   ☐ dcmatch   ☐ acmatch

☐ stb   ☐ pz   ☐ lf   ☐ sp

☐ envlp   ☐ pss   ☐ pac   ☐ pstb

☐ pnoise   ☐ pxf   ☐ psp   ☐ qpss

☐ qpac   ☐ qpnoise   ☐ qpxf   ☐ qpss

☐ hb   ☐ hbac   ☐ hbstb   ☐ hbnoise

☐ hbsp   ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative   ☐ moderate   ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

**OK**   Cancel   Defaults   Apply   Help

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

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

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

The screenshot shows the 'Choosing Analyses Form' for setting PSS Analysis. The form is organized into several sections:

- Analysis Selection:** A grid of radio buttons for selecting analysis types. **pss** is selected. Other options include envlp, pac, pstb, pnoise, pxf, psp, qpss, qpac, qpnoise, qpxf, qpss, hb, hbac, hbstb, hbnoise, hbsp, and hbxf.
- Periodic Steady State Analysis:** A section with the title 'Periodic Steady State Analysis'.
- Engine:** Radio buttons for 'Shooting' and 'Harmonic Balance'. 'Harmonic Balance' is selected.
- Tones:** A table with columns 'Name', 'Expr', 'Value', and 'SrcId'. Below the table are input fields for 'Beat Frequency' and 'Oversample Factor', and a checkbox for 'Auto Calculate'.
- Number of Harmonics:** An input field.
- Accuracy Defaults (errpreset):** Radio buttons for 'conservative', 'moderate', and 'liberal'.
- Convergence:** A section with 'Additional Time for Transient-Aided HB (tstab)' input field, 'Save Initial Transient Results (saveinit)' checkbox (set to 'no'), and 'Harmonic Balance Homotopy Method' dropdown (set to 'default').
- Dynamic Parameter:** A checkbox.
- Oscillator:** A checkbox.
- Sweep:** A checkbox.
- New Initial Value For Each Point (restart):** Radio buttons for 'no' and 'yes'.
- Loadpull:** A checkbox.
- Enabled:** A checkbox that is checked.
- Buttons:** 'Options...', 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help' buttons at the bottom.

- 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.

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

---

- 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.

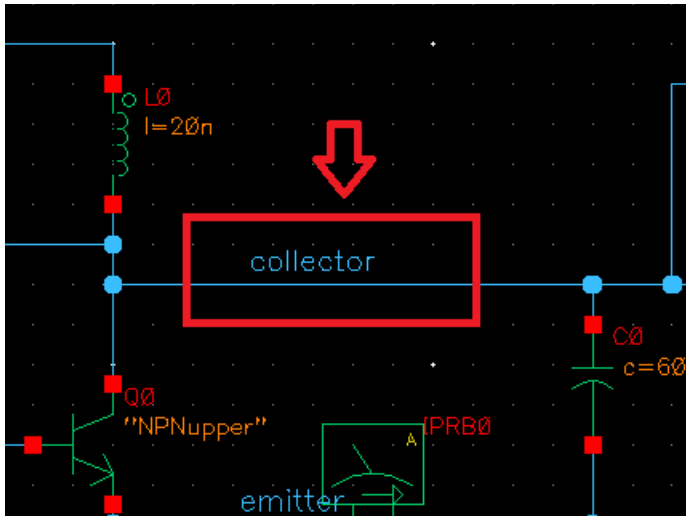
**Figure 3-15 The Choosing Analyses Form - Oscillator Section**

The screenshot shows a dialog box titled "Oscillator" with a red checkmark icon. It contains the following elements:

- A label "Oscillator" followed by a red checkmark icon.
- A field "Oscillator node+" with an empty text box and a "Select" button to its right.
- A field "Oscillator node-" with an empty text box and a "Select" button to its right.
- A checked checkbox labeled "Calculate initial conditions (ic) automatically".
- An unchecked checkbox labeled "Enable tuning mode analysis".
- An unchecked checkbox labeled "Use the probe-based solution method".

- 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.



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

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

Engine ☐ Shooting ☒ Harmonic Balance

Tones

| Name | Expr | Value | SrcId |
|------|------|-------|-------|
|------|------|-------|-------|

Beat Frequency  Auto Calculate ☐

Oversample Factor

Number of Harmonics

Accuracy Defaults (errpreset)  
☒ conservative ☐ moderate ☐ liberal

Convergence  
Additional Time for Transient-Aided HB (tstab)   
Save Initial Transient Results (saveinit) ☐ no ☒ yes  
Harmonic Balance Homotopy Method

Dynamic Parameter ☐

Oscillator ☒ Oscillator node+    
Oscillator node-

☒ Calculate initial conditions (ic) automatically  
☐ Enable tuning mode analysis  
☐ Use the probe-based solution method

Sweep ☐  
New Initial Value For Each Point (restart) ☐ no ☐ yes

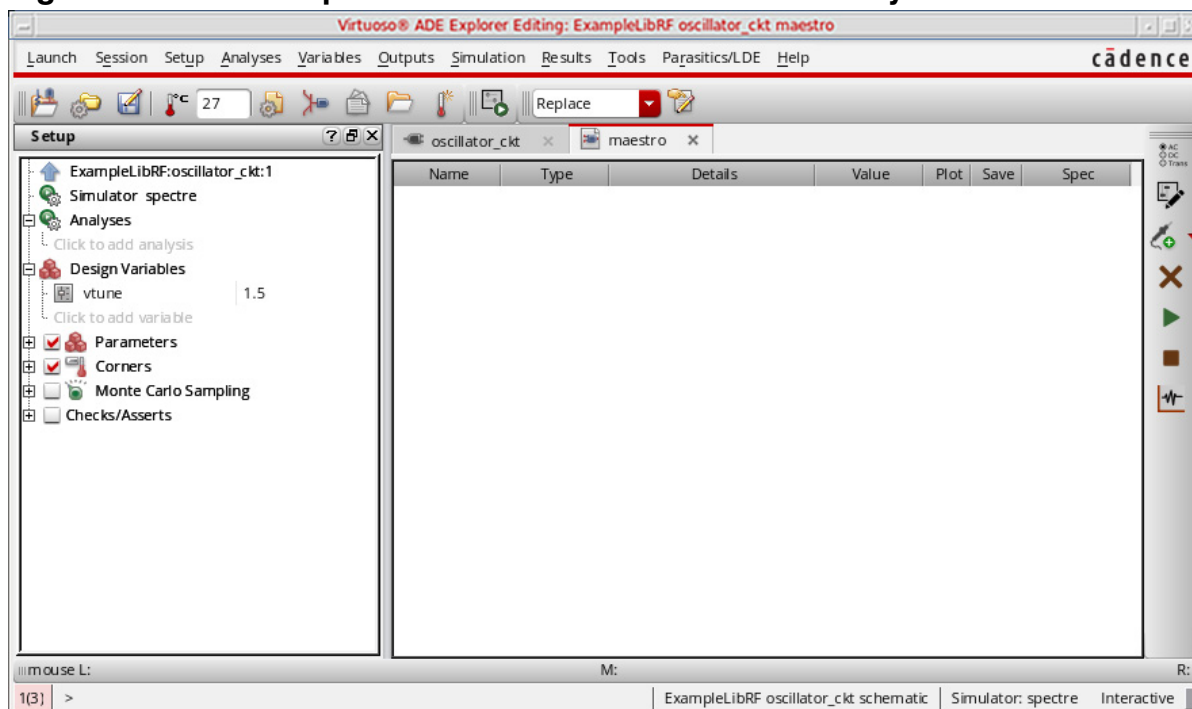
Loadpull ☐

Enabled ☒

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

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

**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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☒ pss ☐ tstab

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Harmonic Frequency ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp. ☐ THD

Select: Net

**Sweep**

☒ spectrum ☐ time

Signal Level: ☒ peak ☐ rms

**Modifier**

☒ Magnitude ☐ Phase ☐ dB20  
☐ Real ☐ Imaginary

**Scalar Expression**

Value At (harms):

Add To Outputs: ☐

> Select Net on schematic...

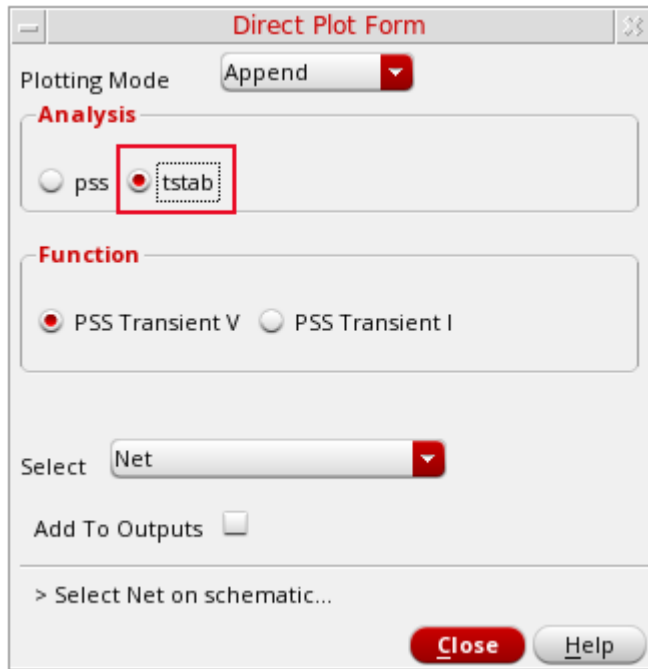
Close Help

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).

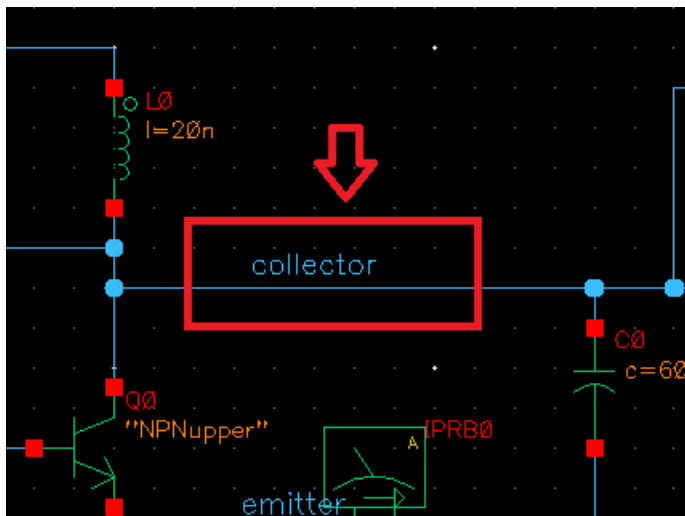
4. The *Direct Plot Form* window should look like the following:

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



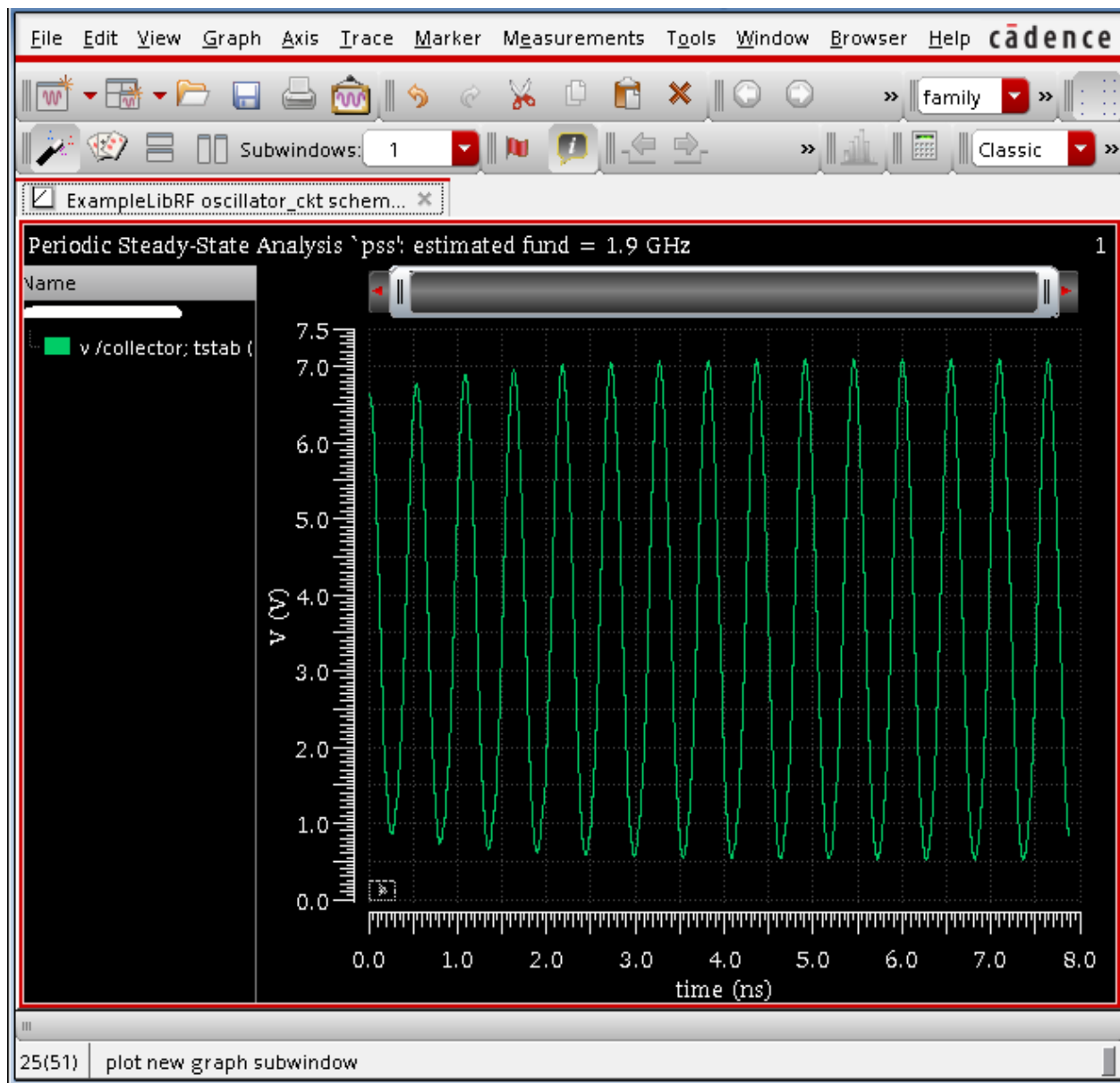
5. Select the *collector* net in the schematic. It is located just below the *collector* label.

**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*.

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

---

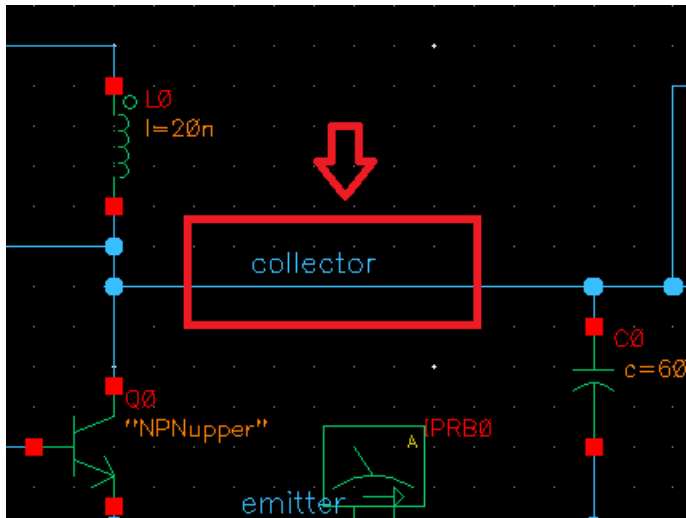
2. Select *pss* as Analysis.
3. Leave *Function* as *Voltage*, which is set by default.
4. Leave the *Select* section of the form set to the default value, *Net* (You can also select differential nets).
5. Select *Sweep* as *spectrum*. (This is the default)
6. Select *rms* as Signal Level (the default 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**

The image shows the 'Direct Plot Form' dialog box in a software application. The form is organized into several sections with red headers. The 'Analysis' section has two radio buttons: 'pss' (selected and highlighted with a red box) and 'tstab'. The 'Function' section contains a grid of radio buttons for various analysis types: 'Voltage' (selected and highlighted with a red box), 'Current', 'Power', 'Voltage Gain', 'Current Gain', 'Power Gain', 'Transconductance', 'Transimpedance', 'Compression Point', 'IPN Curves', 'Power Contours', 'Reflection Contours', 'Harmonic Frequency', 'Power Added Eff.', 'Power Gain Vs Pout', 'Comp. Vs Pout', 'Node Complex Imp.', and 'THD'. Below this is a 'Select' dropdown menu set to 'Net'. The 'Sweep' section has two radio buttons: 'spectrum' (selected and highlighted with a red box) and 'time'. The 'Signal Level' section has two radio buttons: 'peak' and 'rms' (selected and highlighted with a red box). The 'Modifier' section has four radio buttons: 'Magnitude', 'Phase', 'dB20' (selected and highlighted with a red box), and 'Real'. Below this is a 'Scalar Expression' section with a text input field labeled 'Value At (harms)'. At the bottom, there are buttons for 'Add To Outputs' (with a small square icon), 'Replot', '> Select Net on schematic...', 'Close', and 'Help'.

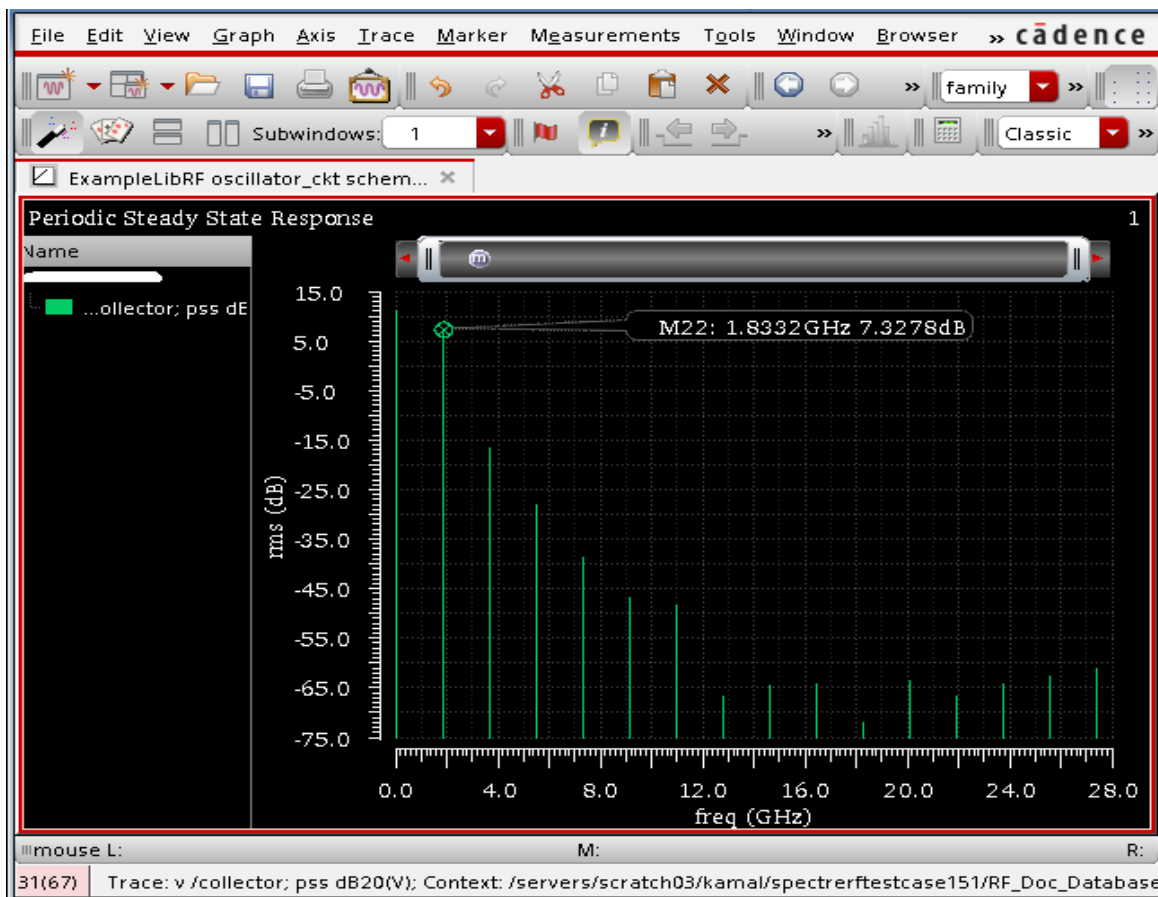
9. Select the *collector* net in the schematic. It is located just below the *collector* label.

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*.

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

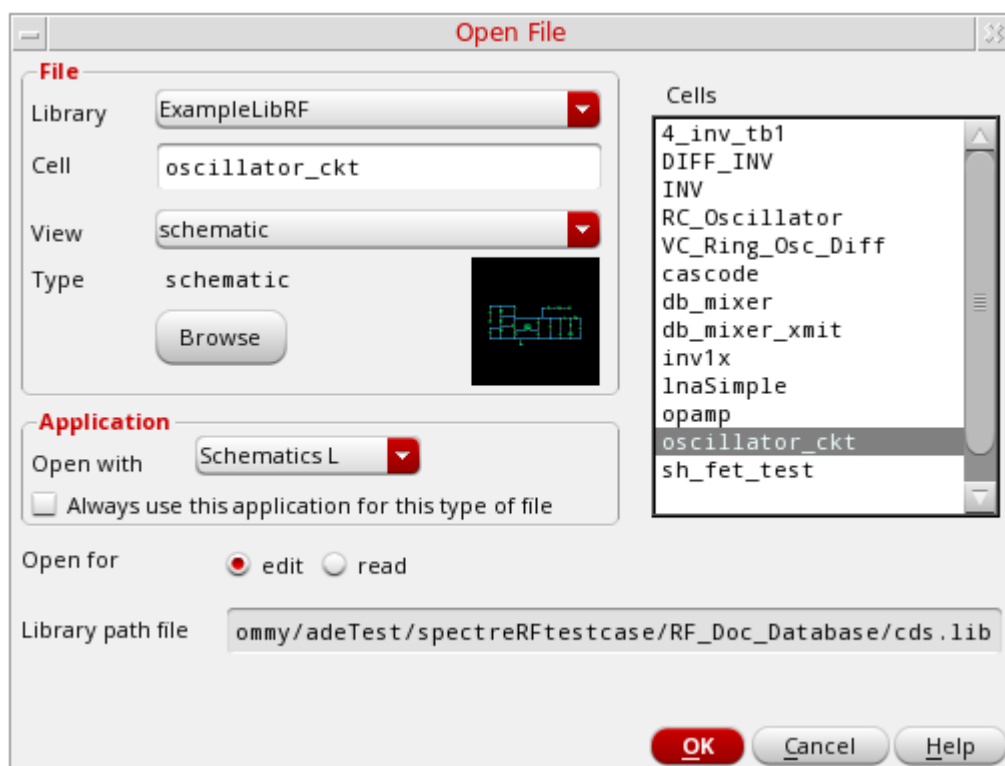
**Figure 3-26 Virtuoso CIW Window - Opening Cellview**



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**

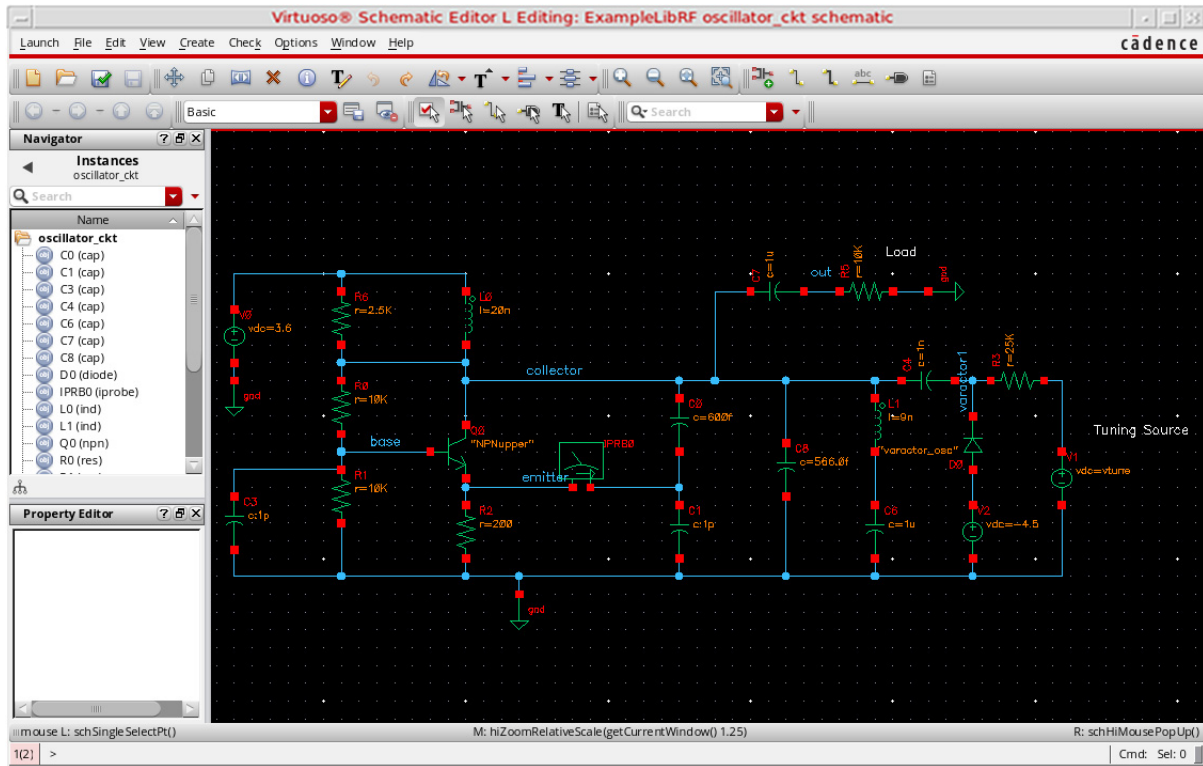


## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

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



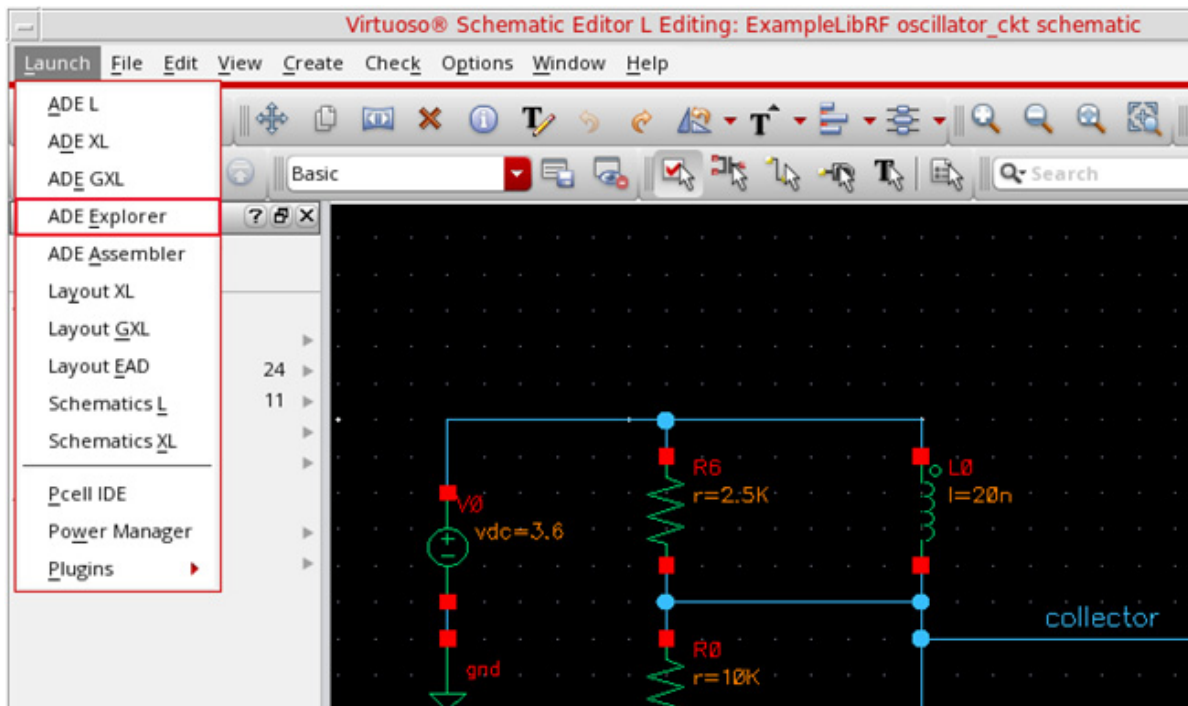
### Setting up the stb Analysis

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

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

---

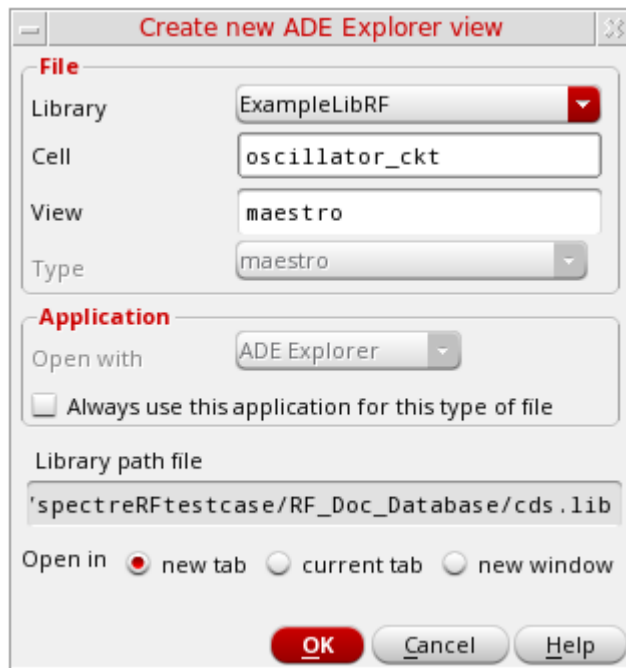
**Figure 3-29 Opening ADE Explorer window from VSE window**



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**

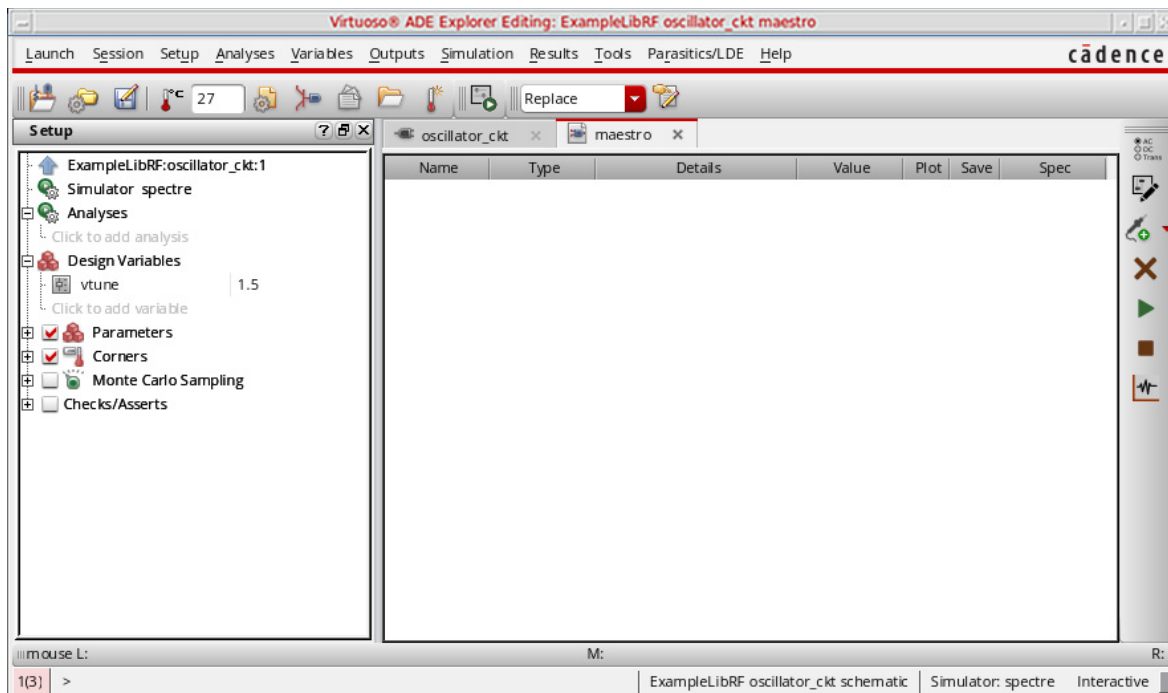


3. Leave each option to the default selections and click *OK*.

Virtuoso ADE Explorer is opened, as shown below.

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

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**



The dialog box is titled "High-Performance Simulation Options". It contains two main sections: "General" and "Post-Layout Settings".

**General Section:**

- Simulation Performance Mode:** Radio buttons for Spectre, APS (selected), Spectre X, and XPS MS.
- Accuracy + Speed:**
  - Error Preset:** Radio buttons for Do not override (selected), Liberal, Moderate, and Conservative.
  - APS Options:**
    - Use ++aps:** A checkbox that is currently unchecked.
  - Multi-Threading:**
    - Radio buttons for Auto (selected), Disable, and Manual.
    - # Threads:** A text input field.
    - Processor affinity (e.g. 0-3 or 0,2,4,6):** A text input field.

**Post-Layout Settings Section:**

- Enable Post-Layout Optimization:** A checkbox that is currently unchecked.
- Post-Layout Preset Mode:** Radio buttons for Default (selected), High Precision Analog, Ultra Precision Analog, Legacy RCR, and Legacy-RF RCR.
- Instance Preservation:** Radio buttons for None (selected) and Selected.
- Selected Instances:** A list box with "Select" and "Clear" buttons.
- Savefilter:** Checkboxes for none and rc.

At the bottom of the dialog are buttons for OK, Cancel, Defaults, Apply, and Help.

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.

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

Figure 3-34 Save Options Form

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

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.



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

---

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.

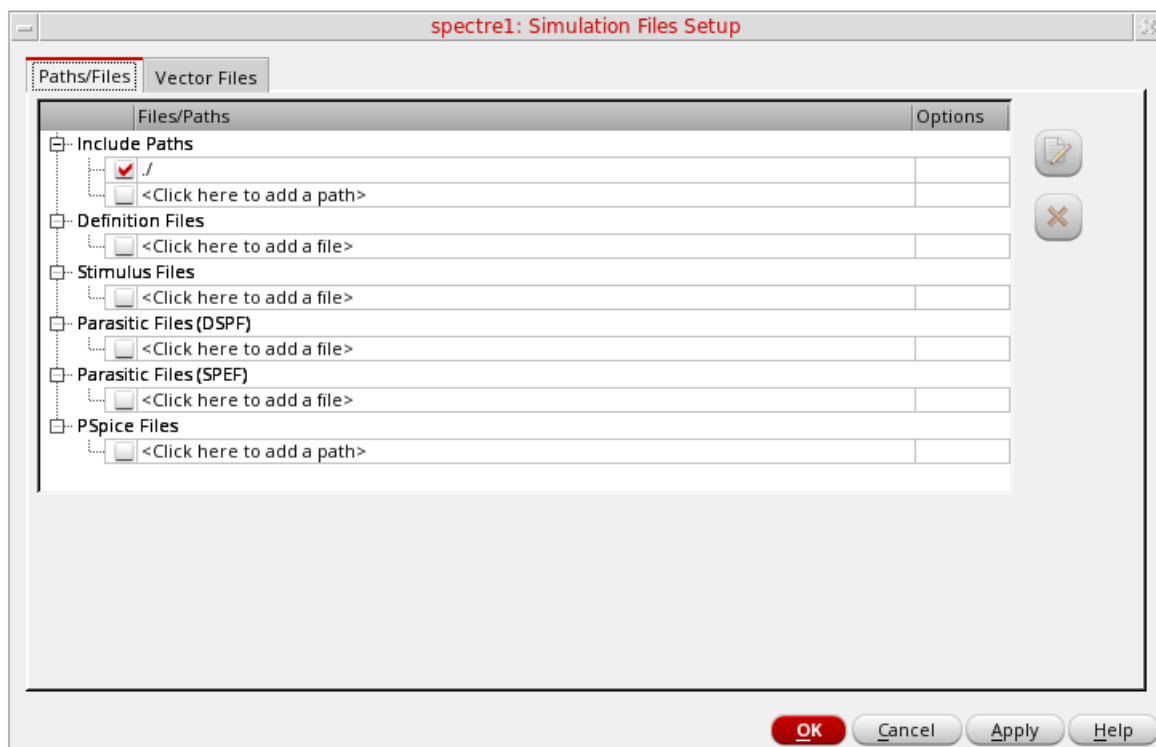
11. Click *OK*.

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.

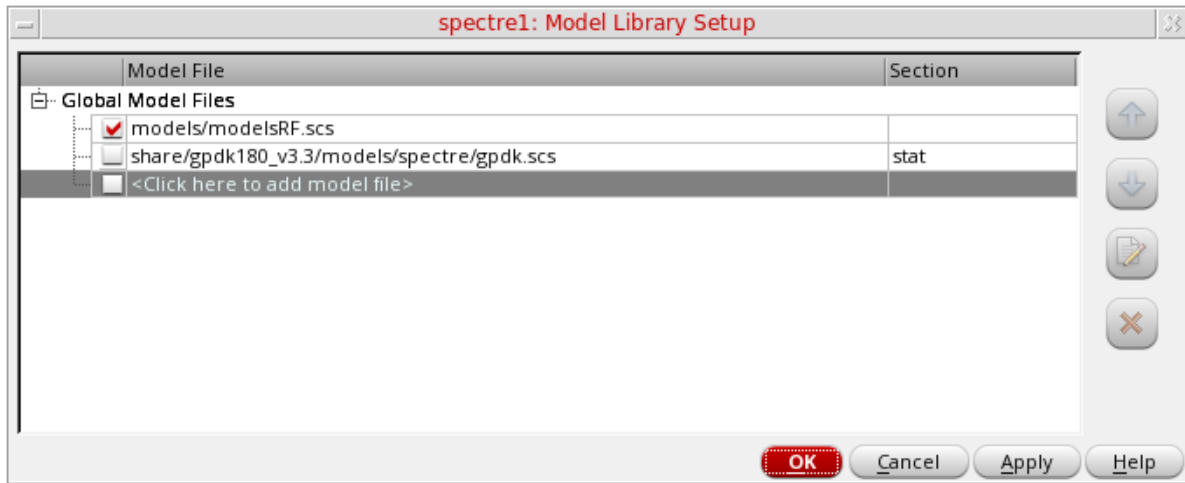
15. Select *Setup – Model Libraries*.

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**

Choosing Analyses -- ADE Explorer

Analysis

☒ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpsp

☐ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative ☐ moderate ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

OK Cancel Defaults Apply Help

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

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

---

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

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☒ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpss

☐ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

Stability Analysis

Sweep Variable

☒ Frequency

☐ Design Variable

☐ Temperature

☐ Component Parameter

☐ Model Parameter

☐ None

Sweep Range

☒ Start-Stop Start  Stop

☐ Center-Span

Sweep Type

Automatic

Add Specific Points ☐

Add Points By File ☐

Probe Instance/Terminal  Select

Local Ground Name  Select

Enabled ☐

Options...

OK Cancel Defaults Apply Help

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.

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

---

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:

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

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

The screenshot shows the 'Choosing Analyses -- ADE Explorer' dialog box. The 'Analysis' section has a grid of radio buttons for various analysis types. The 'stb' option is selected and highlighted with a red box. Below this, the 'Stability Analysis' section is visible. The 'Sweep Variable' section has 'Frequency' selected. The 'Sweep Range' section has 'Start-Stop' selected, with 'Start' set to '1m' and 'Stop' set to '1T'. The 'Sweep Type' section has 'Logarithmic' selected in a dropdown menu. The 'Points Per Decade' option is selected, with a value of '100' entered in the adjacent field. The 'Add Specific Points' and 'Add Points By File' checkboxes are unchecked. The 'Probe Instance/Terminal' field is set to '/IPRB0' and is highlighted with a red box. The 'Local Ground Name' field is empty. At the bottom, the 'Enabled' checkbox is checked, and there are buttons for 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'.

Choosing Analyses -- ADE Explorer

Analysis

- ☐ tran
- ☐ dc
- ☐ ac
- ☐ noise
- ☐ xf
- ☐ sens
- ☐ dcmatch
- ☐ acmatch
- ☒ stb
- ☐ pz
- ☐ lf
- ☐ sp
- ☐ envlp
- ☐ pss
- ☐ pac
- ☐ pstb
- ☐ pnoise
- ☐ pxf
- ☐ psp
- ☐ qpss
- ☐ qpac
- ☐ qpnoise
- ☐ qpxf
- ☐ qpssp
- ☐ hb
- ☐ hbac
- ☐ hbstb
- ☐ hbnoise
- ☐ hbasp
- ☐ hbxf

Stability Analysis

Sweep Variable

- ☒ Frequency
- ☐ Design Variable
- ☐ Temperature
- ☐ Component Parameter
- ☐ Model Parameter
- ☐ None

Sweep Range

- ☒ Start-Stop
- ☐ Center-Span

Start: 1m Stop: 1T

Sweep Type

Logarithmic

- ☒ Points Per Decade
- ☐ Number of Steps

100

Add Specific Points ☐

Add Points By File ☐

Probe Instance/Terminal: /IPRB0 Select

Local Ground Name: Select

Enabled ☒

Options...

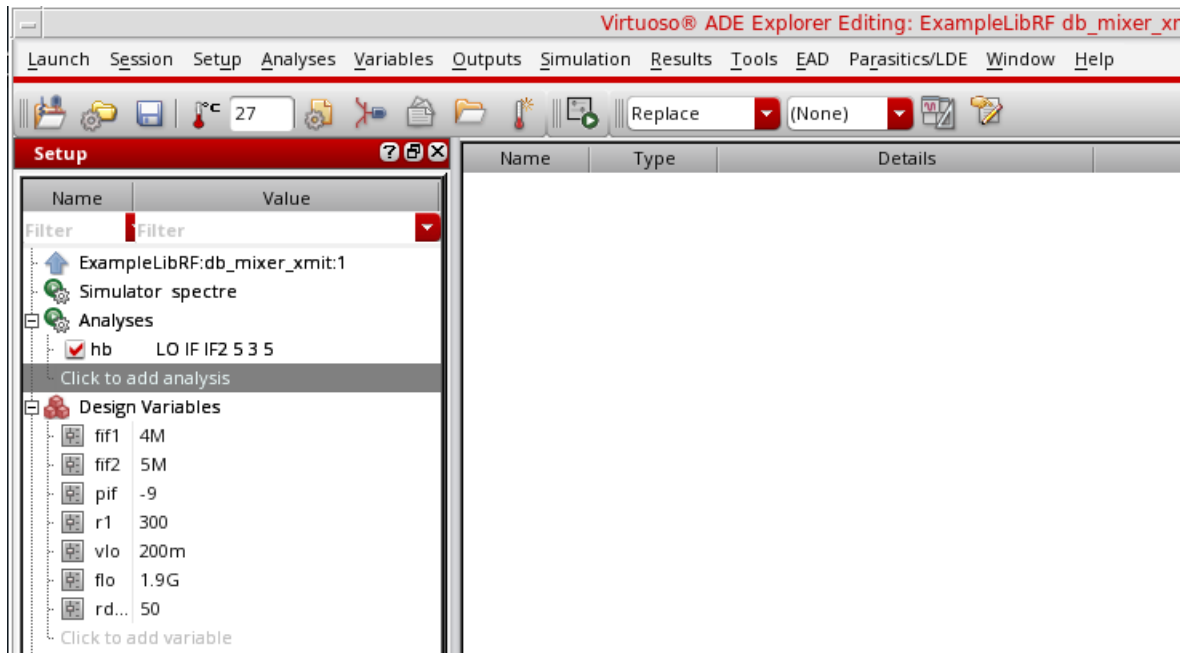
OK Cancel Defaults Apply Help

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:

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

---

**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**

☐ qnoise ☐ qpxf ☐ qpsp ☒ hb

☐ hbac ☐ hbstb ☐ hbnoise ☐ hbasp

☐ hbx

Harmonic Balance Analysis

Transient-Aided Options

Run transient? Decide automatically

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

osc!

Fundamental Frequency

Number of Harmonics

Oversample Factor

Freqdivide Ratio for Tone 1

Harmonics Default

Accuracy Defaults (errpreset)

☒ conservative ☐ moderate ☐ liberal

Oscillator ☒ Oscillator node+  Select

Oscillator node-  Select

☒ Calculate initial conditions (ic) automatically

☐ Enable tuning mode analysis

☐ Use the probe-based solution method

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☒

Options...

OK Cancel Defaults Apply Help

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



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

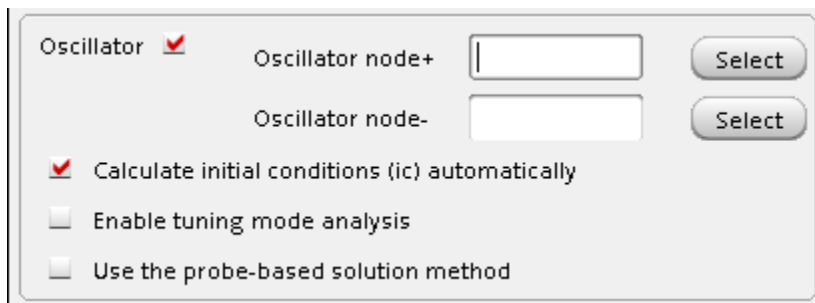
---

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**



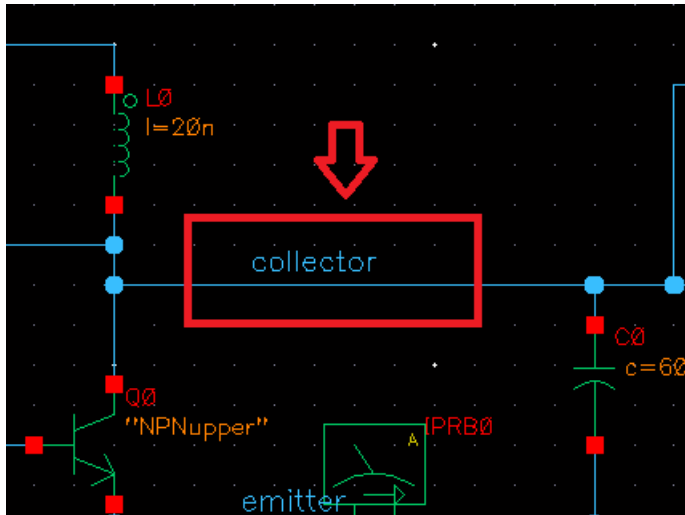
The screenshot shows a dialog box titled "Oscillator" with a red checkmark icon. It contains the following elements:

- A label "Oscillator node+" followed by a text input field and a "Select" button.
- A label "Oscillator node-" followed by a text input field and a "Select" button.
- A checked checkbox labeled "Calculate initial conditions (ic) automatically".
- An unchecked checkbox labeled "Enable tuning mode analysis".
- An unchecked checkbox labeled "Use the probe-based solution method".

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 *reference* 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:

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

Figure 3-44 Choosing Analyses Form - Harmonic Balance Setup

qnoise qpxf qpdp hb  
hbac hbstd hbnoise hbss  
hbxf

Harmonic Balance Analysis

Transient-Aided Options

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) no **yes**

Dynamic Parameter

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

oscl

Fundamental Frequency **1.96**

Number of Harmonics **15**

Oversample Factor **1**

Freqdivide Ratio for Tone 1 **1**

Harmonics **Default**

Accuracy Defaults (errpreset)

**conservative** moderate liberal

**Oscillator** ☒ Oscillator node+ /collector **Select**

Oscillator node- **Select**

**Calculate initial conditions (ic) automatically**

☐ Enable tuning mode analysis

☐ Use the probe-based solution method

Sweep

Loadpull

LSSP

Compression

Enabled ☒ Options...

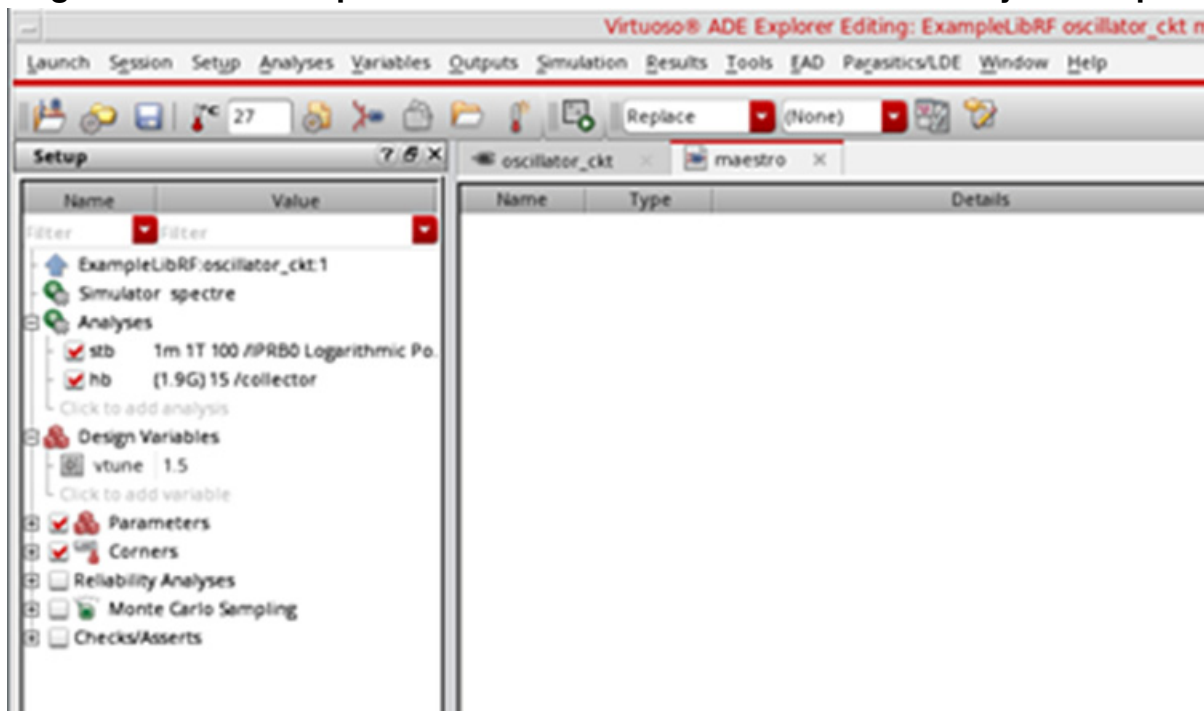
**OK** Cancel Defaults Apply Help

- 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:

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

---

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*.

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

---

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

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpss

☐ hb ☐ hbac ☒ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

Harmonic Balance Stability Analysis

Harmonic Balance Stability Analysis Notification

Start-Stop  Start  Stop

Sweep Type

Automatic

Add Specific Points ☐

Add Points By File ☐

Probe Instance

Local Ground Name

Enabled ☐

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.

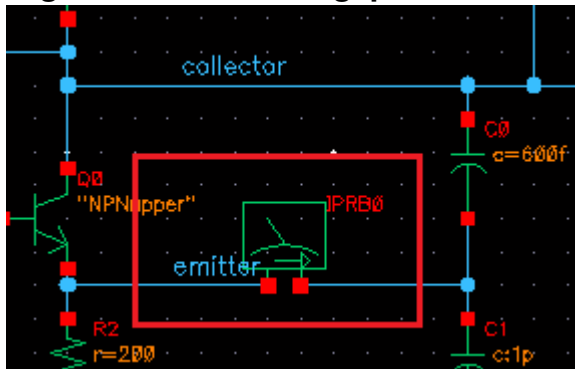
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

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

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

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

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpsp

☐ hb ☐ hbac ☒ hbstb ☐ hbnoise

☐ hbzp ☐ hbxf

Harmonic Balance Stability Analysis

Harmonic Balance Stability Analysis Notification

Start-Stop  Start 1.7G Stop 2G

Sweep Type  Linear

☐ Step Size

☒ Number of Steps 100

Add Specific Points ☐

Add Points By File ☐

Probe Instance /IPRB0

Local Ground Nam

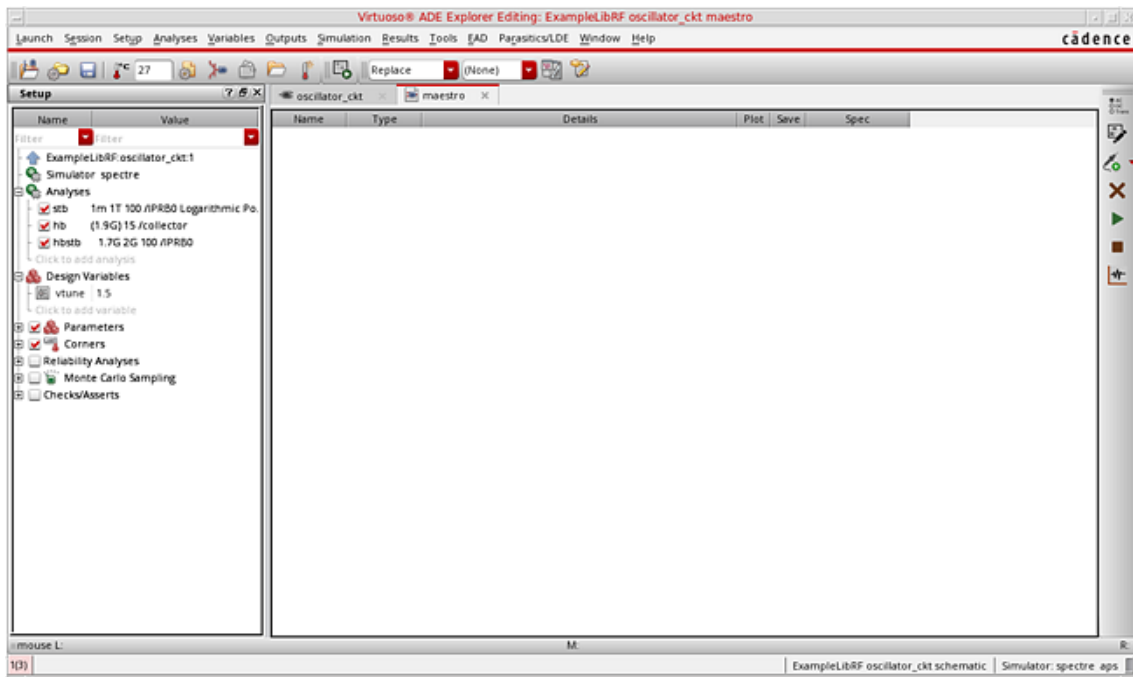
Enabled ☒

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.

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

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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☒ stb ☐ hb ☐ hbstb ☐ tstab

**Function**

☒ Loop Gain ☐ Stability Summary  
☐ Phase Margin ☐ Gain Margin  
☐ PM Frequency ☐ GM Frequency

**Modifier**

☐ Magnitude ☐ Phase ☒ Magnitude and Phase

**Magnitude Modifier**

☐ None ☐ dB10 ☒ dB20

Add To Outputs ☐ Plot

> Press plot button on this form...

Close Help

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.

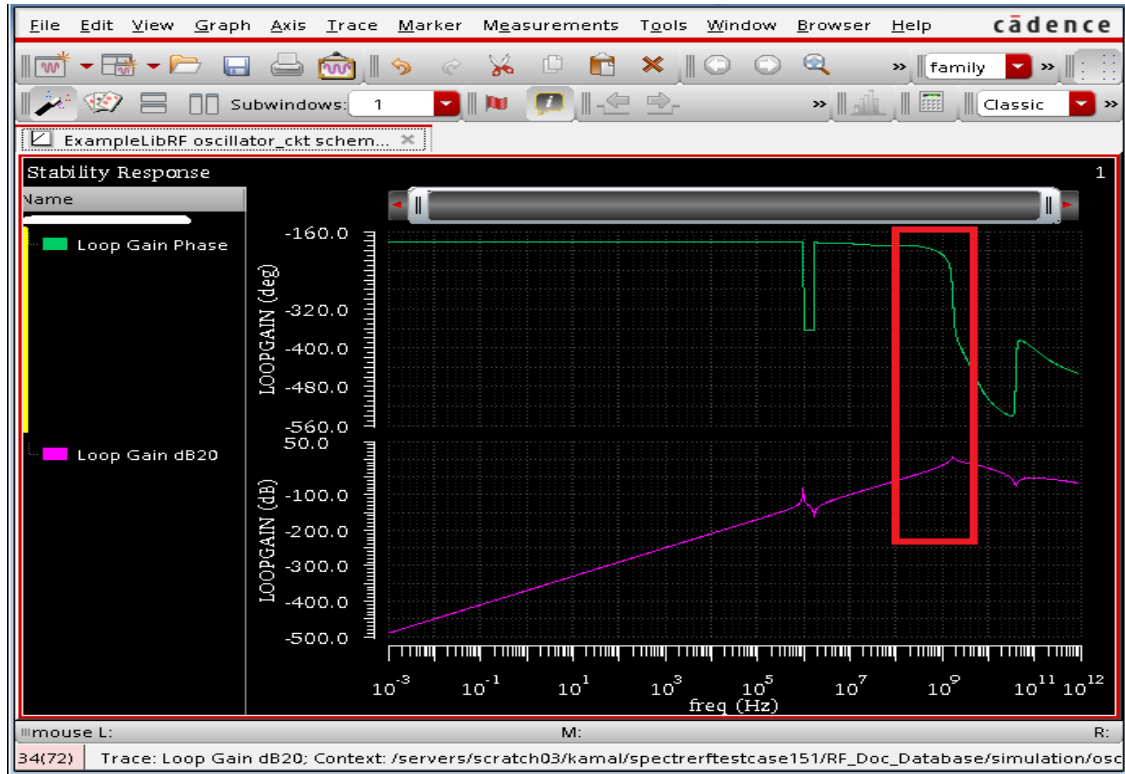
The Direct Plot Form looks like the following:

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

The image shows the 'Direct Plot Form' dialog box for setting up an *stb* analysis. The window has a title bar 'Direct Plot Form' with standard OS controls. Inside, the 'Plotting Mode' is set to 'Append'. The 'Analysis' section has radio buttons for 'stb' (selected), 'hb', 'hbstd', and 'tstab'. The 'Function' section has radio buttons for 'Loop Gain' (selected), 'Stability Summary', 'Phase Margin', 'Gain Margin', 'PM Frequency', and 'GM Frequency'. The 'Modifier' section has radio buttons for 'Magnitude', 'Phase', and 'Magnitude and Phase' (selected). The 'Magnitude Modifier' section has radio buttons for 'None', 'dB10', and 'dB20' (selected). At the bottom, there is an 'Add To Outputs' checkbox, a 'Plot' button, and a red 'Close' button. A help button is also present. A note at the bottom says '> Press plot button on this form...'.

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 Output Graph Window - Loop Gain Magnitude and Phase Plot**



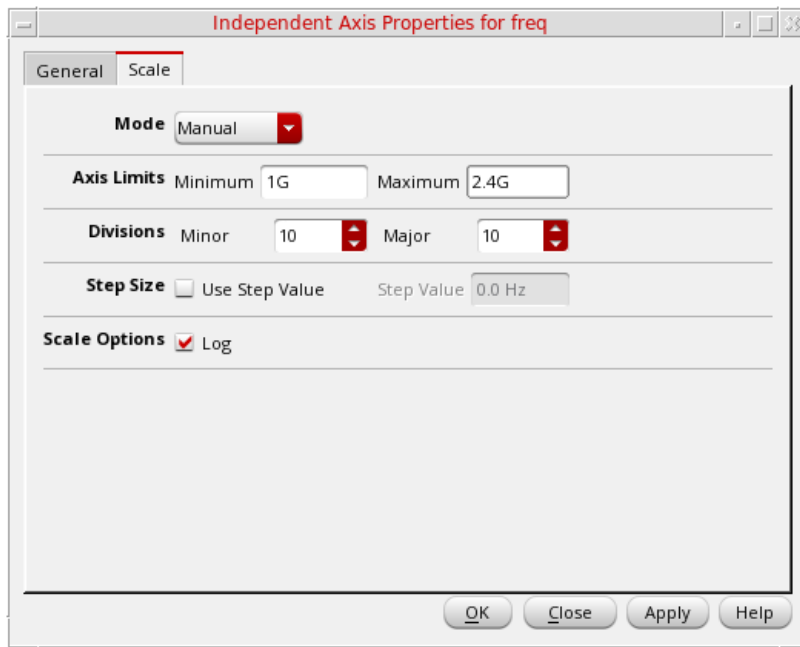
On the top of the graph is a plot of loop gain phase in degrees. On the bottom of the graph is a plot of loop gain magnitude in dB. The area in the red box is the unity gain area.

**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:

- Move your mouse cursor over one of the numbers on the X axis.
- Click the right mouse button and select *Axis Properties*.
- Click the *Scale* tab and set *Mode* to *Manual*.
- Type 1G in the *Minimum cyclic* field. This is near to the frequency before the unity gain area starts.
- Type 2.4G in the *Maximum cyclic* field. This is frequency near to which the unity gain area ends.

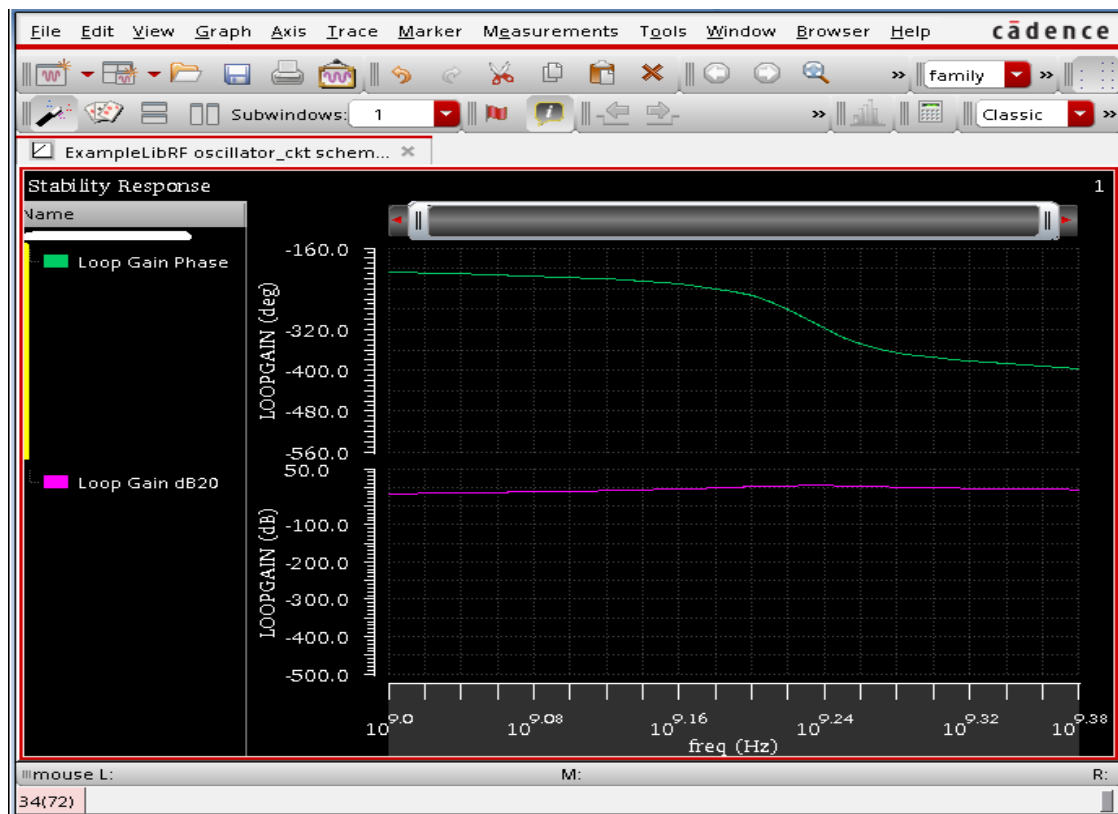
**Figure 3-53 Graph Window X-Axis Setup**



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:

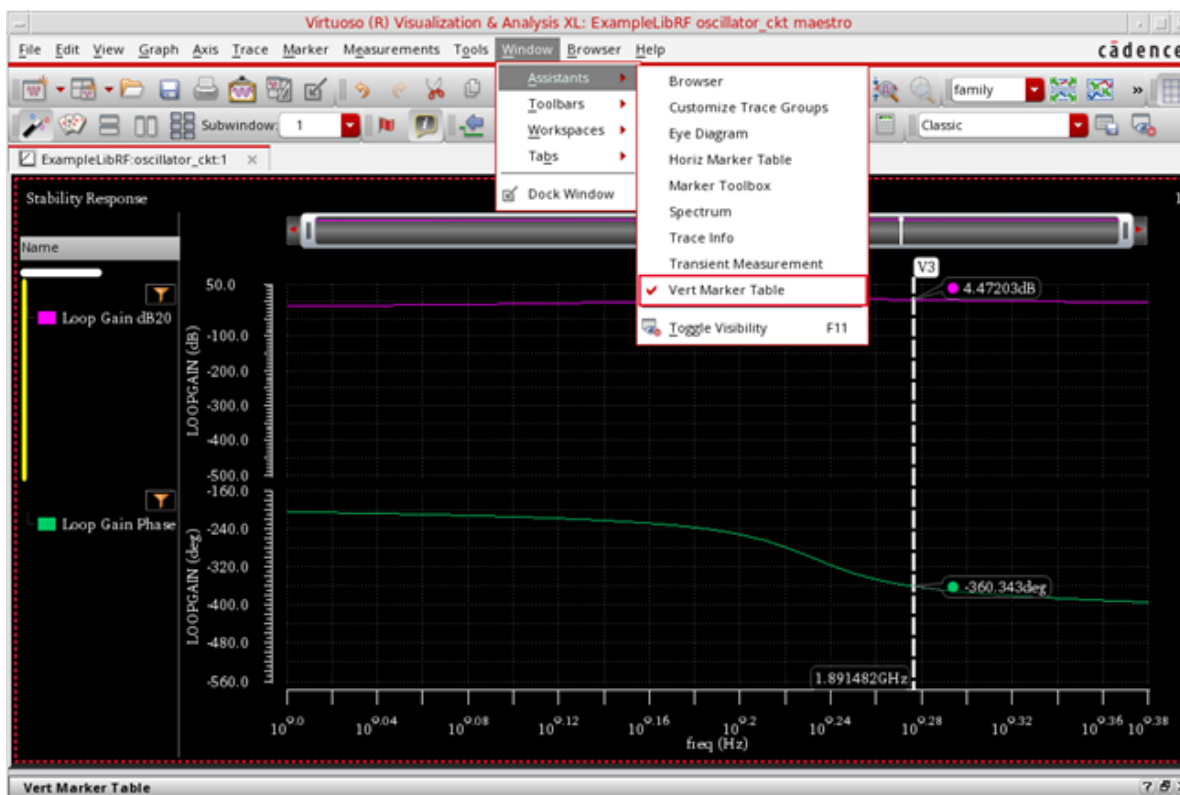
Figure 3-55 stb Analysis - 360 degree Marker Placement



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

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

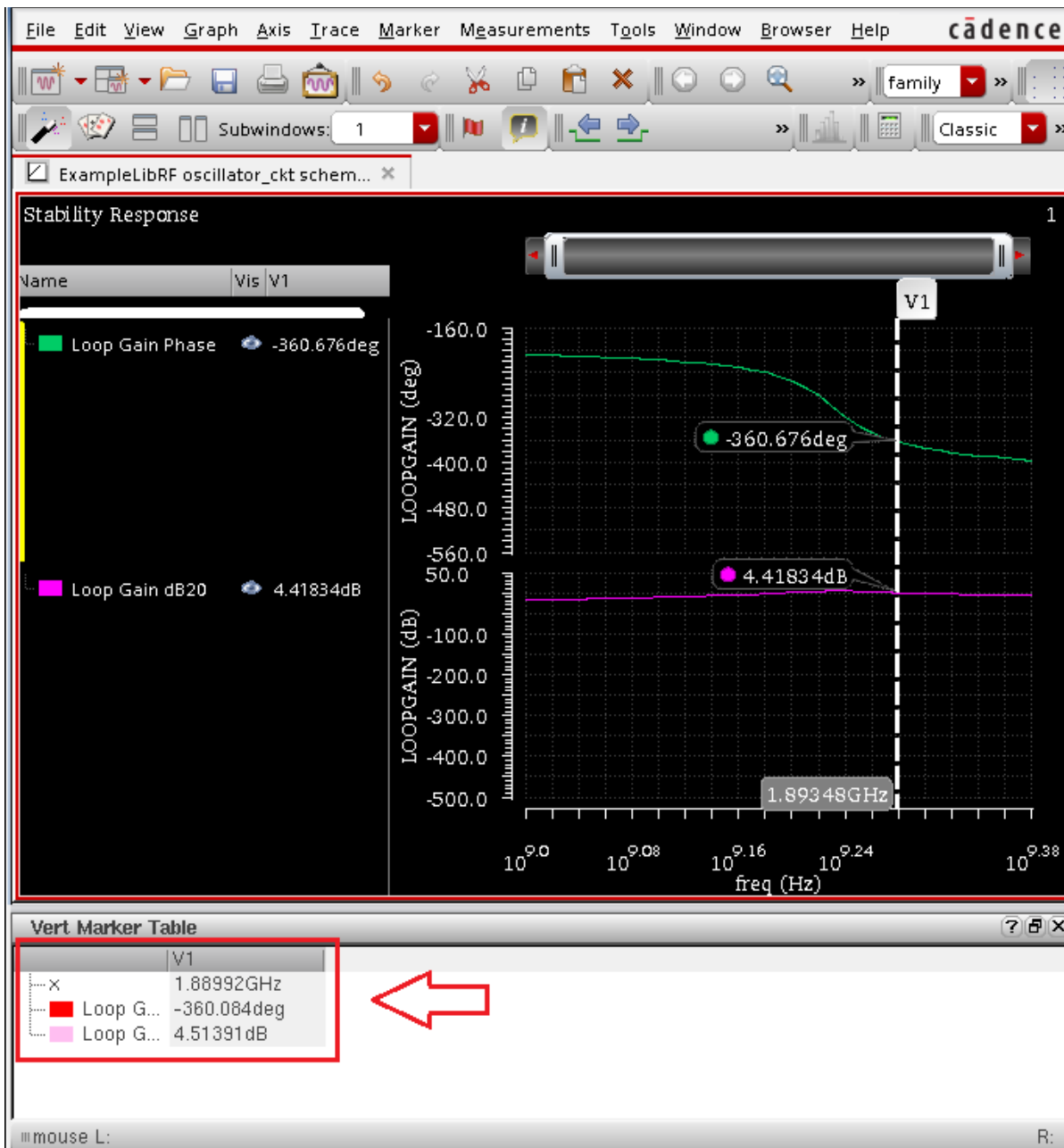
Figure 3-56 ViVA XL - Selecting Marker Toolbox



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*.

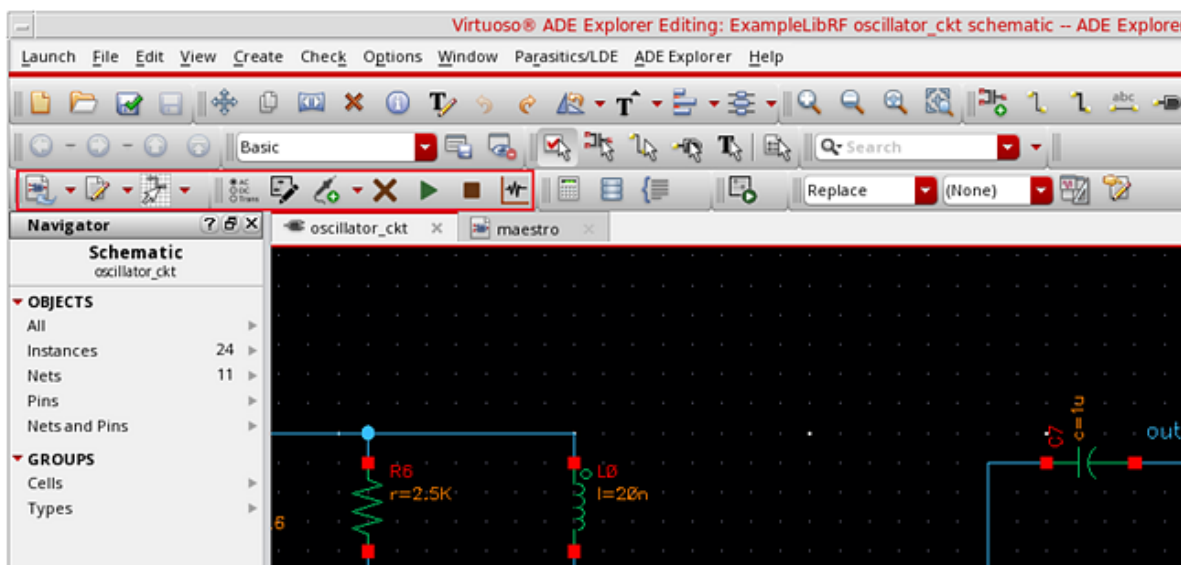


## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

### 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**

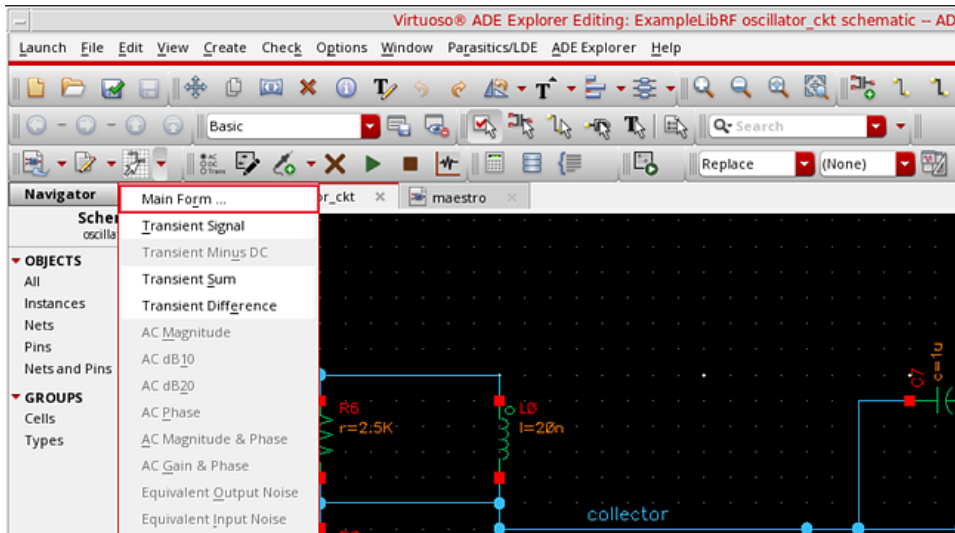


You may refer to *Virtuoso ADE Explorer User Guide* for more information regarding these simulation menu icons.

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:

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

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**

The image shows the 'Direct Plot Form' dialog box in a software application. The dialog is titled 'Direct Plot Form' and has a close button in the top right corner. It contains several sections with radio buttons and checkboxes. The 'Plotting Mode' is set to 'Append'. The 'Analysis' section has three radio buttons: 'stb', 'hb', and 'hbstb', with 'hbstb' selected and highlighted by a red box. The 'Function' section has two radio buttons: 'Loop Gain' and 'Stability Summary', with 'Loop Gain' selected and highlighted by a red box. The 'Modifier' section has four radio buttons: 'Magnitude', 'Phase', 'Magnitude and Phase', and 'Imag vs. Real', with 'Magnitude' selected and highlighted by a red box. The 'Magnitude Modifier' section has three radio buttons: 'None', 'dB10', and 'dB20', with 'dB20' selected and highlighted by a red box. There is a 'Loadpull Contours' checkbox which is unchecked. At the bottom, there is an 'Add To Outputs' checkbox which is unchecked, and a 'Plot' button. Below the 'Plot' button is a text prompt '> Press plot button on this form...'. At the very bottom are 'Close' and 'Help' buttons.

Direct Plot Form

Plotting Mode: Append

**Analysis**

☐ stb ☐ hb ☒ hbstb ☐ tstab

**Function**

☒ Loop Gain ☐ Stability Summary

**Modifier**

☒ Magnitude ☐ Phase ☐ Magnitude and Phase  
☐ Imag vs. Real

**Magnitude Modifier**

☐ None ☐ dB10 ☒ dB20

Loadpull Contours ☐

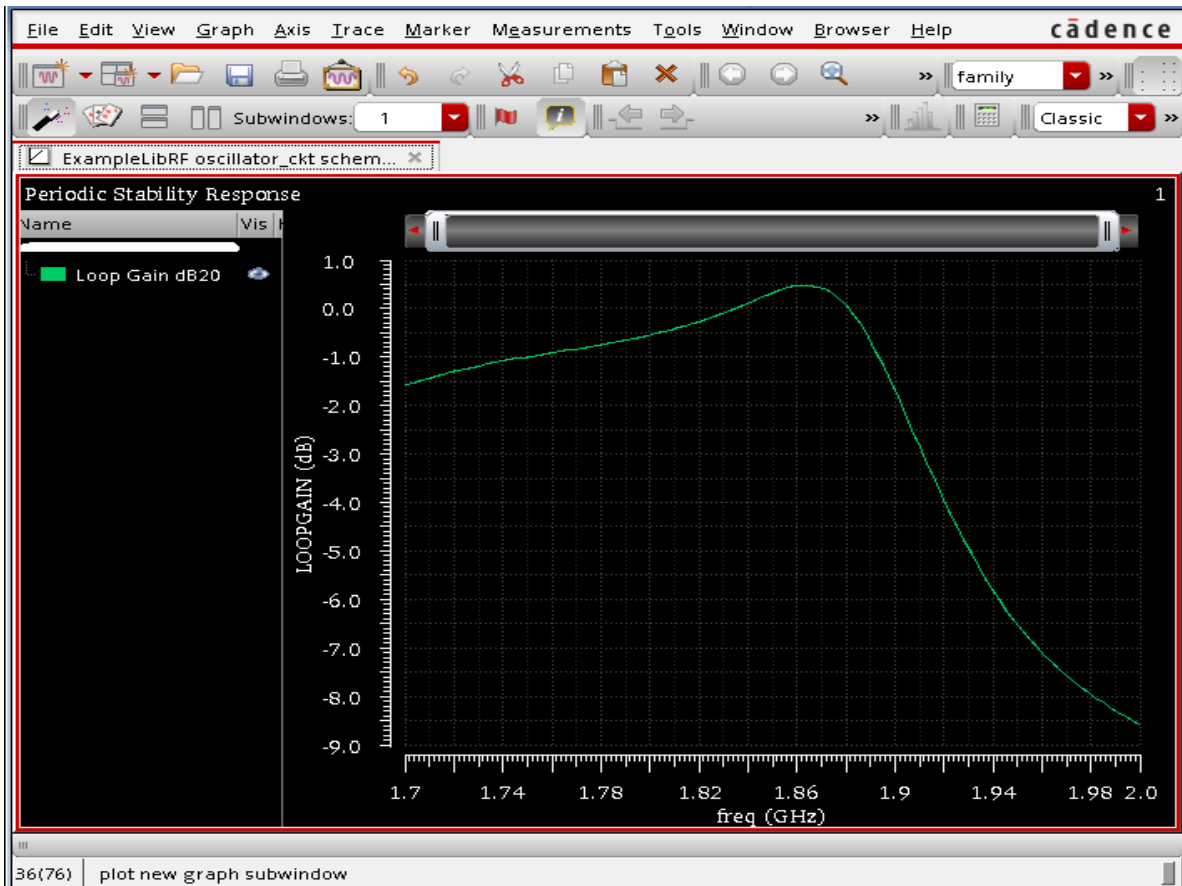
Add To Outputs ☐ Plot

> Press plot button on this form...

Close Help

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

Figure 3-61 HBSTB Loop Gain Magnitude Plot



- 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**

The image shows the 'Direct Plot Form' dialog box in a software application. The title bar is 'Direct Plot Form'. Inside, the 'Plotting Mode' is set to 'Append'. Under the 'Analysis' section, there are four radio buttons: 'stb', 'hb', 'hbstb' (which is selected and highlighted with a red box), and 'tstab'. Under the 'Function' section, there are two radio buttons: 'Loop Gain' (selected) and 'Stability Summary'. Under the 'Modifier' section, there are three radio buttons: 'Magnitude', 'Phase' (selected and highlighted with a red box), and 'Magnitude and Phase'. There is also an 'Imag vs. Real' radio button. Below these sections, there is a 'Loadpull Contours' checkbox which is unchecked. At the bottom, there is an 'Add To Outputs' checkbox which is unchecked, and a 'Plot' button. At the very bottom, there is a red 'Close' button and a 'Help' button. A text prompt at the bottom says '> Press plot button on this form...'.

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

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

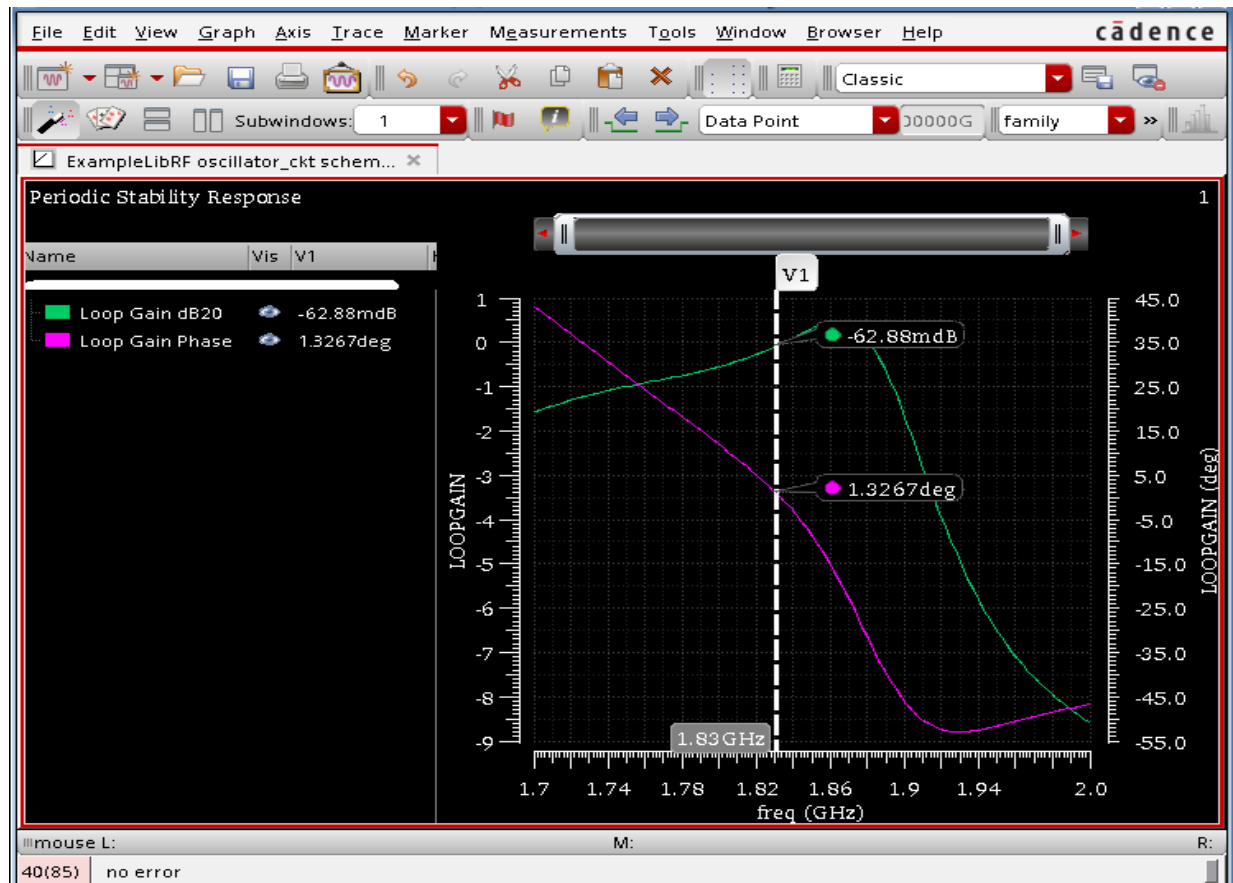


**Note:** Loop Gain's Magnitude and Phase are plotted separately as when *Magnitude and Phase* is selected, there is no dB20 modifier for the loop gain.

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:

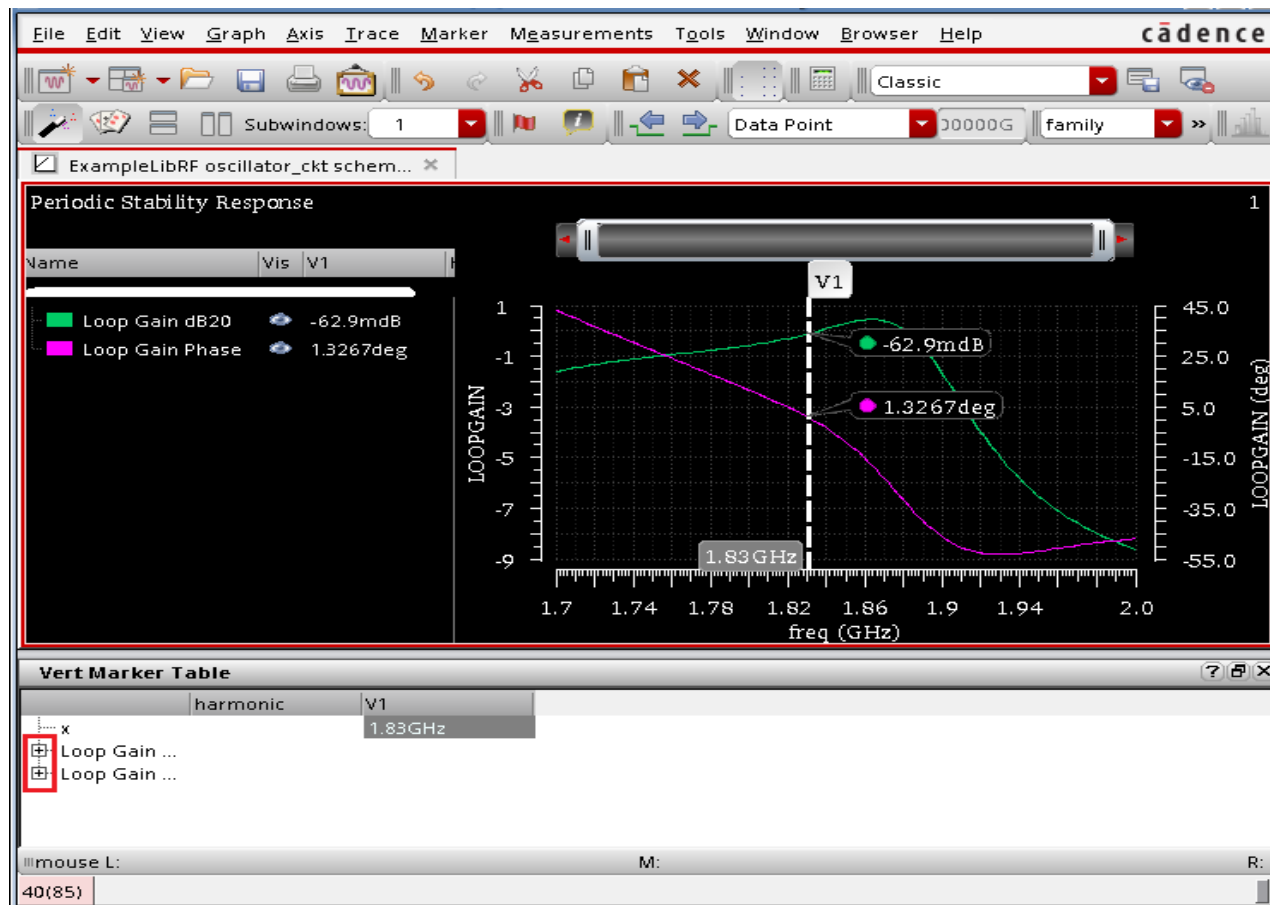
Figure 3-64 hbstb Loop Gain magnitude and phase plot with vertical marker



### 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.

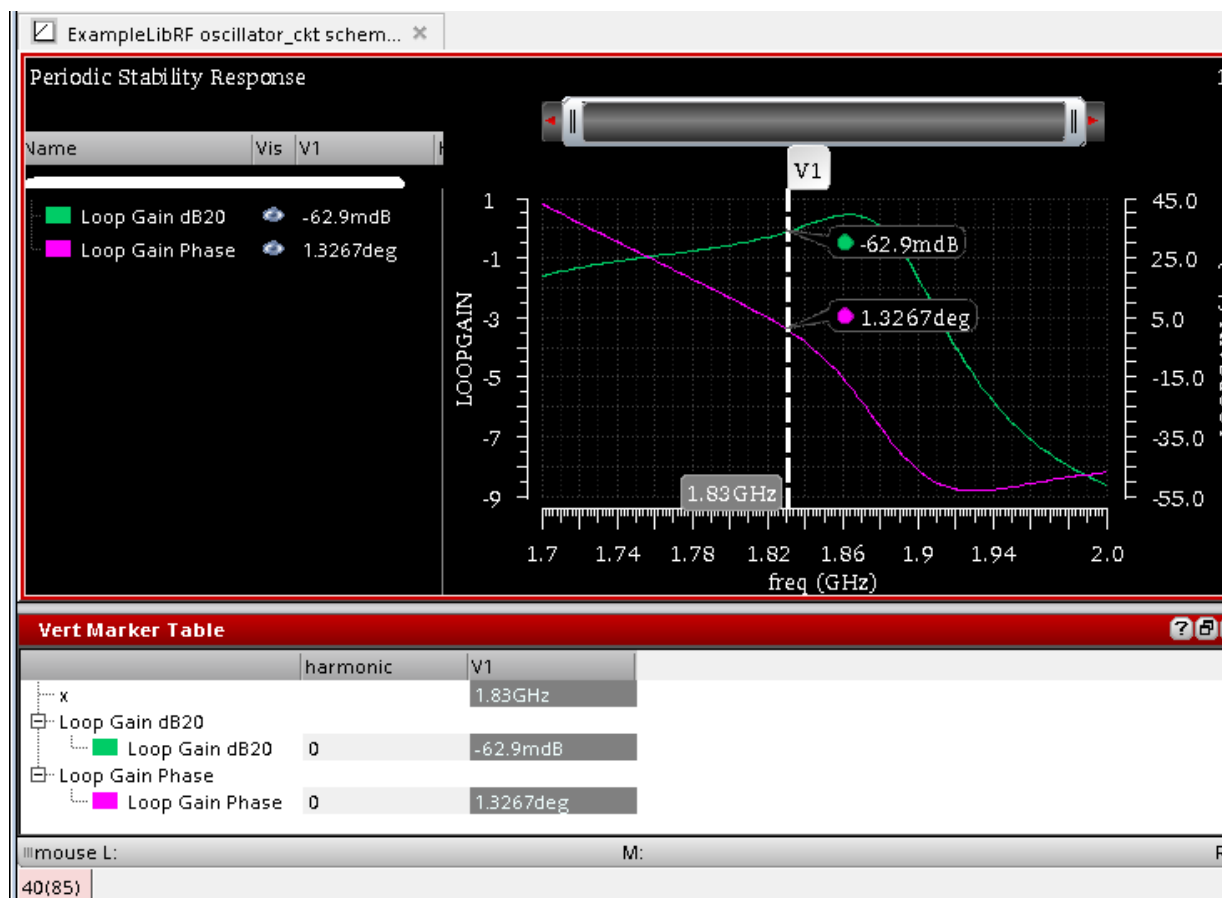
**Figure 3-65 hbstb Analysis Loop Gain Plot with unexpanded Verticle Marker Table**



- 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*.

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

---

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**

Direct Plot Form

Plotting Mode: New Win

**Analysis**

☐ stb ☒ hb ☐ hbstb ☐ tstab

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Harmonic Frequency ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp. ☐ THD

Select: Net

**Sweep**

☒ spectrum ☐ time

Signal Level: ☐ peak ☒ rms

**Modifier**

☐ Magnitude ☐ Phase ☒ dB20  
☐ Real ☐ Imaginary

**Scalar Expression**

Value At (harms):

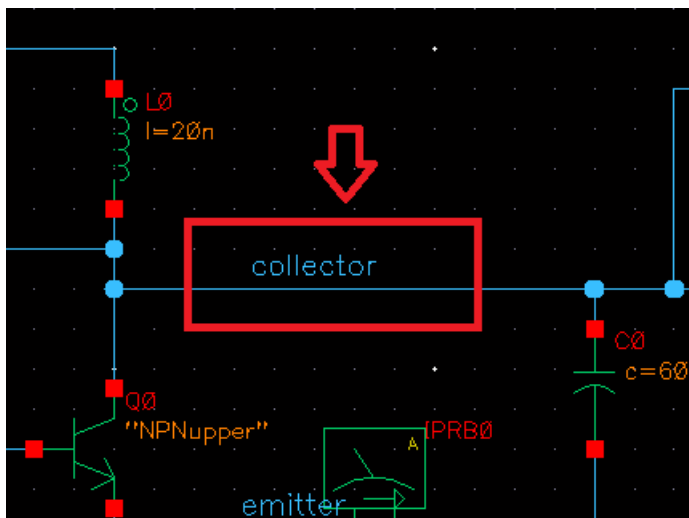
Add To Outputs: ☐

> Select Net on schematic...

Close Help

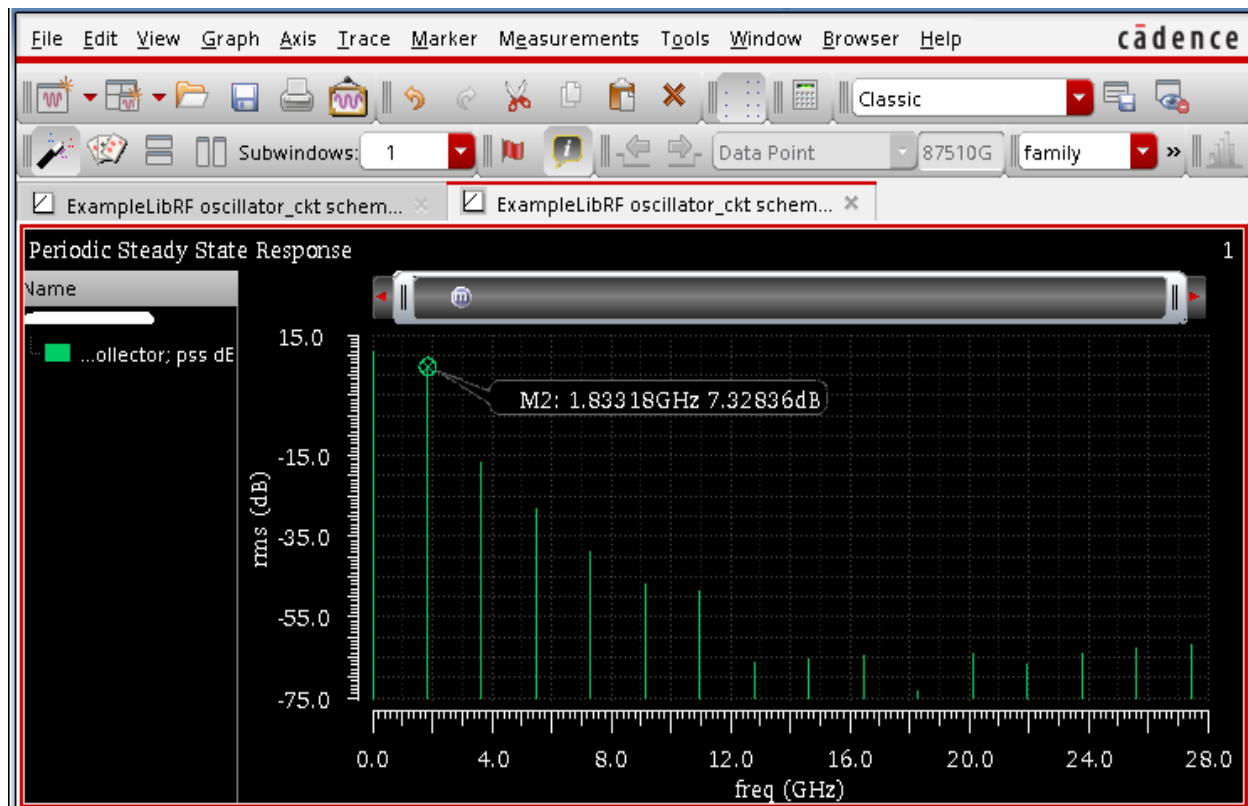
9. Select the *collector* net in the Schematic. It is located just below the *collector* label.

**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



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

---

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.

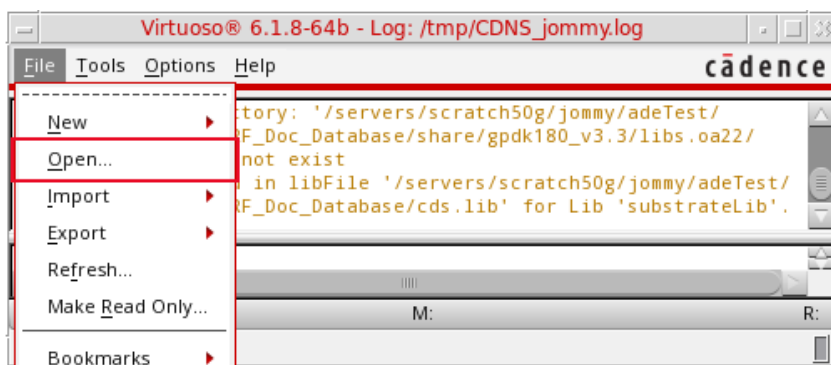
Next, the phase noise measurement which is one of the key measurement in oscillator design will be done. Also, the Noise Summary Table will be obtained.

## 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**



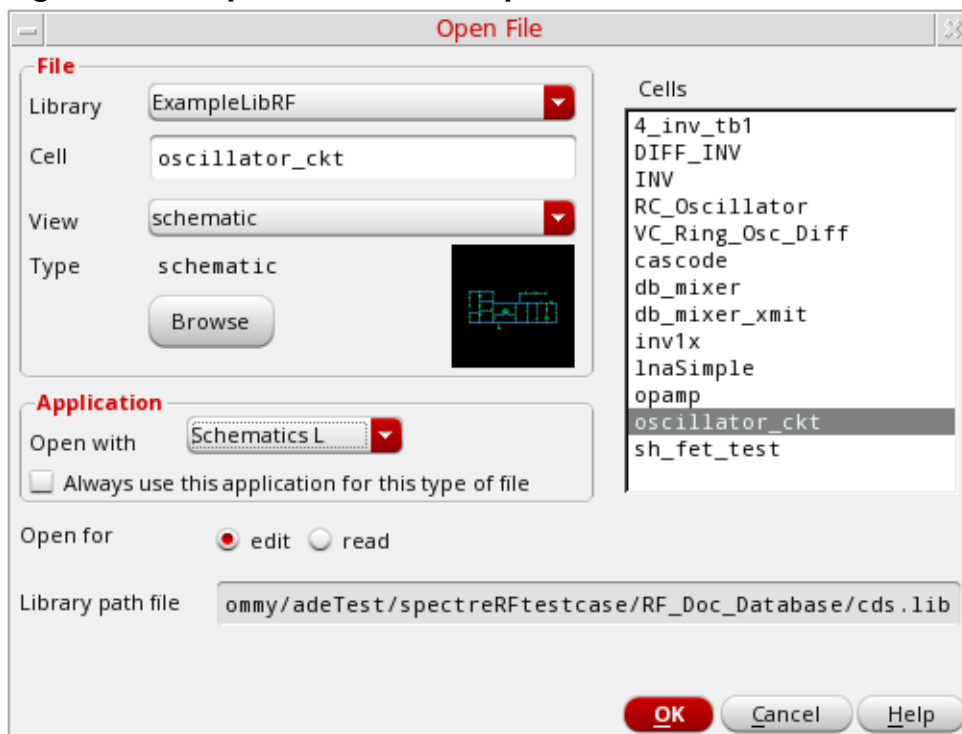
The *Open File* form is displayed.

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

---

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).

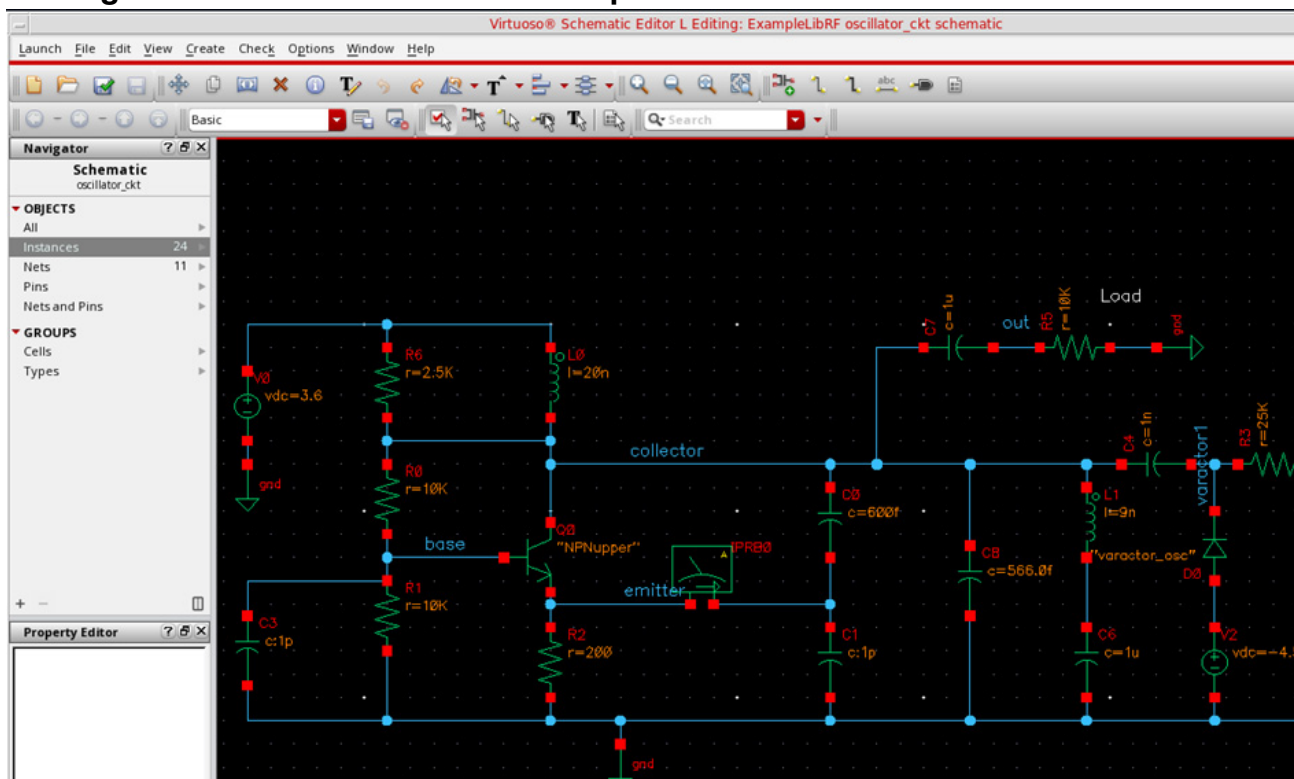
**Figure 3-71 Open File Form to open the oscillator\_ckt cell's Schematic View**



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

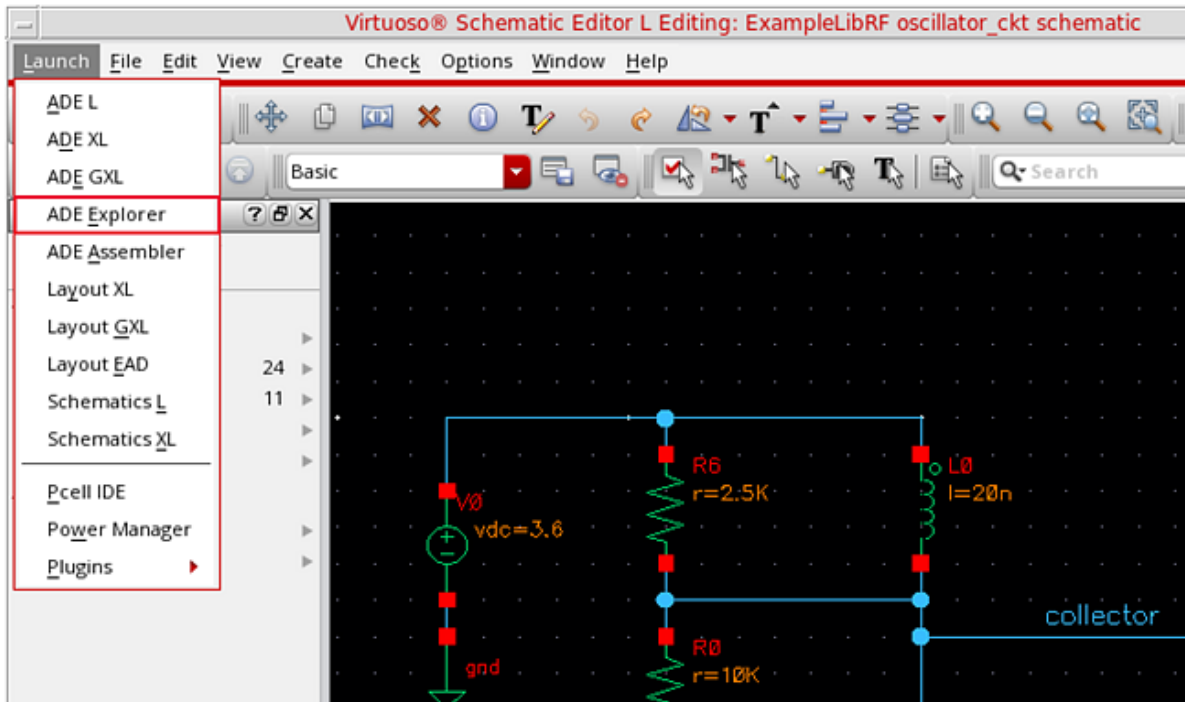
In this section, for performing phase noise measurement, you would be using the *hb* and *hbnoise* analyses.

## Setting up the HB Analysis

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

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

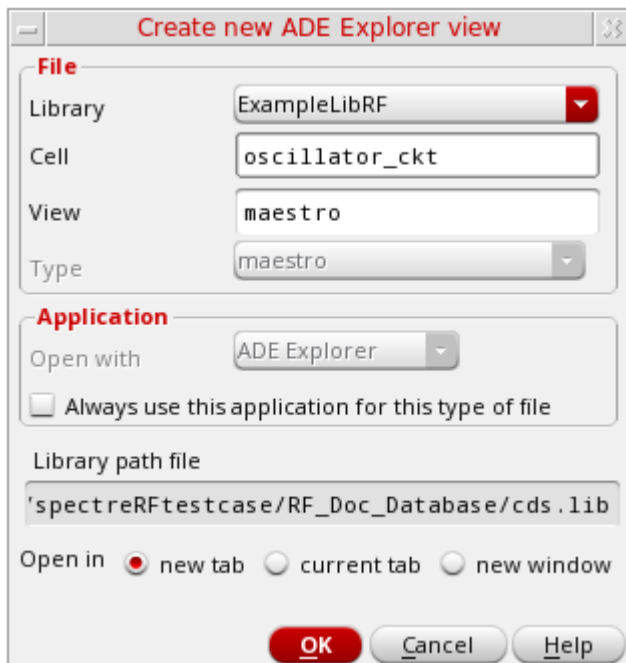
**Figure 3-73 Opening ADE Explorer window from VSE window**



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**



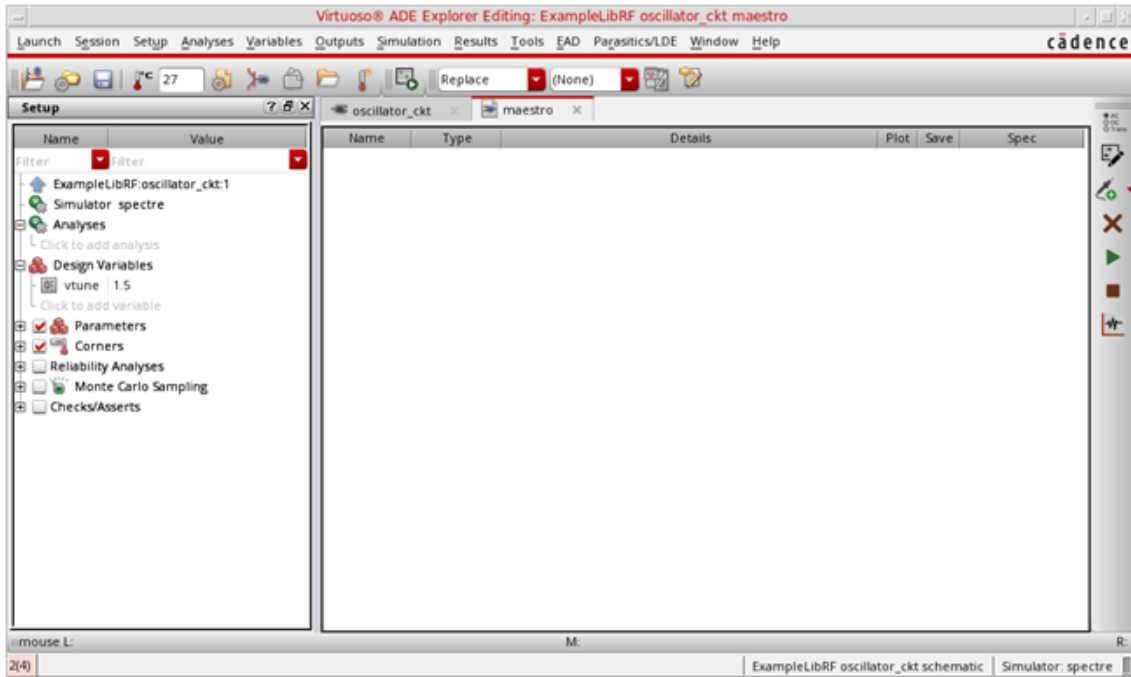
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

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*.

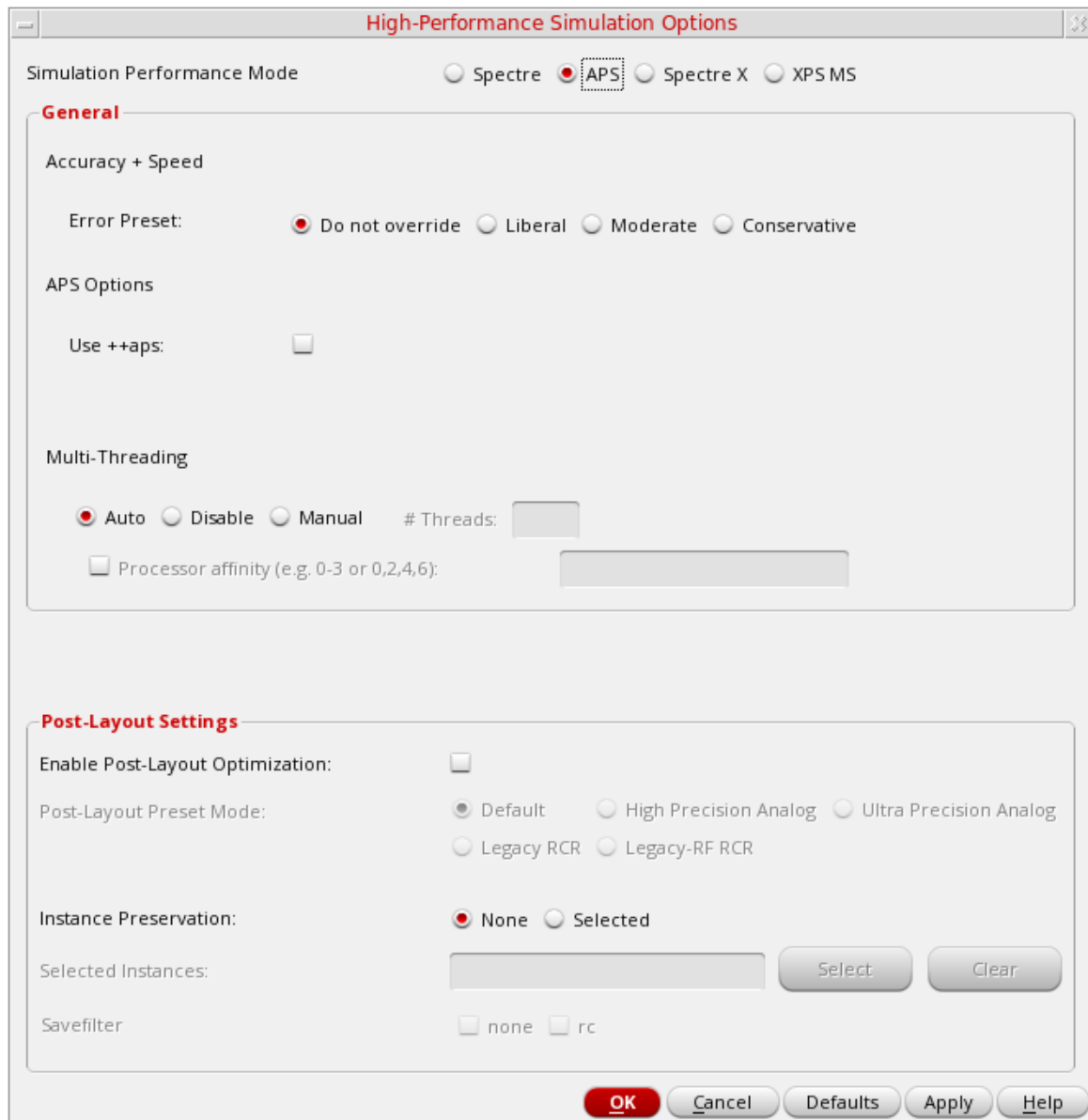
**Figure 3-76 Choosing Simulator Form**



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**



The image shows a software dialog box titled "High-Performance Simulation Options". At the top, under "Simulation Performance Mode", there are four radio buttons: "Spectre", "APS" (which is selected and highlighted with a dashed box), "Spectre X", and "XPS MS". Below this is a "General" section. Under "Accuracy + Speed", the "Error Preset:" has four radio buttons: "Do not override" (selected), "Liberal", "Moderate", and "Conservative". Under "APS Options", there is a checkbox for "Use ++aps:" which is currently unchecked. Under "Multi-Threading", there are three radio buttons: "Auto" (selected), "Disable", and "Manual", followed by a "# Threads:" text input field. Below that is a checkbox for "Processor affinity (e.g. 0-3 or 0,2,4,6):" which is unchecked, followed by another text input field. The bottom section is "Post-Layout Settings". It has a checkbox for "Enable Post-Layout Optimization:" which is unchecked. Below that, "Post-Layout Preset Mode:" has five radio buttons: "Default" (selected), "High Precision Analog", "Ultra Precision Analog", "Legacy RCR", and "Legacy-RF RCR". "Instance Preservation:" has two radio buttons: "None" (selected) and "Selected". Below this is a "Selected Instances:" text input field, followed by "Select" and "Clear" buttons. At the bottom of this section is a "Savefilter" with two checkboxes: "none" and "rc". At the very bottom of the dialog are five buttons: "OK" (highlighted in red), "Cancel", "Defaults", "Apply", and "Help".

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**

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHD variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

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**

The screenshot shows the 'spectre1: Simulation Files Setup' dialog box. It has two tabs: 'Paths/Files' (selected) and 'Vector Files'. The 'Paths/Files' tab contains a table with columns 'Files/Paths' and 'Options'. The 'Files/Paths' column is expanded, showing a tree view with the following items:

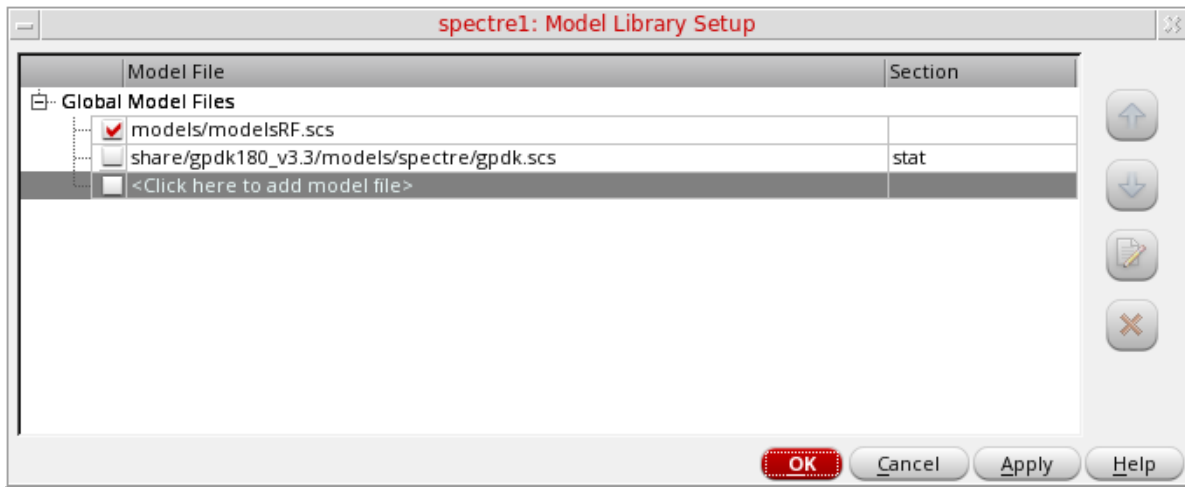
- ☒ Include Paths
- ☐ Definition Files
- ☐ Stimulus Files
- ☐ Parasitic Files (DSPF)
- ☐ Parasitic Files (SPEF)
- ☐ P Spice Files

Each item has a corresponding input field with a '<Click here to add a path>' or '<Click here to add a file>' prompt. The 'Options' column is empty. On the right side of the dialog, there are two buttons: a plus sign in a circle (Add) and a minus sign in a circle (Remove).

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.

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

---

**Figure 3-81 The Choosing Analyses Form**

Choosing Analyses -- ADE Explorer

Analysis

☒ tran   ☐ dc   ☐ ac   ☐ noise

☐ xf   ☐ sens   ☐ dcmatch   ☐ acmatch

☐ stb   ☐ pz   ☐ lf   ☐ sp

☐ envlp   ☐ pss   ☐ pac   ☐ pstb

☐ pnoise   ☐ pxf   ☐ psp   ☐ qpss

☐ qpac   ☐ qpnoise   ☐ qpxf   ☐ qpss

☐ hb   ☐ hbac   ☐ hbstb   ☐ hbnoise

☐ hbsp   ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative   ☐ moderate   ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

**OK**   Cancel   Defaults   Apply   Help

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

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

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

The image shows a software dialog box titled "Choosing Analyses". At the top, under the "Analysis" section, there is a grid of radio buttons for various analysis types. The "hb" (Harmonic Balance) option is selected, indicated by a red dot. Other options include tran, dc, ac, noise, xf, sens, dcmatch, acmatch, stb, pz, sp, envlp, pss, pac, pstb, pnoise, pxf, psp, qpss, qpac, qpnoise, qpxf, qpss, qpac, hbac, hbnoise, and hbasp.

Below the analysis selection, the "Harmonic Balance Analysis" section is expanded. It contains several sub-sections:

- Transient-Aided Options:** Includes a "Run transient?" dropdown set to "Decide automatically", a "Detect Steady State" checkbox checked, a "Stop Time(tstab)" dropdown set to "auto", and a "Save Initial Transient Results (saveinit)" checkbox with "no" and "yes" options.
- Dynamic Parameter:** A checkbox that is currently unchecked.
- Tones:** Radio buttons for "Frequencies" (selected) and "Names".
- Number of Tones:** Radio buttons for 1, 2, 3, and 4. "1" is selected.
- Tone 1 settings:** Includes input fields for "Fundamental Frequency", "Number of Harmonics" (set to "auto"), "Oversample Factor" (set to "1"), and "Freqdivide Ratio for Tone 1" (set to "1").
- Harmonics:** A dropdown menu set to "Default".
- Accuracy Defaults (errpreset):** Radio buttons for "conservative", "moderate" (selected), and "liberal".
- Oscillator:** A checkbox that is currently unchecked.
- Sweep:** A checkbox that is currently unchecked.
- Loadpull:** A checkbox that is currently unchecked.
- LSSP:** A checkbox that is currently unchecked.
- Compression:** A checkbox that is currently unchecked.
- Enabled:** A checkbox that is currently unchecked.

At the bottom right of the dialog, there is an "Options..." button. At the very bottom, there are five buttons: "OK" (highlighted in red), "Cancel", "Defaults", "Apply", and "Help".

- 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.

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

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

☐ xf ☐ sens ☐ dcmatch ☐ acmatch  
☐ stb ☐ pz ☐ sp ☐ envlp  
☐ pss ☐ pac ☐ pstb ☐ pnoise  
☐ pxf ☐ psp ☐ qpss ☐ qpac  
☐ qpnoise ☐ qpxf ☐ qpsp ☒ hb  
☐ hbac ☐ hbnoise ☐ hbasp

Harmonic Balance Analysis

Transient-Aided Options

Run transient? Decide automatically

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ Names

Tones

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
|---|------|------|-------|-------|-------|-------|-------|

yes

Change Delete Update From Hierarchy

Frequency Ratio for tone with Tstab 1

Harmonics Default

Accuracy Defaults (errpreset)  
☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

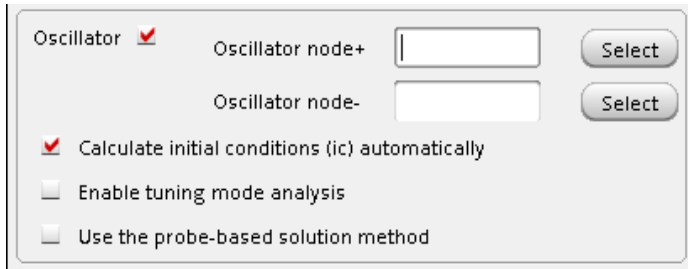
Enabled ☐ Options...

OK Cancel Defaults Apply Help



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

**Figure 3-84 The Choosing Analyses Form - Oscillator Section**



The screenshot shows a form titled "Oscillator" with a red checkmark icon. It contains two input fields for "Oscillator node+" and "Oscillator node-", each with a "Select" button to its right. Below these fields are three checkboxes: "Calculate initial conditions (ic) automatically" (checked), "Enable tuning mode analysis" (unchecked), and "Use the probe-based solution method" (unchecked).

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.

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

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

☐ pxf ☐ psp ☐ qpss ☐ qpac  
☐ qpnoise ☐ qpxf ☐ qpss ☒ hb  
☐ hbac ☐ hbnoise ☐ hbss

Harmonic Balance Analysis

Transient-Aided Options

Run transient? Decide automatically

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ Names

Tones

| # | Name | Expr | Value   | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|---------|-------|-------|-------|-------|
| 0 | osc! |      | * Nan * | auto  | 1     | yes   |       |

Freqdivide Ratio for tone with Tstab 1

Harmonics Default

Accuracy Defaults (errpreset)  
☐ conservative ☒ moderate ☐ liberal

Oscillator ☒ Oscillator node+    
Oscillator node-

☒ Calculate initial conditions (ic) automatically  
☐ Enable tuning mode analysis  
☐ Use the probe-based solution method

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

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.

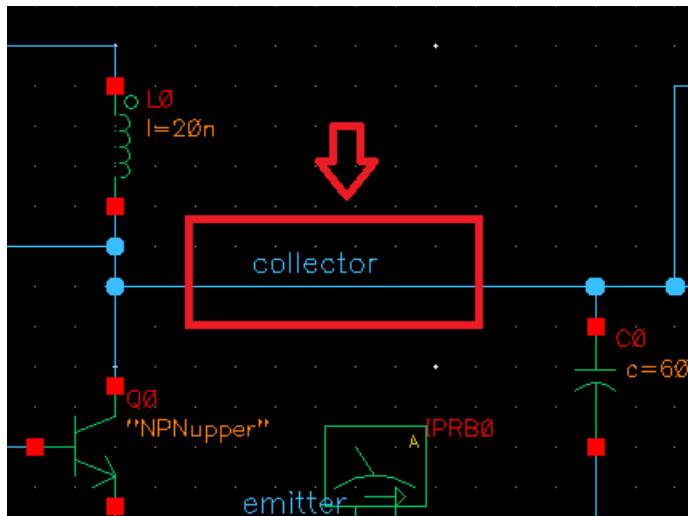
30. In the *Oscillator* section, in *Oscillator node+* field, click *Select* just to the right of this field. In the schematic, select the *collector* node. Instead of selecting the node from the schematic you can also type `/collector` in the *Oscillator node+* field.

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:

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

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

pxf psp qpss qpac  
qpnoise qpxf qpsh **hb**  
hbac hbnoise hbshp

Harmonic Balance Analysis

Transient-Aided Options

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) **auto**

Save Initial Transient Results (saveinit) ☐ no ☒ **yes**

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ **Names**

Tones

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 0 | osc! | 1.9G | 1.9G  | auto  | 1     | yes   |       |

osc! **1.9G** 1.9G auto 1 yes ☒

**Change** Delete Update From Hierarchy

Frequdivide Ratio for tone with Tstab **1**

Harmonics **Default**

Accuracy Defaults (errpreset)

☒ **conservative** ☐ moderate ☐ liberal

Oscillator ☒ Oscillator node+ **/collector** Select

Oscillator node- Select

☒ Calculate initial conditions (ic) automatically

☐ Enable tuning mode analysis

☐ Use the probe-based solution method

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

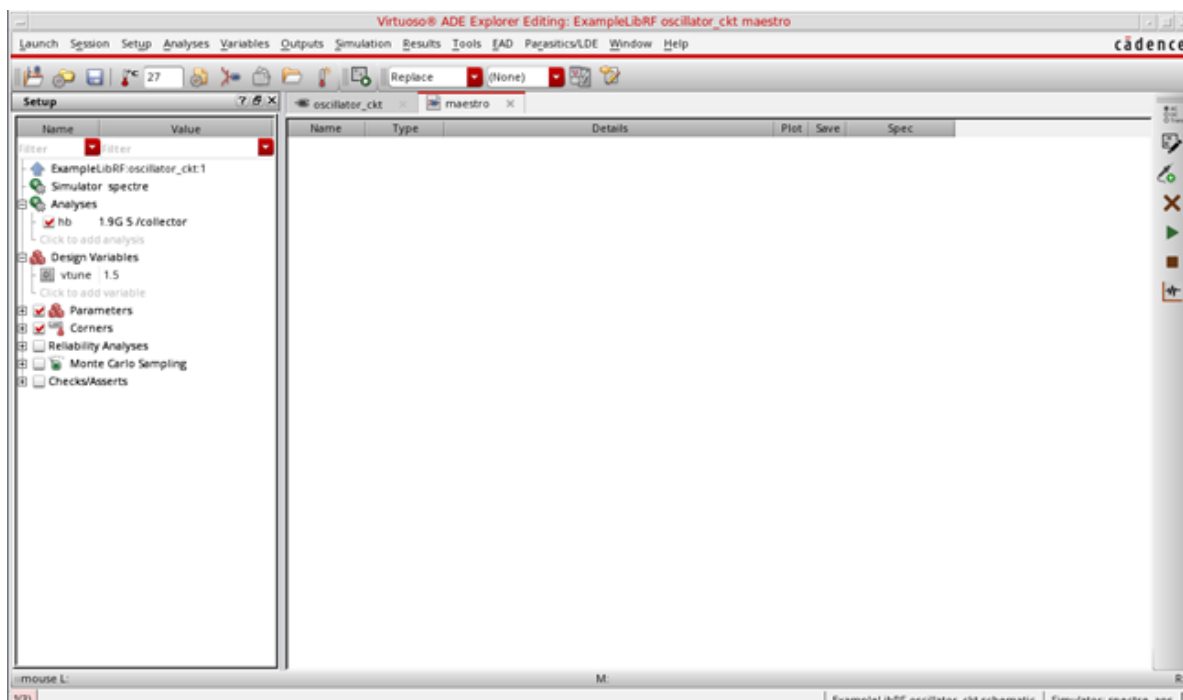
**OK** Cancel Defaults Apply Help

**34.** Click *Apply*.

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

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.

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

Figure 3-90 *hbnoise* Choosing Analyses Form

The screenshot shows the 'hbnoise' dialog box in the Spectre Circuit Simulator. The 'hbnoise' radio button is selected and highlighted with a red rectangle. The dialog is titled 'Harmonic Balance Noise Analysis'. It contains several sections: 'Multiple hbnoise' with a checkbox; 'SweepType' with a dropdown set to 'default' and a 'Relative Harmonic' input field; 'Output Frequency Sweep Range (Hz)' with a 'Start-Stop' dropdown and 'Start'/'Stop' input fields; 'Sweep Type' with a dropdown set to 'Automatic'; 'Add Specific Points' with a checkbox; 'Sidebands' with a dropdown set to 'Maximum sideband' and a text input field; 'Output' with a dropdown set to 'probe' and an 'Output Probe Instance' input field; 'Do Noise' with a checked checkbox; 'Noise Type' with a dropdown set to 'timeaverage'; 'Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis'; 'Contribution Type' with radio buttons for 'USB', 'AM', 'PM' (selected), 'AM&PM', and 'ALL(AM,PM,USB,LSB)'; 'For fmjitter, PM noise must be enabled'; 'Noise Separation' with a checkbox; 'Separate noise into source and gain'; 'Enabled' with a checkbox; and an 'Options...' button. At the bottom are 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help' buttons.

### 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.

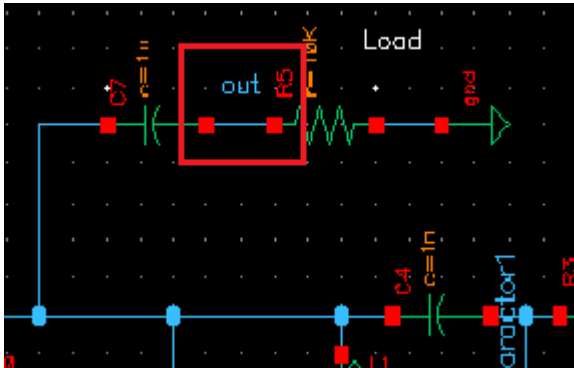
3. Leave the *Maximum Sideband* field blank. In general, *Maximum sideband* needs to be set high enough to include all the frequencies that could mix down to the oscillator output frequency. By default, when this field is left blank, all the mixing with all the *hb* harmonics are present in the *hbnoise*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

Harmonic Balance Noise Analysis

Multiple hbnoise ☐

Sweeptype: relative  Relative Harmonic: 1

Output Frequency Sweep Range (Hz)

Start-Stop  Start: 10 Stop: 100M

Sweep Type: Logarithmic  Points Per Decade: 3

Add Specific Points ☐

Sidebands: Maximum sideband

When using hb engine, default value is harms of 1st tone.

Output: voltage  Positive Output Node: /out  Negative Output Node:

Do Noise ☒

Noise Type: timeaverage

Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis

Contribution Type: ☐ USB ☐ AM ☐ PM ☐ AM&PM ☒ ALL(AM,PM,USB,LSB)

For fmjitter, PM noise must be enabled

Noise Separation: ☒ yes ☐ no

Separate noise into source and gain

Enabled ☒

Options...

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

The screenshot shows the 'Harmonic Balance Noise Options' dialog box. It is organized into four main sections:

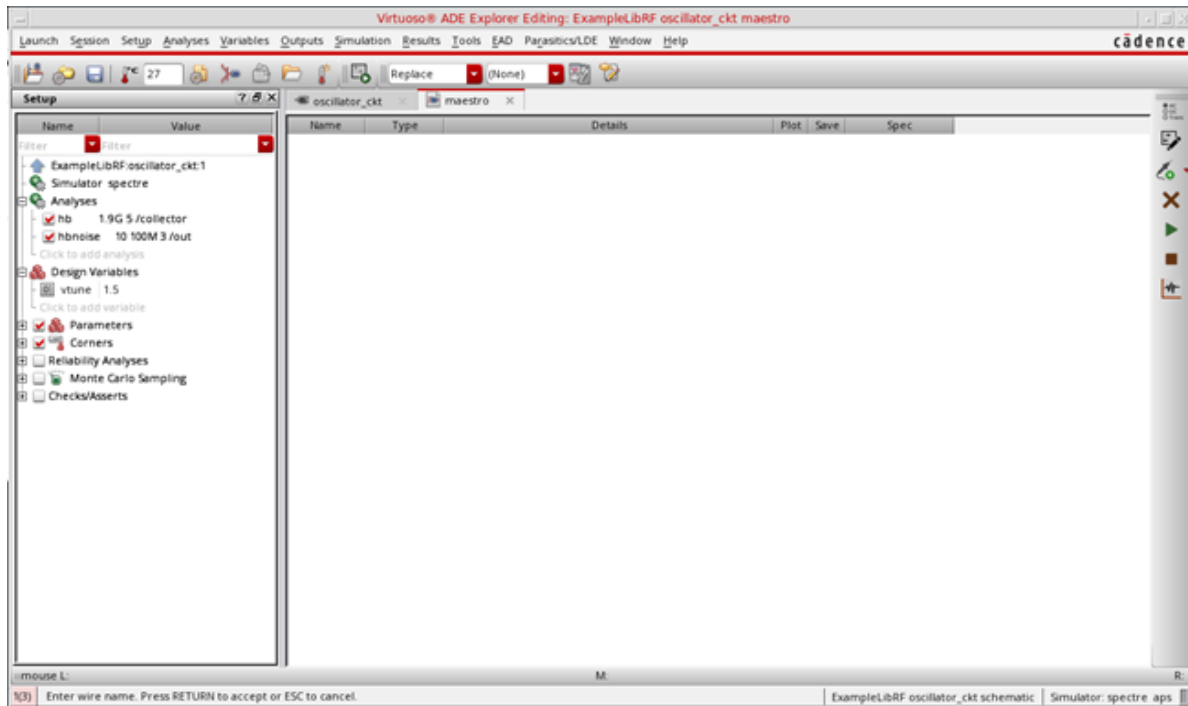
- CONVERGENCE PARAMETERS:** Includes input fields for 'relative tolerance' and 'tolerance'. The 'resgmrescycle' section has checkboxes for 'instant', 'short', 'long', 'recycleinstant', 'recycleshort', and 'recyclelong'. The 'hbprecond\_solver' section has checkboxes for 'basicsolver' and 'autoset'.
- ANNOTATION PARAMETERS:** Includes an 'annotate' section with checkboxes for 'no', 'title', 'sweep', 'status' (checked), and 'steps'.
- OUTPUT PARAMETERS:** Includes a 'stimuli' section with checkboxes for 'sources' and 'nodes\_and\_terminals'. The 'save' section has checkboxes for 'selected', 'lvlpub', 'lvl', 'allpub', and 'all'. There is a 'nestlvl' input field. The 'saveallsidebands' section has checkboxes for 'yes' and 'no'. The 'lorentzian' section has checkboxes for 'yes' (checked and highlighted with a red box), 'cornerfreqonly', and 'no'. The 'enable osc ppv' section has checkboxes for 'yes' and 'no'.
- ADDITIONAL PARAMETERS:** Includes an 'additionalParams' input field.

At the bottom of the dialog are five buttons: 'OK' (highlighted in red), 'Cancel', 'Defaults', 'Apply', and 'Help'.

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.

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

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

Direct Plot Form

Plotting Mode: Append

**Analysis**

☐ hb ☒ hbnoise  
☐ tstab ☐ hbnoise separation

**Noise Type**

☐ USB ☐ LSB ☐ AM ☒ PM

**Function**

☒ Output Noise ☐ -20dB/dec Line  
☐ Jc ☐ Jcc  
☐ Phase Noise

**Units**

☒ V/sqrt(Hz) ☐ V\*\*2/Hz ☐ dBc/Hz ☐ dBV/Hz

**Scalar Expression**

Value At (Hz):

**Noise convention**

☒ SSB ☐ DSB

Add To Outputs ☐ Plot

> Press plot button on this form...

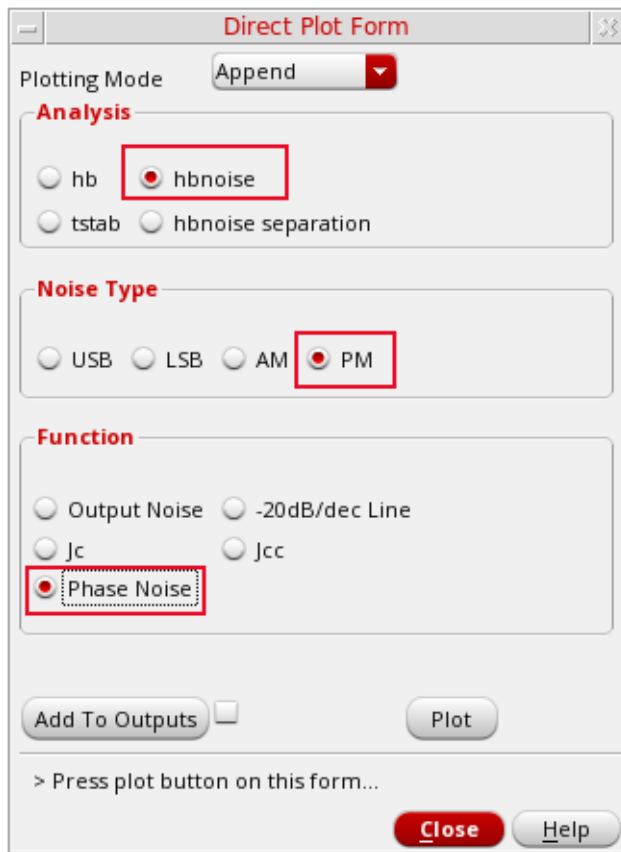
Close Help

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



The image shows a software window titled "Direct Plot Form". It contains several sections for configuring an analysis plot:

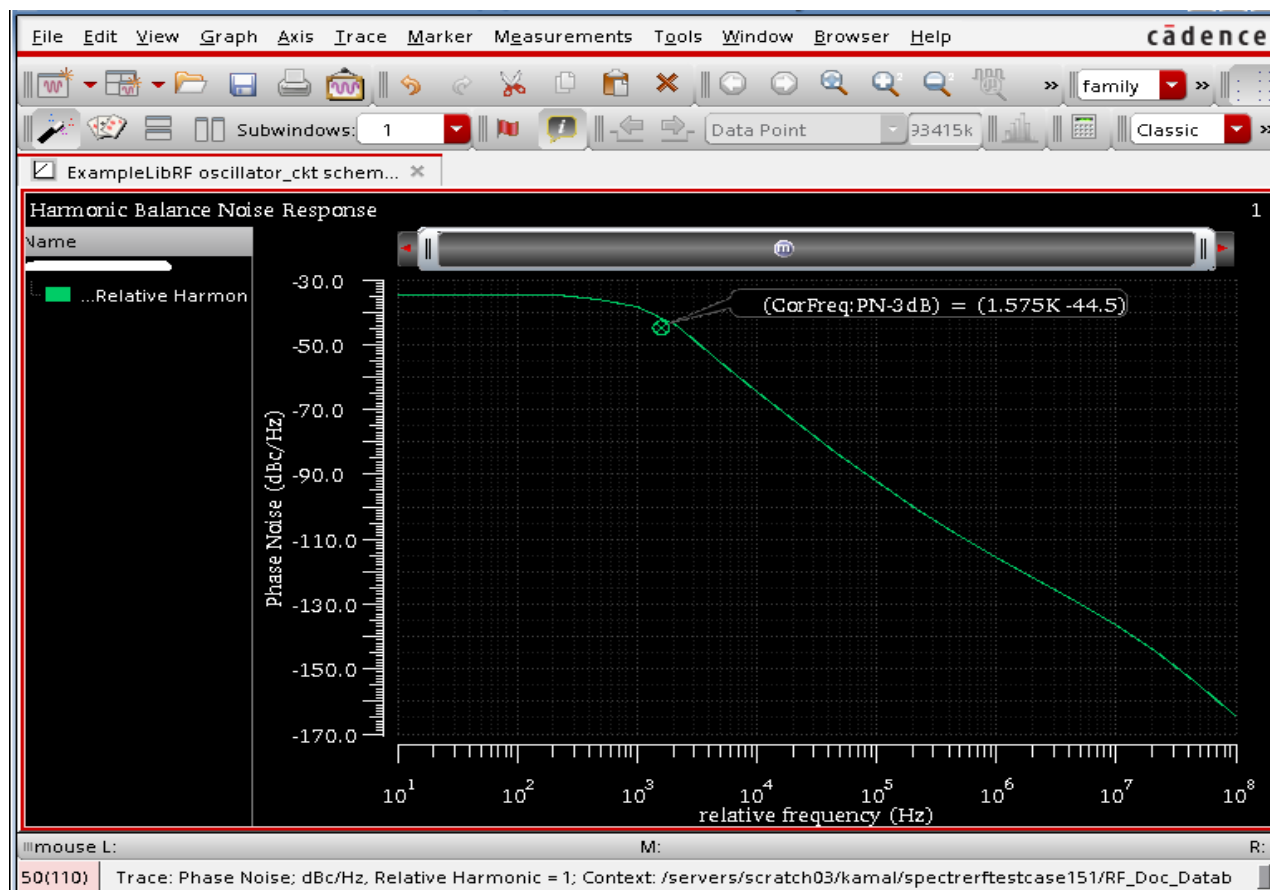
- Plotting Mode:** A dropdown menu set to "Append".
- Analysis:** A group box containing three radio buttons: "hb", "hbnoise" (which is selected and highlighted with a red box), and "tstab hbnoise separation".
- Noise Type:** A group box containing four radio buttons: "USB", "LSB", "AM", and "PM" (which is selected and highlighted with a red box).
- Function:** A group box containing four radio buttons: "Output Noise", "-20dB/dec Line", "Jc", and "Phase Noise" (which is selected and highlighted with a red box).

At the bottom of the form, there is an "Add To Outputs" checkbox (unchecked), a "Plot" button, and a "Close" button. A text prompt "> Press plot button on this form..." is located above the "Close" button.

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.



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

---

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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☐ hb ☐ hbnoise  
☐ tstab ☒ hbnoise separation

**Noise Type**

☐ USB ☐ LSB ☐ AM ☒ PM

**Function**

☒ Sideband Output ☐ Instance Output  
☐ Source Output ☐ Instance Source  
☐ Primary Source ☐ Src. Noise Gain

Noise contribution of sidebands to output

Currently, only sideband data is available

Signal Level: ☒ V / sqrt(Hz) ☐ V\*\*2 / Hz

**Modifier**

☒ Magnitude ☐ dB20

Output Sideband

-5  
-4  
-3  
-2  
-1  
0  
4

Add To Outputs: ☐

> Select Output Sideband on this form...

Close Help

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

---

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  $V / \sqrt{\text{Hz}}$ .
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 the output of the circuit.

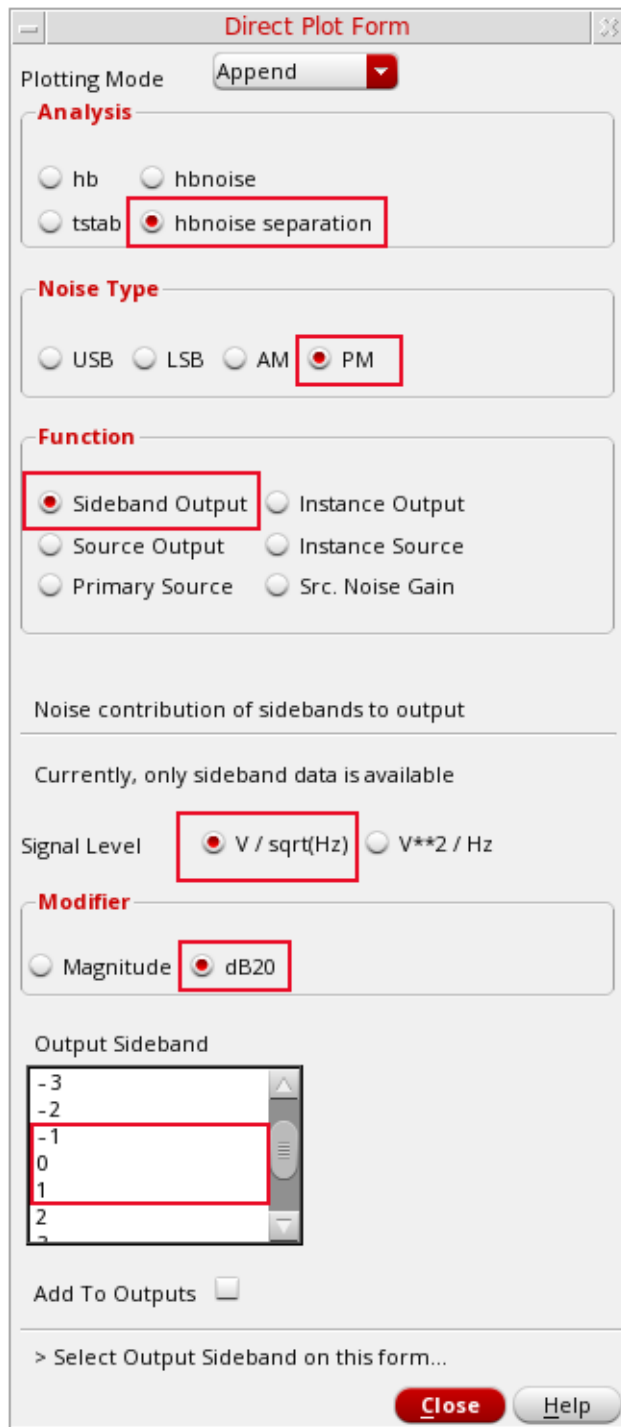


### *Tip*

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**



The image shows the 'Direct Plot Form' dialog box in a software application. The form is divided into several sections with red headers: 'Analysis', 'Noise Type', 'Function', 'Signal Level', 'Modifier', and 'Output Sideband'. In the 'Analysis' section, the 'hbnoise separation' radio button is selected and highlighted with a red box. In the 'Noise Type' section, the 'PM' radio button is selected and highlighted with a red box. In the 'Function' section, the 'Sideband Output' radio button is selected and highlighted with a red box. In the 'Signal Level' section, the 'V / sqrt(Hz)' radio button is selected and highlighted with a red box. In the 'Modifier' section, the 'dB20' radio button is selected and highlighted with a red box. In the 'Output Sideband' section, a list box shows values from -3 to 2, with '-1' selected and highlighted with a red box. At the bottom, there is an 'Add To Outputs' checkbox and a text prompt '> Select Output Sideband on this form...'. The dialog has 'Close' and 'Help' buttons at the bottom right.

Direct Plot Form

Plotting Mode: Append

**Analysis**

☐ hb ☐ hbnoise ☒ hbnoise separation

**Noise Type**

☐ USB ☐ LSB ☐ AM ☒ PM

**Function**

☒ Sideband Output ☐ Instance Output  
☐ Source Output ☐ Instance Source  
☐ Primary Source ☐ Src. Noise Gain

Noise contribution of sidebands to output

Currently, only sideband data is available

Signal Level: ☒ V / sqrt(Hz) ☐ V\*\*2 / Hz

**Modifier**

☐ Magnitude ☒ dB20

Output Sideband

-3  
-2  
-1  
0  
1  
2

Add To Outputs ☐

> Select Output Sideband on this form...

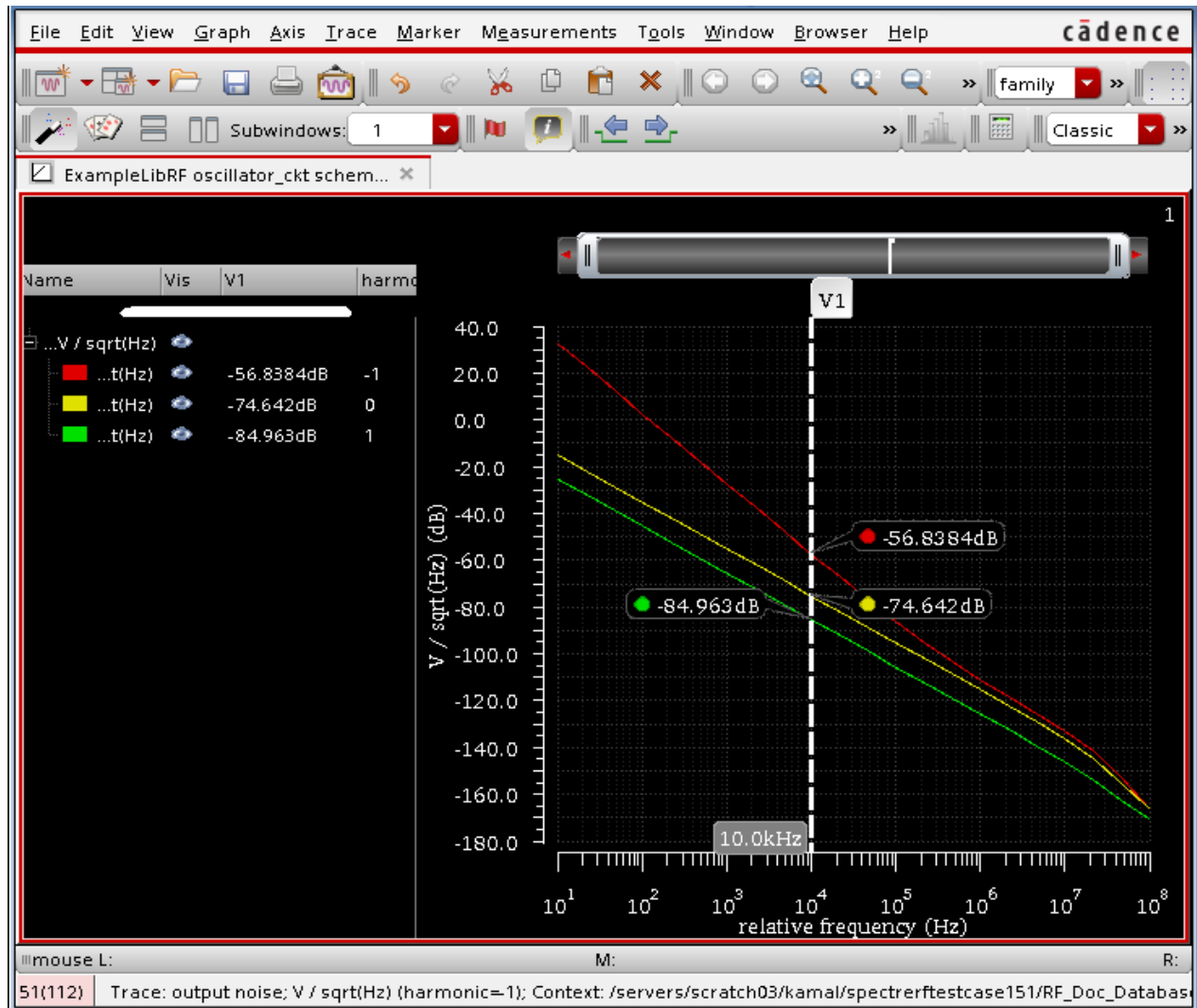
Close Help

13. Click *Plot*.

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

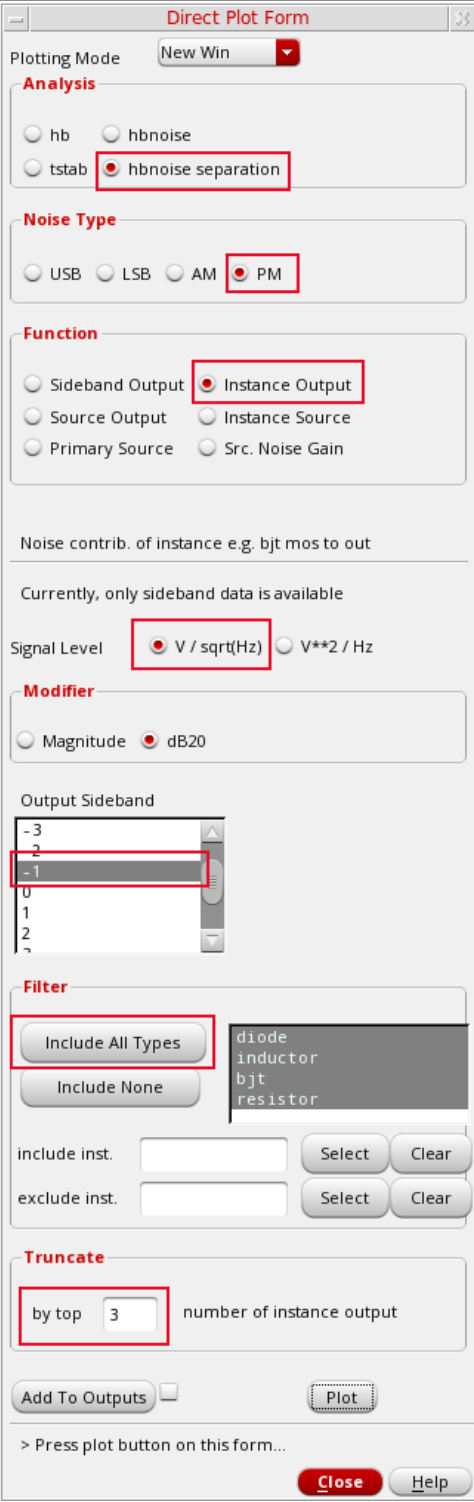
The hbnoise separation plot is shown below. Note that the harmonic numbers plotted are in the legend on the left side of the screen.

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



- 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  $V/\sqrt{\text{Hz}}$ . Set the Modifier to  $\text{dB}_{20}$ . 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**



The image shows the 'Direct Plot Form' window in a circuit simulator. It contains several sections with radio buttons and checkboxes for configuring a noise analysis plot. Red boxes highlight specific settings: 'hbnoise separation' under Analysis, 'PM' under Noise Type, 'Instance Output' under Function, 'V / sqrt(Hz)' under Signal Level, 'dB20' under Modifier, '-1' in the Output Sideband list, 'Include All Types' under Filter, and '3' in the Truncate section.

**Direct Plot Form**

Plotting Mode: New Win

**Analysis**

☐ hb ☐ hbnoise  
☐ tstab ☒ hbnoise separation

**Noise Type**

☐ USB ☐ LSB ☐ AM ☒ PM

**Function**

☐ Sideband Output ☒ Instance Output  
☐ Source Output ☐ Instance Source  
☐ Primary Source ☐ Src. Noise Gain

Noise contrib. of instance e.g. bjt mos to out

Currently, only sideband data is available

Signal Level: ☒ V / sqrt(Hz) ☐ V\*\*2 / Hz

**Modifier**

☐ Magnitude ☒ dB20

Output Sideband

-3  
-2  
-1  
0  
1  
2  
3

**Filter**

Include All Types  Include None

diode  
inductor  
bjt  
resistor

include inst.   Select  Clear  
exclude inst.   Select  Clear

**Truncate**

by top  number of instance output

Add To Outputs  Plot

> Press plot button on this form...

Close  Help

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  $V/\sqrt{\text{Hz}}$ . 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**

**Direct Plot Form**

Plotting Mode: **New Win**

**Analysis**

☐ hb ☐ hbnoise  
☐ tstab ☒ hbnoise separation

**Noise Type**

☐ USB ☐ LSB ☐ AM ☒ PM

**Function**

☐ Sideband Output ☐ Instance Output  
☒ Source Output ☐ Instance Source  
☐ Primary Source ☐ Src. Noise Gain

Noise contrib. of instance e.g. bjt mos to out

Currently, only sideband data is available

Signal Level: ☒ V / sqrt(Hz) ☐ V\*\*2 / Hz

**Modifier**

☐ Magnitude ☒ dB20

Output Sideband

-3  
-2  
**-1**  
0  
1  
2  
3

**Filter**

Include All Types  Include None

diode  
inductor  
bjt  
resistor

include inst.   Select  Clear  
exclude inst.   Select  Clear

**Truncate**

☒ by top  number of instance output

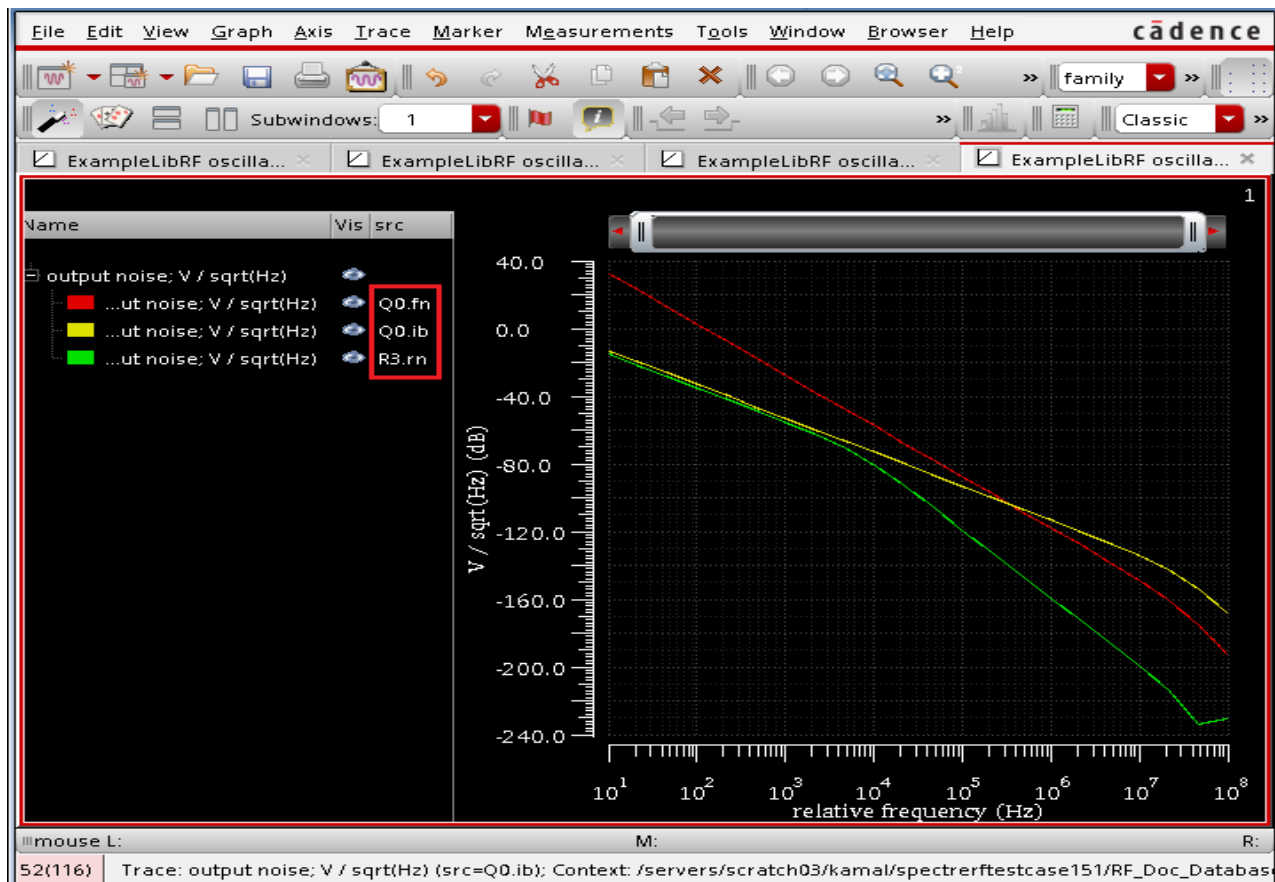
Add To Outputs  Plot

> Press plot button on this form...

Close  Help

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

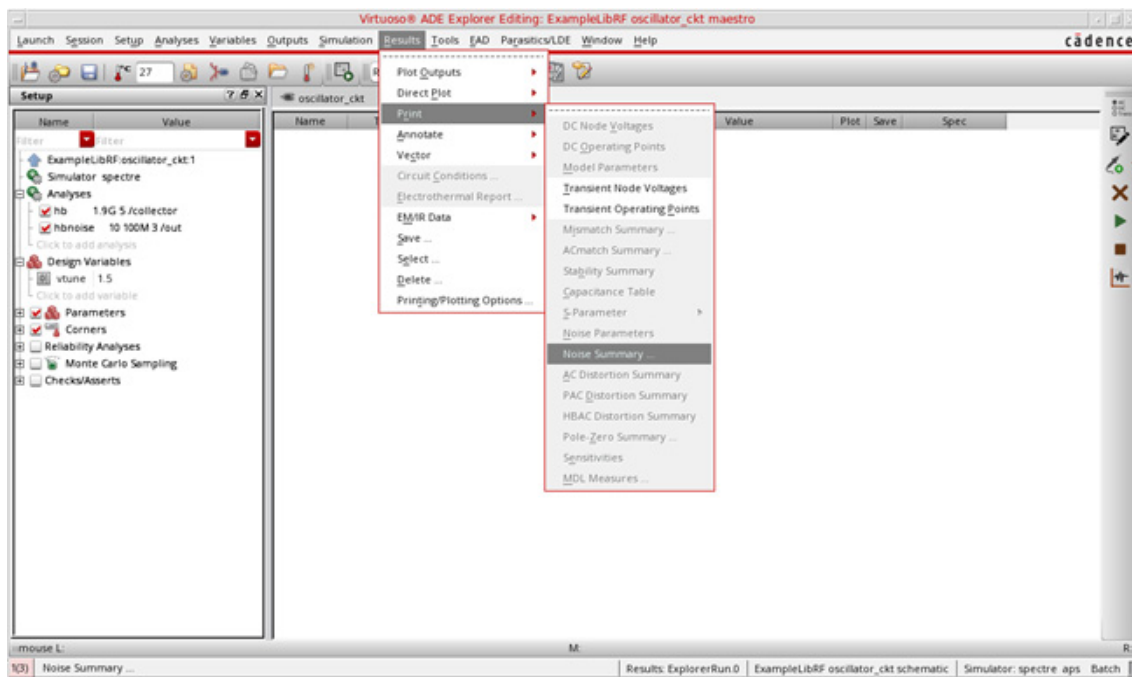
This table provides the list of noise contributors based on the selections made on the Noise Summary Table form.

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



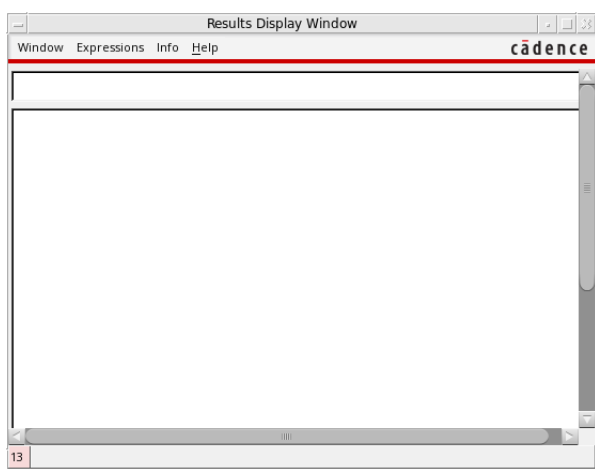
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

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**

The screenshot shows the 'Noise Summary' dialog box. The 'Data from' dropdown is set to 'hbnoise\_pm'. The 'Type' section has 'spot noise' selected. The 'Frequency Spot (Hz)' is set to '10K'. In the 'FILTER' section, the 'Include All Types' button is highlighted, and a list of components (diode, inductor, bjt, resistor) is shown. The 'include instances' and 'exclude instances' fields are empty. In the 'TRUNCATE & SORT' section, the 'truncate' dropdown is set to 'by number', and the 'top' field is set to '10'. The 'sort by' section has 'noise contributors' checked. At the bottom are 'OK', 'Cancel', 'Apply', and 'Help' buttons.

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

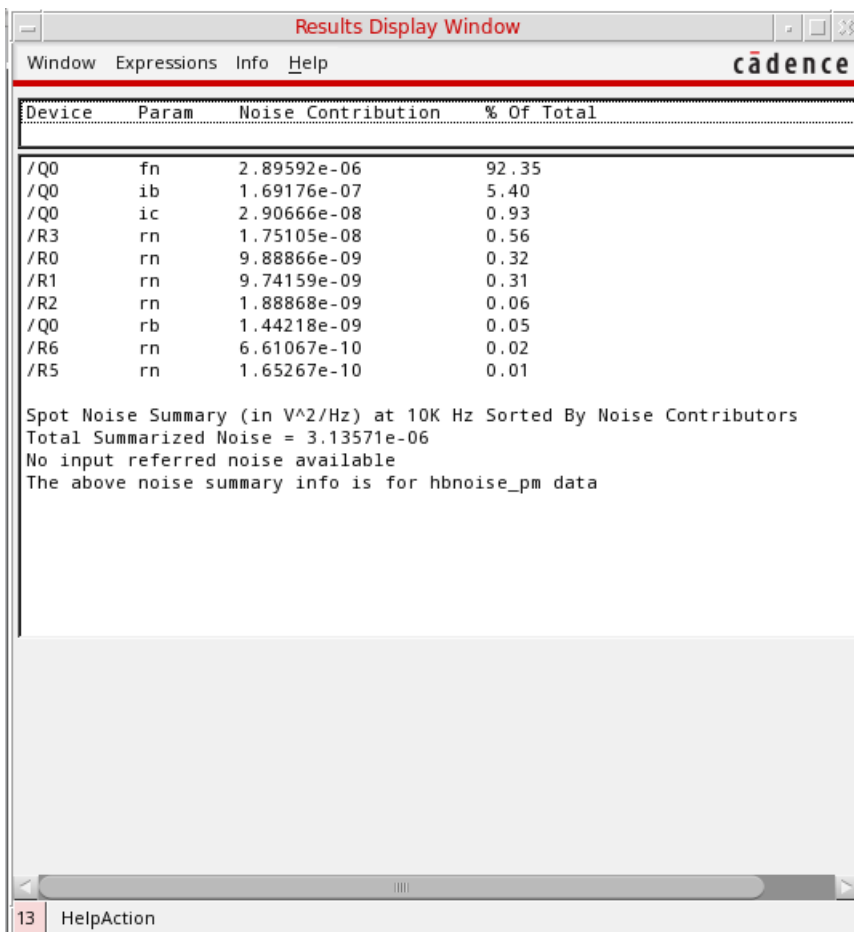
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

- ❑ *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 screenshot shows the 'Results Display Window' from Cadence. It contains a table of noise contributors and a summary section below it.

| Device | Param | Noise Contribution | % Of Total |
|--------|-------|--------------------|------------|
| /Q0    | fn    | 2.89592e-06        | 92.35      |
| /Q0    | ib    | 1.69176e-07        | 5.40       |
| /Q0    | ic    | 2.90666e-08        | 0.93       |
| /R3    | rn    | 1.75105e-08        | 0.56       |
| /R0    | rn    | 9.88866e-09        | 0.32       |
| /R1    | rn    | 9.74159e-09        | 0.31       |
| /R2    | rn    | 1.88868e-09        | 0.06       |
| /Q0    | rb    | 1.44218e-09        | 0.05       |
| /R6    | rn    | 6.61067e-10        | 0.02       |
| /R5    | rn    | 1.65267e-10        | 0.01       |

Spot Noise Summary (in V<sup>2</sup>/Hz) at 10K Hz Sorted By Noise Contributors  
Total Summarized Noise = 3.13571e-06  
No input referred noise available  
The above noise summary info is for hbnoise\_pm 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 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.

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

---

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.

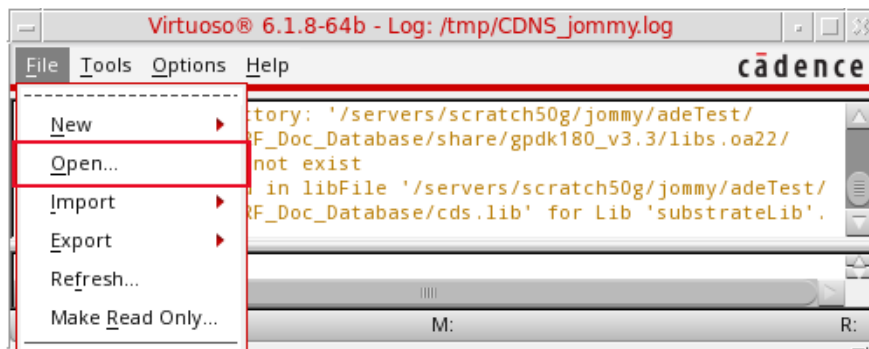
## Oscillator Swept Tuning Range and Phase Noise Measurement

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

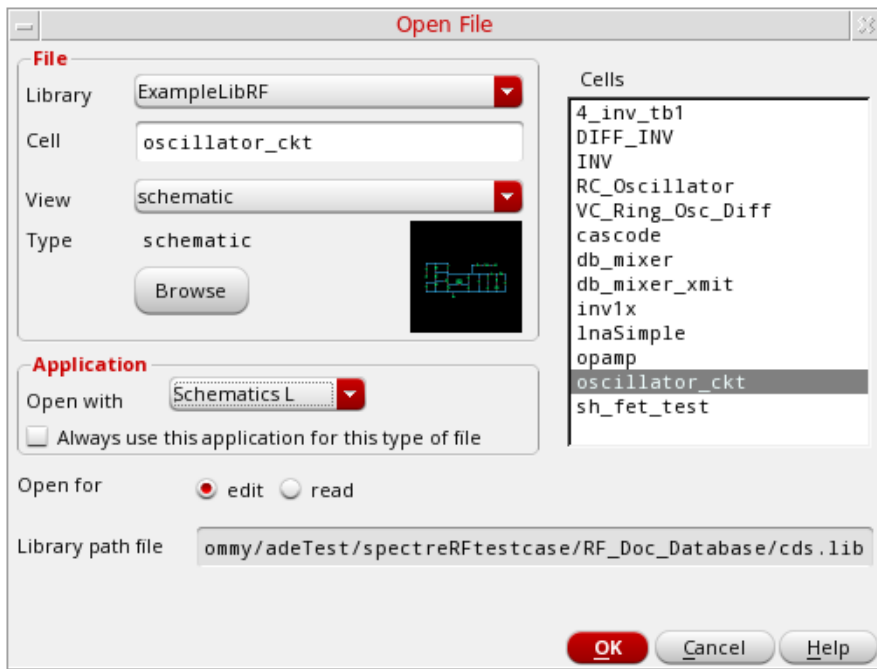
1. In CIW, select *File - Open*.

**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.

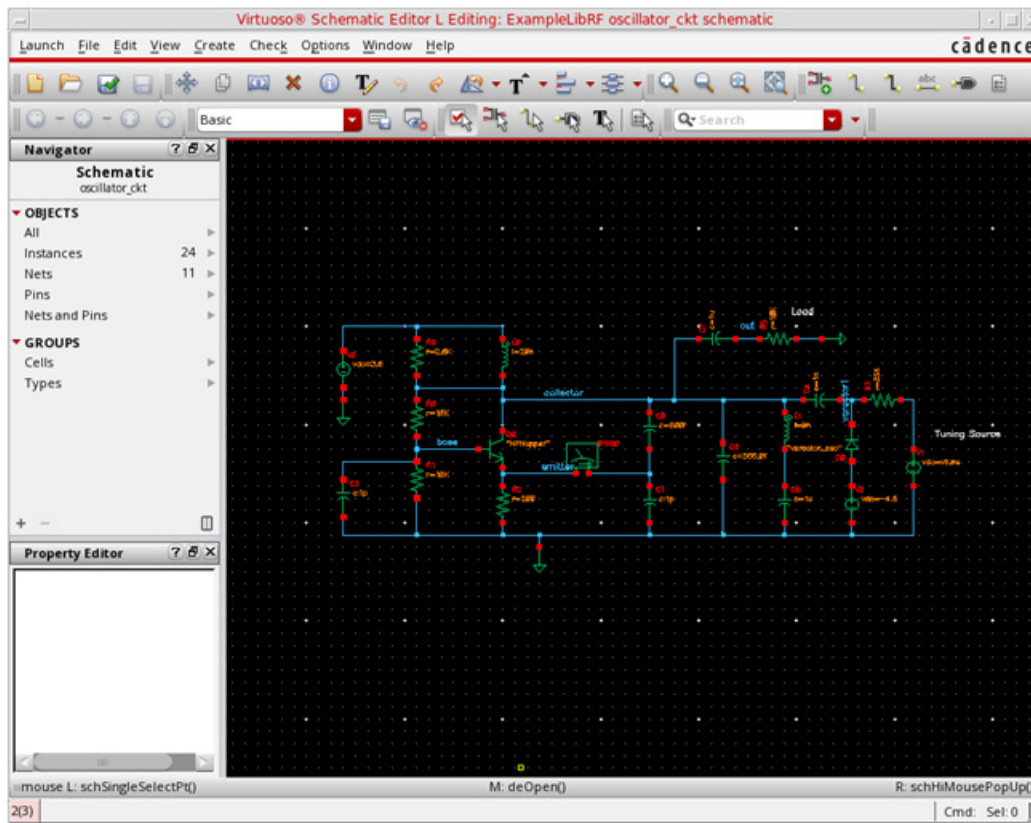


2. Select *ExampleLibRF* from the *Library* drop-down list.
3. In the *Cells* field, type `oscillator_ckt`.
4. Choose *schematic* from the *View* drop-down list.
5. In the *Application* section, select *Schematic L* from the *Open With* drop-down list.
6. Leave *Open For* to *Edit* (which is set by default)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

This will open the oscillator\_ckt schematic in Virtuoso Schematic Editor L window.

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

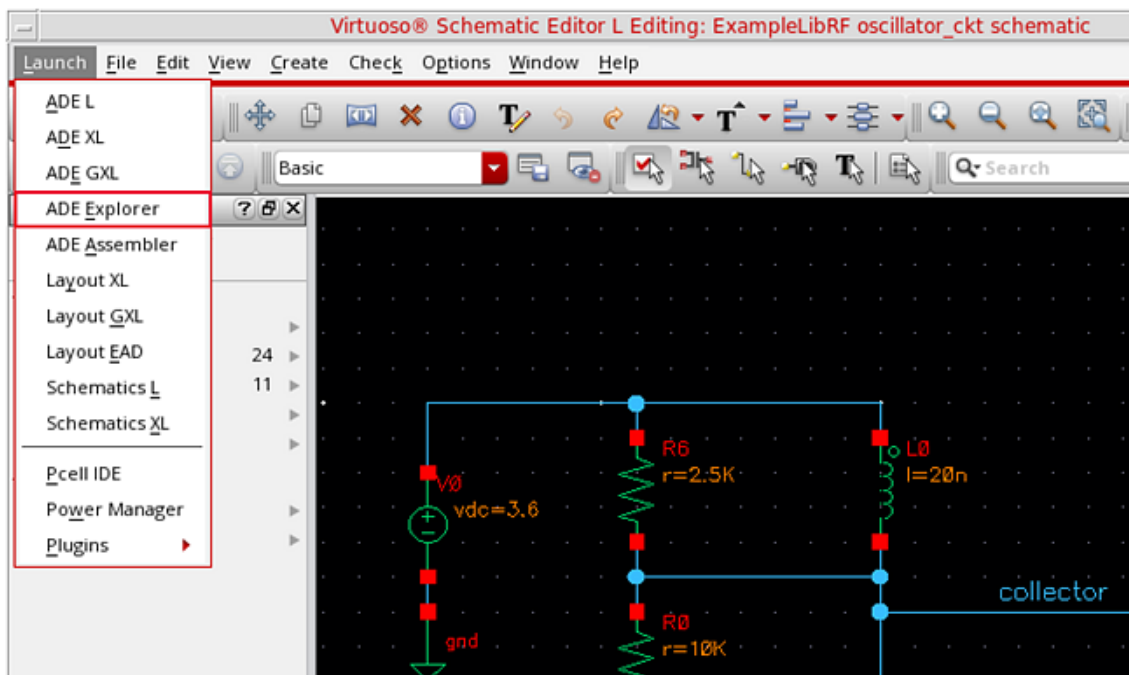
Figure 3-111 Oscillator Schematic in VSE-L Window



### Setting up the HB and HBnoise Analysis

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

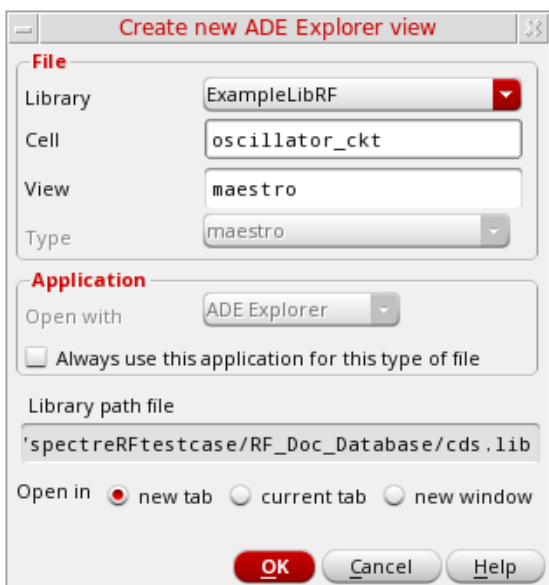
Figure 3-112 Opening ADEL window from VSE window



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*.

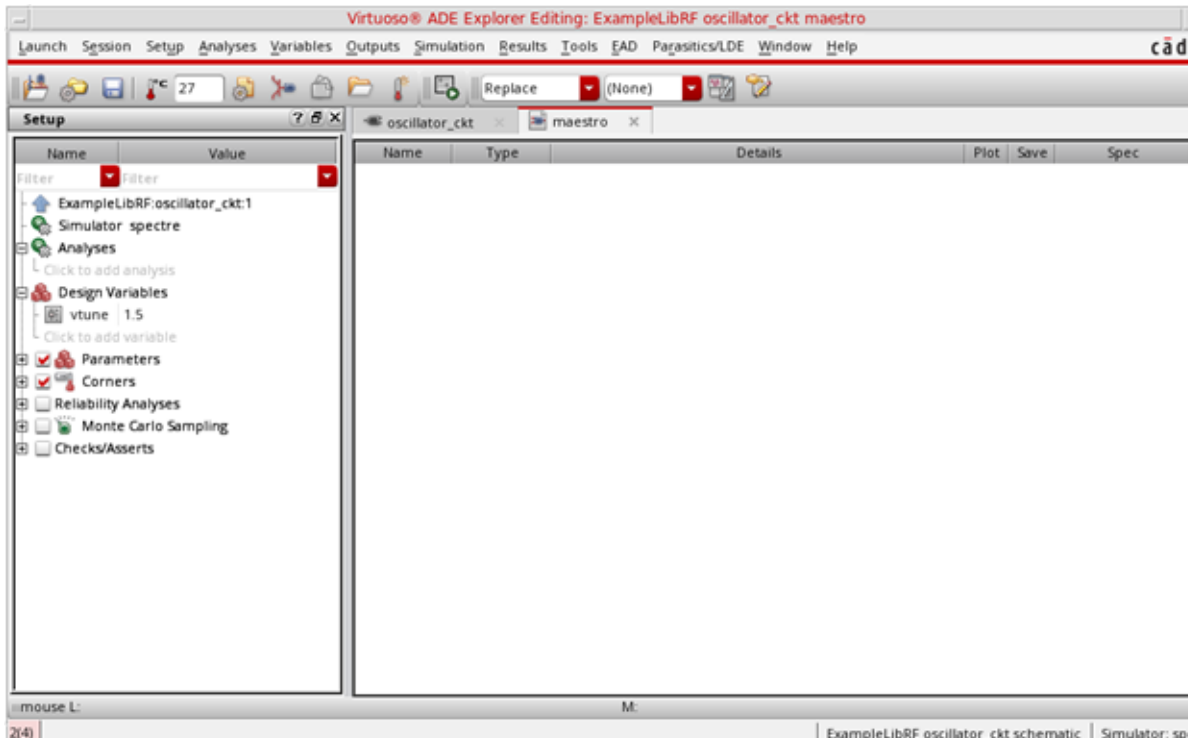


## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

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**



The image shows the 'High-Performance Simulation Options' dialog box. At the top, the title bar reads 'High-Performance Simulation Options'. Below it, the 'Simulation Performance Mode' section has four radio buttons: 'Spectre', 'APS' (which is selected and highlighted with a dashed box), 'Spectre X', and 'XPS MS'. The 'General' section contains 'Accuracy + Speed' settings with an 'Error Preset' of 'Do not override' (selected) and other options 'Liberal', 'Moderate', and 'Conservative'. There is a 'Use ++aps:' checkbox which is unchecked. The 'Multi-Threading' section has 'Auto' selected, with 'Disable' and 'Manual' as other options. A '# Threads:' text box is empty. Below it is a 'Processor affinity (e.g. 0-3 or 0,2,4,6):' text box which is also empty. The 'Post-Layout Settings' section includes an 'Enable Post-Layout Optimization:' checkbox (unchecked). The 'Post-Layout Preset Mode:' has four radio buttons: 'Default' (selected), 'High Precision Analog', 'Ultra Precision Analog', 'Legacy RCR', and 'Legacy-RF RCR'. The 'Instance Preservation:' has 'None' selected, with 'Selected' as the other option. Below this is a 'Selected Instances:' text box with 'Select' and 'Clear' buttons. At the bottom of this section is a 'Savefilter' with 'none' and 'rc' checkboxes. The bottom of the dialog has five buttons: 'OK' (highlighted in red), 'Cancel', 'Defaults', 'Apply', and 'Help'.

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*.

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

The *Save Options* form is displayed.

**Figure 3-117 Save Options Form**

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ net2 ☐ ref ☐ ref with Rstz ☒ ref1 ☐ ref2

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.

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

---

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*.

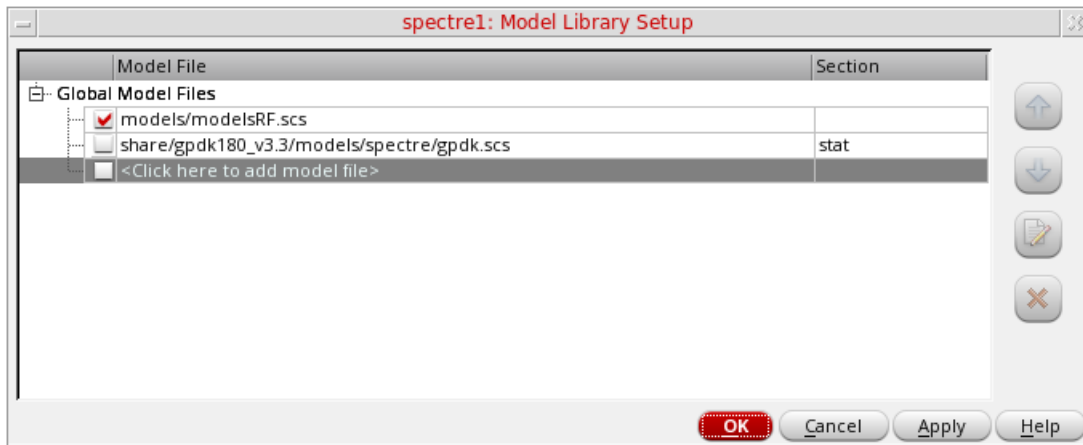
The *Simulation Files Setup* form is displayed, as shown below.

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

| Files/Paths                                         | Options |
|-----------------------------------------------------|---------|
| <input checked="" type="checkbox"/> Include Paths   |         |
| <input type="checkbox"/> <Click here to add a path> |         |
| <input type="checkbox"/> Definition Files           |         |
| <input type="checkbox"/> <Click here to add a file> |         |
| <input type="checkbox"/> Stimulus Files             |         |
| <input type="checkbox"/> <Click here to add a file> |         |
| <input type="checkbox"/> Parasitic Files (DSPF)     |         |
| <input type="checkbox"/> <Click here to add a file> |         |
| <input type="checkbox"/> Parasitic Files (SPEF)     |         |
| <input type="checkbox"/> <Click here to add a file> |         |
| <input type="checkbox"/> PSpice Files               |         |
| <input type="checkbox"/> <Click here to add a path> |         |

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.

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

---

**Figure 3-120 The Choosing Analyses Form**

Choosing Analyses -- ADE Explorer

Analysis

☒ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpsp

☐ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative ☐ moderate ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

OK Cancel Defaults Apply Help

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

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

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

The dialog box is titled "Choosing Analyses" and contains the following sections:

- Analysis:** A grid of radio buttons for selecting an analysis type. The selected option is **hb** (Harmonic Balance).
- Harmonic Balance Analysis:**
  - Transient-Aided Options:**
    - Run transient?: **Decide automatically** (dropdown menu)
    - Detect Steady State: ☒ (checkbox)
    - Stop Time(tstab): **auto** (text field)
    - Save Initial Transient Results (saveinit): ☐ no ☐ yes
  - Dynamic Parameter:** ☐ (checkbox)
  - Tones:** ☒ Frequencies ☐ Names
  - Number of Tones:** ☒ 1 ☐ 2 ☐ 3 ☐ 4
    - Tone 1:**
      - Fundamental Frequency:
      - Number of Harmonics: **auto** (text field)
      - Oversample Factor: **1** (text field)
      - Freqdivide Ratio for Tone 1: **1** (text field)
  - Harmonics:** **Default** (dropdown menu)
  - Accuracy Defaults (errpreset):** ☐ conservative ☒ moderate ☐ liberal
  - Oscillator:** ☐ (checkbox)
  - Sweep:** ☐ (checkbox)
  - Loadpull:** ☐ (checkbox)
  - LSSP:** ☐ (checkbox)
  - Compression:** ☐ (checkbox)
  - Enabled:** ☐ (checkbox)
  - Options...** (button)
- Buttons:** **OK** (red button), **Cancel**, **Defaults**, **Apply**, **Help**

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

---

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**

Oscillator ☒ Oscillator node+  Select

Oscillator node-  Select

☒ Calculate initial conditions (ic) automatically

☐ Enable tuning mode analysis

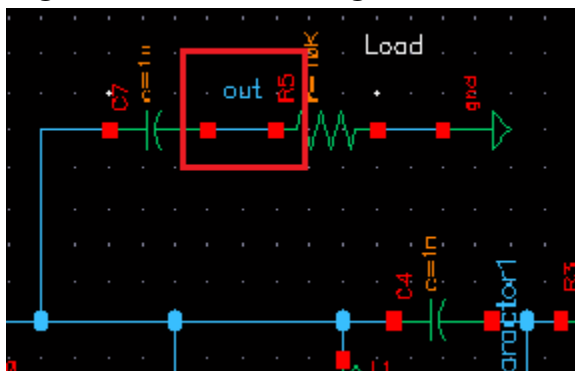
☐ Use the probe-based solution method

Selecting the Oscillator button changes the Tone1 name in the Tones section to osc1.

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.

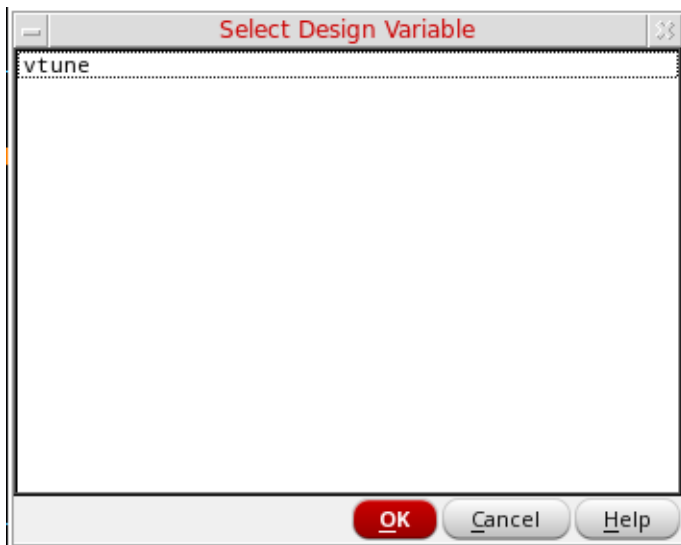
34. Note that by default *Use the probe-based solution method* (oscmethod) option is deselected. Spectre will use the *onetier* method. In *onetier* method, the frequency and voltage spectrum are solved simultaneously in one single set of nonlinear equations.

**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.

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

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

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

**Choosing Analyses -- ADE Explorer**

Transient-Aided Options

Run transient? ☒ Decide automatically ☐ ☐ No ☐ Yes

Detect Steady State ☒ Stop Time(tstab)

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

Fundamental Frequency

Number of Harmonics

Oversample Factor

Freqdivide Ratio for Tone 1

Harmonics

Accuracy Defaults (errpreset) ☒ conservative ☐ moderate ☐ liberal

Oscillator ☒ Oscillator node+

Oscillator node-

☒ Calculate initial conditions (ic) automatically

☐ Enable tuning mode analysis

☐ Use the probe-based solution method

Sweep ☒ Variable  Frequency Variable? ☒ no ☐ yes

Variable Name

Sweep Range

☒ Start-Stop ☐ Center-Span

Start  Stop

Sweep Type

☒ Linear ☐ Logarithmic

☒ Step Size ☐ Number of Steps

Step Size

Add Specific Points ☐

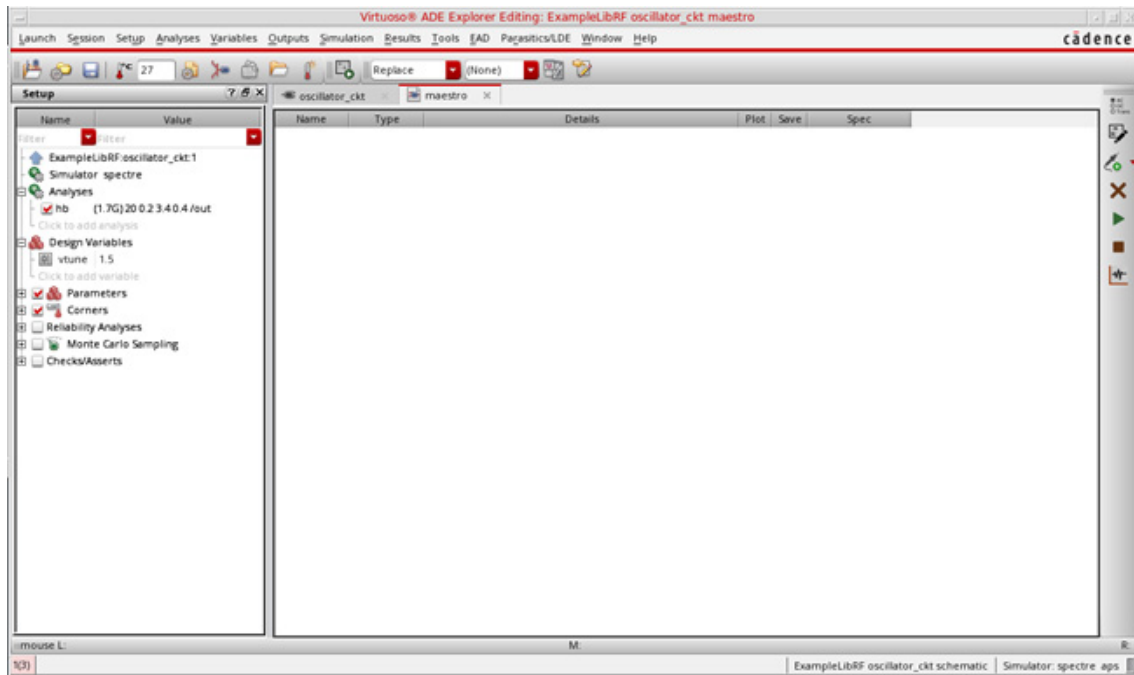
Loadpull ☐

LSSP ☐

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

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.

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

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

The screenshot shows the 'Choosing Analyses -- ADE Explorer' dialog box. The 'Analysis' section at the top has a grid of radio buttons for various analysis types. The 'hbnoise' option is selected and highlighted with a red rectangle. Below this, the 'Harmonic Balance Noise Analysis' section contains several sub-sections: 'Multiple hbnoise' (checkbox), 'SweepType' (dropdown set to 'default') and 'Relative Harmonic' (text field), 'Output Frequency Sweep Range (Hz)' with 'Start-Stop' (dropdown), 'Start' (text field), and 'Stop' (text field), 'Sweep Type' (dropdown set to 'Automatic'), 'Add Specific Points' (checkbox), and 'Add Points By File' (checkbox). The 'Sidebands' section has a 'Maximum sideband' (dropdown) and a note: 'When using hb engine, default value is harms of 1st tone.' The 'Output' section has a 'probe' (dropdown), 'Output Probe Instance' (text field), and a 'Select' button. The 'Noise Type' section has a 'timeaverage' (dropdown), a description: 'Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis', 'Contribution Type:' (text), and radio buttons for 'USB', 'AM', 'PM' (selected), 'AM&PM', and 'ALL(AM,PM,USB,LSB)'. Below these are checkboxes for 'Noise Separation' and 'Separate noise into source and gain'. At the bottom, there is a 'Generate Osc Macro Source' (checkbox), an 'Enabled' checkbox, an 'Options...' button, and a row of buttons: 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'.

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.

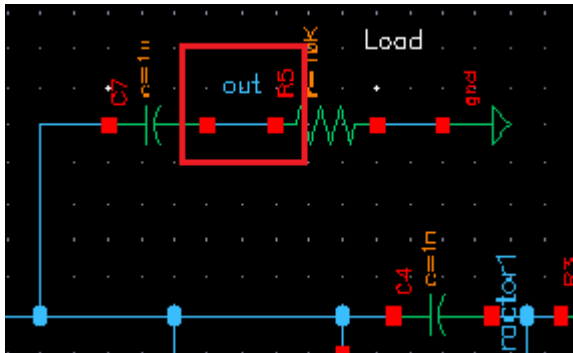
3. Leave the *Maximum Sideband* field blank.

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**

Harmonic Balance Noise Analysis

Multiple hbnoise ☐

Sweeptype: relative  Relative Harmonic: 1

Output Frequency Sweep Range (Hz)

Single-Point  Freq: 1M

Add Specific Points ☐

Sidebands

Maximum sideband

When using hb engine, default value is harms of 1st tone.

Output

voltage  Positive Output Node: /out

Negative Output Node:

Do Noise ☒

Noise Type: timeaverage

Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis

Contribution Type:

USB ☐ AM ☐ PM ☐ AM&PM ☒ ALL(AM,PM,USB,LSB)

For fmjitter, PM noise must be enabled

Noise Separation ☐

Separate noise into source and gain

Enabled ☒

Options...

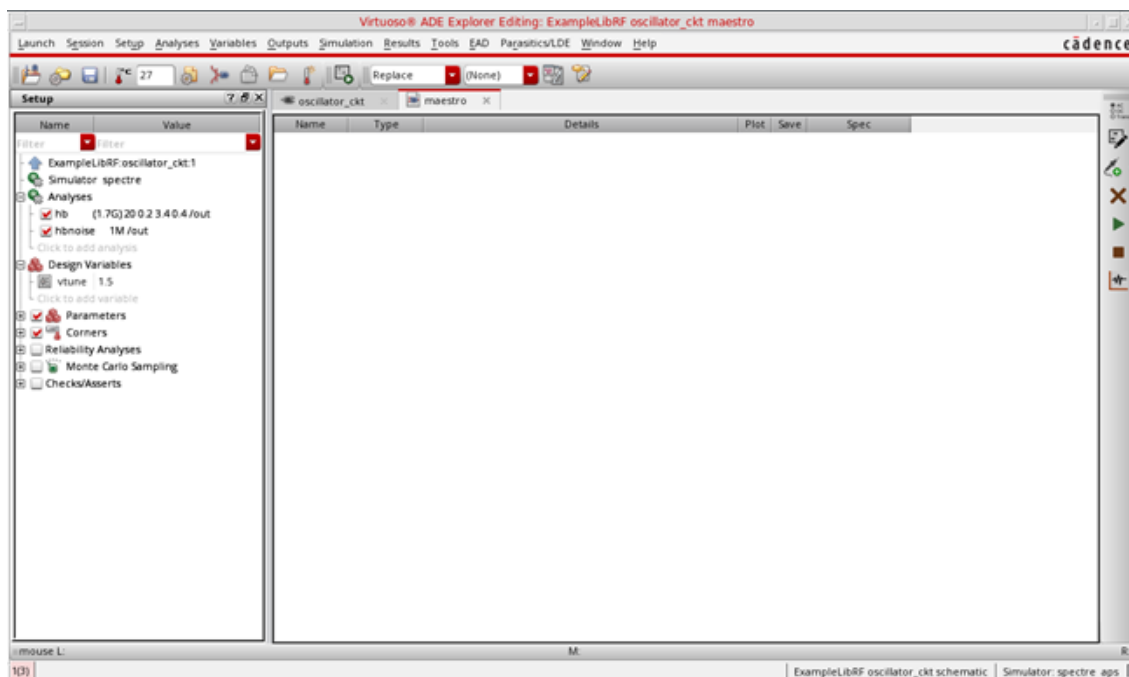
6. Click **OK** to close the *Choosing Analyses* form.



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

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**

**Direct Plot Form**

Plotting Mode: **Append**

**Analysis**

☒ hb ☐ hbnoise

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Harmonic Frequency ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp. ☐ THD

Select: **Net**

**Sweep**

☒ spectrum ☐ variable ☐ freq  
☐ time

Signal Level: ☒ peak ☐ rms

**Modifier**

☒ Magnitude ☐ Phase ☐ dB20  
☐ Real ☐ Imaginary

**Scalar Expression**

Value At (harms):

Variable Value (vtune)

|      |
|------|
| 200m |
| 600m |
| 1    |
| 1.4  |
| 1.8  |
| 2.2  |
| ...  |

Add To Outputs ☐

> Select Variable Value (vtune) on this form...

**Close** **Help**

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.

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

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**

The screenshot shows the 'Direct Plot Form' dialog box. The 'Plotting Mode' is set to 'New Win'. Under the 'Analysis' section, the 'hb' radio button is selected. Under the 'Function' section, the 'Harmonic Frequency' radio button is selected. The 'Harmonic Frequency' table shows the following data:

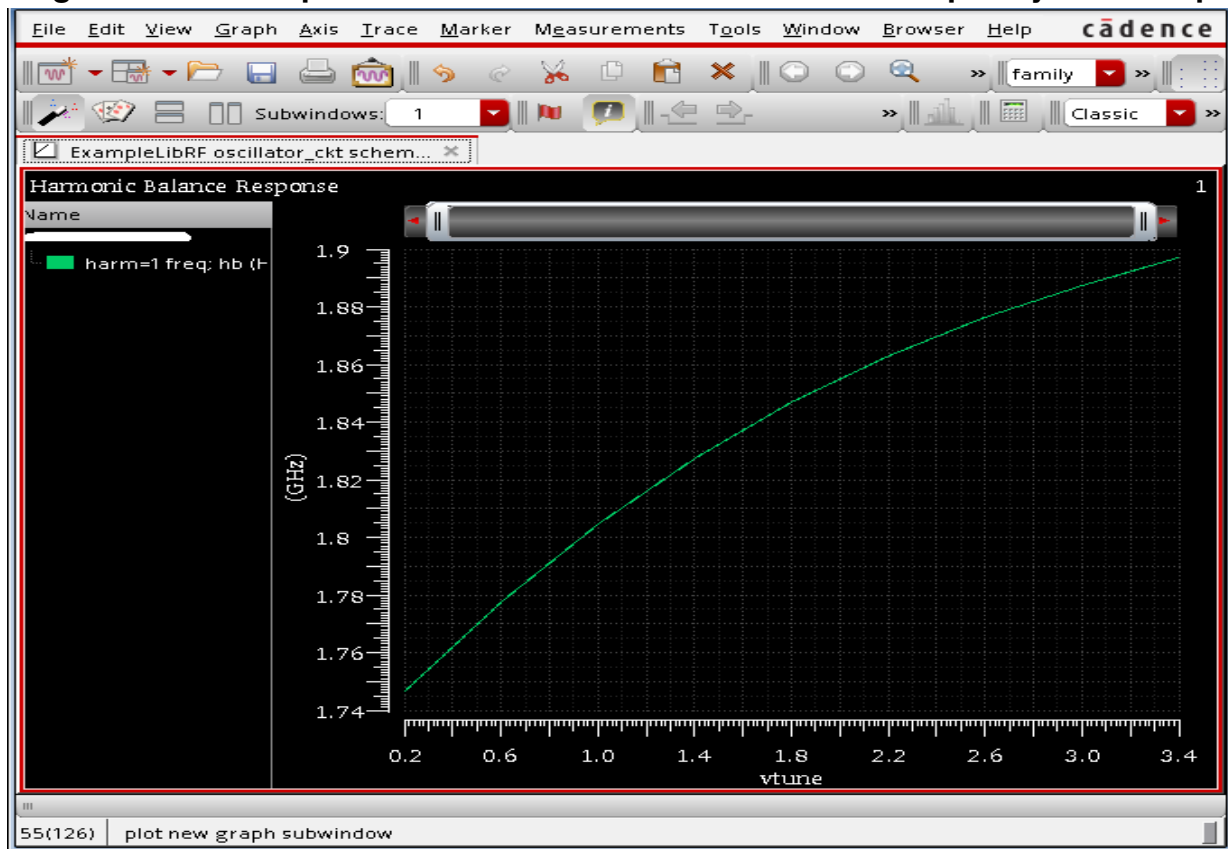
| 0 | 0            |
|---|--------------|
| 1 | 1.747036268G |
| 2 | 3.494072536G |
| 3 | 5.241108804G |
| 4 | 6.988145072G |

The first row of the table is highlighted. Below the table, the 'Loadpull Contours' checkbox is unchecked. The 'Add To Outputs' checkbox is also unchecked. The 'Plot' button is visible. At the bottom, there is a red 'Close' button and a 'Help' button.

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.

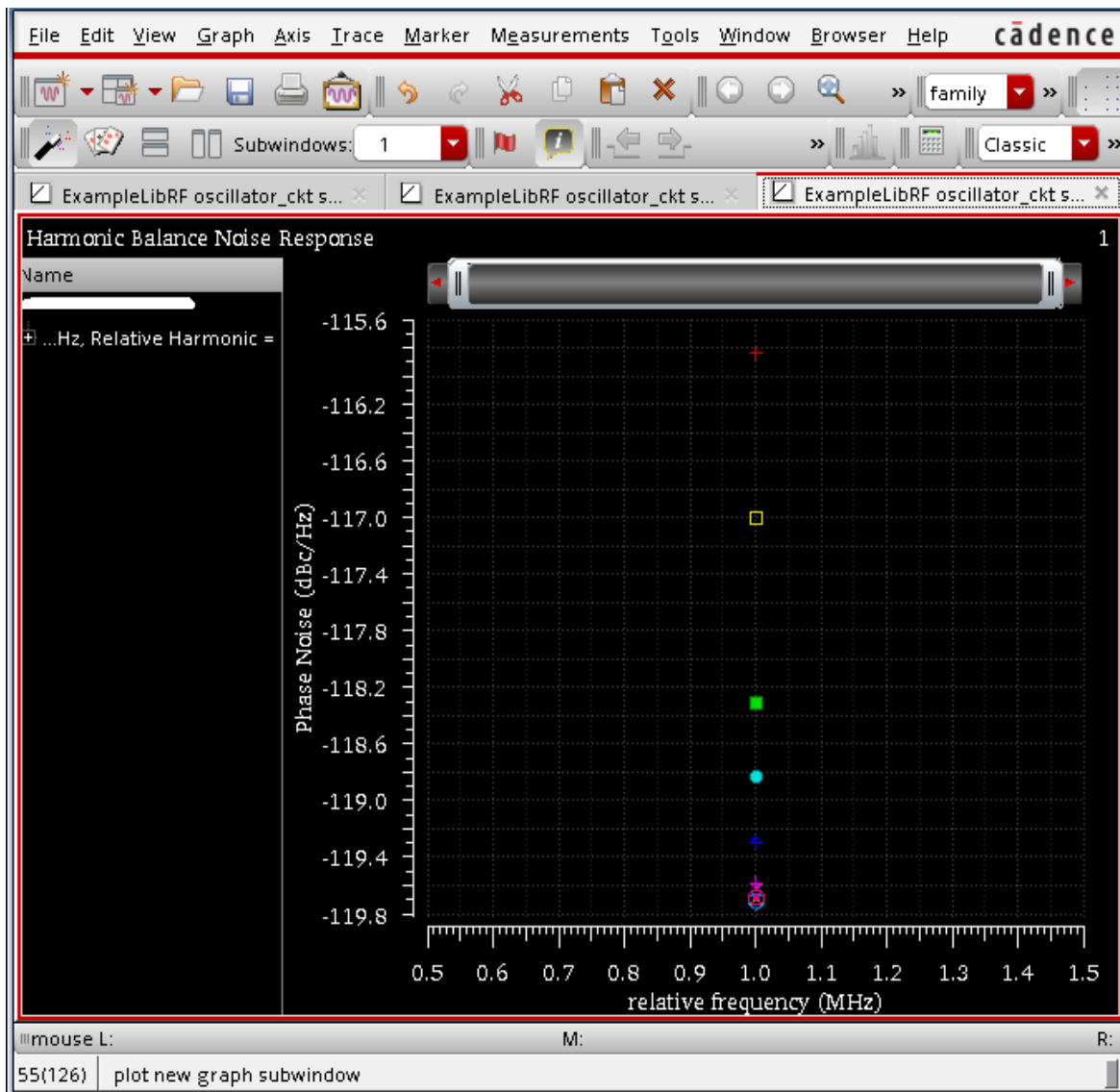
The *Direct Plot Form* should look like the following:

**Figure 3-134** *hbnoise* Direct Plot Setup

The image shows the 'Direct Plot Form' dialog box in a software application. The dialog has a title bar with the text 'Direct Plot Form'. Inside, there are several sections with red headers: 'Plotting Mode' with a dropdown menu set to 'New Win'; 'Analysis' with radio buttons for 'hb' and 'hbnoise' (the latter is selected and highlighted with a red box); 'Noise Type' with radio buttons for 'USB', 'LSB', 'AM', and 'PM' (the latter is selected and highlighted with a red box); and 'Function' with radio buttons for 'Output Noise', '-20dB/dec Line', 'Jc', 'Jcc', and 'Phase Noise' (the latter is selected and highlighted with a red box). Below these sections, there is a text label 'Currently, only variable data is available'. At the bottom, there are two buttons: 'Add To Outputs' with a small square icon to its right, and 'Plot'. Below these buttons is a text prompt '> Press plot button on this form...'. At the very bottom are two buttons: 'Close' (highlighted in red) and 'Help'.

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

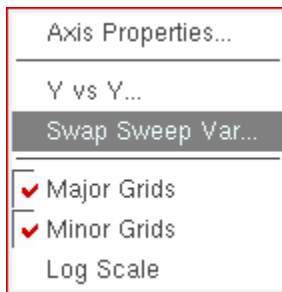
Figure 3-135 Phase Noise vs. Relative Frequency Plot



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.

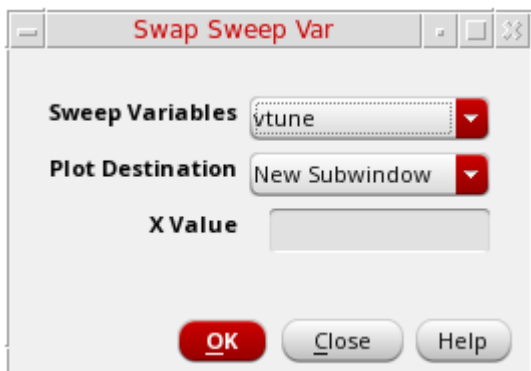
6. In the plot window, right-click one of the numbers on the X Axis, and select *Swap Sweep Var* from the context menu.

**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*.

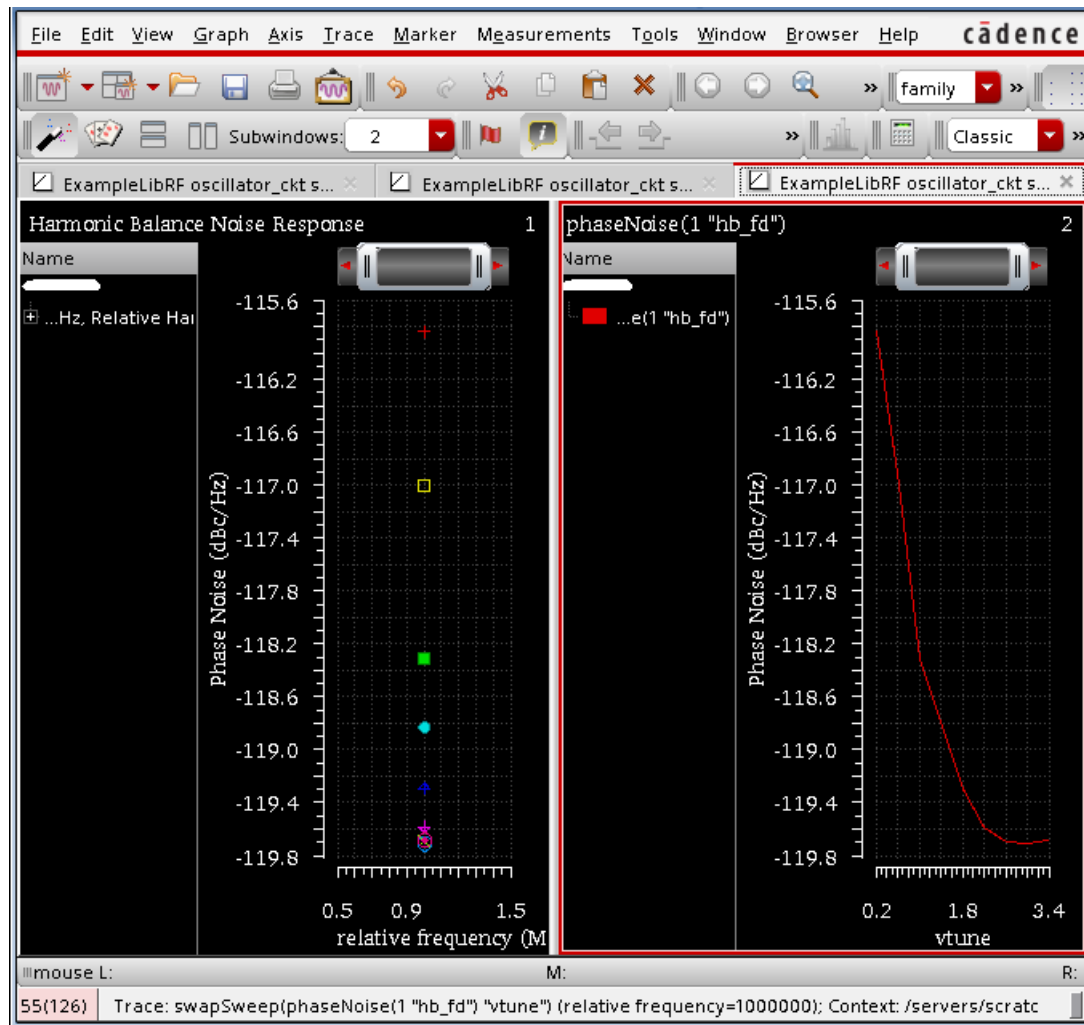
**Figure 3-137 Swap Sweep Var Dialog Box Window**



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*.



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

---

### **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.

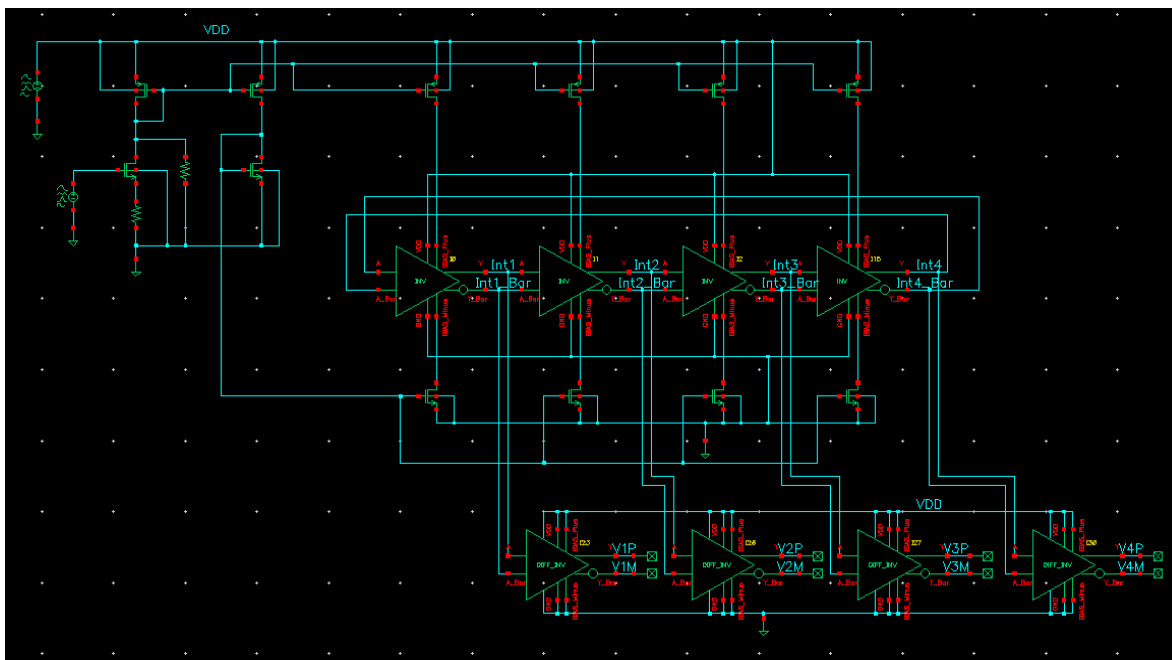
See *Oscillators and Autonomous PSS Analysis* in the [Spectre Circuit Simulator RF Analysis Theory](#) for more information on oscillator simulation.

### 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**

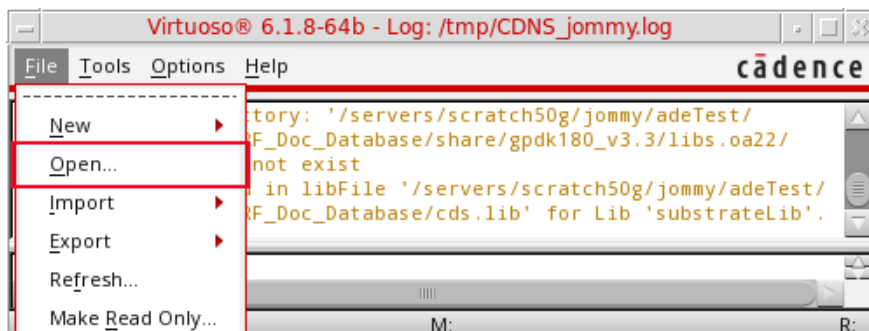


## Simulating the Oscillator Circuit

### Opening the Oscillator Circuit in the Schematic Window

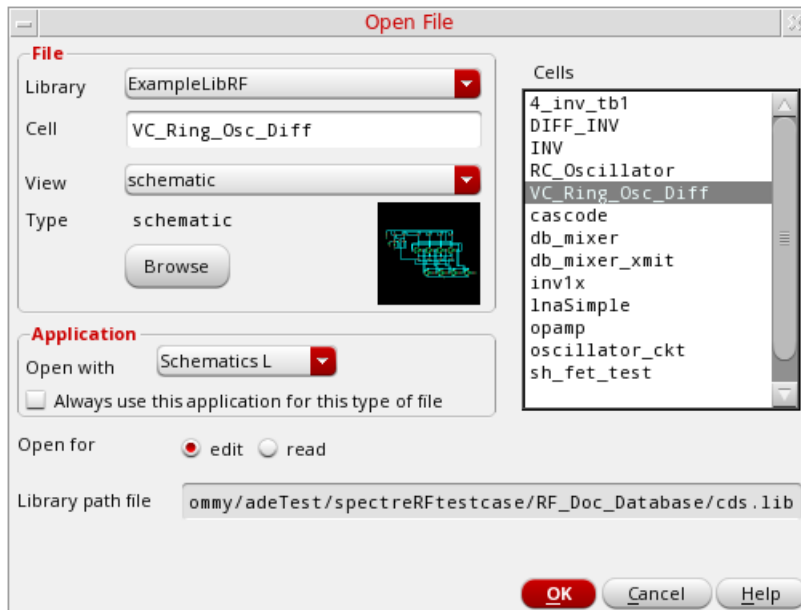
1. In the Command Interpreter Window (CIW), choose *File* → *Open*.

**Figure 3-140 Virtuoso CIW Window - Opening Cellview**



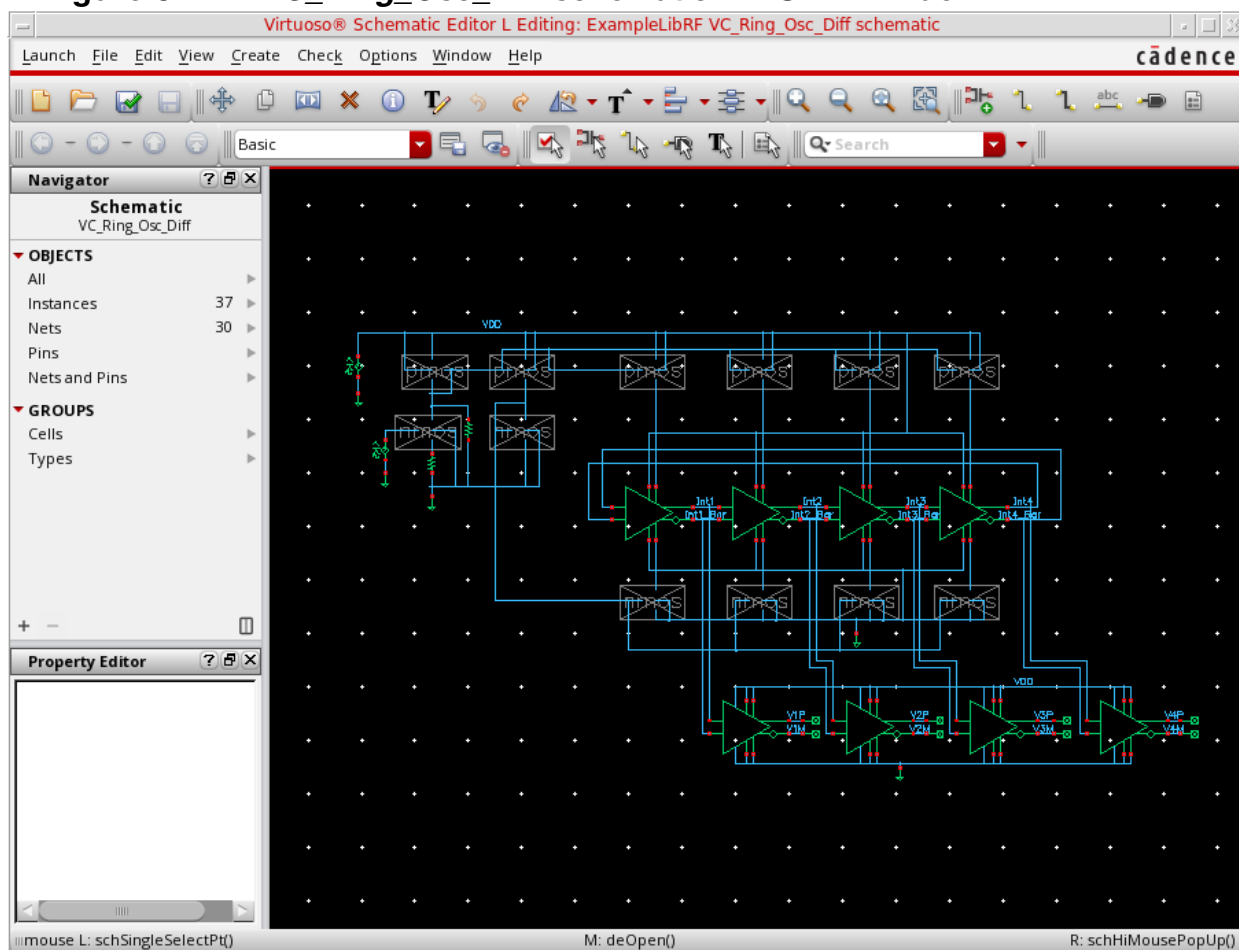
The *Open File* form is displayed.

**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

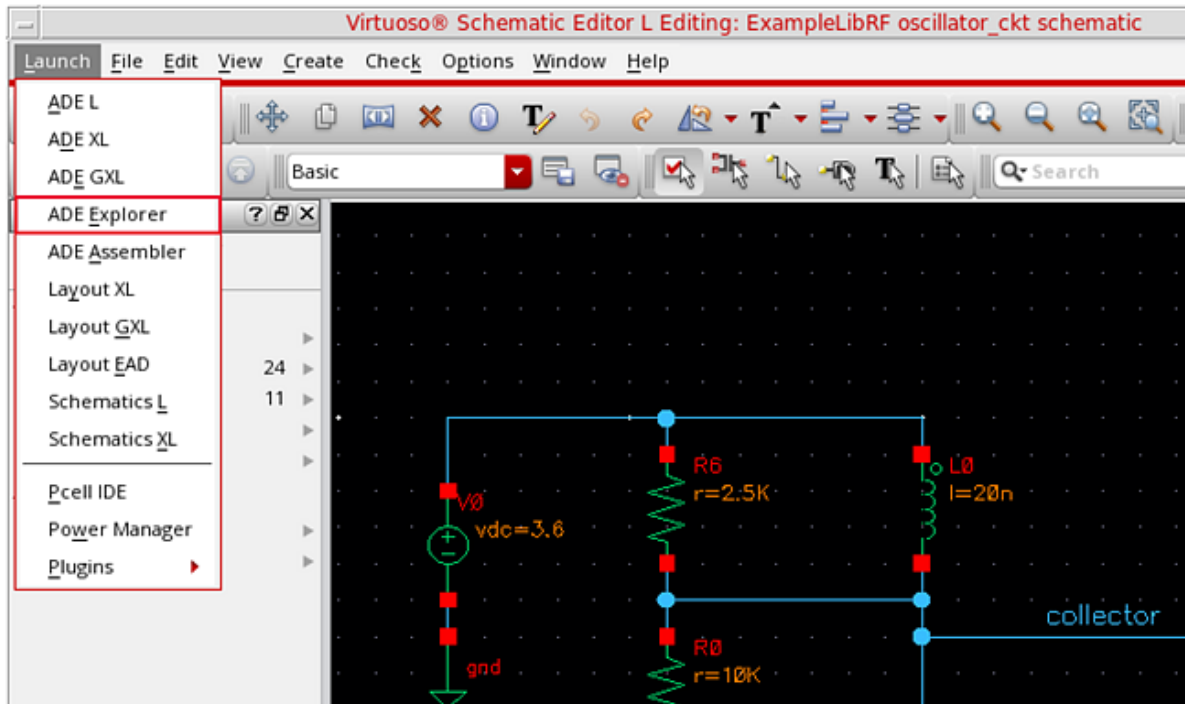
This example computes the periodic steady state solution for the *VC\_Ring\_Osc\_Diff* oscillator circuit. You perform a PSS-Shooting analysis first because the periodic steady state solution must be determined before you can perform any other periodic small-signal analysis, such as pnoise to determine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

### Setting up the PSS Analysis

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

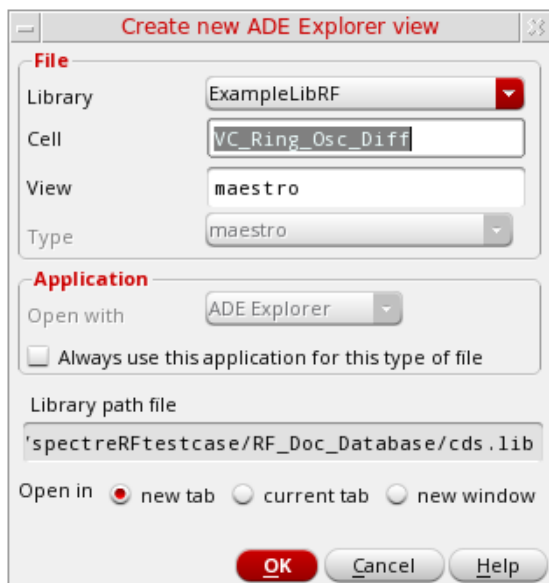
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

**Figure 3-143 Opening ADE Explorer window from VSE window**



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

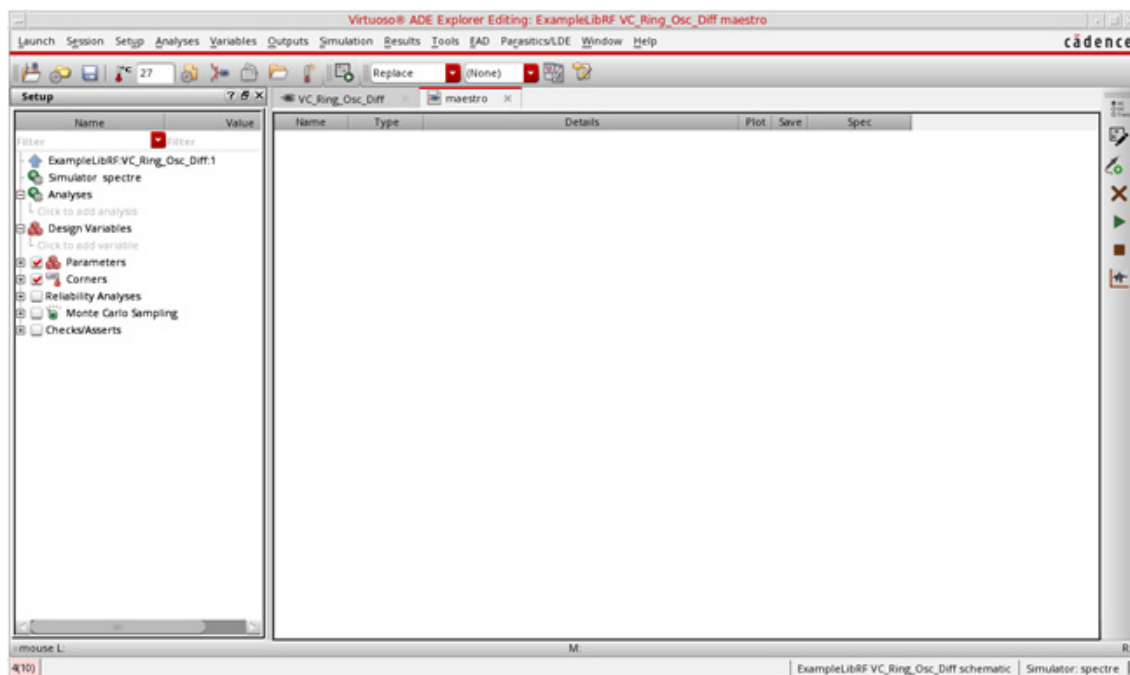
**Figure 3-144 Create new ADE Explorer view Form**



ADE Explorer Window opens, as shown below.

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

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

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

---

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**



The image shows the 'High-Performance Simulation Options' dialog box. At the top, the title bar reads 'High-Performance Simulation Options'. Below the title bar, the 'Simulation Performance Mode' section has four radio buttons: 'Spectre', 'APS' (which is selected and highlighted with a dashed box), 'Spectre X', and 'XPS MS'. The 'General' section contains three sub-sections: 'Accuracy + Speed' with an 'Error Preset' of 'Do not override' (selected); 'APS Options' with a 'Use ++aps' checkbox that is unchecked; and 'Multi-Threading' with 'Auto' selected for the thread mode and a '# Threads' input field. Below this is a 'Processor affinity' checkbox and an input field. The 'Post-Layout Settings' section includes an 'Enable Post-Layout Optimization' checkbox (unchecked), a 'Post-Layout Preset Mode' with 'Default' selected, and an 'Instance Preservation' section with 'None' selected. There is a 'Selected Instances' list box with 'Select' and 'Clear' buttons. At the bottom, there is a 'Savefilter' section with 'none' and 'rc' checkboxes. The bottom of the dialog features a row of buttons: 'OK' (highlighted in red), 'Cancel', 'Defaults', 'Apply', and 'Help'.

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.



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

**Figure 3-148 Save Options Form**

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

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.

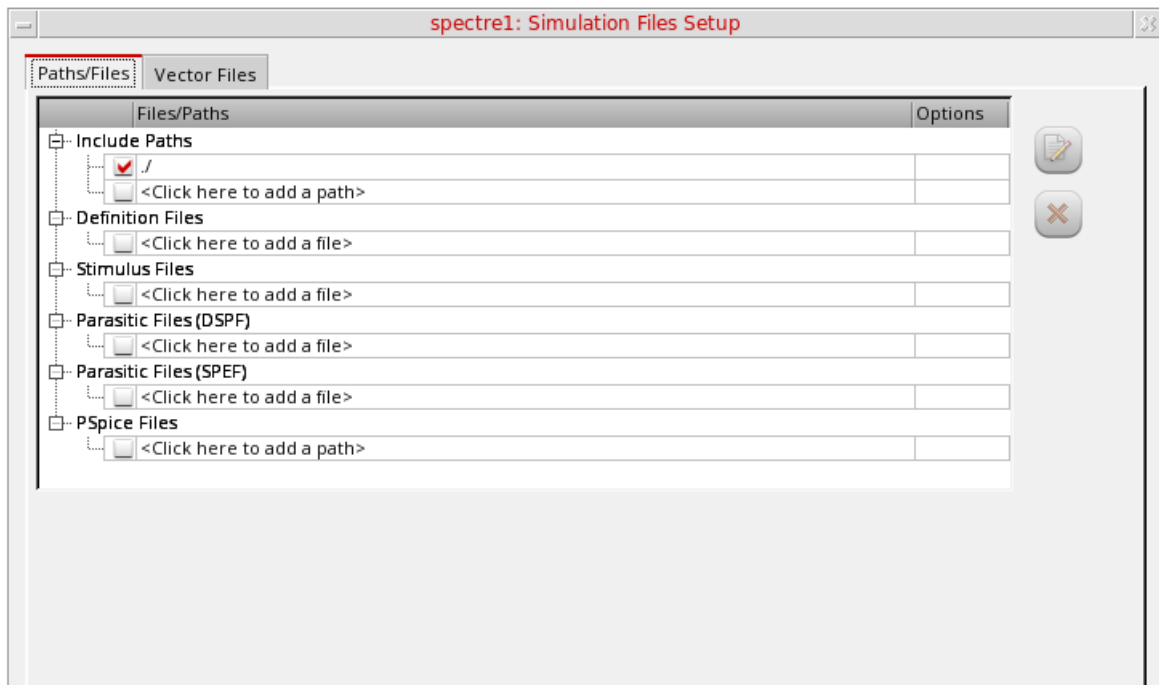
To save the currents, select the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or all if you want to save all the currents in the circuit. When you save currents, more disk space is required for the results file.

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

---

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:

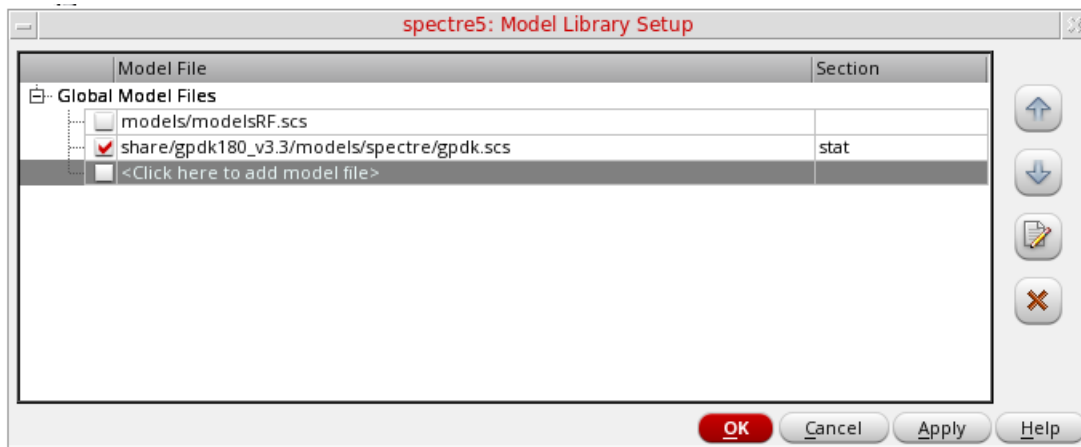
**Figure 3-149** *Simulation Files Setup* Form



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.

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

---

**Figure 3-151 The Choosing Analyses Form**

Choosing Analyses -- ADE Explorer

Analysis

☒ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpss

☐ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative ☐ moderate ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

**OK** Cancel Defaults Apply Help

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

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

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

The screenshot shows the 'Choosing Analyses' form in the Spectre Circuit Simulator. The 'PSS' radio button is selected and highlighted with a red box. Below the analysis type selection, the 'Periodic Steady State Analysis' section is visible, with the 'Shooting' engine selected. The 'Fundamental Tones' section contains a table with columns: #, Name, Expr, Value, Signal, and SrcId. Below the table are input fields for 'Large' and buttons for 'Clear/Add', 'Delete', and 'Update From Hierarchy'. The 'Beat Frequency' radio button is selected, and the 'Auto Calculate' checkbox is checked. The 'Output harmonics' section has a dropdown menu set to 'Number of harmonics' and an input field. The 'Accuracy Defaults (errpreset)' section has three checkboxes: 'conservative', 'moderate', and 'liberal'. The 'Transient-Aided Options' section has a 'Run transient?' section with 'Yes' selected, a 'Detect Steady State' checkbox, a 'Stop Time (tstab)' input field, and a 'Save Initial Transient Results (saveinit)' section with 'no' and 'yes' checkboxes. The 'Dynamic Parameter' checkbox is unchecked. The 'Oscillator' checkbox is unchecked. The 'Sweep' checkbox is unchecked, and the 'New Initial Value For Each Point (restart)' section has 'no' and 'yes' checkboxes. The 'Loadpull' checkbox is unchecked. The 'Enabled' checkbox is unchecked. At the bottom are buttons for 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help', along with an 'Options...' button.

Analysis Type Selection:

- ☒ pss
- ☐ envlp
- ☐ pnoise
- ☐ qpac
- ☐ hb
- ☐ hbnoise
- ☐ pac
- ☐ pxf
- ☐ qpnoise
- ☐ hbac
- ☐ hbsp
- ☐ hbsb
- ☐ psp
- ☐ qpac
- ☐ qpnoise
- ☐ hbac
- ☐ hbnoise
- ☐ pstb
- ☐ qpss
- ☐ qpss
- ☐ hbnoise

Periodic Steady State Analysis

Engine: ☒ Shooting ☐ Harmonic Balance

Fundamental Tones

| # | Name | Expr | Value | Signal | SrcId |
|---|------|------|-------|--------|-------|
|---|------|------|-------|--------|-------|

Large

Clear/Add Delete Update From Hierarchy

☒ Beat Frequency  Auto Calculate ☒

☐ Beat Period

Output harmonics

Number of harmonics

Accuracy Defaults (errpreset)

☐ conservative ☐ moderate ☐ liberal

Transient-Aided Options

Run transient? ☒ Yes ☐ No ☐ Decide automatically

Detect Steady State ☐ Stop Time (tstab)

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Oscillator ☐

Sweep ☐

New Initial Value For Each Point (restart) ☐ no ☐ yes

Loadpull ☐

Enabled ☐

Options...

OK Cancel Defaults Apply Help

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.

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

---

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.

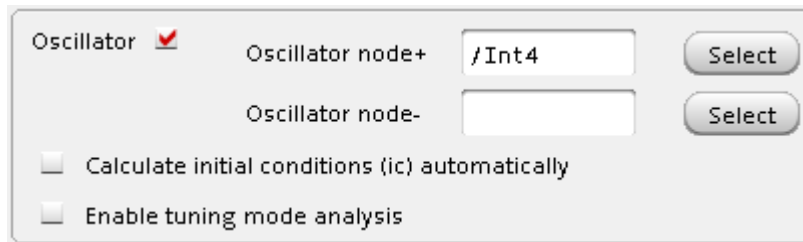
23. Select *yes* for *Run Transient*.

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**



Oscillator ☒ Oscillator node+ /Int4 Select

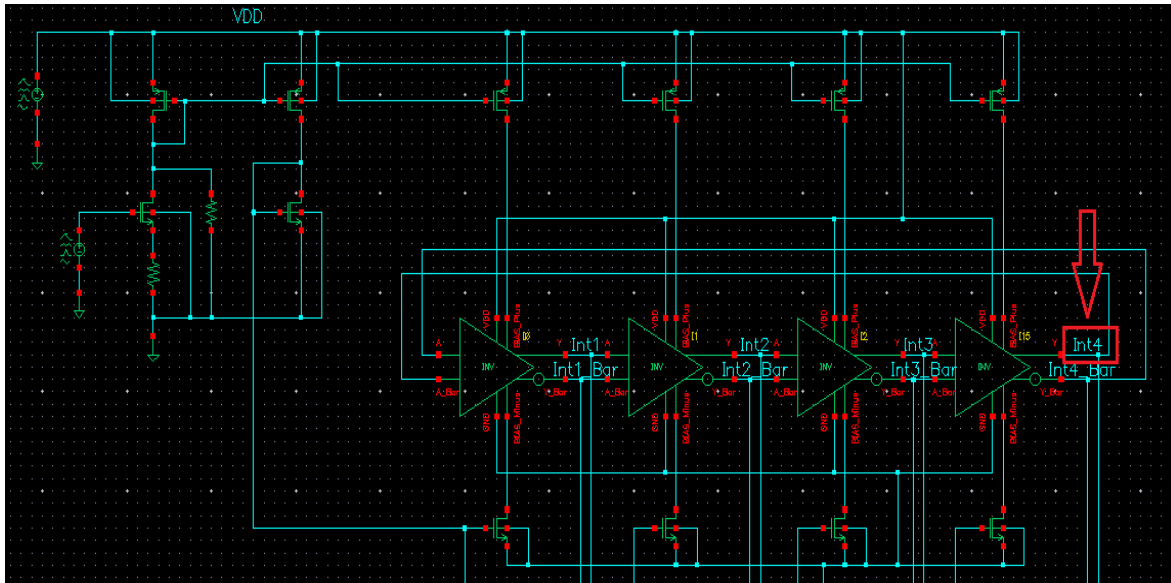
Oscillator node- Select

☐ Calculate initial conditions (ic) automatically

☐ Enable tuning mode analysis

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:

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

**Figure 3-155 Choosing Analyses Form - PSS-Shooting Setup**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ sp ☐ envlp

☒ pss ☐ pac ☐ pstb ☐ pnoise

☐ pxf ☐ psp ☐ qpss ☐ qpac

☐ qpnoise ☐ qpxf ☐ qpssp ☐ hb

☐ hbac ☐ hbnoise ☐ hbbsp

Periodic Steady State Analysis

Engine ☒ Shooting ☐ Harmonic Balance

Fundamental Tones

| # | Name | Expr | Value | Signal | SrcId |
|---|------|------|-------|--------|-------|
|---|------|------|-------|--------|-------|

Clear/Add Delete Update From Hierarchy

☒ Beat Frequency ☐ Beat Period

1.96 Auto Calculate ☐

Output harmonics

Number of harmonics 28

Accuracy Defaults (errpreset)

☒ conservative ☐ moderate ☐ liberal

Transient-Aided Options

Run transient? ☒ Yes ☐ No ☐ Decide automatically

Detect Steady State ☐ Stop Time (tstab) 2n

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter ☐

Oscillator ☒

Oscillator node+ /Int4 Select

Oscillator node- Select

☒ Calculate initial conditions (ic) automatically

☐ Enable tuning mode analysis

OK Cancel Defaults Apply Help

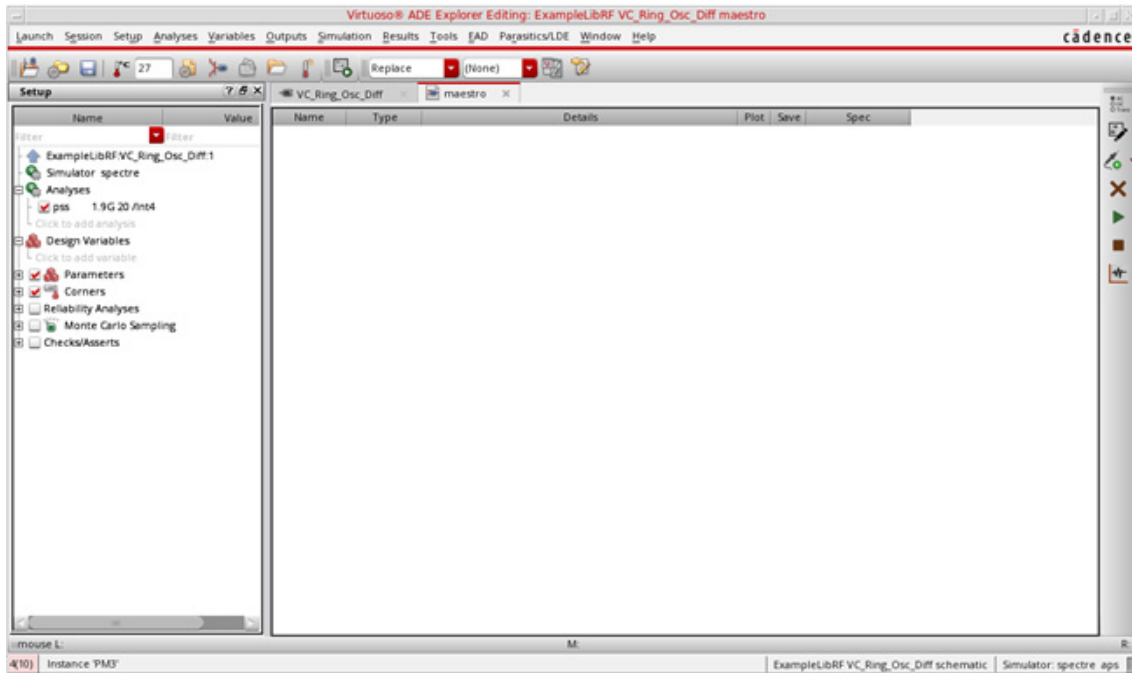
29. Click **OK**.



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

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.

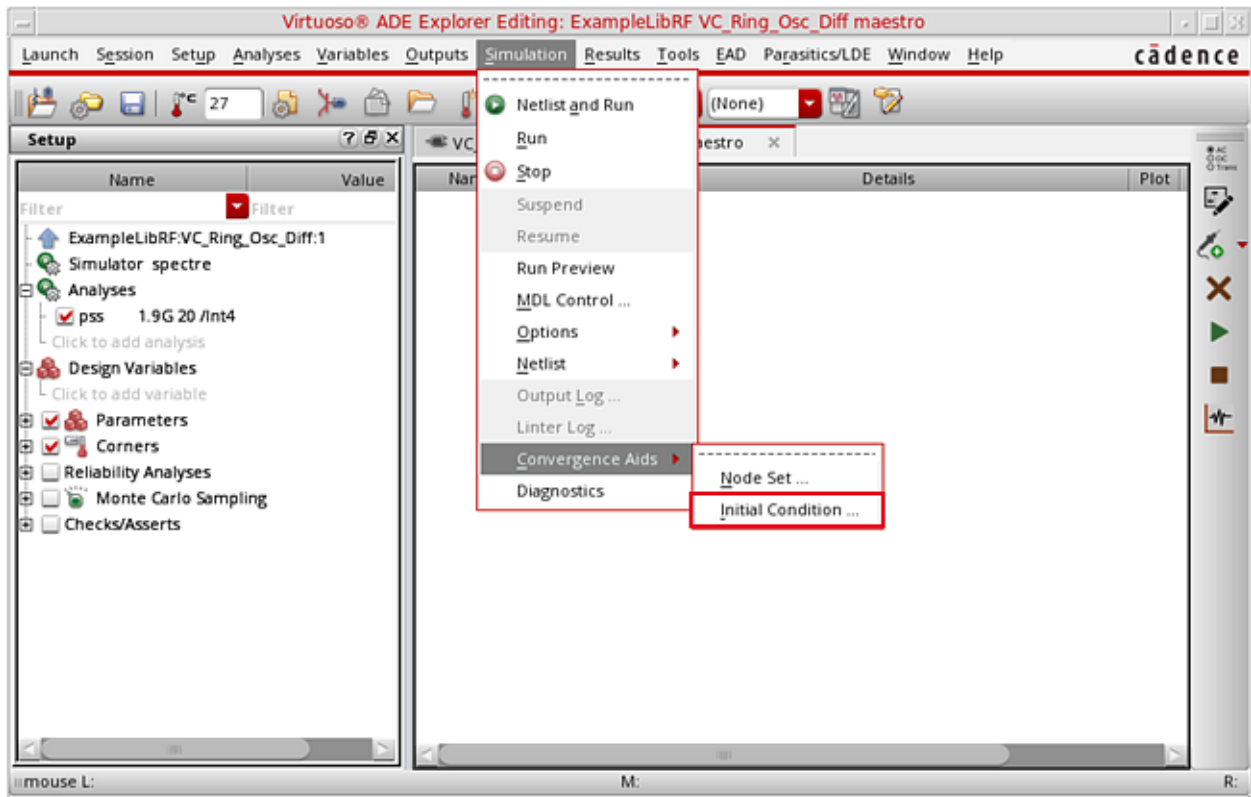
In this case, we will set one stage of the oscillator so that one differential output is high, and one differential output (in the same stage) is low.

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 -

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

Figure 3-157 PSS Analysis - Setting Initial Condition in ADE

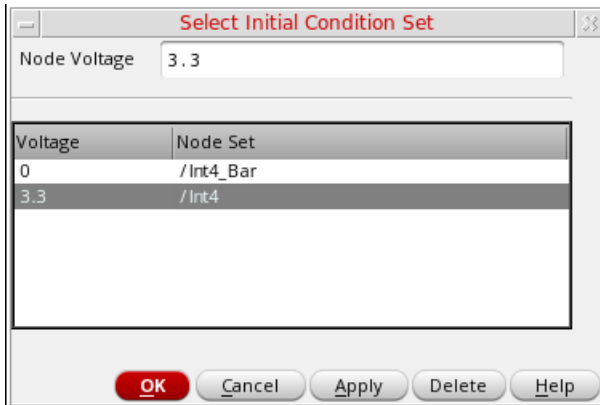


The *Select Initial Condition Set* form is displayed.

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.

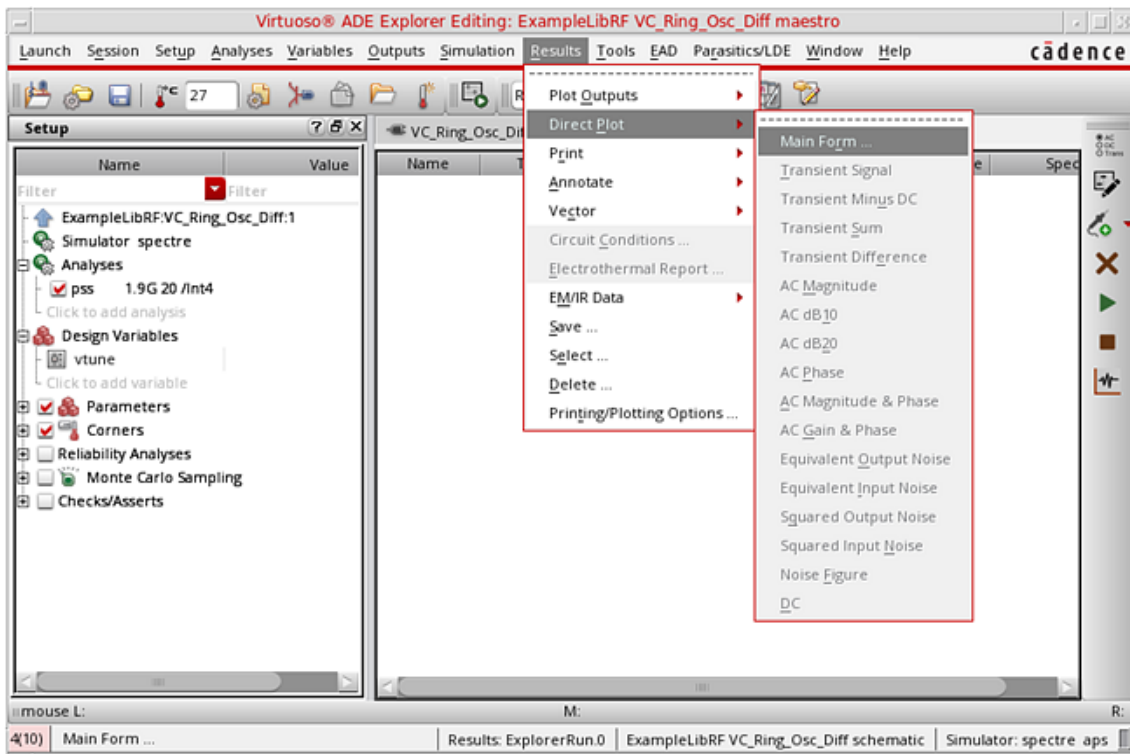
Next, you will plot the results.

### Plotting the PSS Analysis Results

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

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

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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☒ pss ☐ tstab

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Harmonic Frequency ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp. ☐ THD

Select: Net

**Sweep**

☒ spectrum ☐ time

Signal Level: ☒ peak ☐ rms

**Modifier**

☒ Magnitude ☐ Phase ☐ dB20  
☐ Real ☐ Imaginary

**Scalar Expression**

Value At (harms):

Add To Outputs: ☐

> Select Net on schematic...

Close Help

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).

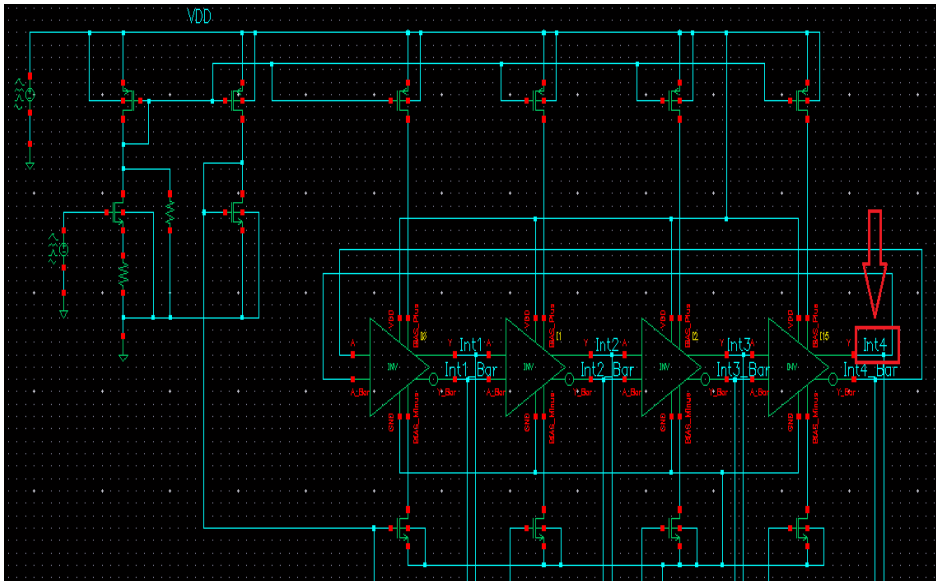
The *Direct Plot Form* window should like the following:

**Figure 3-161 PSS Analysis Direct Plot Setup - Initial Transient**



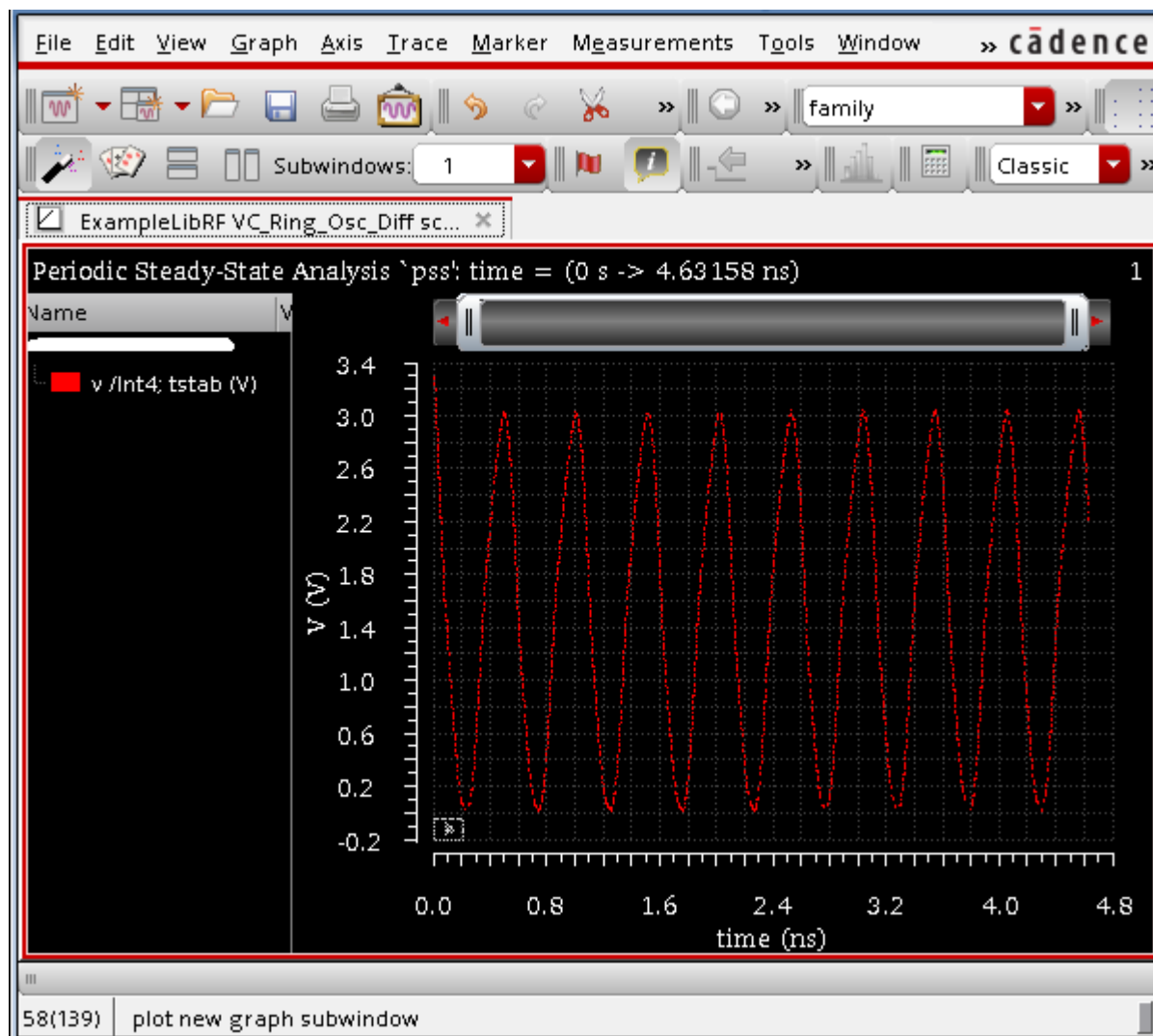
5. Select the *Int4* net in the schematic. It is located just below the *Int4* label.

**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).

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

---

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**

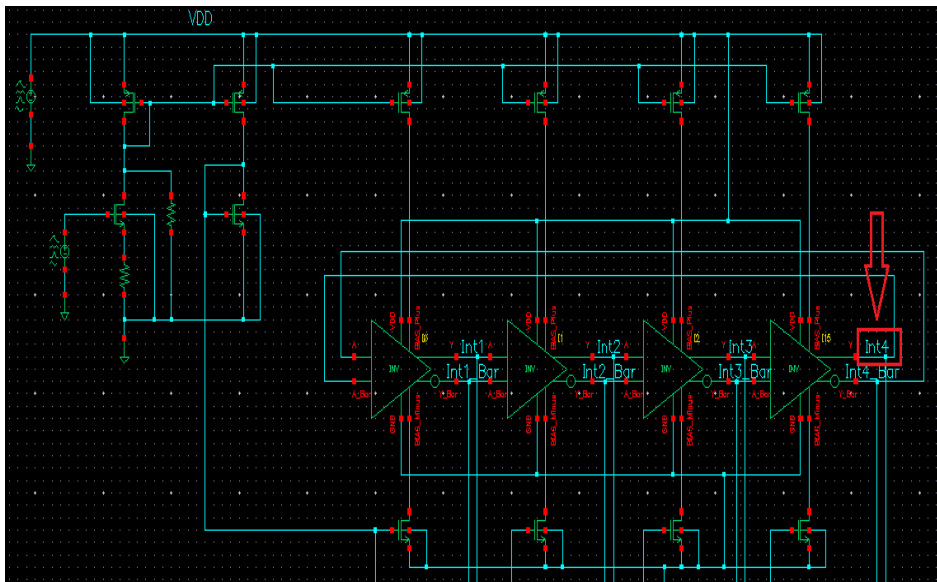
The screenshot shows the 'Direct Plot Form' window with the following settings:

- Plotting Mode:** New Win
- Analysis:** ☒ pss, ☐ tstab
- Function:** ☒ Voltage, ☐ Current, ☐ Power, ☐ Voltage Gain, ☐ Current Gain, ☐ Power Gain, ☐ Transconductance, ☐ Transimpedance, ☐ Compression Point, ☐ IPN Curves, ☐ Power Contours, ☐ Reflection Contours, ☐ Harmonic Frequency, ☐ Power Added Eff., ☐ Power Gain Vs Pout, ☐ Comp. Vs Pout, ☐ Node Complex Imp., ☐ THD
- Select:** Net
- Sweep:** ☒ spectrum, ☐ time
- Signal Level:** ☐ peak, ☒ rms
- Modifier:** ☐ Magnitude, ☐ Phase, ☒ dB20, ☐ Real, ☐ Imaginary
- Scalar Expression:** Value At (harms)
- Buttons:** Add To Outputs, Replot, Close, Help
- Footer:** > Select Net on schematic...

8. Select *Int4* net in the schematic. It is located just below the *Int4* label.

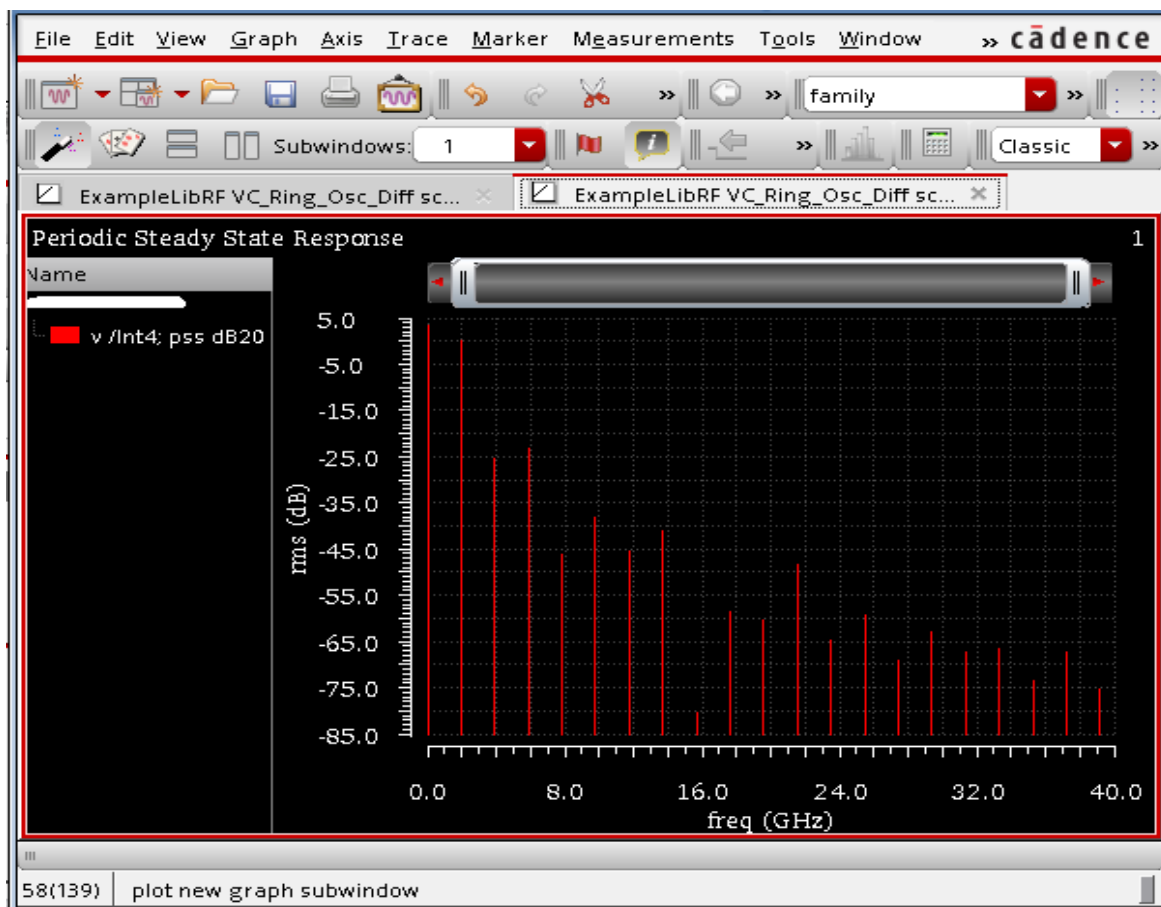


**Figure 3-165** Selecting *Int4* net on schematic



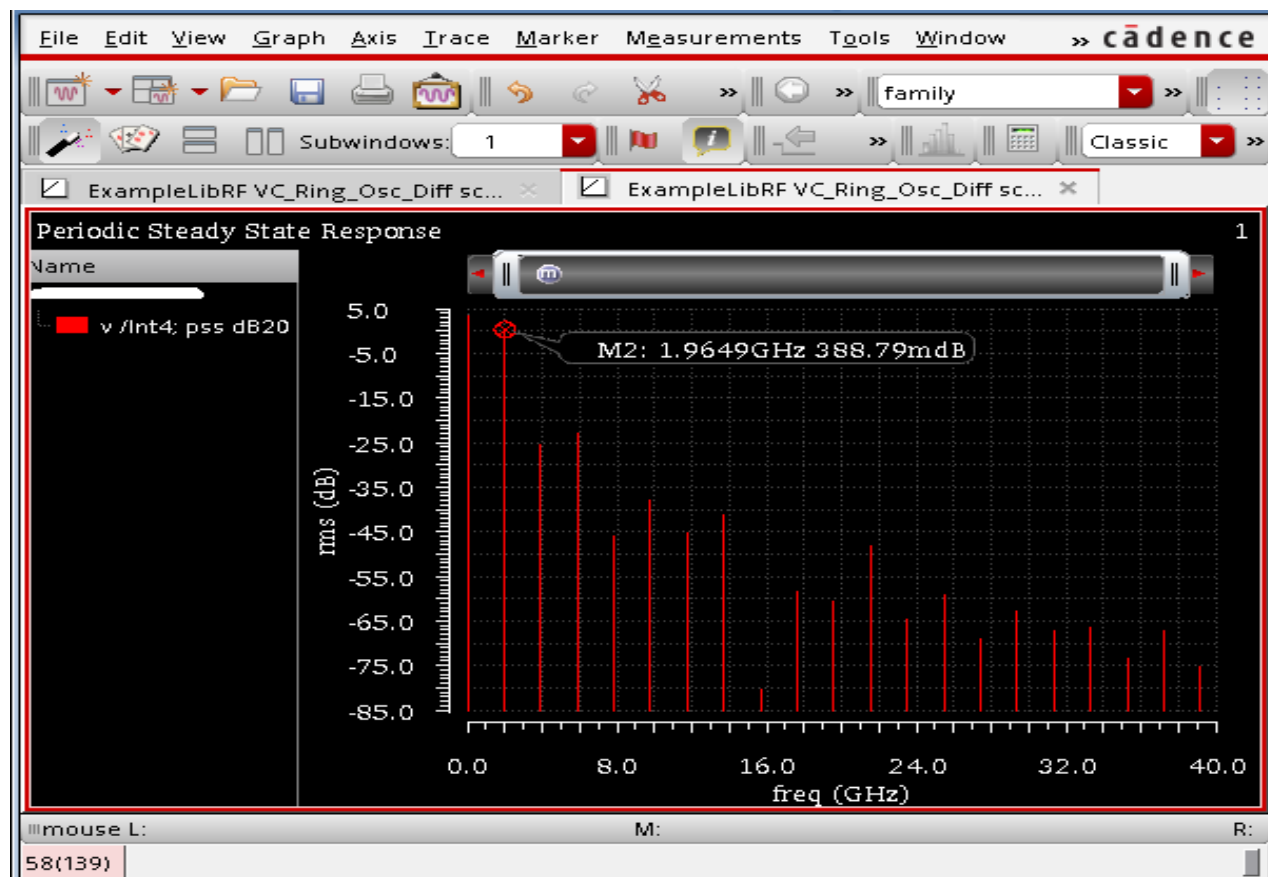
The waveform window is displayed, as shown below.

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.

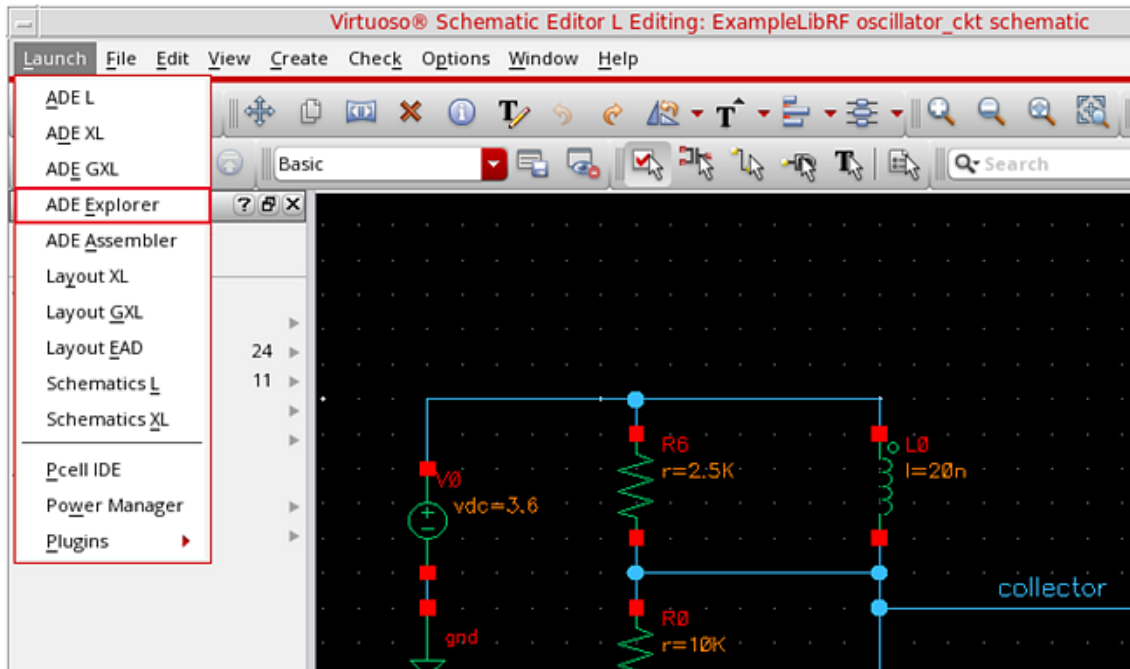
In this measurement, you will perform the FM jitter measurement and plot the cycle jitter Jc.

### Setting up the PSS Analysis

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

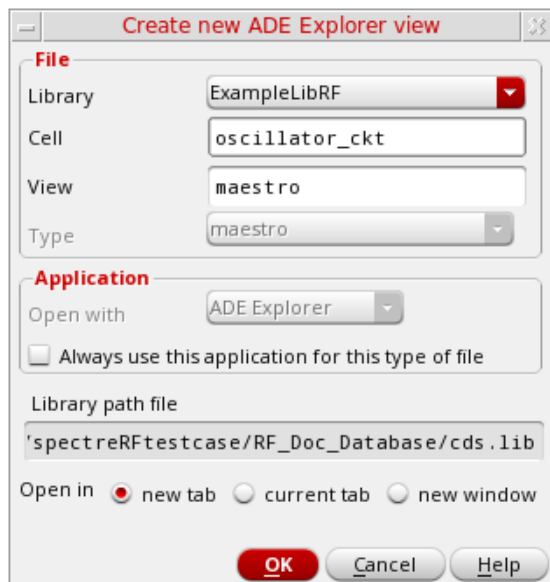
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

**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.

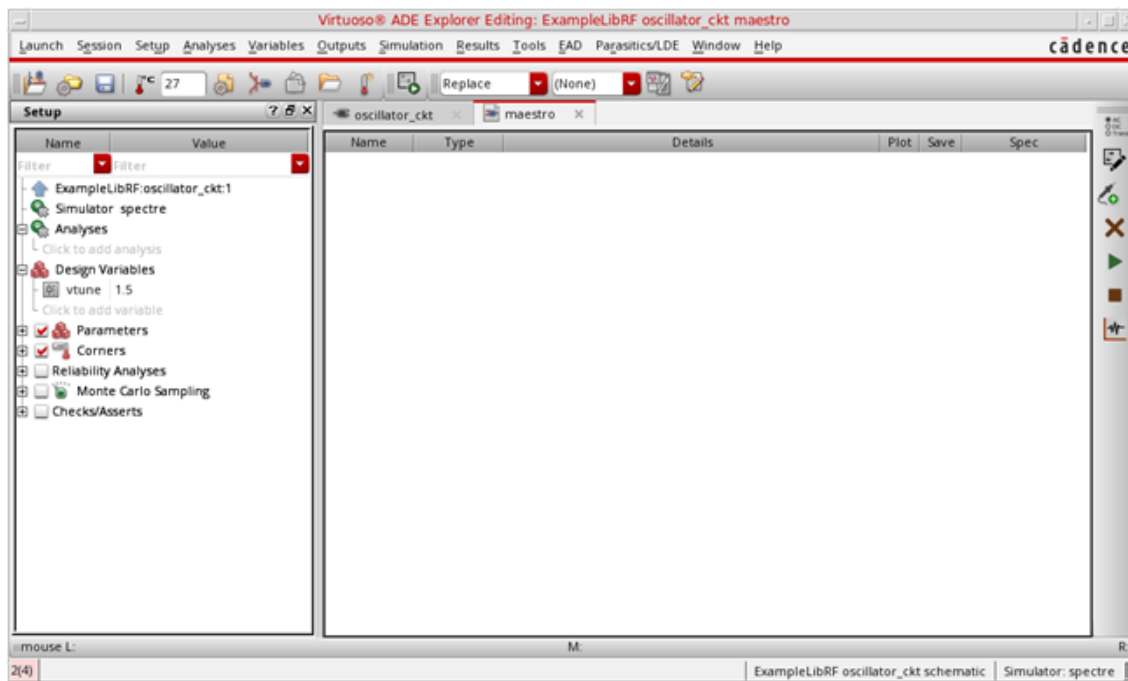
**Figure 3-169 Create new ADE Explorer view**



3. Leave each option to the default selections and click *OK*.  
ADE Explorer is displayed, as shown below.

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

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

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

---

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**



The image shows the 'High-Performance Simulation Options' dialog box. At the top, the title bar reads 'High-Performance Simulation Options'. Below the title bar, the 'Simulation Performance Mode' section has four radio buttons: 'Spectre', 'APS' (which is selected and highlighted with a dashed box), 'Spectre X', and 'XPS MS'. The 'General' section contains three sub-sections: 'Accuracy + Speed' with an 'Error Preset' of 'Do not override' (selected); 'APS Options' with a 'Use ++aps' checkbox that is unchecked; and 'Multi-Threading' with 'Auto' selected for the threading mode, a '# Threads' input field, and a 'Processor affinity' checkbox that is unchecked. The 'Post-Layout Settings' section includes an 'Enable Post-Layout Optimization' checkbox (unchecked), a 'Post-Layout Preset Mode' with 'Default' selected, 'Instance Preservation' set to 'None', a 'Selected Instances' list box with 'Select' and 'Clear' buttons, and a 'Savefilter' section with 'none' and 'rc' options.

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.

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

Figure 3-173 Save Options Form

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

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**.

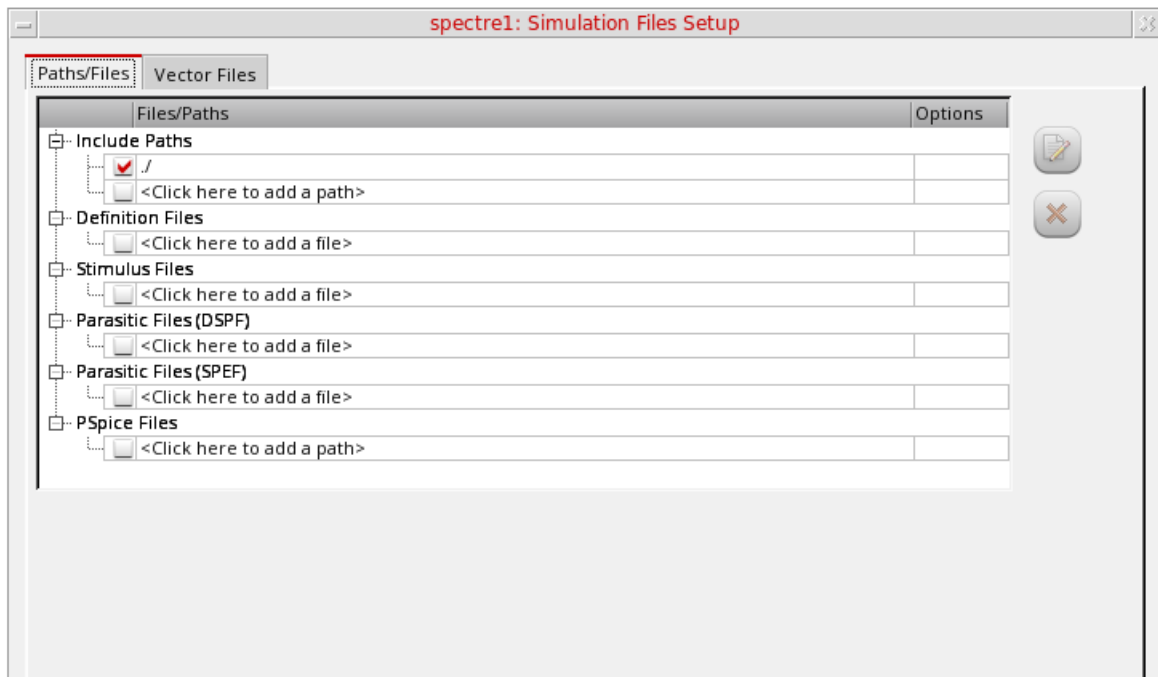


## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

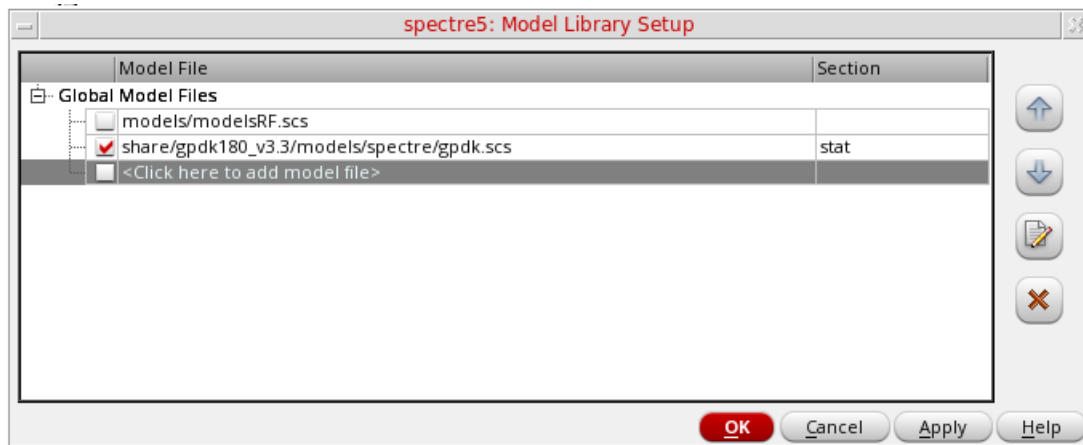
12. Select *Setup - Simulation Files*.
13. In the *Simulation Files Setup* form which gets opened, enter . / by clicking in the *Include Paths* section. It should look like as shown below:

**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.

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

---

**Figure 3-176 The Choosing Analyses Form**

Choosing Analyses -- ADE Explorer

Analysis

☒ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpsp

☐ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative ☐ moderate ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

**OK** Cancel Defaults Apply Help

**20.** Select *pss* as *Analysis*. The form expands, as shown below.

**Figure 3-177 The Choosing Analyses Form- Setting PSS Analysis**

☐ envlp   ☒ pss   ☐ pac   ☐ pstb  
☐ pnoise   ☐ pxf   ☐ psp   ☐ qpss  
☐ qpac   ☐ qpnoise   ☐ qpxf   ☐ qpssp  
☐ hb   ☐ hbac   ☐ hbstb   ☐ hbnoise  
☐ hbasp   ☐ hbxp

Periodic Steady State Analysis

Engine   ☐ Shooting   ☒ Harmonic Balance

Tones

| Name | Expr | Value | SrcId |
|------|------|-------|-------|
|      |      |       |       |

Beat Frequency      Auto Calculate   ☐

Oversample Factor  

Number of Harmonics  

Accuracy Defaults (errpreset)

☐ conservative   ☐ moderate   ☐ liberal

Convergence

Additional Time for Transient-Aided HB (tstab)  

Save Initial Transient Results (saveinit)   ☐ no   ☐ yes

Harmonic Balance Homotopy Method    ▼

Dynamic Parameter   ☐

Oscillator   ☐

Sweep   ☐

New Initial Value For Each Point (restart)   ☐ no   ☐ yes

Loadpull   ☐

Enabled   ☒

Options...

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.

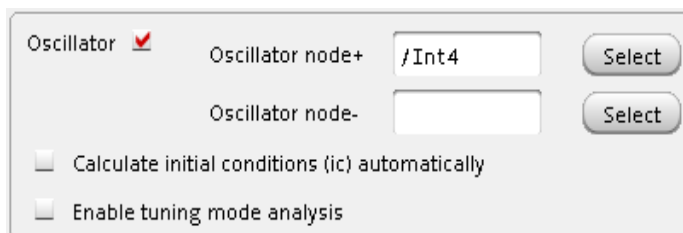
25. Select *yes* for *Run Transient*.

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.

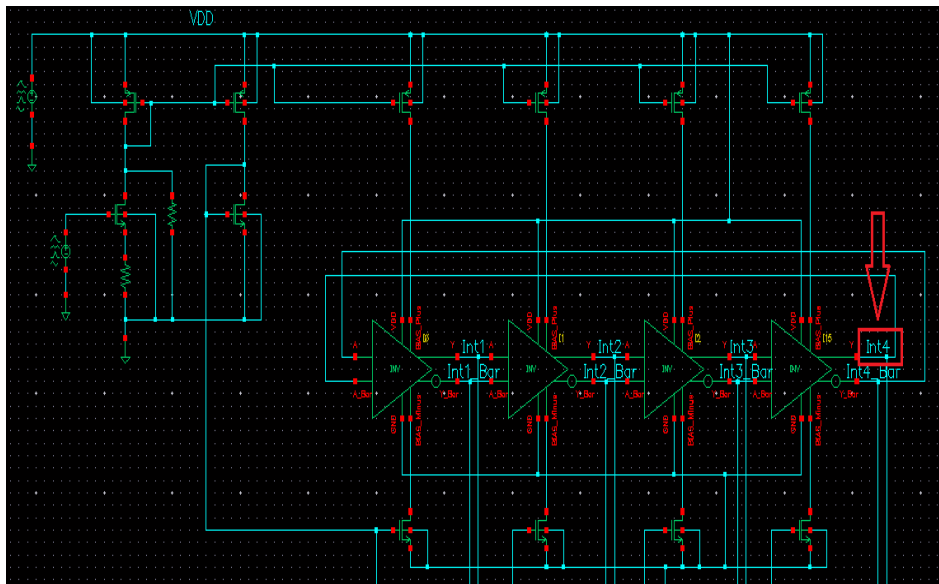
**Figure 3-178 The Choosing Analyses Form - Oscillator Section**



Oscillator ☒ Oscillator node+ /Int4 Select  
Oscillator node- Select  
☐ Calculate initial conditions (ic) automatically  
☐ Enable tuning mode analysis

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 look like the following:

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

**Figure 3-180 Choosing Analyses Form - PSS-Shooting Method Setup**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ sp ☐ envlp

☒ pss ☐ pac ☐ pstb ☐ pnoise

☐ pxf ☐ psp ☐ qpss ☐ qpac

☐ qpnoise ☐ qpxf ☐ qpssp ☐ hb

☐ hbac ☐ hbnoise ☐ hbssp

Periodic Steady State Analysis

Engine ☒ Shooting ☐ Harmonic Balance

Fundamental Tones

| # | Name | Expr | Value | Signal | SrcId |
|---|------|------|-------|--------|-------|
|---|------|------|-------|--------|-------|

Large

Clear/Add Delete Update From Hierarchy

☒ Beat Frequency ☐ Beat Period 1.9G Auto Calculate

Output harmonics

Number of harmonics 20

Accuracy Defaults (errpreset)

☒ conservative ☐ moderate ☐ liberal

Transient-Aided Options

Run transient? ☒ Yes ☐ No ☐ Decide automatically

Detect Steady State ☐ Stop Time (tstab) 2n

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter

Oscillator ☒ Oscillator node+ /Int4 Select

Oscillator node- Select

☒ Calculate initial conditions (ic) automatically

☐ Enable tuning mode analysis

OK Cancel Defaults Apply Help

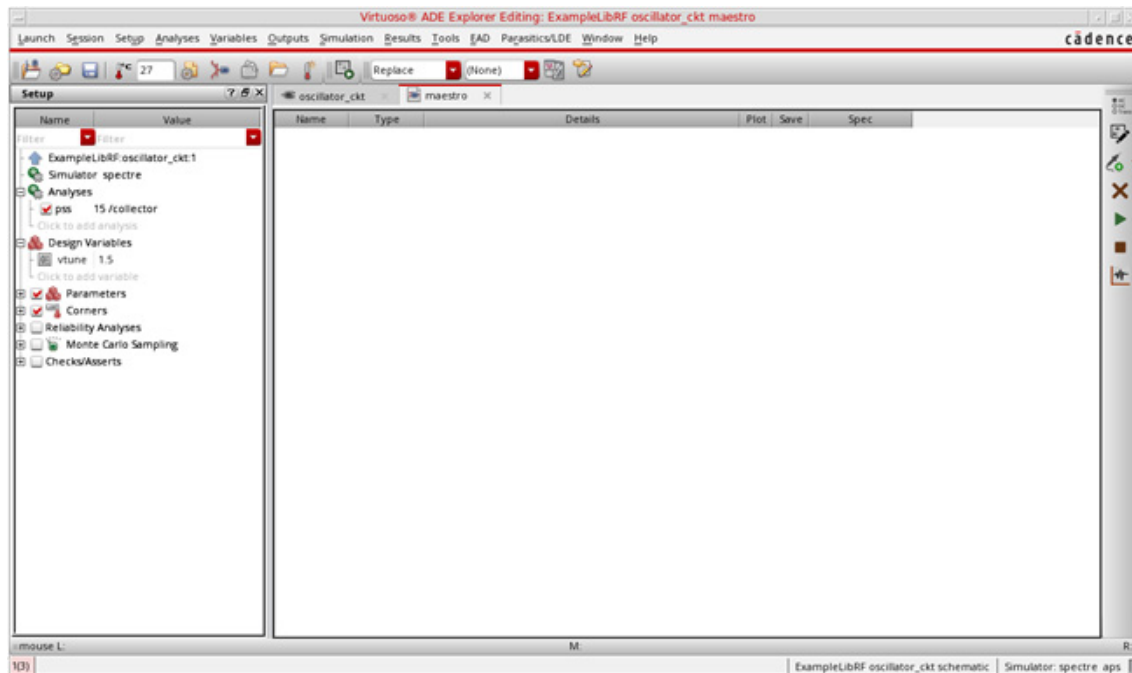
31. Click *Ok*.

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

---

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.

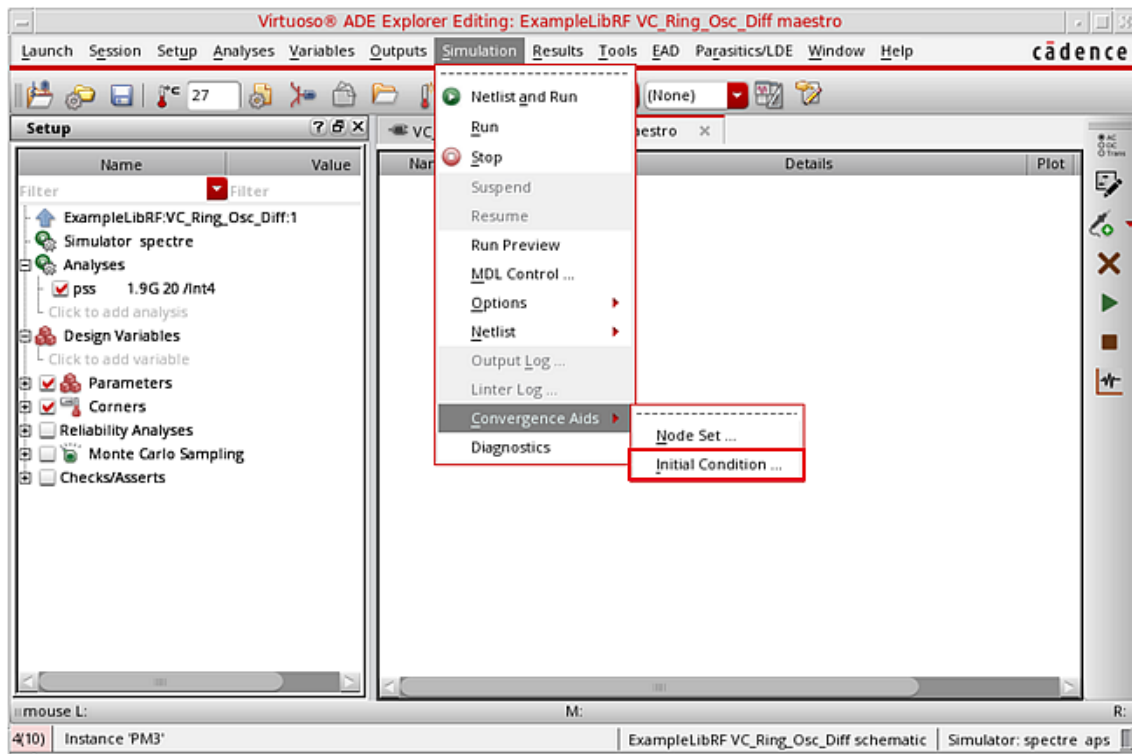
This defines all the nodes in the circuit, and the nodes that are connected to the nodes that are forced are pulling as hard as they can to get the forced nodes to change to the other state. When the forcing condition is removed, this action starts the oscillations.

Set the initial conditions by choosing *Simulation - Convergence Aids - Initial Condition* in ADE Explorer, as shown below.



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

**Figure 3-182 PSS Analysis - Setting Initial Condition in ADE Explorer**

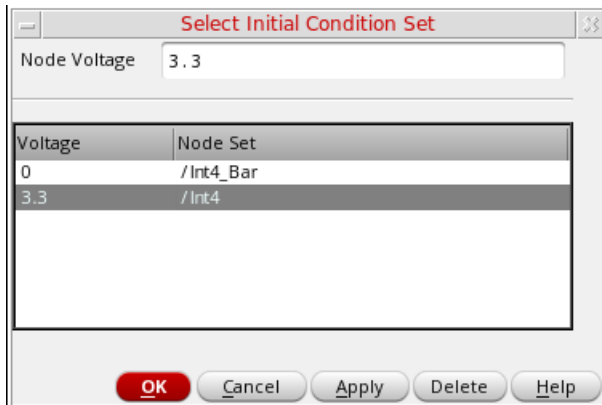


This will open the *Select Initial Condition Set* dialog box.

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**



| Voltage | Node Set  |
|---------|-----------|
| 0       | /Int4_Bar |
| 3.3     | /Int4     |

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**

The image shows the 'Choosing Analyses Form - *pnoise* Analysis Setup' dialog box. At the top, there is a grid of radio buttons for selecting analysis types: *pss*, *pac*, *pstb*, *pnoise* (selected and highlighted with a red box), *pxf*, *psp*, *qpss*, *qpac*, *qnoise*, *qpxf*, *qpss*, *hb*, *hbac*, *hbnoise*, and *hbss*. Below this, the 'Periodic Noise Analysis' section contains a text field for 'PSS Beat Frequency (Hz)' with the value '1.9G'. A 'Multiple noise' checkbox is present. The 'Sweep Type' section includes a 'Sweep Type' dropdown set to 'default', a 'Relative Harmonic' text field, and an 'Output Frequency Sweep Range (Hz)' section with a 'Start-Stop' dropdown, 'Start' and 'Stop' text fields, and a 'Sweep Type' dropdown set to 'Automatic'. An 'Add Specific Points' checkbox is also shown. The 'Sidebands' section has a 'Method' dropdown set to 'default', a 'Maximum sideband' dropdown, and a note: 'When using shooting engine, default value is 7.' The 'Output' section includes an 'Output' dropdown set to 'probe', an 'Output Probe Instance' text field, and a 'Select' button. The 'Noise Type' section shows a 'Noise Type' dropdown set to 'timeaverage', a description: 'Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis', and a 'Contribution Type' section with radio buttons for 'USB' (selected), 'AM', 'PM', 'AM&PM', and 'ALL(AM,PM,USB,LSB)'. A note states: 'For fmjitter, PM noise must be enabled'. There is also a 'Noise Separation' section with 'yes' and 'no' checkboxes, and a 'Separate noise into source and gain' checkbox. At the bottom, there is an 'Enabled' checkbox, an 'Options...' button, and a row of buttons: 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'.

### 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*.

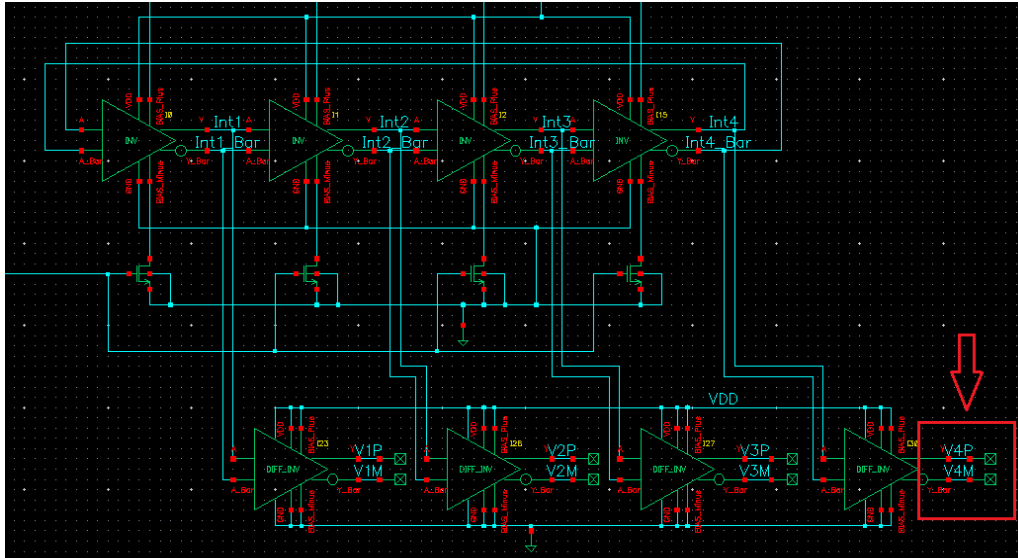
- a. Type /V4P in the *Positive Output Node* field. You can also select V4P net from schematic by clicking *Select* on the right of the *Positive Output Node* field and then selecting the net just below the *out* label in schematic.
- b. Type /V4M in the *Negative Output Node* field. If the second node, 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.

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

---

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**

☐ stb   ☐ pz   ☐ sp   ☐ envlp  
☐ pss   ☐ pac   ☐ pstb   ☒ pnoise  
☐ pxf   ☐ psp   ☐ qpss   ☐ qpac  
☐ qpnoise   ☐ qpxf   ☐ qpsp   ☐ hb  
☐ hbac   ☐ hbnoise   ☐ hbasp

Periodic Noise Analysis

PSS Beat Frequency (Hz)

Multiple pnoise ☐

Sweeptype ☒ relative   Relative Harmonic

Output Frequency Sweep Range (Hz)

☒ Start-Stop   Start    Stop

Sweep Type

☒ Logarithmic   ☐ Points Per Decade    ☐ Number of Steps

Add Specific Points ☐

Sidebands

Method ☐ default   ☒ fullspectrum

Maximum sideband

Calculates noise contributions up to the frequency determined by PSS time point resolution

Output

☒ voltage   Positive Output Node   

Negative Output Node   

Noise Type ☒ timeaverage

Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis

Contribution Type:

☐ USB   ☐ AM   ☐ PM   ☐ AM&PM   ☒ ALL(AM,PM,USB,LSB)

For fmjitter, PM noise must be enabled

Enabled ☒

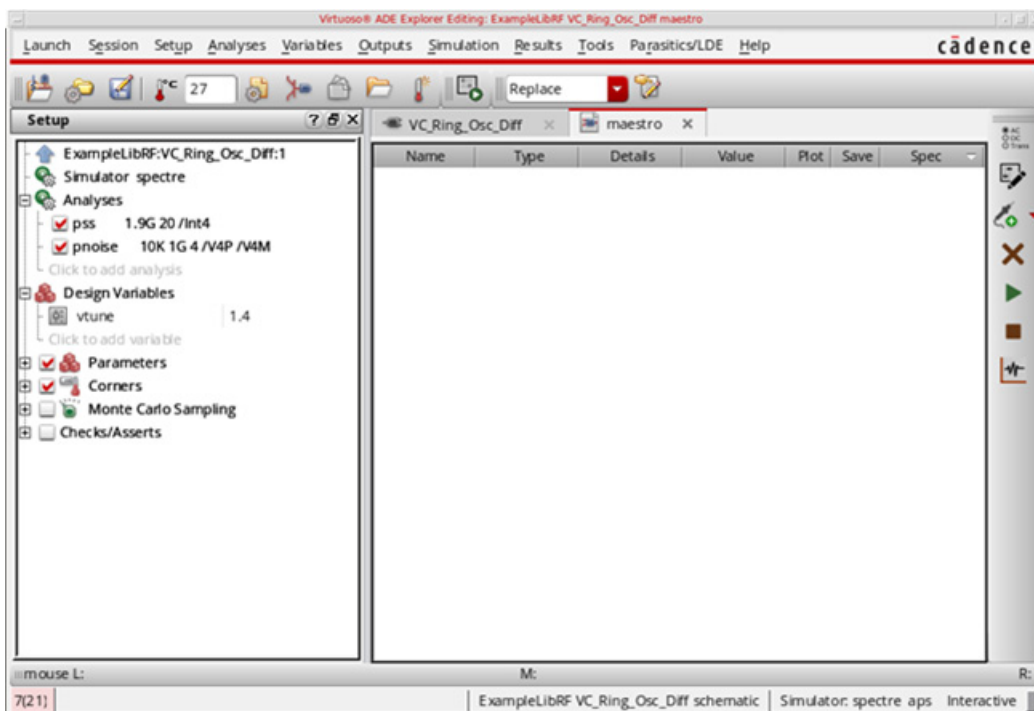
          

8. Click *OK* to close the *Choosing Analyses* form.

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

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

The image shows the 'Direct Plot Form' dialog box in a software application. The title bar is 'Direct Plot Form'. The 'Plotting Mode' is set to 'Append'. The 'Analysis' section has two radio buttons: 'pss' (selected) and 'pnoise'. The 'Function' section has a list of radio buttons: 'Voltage' (selected), 'Current', 'Power', 'Voltage Gain', 'Current Gain', 'Power Gain', 'Transconductance', 'Transimpedance', 'Compression Point', 'IPN Curves', 'Power Contours', 'Reflection Contours', 'Harmonic Frequency', 'Power Added Eff.', 'Power Gain Vs Pout', 'Comp. Vs Pout', 'Node Complex Imp.', and 'THD'. The 'Select' dropdown is set to 'Net'. The 'Sweep' section has two radio buttons: 'spectrum' (selected) and 'time'. The 'Signal Level' section has two radio buttons: 'peak' (selected) and 'rms'. The 'Modifier' section has five radio buttons: 'Magnitude' (selected), 'Phase', 'dB20', 'Real', and 'Imaginary'. There is an 'Add To Outputs' checkbox which is unchecked. At the bottom, there is a link '> Select Net on schematic...' and three buttons: 'OK', 'Cancel', and 'Help'.

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

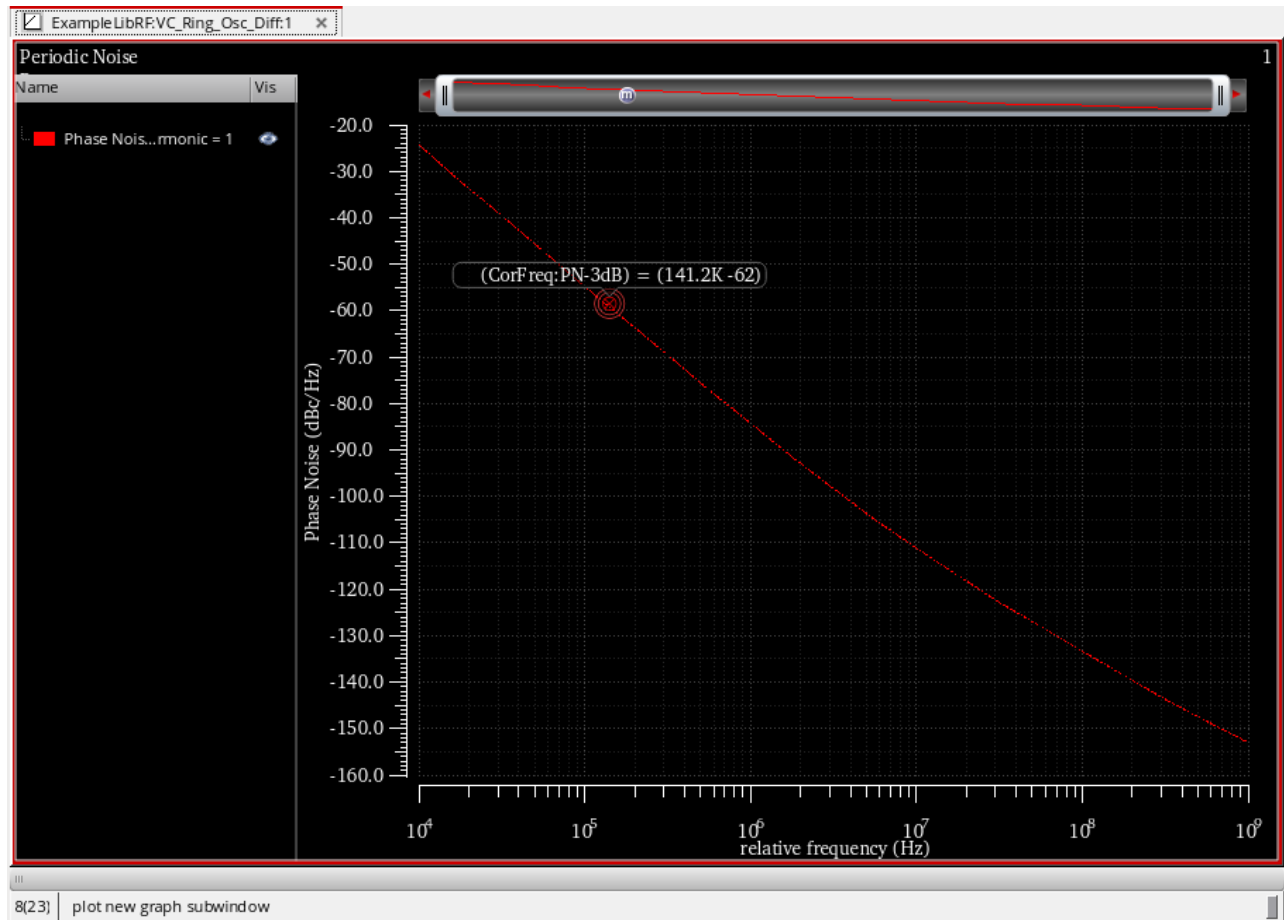
The image shows a software dialog box titled "Direct Plot Form". It contains several sections for configuring an analysis plot:

- Plotting Mode:** A dropdown menu set to "Append".
- Analysis:** Two radio buttons, "pss" and "pnoise". The "pnoise" button is selected.
- Noise Type:** Four radio buttons: "USB", "LSB", "AM", and "PM". The "PM" button is selected.
- Function:** Six radio buttons arranged in two rows: "Output Noise", "-20dB/dec Line", "Jc", "Jcc", and "Phase Noise". The "Phase Noise" button is selected and has a dashed border.
- Buttons:** "Add To Outputs" (with an unchecked checkbox), "Plot", "OK", "Cancel", and "Help".
- Text:** A note at the bottom says "> Press plot button on this form...".

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  $J_c$  (cycle or period) jitter value.

1. In the *Direct Plot Form*, select  $J_c$  in the *Function* section.

The form expands.  $J_c$  is cycle or period jitter while  $J_{cc}$  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  $J_c$  (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.

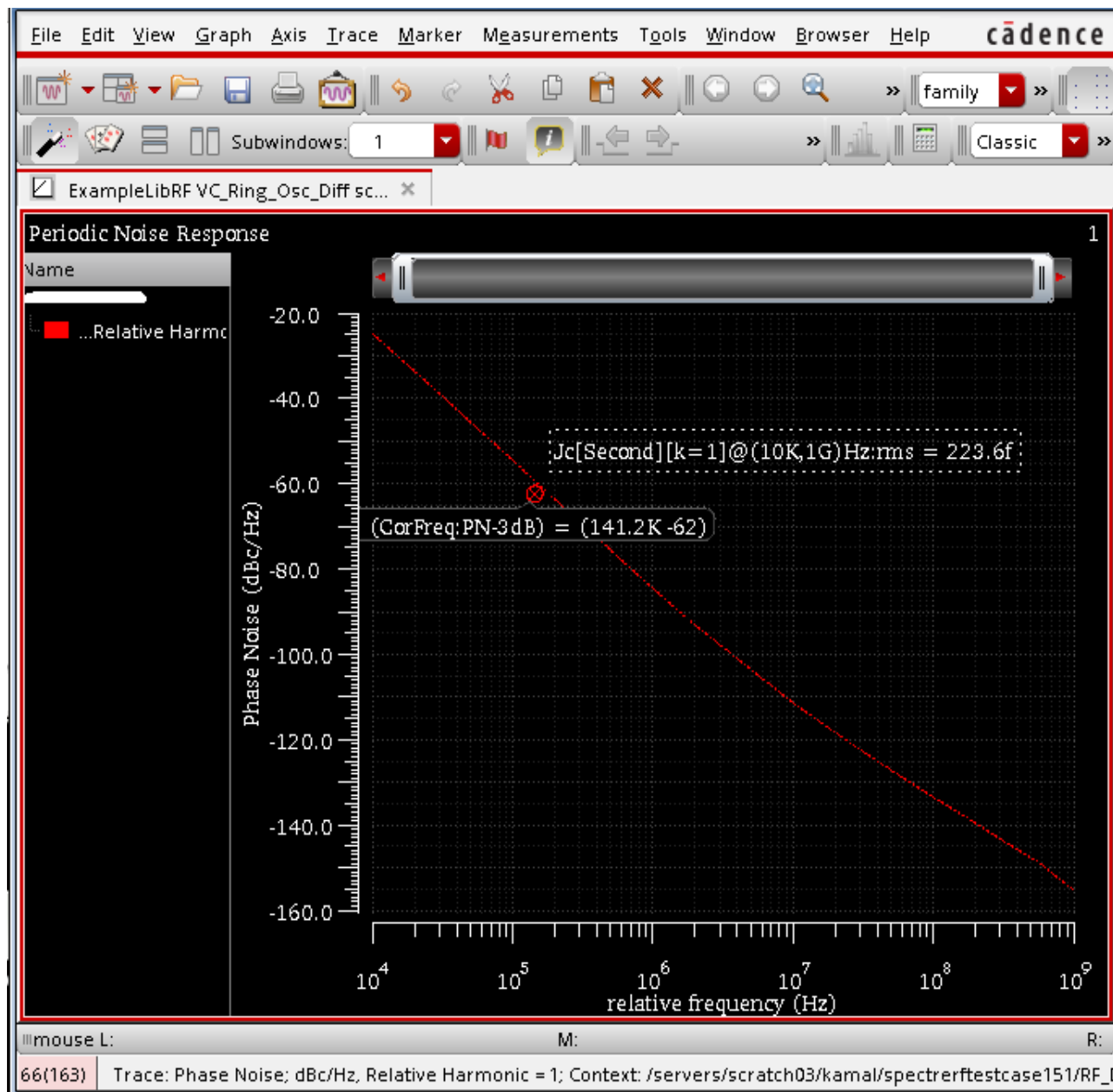
**Figure 3-191 Direct Plot pnoise jitter Jc Setup**

The image shows the 'Direct Plot Form' dialog box in a software application. The dialog is titled 'Direct Plot Form' and has a close button in the top right corner. It contains several sections for configuring a plot:

- Plotting Mode:** A dropdown menu set to 'Append'.
- Analysis:** Two radio buttons: 'pss' and 'pnoise'. The 'pnoise' button is selected and highlighted with a red box.
- Noise Type:** Four radio buttons: 'USB', 'LSB', 'AM', and 'PM'. The 'PM' button is selected and highlighted with a red box.
- Function:** Four radio buttons: 'Output Noise', '-20dB/dec Line', 'Jc', and 'Phase Noise'. The 'Jc' button is selected and highlighted with a red box.
- Number of Cycles [k]:** A text input field containing the value '1'.
- Signal Level:** Two radio buttons: 'rms' and 'peak-to-peak'. The 'rms' button is selected.
- Modifier:** Three radio buttons: 'Second', 'UI', and 'ppm'. The 'Second' button is selected.
- Freq. Multiplier:** A text input field containing the value '1'.
- Integration Limits:** Two text input fields: 'Start Frequency (Hz)' containing '10K' and 'Stop Frequency (Hz)' containing '1G'.
- Buttons:** 'Add To Outputs' (disabled), 'Plot' (active), 'OK' (red), 'Cancel', and 'Help'.
- Footer:** A note that says '> Press plot button on this form...'.

- 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.

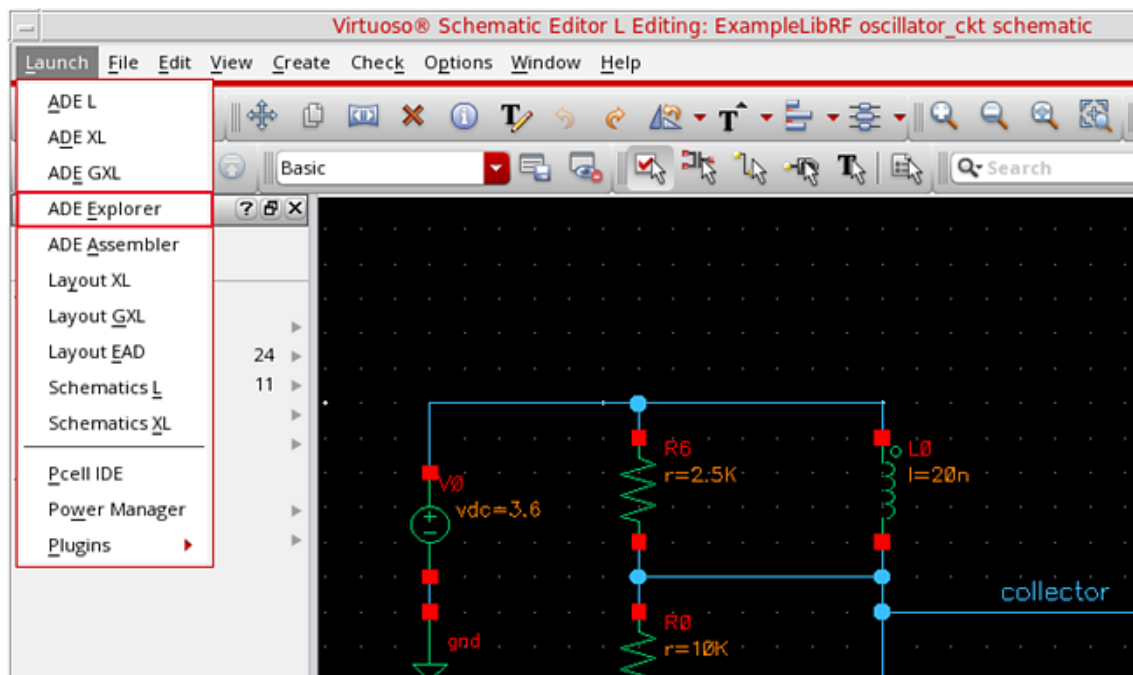
## Calculating the Swept Tuning Range and Phase Noise for the Ring Oscillator

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

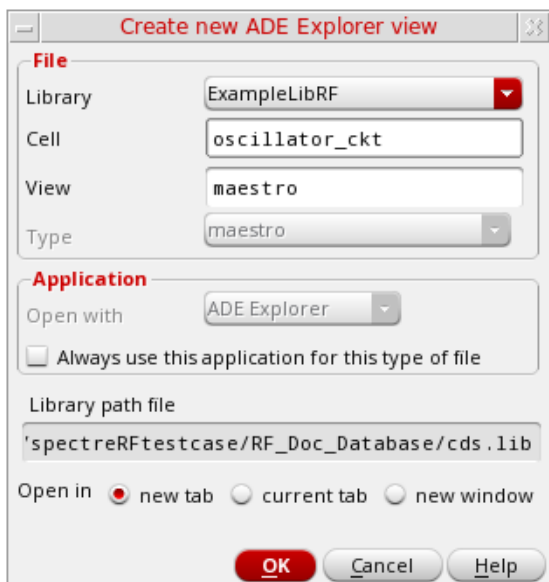


2. In the *Launch ADE Explorer* dialog, select *Create New View*.

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

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

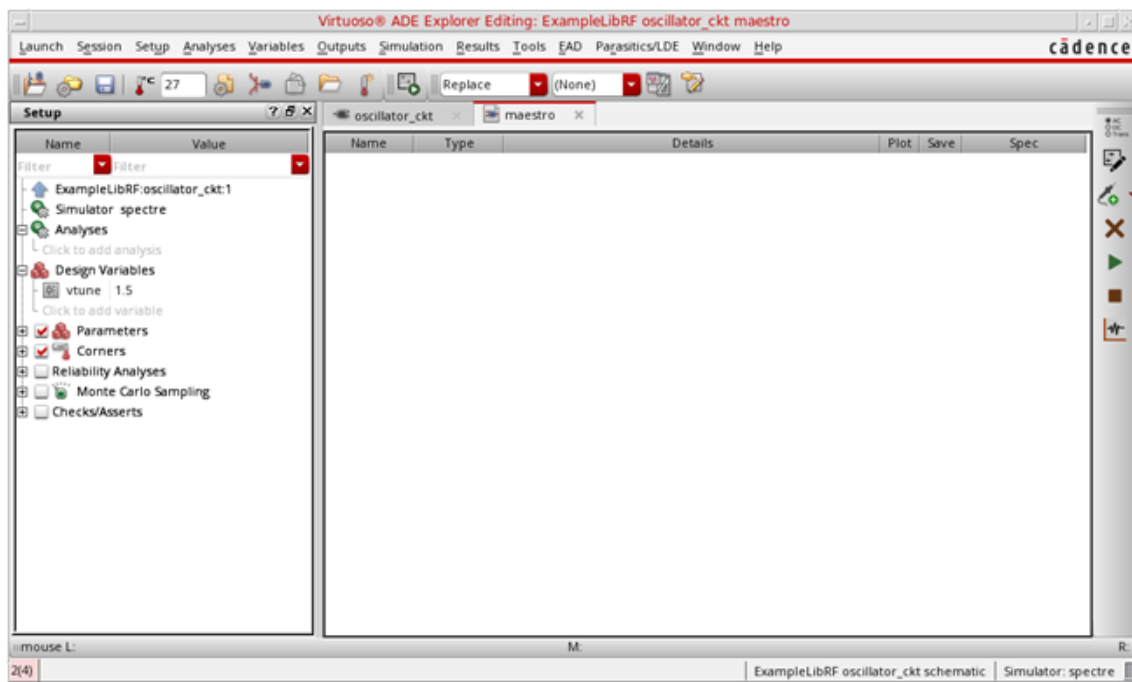
**Figure 3-194 Create new ADE Explorer view**



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.

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

---

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.



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

---

**Figure 3-197 High Performance Simulation Options Form**



The dialog box is titled "High-Performance Simulation Options". It contains two main sections: "General" and "Post-Layout Settings".

**General Section:**

- Simulation Performance Mode:** Radio buttons for Spectre, **APS** (selected), Spectre X, and XPS MS.
- Accuracy + Speed:**
  - Error Preset:** Radio buttons for **Do not override** (selected), Liberal, Moderate, and Conservative.
  - APS Options:**
    - Use ++aps:** A checkbox that is currently unchecked.
  - Multi-Threading:**
    - Radio buttons for **Auto** (selected), Disable, and Manual.
    - # Threads:** A text input field.
    - Processor affinity (e.g. 0-3 or 0,2,4,6):** A text input field.

**Post-Layout Settings Section:**

- Enable Post-Layout Optimization:** A checkbox that is currently unchecked.
- Post-Layout Preset Mode:** Radio buttons for **Default** (selected), High Precision Analog, Ultra Precision Analog, Legacy RCR, and Legacy-RF RCR.
- Instance Preservation:** Radio buttons for **None** (selected) and Selected.
- Selected Instances:** A text input field with **Select** and **Clear** buttons.
- Savefilter:** Checkboxes for **none** (selected) and rc.

**Buttons:** At the bottom right, there are buttons for **OK** (highlighted in red), **Cancel**, **Defaults**, **Apply**, and **Help**.

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.

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

**Figure 3-198 Save Options Form**

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

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.

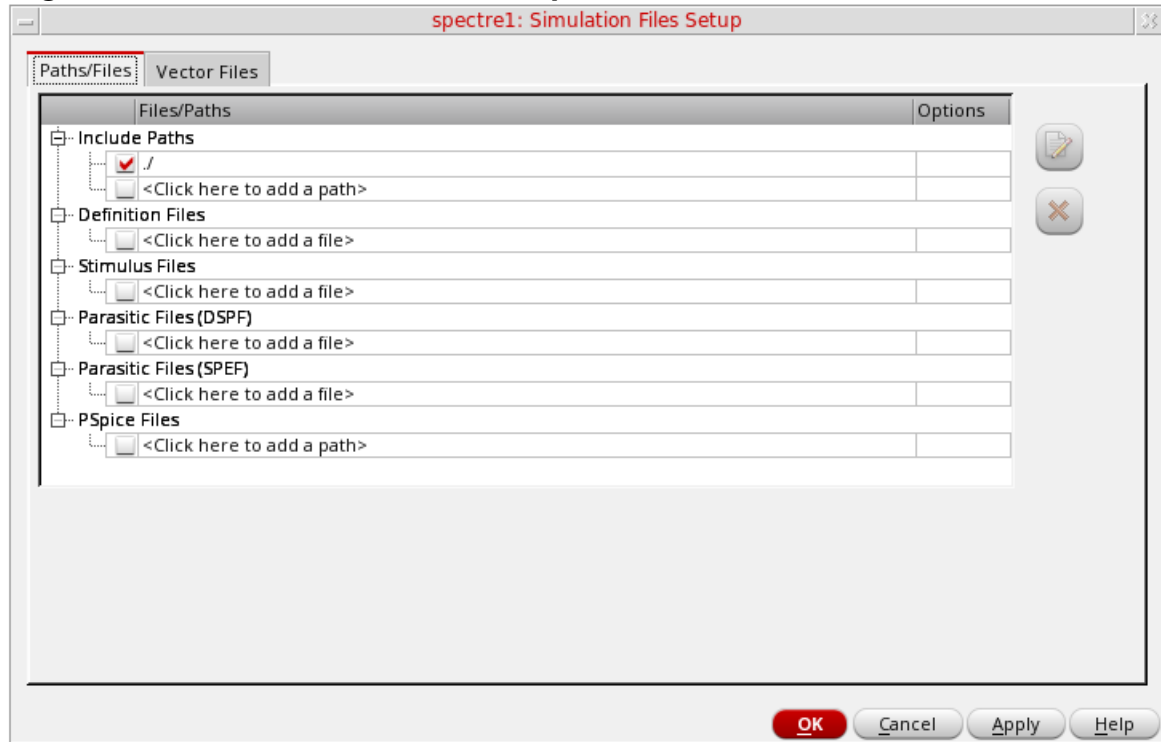
To save the currents, use the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or select *all* if you want to save all the currents in the circuit. When you save currents, you require more disk space for the results file.

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

---

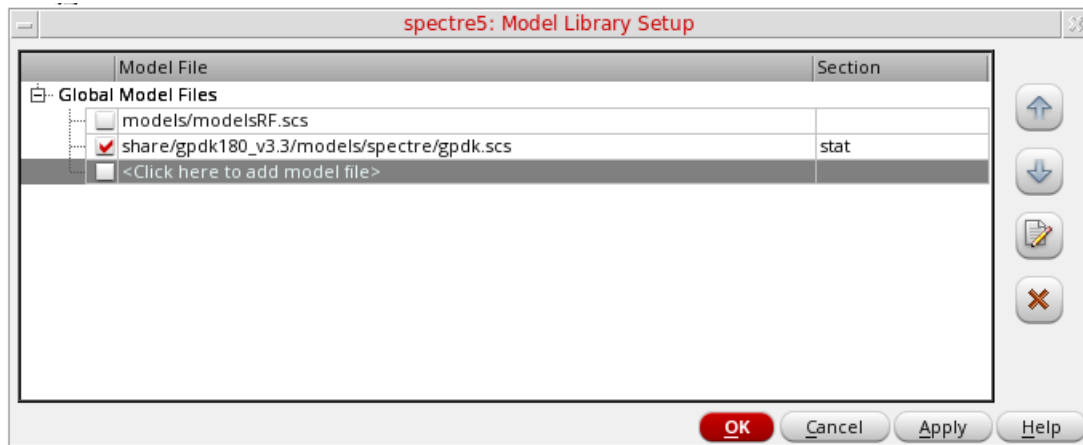
11. Click *OK*.
12. Select *Setup - Simulation Files*.
13. In the *Simulation Files Setup* form which opens, enter . / by clicking in the *Include Paths* section. The *Simulation Files Setup* form should look like the following:

**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**

Choosing Analyses -- ADE Explorer

Analysis

☒ tran   ☐ dc   ☐ ac   ☐ noise

☐ xf   ☐ sens   ☐ dcmatch   ☐ acmatch

☐ stb   ☐ pz   ☐ lf   ☐ sp

☐ envlp   ☐ pss   ☐ pac   ☐ pstb

☐ pnoise   ☐ pxf   ☐ psp   ☐ qpss

☐ qpac   ☐ qpnoise   ☐ qpxf   ☐ qpsp

☐ hb   ☐ hbac   ☐ hbstb   ☐ hbnoise

☐ hbzp   ☐ hbxp

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative   ☐ moderate   ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

OK Cancel Defaults Apply Help

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

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

**Figure 3-202 The Choosing Analyses Form- Setting PSS Analysis**

The screenshot shows the 'Choosing Analyses Form' for setting PSS Analysis. The form is organized into several sections:

- Analysis Selection:** A grid of radio buttons for selecting analysis types. ☒ pss is selected. Other options include envlp, pac, pstb, pnoise, pxf, psp, qpss, qpac, qpnoise, qpxf, qpss, hb, hbac, hbstb, hbnoise, hbasp, and hbxf.
- Periodic Steady State Analysis:** A section with the title 'Periodic Steady State Analysis'.
- Engine:** Radio buttons for 'Shooting' and 'Harmonic Balance'. 'Shooting' is selected.
- Tones:** A table with columns 'Name', 'Expr', 'Value', and 'SrcId'. Below the table are input fields for 'Beat Frequency' and 'Oversample Factor', and an 'Auto Calculate' checkbox.
- Number of Harmonics:** An input field.
- Accuracy Defaults (errpreset):** Radio buttons for 'conservative', 'moderate', and 'liberal'.
- Convergence:** A section with 'Additional Time for Transient-Aided HB (tstab)' input field, 'Save Initial Transient Results (saveinit)' radio buttons (no, yes), and 'Harmonic Balance Homotopy Method' dropdown menu (default).
- Dynamic Parameter:** A checkbox.
- Oscillator:** A checkbox.
- Sweep:** A checkbox.
- New Initial Value For Each Point (restart):** Radio buttons for 'no' and 'yes'.
- Loadpull:** A checkbox.
- Enabled:** A checkbox with a checkmark.
- Buttons:** 'OK', 'Cancel', 'Defaults', 'Apply', 'Help', and 'Options...'.

**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.

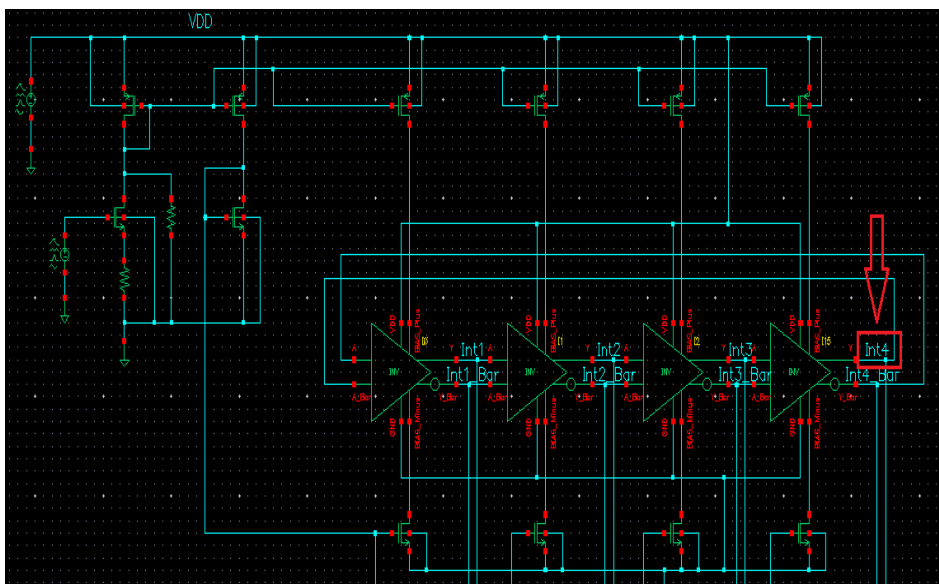
- 27.** Select the *Oscillator* option.

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.

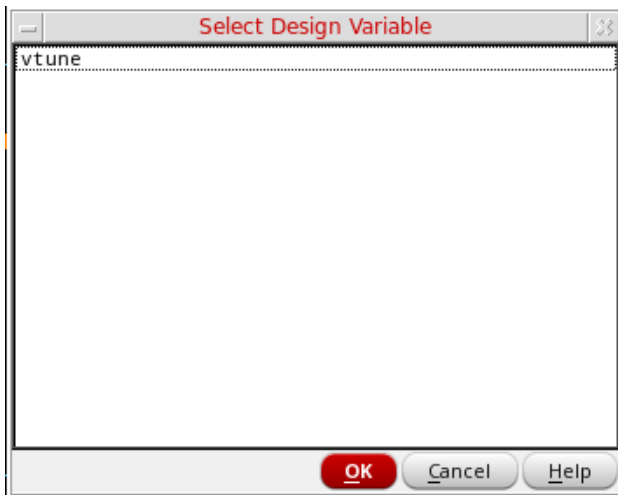
**Figure 3-204 Selecting *Int4* net on schematic**



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.

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

**Figure 3-206 Choosing Analyses Form - PSS-Shooting Method Setup - Part1**

Analysis

- ☐ tran
- ☐ dc
- ☐ ac
- ☐ noise
- ☐ xf
- ☐ sens
- ☐ dcmatch
- ☐ acmatch
- ☐ stb
- ☐ pz
- ☐ sp
- ☐ envlp
- ☒ pss
- ☐ pac
- ☐ pstb
- ☐ pnoise
- ☐ pxf
- ☐ psp
- ☐ qpss
- ☐ qpac
- ☐ qpnoise
- ☐ qpxf
- ☐ qpdp
- ☐ hb
- ☐ hbac
- ☐ hbnoise
- ☐ hbdp

Periodic Steady State Analysis

Engine

- ☒ Shooting
- ☐ Harmonic Balance

Fundamental Tones

| # | Name | Expr | Value | Signal | SrcId |
|---|------|------|-------|--------|-------|
|---|------|------|-------|--------|-------|

Large

Clear/Add Delete Update From Hierarchy

Beat Frequency ☐ Beat Period

16 Auto Calculate ☐

Output harmonics

Number of harmonics 28

Accuracy Defaults (errpreset)

- ☒ conservative
- ☐ moderate
- ☐ liberal

Transient-Aided Options

Run transient? ☒ Yes ☐ No ☐ Decide automatically

Detect Steady State ☐ Stop Time (tstab) 10n

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter ☐

Oscillator ☒

Oscillator node+ /Int4 Select

Oscillator node- Select

☐ Calculate initial conditions (ic) automatically

☐ Enable tuning mode analysis

Sweep 1 Variable

Frequency Variable? ☒ no ☐ yes

Variable Name vtune Select Design Variable

Sweep Range

Start-Stop Start 0.0 Stop 1.0

### 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**

The screenshot shows the 'Choosing Analyses' dialog box with the 'pnoise' option selected and highlighted by a red rectangle. The dialog is divided into several sections:

- Analysis Selection:** A grid of radio buttons for various analyses. 'pnoise' is selected and highlighted.
- Periodic Noise Analysis:**
  - PSS Beat Frequency (Hz): 1.9G
  - Multiple pnoise: ☐
  - Sweep type: default (dropdown)
  - Relative Harmonic:
  - Output Frequency Sweep Range (Hz):
    - Start-Stop:  Start:  Stop:
  - Sweep Type: Automatic (dropdown)
  - Add Specific Points: ☐
- Sidebands:**
  - Method: default (radio), fullspectrum (radio)
  - Maximum sideband:  (dropdown)
  - When using shooting engine, default value is 7.
- Output:**
  - probe (dropdown)
  - Output Probe Instance:  Select
- Noise Type:**
  - timeaverage (dropdown)
  - Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis
  - Contribution Type:
    - USB (radio), AM (radio), PM (radio), AM&PM (radio), ALL(AM,PM,USB,LSB) (radio)
    - For fmjitter, PM noise must be enabled
  - Noise Separation: ☐ yes ☐ no
  - Separate noise into source and gain
- Buttons:** Enabled ☐ Options... OK Cancel Defaults Apply Help

### 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.

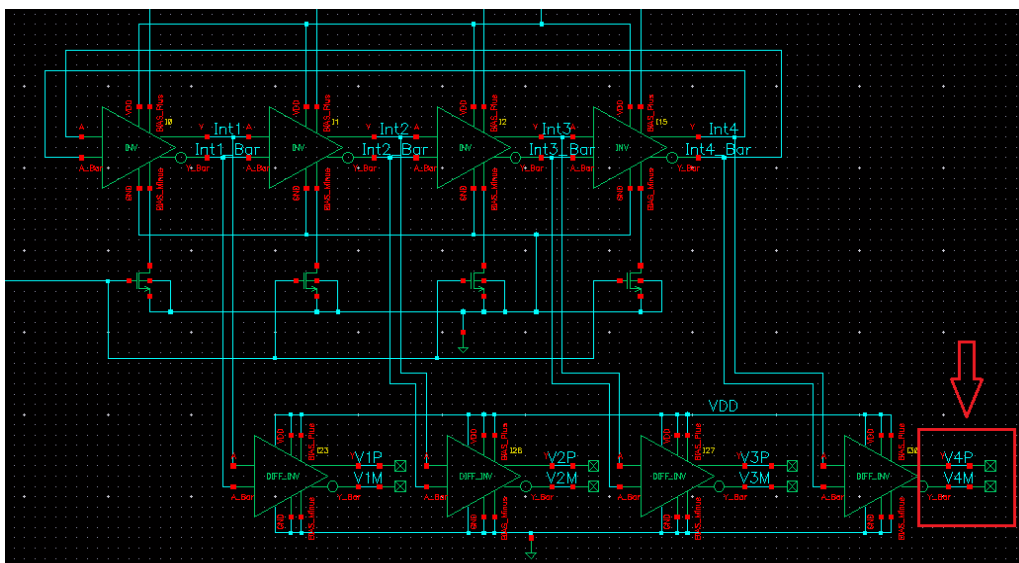
**Note:** When the default method is selected, specify the number of desired frequency translations in the *Maximum sideband* field. This will always be specific to the design.

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

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**

☐ stb   ☐ pz   ☐ sp   ☐ envlp  
☐ pss   ☐ pac   ☐ pstb   ☒ pnoise  
☐ pxf   ☐ psp   ☐ qpss   ☐ qpac  
☐ qpnoise   ☐ qpxf   ☐ qpss   ☐ hb  
☐ hbac   ☐ hbnoise   ☐ hbss

Periodic Noise Analysis

PSS Beat Frequency (Hz)

Multiple pnoise ☐

SweepType  Relative Harmonic

Output Frequency Sweep Range (Hz)

Freq

Add Specific Points ☐

Sidebands

Method ☐ default ☒ fullspectrum

Maximum sideband

Calculates noise contributions up to the frequency determined by PSS time point resolution

Output

Positive Output Node

Negative Output Node

Noise Type

Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis

Contribution Type:

☐ USB ☐ AM ☐ PM ☐ AM&PM ☒ ALL(AM,PM,USB,LSB)

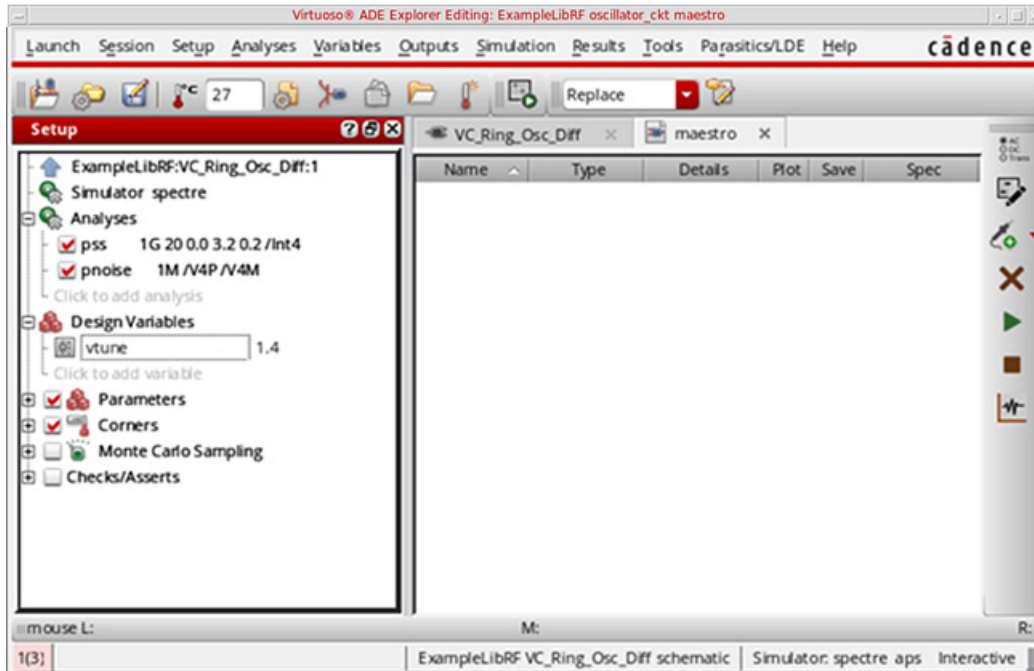
For fmjitter, PM noise must be enabled

Enabled ☒

- 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.

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

**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**

**Direct Plot Form**

Plotting Mode: **Append**

**Analysis**

☒ pss ☐ pnoise ☐ tstab

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Harmonic Frequency ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp. ☐ THD

Select: **Net**

**Sweep**

☒ spectrum ☐ variable ☐ freq  
☐ time

Signal Level: ☒ peak ☐ rms

**Modifier**

☒ Magnitude ☐ Phase ☐ dB20  
☐ Real ☐ Imaginary

Variable Value (vtune)

0  
200m  
400m  
600m  
800m  
1

Add To Outputs ☐

> Select Variable Value (vtune) on this form...

**OK** **Cancel** **Help**

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.



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

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**

**Direct Plot Form**

Plotting Mode: **Append**

**Analysis**

☒ pss ☐ pnoise ☐ tstab

**Function**

☐ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☒ **Harmonic Frequency** ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp. ☐ THD

**Harmonic Frequency**

|   | 0        | 0     |
|---|----------|-------|
| 1 | 335.348M | - 2.3 |
| 2 | 670.697M | - 4.1 |
| 3 | 1.00605G | - 7.0 |
| 4 | 1.34139G | - 9.4 |

Loadpull Contour: ☐

Add To Outputs ☐ **Plot**

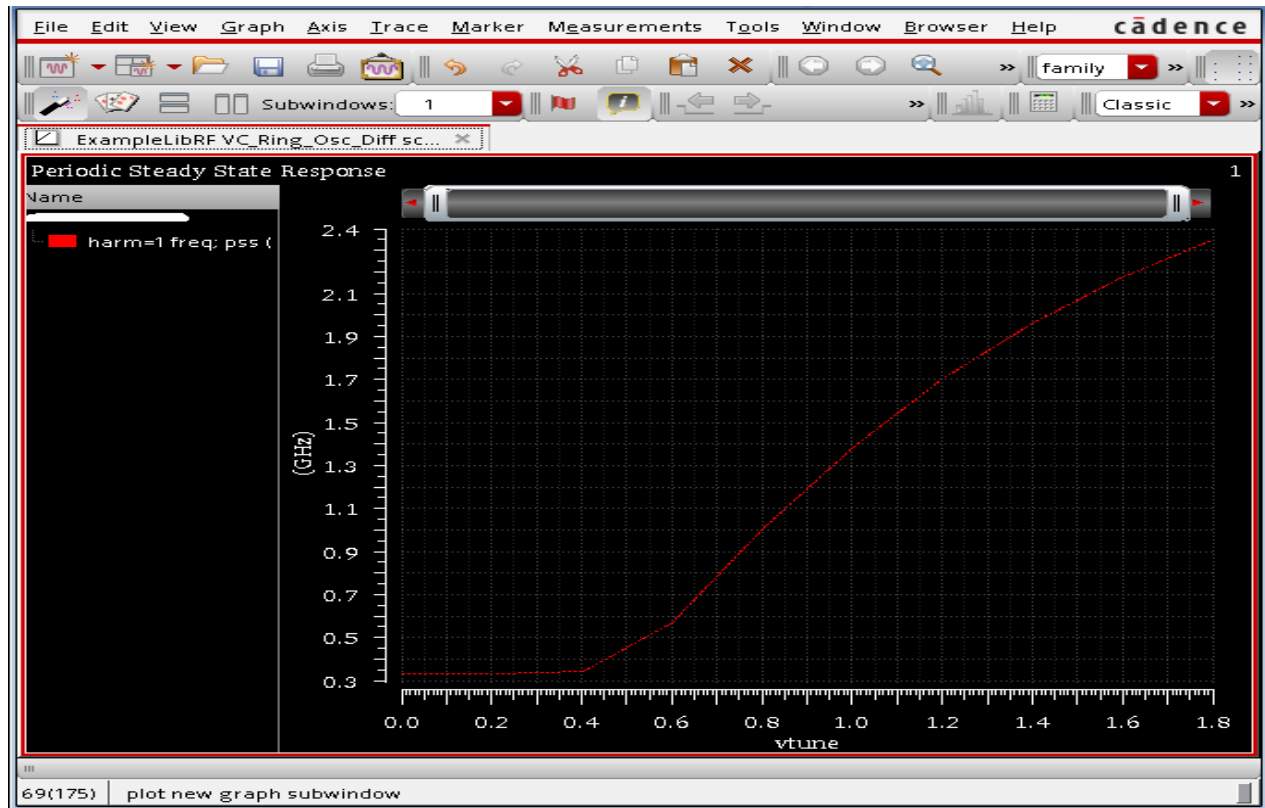
> Press plot button on this form...

**OK** **Cancel** **Help**

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**

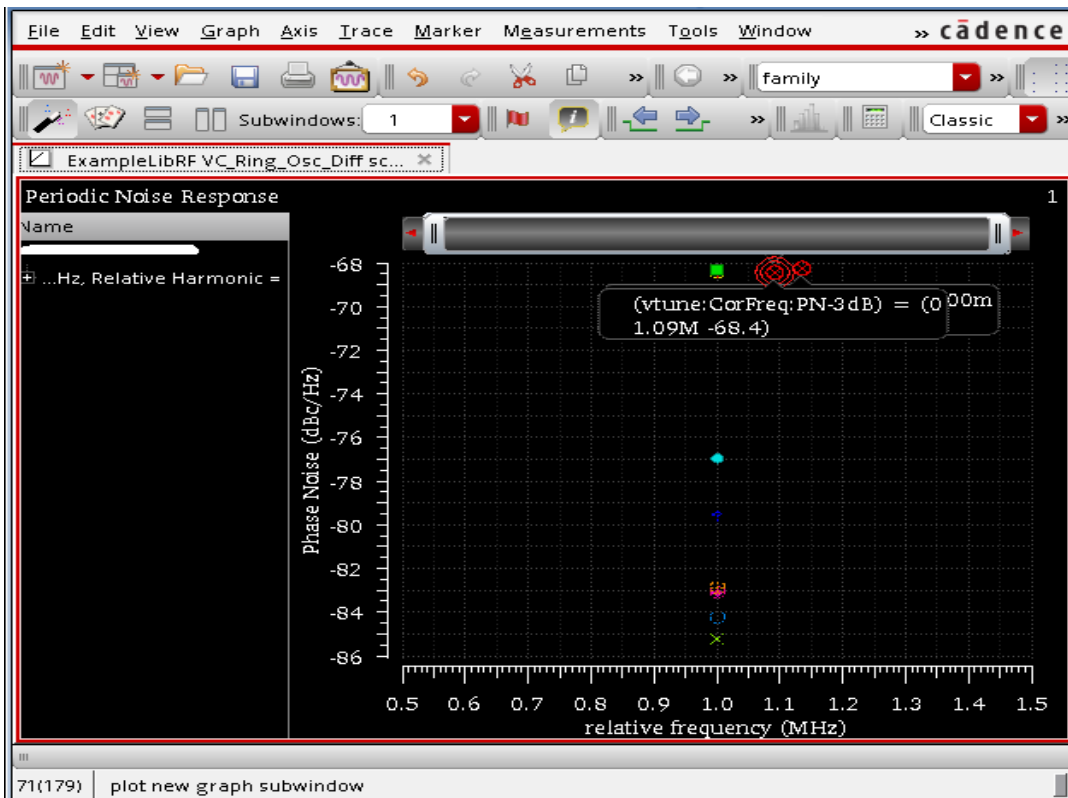
The image shows a software dialog box titled "Direct Plot Form". It contains several sections for configuring a plot:

- Plotting Mode:** A dropdown menu set to "Append".
- Analysis:** Three radio buttons: "pss", "pnoise" (which is selected), and "tstab".
- Noise Type:** Four radio buttons: "USB", "LSB", "AM", and "PM" (which is selected).
- Function:** Six radio buttons arranged in two rows: "Output Noise", "-20dB/dec Line", "Jc", "Jcc", "Phase Noise" (which is selected), and an unlabeled one.
- Text:** "Currently, only variable data is available".
- Buttons:** "Add To Outputs" (disabled), "Plot" (disabled), "OK" (highlighted in red), "Cancel", and "Help".
- Footer:** "> Press plot button on this form..."

**4. Click *Plot*.**

This will plot the Single Sideband Phase Noise vs. Relative Frequency graph, as shown below:

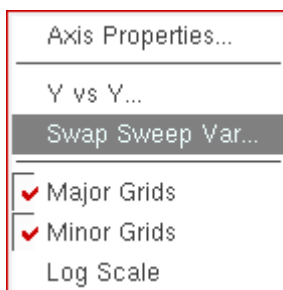
**Figure 3-215 Phase Noise vs. Relative Frequency Plot**



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.

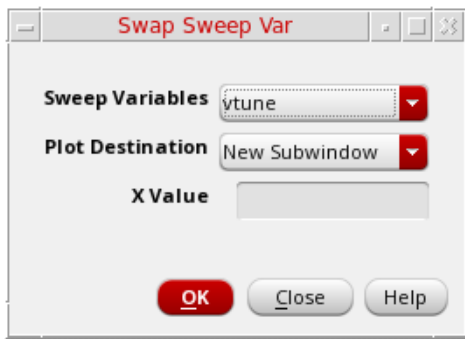
1. In the plot window, right-click on one of the numbers on the X Axis and select *Swap Sweep Var* from the context menu.

**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*.

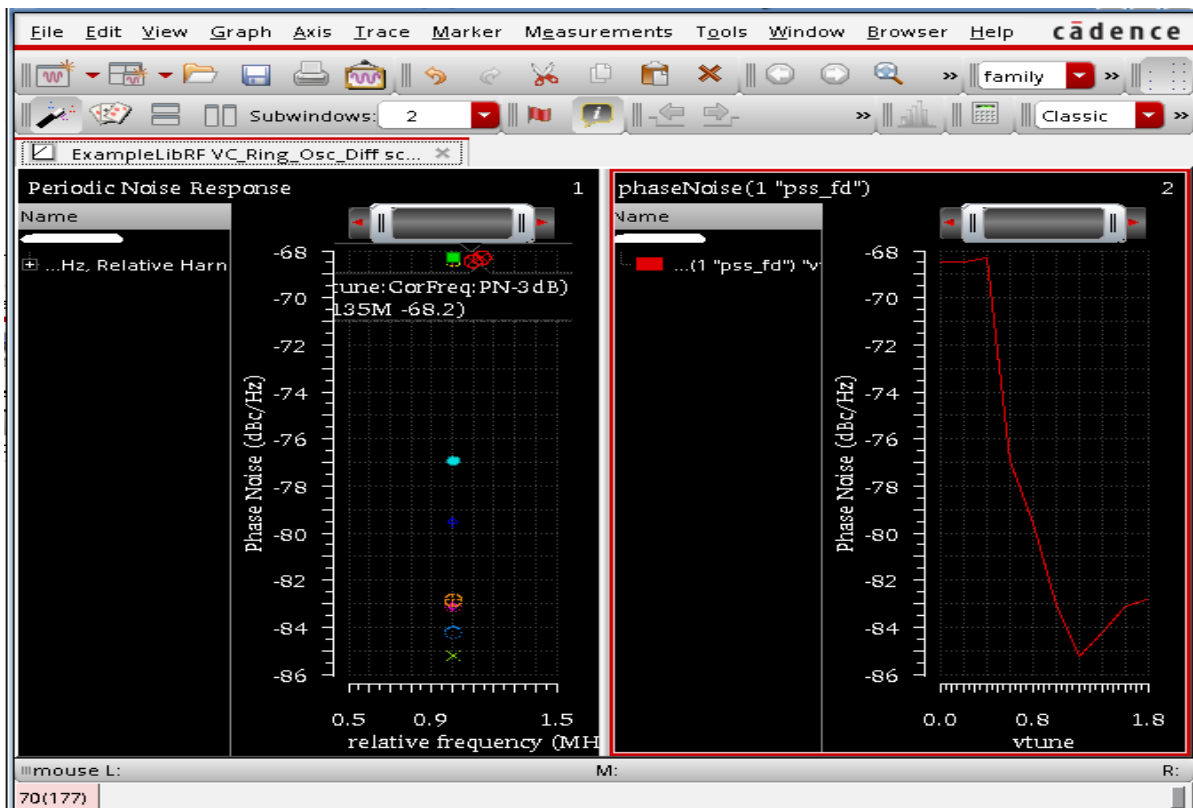
Figure 3-217 Swap Sweep Var Dialog Box Window



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.

---

## 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 ( <code>db_mixer</code> )          | Analyses              |
|---------------------------------------------------------------|-----------------------|
| <u>Mixer Conversion Gain and RF to IF Isolation</u>           | HB and HBAC           |
| <u>LO Leakage (LO to IF Leakage)</u>                          | HB                    |
| <a href="#">Noise Figure Measurement, Noise Summary Table</a> | HB and HBnoise        |
| <u>1dB Compression Point, Desensitization and Blocking</u>    | HB, HBAC, and HBnoise |
| <u>Third-Order Intercept measurement with HB</u>              | HB                    |
| <u>Rapid IP2, Rapid IP3</u>                                   | Specialized HBAC      |
| <u>Compression Distortion Summary</u>                         | HB, HBAC              |
| Transmit Mixer Measurements ( <code>db_mixer_xmit</code> )    | Analyses              |
| <u>Image Rejection</u>                                        | HB                    |
| <u>Three tone Swept IP3 (large signal)</u>                    | HB                    |
| <a href="#">Noise Figure</a>                                  | 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](#)*.

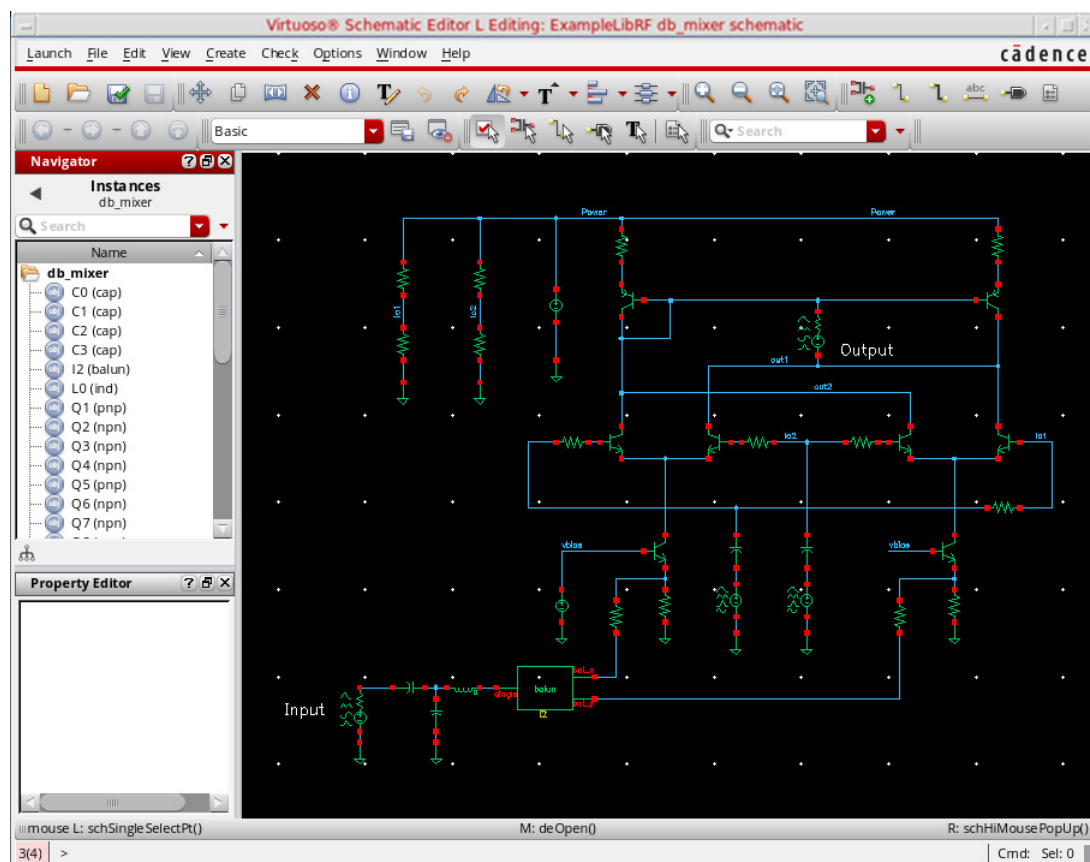
### The db\_mixer and db\_mixer\_xmit Mixer Circuits

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.



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

---

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                | <i>measurement needed</i> |
| Double Sideband Noise figure   | <i>measurement needed</i> |
| Input 1dB compression point    | <i>measurement needed</i> |
| Input IP3 (from swept power)   | <i>measurement needed</i> |
| Input IP3 (from HBAC analysis) | <i>measurement needed</i> |

| 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 [Introduction](#) chapter.)

### Opening the db\_mixer Mixer Circuit in the Schematic Window

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

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

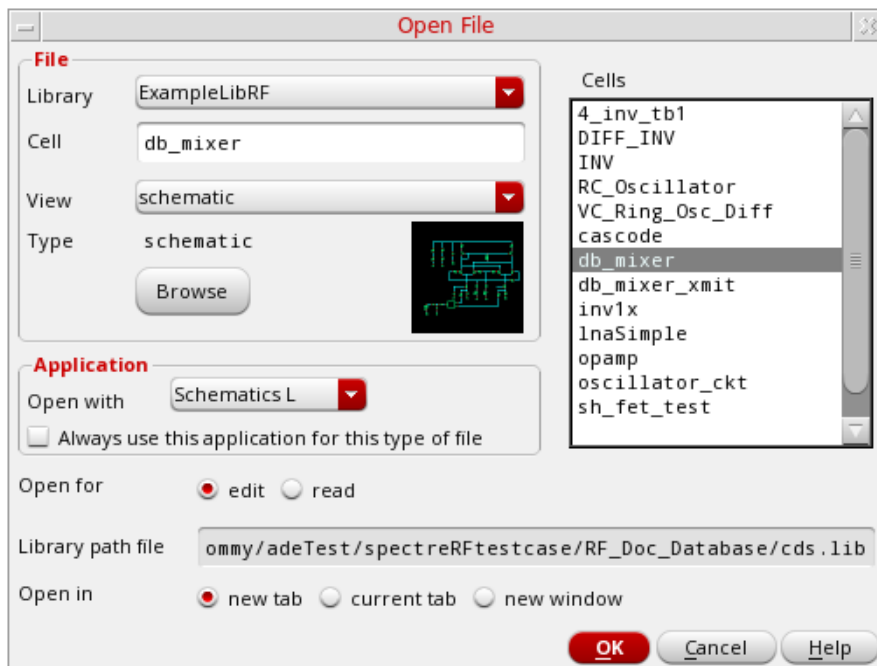
---

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**

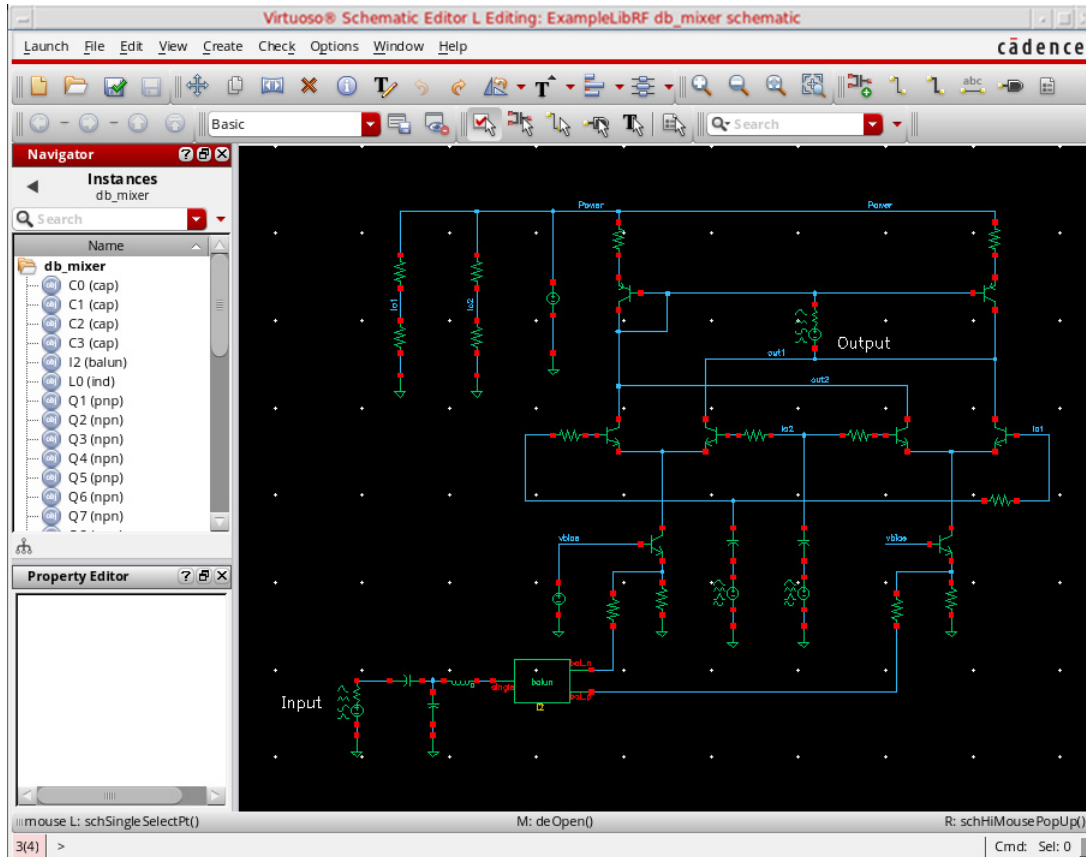


4. Click *OK*.

The Schematic window for *db\_mixer* mixer is displayed.

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

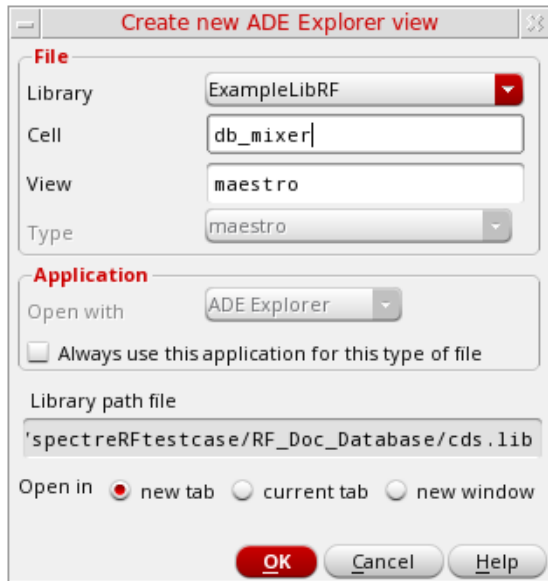
Figure 4-3 db\_mixer Schematic



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.

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

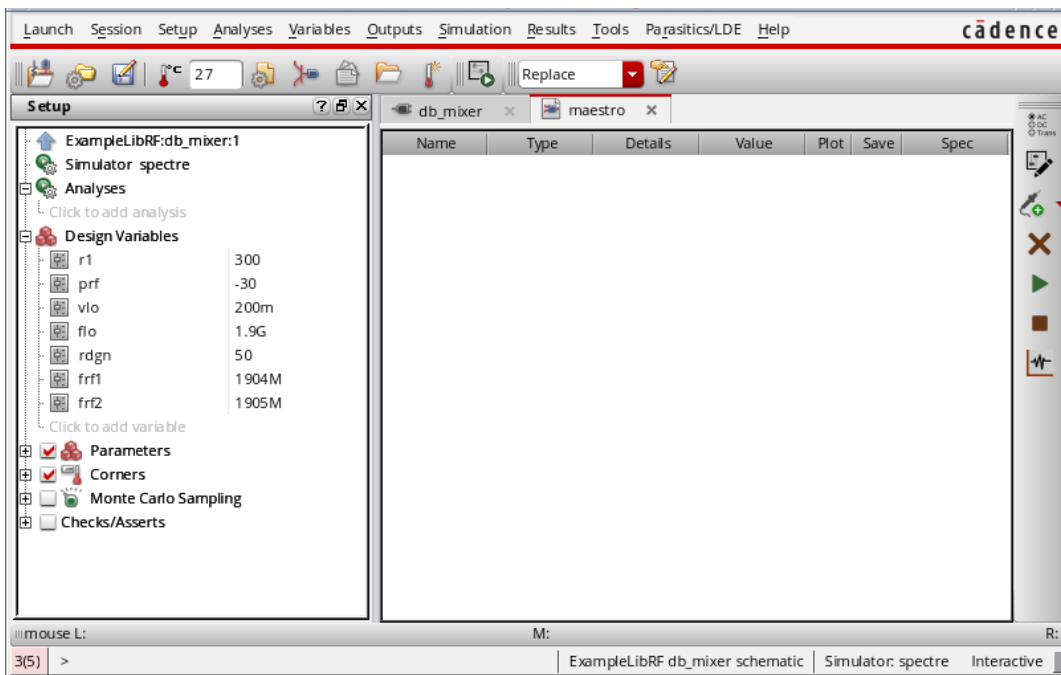
Figure 4-4 Create new ADE Explorer view



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*.

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

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

**Figure 4-8 Save Options Form**

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lv/pub ☐ lv ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlv)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

6. In the *Select signals to output* section, make sure that *allpub* is selected.

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

---

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.

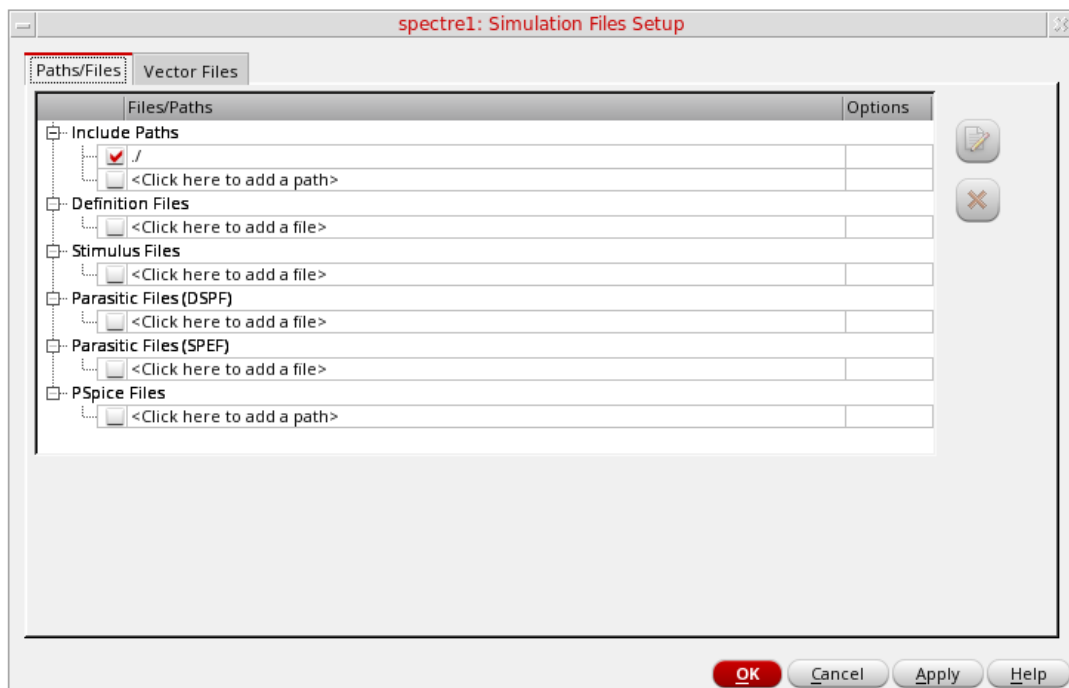
To save the currents, use the *Select device currents (currents)* option, and select *nonlinear* if you just want to save the device currents, or *all* if you want to save all the currents in the circuit.

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:

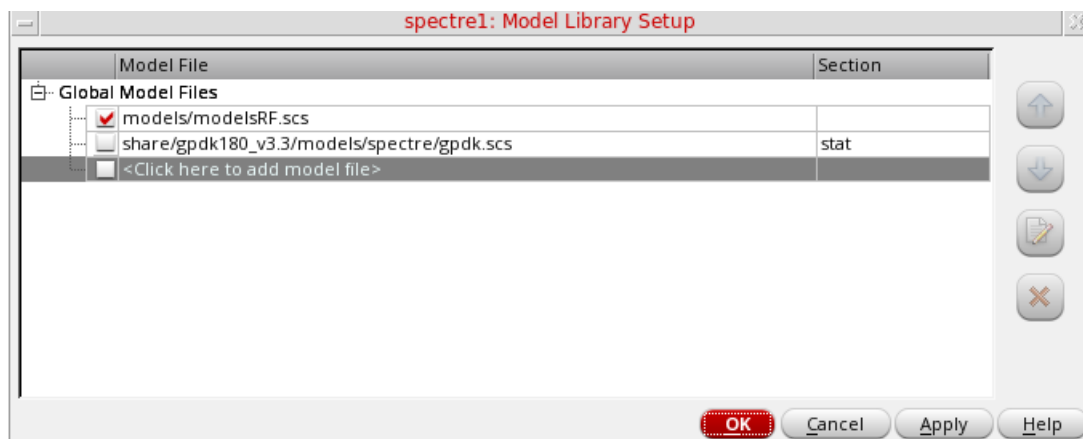
```
models/modelsRF.scs
```



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.

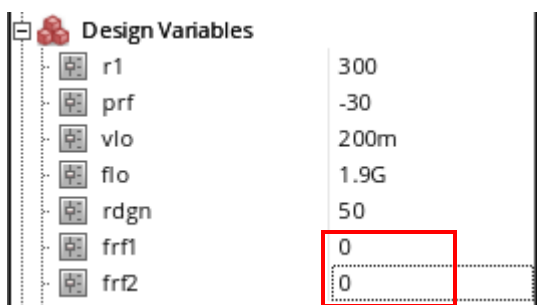
Hbac and hbnoise are both small-signal analyses. Both are nonlinear in the sense of being able to take into account the frequency conversions created by the LO. Noise is almost certainly a small-signal problem. In practice, hbac calculates the correct conversion gain for the cases where the RF input 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.

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

---

**Figure 4-12 The hb Choosing Analyses Form**

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☒ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbasp ☐ hbxf

Harmonic Balance Analysis

Transient-Aided Options

Run transient? Decide automatically

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results(saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

Tone 1

Fundamental Frequency

Number of Harmonics auto

Oversample Factor 1

Freqdivide Ratio for Tone 1 1

Harmonics Default

Accuracy Defaults (errpreset)

☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☐

Options...

OK Cancel Defaults Apply Help

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.

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

---

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:

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

Figure 4-13 hb Choosing Analyses Form

**Choosing Analyses -- ADE Explorer**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☒ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbisp ☐ hbxf

**Harmonic Balance Analysis**

Transient-Aided Options

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) **auto**

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☒ **Frequencies** ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

Tone 1

Fundamental Frequency **1.9G**

Number of Harmonics **auto**

Oversample Factor **1**

Freqdivide Ratio for Tone 1 **1**

Harmonics **Default**

Accuracy Defaults (errpreset)

☐ conservative ☒ **moderate** ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☒

**OK** Cancel Defaults Apply Help

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

---

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**

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☐ hb ☒ hbac ☐ hbstb ☐ hbnoise

☐ hbasp ☐ hbxf

Harmonic Balance AC Analysis

Sweeptype **default** Sweep is currently absolute

Input Frequency Sweep Range (Hz)

**Start-Stop** Start  Stop

Sweep Type

**Automatic**

Add Specific Points ☐

Add Points By File ☐

Sidebands

**Maximum sideband**

When using hb engine, default value is harms of 1st tone.

Specialized Analyses

**None**

Enabled ☐

Options...

**OK** Cancel Defaults Apply Help

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.



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

---

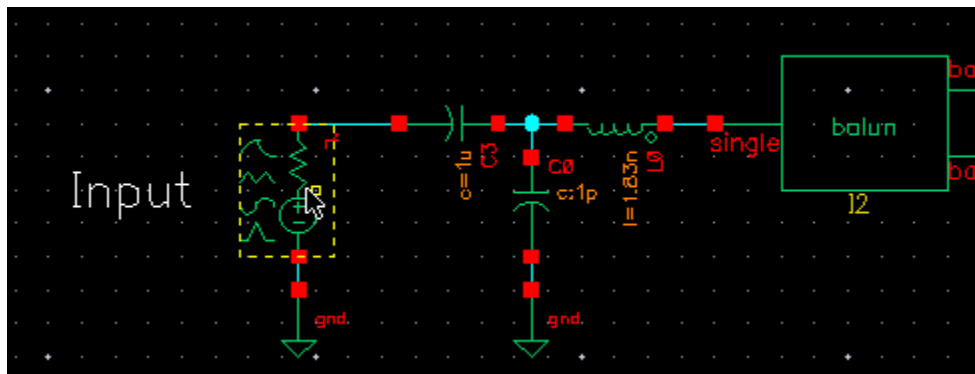
The frequency sweep range is always the input frequency range in hbac. Because hbac has the ability to calculate the outputs at different frequencies based on the nonlinearity caused by the LO, we choose which output frequencies to calculate in the Sidebands area of the Choosing Analyses form. Sideband 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

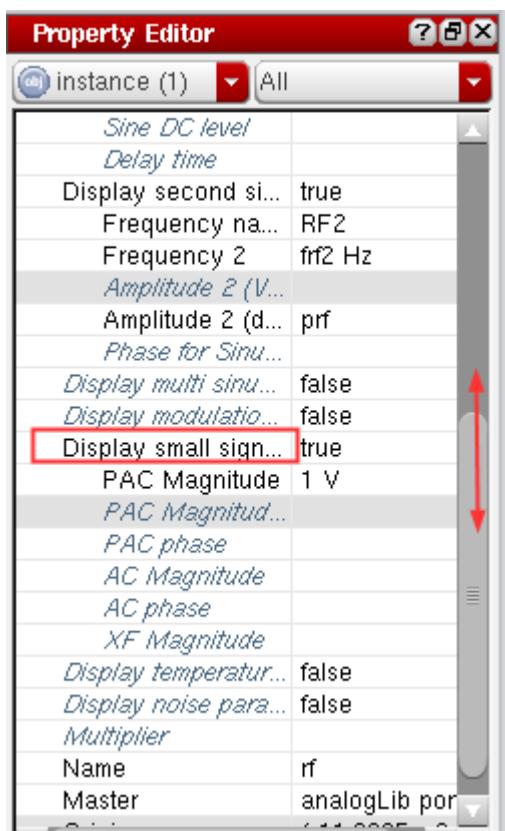
- a. To set the amplitude for hbac, select the *Input* port in the schematic.

**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 hba *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 ( $1.9001\text{G} \sim 2\text{G} - 1 \times 1.9\text{G} = 100\text{K} \sim 100\text{M}$ ) is selected. In addition, hba also calculates the outputs with no frequency translation ( $1.9001\text{G} \sim 2\text{G} - 0 \times 1.9\text{G} = 1.9001\text{G} \sim 2\text{G}$ ). You will be choosing these two sidebands.

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

---

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**

**Choosing Analyses -- ADE Explorer**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☐ hb ☒ hbac ☐ hbstb ☐ hbnoise

☐ hbasp ☐ hbxf

Harmonic Balance AC Analysis

Sweeptype **default** Sweep is currently absolute

Input Frequency Sweep Range (Hz)

Start-Stop **Start** 1.9001G **Stop** 2G

Sweep Type

**Linear** ☐ Step Size 10

☒ Number of Steps

Add Specific Points ☐

Add Points By File ☐

Sidebands **Select from range** From (Hz) 0 To (Hz) 1e12 Max. Order 1

| side | Frequencies | 1.9G |
|------|-------------|------|
| 1    | 100K        | 100M |
| u    | 1.9001G     | 2G   |
| u    | 3.8001G     | 3.9G |

Specialized Analyses

**None**

Enabled ☒

Options...

**OK** Cancel Defaults Apply Help

6. Click *Apply*.

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

---

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**

**Choosing Analyses -- ADE Explorer**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☐ hb ☐ hbac ☐ hbstb ☒ hbnoise

☐ hbsp ☐ hbxf

Harmonic Balance Noise Analysis

Multiple hbnoise ☐

Sweeptype **default** Sweep is currently absolute

Output Frequency Sweep Range (Hz)

Start-Stop **Start**  **Stop**

Sweep Type

**Automatic**

Add Specific Points ☐

Add Points By File ☐

Sidebands

**Maximum sideband**

When using hb engine, default value is harms of 1st tone.

Noise Figure ☐

Output

**probe** Output Probe Instance  **Select**

Noise Type **timeaverage**

Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis

Contribution Type:

☒ USB ☐ AM ☐ PM ☐ AM&PM ☐ ALL(AM,PM,USB,LSB)

Noise Separation ☐

Separate noise into source and gain

Enabled ☐

**OK** **Cancel** **Defaults** **Apply** **Help**

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= $k$ ,

$$|f_{in}| = |f_{out}| + k * \text{fundamental frequency}.$$

Here,  $f_{out}=1K\sim 100M$  and the fundamental frequency is 1.9G.

$f_{in}$  should be  $(1K\sim 100M) + 1.9G$ ,

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 *Sweep type* is set as *relative*. For *db\_mixer*, the *Output Frequency Sweep Range* is set as  $f_{out}=1K\sim 100M$ , 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.

The image shows a dialog box titled "hbnoise Noise Separation Check Box". It contains the following elements:

- Do Noise:** A checked checkbox.
- Noise Type:** A dropdown menu set to "timeaverage".
- Timeaverage:** A text label describing the noise type: "single-sided spectrum and harmonic-referred (modulated) noise analysis".
- Contribution Type:** A group of radio buttons with the following options: USB, AM, PM, AM&PM, and ALL (AM,PM,USB,LSB). The "ALL" option is selected.
- Noise Separation:** A group of checkboxes with the options "yes" (checked) and "no". This entire group is highlighted with a red rectangular box.
- Separate noise into source and gain:** A text label at the bottom of the dialog.

Your completed hbnoise *Choosing Analyses* form should look like the following figure.



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

Figure 4-20 hbnoise Choosing Analyses Form

Choosing Analyses -- ADE Explorer

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpsp  
☐ hb ☐ hbac ☐ hbstb ☒ hbnoise  
☐ hbsp ☐ hbxf

Harmonic Balance Noise Analysis

Multiple hbnoise ☐

Sweep type: relative Relative Harmonic: 0

Output Frequency Sweep Range (Hz)

Start-Stop Start: 1k Stop: 100M

Sweep Type: Logarithmic Points Per Decade: 3  
☐ Number of Steps

Add Specific Points ☐  
Add Points By File ☐

Sidebands: Maximum sideband

When using hb engine, default value is harms of 1st tone.

Noise Figure ☒

Output: probe Output Probe Instance: /rif Select

Input Source: port Input Port Source: /rf Select

Reference Side-Band:  $|f(in)| = |f(out) + \text{refsideband freq shift}|$   
Select from list

From (Hz): 0 To (Hz): 1e12 Max. Order: 1

| side | Frequencies    | 1.9G |
|------|----------------|------|
| 1    | 1K 100M        | -1   |
| u    | 1.900001G 2G   | 0    |
| u    | 3.800001G 3.9G | 1    |

Noise Type: timeaverage  
Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis  
Contribution Type: ☐ USB ☐ AM ☐ PM ☐ AM&PM ☒ ALL(AM,PM,USB,LSB)  
Noise Separation ☒  
Separate noise into source and gain

Enabled ☒

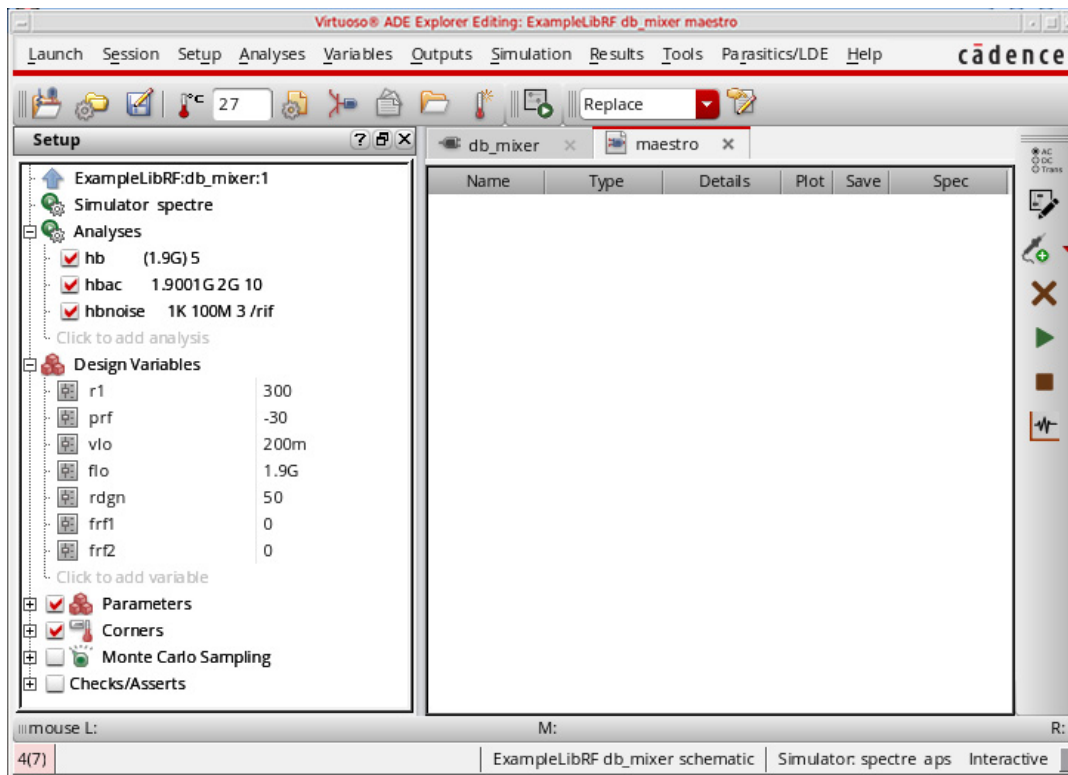
Options...

OK Cancel Defaults Apply Help

18. Click **OK** at the bottom of the form.

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

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*.

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

---

The *Direct Plot Form* is displayed.

**Figure 4-22 hb Direct Plot Form**

The **Direct Plot Form** dialog box is shown with the following settings:

- Plotting Mode:** Append
- Analysis:**
  - ☒ hb
  - ☐ hbac
  - ☐ hbnoise
  - ☐ hbnoise separation
- Function:**
  - ☒ Voltage
  - ☐ Current
  - ☐ Power
  - ☐ Voltage Gain
  - ☐ Current Gain
  - ☐ Power Gain
  - ☐ Transconductance
  - ☐ Transimpedance
  - ☐ Compression Point
  - ☐ IPN Curves
  - ☐ Power Contours
  - ☐ Reflection Contours
  - ☐ Harmonic Frequency
  - ☐ Power Added Eff.
  - ☐ Power Gain Vs Pout
  - ☐ Comp. Vs Pout
  - ☐ Node Complex Imp.
  - ☐ THD
- Select:** Net
- Sweep:**
  - ☒ spectrum
  - ☐ time
- Signal Level:**
  - ☐ peak
  - ☒ rms
- Modifier:**
  - ☒ Magnitude
  - ☐ Phase
  - ☐ dB20
  - ☐ Real
  - ☐ Imaginary
- Scalar Expression:**
  - Value At (harms):
- Add To Outputs:** ☐
- > Select Net on schematic...**
- Buttons:** Close, Help

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* section, select *Voltage*.
3. *SelectNet* 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**

**Direct Plot Form**

Plotting Mode: **Append**

**Analysis**

☒ hb ☐ hbac  
☐ hbnoise ☐ hbnoise separation

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Harmonic Frequency ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp. ☐ THD

Select: **Net**

**Sweep**

☒ spectrum ☐ time

Signal Level: ☐ peak ☒ rms

**Modifier**

☐ Magnitude ☐ Phase ☒ dB20  
☐ Real ☐ Imaginary

**Scalar Expression**

Value At (harms):

Add To Outputs: ☐

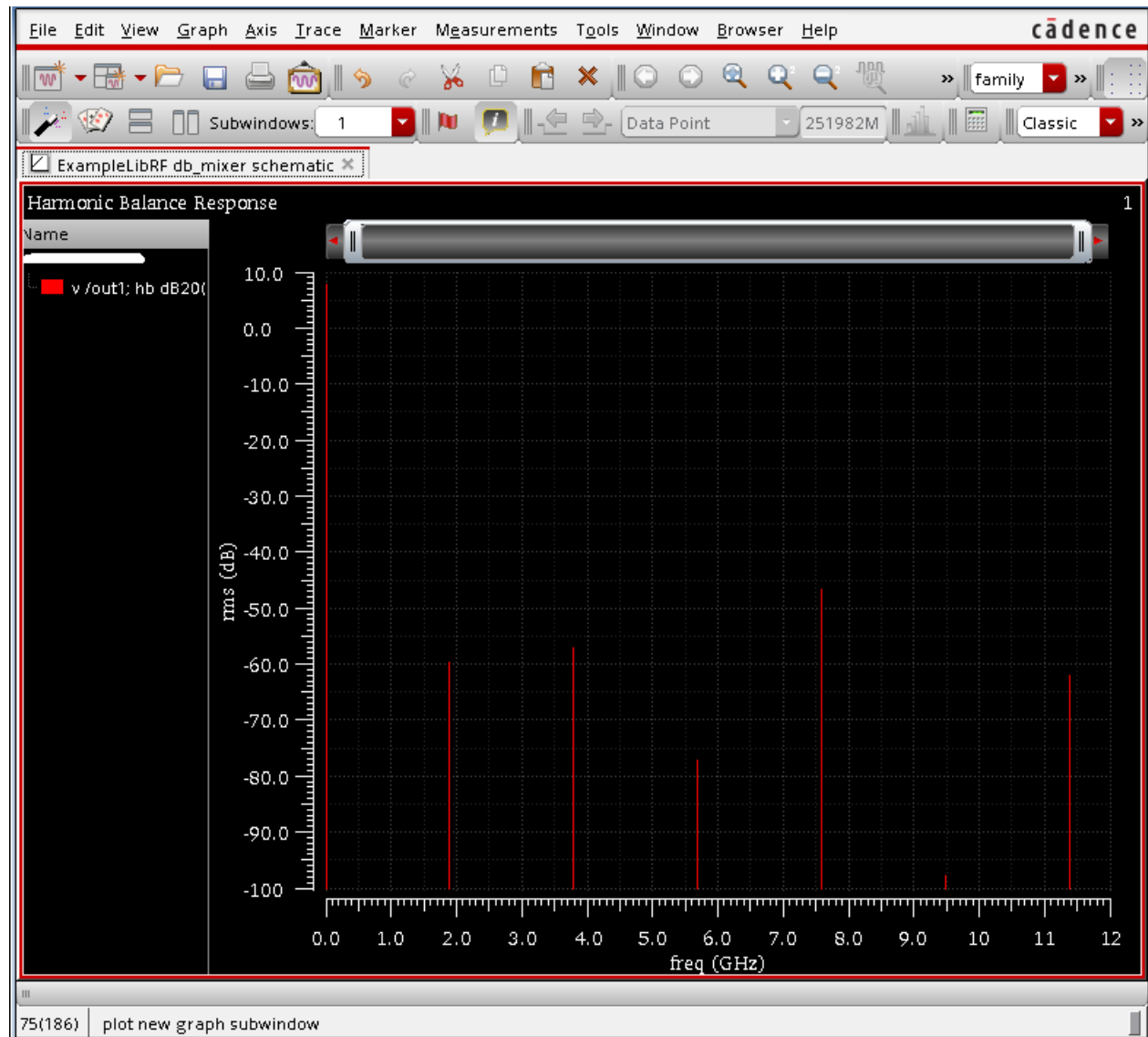
> Select Net on schematic...

**Close** **Help**

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

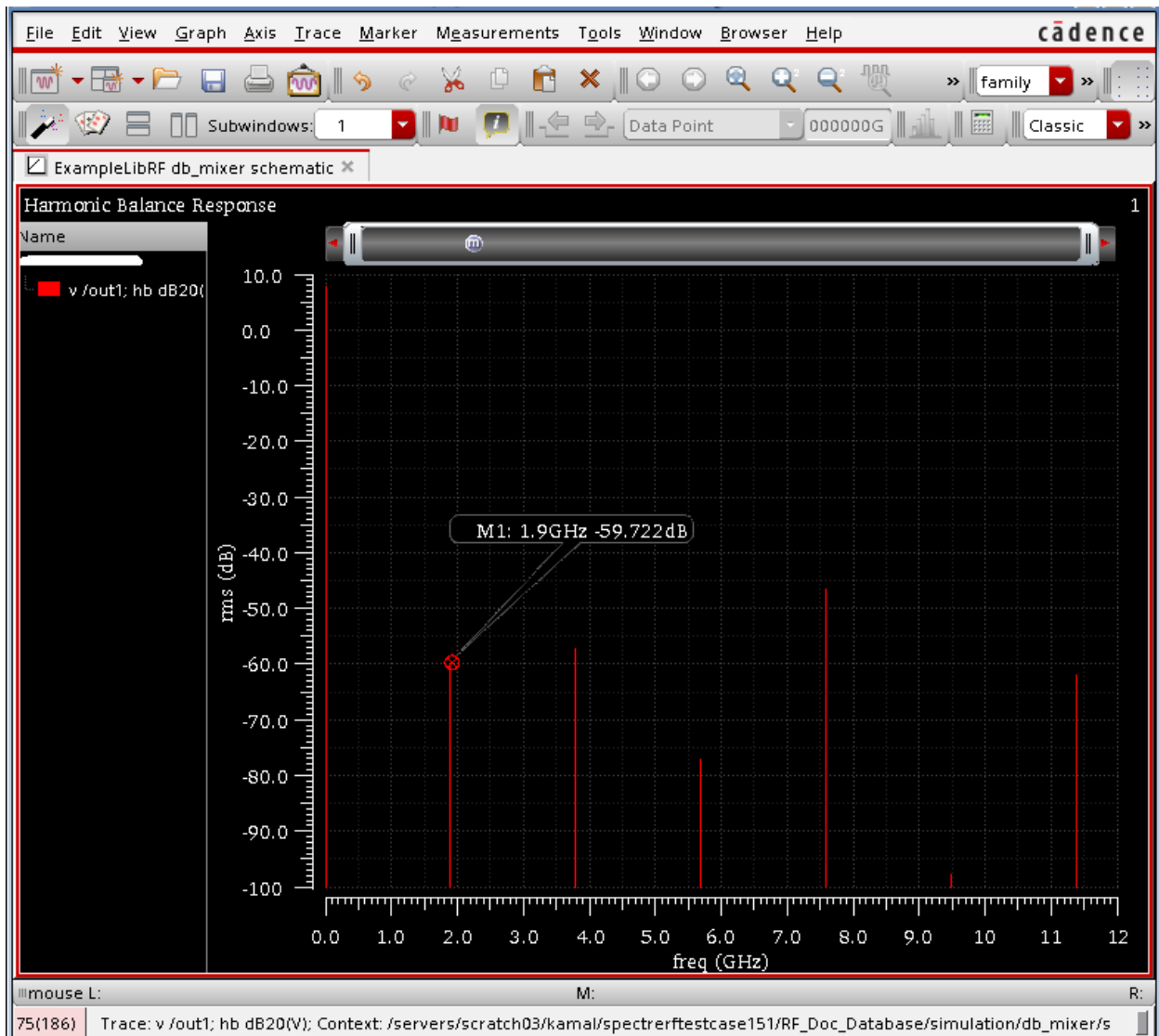


### Measure LO leakage on the IF Output

8. Move the mouse cursor to the tip of the spike at  $1.9\text{GHz}$  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  $-59.72\text{ dB}$ .

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).

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

---

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**

Direct Plot Form

Plotting Mode: New SubWin

**Analysis**

☐ hb ☒ hbac ☐ hbnoise ☐ hbnoise separation

**Function**

☒ Voltage ☐ Voltage Gain ☐ Current ☐ IPN Curves

Select: Net

**Sweep**

☒ spectrum ☐ sideband

**Modifier**

☐ Magnitude ☐ Phase ☒ dB20 ☐ Real ☐ Imaginary

Add To Outputs ☐ Replot

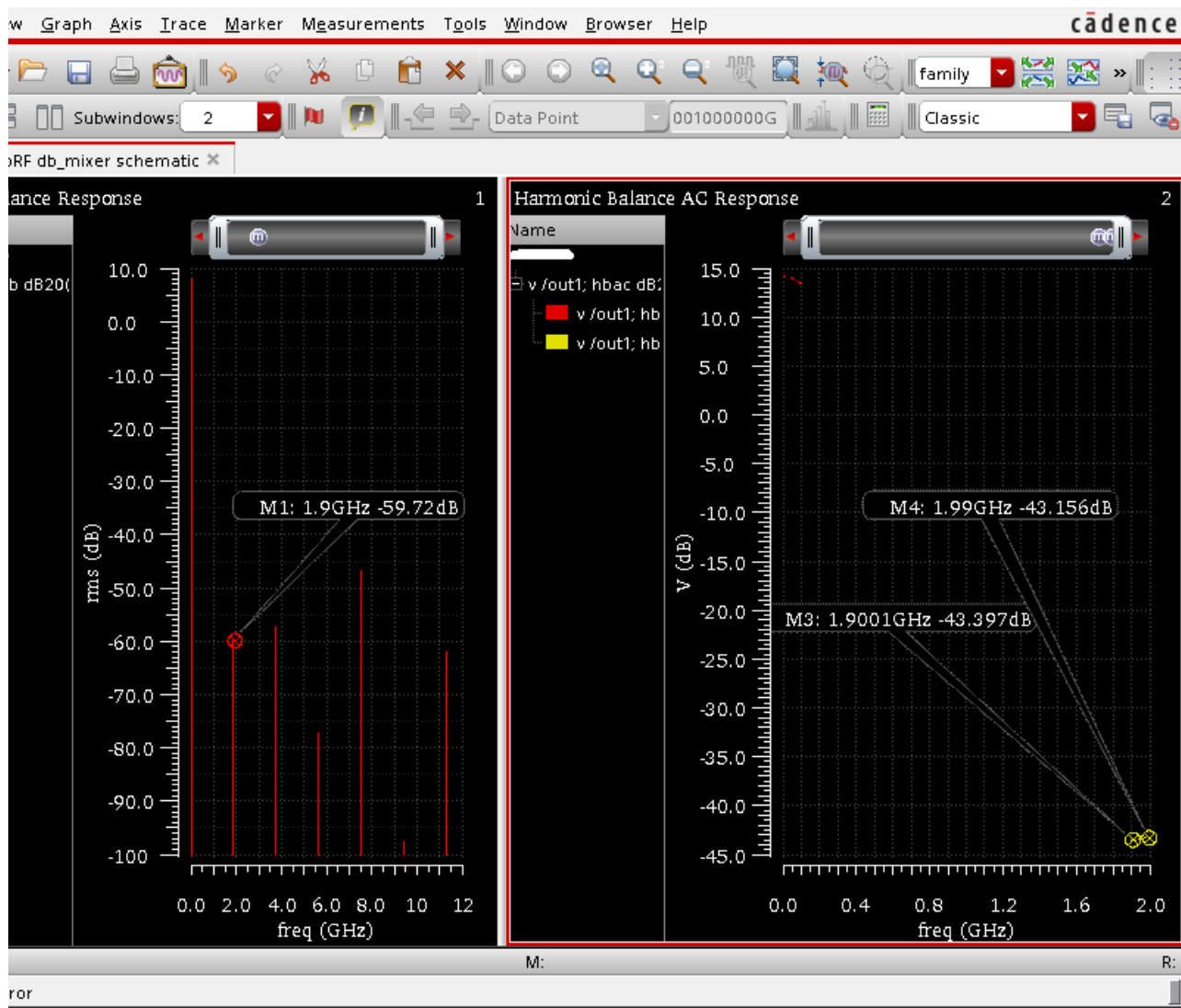
> Select Net on schematic...

Close Help

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.

- Note that the trace near 2GHz is about  $-43.4\text{dB}$ , 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.

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

---

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.

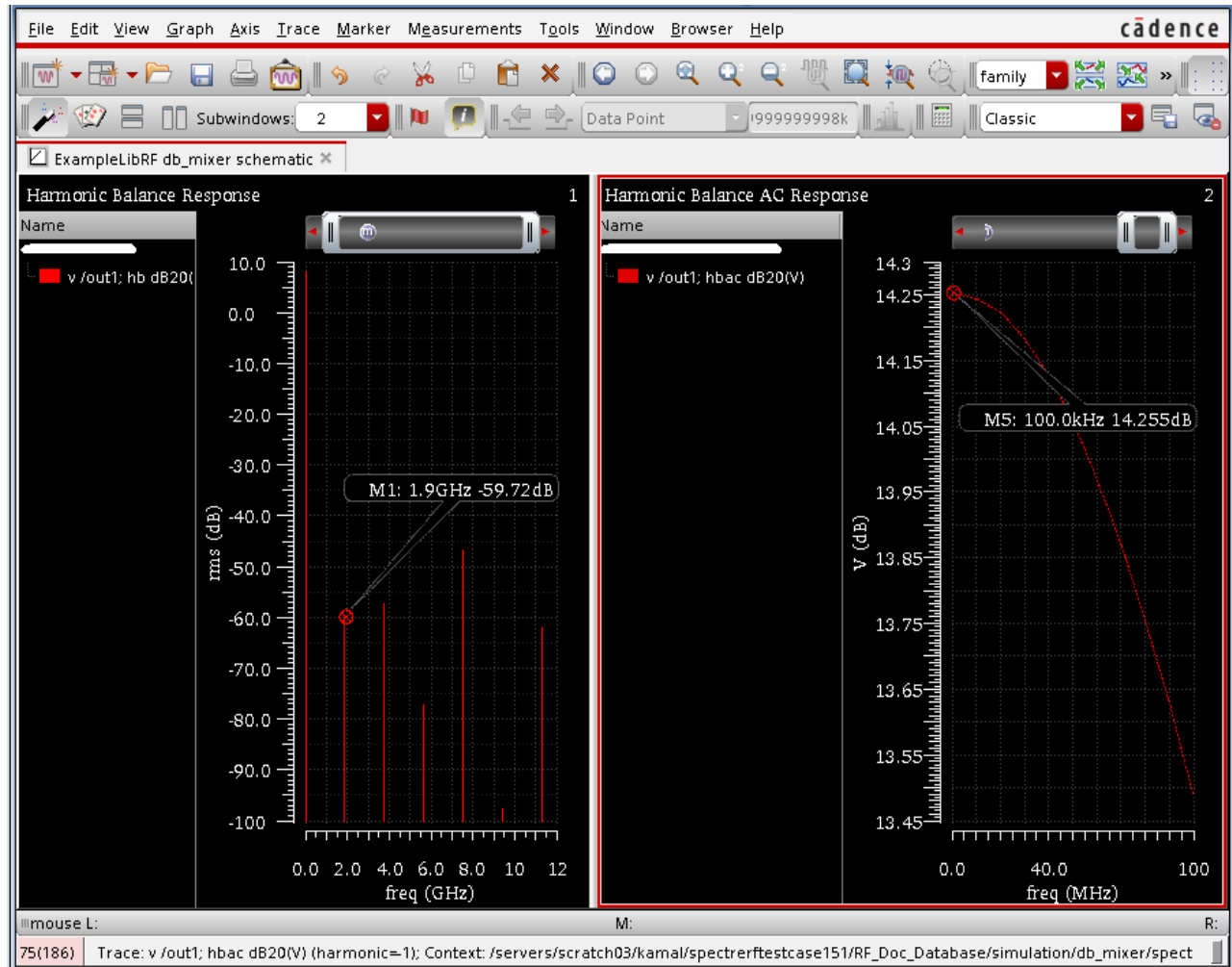
**Figure 4-28 : Choosing Sideband of Interest**



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**

The screenshot shows the 'Direct Plot Form' dialog box. At the top, the title bar says 'Direct Plot Form'. Below it, the 'Plotting Mode' is set to 'New SubWin'. The 'Analysis' section has four radio buttons: 'hb', 'hbac', 'hbnoise' (which is selected), and 'hbnoise separation'. The 'Noise Type' section has four radio buttons: 'USB' (selected), 'LSB', 'AM', and 'PM'. The 'Function' section has eight radio buttons: 'Output Noise' (selected), 'Input Noise', 'Noise Figure', 'Noise Factor', 'NFdsb', 'Fdsb', 'NFieee', 'Fieee', and 'Transfer Function'. The 'Units' section has four radio buttons: 'V/sqrt(Hz)' (selected), 'V\*\*2/Hz', 'dBc/Hz', and 'dBV/Hz'. The 'Scalar Expression' section has a text field labeled 'Value At (Hz)'. At the bottom, there are two buttons: 'Add To Outputs' and 'Plot'. Below these buttons is a message: '> Press plot button on this form...'. At the very bottom are two buttons: 'Close' and 'Help'.

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**

Direct Plot Form

Plotting Mode: New Win

**Analysis**

☐ hb ☐ hbac  
☒ hbnoise ☐ hbnoise separation

**Noise Type**

☒ USB ☐ LSB ☐ AM ☐ PM

**Function**

☐ Output Noise ☐ Input Noise  
☐ Noise Figure ☐ Noise Factor  
☒ NFdsb ☐ Fdsb  
☐ NFieee ☐ Fieee  
☐ Transfer Function

Description: Double Sideband Noise Figure

Integrated Over Bandwidth: ☒

Start Frequency (Hz): 1K  
Stop Frequency (Hz): 1M

Add To Outputs: ☐ Plot

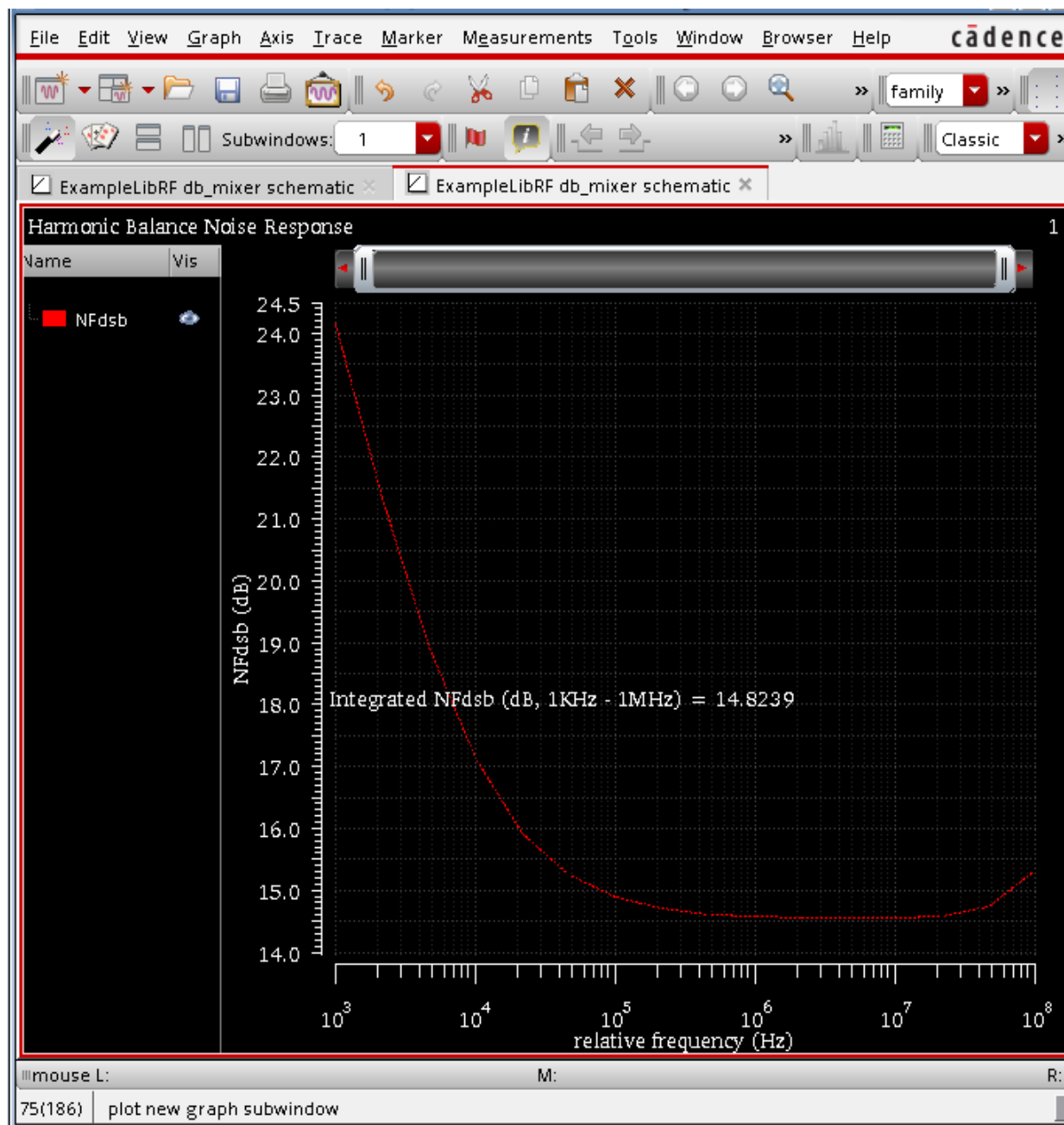
> Press plot button on this form...

Close Help

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 *Choosing 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 the output of the circuit.

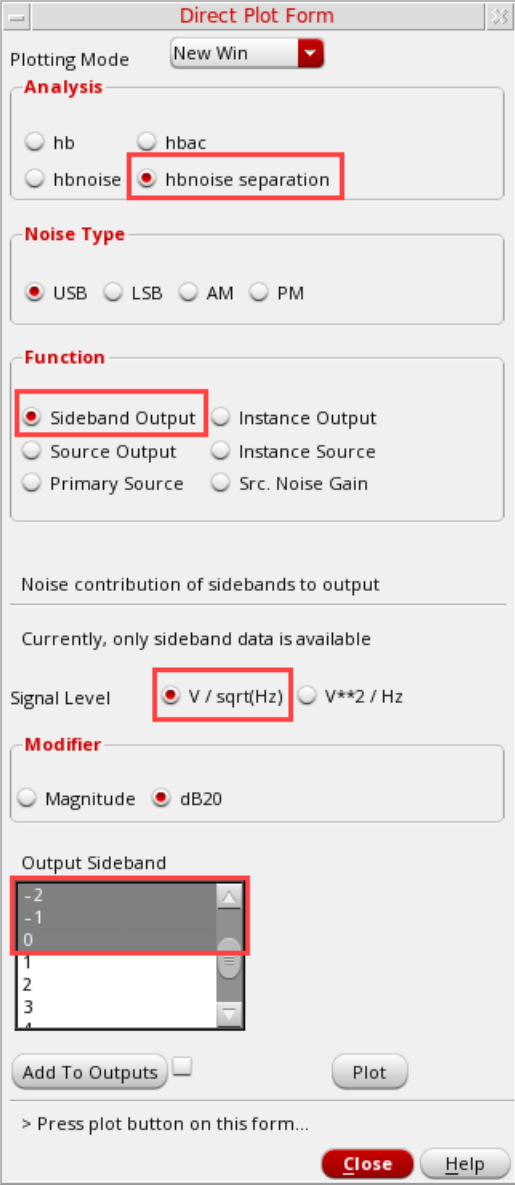


### Tip

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**



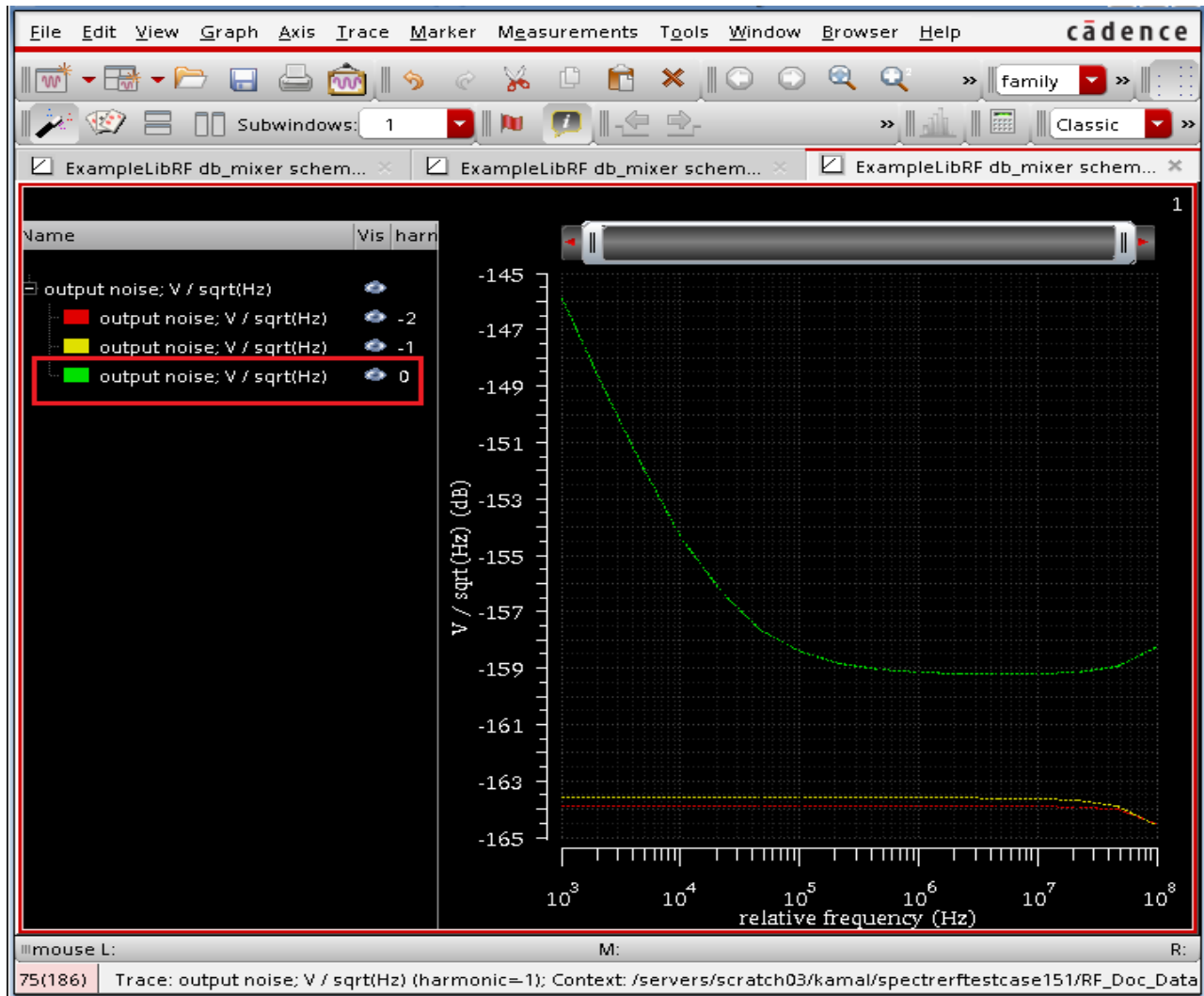
The image shows the 'Direct Plot Form' window in a circuit simulator. The window has several sections with radio button options:

- Plotting Mode:** A dropdown menu set to 'New Win'.
- Analysis:** Radio buttons for 'hb', 'hbac', 'hbnoise', and 'hbnoise separation'. The 'hbnoise separation' option is selected and highlighted with a red box.
- Noise Type:** Radio buttons for 'USB', 'LSB', 'AM', and 'PM'. The 'USB' option is selected.
- Function:** Radio buttons for 'Sideband Output', 'Instance Output', 'Source Output', 'Instance Source', 'Primary Source', and 'Src. Noise Gain'. The 'Sideband Output' option is selected and highlighted with a red box.
- Noise contribution of sidebands to output:** A text area containing the message 'Currently, only sideband data is available'.
- Signal Level:** Radio buttons for 'V / sqrt(Hz)' and 'V\*\*2 / Hz'. The 'V / sqrt(Hz)' option is selected and highlighted with a red box.
- Modifier:** Radio buttons for 'Magnitude' and 'dB20'. The 'dB20' option is selected.
- Output Sideband:** A list box showing sideband numbers: -2, -1, 0, 1, 2, 3, 4. The values -2, -1, and 0 are highlighted with a red box.
- Buttons:** 'Add To Outputs' (disabled), 'Plot', 'Close', and 'Help'.
- Footer:** A message '> Press plot button on this form...'.

- 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



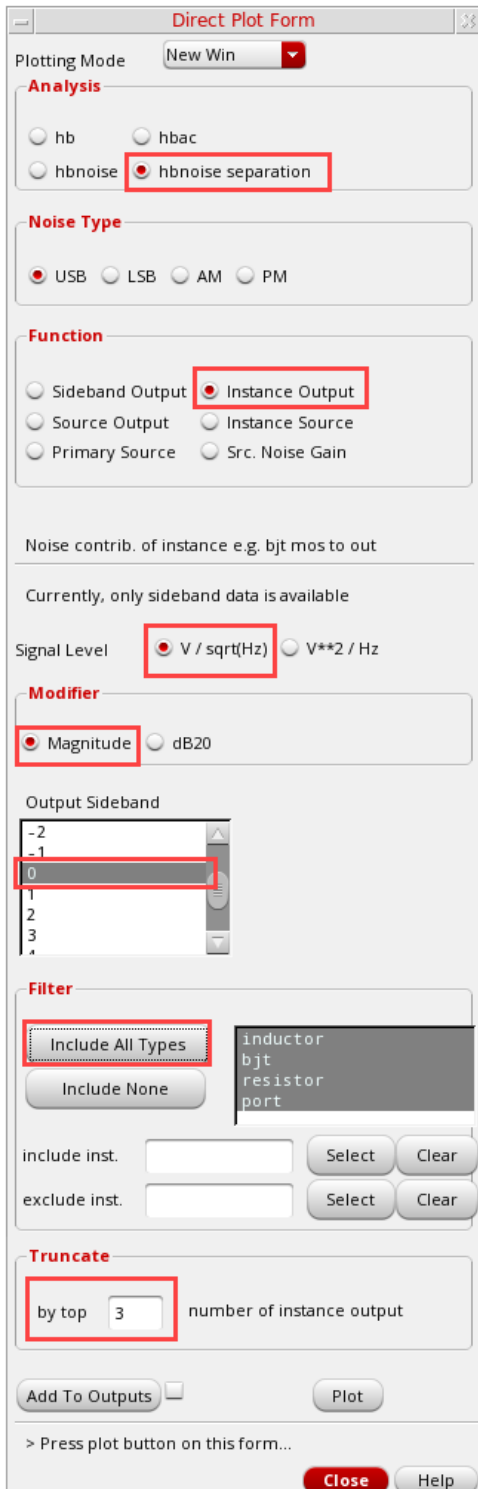
14. Click the **+** sign to the left of *output noise* in the legend on the left side of the waveform window. The legend expands. You can see that the *0 Output sideband* corresponds to the green trace. The *0 Output sideband* contributes the most noise and is the worst noise offender. Next, plot *Instance 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.

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

**Figure 4-35 Direct Plot Form for Plotting Instance Output for Noise Contribution of instances**



The image shows the 'Direct Plot Form' dialog box in a software application. The form is organized into several sections with red headers: 'Analysis', 'Noise Type', 'Function', 'Modifier', 'Filter', and 'Truncate'. In the 'Analysis' section, the 'hbnoise separation' radio button is selected and highlighted with a red box. In the 'Noise Type' section, the 'USB' radio button is selected. In the 'Function' section, the 'Instance Output' radio button is selected and highlighted with a red box. Below this, the text 'Noise contrib. of instance e.g. bjt mos to out' is displayed. In the 'Signal Level' section, the 'V / sqrt(Hz)' radio button is selected and highlighted with a red box. In the 'Modifier' section, the 'Magnitude' radio button is selected and highlighted with a red box. In the 'Output Sideband' section, a list box shows values from -2 to 4, with '0' selected and highlighted with a red box. In the 'Filter' section, the 'Include All Types' button is highlighted with a red box, and a list box shows 'inductor', 'bjt', 'resistor', and 'port'. Below this are fields for 'include inst.' and 'exclude inst.' with 'Select' and 'Clear' buttons. In the 'Truncate' section, the 'by top' checkbox is checked and highlighted with a red box, and the value '3' is entered in the adjacent text box. At the bottom, there are 'Add To Outputs' and 'Plot' buttons, and a note that says '> Press plot button on this form...'. The 'Close' and 'Help' buttons are at the very bottom.

Direct Plot Form

Plotting Mode: New Win

**Analysis**

☐ hb ☐ hbac  
☐ hbnoise ☒ hbnoise separation

**Noise Type**

☒ USB ☐ LSB ☐ AM ☐ PM

**Function**

☐ Sideband Output ☒ Instance Output  
☐ Source Output ☐ Instance Source  
☐ Primary Source ☐ Src. Noise Gain

Noise contrib. of instance e.g. bjt mos to out

Currently, only sideband data is available

Signal Level: ☒ V / sqrt(Hz) ☐ V\*\*2 / Hz

**Modifier**

☒ Magnitude ☐ dB20

Output Sideband

-2  
-1  
0  
1  
2  
3  
4

**Filter**

Include All Types  Include None

inductor  
bjt  
resistor  
port

include inst.   Select  Clear  
exclude inst.   Select  Clear

**Truncate**

☒ by top  3 number of instance output

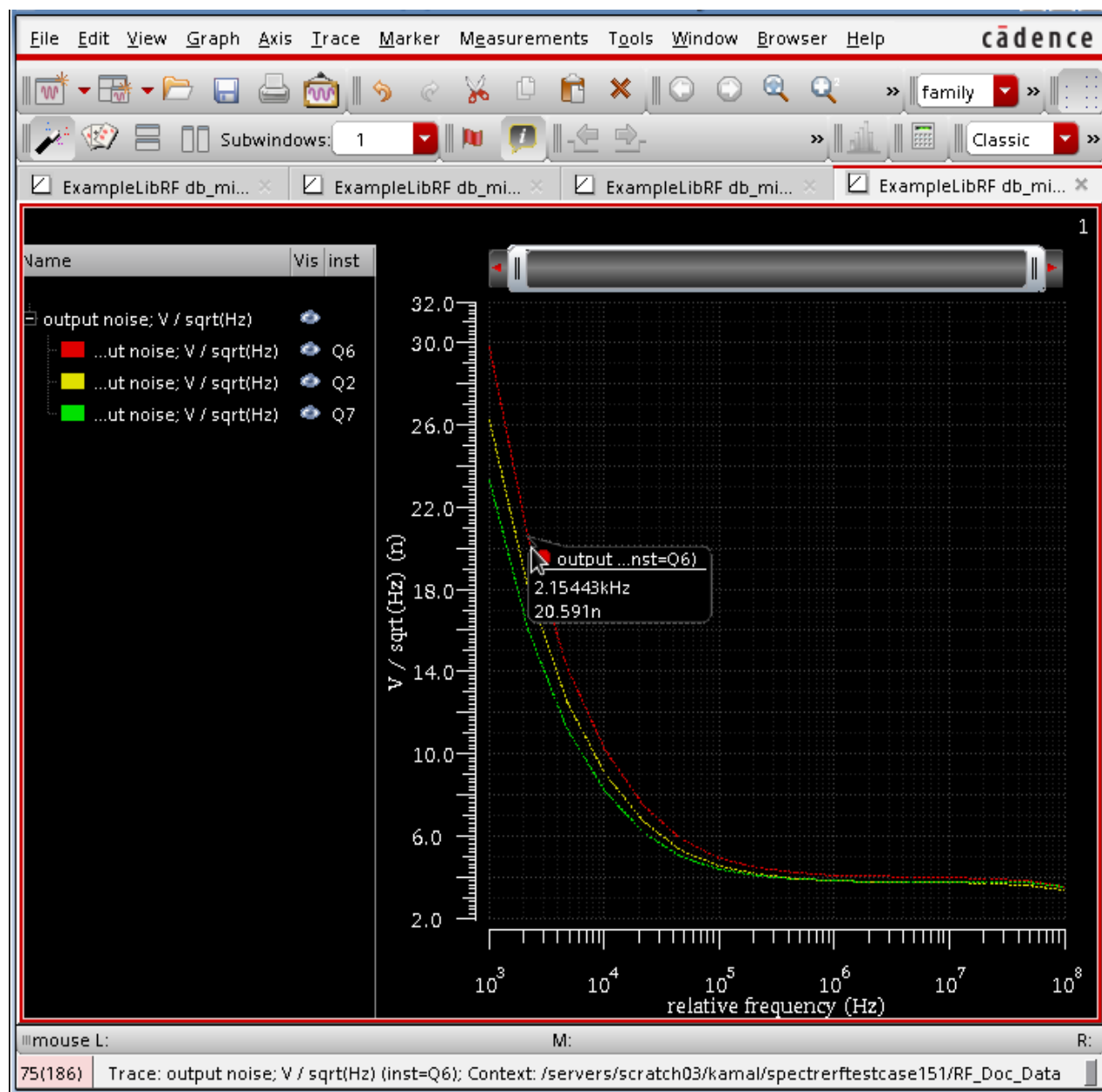
Add To Outputs  Plot

> Press plot button on this form...

Close  Help

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.

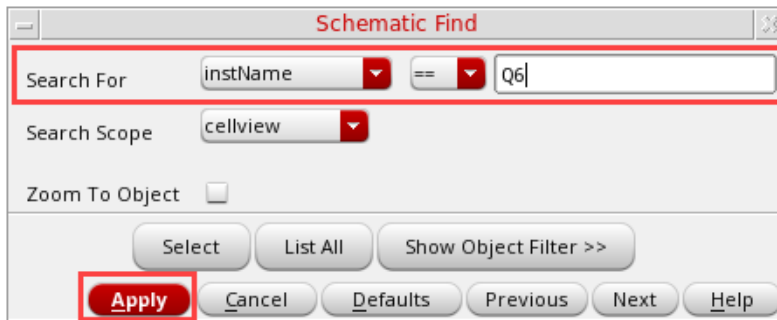
1. If you move your mouse cursor over one of the traces, you can read the instance from the cursor readout and identify which instance contributes the most amount of noise.

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

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.

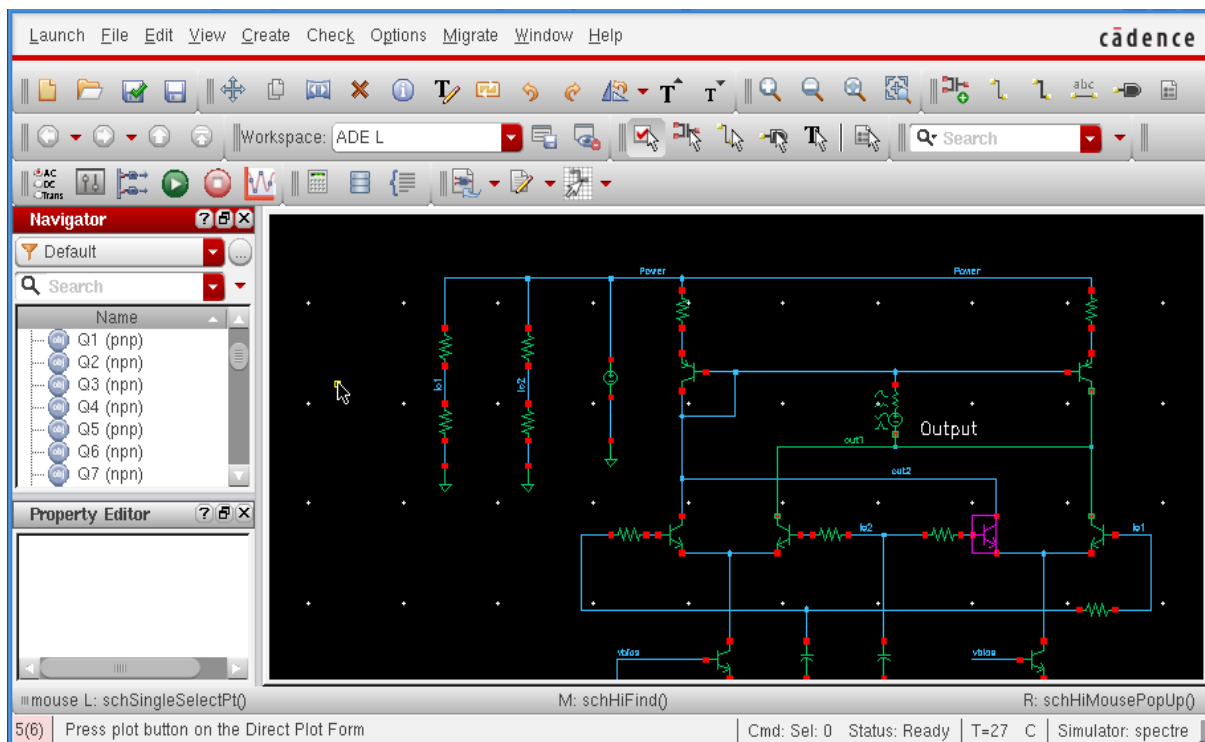
2. To find Q6 in the schematic, choose *Edit - Find* in the schematic. The *Schematic Find* form is displayed. Search for *instanceName == Q6* in the *cellview* and click *Apply*.

**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**

☐ Sideband Output   ☐ Instance Output  
☒ **Source Output**   ☐ Instance Source  
☐ Primary Source   ☐ Src. Noise Gain

Noise contrib. of primary source in instance to out

Currently, only sideband data is available

Signal Level   ☒ **V / sqrt(Hz)**   ☐ V\*\*2 / Hz

**Modifier**

☒ Magnitude   ☐ dB20

Output Sideband

-2  
-1  
**0**  
1  
2  
3

**Filter**

**Include All Types**    Include None

inductor  
bjt  
resistor  
port

include inst.     Select    Clear

exclude inst.     Select    Clear

**Truncate**

by top  number of source output

Add To Outputs    Plot

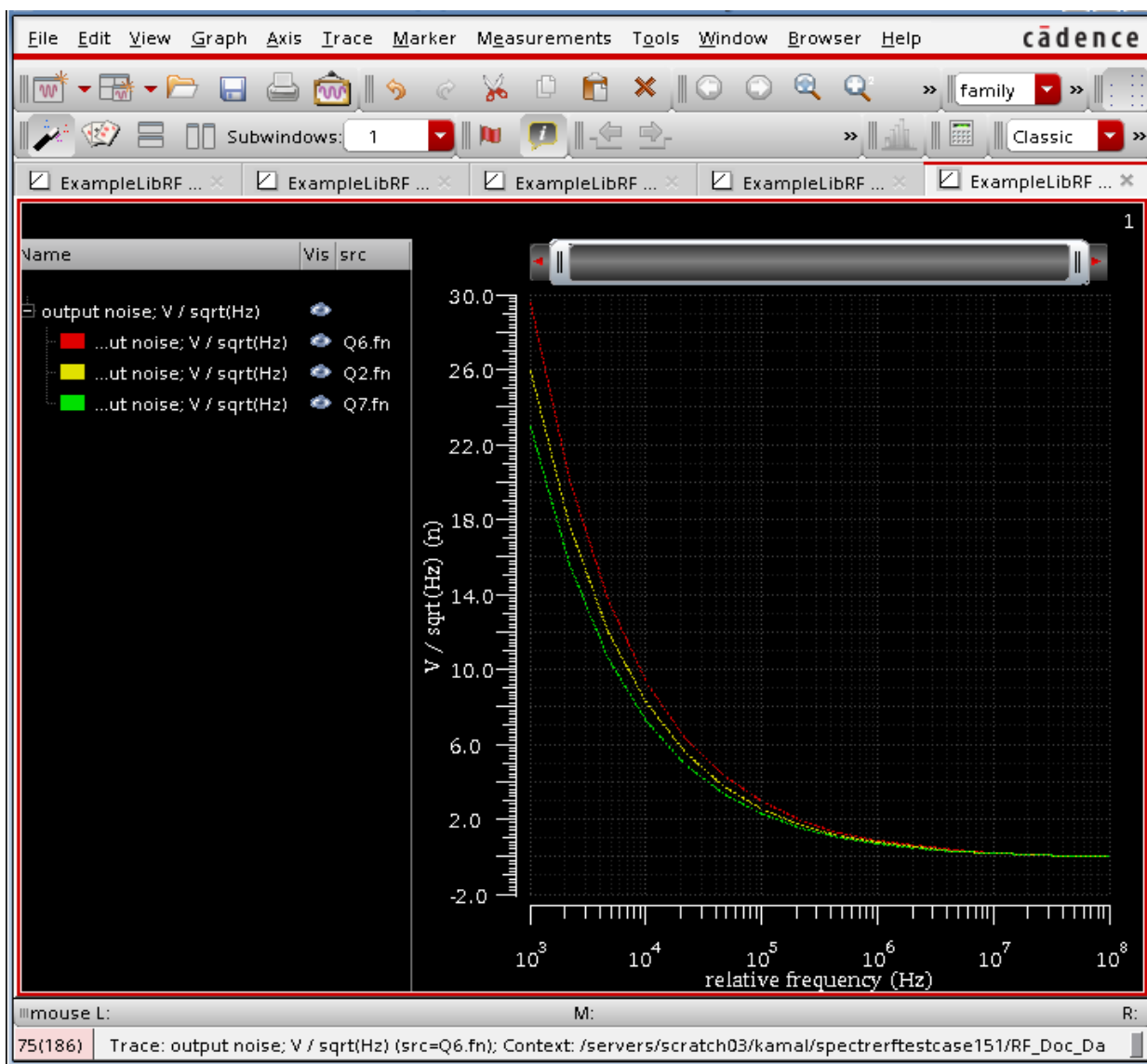
> Press plot button on this form...

**OK**    Cancel    Help

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**



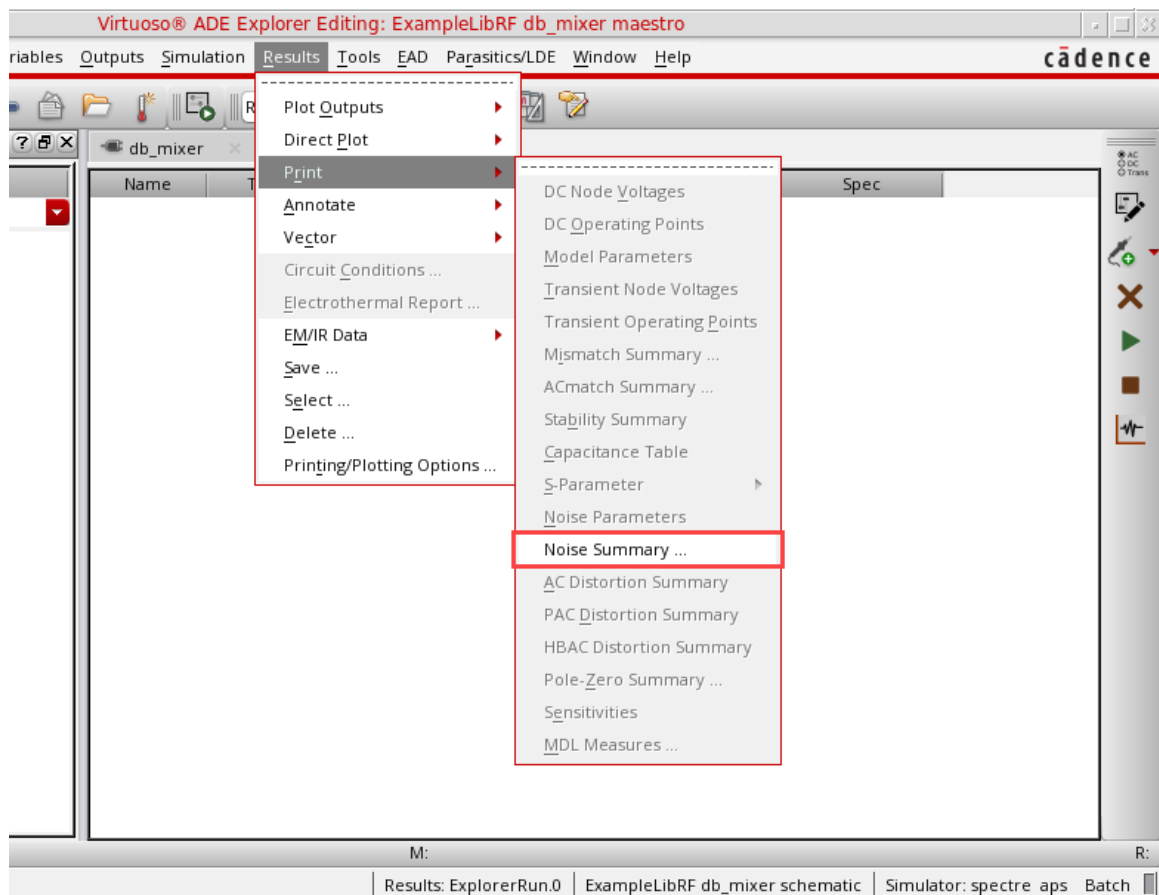
Note that the top three Source Output contributors are displayed in the legend on the left side of the graph: *Q6: fn*, *Q2: fn*, and *Q7: fn*. *fn* refers to flicker noise. Refer to the Noise Summary section in [Chapter 3: Frequency Domain Analyses: Harmonic Balance](#) of the Spectre® Circuit Simulator and Accelerated Parallel Simulator RF Analysis User Guide to get a description of the parameter names for the different device types.



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

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**

**Noise Summary**

Data from **hbnoise\_am**  
Print the output noise of 'hbnoise\_am-rif' analysis

Type ☒ spot noise ☐ integrated noise noise unit **V^2**

Frequency Spot (Hz) **1k**

**FILTER**

hierarchy level

**Include All Types**  
**Include None**

inductor  
bjt  
resistor  
port

include instances  **Select** **Clear**

exclude instances  **Select** **Clear**

**TRUNCATE & SORT**

truncate **by number** top **3**

sort by ☒ noise contributors ☐ composite noise ☐ device name

**OK** **Cancel** **Apply** **Help**

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.

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

---

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**

**Noise Summary**

Data from **hbnoise\_src**  
Print the noise source of 'hbnoise' analysis

Type ☐ spot noise ☒ integrated noise noise unit **A**

From (Hz) **1K** To (Hz) **1M**

weighting ☒ flat ☐ from weight file

**FILTER**

hierarchy level **▼**

**Include All Types** **Include None**

**inductor**  
**bjt**  
**resistor**  
**port**

include instances  **Select** **Clear**

exclude instances  **Select** **Clear**

**TRUNCATE & SORT**

truncate **by number** **top** **20**

sort by ☒ noise contributors ☐ composite noise ☐ device name

**OK** **Cancel** **Apply** **Help**

**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 [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.

**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 which 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.

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

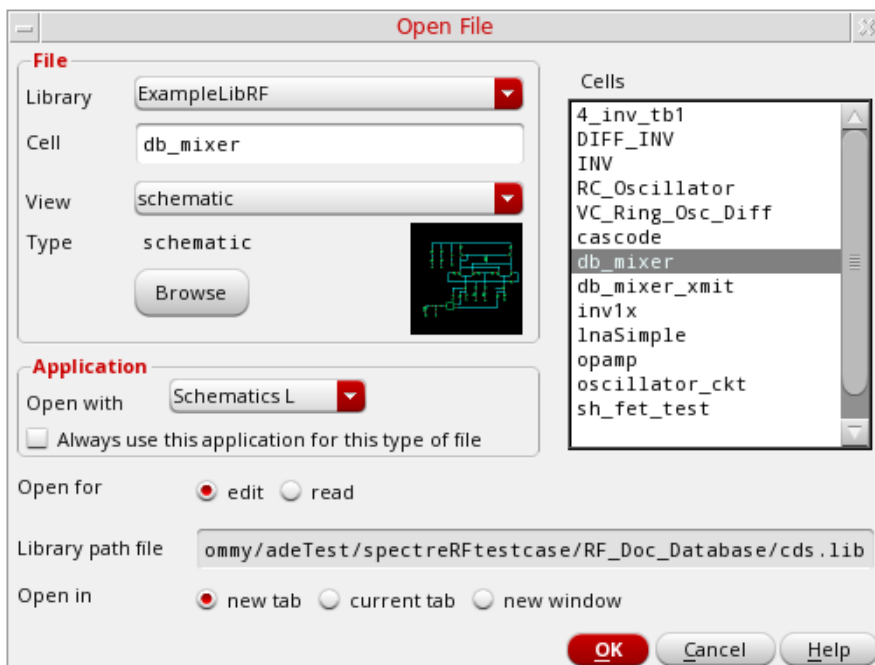
---

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.

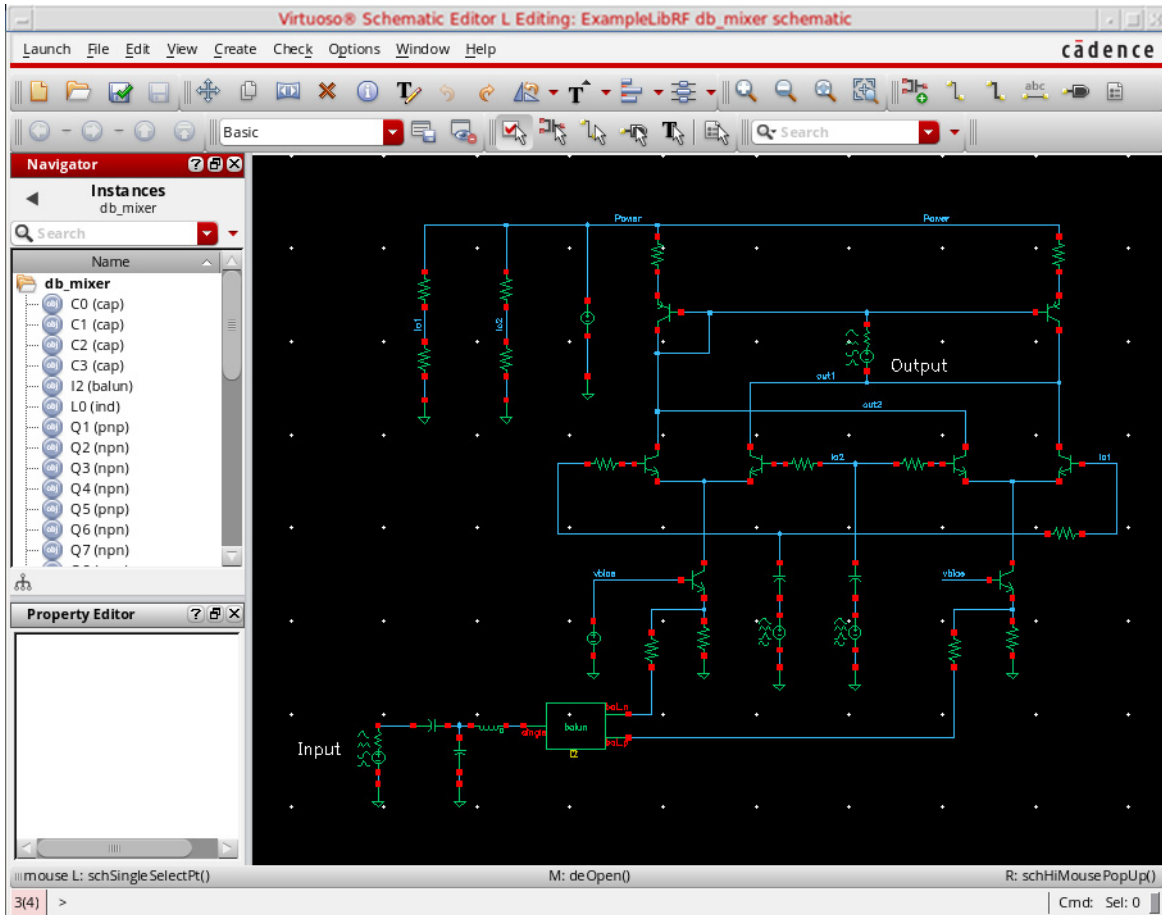
**Figure 4-45 Open File Form**



2. Click *OK*. The *db\_mixer* schematic appears, as shown below.

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

Figure 4-46 db\_mixer Schematic

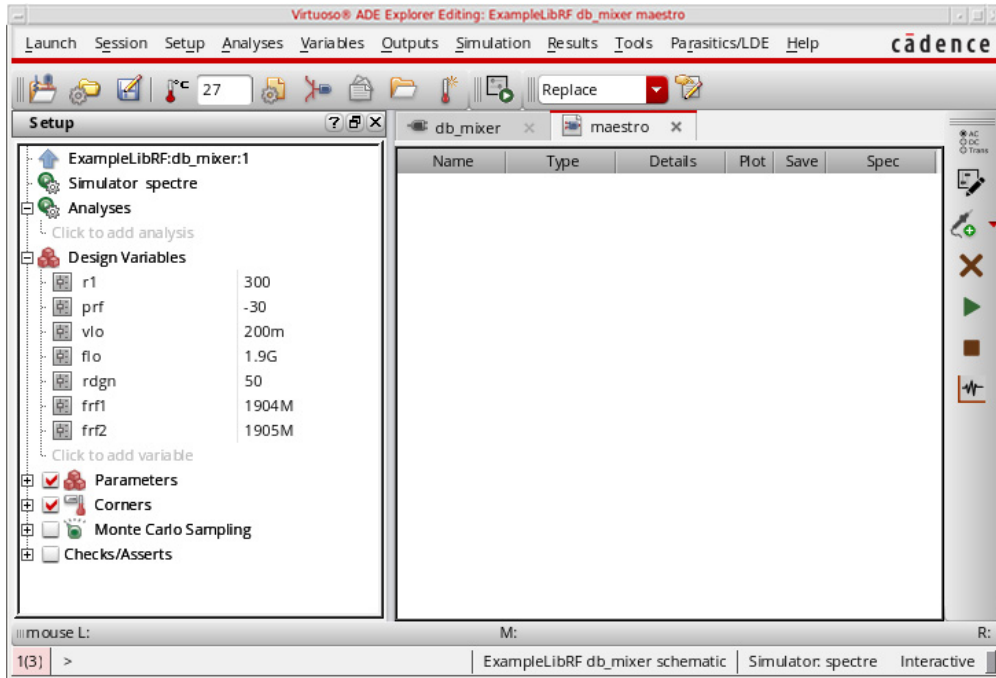


3. Start the *Analog Design Environment* from the schematic by choosing *Launch - ADE Explorer*. The Simulation Window is displayed, as shown below.



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

**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**

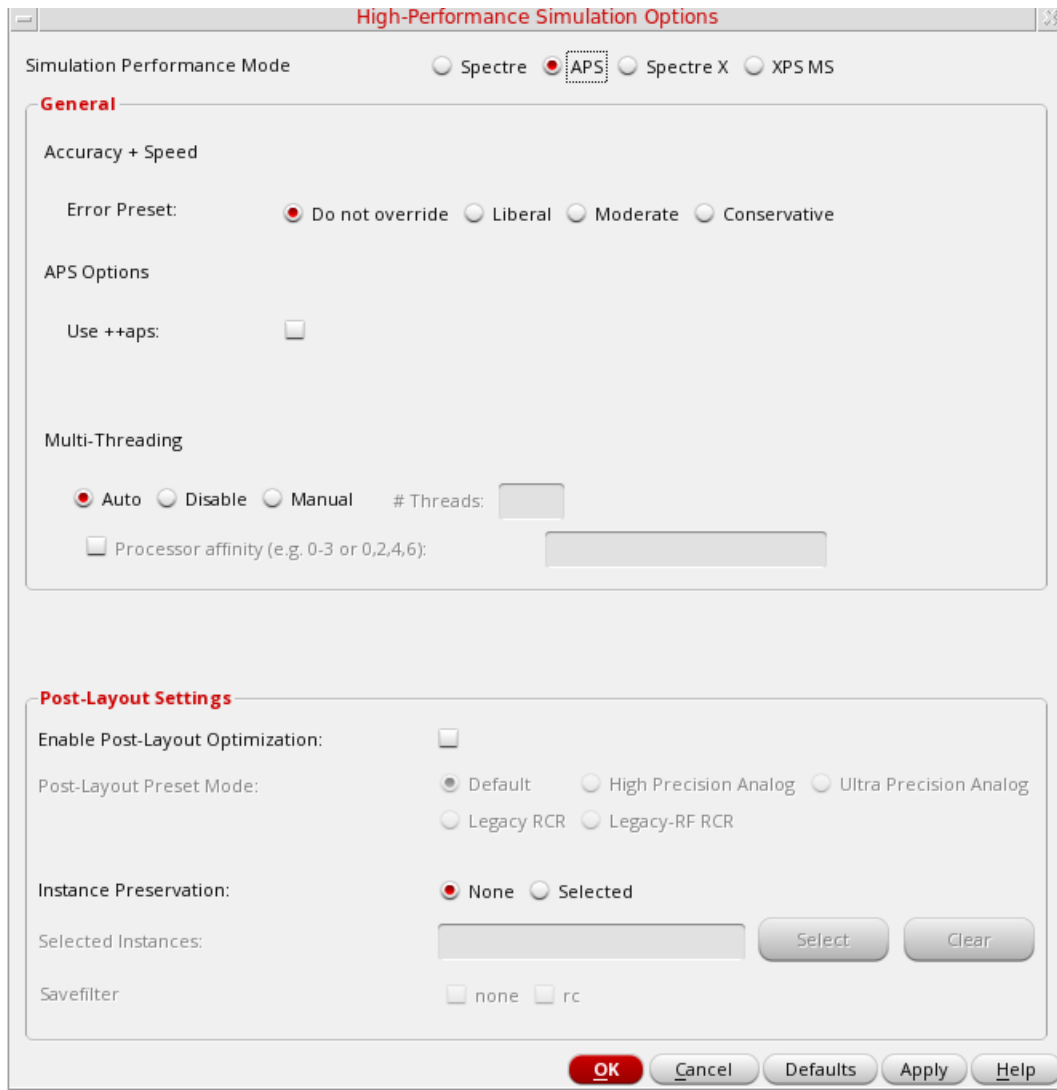


6. Click *OK* to close the *Choosing Simulator* form.
7. Set up the High Performance Simulation Options, as follows:

In ADE Explorer, choose *Setup - High Performance Simulation*. The High Performance Simulation Options window is displayed, as shown below.

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

Figure 4-49 High Performance Simulation Options Form



The image shows the 'High-Performance Simulation Options' dialog box. At the top, the title bar reads 'High-Performance Simulation Options'. Below the title bar, the 'Simulation Performance Mode' section has four radio buttons: 'Spectre', 'APS' (which is selected and highlighted with a dashed box), 'Spectre X', and 'XPS MS'. The 'General' section contains the following options: 'Accuracy + Speed' with an 'Error Preset' of 'Do not override' (selected), 'Liberal', 'Moderate', and 'Conservative'; 'APS Options' with a 'Use ++aps:' checkbox that is unchecked; and 'Multi-Threading' with 'Auto' (selected), 'Disable', and 'Manual' radio buttons, a '# Threads:' text field, and a 'Processor affinity (e.g. 0-3 or 0,2,4,6):' text field. The 'Post-Layout Settings' section includes 'Enable Post-Layout Optimization:' (unchecked), 'Post-Layout Preset Mode:' with radio buttons for 'Default' (selected), 'High Precision Analog', 'Ultra Precision Analog', 'Legacy RCR', and 'Legacy-RF RCR'; 'Instance Preservation:' with 'None' (selected) and 'Selected' radio buttons; a 'Selected Instances:' text field with 'Select' and 'Clear' buttons; and a 'Savefilter' section with 'none' and 'rc' checkboxes. At the bottom, there are five buttons: 'OK' (highlighted in red), 'Cancel', 'Defaults', 'Apply', and 'Help'.

- 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*.

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

### 8. Select *Outputs* – *Save All*.

The *Save Options* form is displayed.

**Figure 4-50 Save Options Form**

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

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.

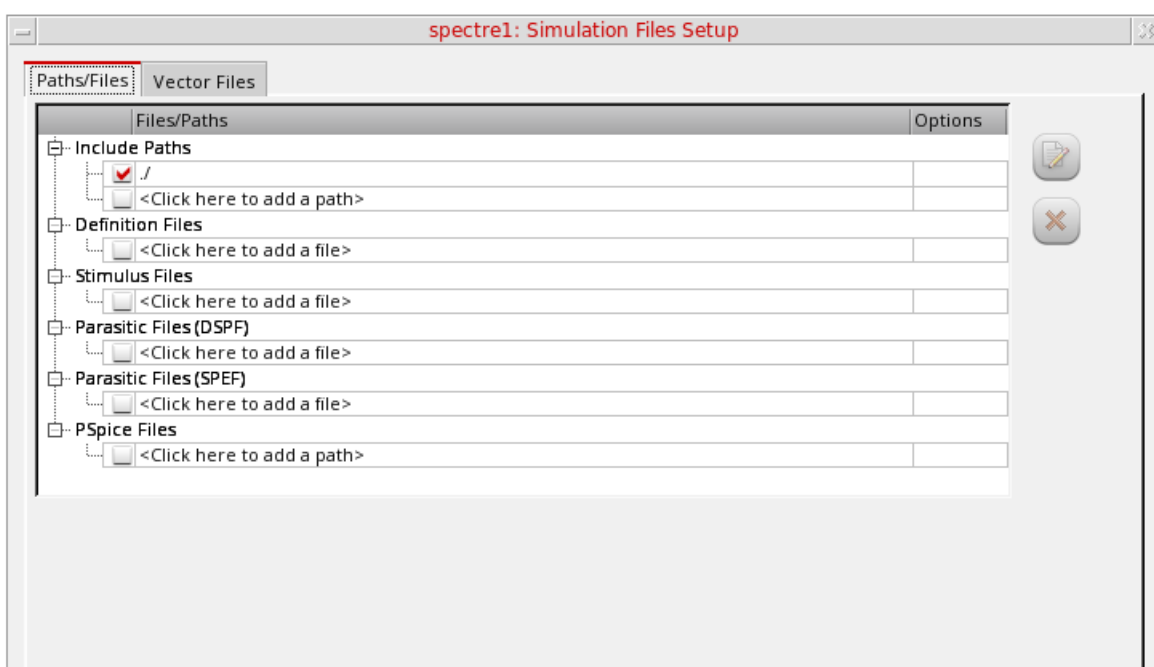
10. Click *OK* to close the *Save Options* form.

### 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.

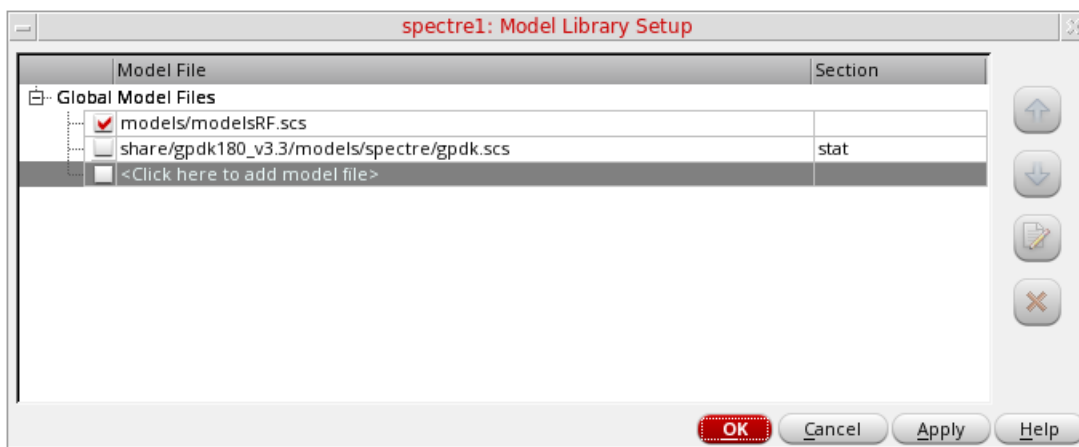
4. In the *Model Library File* field, type the following path to the model file including the file name:

```
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.

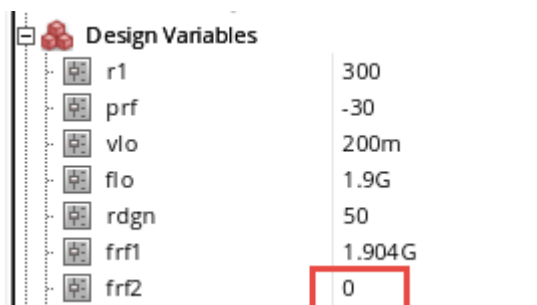
## Setting Design Variables

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.



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

---

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

**Figure 4-54 hb Choosing Analyses Form**

**Choosing Analyses -- ADE Explorer**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☒ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbasp ☐ hbxp

Harmonic Balance Analysis

Transient-Aided Options

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) **auto**

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

Tone 1

Fundamental Frequency

Number of Harmonics **auto**

Oversample Factor

Freqdivide Ratio for Tone 1

Harmonics **Default**

Accuracy Defaults (errpreset)

☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☐ **Options...**

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

---

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

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

---

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. Set the *Sweep Type* to *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:



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

Figure 4-55 hb Choosing Analyses Form

☐ qpnoise ☐ qpxf ☐ qpnp ☒ hb  
☐ hbac ☐ hbnoise ☐ hbnp

Harmonic Balance Analysis

Transient-Aided Options

Run transient? Decide automatically

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Multi-divider Mode ☐

Number of Tones ☐ 1 ☒ 2 ☐ 3 ☐ 4

Tone 1 Tone 2

Fundamental Frequency 1.9G 1.904G

Number of Harmonics auto 5

Oversample Factor 1 1

Tone 1 be LO or signal which causes the most nonlinearity.

Freqdivide Ratio for Tone 1 1

Harmonics Default

Accuracy Defaults (errpreset)

☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

Sweep 1 ☒

Variable Variable

Frequency Variable? ☒ no ☐ yes

Variable Name prf

Select Design Variable

Sweep Range

☒ Start-Stop Start -30 Stop -10

☐ Center-Span

Sweep Type

☒ Linear ☐ Logarithmic

☒ Step Size 5

☐ Number of Steps

Add Specific Points ☒ -8 -7.25 -6.5 -5.75

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**

The screenshot shows the 'hbnoise Choosing Analyses Form' dialog box. At the top, there are radio buttons for selecting the analysis type: pxf, psp, qpss, qpac, qpnoise, qpxf, qpsp, hb, hbac, **hbnoise** (selected and highlighted with a red box), and hbasp. Below this, the title 'Harmonic Balance Noise Analysis' is displayed. The 'Multiple hbnoise' checkbox is unchecked. The 'Sweeptype' is set to 'default' with a dropdown arrow, and a note states 'Sweep is currently absolute'. The 'Output Frequency Sweep Range (Hz)' section includes a 'Start-Stop' dropdown, 'Start' and 'Stop' input fields, and a 'Sweep Type' dropdown set to 'Automatic'. The 'Add Specific Points' checkbox is unchecked. The 'Sidebands' section has a 'Maximum sideband' dropdown and an input field, with a note: 'When using hb engine, default value is harms of 1st tone.' The 'Noise Figure' checkbox is unchecked. The 'Output' section has a 'probe' dropdown, an 'Output Probe Instance' input field, and a 'Select' button. The 'Do Noise' checkbox is checked. The 'Noise Type' is set to 'timeaverage' with a dropdown arrow. Below this, a description reads: 'Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis'. The 'Contribution Type' section has radio buttons for USB (selected), AM, PM, AM&PM, and ALL(AM,PM,USB,LSB). The 'Noise Separation' section has 'yes' and 'no' checkboxes, with 'no' selected. A note states: 'Separate noise into source and gain'. At the bottom left, the 'Enabled' checkbox is unchecked. At the bottom right, there is an 'Options...' button. The bottom of the dialog features a row of buttons: 'OK' (highlighted in red), 'Cancel', 'Defaults', 'Apply', and 'Help'.

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

---

### Discussion

In hbnoise, the output frequency is  $f_{out}=200K$ . The LO is at  $1.9GHz$ . The RF signal frequency is  $1.9002G$ , and calculated as follows:

$$f_{in}=f_{out}+1*f_{lo}+0*f_{rf1}=200K+1*1.9G+0*1.904G=1.9002G$$

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  $1.904GHz$  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 *Sweep type* 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 ( $f_{out}$ ) 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.

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

---

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:

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

Figure 4-57 hbnoise Choosing Analyses form

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ sp ☐ envlp

☐ pss ☐ pac ☐ pstb ☐ pnoise

☐ pxf ☐ psp ☐ qpss ☐ qpac

☐ qpnoise ☐ qpxf ☐ qpss ☐ qb

☐ hbac ☒ hbnoise ☐ hbss

Harmonic Balance Noise Analysis

Multiple hbnoise ☐

Sweep type  Sweep is currently absolute

Output Frequency Sweep Range (Hz)

Add Specific Points ☐

Sidebands

When using hb engine, default value is harms of 1st tone.

Noise Figure ☒

Output

Output Probe Instance

Input Source

Input Port Source

Reference Side-Band  $|f(in)| = |f(out) + refsideband\ freq\ shift|$

From (Hz)  To (Hz)  Max. Order

| side | Frequency | 1  | 1.96 | 1.9846 |
|------|-----------|----|------|--------|
| l    | 3.8M      | 1  | -1   |        |
| u    | 4.2M      | -1 | 1    |        |
| l    | 1.8998G   | -1 | 0    |        |
| u    | 1.9882G   | 1  | 0    |        |
| l    | 1.988G    | 0  | -1   |        |

Do Noise ☒

Noise Type

Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis

Contribution Type:

☒ USB ☐ AM ☐ PM ☐ AM&PM ☐ ALL(AM,PM,USB,LSB)

Enabled ☒

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

---

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**

The screenshot shows the 'Choosing Analyses -- ADE Explorer' dialog box. The 'Analysis' section contains a grid of radio buttons for various analysis types: tran, dc, ac, noise, xf, sens, dcmatch, acmatch, stb, pz, lf, sp, envlp, pss, pac, pstb, pnoise, pxf, psp, qpss, qpac, qpnoise, qpxf, qpssp, hb, hbac (selected), hbstb, hbnoise, hbasp, and hbxp. Below this is the 'Harmonic Balance AC Analysis' section. It includes a 'Sweep type' dropdown set to 'default' with a note 'Sweep is currently absolute'. The 'Input Frequency Sweep Range (Hz)' section has a 'Start-Stop' dropdown, 'Start' and 'Stop' input fields. The 'Sweep Type' dropdown is set to 'Automatic'. There are checkboxes for 'Add Specific Points' and 'Add Points By File'. The 'Sidebands' section has a 'Maximum sideband' dropdown and a text box. A note states 'When using hb engine, default value is harms of 1st tone.' The 'Specialized Analyses' dropdown is set to 'None'. At the bottom left is an 'Enabled' checkbox. At the bottom right is an 'Options...' button. The bottom of the dialog features 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help' buttons.

Fill in the hbac Choosing Analyses form as follows:

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

---

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 *1 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:



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

Figure 4-59 The hbac Choosing Analyses form

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☐ hb ☒ hbac ☐ hbstb ☐ hbnoise

☐ hbsp ☐ hbxf

Harmonic Balance AC Analysis

Sweep type: default Sweep is currently absolute

Input Frequency Sweep Range (Hz)

Single-Point Freq: 1.9002G

Add Specific Points ☐

Add Points By File ☐

Sidebands

Select from range From (Hz): 0 To (Hz): 1e12 Max. Order: 1

| side | Frequency | 1.9G | 1.904G |
|------|-----------|------|--------|
| 1    | 200K      | -1   | 0      |
| 1    | 3.8M      | 0    | -1     |
| 1    | 1.8962G   | 1    | -1     |
| u    | 1.9002G   | 0    | 0      |
| 1    | 1.9038G   | -1   | -1     |
| u    | 1.9042G   | -1   | 1      |
| u    | 3.8002G   | 1    | 0      |

Specialized Analyses

None

Enabled ☒

Options...

OK Cancel Defaults Apply Help

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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☐ tstab ☒ hb\_mt

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Power Added Eff. ☐ Power Gain Vs Pout  
☐ Comp. Vs Pout ☐ Node Complex Imp.

Select: Net

**Sweep**

☒ spectrum ☐ frequency ☐ variable  
☐ ifft

Signal Level: ☒ peak ☐ rms

**Modifier**

☒ Magnitude ☐ Phase ☐ dB20  
☐ Real ☐ Imaginary

**Scalar Expression**

Value At (harms):

Variable Value (prf):

- 40
- 35
- 30
- 25
- 20

Add To Outputs: ☐

> Select Variable Value (prf) on this form...

Close Help

3. Select *Compression Point* in the *Function* section.
4. Ensure that *Port (fixed R(port))* is selected from the *Select* drop-down list.

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

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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☐ tstab ☒ hb\_mt

**Function**

☐ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☒ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Power Added Eff. ☐ Power Gain Vs Pout  
☐ Comp. Vs Pout ☐ Node Complex Imp.

Select: Port (fixed R(port))

Format: Output Power

Gain Compression (dB): 1

"prf" ranges from -40 to -20

Input Power Extrapolation Point (dBm): (Defaults to -40)

Input Referred 1dB Compression

|                    | Freq. (Hz) | LO | RF | RF2 |
|--------------------|------------|----|----|-----|
| 1st Order Harmonic | 3M         | 0  | -3 | 3   |
|                    | 4M         | -1 | 1  | 0   |
|                    | 5M         | -1 | 0  | 1   |
|                    | 6M         | -1 | -1 | 2   |
|                    | 7M         | -2 | 3  | -1  |

Loadpull Contours ☐

Add To Outputs ☐

> Select Port on schematic...

Close Help

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

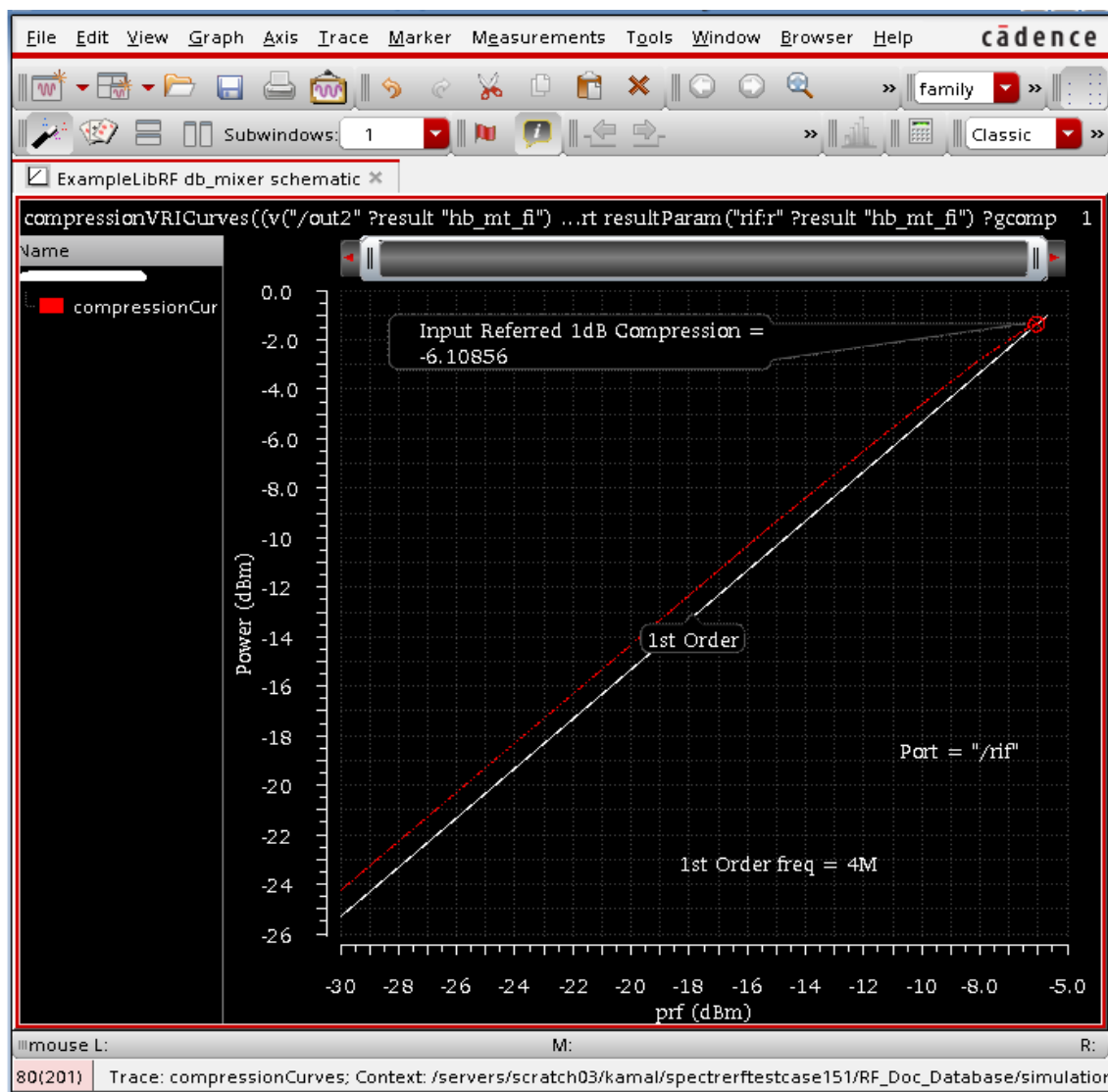
7. In the schematic, select the signal source just to the left of the *Output* label.

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  $-6.1\text{ dB}$ .

**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 *h<sub>bac</sub>\_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 *199.999999998K -1 0* line.

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

**Figure 4-63 Direct Plot Form hbac\_mt**

The image shows the 'Direct Plot Form' dialog box for the 'hbac\_mt' analysis. The 'Plotting Mode' is set to 'New SubWin'. Under the 'Analysis' section, 'hbac\_mt' is selected. Under the 'Function' section, 'Voltage' is selected. The 'Select' dropdown is set to 'Net'. Under the 'Sweep' section, 'variable' is selected. Under the 'Modifier' section, 'dB20' is selected. The 'Output' section shows a table with the following data:

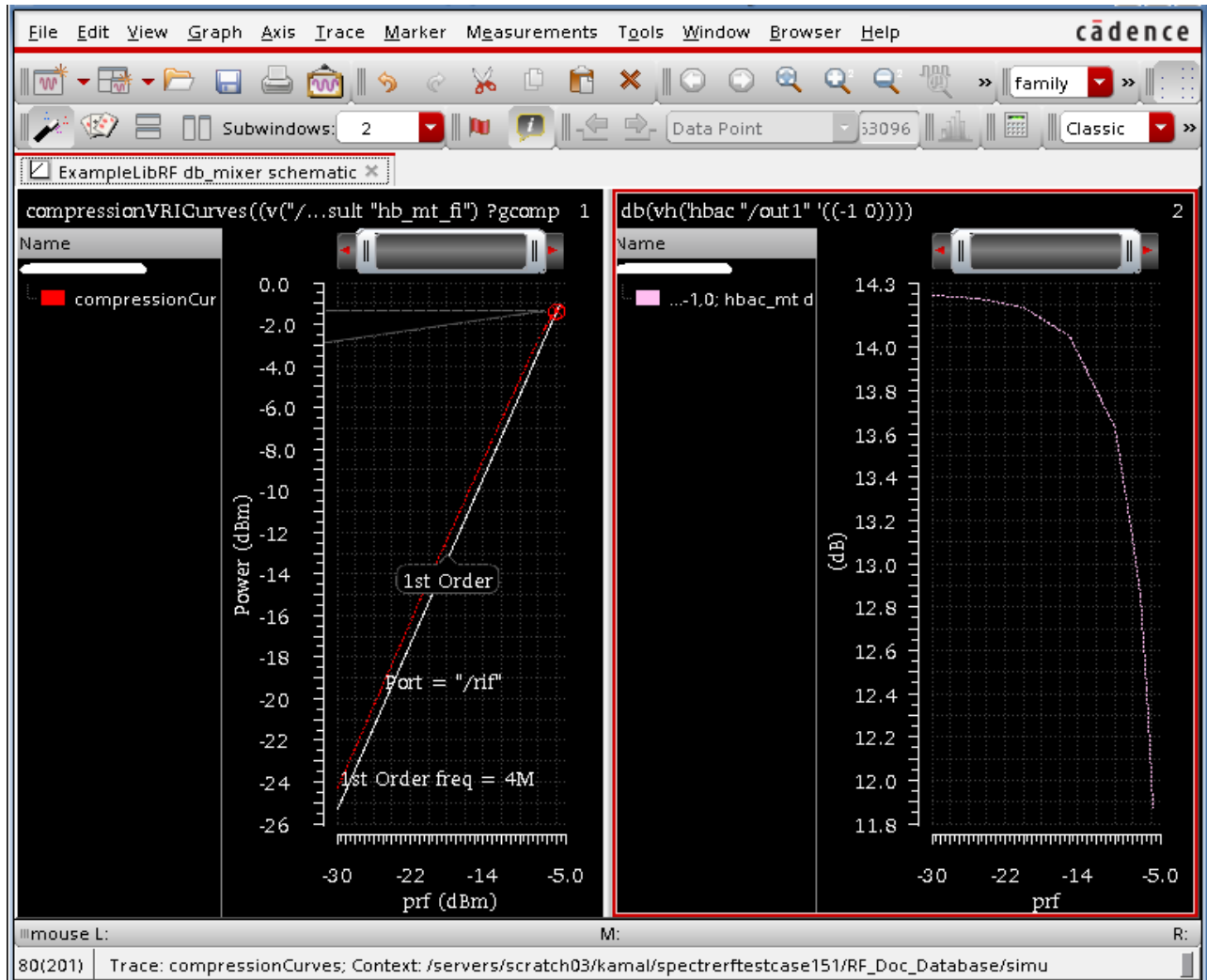
| Freq. (Hz)     | _hb_ton | _hb_ton |
|----------------|---------|---------|
| 199.999999998K | -1      | 0       |

The 'Loadpull Contours' and 'Add To Outputs' checkboxes are unchecked. The 'OK', 'Cancel', and 'Help' buttons are at the bottom.

**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:



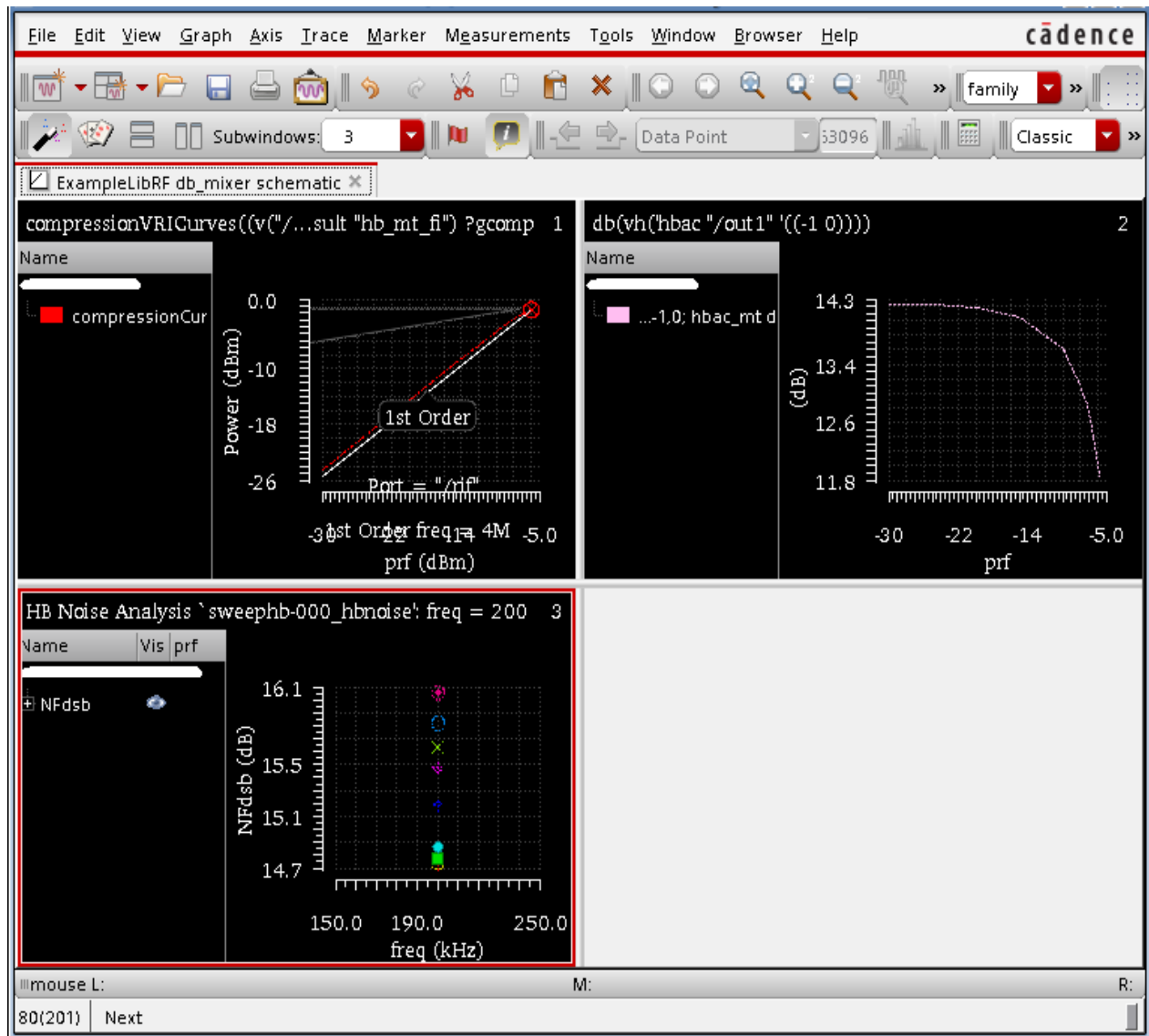
**Figure 4-65 Direct Plot Form Plotting NFdsb**

The image shows the 'Direct Plot Form' dialog box. At the top, the title bar says 'Direct Plot Form'. Below it, 'Plotting Mode' is set to 'NewSubWin'. The 'Analysis' section has four radio buttons: 'tstab', 'hb\_mt', 'hbac\_mt', and 'hbnoise\_mt'. The 'hbnoise\_mt' button is selected and highlighted with a red rectangle. The 'Function' section has eight radio buttons: 'Output Noise', 'Input Noise', 'Noise Figure', 'Noise Factor', 'NFdsb', 'Fdsb', 'NFeee', 'Feee', and 'Transfer Function'. The 'NFdsb' button is selected and highlighted with a red rectangle. Below the function section, there is a 'Description' field with the text 'Double Sideband Noise Figure'. Below that, a status message says 'Currently, only variable data is available'. There are three checkboxes: 'Integrated Over Bandwidth', 'Loadpull Contours', and 'Add To Outputs', all of which are currently unchecked. At the bottom right, there is a 'Plot' button. At the very bottom, there are three buttons: 'OK' (highlighted in red), 'Cancel', and 'Help'. A note at the bottom left says '> Press plot button on this form...'

**18.** Click *Plot*.

The noise figure plot is added to the previous plot in a new sub window, as shown below.

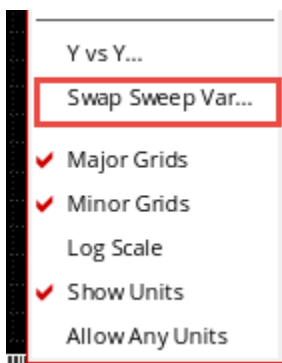
**Figure 4-66 1dB Compression Point, Conversion Gain, and Double Sided Noise Figure**



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.

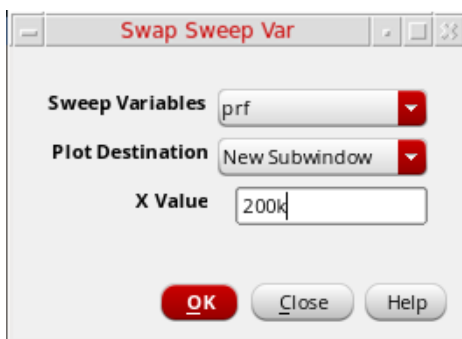
**Figure 4-67 X-Axis Properties Form**



The Swap Sweep Var form is displayed.

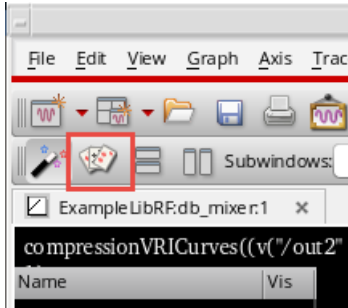
20. In the *Swap Sweep Var* form, select *prf* from the *Sweep Variable* drop-down list. Leave *Plot Destination* set to *New Subwindow*. Enter 200k in the *X Value* field. Note that for this menu, you must enter k, not K.

**Figure 4-68 Swap Sweep Variable Form**



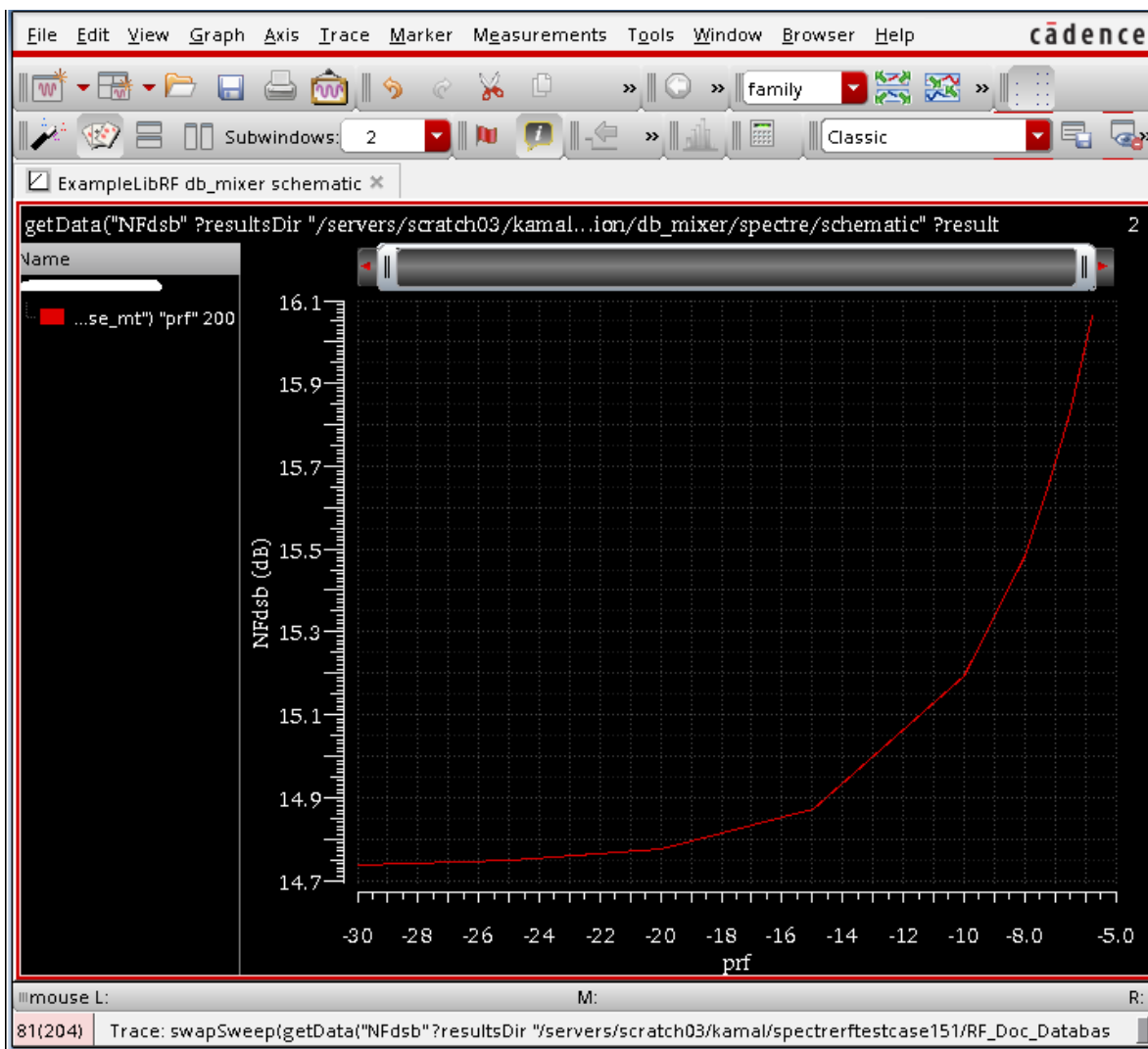
21. Click *OK*.
22. The waveform window is now fairly difficult to read. In the waveform window, select *Card* icon from the *Graph* toolbar. The *Card* mode displays one subwindow at a time so that the display area becomes larger. You can switch which subwindow is displayed at any time.

**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.

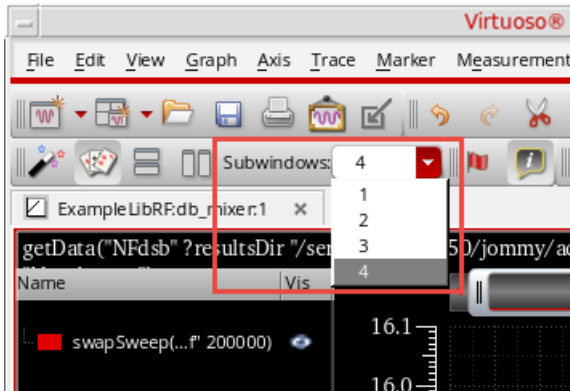
Figure 4-70 Noise Figure



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:

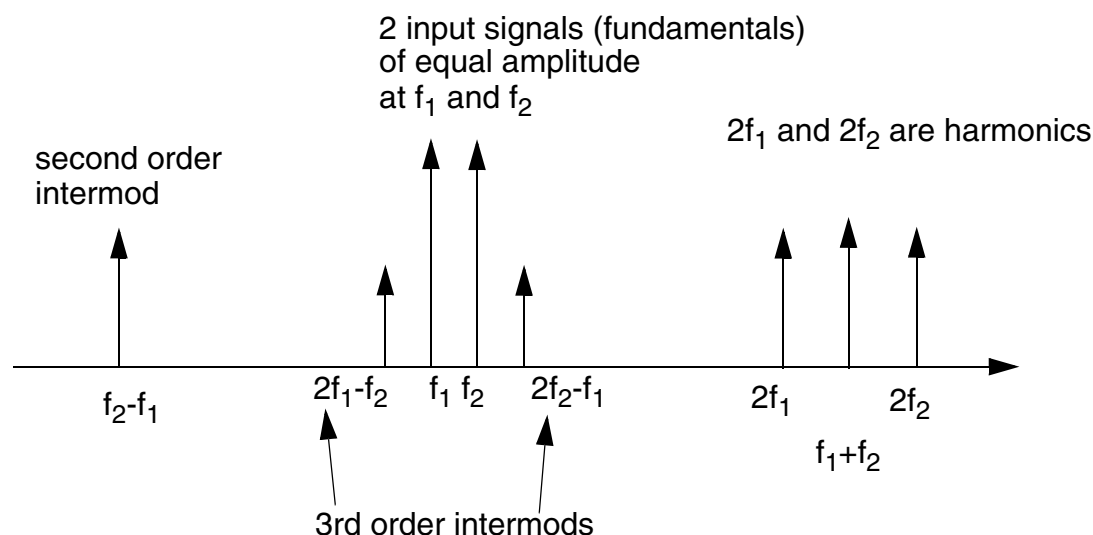
In this section, you measured 1dB compression point, conversion gain, and desensitization with an RF blocking signal present using hb, hbnoise, and hbac analyses.

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

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

---

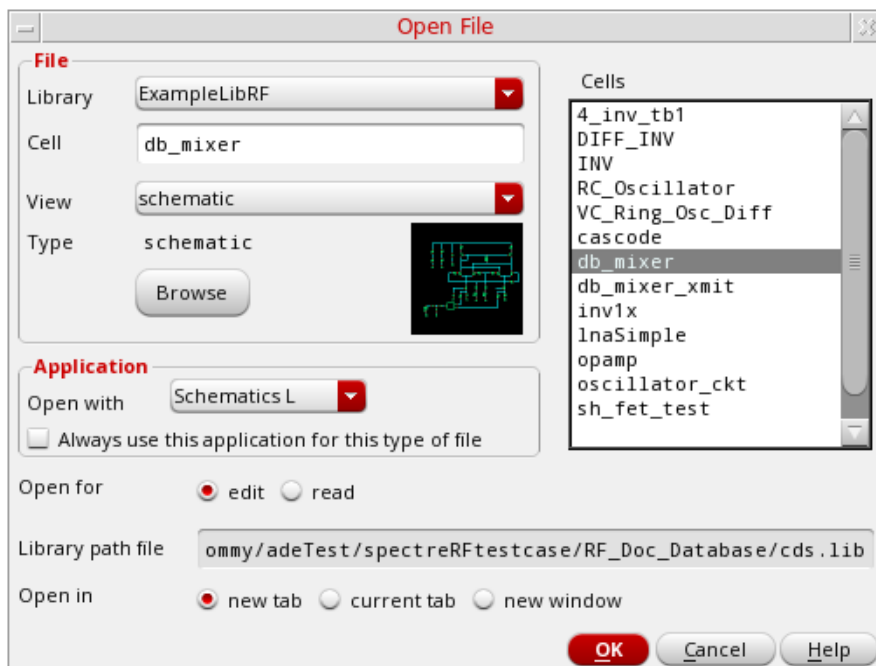
1. In the CIW, choose *File – Open*.

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**



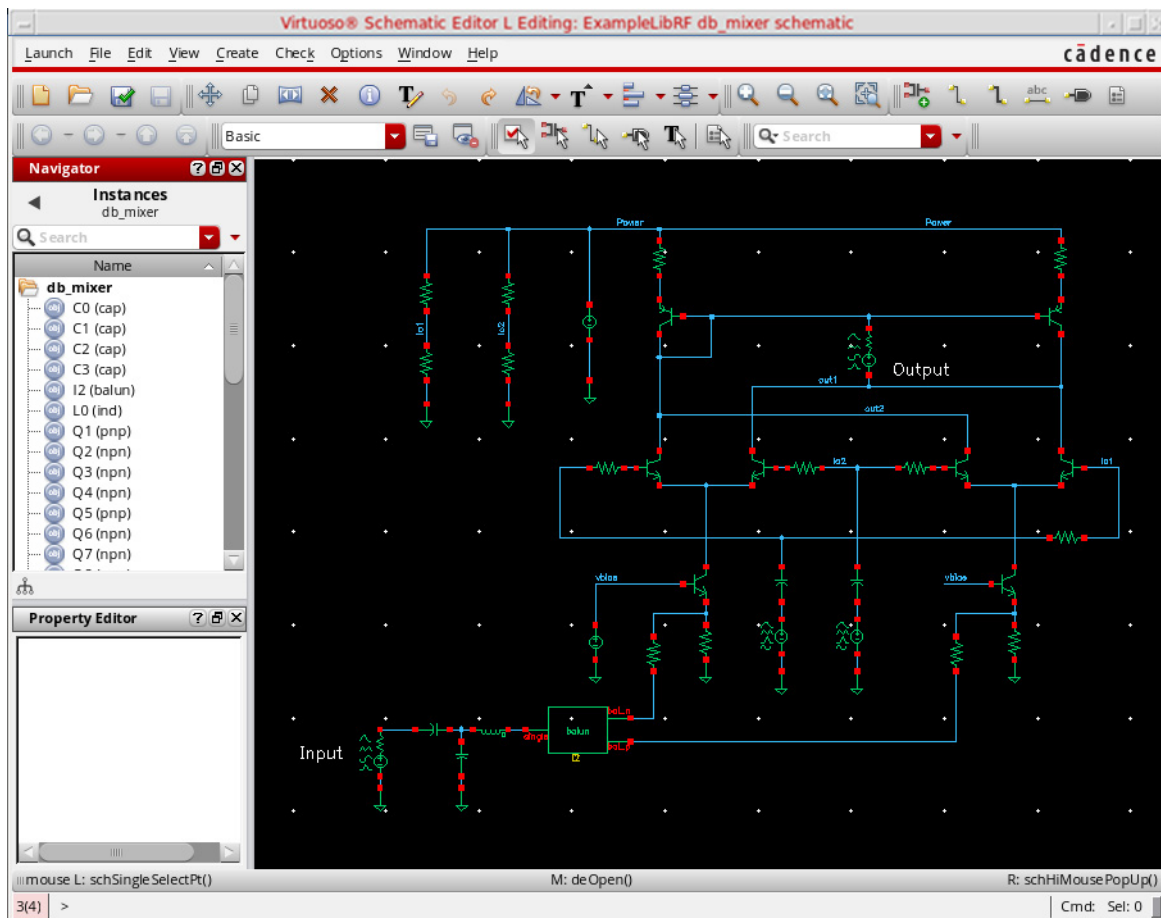
4. Click *OK*.
5. Click *OK* in the Create new ADE Explorer view form.

The Schematic window for the *db\_mixer* mixer is displayed.



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

Figure 4-74 db\_mixer Schematic

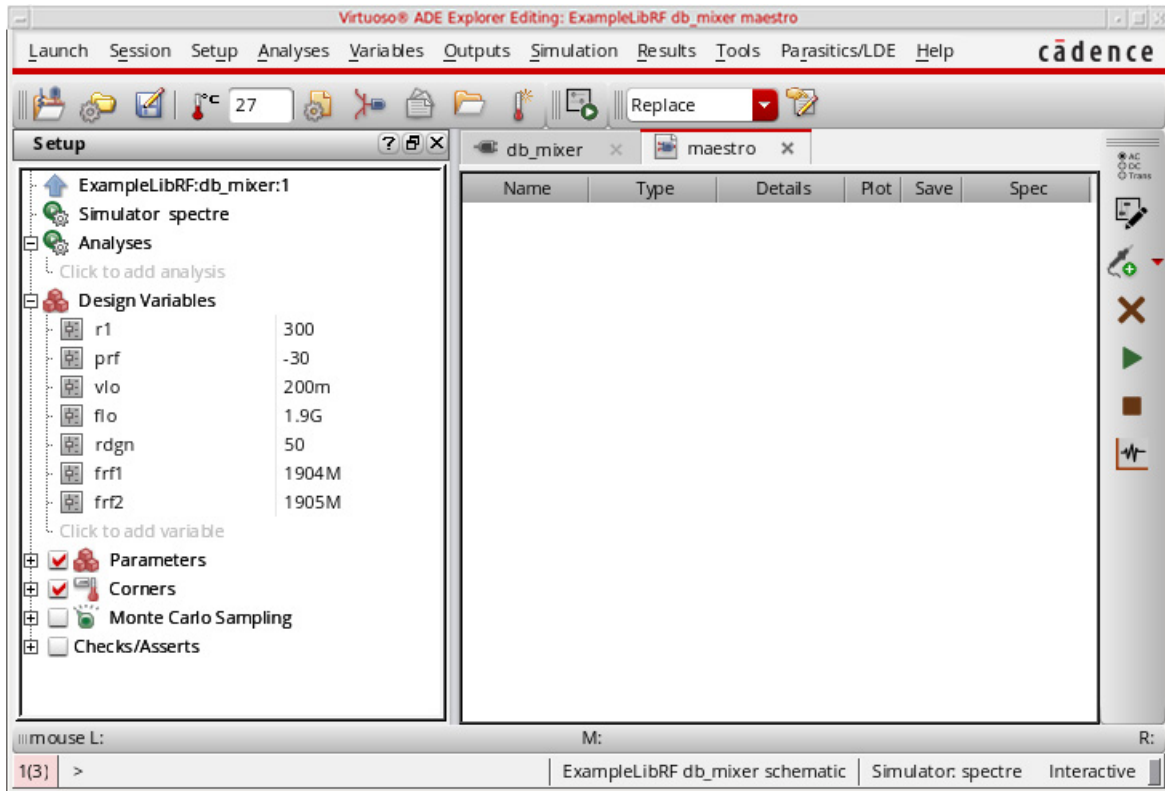


6. In the Schematic window, choose *Launch – ADE Explorer*.

ADE Explorer is displayed, as shown below.

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

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

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

---

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](#).

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

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**

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvi ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

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

---

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.

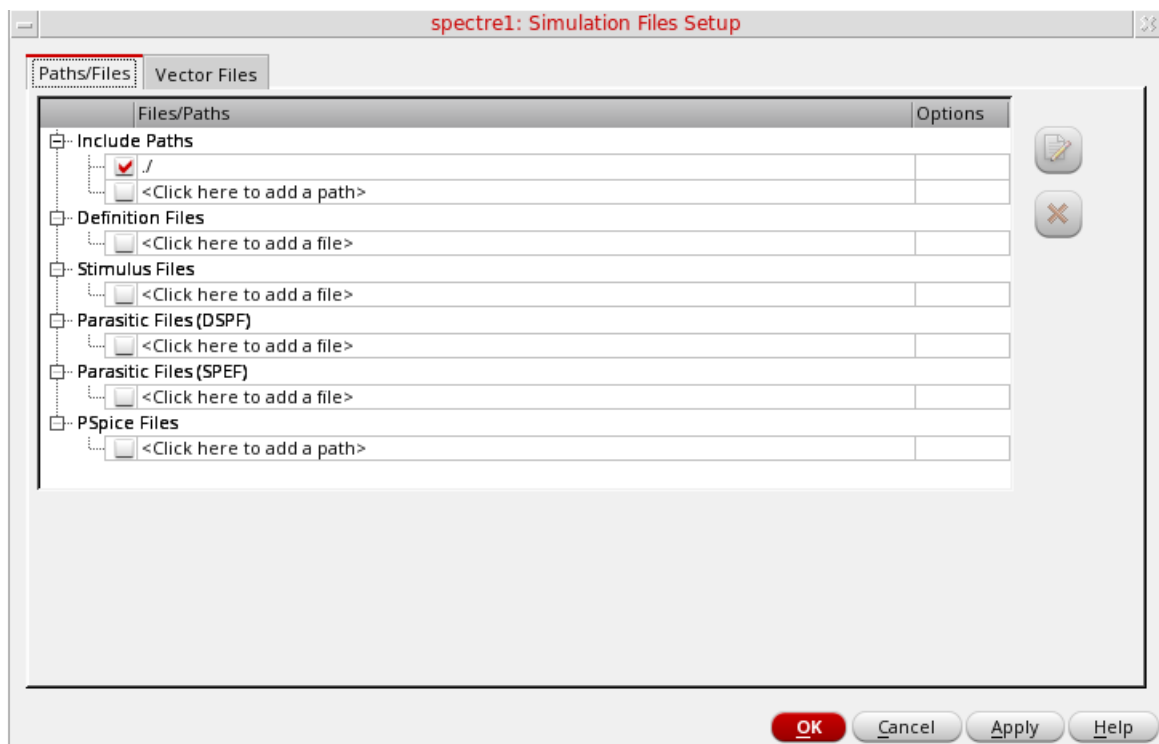
In the *Select signals to output* section, make sure that *allpub* is highlighted. This is the default which saves all node voltages at all levels of hierarchy, but it does not include the node voltages inside the device models.

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
```

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

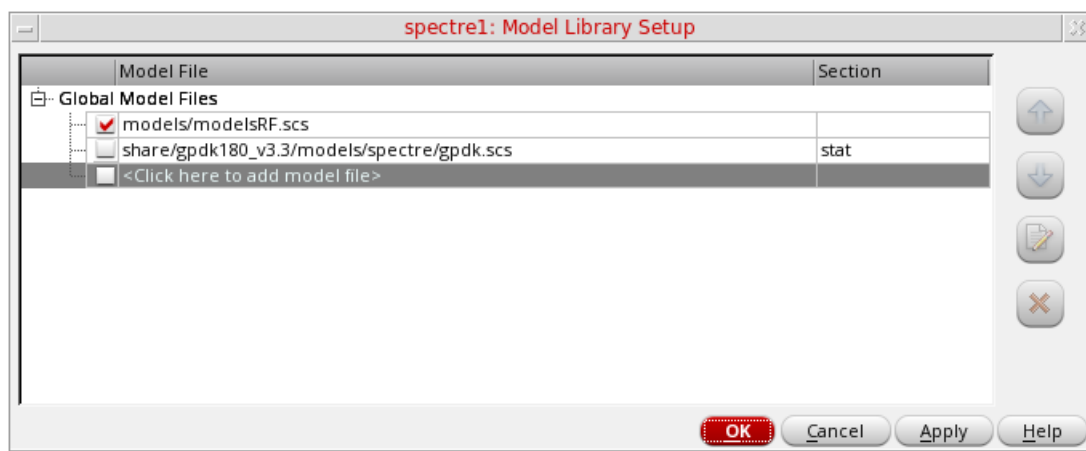
---

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**



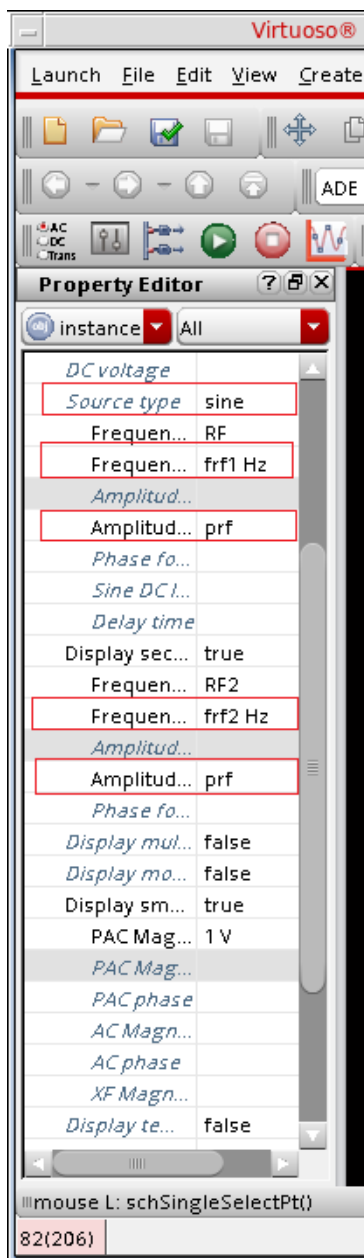
7. Click *OK* to close the *Model Library Setup* form.

### 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*.

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

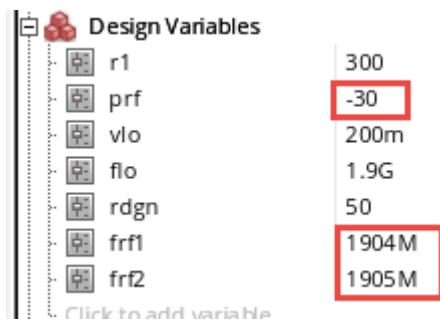
---

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.

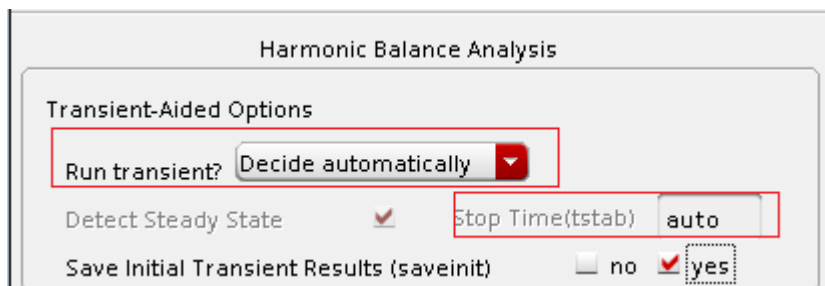
When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide 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**

The screenshot shows the 'Tones' section of a software interface. At the top, there are three radio buttons: 'Tones' (selected), 'Frequencies', and 'Names'. Below this is a table with the following data:

| # | Name | Expr | Value  | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|--------|-------|-------|-------|-------|
| 1 | LO   | flo  | 1.9G   | auto  | 1     | yes   | V0    |
| 2 | LO   | flo  | 1.9G   | auto  | 1     | yes   | V0    |
| 3 | RF   | frf1 | 1.904G | 3     | 1     | no    | rf    |

Below the table, there are input fields for 'LO', 'flo', '1.9G', 'auto', '1', 'yes', and 'V0'. The 'auto' field is highlighted with a red box. Below these fields are three buttons: 'Change' (highlighted with a red box), 'Delete', and 'Update From Hierarchy'. At the bottom, there is a label 'Freqdivide Ratio for tone with Tstab' and a value of '1'.

- 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).
- 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*.
- 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*

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

---

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**

Accuracy Defaults (errpreset)  
☒ conservative ☐ moderate ☐ liberal

Oscillator ☐

Sweep ☒ 1 ☒  
Variable

Frequency Variable? ☒ no ☐ yes  
Variable Name

Sweep Range  
☒ Start-Stop  
☐ Center-Span

Sweep Type  
☒ Linear  
☐ Logarithmic

Select Design Variable  
r1  
prf  
v1o  
rdgn

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

Harmonic Balance Analysis

Transient-Aided Options

Run transient?

Detect Steady State ☒ Stop Time(tstab)

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ Names

Tones

| # | Name | Expr | Value  | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|--------|-------|-------|-------|-------|
| 3 | RF   | frf1 | 1.984G | 3     | 1     | no    | rf    |
| 4 | RF2  | frf2 | 1.985G | 3     | 1     | no    | rf    |

RF2 frf2 1.985G 3 1 no rf

Freqdivide Ratio for tone with Tstab

Harmonics

Accuracy Defaults (errpreset)

☒ conservative ☐ moderate ☐ liberal

Oscillator ☐

Sweep  ☒

Variable

Frequency Variable? ☒ no ☐ yes

Variable Name

Sweep Range

☒ Start-Stop ☐ Center-Span

Start  Stop

Sweep Type

☒ Linear ☐ Logarithmic

☒ Step Size ☐ Number of Steps

23. Click **OK**.

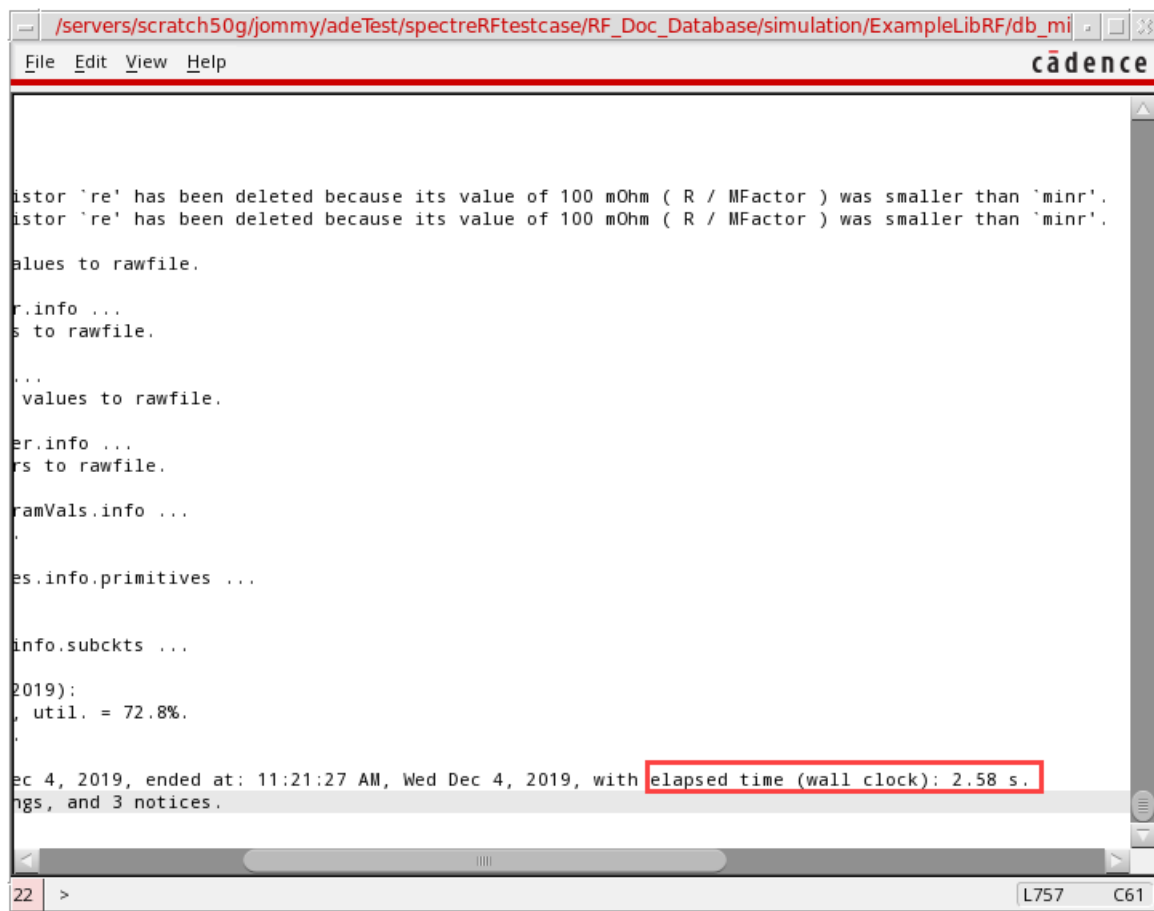
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

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-down, choose *Net (Specify R)*.
29. Leave the *Resistance* set to the default value of 50.

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

---

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**

**Direct Plot Form**

Plotting Mode: Append

**Analysis**

☐ tstab ☒ hb\_mt

**Function**

☐ Voltage ☐ Current

☐ Power ☐ Voltage Gain

☐ Current Gain ☐ Power Gain

☐ Transconductance ☐ Transimpedance

☐ Compression Point ☒ IPN Curves

☐ Power Contours ☐ Reflection Contours

☐ Power Added Eff. ☐ Power Gain Vs Pout

☐ Comp. Vs Pout ☐ Node Complex Imp.

Select: Net ( specify R )

Resistance (Default is 50.):

Circuit Input Power: ☐ Single Point ☒ Variable Sweep ("prf")

"prf" ranges from -40 to -20

Input Power Extrapolation Point (dBm):

(Defaults to -40)

Input Referred IP3: Order 3rd

|     | Freq. (Hz) | L0 | RF | RF2 |
|-----|------------|----|----|-----|
| 3rd | 2M         | -1 | 3  | -2  |
|     | 3M         | -1 | 2  | -1  |
|     | 3M         | 0  | -3 | 3   |
|     | 4M         | -1 | 1  | 0   |
|     | 5M         | -1 | 0  | 1   |
| 1st | 2M         | 0  | -2 | 2   |
|     | 2M         | -1 | 3  | -2  |
|     | 3M         | -1 | 2  | -1  |
|     | 3M         | 0  | -3 | 3   |
|     | 4M         | -1 | 1  | 0   |

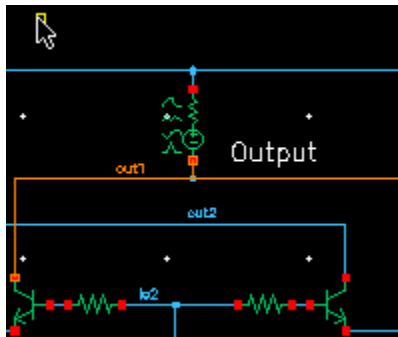
Add To Outputs: ☐

> Select Net on schematic...

OK Cancel Help

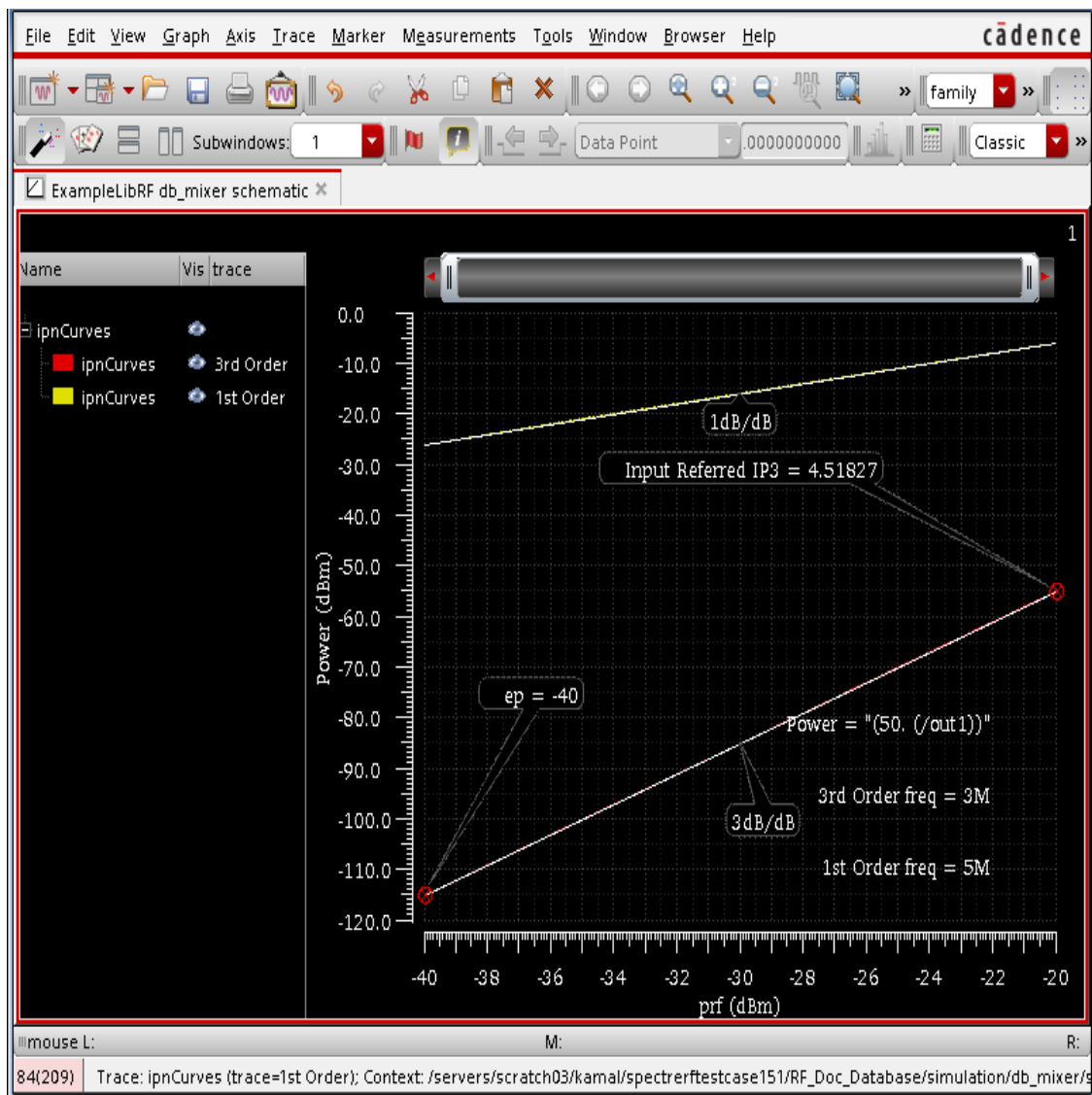
35. Click the *out1* net on the schematic just below the *Output* label, as shown below. Then press the *Esc* key, with the mouse cursor in the schematic window. This closes the *Direct Plot Form*.

**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**

☐ Compression Point   ☒ IPN Curves  
☐ Power Contours   ☐ Reflection Contours  
☐ Power Added Eff.   ☐ Power Gain Vs Pout  
☐ Comp. Vs Pout   ☐ Node Complex Imp.

Select Net ( specify R )

Resistance (Default is 50.)

Circuit Input Power   ☐ Single Point  
☒ Variable Sweep ("prf")

"prf" ranges from -40 to -20  
 Input Power Extrapolation Point (dBm)   
 (Defaults to -40)

Input Referred IP2   Order 2nd

|                    | Freq. (Hz) | L0 | RF | RF2 |
|--------------------|------------|----|----|-----|
| 2nd Order Harmonic | 0          | 0  | 0  | 0   |
|                    | 1M         | 0  | -1 | 1   |
|                    | 2M         | 0  | -2 | 2   |
|                    | 2M         | -1 | 3  | -2  |
|                    | 3M         | -1 | 2  | -1  |
| 1st Order Harmonic | 3M         | 0  | -3 | 3   |
|                    | 2M         | 0  | -2 | 2   |
|                    | 2M         | -1 | 3  | -2  |
|                    | 3M         | -1 | 2  | -1  |
|                    | 3M         | 0  | -3 | 3   |
| 1st Order Harmonic | 4M         | -1 | 1  | 0   |
|                    | 5M         | -1 | 0  | 1   |

> Select Net on schematic...

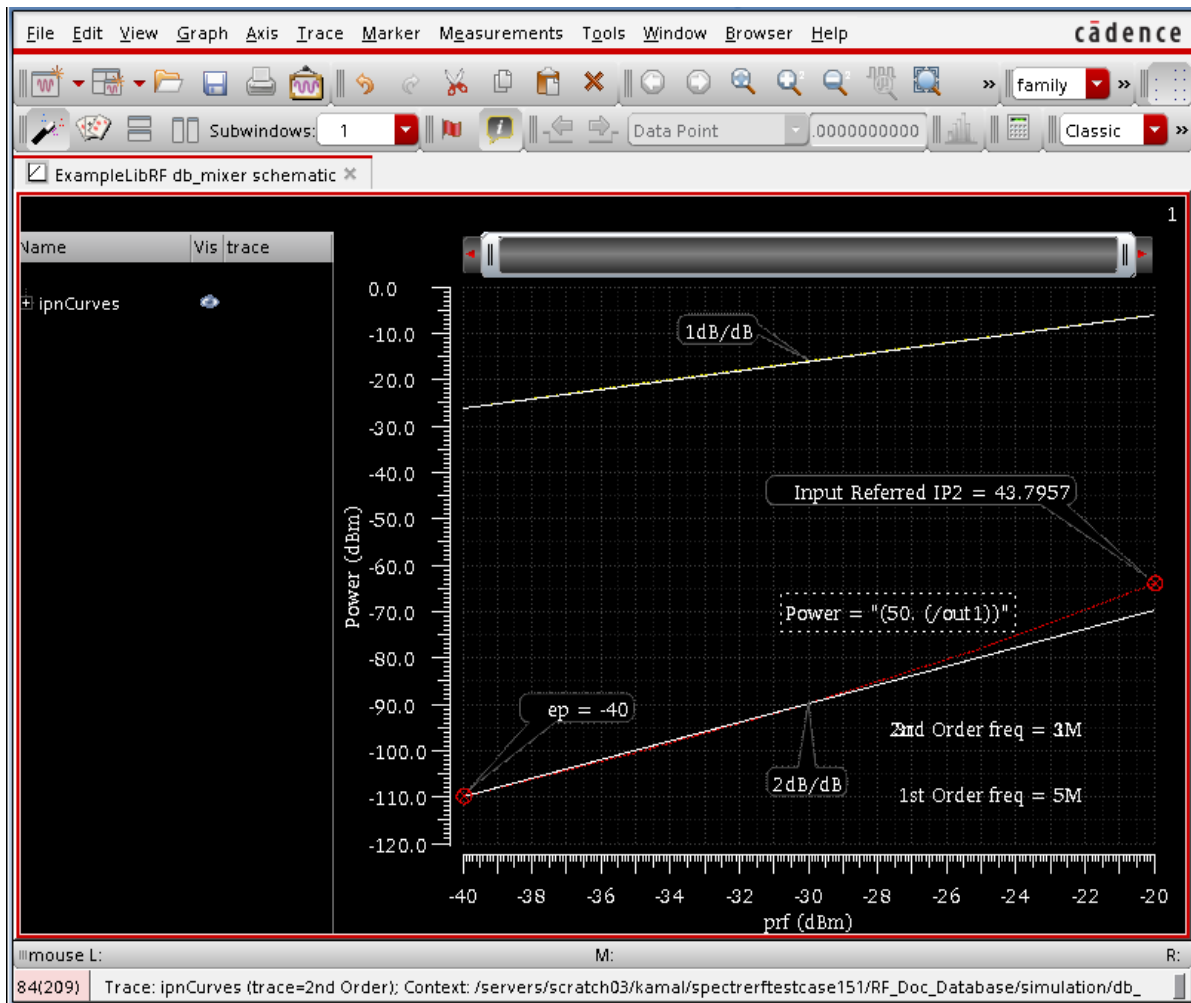
## 6. Click *Replot*.

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

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

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.

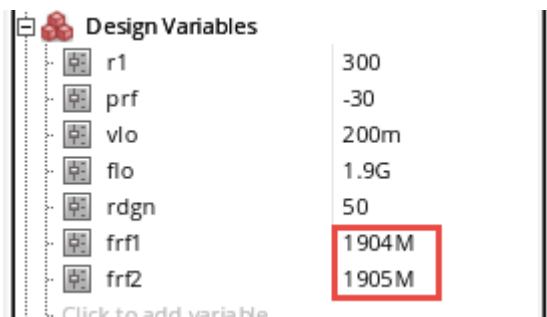
### Rapid IP2 and IP3 Measurement

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.

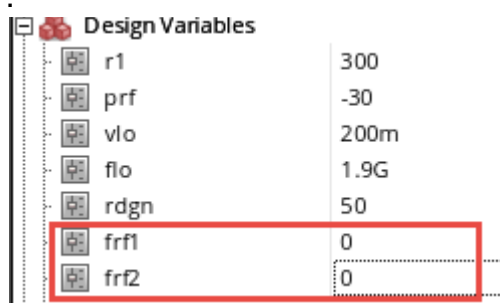
**Figure 4-94 Design Variables Section of ADE Simulation Window**



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**



| Design Variables |      |
|------------------|------|
| r1               | 300  |
| prf              | -30  |
| vlo              | 200m |
| flo              | 1.9G |
| rdgn             | 50   |
| frf1             | 0    |
| frf2             | 0    |

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:



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

Figure 4-96 Choosing Analyses Form for hb Analysis

The dialog box 'Choosing Analyses -- ADE Explorer' is shown. It contains several sections for configuring the analysis:

- Analysis Selection:** Radio buttons for `qnoise`, `qpxf`, `qp sp`, `hb` (selected), `hbac`, `hbstb`, `hbnoise`, `hb sp`, and `hbxf`.
- Harmonic Balance Analysis:**
  - Transient-Aided Options:**
    - Run transient?: `Decide automatically` (dropdown)
    - Detect Steady State: ☒ (checked)
    - Stop Time(tstab): `auto` (button)
    - Save Initial Transient Results (saveinit): ☐ no, ☒ yes
  - Dynamic Parameter: ☐
  - Tones: ☐ Frequencies, ☒ Names
  - Tones Table:**

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 2 | L0   | f1o  | 1.9G  | auto  | 1     | yes   | V1    |
| 1 | L0   | f1o  | 1.9G  | auto  | 1     | yes   | V0    |
  - RF2 Section:** `fr f2`, `1.905G`, `3`, `1`, `no` (dropdown), `rf`. Buttons: `Change`, `Delete`, `Update From Hierarchy`.
  - Freqdivide Ratio for tone with Tstab: `1` (input field)
  - Harmonics: `Default` (dropdown)
  - Accuracy Defaults (errpreset): ☐ conservative, ☒ moderate, ☐ liberal
  - Oscillator: ☐
  - Sweep: `1` (dropdown), ☒ (checkbox). Frequency Variable?: ☒ no, ☐ yes.
- Buttons:** `OK`, `Cancel`, `Defaults`, `Apply`, `Help`.

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* 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, 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.

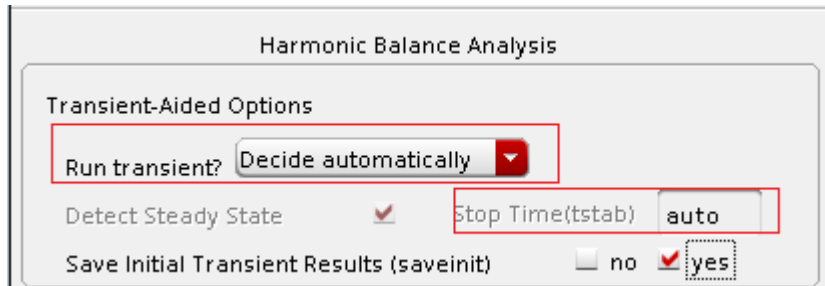
When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide 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:

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

Figure 4-98 Choosing Analyses Form - hb Setup for Rapid IP3

The screenshot shows the 'Choosing Analyses Form' for Harmonic Balance Analysis (hb). The 'hb' radio button is selected and highlighted with a red box. Below it, the 'Transient-Aided Options' section contains a 'Run transient?' dropdown menu set to 'Decide automatically' (highlighted with a red box), a 'Detect Steady State' checkbox checked, a 'Stop Time(tstab)' dropdown set to 'auto', and a 'Save Initial Transient Results (saveinit)' checkbox with 'yes' selected (highlighted with a red box). The 'Dynamic Parameter' checkbox is unchecked. The 'Tones' section has 'Frequencies' selected. The 'Number of Tones' section has '1' selected. The 'Fundamental Frequency' is set to '1.9G' (highlighted with a red box). The 'Number of Harmonics' is set to 'auto', and the 'Oversample Factor' is set to '1'. The 'Freqdivide Ratio for Tone 1' is set to '1'. The 'Harmonics' dropdown is set to 'Default'. The 'Accuracy Defaults (errpreset)' section has 'moderate' selected (highlighted with a red box). The 'Oscillator', 'Sweep', 'Loadpull', 'LSSP', and 'Compression' checkboxes are all unchecked. The 'Enabled' checkbox is checked. At the bottom, there are buttons for 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'.

pxf psp qpss qpac  
qpnoise qpxf qpss **hb**  
hbac hbnoise hbasp

Harmonic Balance Analysis

Transient-Aided Options

Run transient? **Decide automatically**  
Detect Steady State ☒ Stop Time(tstab) auto  
Save Initial Transient Results (saveinit) ☐ no ☒ **yes**

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

**Tone 1**  
Fundamental Frequency **1.9G**  
Number of Harmonics auto  
Oversample Factor 1  
Freqdivide Ratio for Tone 1 1

Harmonics Default

Accuracy Defaults (errpreset)  
☐ conservative ☒ **moderate** ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☒

Options...

**OK** Cancel Defaults Apply Help

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

---

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**

The screenshot shows the 'Choosing Analyses -- ADE Explorer' dialog box. In the 'Analysis' section, the 'hbac' radio button is selected and highlighted with a red box. Below this, the 'Harmonic Balance AC Analysis' section contains several settings: 'Sweep type' is set to 'default', 'Sweep is currently absolute' is displayed, 'Input Frequency Sweep Range (Hz)' has 'Start-Stop' selected, 'Sweep Type' is 'Automatic', 'Add Specific Points' and 'Add Points By File' are unchecked, 'Sidebands' is 'Maximum sideband', 'When using hb engine, default value is harms of 1st tone.' is shown, 'No specialized analyses support for multi-tone HBAC analysis' is a note, 'Lowmem' is 'no', and 'Enabled' is unchecked. At the bottom, there are buttons for 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'.

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

---

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 measurements.
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

Choosing Analyses -- ADE Explorer

☐ qpnoise ☐ qpwt ☐ qpwp ☐ no

☒ hbac ☐ hbstb ☐ hbnoise ☐ hbsp

☐ hbxf

Harmonic Balance AC Analysis

Specialized Analyses

Rapid IP3

Source Type ☒ port ☐ isource ☐ vsource

Select Clear

Input Sources 1 /rf Freq 1.904G

Select Clear

Input Sources 2 /rf Freq 1.905G

Input Power (dBm) -40 Power 2 3

Frequency of IM Output Signal 3M

Frequency of Linear Output Signal 5M

Maximum Non-linear Harmonics

Output ☒ Voltage ☐ Current

Out+ /out1 Select

Out- Select

Enabled ☒ Options...

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.

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

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.

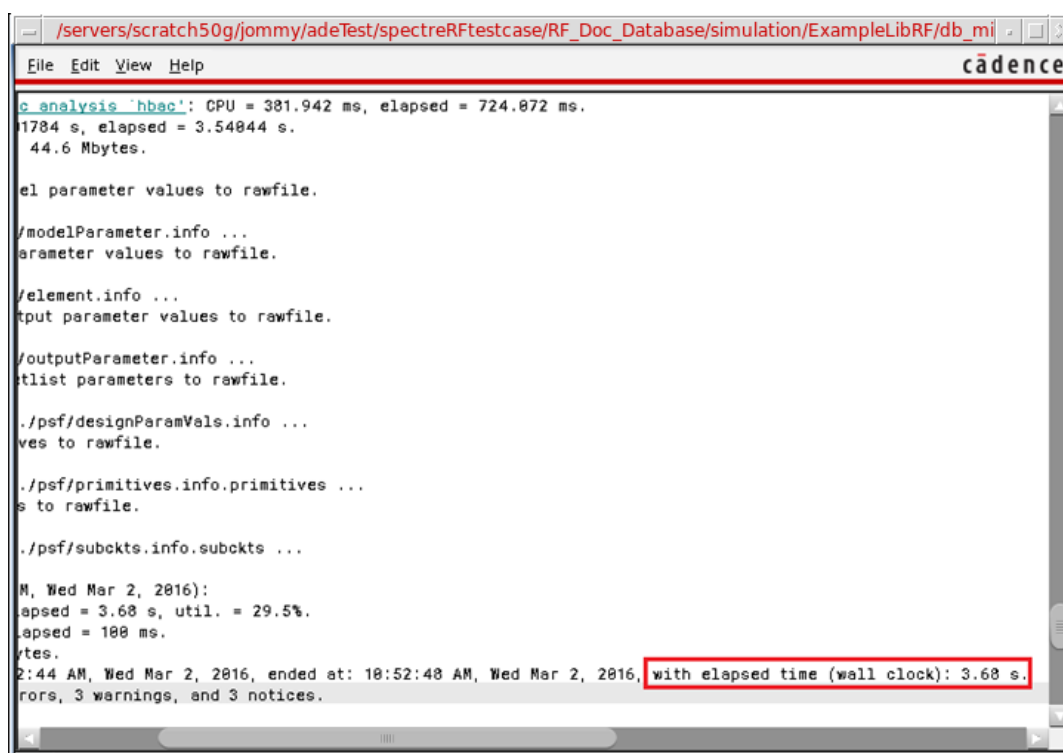
11. Click *OK* at the bottom of the *Choosing Analyses* form.

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**





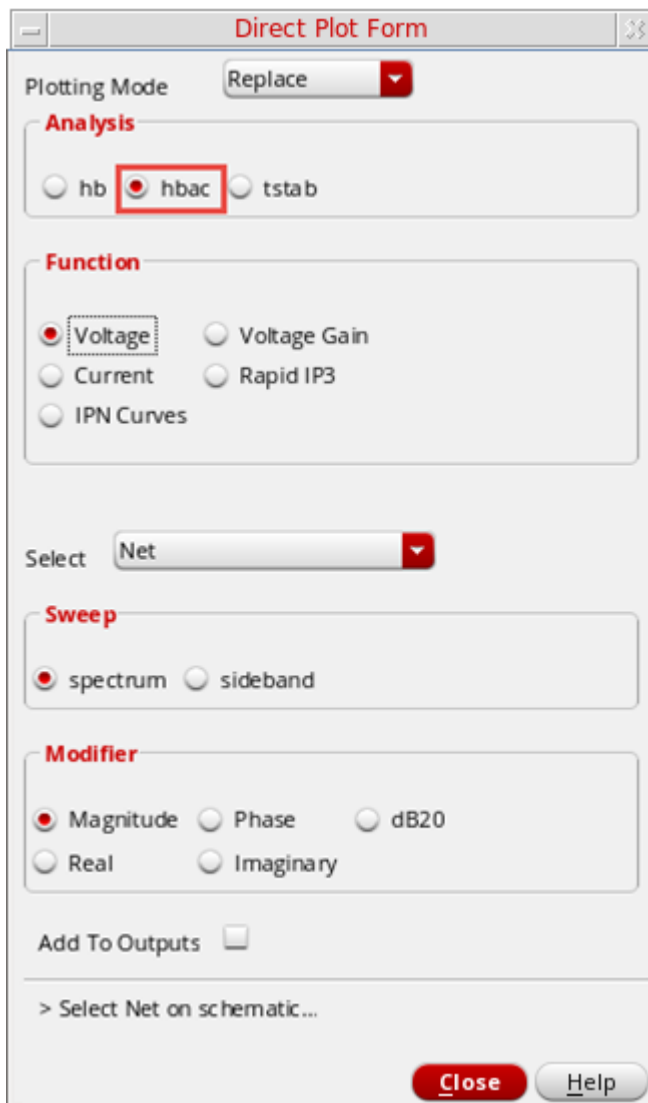
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

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**



The screenshot shows the 'Direct Plot Form' dialog box. At the top, the title bar reads 'Direct Plot Form'. Below it, the 'Plotting Mode' is set to 'Replace'. The 'Analysis' section has three radio buttons: 'hb', 'hbac' (which is selected and highlighted with a red box), and 'tstab'. The 'Function' section has five radio buttons: 'Voltage' (selected), 'Voltage Gain', 'Current', 'Rapid IP3', and 'IPN Curves'. Below this, the 'Select' dropdown menu is set to 'Net'. The 'Sweep' section has two radio buttons: 'spectrum' (selected) and 'sideband'. The 'Modifier' section has five radio buttons: 'Magnitude' (selected), 'Phase', 'dB20', 'Real', and 'Imaginary'. At the bottom, there is an 'Add To Outputs' checkbox which is unchecked, and a text prompt '> Select Net on schematic...'. The dialog box has 'Close' and 'Help' buttons at the bottom right.

16. In the *Function* section, select *Rapid IP3*.  
Leave the *Resistance* set to the default.

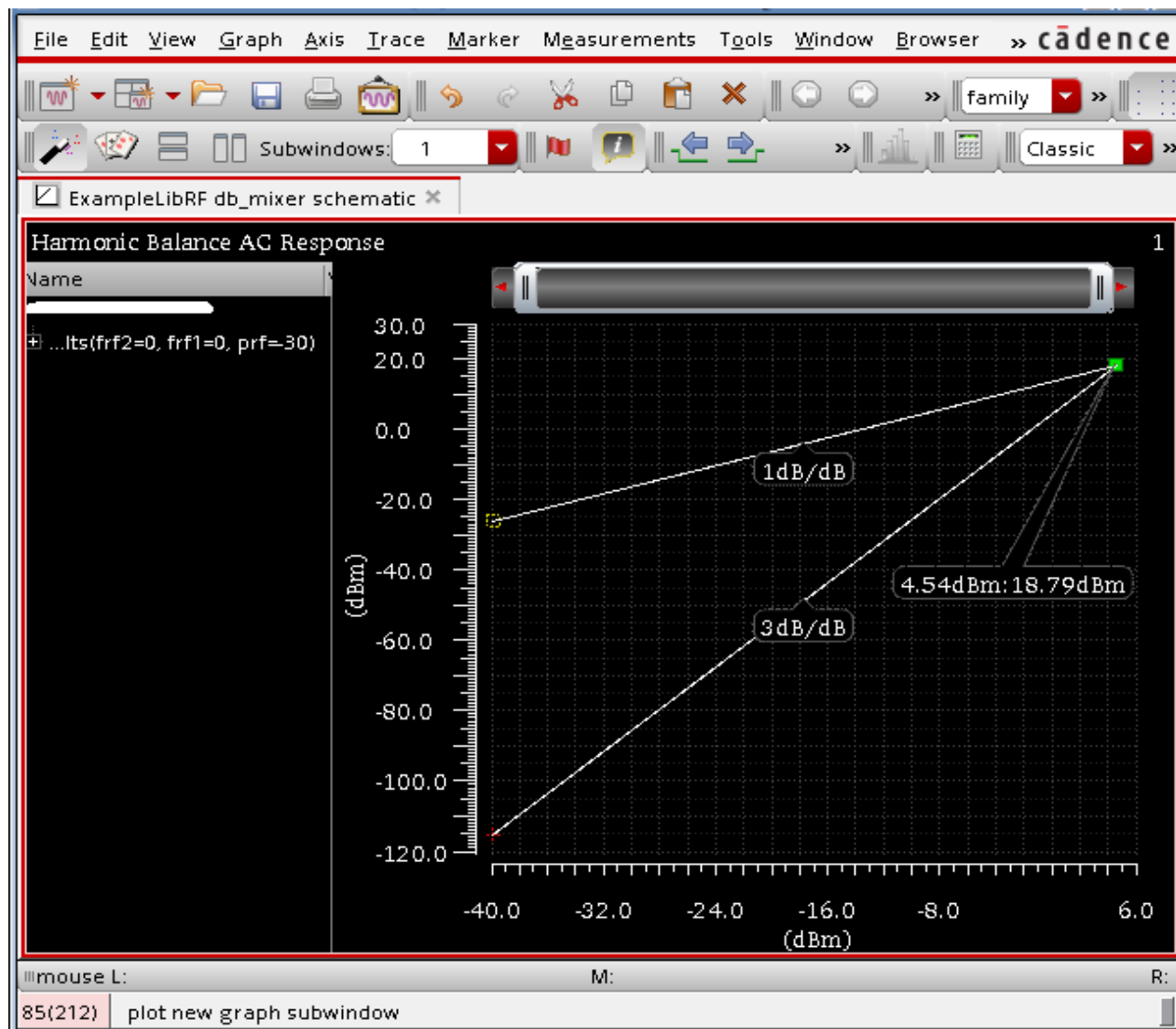
The *Direct Plot Form* should look like the following:

**Figure 4-103** hbac Direct Plot Form for Rapid IP3

The screenshot shows the 'Direct Plot Form' dialog box. It has a title bar with the text 'Direct Plot Form'. Inside, there is a 'Plotting Mode' dropdown menu set to 'Replace'. Below this is a section titled 'Analysis' with three radio buttons: 'hb', 'hbac' (which is selected), and 'tstab'. Another section titled 'Function' contains four radio buttons: 'Voltage', 'Current', 'IPN Curves', and 'Rapid IP3' (which is selected and highlighted with a red rectangle). Below the 'Function' section is a 'Resistance (Default is 50.)' text box. Further down is a 'Loadpull Contours' checkbox, which is unchecked. At the bottom left is an 'Add To Outputs' checkbox, also unchecked. To the right of this is a 'Plot' button. Below these options is a line of text: '> Press plot button on this form...'. At the very bottom are two buttons: 'Close' (in red) and 'Help'.

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

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

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 image shows a screenshot of the 'Harmonic Balance AC Analysis' dialog box. The 'Specialized Analyses' section is highlighted with a red box, and 'Rapid IP2' is selected from the dropdown menu. Below this, the 'Source Type' is set to 'port'. The 'Input Sources 1' and 'Input Sources 2' are both set to '/rf', with frequencies of 1.984G and 1.985G respectively. The 'Input Power (dBm)' is set to -40. The 'Frequency of IM Output Signal' is set to 1M, and the 'Frequency of Linear Output Signal' is set to 5M. The 'Maximum Non-linear Harmonics' is set to 3. The 'Output' section shows 'Voltage' selected, with 'Out+' set to '/out1' and 'Out-' set to an empty field. The 'Select' buttons are visible for each of these fields.

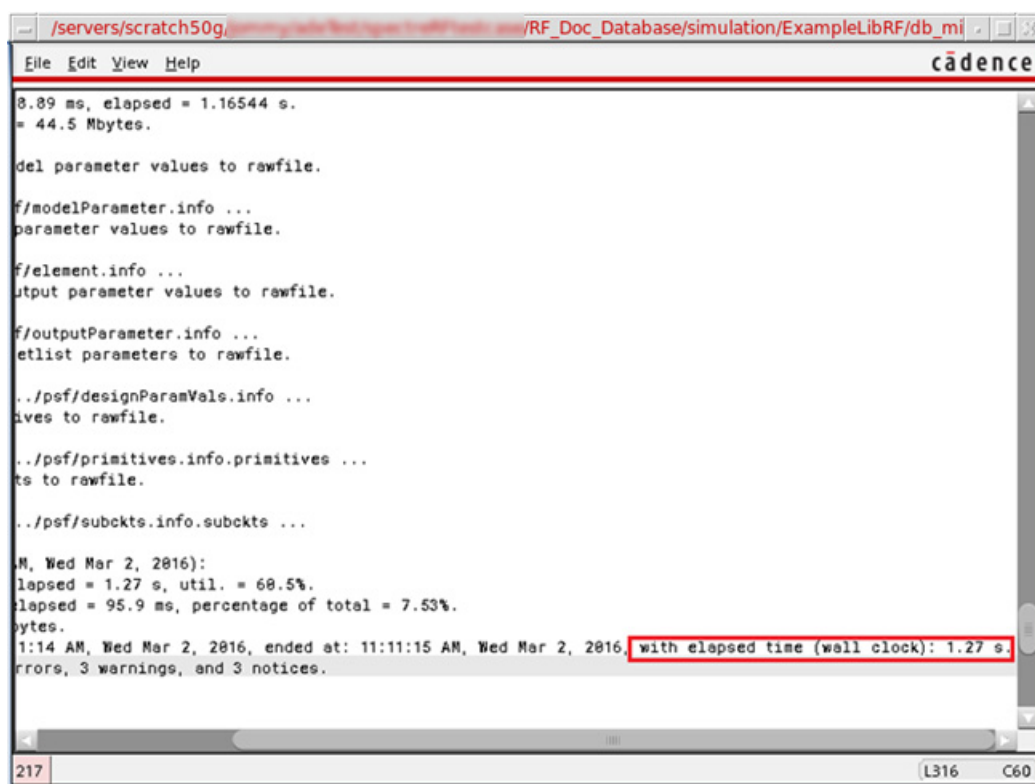
The form retains all the settings that were set for the rapid IP3. The only thing that needs to be changed is the frequency of the intermodulation product. The main down conversion frequencies are at 4MHz and 5MHz. The second order intermodulation product is at 1MHz. The frequencies to be calculated depend on the choice of frequencies at the input and the LO frequency. Individual choices are provided for the

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

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 is displayed 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*.

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

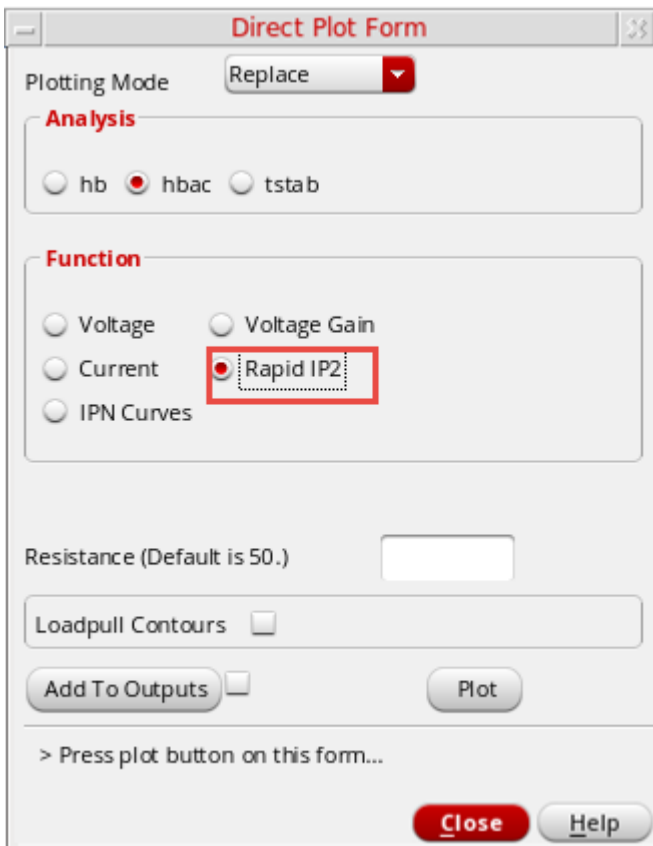
---

You may also invoke the Direct Plot Form by clicking the *Direct Plot* icon (  ) in the schematic.

9. In the *Direct Plot Form*, select *hvac* 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.

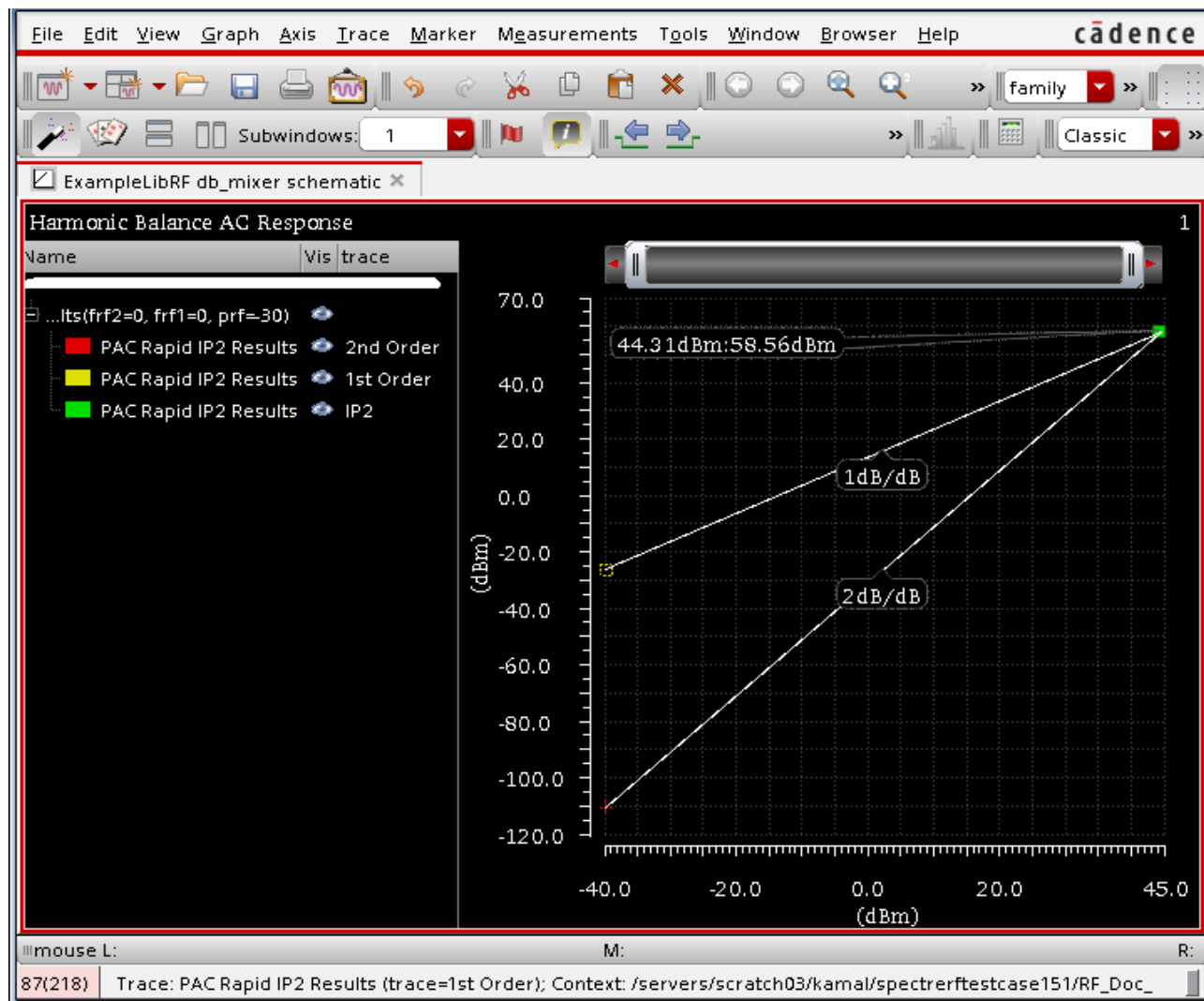
**Figure 4-107 Rapid IP2 Direct Plot Form**



The image shows the 'Direct Plot Form' dialog box. At the top, the title bar says 'Direct Plot Form'. Below it, 'Plotting Mode' is set to 'Replace'. The 'Analysis' section has three radio buttons: 'hb', 'hvac' (selected), and 'tstab'. The 'Function' section has five radio buttons: 'Voltage', 'Voltage Gain', 'Current', 'Rapid IP2' (selected and highlighted with a red box), and 'IPN Curves'. Below these is a 'Resistance (Default is 50.)' text box. There are checkboxes for 'Loadpull Contours' and 'Add To Outputs'. A 'Plot' button is located to the right of the 'Add To Outputs' checkbox. At the bottom, there is a red 'Close' button and a 'Help' button. A note at the bottom says '> Press plot button on this 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.

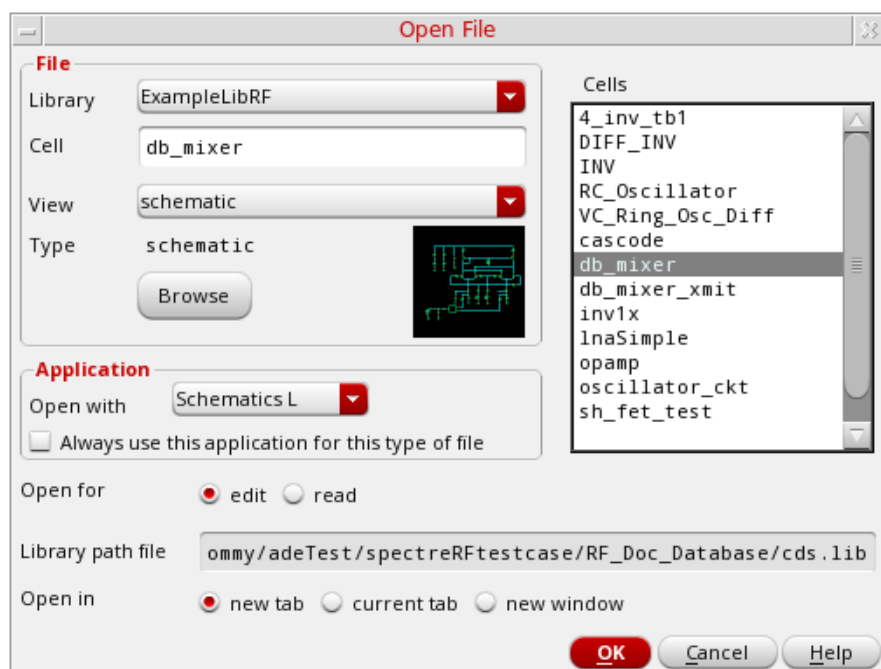


### Mixer Distortion Measurement

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*.

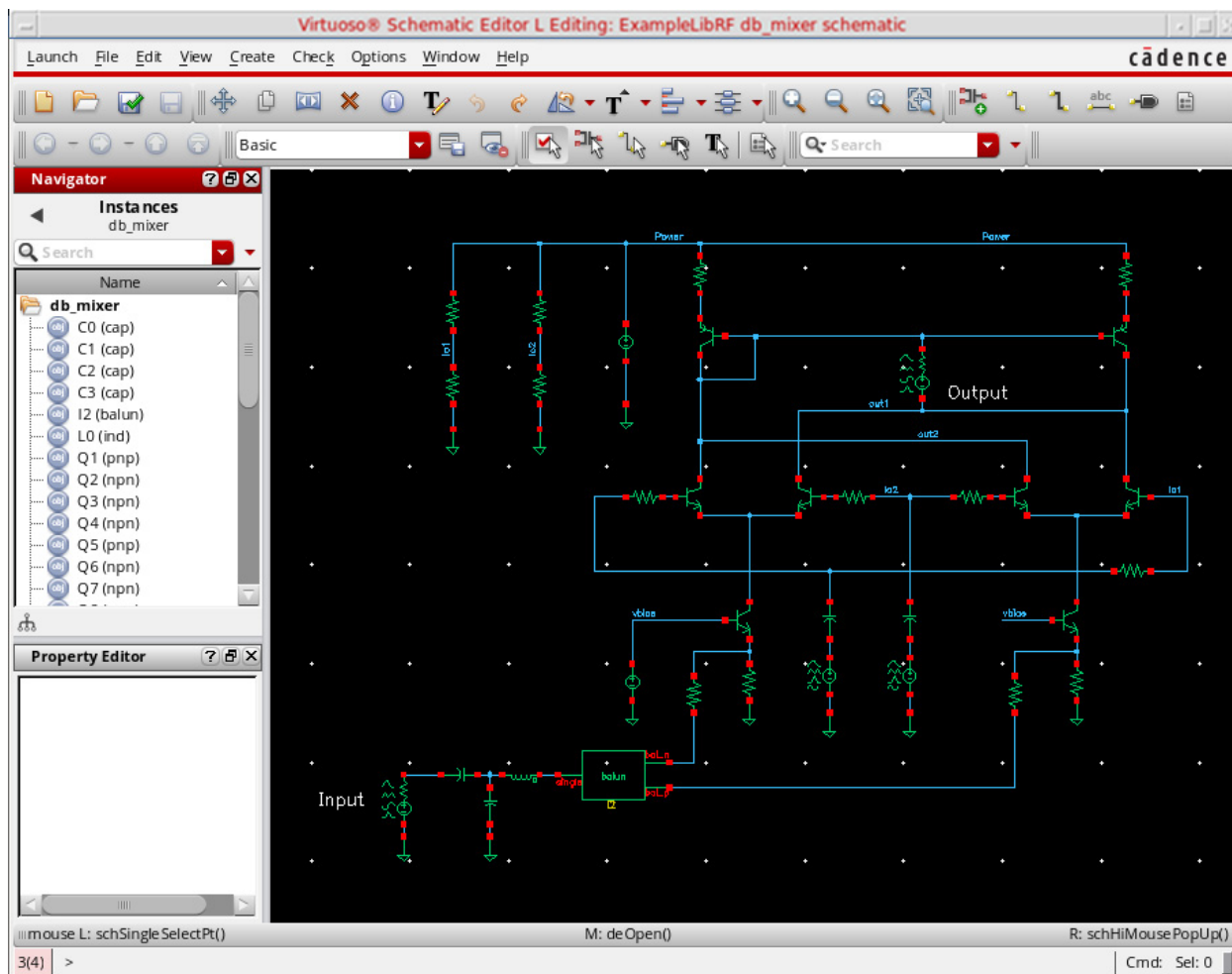
**Figure 4-109 Open File Form.**



2. Click *OK*. The *db\_mixer* schematic is displayed, as shown below.

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

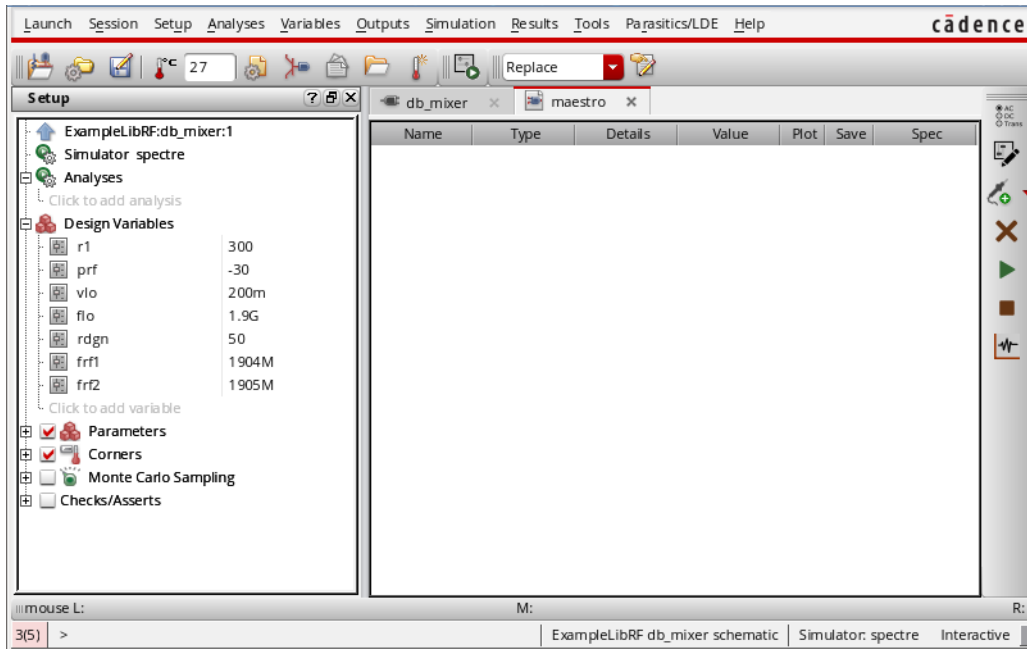
Figure 4-110 db\_mixer Schematic.



3. Open the Analog Design Environment from the schematic by selecting *Launch - ADE Explorer*. The simulation window is displayed, as shown below.

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

**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**



The image shows a dialog box titled "High-Performance Simulation Options". At the top, under "Simulation Performance Mode", there are four radio buttons: "Spectre", "APS" (which is selected), "Spectre X", and "XPS MS". Below this is a section titled "General" in red. It contains three sub-sections: "Accuracy + Speed" with an "Error Preset:" label and four radio buttons ("Do not override" is selected, followed by "Liberal", "Moderate", and "Conservative"); "APS Options" with a "Use ++aps:" label and an unchecked checkbox; and "Multi-Threading" with three radio buttons ("Auto" is selected, followed by "Disable" and "Manual"), a "# Threads:" label with a text input field, and an unchecked checkbox for "Processor affinity (e.g. 0-3 or 0,2,4,6):" with a corresponding text input field. Below the "General" section is a section titled "Post-Layout Settings" in red. It contains: "Enable Post-Layout Optimization:" with an unchecked checkbox; "Post-Layout Preset Mode:" with four radio buttons ("Default" is selected, followed by "High Precision Analog", "Ultra Precision Analog", "Legacy RCR", and "Legacy-RF RCR"); "Instance Preservation:" with two radio buttons ("None" is selected, followed by "Selected"); "Selected Instances:" with a text input field, "Select", and "Clear" buttons; and "Savefilter" with two checkboxes ("none" and "rc", both unchecked). At the bottom of the dialog are five buttons: "OK" (highlighted in red), "Cancel", "Defaults", "Apply", and "Help".

- 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*.

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

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

**Figure 4-114 Save Options Form**

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

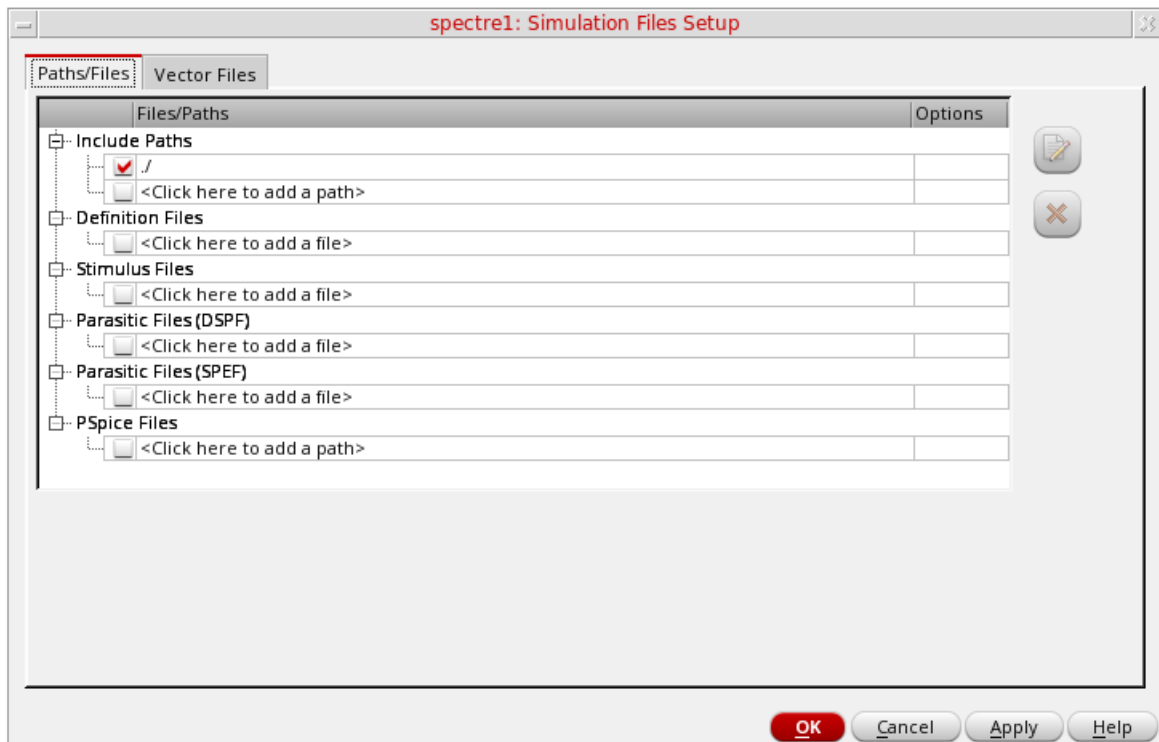
Use Fast Viewing Extensions ☐

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:

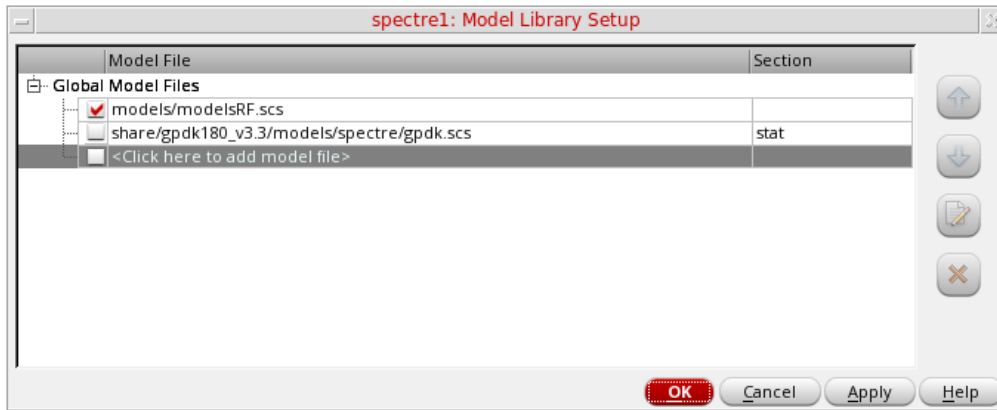
```
models/modelsRF.scs
```

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:

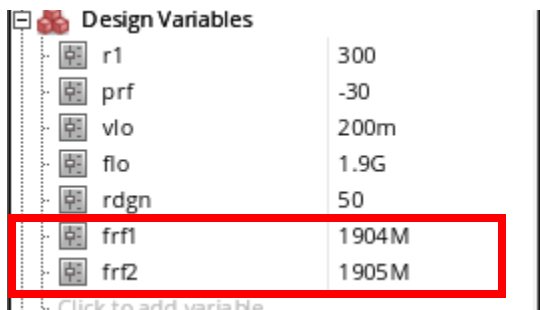
**Figure 4-116 Model Library Setup Form**



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

### Setting the Design Variables

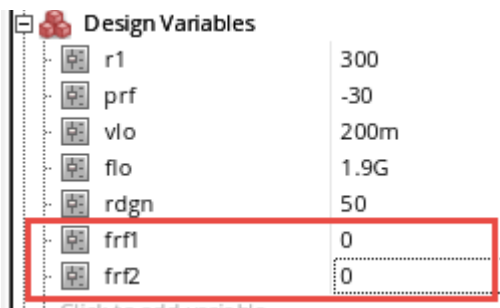
**Figure 4-117 Design Variables Section**



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**

**Choosing Analyses -- ADE Explorer**

☐ qpnoise
 ☐ qpxf
 ☐ qpss
 ☒ hb
 ☐ hbac
 ☐ hbstd
 ☐ hbnoise
 ☐ hbss
 ☐ hbxf

**Harmonic Balance Analysis**

**Transient-Aided Options**

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) **auto**

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter ☐

**Tones** ☐ Frequencies ☒ Names

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 2 | L0   | f1o  | 1.9G  | auto  | 1     | yes   | V1    |
| 1 | L0   | f1o  | 1.9G  | auto  | 1     | yes   | V0    |

RF2 **frf2** **1.905G** **3** **1** **no** **rf**

Change Delete Update From Hierarchy

Freqdivide Ratio for tone with Tstab **1**

Harmonics **Default**

**Accuracy Defaults (errpreset)**

☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

Sweep **1** ☒ Frequency Variable? ☒ no ☐ yes

Variable **Variable**

**OK** Cancel Defaults Apply Help

### 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.

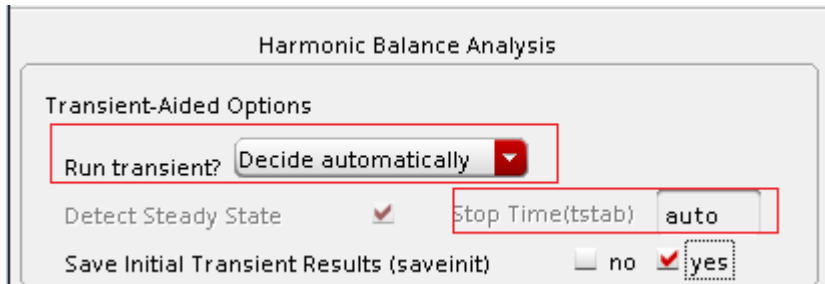
When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide 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 *Tone 1* 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**

pxf psp qpss qpac  
qpnoise qpxf qpssp **hb**  
hbac hbnoise hbbsp

Harmonic Balance Analysis

Transient-Aided Options

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☐ no ☒ **yes**

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

**Tone 1**

Fundamental Frequency **1.9G**

Number of Harmonics auto

Oversample Factor 1

Freqdivide Ratio for Tone 1 1

Harmonics Default

Accuracy Defaults (errpreset)

☐ conservative ☒ **moderate** ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☒

Options...

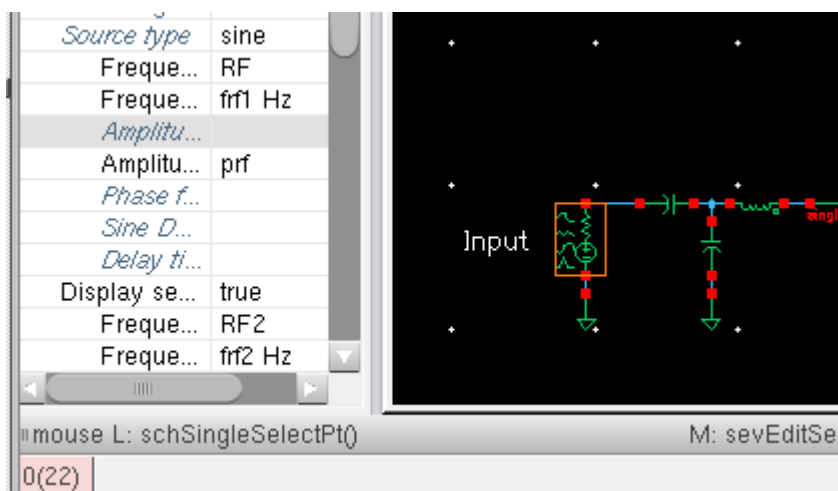
**OK** Cancel Defaults Apply Help

**10.** Click *Apply*.

11. In the Schematic window, set up the input port to perform a compression measurement.

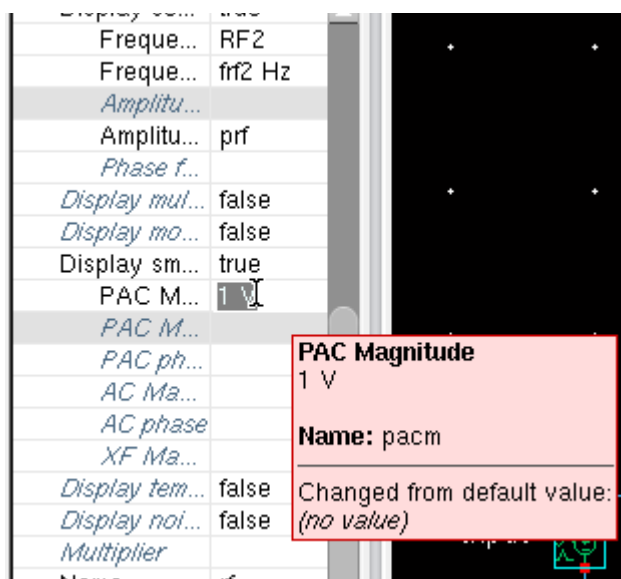
In the schematic, select the *Input* port. Once the input port is selected, the *Property Editor* on the left side of the schematic is populated with the properties of the selected instance, as shown below.

**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**



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

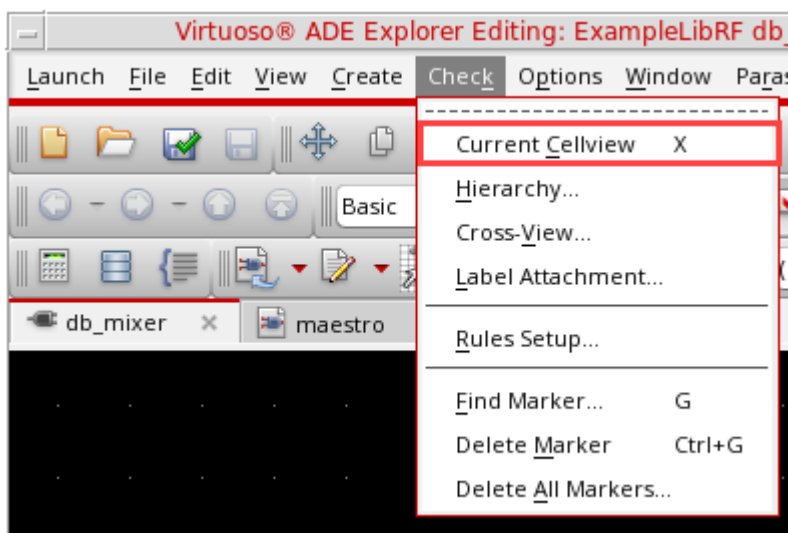
---

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**

The image shows a dialog box titled "Harmonic Balance AC Analysis". At the top, there are several radio buttons for different analysis types: qnoise, qpzf, qpzp, hb, hbstb, hbnoise, hbzp, and hbxf. The "hbac" radio button is selected and highlighted with a red box. Below this, the "Sweepertype" is set to "default" with a dropdown arrow, and a note says "Sweep is currently absolute". The "Input Frequency Sweep Range (Hz)" section has a "Start-Stop" dropdown, and "Start" and "Stop" input fields. The "Sweep Type" is set to "Automatic" with a dropdown arrow. There is a checkbox for "Add Specific Points". The "Sidebands" section has a "Maximum sideband" dropdown and an input field, with a note: "When using hb engine, default value is harms of 1st tone." The "Specialized Analyses" section has a dropdown menu currently showing "None", which is highlighted with a red box. At the bottom, there is an "Enabled" checkbox, an "Options..." button, and a row of buttons: "OK", "Cancel", "Defaults", "Apply", and "Help".

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.

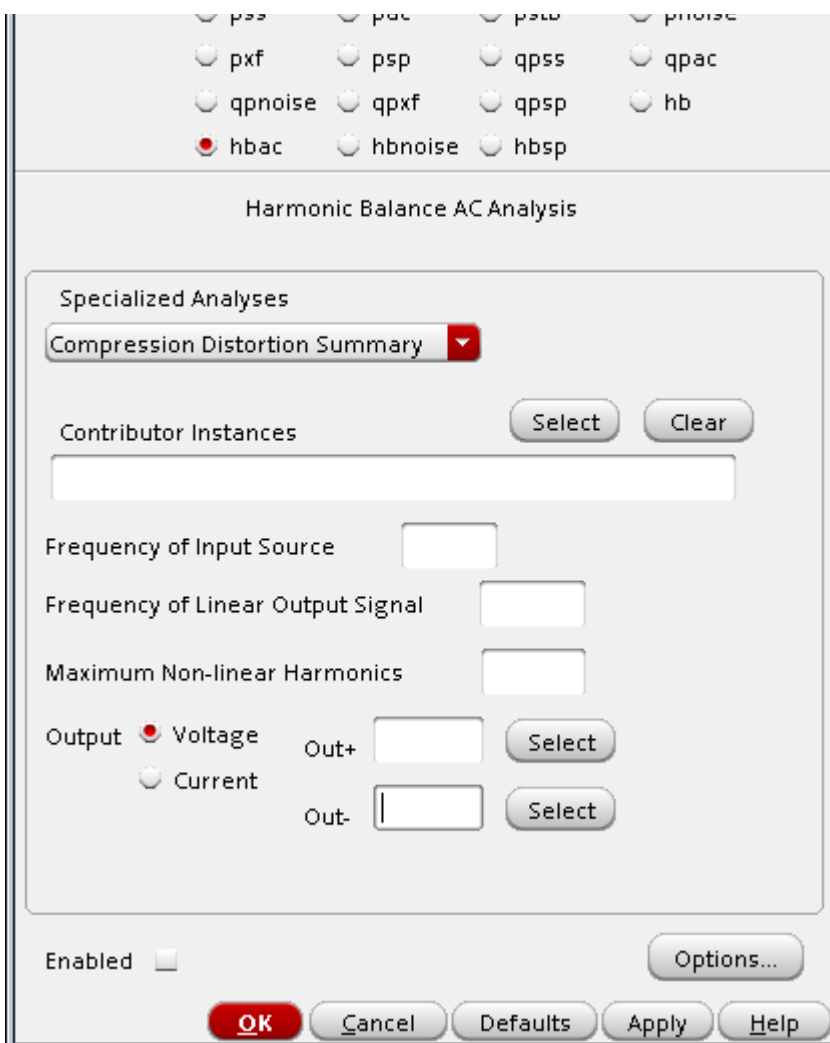
**Figure 4-126 Compression Distortion Summary Dialog Box**



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**





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

---

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**

**Choosing Analyses -- ADE Explorer**

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ sp ☐ envlp

☐ pss ☐ pac ☐ pstb ☐ pnoise

☐ pxf ☐ psp ☐ qpss ☐ qpac

☐ qpnoise ☐ qpxf ☐ qpssp ☐ hb

☒ hbac ☐ hbstb ☐ hbnoise ☐ hbbsp

☐ hbxf

Harmonic Balance AC Analysis

Specialized Analyses

Compression Distortion Summary

Contributor Instances

Frequency of Input Source

Frequency of Linear Output Signal

Maximum Non-linear Harmonics

Output ☒ Voltage ☐ Current

Out-

Enabled ☒

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.

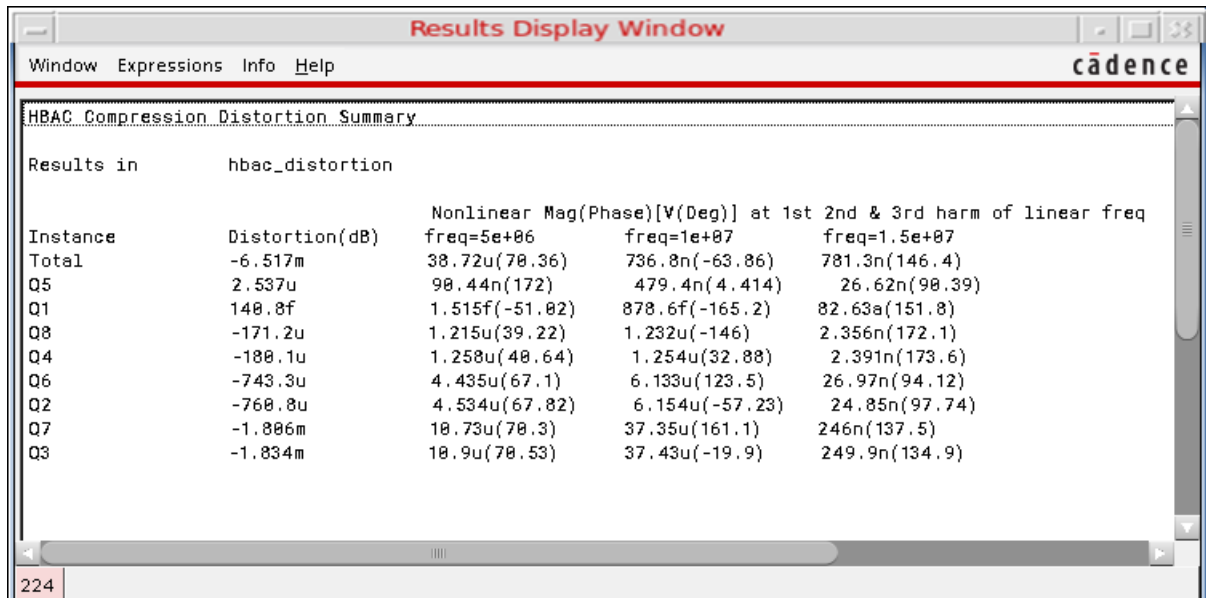
The compression distortion summary provides a relative measure of which components in a gain path contribute more or less compression. Because compression and IP3 are related mathematically, the compression distortion provides information about which components contribute to IP3. Start the simulation. In the ADE Explorer main window or the Schematic, click the green arrow icon (▶) on the right side of the window.

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

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**



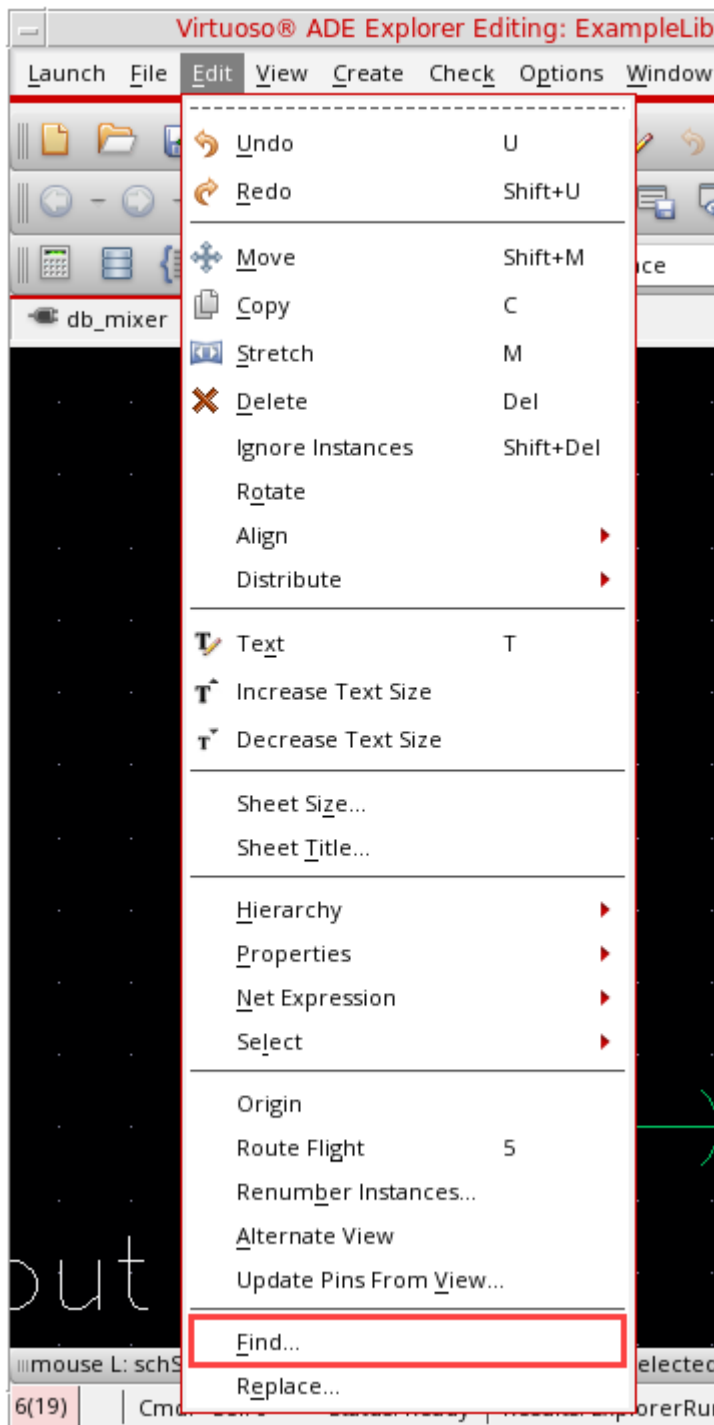
12. Note that Q3, Q7, Q2, and Q6 have the largest distortion contributors.
13. Find Q3 in the schematic.

In the schematic window, choose *Edit - Find*.

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

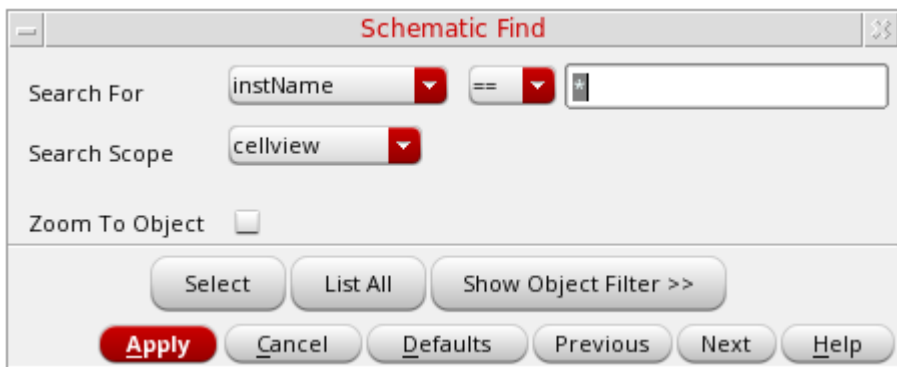
---

**Figure 4-130 Edit-Find to bring up Schematic Find Form**



The *Schematic Find* form is displayed, as shown below.

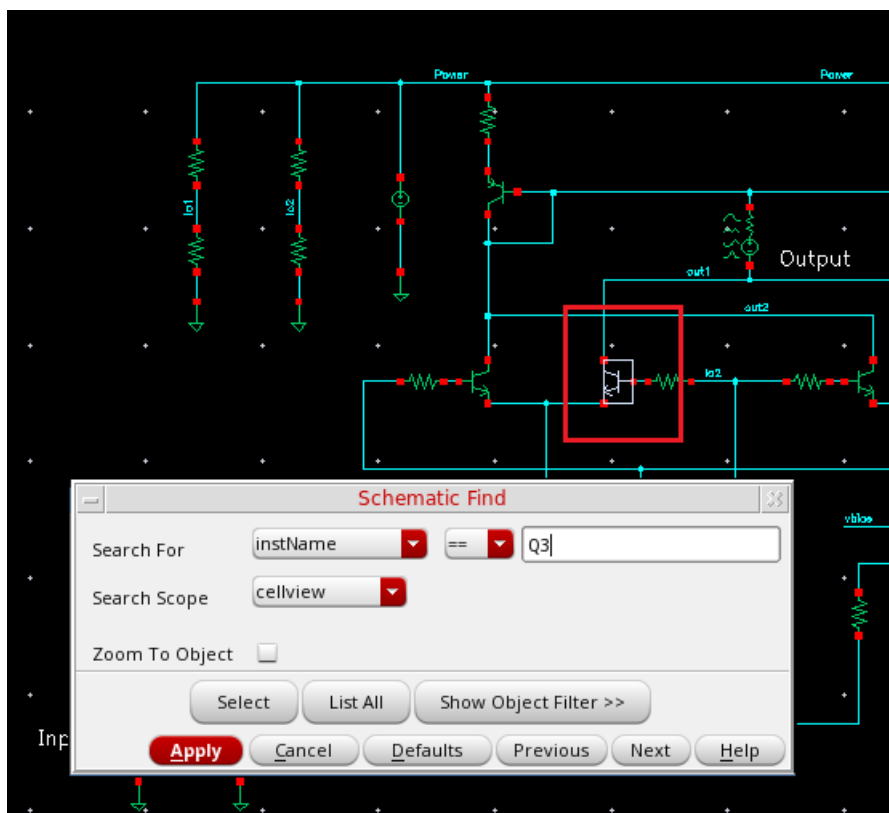
**Figure 4-131 Schematic Find Form**



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.**



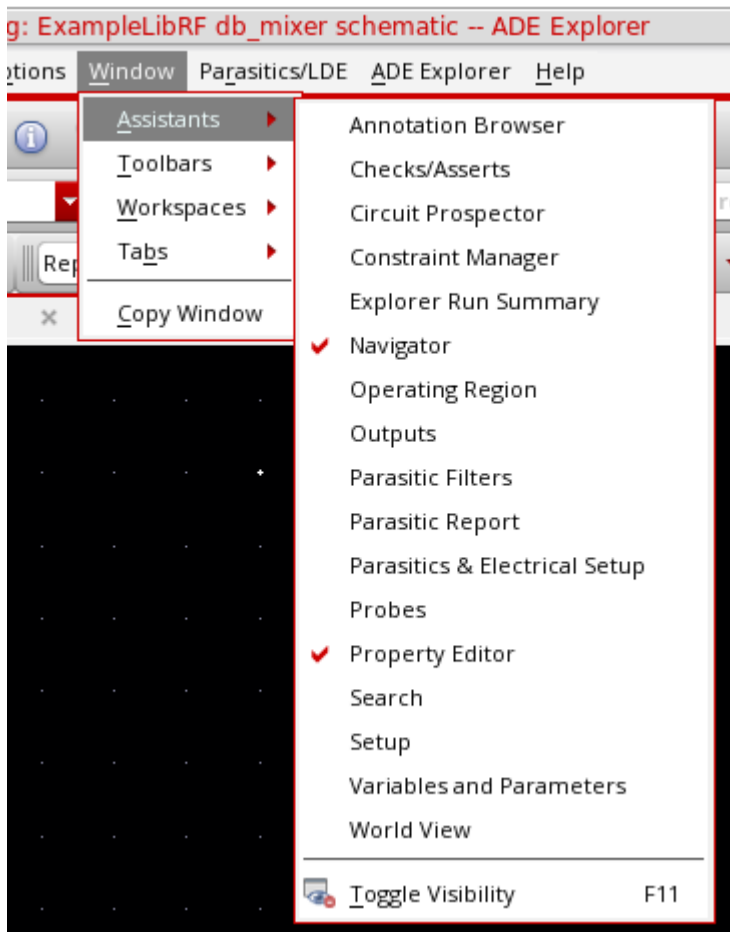
15. In a similar manner, find Q2, Q6, and Q7.

In the *Schematic Find* window, click *Cancel*.

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

---

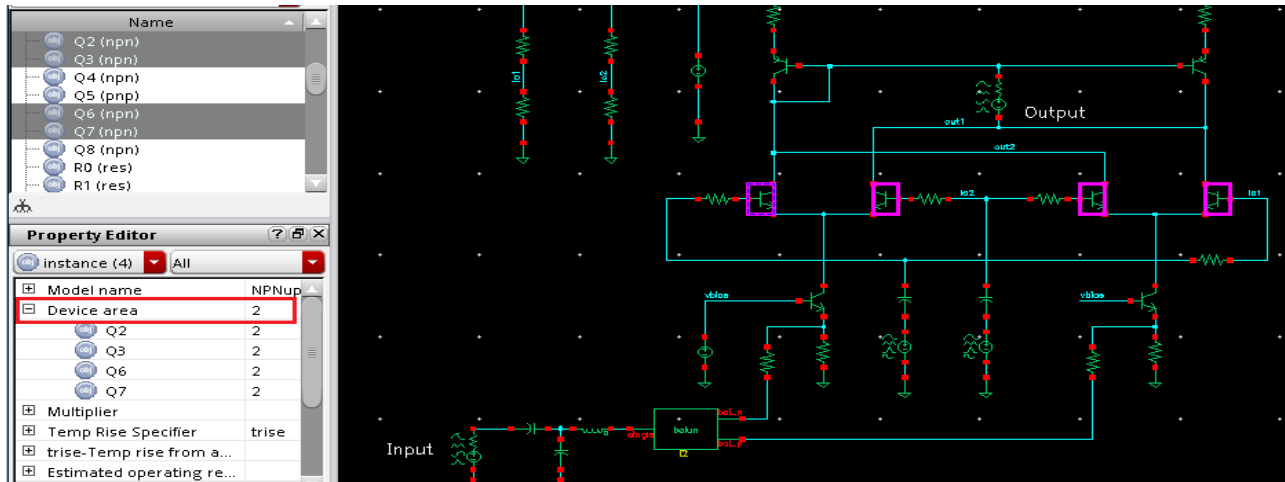
16. Click in the graphics area away from any component so the input port is deselected.
17. If you deleted the Navigator in an earlier step, recall the Window Navigator by choosing *Window - Assistants - Navigator*.



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 cadence                              |                |                |                |                |  |
| 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)  |  |
| 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(170.9)  |  |
| 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                                                                | -258.6u        | 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](#).



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

---

The remaining sections focus on simulating Transmit Mixers.

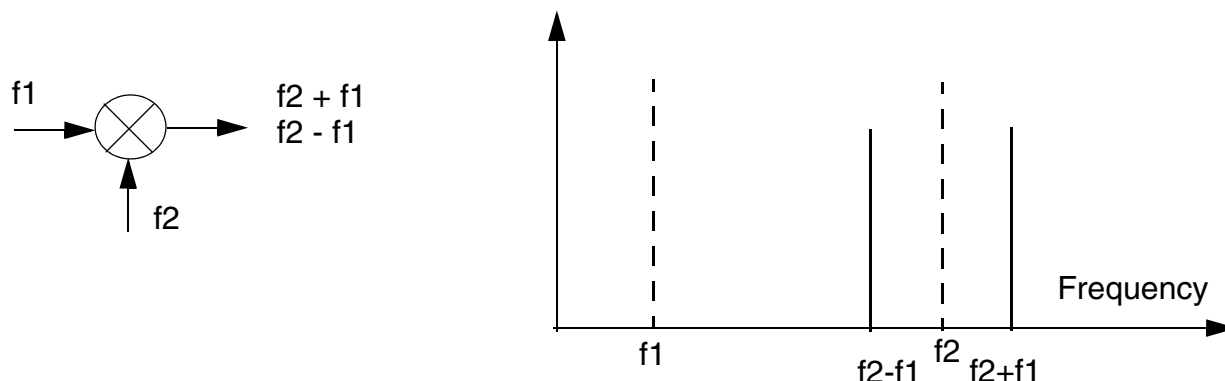
## Setting Up to Simulate the db\_mixer\_xmit Mixer

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 ( $f_1$  and  $f_2$ ), the sum and difference signals are produced, as shown below.



The output signals are  $(f_1 + f_2)$  and  $(f_1 - f_2)$ .

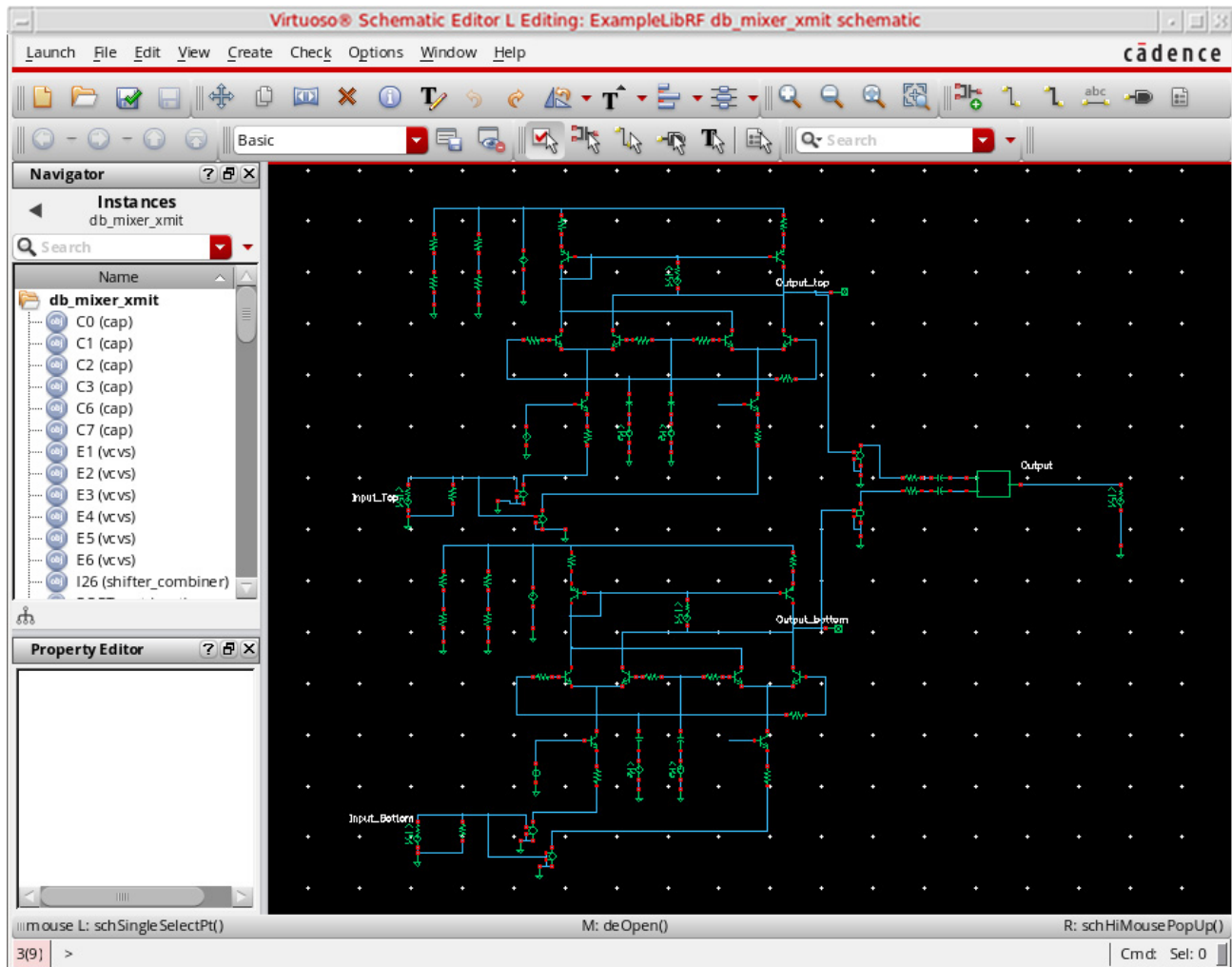
Of the two products from a mixer, normally only one is required. Often, the unwanted one will fall outside the required bandwidths and can be removed very easily. If the unwanted or image product is close to the desired signal, it can require complicated filtering to remove it sufficiently.

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.

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

Figure 4-135 db\_mixer\_xmit Schematic



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

---

| 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              | <i>measurement needed</i> |
| Input IP3 (from swept power) | <i>measurement needed</i> |
| Noise figure/SNR             | <i>measurement needed</i> |

### 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  $f_1 + f_2$  and  $|f_1 - f_2|$ . 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.

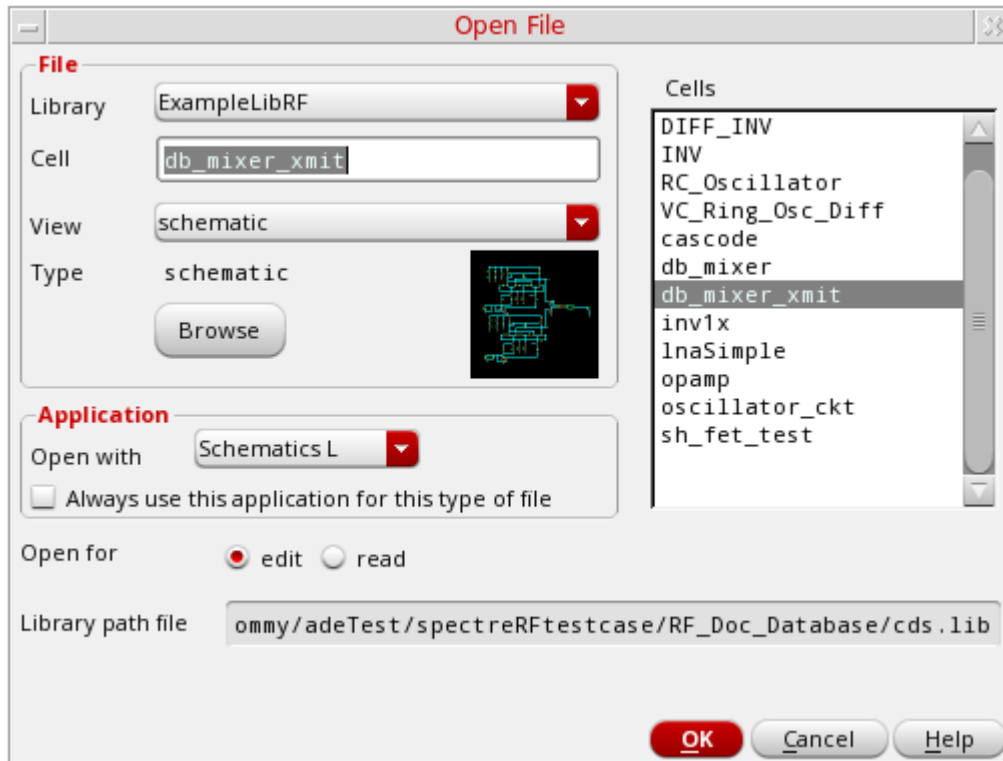
### Setting Up the Simulation

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**

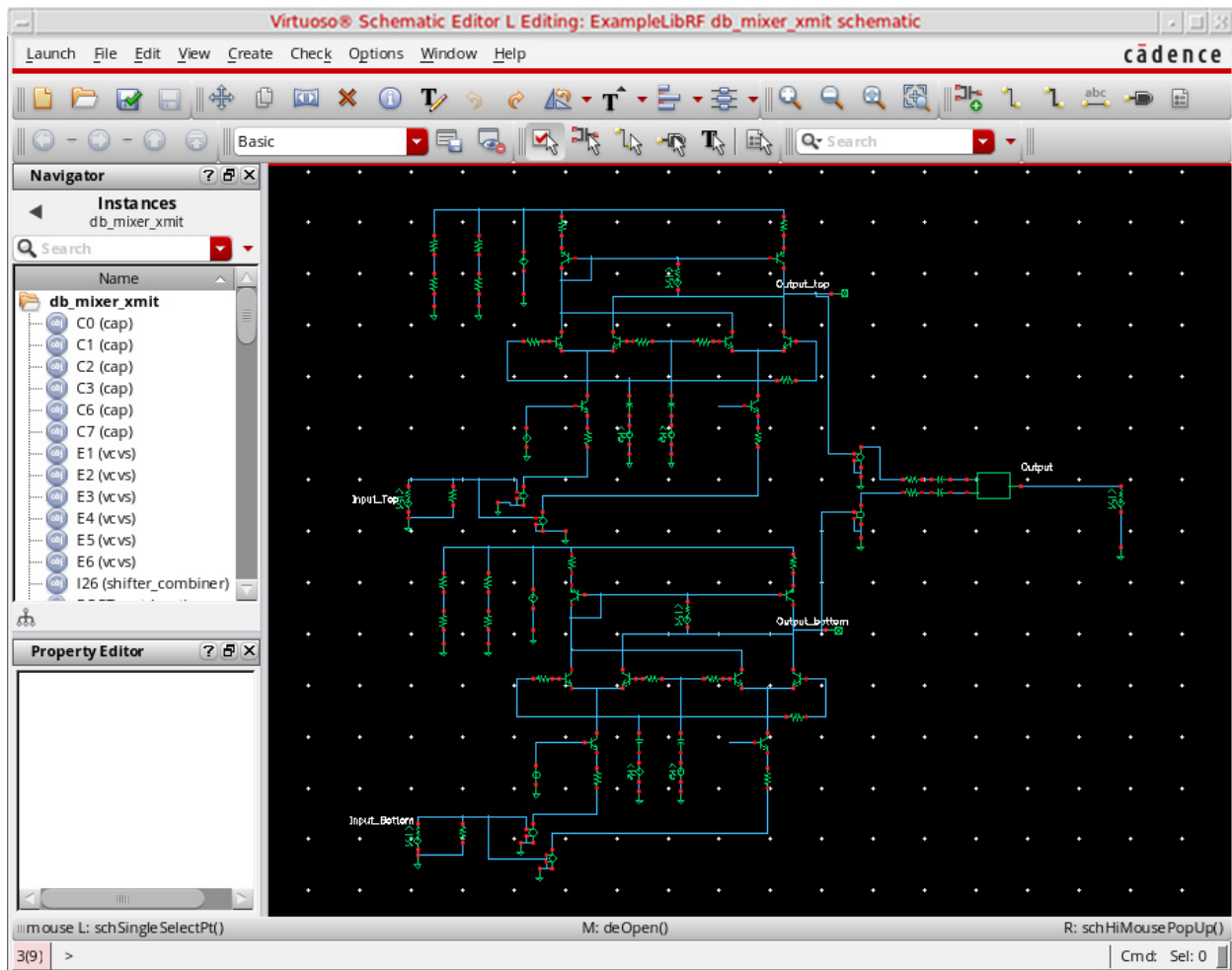


4. Click **OK**.

The Schematic window for the `db_mixer_xmit` mixer is displayed, as shown below.

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

Figure 4-137 db\_mixer\_xmit Schematic

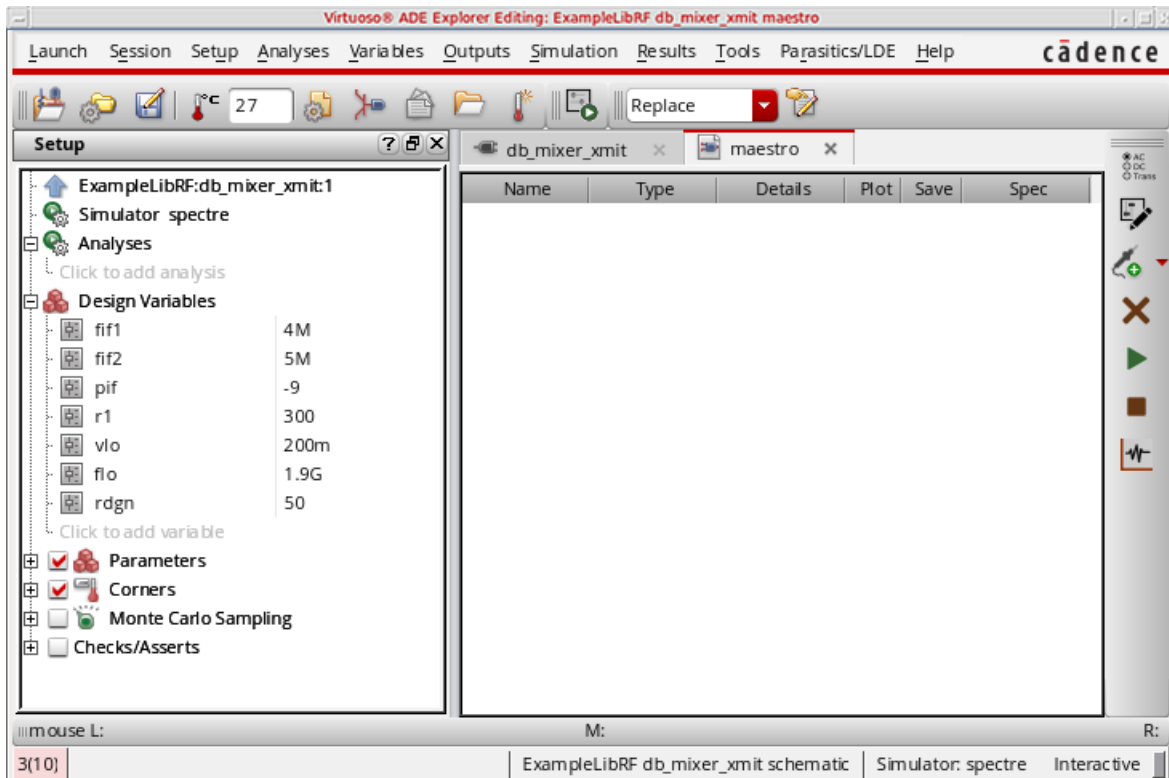


5. In the Schematic window, choose *Launch – ADE Explorer*.

The Virtuoso Analog Design Environment Explorer window is displayed, as shown below.

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

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**



The image shows the 'High-Performance Simulation Options' dialog box. At the top, the title bar reads 'High-Performance Simulation Options'. Below the title bar, the 'Simulation Performance Mode' section has four radio buttons: 'Spectre', 'APS' (which is selected), 'Spectre X', and 'XPS MS'. The 'General' section contains three sub-sections: 'Accuracy + Speed' with an 'Error Preset' of 'Do not override' (selected), 'Liberal', 'Moderate', and 'Conservative'; 'APS Options' with a 'Use ++aps:' checkbox that is unchecked; and 'Multi-Threading' with 'Auto' (selected), 'Disable', and 'Manual' radio buttons, a '# Threads:' text field, and a 'Processor affinity' checkbox that is unchecked. The 'Post-Layout Settings' section includes 'Enable Post-Layout Optimization:' (unchecked), 'Post-Layout Preset Mode:' with 'Default' (selected), 'High Precision Analog', 'Ultra Precision Analog', 'Legacy RCR', and 'Legacy-RF RCR' radio buttons; 'Instance Preservation:' with 'None' (selected) and 'Selected' radio buttons; 'Selected Instances:' with a text field and 'Select' and 'Clear' buttons; and 'Savefilter' with 'none' and 'rc' checkboxes. At the bottom, there are five buttons: 'OK' (highlighted in red), 'Cancel', 'Defaults', 'Apply', and 'Help'.

**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.

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

**Figure 4-141 Save Options Form**

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options Time Setup

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

OK Cancel Defaults Apply Help

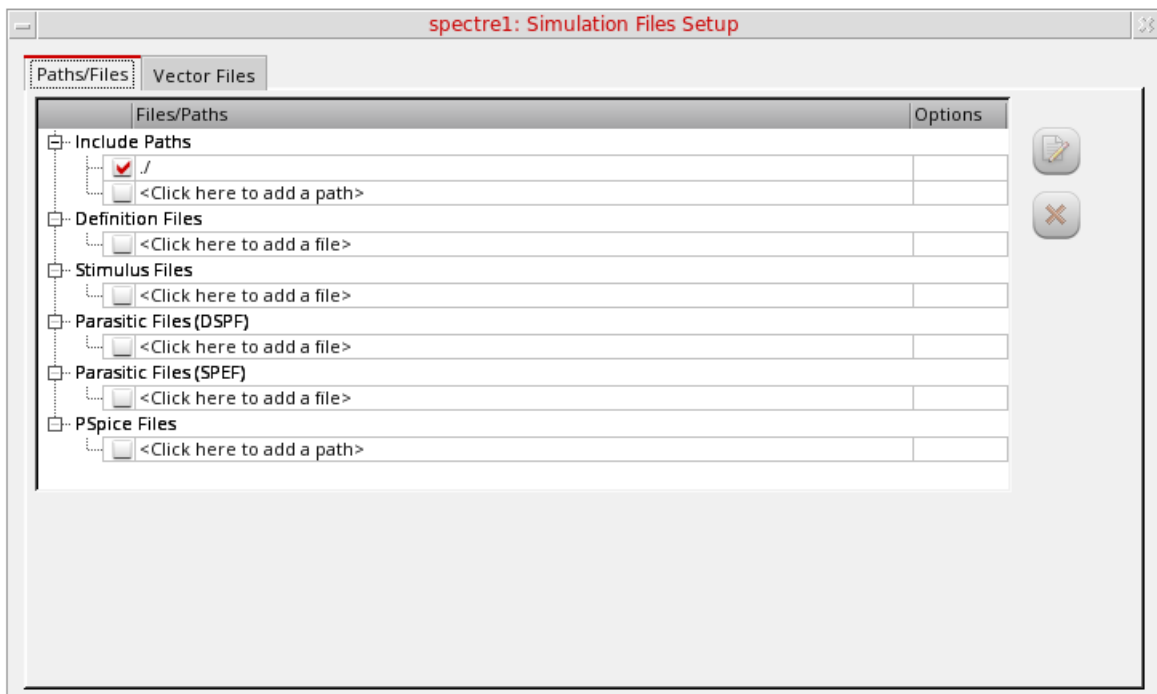
- a. In the *Select signals to output(save)* section, make sure that *allpub* is selected. This is the default which saves all node voltages at all levels of hierarchy, but it does not include the node voltages inside the device models.

- 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`

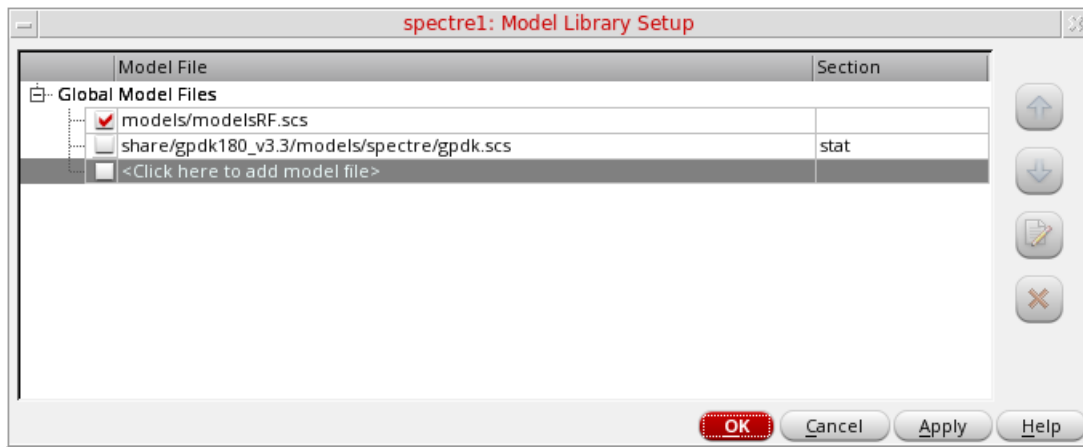
Alternately, you can click *Browse* and browse to the `modelsRF.scs` model file.

15. Make sure that the *Model File* name is selected.

16. Click *Apply*.

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

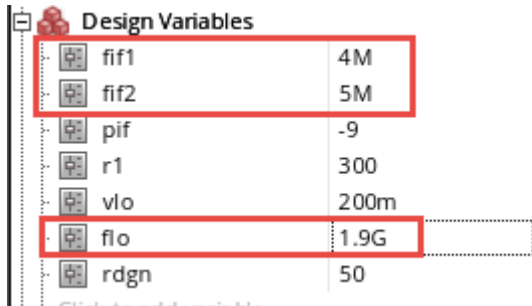
**Figure 4-143 Model Library Setup**



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.

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

Figure 4-145 Edit Properties Forms for IF ports

The figure shows two side-by-side 'Edit Object Properties' dialog boxes for IF ports. The left dialog is for 'port' (Instance Name: r1f1) and the right is for 'port' (Instance Name: r1f2). Both dialogs show parameters for Port mode, Resistance, Reactance, Port number, DC voltage, Source type, and two sinusoid frequencies (Frequency 1 and Frequency 2). The 'Frequency 1' and 'Frequency 2' sections are highlighted with red boxes. The 'Frequency 1' section includes fields for Frequency name 1, Frequency 1, Amplitude 1 (Vpk), Amplitude 1 (dBm), and Phase for Sinusoid 1. The 'Frequency 2' section includes fields for Frequency name 2, Frequency 2, Amplitude 2 (Vpk), Amplitude 2 (dBm), and Phase for Sinusoid 2. Both dialogs have 'Display' checkboxes for each parameter and a 'Display second sinusoid' checkbox. The bottom of each dialog has buttons for OK, Cancel, Apply, Defaults, Previous, Next, and Help.

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.

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

Figure 4-146 The HB Choosing Analyses Form

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☒ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbisp ☐ hbxf

Harmonic Balance Analysis

Transient-Aided Options

Run transient? **Decide automatically**

Detect Steady State ☒ Stop Time(tstab) **auto**

Save Initial Transient Results(saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

Tone 1

Fundamental Frequency

Number of Harmonics **auto**

Oversample Factor **1**

Frequency Divide Ratio for Tone 1 **1**

Harmonics **Default**

Accuracy Defaults (errpreset)

☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☐

**OK** Cancel Defaults Apply Help

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

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

---

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.

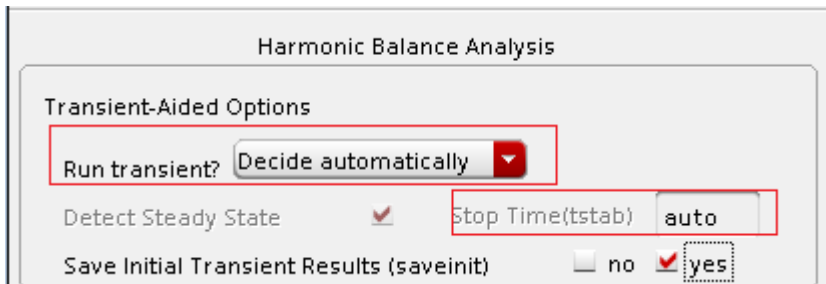
When *Run transient?* is set to *Decide automatically*, the *Detect Steady State* option is checked automatically. When this is set, when steady-state is detected in the *tstab* interval, the simulator stops the transient analysis, runs the FFT, and starts iterating in the frequency domain. Using *Decide 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**



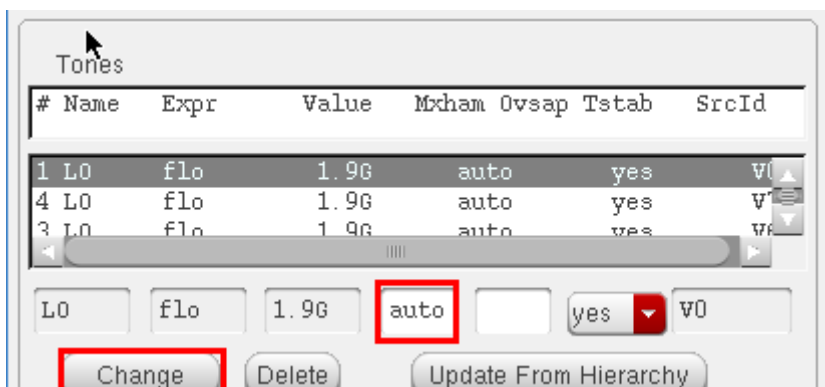
4. In the *Tones* section, select *Names*. When *Names* is selected, the *Tones* portion of the form expands. All the sources in the top-level schematic are read into the form automatically.
5. Select one of the LO sources in the *Tones* section. In the *Mxham* field, type `auto`.

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**

Run transient? Decide automatically

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☐ no ☒ **yes**

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ Names

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 7 | IF   | fif1 | 4M    | 5     | 1     | no    | rif2  |
| 6 | IF2  | fif2 | 5M    | 5     | 1     | no    | rif1  |
| 8 | IF2  | fif2 | 5M    | 5     | 1     | no    | rif2  |

IF2 fif2 5M 5 1 no rif1

**Change** Delete Update From Hierarchy

Freqdivide Ratio for tone with Tstab 1

Harmonics Default

Accuracy Defaults (errpreset)  
☒ **conservative** ☐ moderate ☐ liberal

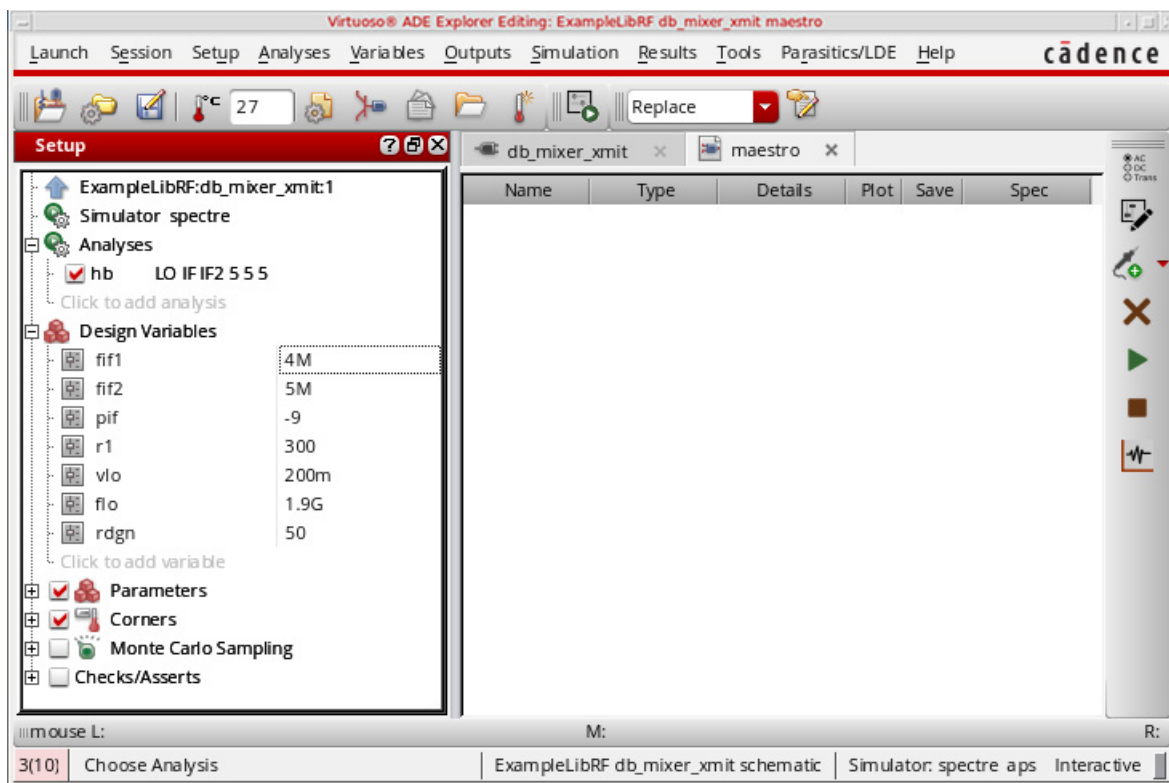
Oscillator ☐

10. Click **OK** at the bottom of the *Choosing Analyses* form.

ADE Explorer will look like the following:

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

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**

Direct Plot Form

Plotting Mode: Append

**Analysis**

☒ tstab ☐ hb\_mt

**Function**

☒ HB Transient V ☐ HB Transient I

Select: Net

Add To Outputs: ☐

> Select Net on schematic...

Close Help

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

**Direct Plot Form**

Plotting Mode: Append

**Analysis**

☐ tstab ☒ hb\_mt

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Power Added Eff. ☐ Power Gain Vs Pout  
☐ Comp. Vs Pout ☐ Node Complex Imp.

Select: Net

**Sweep**

☒ spectrum ☐ frequency ☐ ifft

Signal Level: ☐ peak ☒ rms

**Modifier**

☐ Magnitude ☐ Phase ☒ dB20  
☐ Real ☐ Imaginary

**Scalar Expression**

Value At (harms):

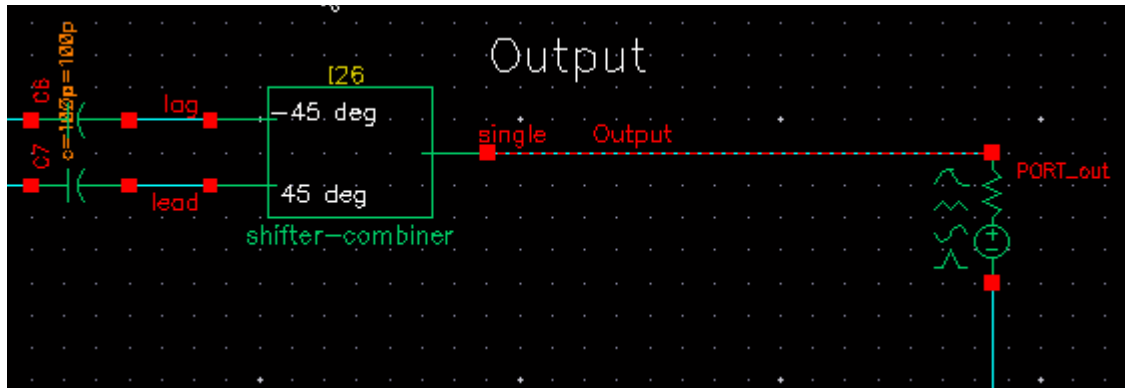
Add To Outputs: ☒

> Select Net on schematic...

Close Help

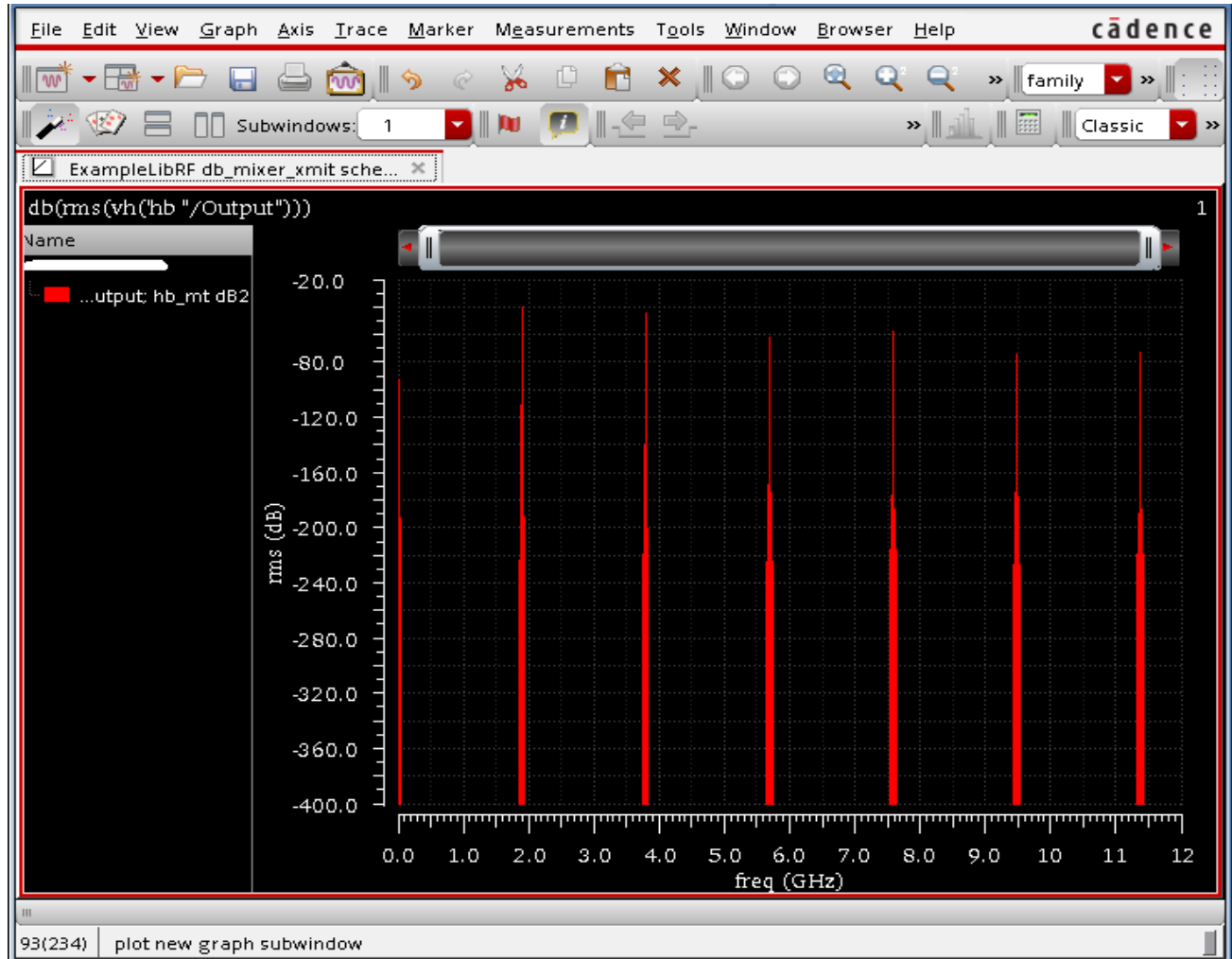
4. Click on the net labeled *Output* in the schematic.

Figure 4-153 Output Net in Schematic



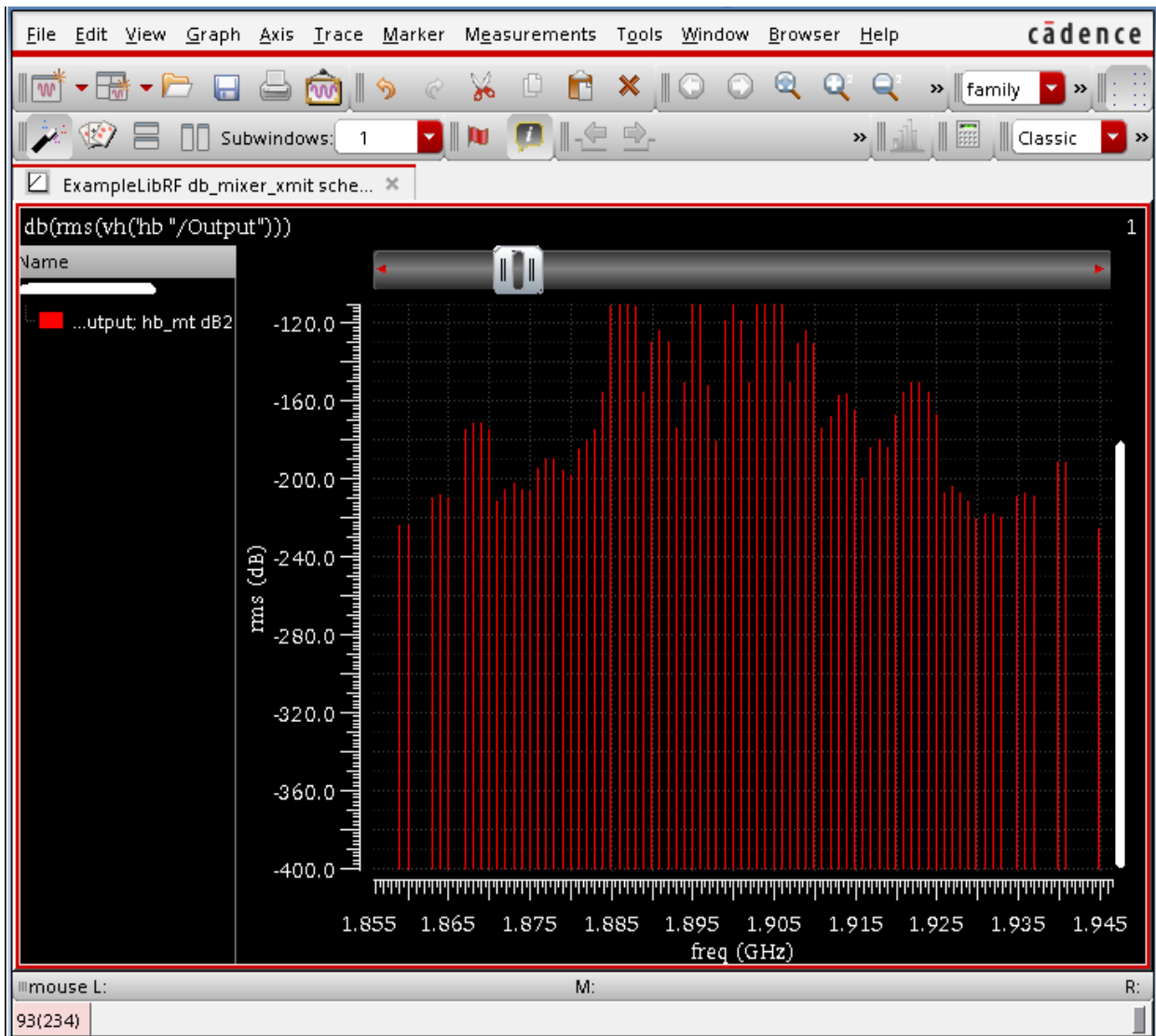
The plot is displayed.

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:



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

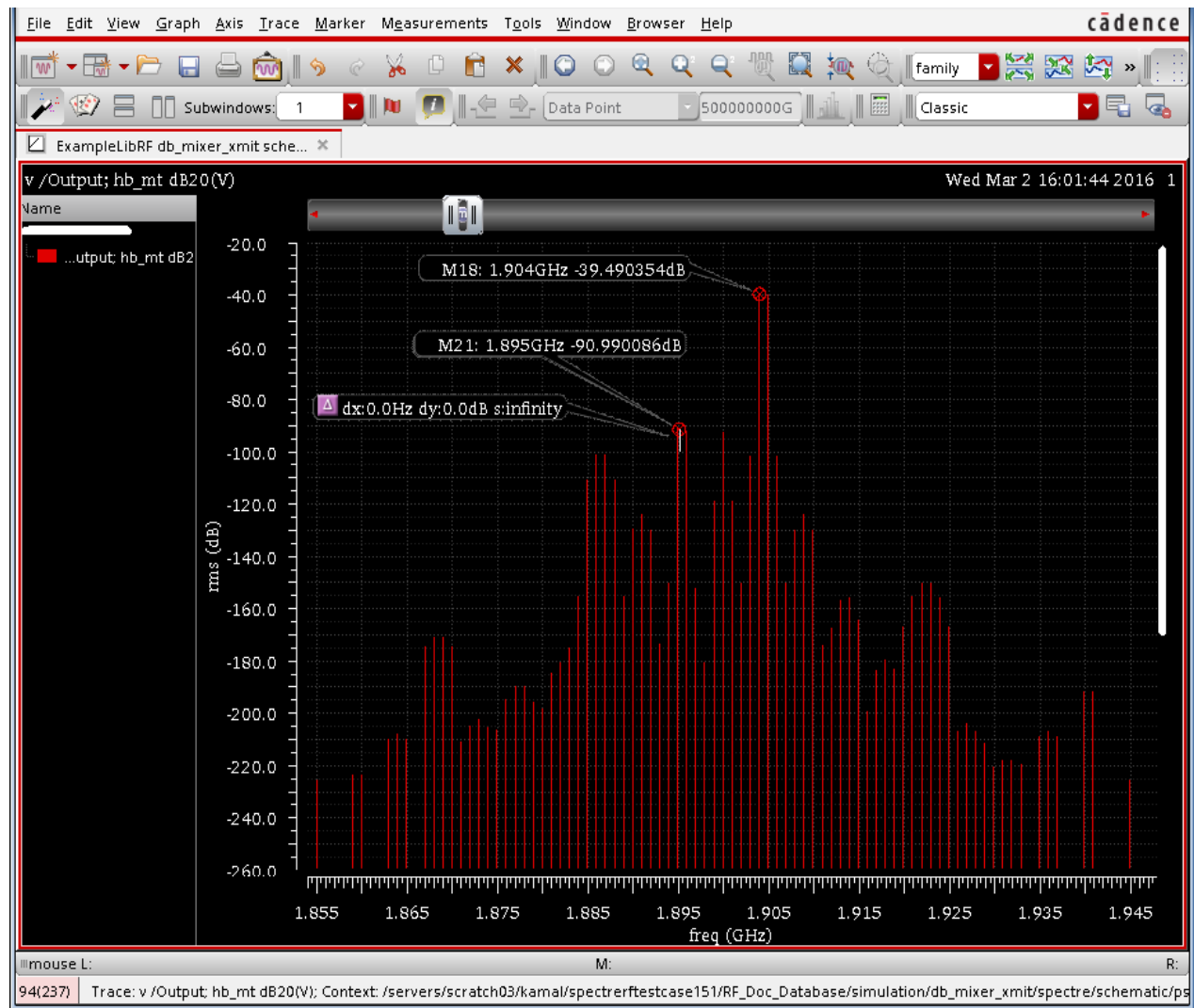
---

|                    | 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*.

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

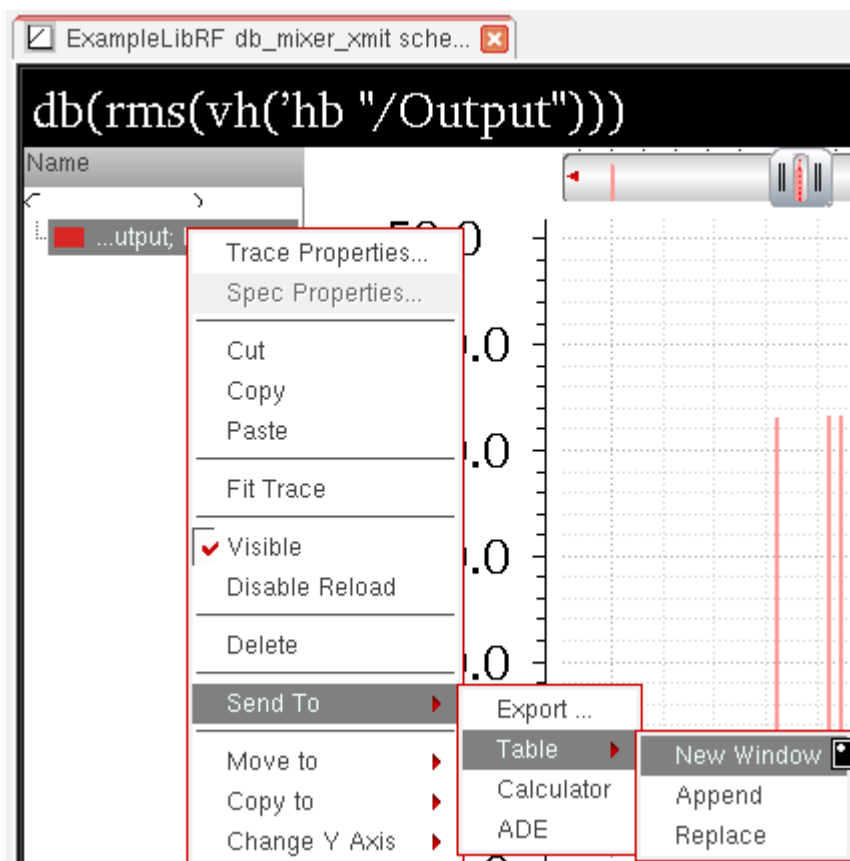
Figure 4-156 Image Rejection



The image rejection is 51.5dB.

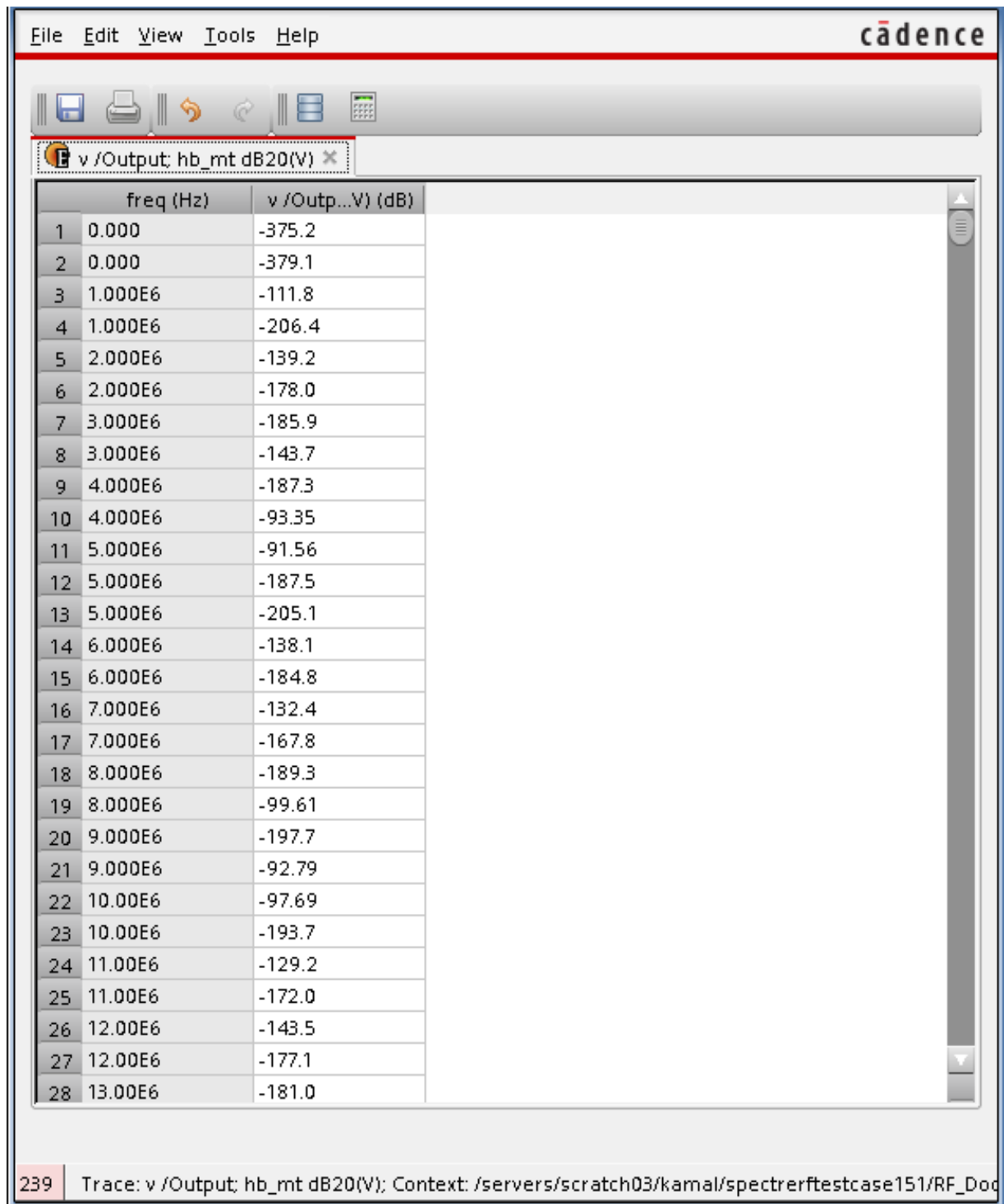
8. You can create tabular data from the results. In the waveform window, select the trace by clicking the legend on the left side of the graph area, clicking the right mouse button, and selecting *Send to - Table - New Window*.

**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**



Close the *Direct Plot Form* and the waveform window.

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

---

### **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**

Transient/AC options

Run transient? Decide automatically

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ Names

Tones

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 5 | IF   | fif1 | 4M    | 5     | 1     | no    | rif1  |
| 7 | IF   | fif1 | 4M    | 5     | 1     | no    | rif2  |
| 6 | IF2  | fif2 | 5M    | 5     | 1     | no    | rif1  |

IF2 fif2 5M 5 1 no rif1

Change Delete Update From Hierarchy

Freqdivide Ratio for tone with Tstab 1

Harmonics Default

Accuracy Defaults (errpreset)  
☒ conservative ☐ moderate ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

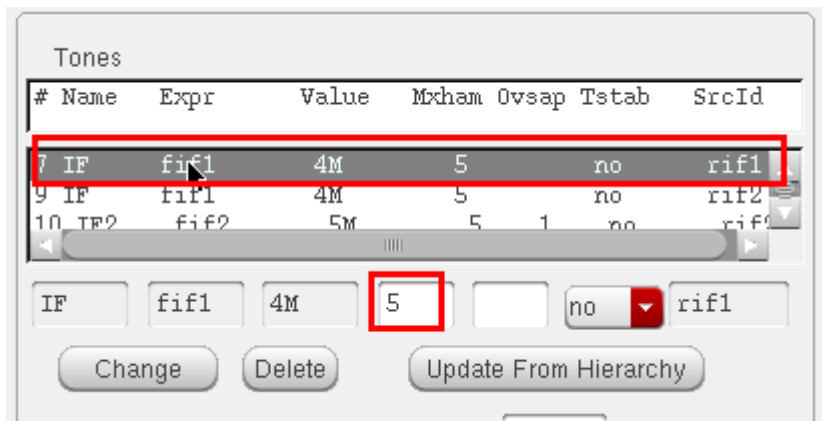
Compression ☐

Enabled ☒ Options...

OK Cancel Defaults Apply Help

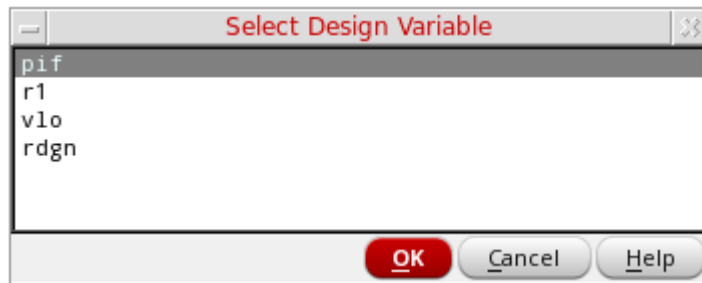
2. Ensure that *Accuracy Defaults (errpreset)* is set to *conservative*. *conservative* is set because third and fifth-order intermodulation distortion will be calculated with this setup. Because these amplitudes are small, high accuracy is needed.
3. In the *Tones* section, select the line containing 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.

The screenshot shows the 'Sweep' section of a software interface. It includes a 'Sweep' dropdown menu set to '1' with a checkmark icon. Below it is a 'Variable' dropdown menu. To the right, there is a 'Frequency Variable?' section with radio buttons for 'no' (selected) and 'yes'. Below this is a 'Variable Name' text field containing 'pif' and a 'Select Design Variable' button. The 'Sweep Range' section has radio buttons for 'Start-Stop' (selected) and 'Center-Span'. The 'Start' text field contains '-14' and the 'Stop' text field contains '-4'. The 'Sweep Type' section has radio buttons for 'Linear' (selected) and 'Logarithmic'. The 'Step Size' text field contains '1' and the 'Number of Steps' radio button is unselected. At the bottom, there is an 'Add Specific Points' checkbox which is unselected.

6. The complete hb *Choosing Analyses* form should look like the following:

Figure 4-163 hb Choosing Analyses Form for 3 Tone IP3

Harmonic Balance Analysis

Transient-Aided Options

Run transient?

Detect Steady State ☒ Stop Time(tstab)

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ Names

Tones

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 5 | IF   | fif1 | 4M    | 5     | 1     | no    | rif1  |
| 7 | IF   | fif1 | 4M    | 5     | 1     | no    | rif2  |
| 8 | IF2  | fif2 | 5M    | 5     | 1     | no    | rif1  |

Freqdivide Ratio for tone with Tstab

Harmonics

Accuracy Defaults (errpreset)

☒ conservative ☐ moderate ☐ liberal

Oscillator ☐

Sweep  ☒

Variable

Frequency Variable? ☒ no ☐ yes

Variable Name

Sweep Range

☒ Start-Stop ☐ Center-Span

Start  Stop

Sweep Type

☒ Linear ☐ Logarithmic

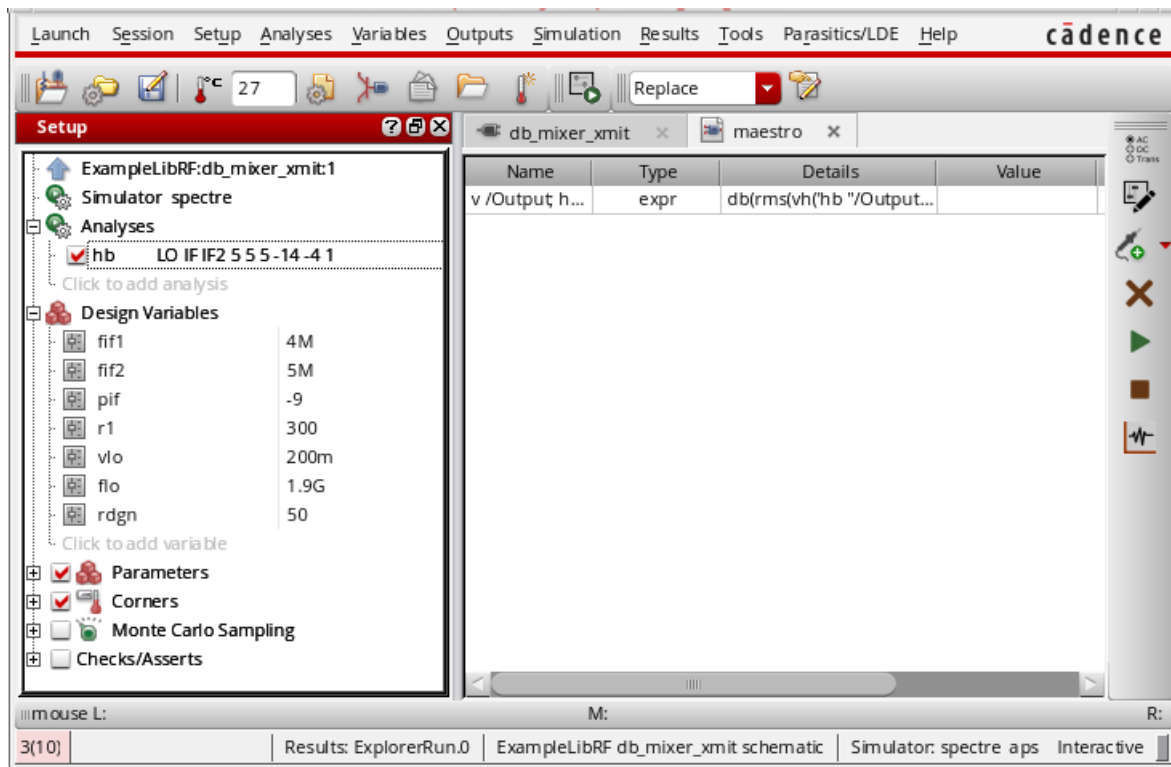
☒ Step Size  ☐ Number of Steps

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

- 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:

- 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.

- 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**

The screenshot shows the 'Harmonic Balance Direct Plot Form' with the following sections and options:

- Analysis:** ☐ tstab, ☒ hb\_mt (highlighted with a red box)
- Function:**
  - ☒ Voltage, ☐ Current
  - ☐ Power, ☐ Voltage Gain
  - ☐ Current Gain, ☐ Power Gain
  - ☐ Transconductance, ☐ Transimpedance
  - ☐ Compression Point, ☐ IPN Curves
  - ☐ Power Contours, ☐ Reflection Contours
  - ☐ Power Added Eff., ☐ Power Gain Vs Pout
  - ☐ Comp. Vs Pout, ☐ Node Complex Imp.
- Select:** Net (dropdown menu)
- Sweep:**
  - ☒ spectrum, ☐ frequency, ☐ variable
  - ☐ ifft
- Signal Level:** ☐ peak, ☒ rms
- Modifier:**
  - ☐ Magnitude, ☐ Phase, ☒ dB20
  - ☐ Real, ☐ Imaginary
- Variable Value (pif):** A list box containing values: -14, -13, -12, -11, -10, -9.
- Add To Outputs:** ☒
- Footer:** > Select Variable Value (pif) on this form... Close Help

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 and 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**

|                          | Freq. (Hz) | L0 | IF | IF2 |
|--------------------------|------------|----|----|-----|
| 3rd<br>Order<br>Harmonic | 1.902G     | 1  | 3  | -2  |
|                          | 1.902G     | 1  | -2 | 2   |
|                          | 1.903G     | 1  | 2  | -1  |
|                          | 1.903G     | 1  | -3 | 3   |
|                          | 1.904G     | 1  | 1  | 0   |

- 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**

|                          |        |   |    |    |
|--------------------------|--------|---|----|----|
| 1st<br>Order<br>Harmonic | 1.904G | 1 | 1  | 0  |
|                          | 1.904G | 1 | -4 | 4  |
|                          | 1.905G | 1 | 5  | -3 |
|                          | 1.905G | 1 | 0  | 1  |
|                          | 1.905G | 1 | -5 | 5  |
|                          | 1.906G | 1 | 4  | -2 |

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

**Figure 4-168 Direct Plot Form for Measuring IP3**

Plotting Mode New Win

**Analysis**

☐ tstab ☒ hb\_mt

**Function**

☐ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☒ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Power Added Eff. ☐ Power Gain Vs Pout  
☐ Comp. Vs Pout ☐ Node Complex Imp.

Select Port ( fixed R(port) )

Circuit Input Power ☐ Single Point ☒ Variable Sweep ("pif")

"pif" ranges from -14 to -4

Input Power Extrapolation Point (dBm) -9

Output Referred IP3 Order 3rd

|          | Freq. (Hz) | L0 | IF | IF2 |
|----------|------------|----|----|-----|
| 3rd      | 1.902G     | 1  | -2 | 2   |
|          | 1.903G     | 1  | 2  | -1  |
|          | 1.903G     | 1  | -3 | 3   |
|          | 1.904G     | 1  | 1  | 0   |
|          | 1.904G     | 1  | -4 | 4   |
| Order    | 1.905G     | 1  | 5  | -3  |
|          | 1.903G     | 1  | -3 | 3   |
|          | 1.904G     | 1  | 1  | 0   |
|          | 1.904G     | 1  | -4 | 4   |
|          | 1.905G     | 1  | 5  | -3  |
| Harmonic | 1.905G     | 1  | 0  | 1   |
|          | 1.905G     | 1  | -5 | 5   |

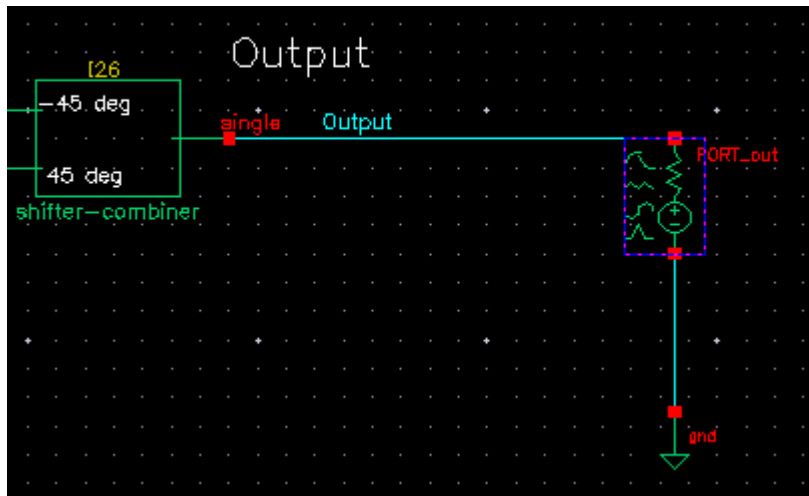
Add To Outputs ☐ Replot

> Select Port on schematic...

OK Cancel Help

4. Select the *Output* port on the schematic.

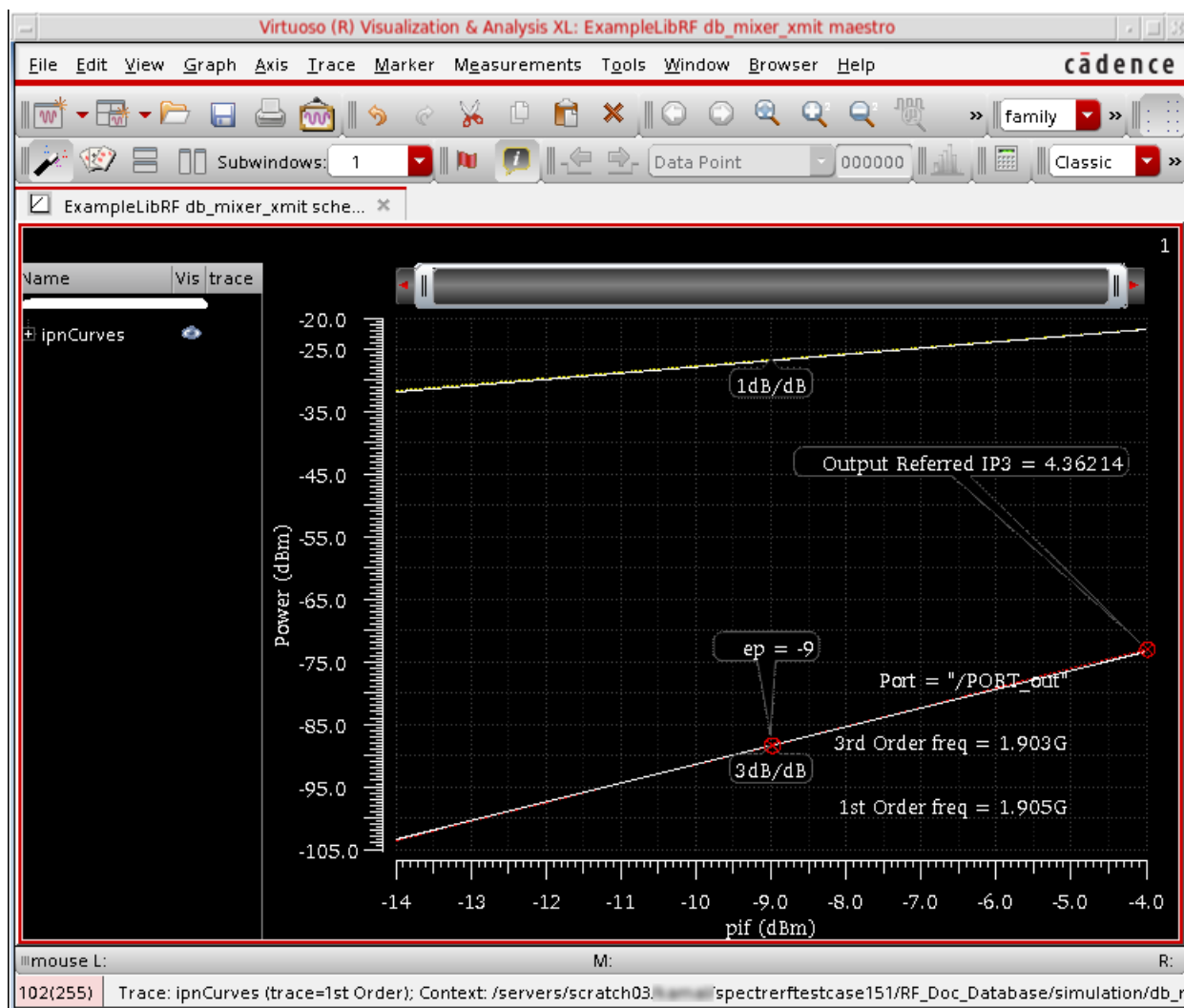
**Figure 4-169** output port



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  $4.36\text{dBm}$ . 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,  $f_2 + f_1$  or  $f_2 - f_1$ ). The only way to achieve image rejection in both legs is to have a  $\pm 45$  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:

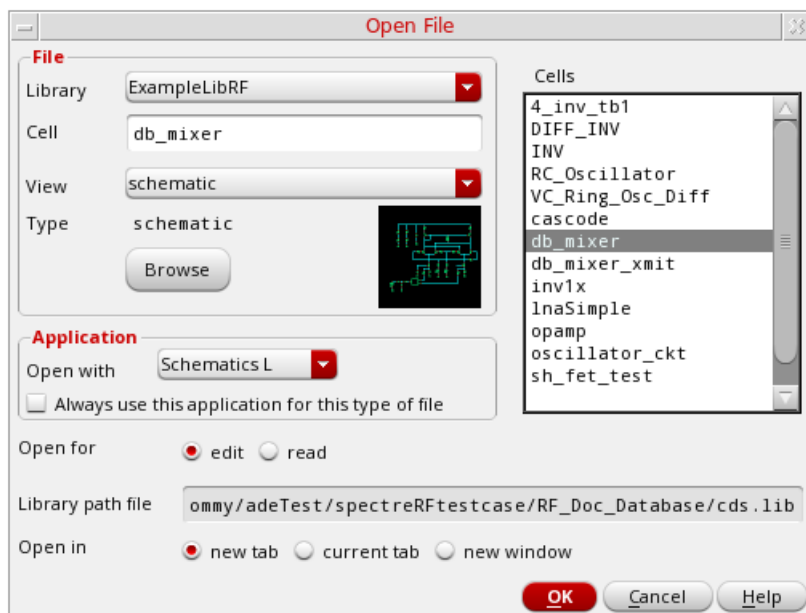
1. In the CIW, choose *File – Open*.

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**

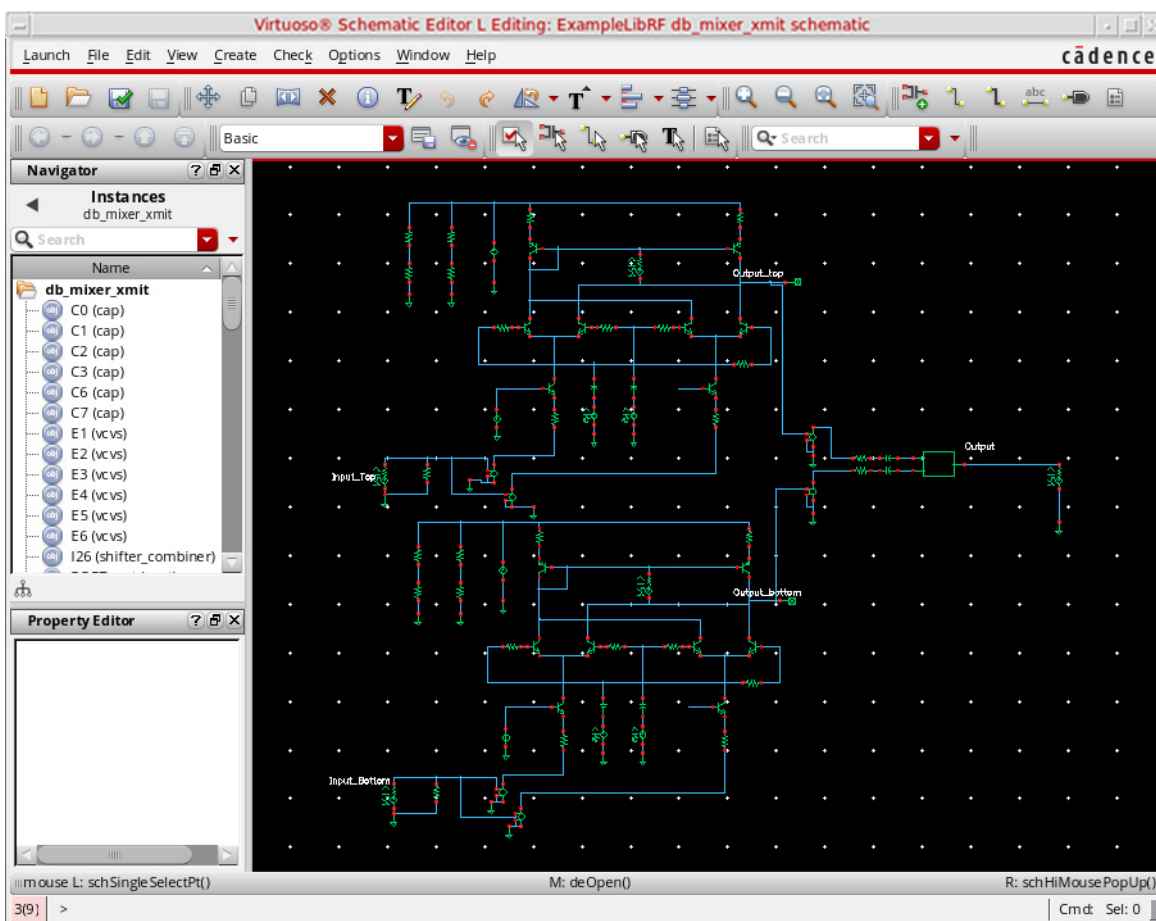


## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

### 4. Click *OK*.

The Schematic window for the *db\_mixer\_xmit* mixer is displayed.

**Figure 4-172 db\_mixer\_xmit Schematic**

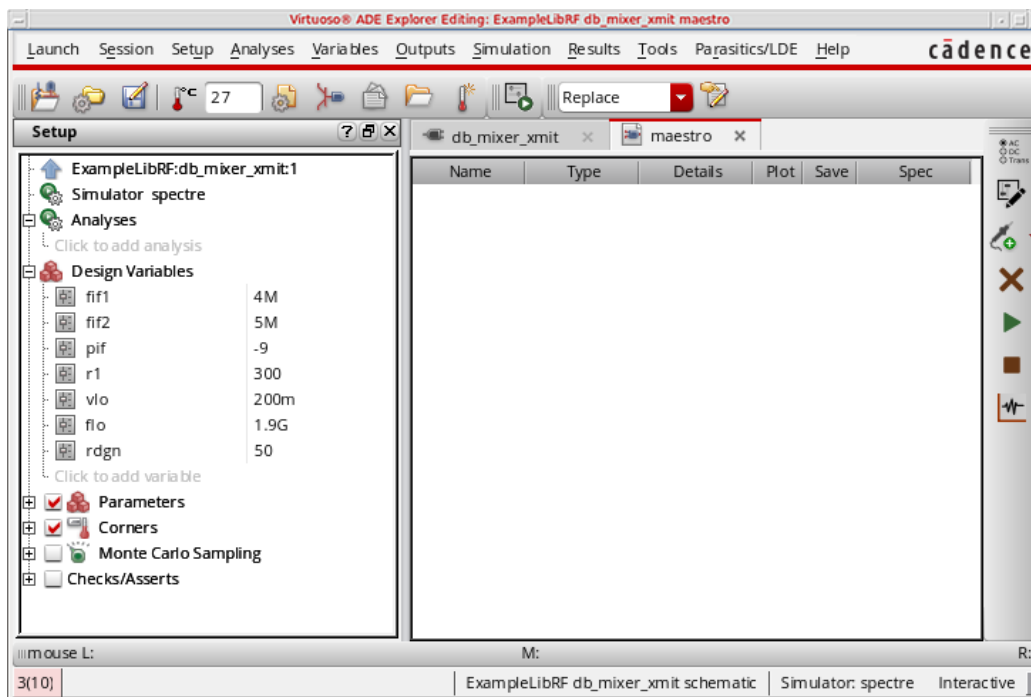


### 5. In the Schematic window, choose *Launch – ADE Explorer*.

The Virtuoso Analog Design Environment Explorer window is displayed, as shown below.

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

**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*.

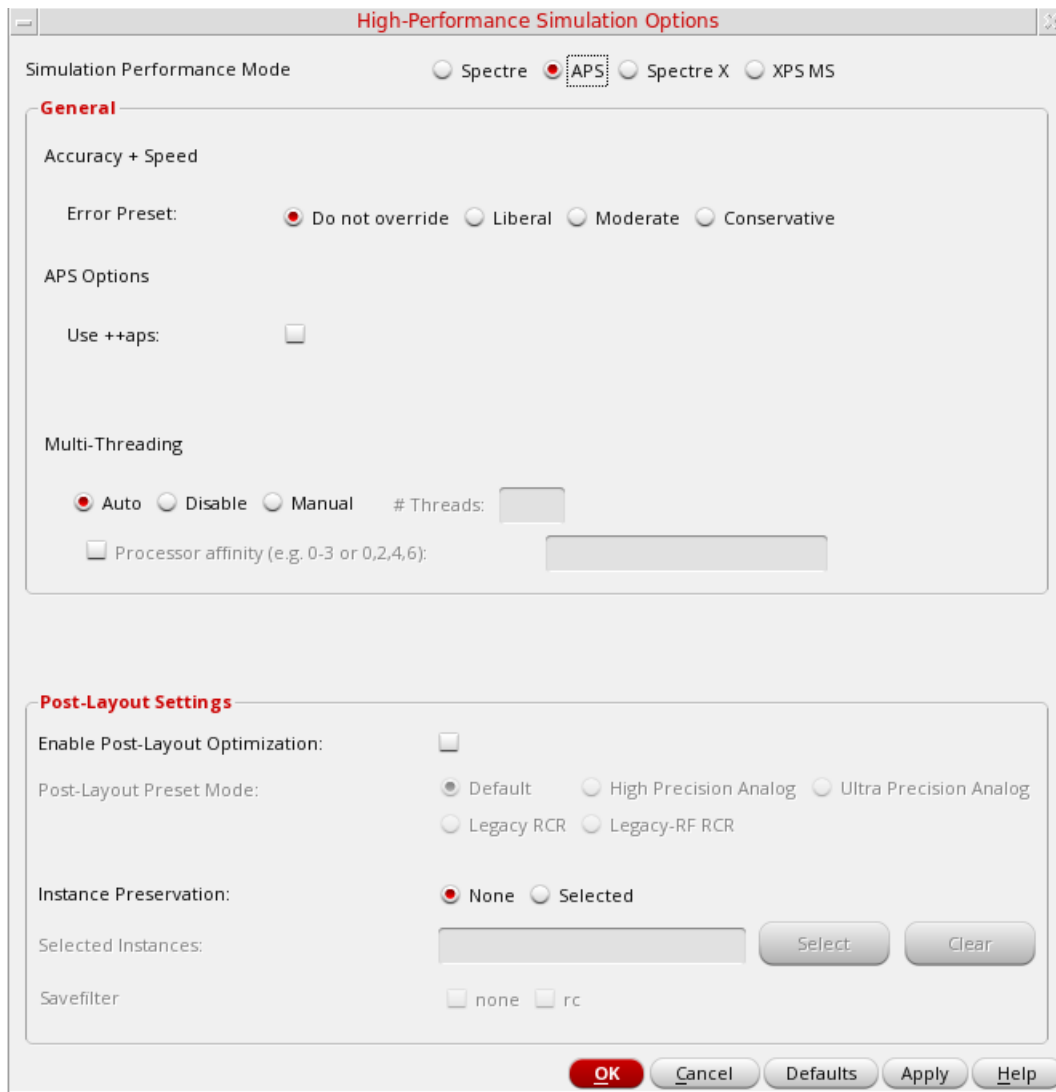
**Figure 4-174 Choosing Simulator/Director/Host Form**



- 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**



The image shows the 'High-Performance Simulation Options' dialog box. At the top, the title bar reads 'High-Performance Simulation Options'. Below the title bar, the 'Simulation Performance Mode' section has four radio buttons: 'Spectre', 'APS' (which is selected and highlighted with a dashed box), 'Spectre X', and 'XPS MS'. The 'General' section contains three sub-sections: 'Accuracy + Speed' with an 'Error Preset' of 'Do not override' (selected); 'APS Options' with a 'Use ++aps' checkbox that is unchecked; and 'Multi-Threading' with 'Auto' selected for the threading mode and an empty text field for '# Threads'. There is also an unchecked checkbox for 'Processor affinity (e.g. 0-3 or 0,2,4,6):'. The 'Post-Layout Settings' section includes an unchecked 'Enable Post-Layout Optimization:' checkbox, a 'Post-Layout Preset Mode' with 'Default' selected, and an 'Instance Preservation' section with 'None' selected. Below this is a 'Selected Instances:' text field with 'Select' and 'Clear' buttons. At the bottom of this section are 'Savefilter' checkboxes for 'none' and 'rc'. The bottom of the dialog box has five buttons: 'OK' (highlighted in red), 'Cancel', 'Defaults', 'Apply', and 'Help'.

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.

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

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

**Figure 4-176 Save Options Form**

**Save Options**

Basic | Save By Subckt

**Save Options**

Select signals to output (save) ☐ none ☐ selected ☐ lvlpub ☐ lvl ☒ allpub ☐ all

Select power signals to output (pwr) ☐ none ☐ total ☐ devices ☐ subckts ☐ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents) ☐ selected ☐ nonlinear ☐ all ☐ none

Select AC terminal currents (useprobes) ☐ yes ☐ no

Set probe terminal level (probelvl)

Select AHDL variables (saveahdlvars) ☐ selected ☐ all

**Transient Time Window Options**

☐ Transient time window save options

**Save circuit information analysis**

| Name            | What       | Where   | File | Extremes | Others | Enabled                             |
|-----------------|------------|---------|------|----------|--------|-------------------------------------|
| modelParameter  | models     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| element         | inst       | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| outputParameter | output     | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| designParamVals | parameters | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| primitives      | primitives | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| subckts         | subckts    | rawfile |      |          |        | <input checked="" type="checkbox"/> |
| asserts         | assert     | rawfile |      |          |        | <input type="checkbox"/>            |
| extremeinfo     | all        | logfile |      | yes      |        | <input type="checkbox"/>            |
| <Click_To_Add>  | none       | rawfile |      |          |        | <input type="checkbox"/>            |

Detail:  Sort:  Threshold:

**Output Options**

Output Format ☐ sst2 ☐ psf ☐ psf with floats ☒ psfxl ☐ fsdb

Use Fast Viewing Extensions ☐

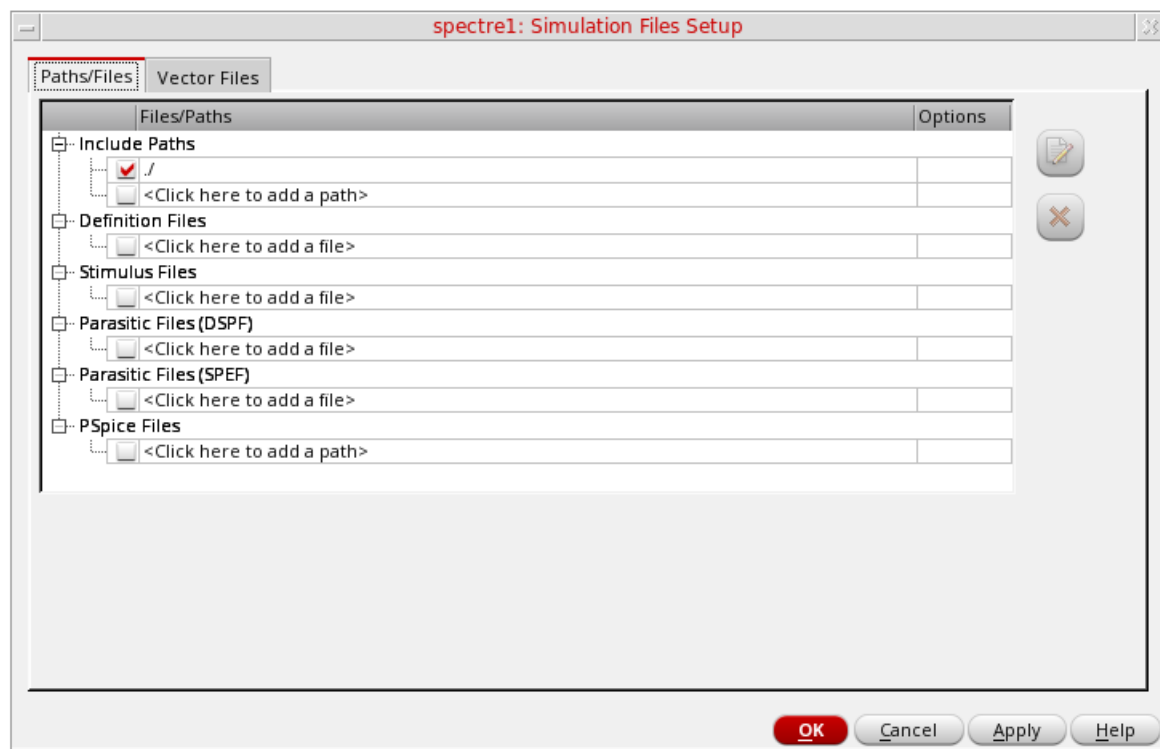
This is the default selection. This saves all of the node voltages at all levels of the hierarchy, but it does not include the node voltages inside the device models.

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**

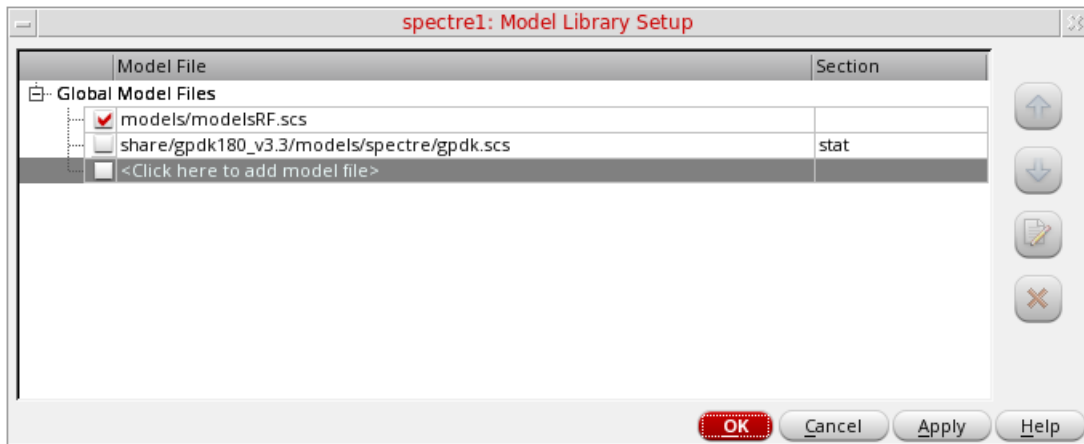


2. Ensure that the *Include Path* is set as shown above.
3. Select *Setup – Model Libraries*.  
The Model Library Setup form is displayed.
4. In the *Model Library File* field, type the following for the name of the model file:  
`models/modelsRF.scs`
5. Click *OK*.

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

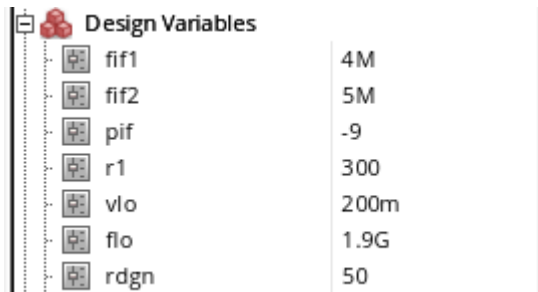
To calculate signal to noise ratio, you first need to calculate the signal using a three-tone hb analysis. You will take the signal value at 1.904G (alternately, you could look at 1.905G) and note that value for later. Then, you will set up a pnoise analysis.

### 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**



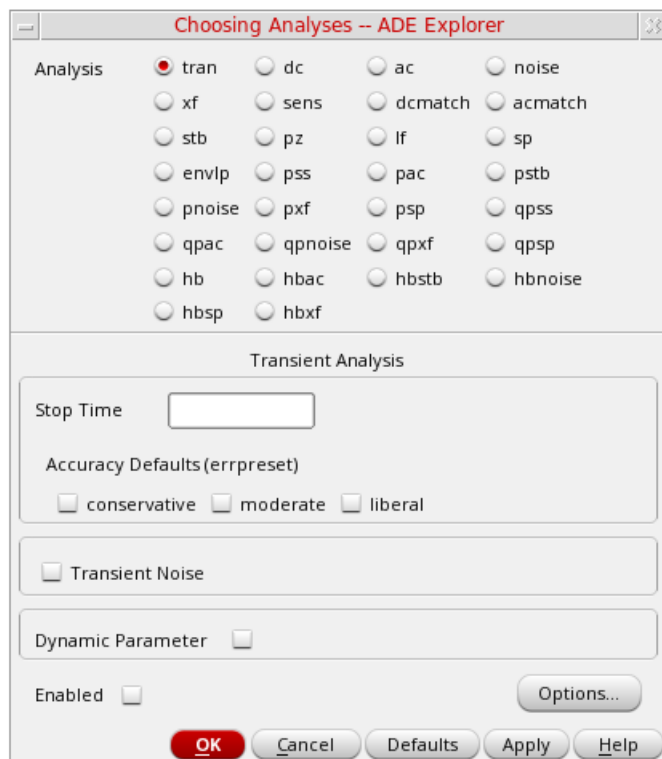
| Design Variables |      |
|------------------|------|
| fif1             | 4M   |
| fif2             | 5M   |
| pif              | -9   |
| r1               | 300  |
| vlo              | 200m |
| flo              | 1.9G |
| rdgn             | 50   |

### 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**



Choosing Analyses -- ADE Explorer

Analysis

☒ tran   ☐ dc   ☐ ac   ☐ noise

☐ xf   ☐ sens   ☐ dcmatch   ☐ acmatch

☐ stb   ☐ pz   ☐ lf   ☐ sp

☐ envlp   ☐ pss   ☐ pac   ☐ pstb

☐ pnoise   ☐ pxf   ☐ psp   ☐ qpss

☐ qpac   ☐ qpnoise   ☐ qpxf   ☐ qpssp

☐ hb   ☐ hbac   ☐ hbstb   ☐ hbnoise

☐ hbasp   ☐ hbxf

Transient Analysis

Stop Time

Accuracy Defaults (errpreset)

☐ conservative   ☐ moderate   ☐ liberal

☐ Transient Noise

Dynamic Parameter ☐

Enabled ☐

Options...

OK Cancel Defaults Apply Help

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.

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

Figure 4-181 The HB Choosing Analyses Form

Choosing Analyses -- ADE Explorer

Analysis

☐ tran ☐ dc ☐ ac ☐ noise

☐ xf ☐ sens ☐ dcmatch ☐ acmatch

☐ stb ☐ pz ☐ lf ☐ sp

☐ envlp ☐ pss ☐ pac ☐ pstb

☐ pnoise ☐ pxf ☐ psp ☐ qpss

☐ qpac ☐ qpnoise ☐ qpxf ☐ qpssp

☒ hb ☐ hbac ☐ hbstb ☐ hbnoise

☐ hbisp ☐ hbxf

Harmonic Balance Analysis

Transient-Aided Options

Run transient?

Detect Steady State ☒ Stop Time(tstab)

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Dynamic Parameter ☐

Tones ☒ Frequencies ☐ Names

Number of Tones ☒ 1 ☐ 2 ☐ 3 ☐ 4

Tone 1

Fundamental Frequency

Number of Harmonics

Oversample Factor

Freqdivide Ratio for Tone 1

Harmonics

Accuracy Defaults (errpreset)

☐ conservative ☒ moderate ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

Enabled ☐

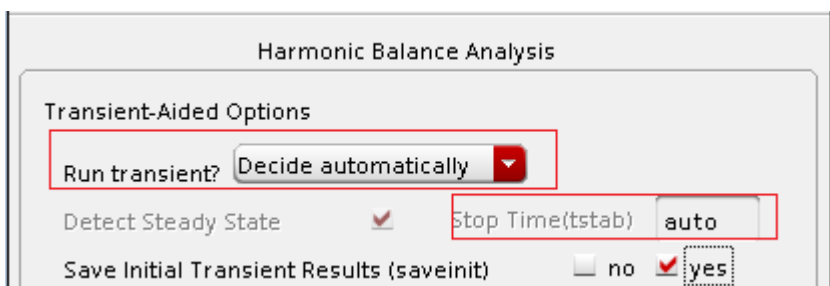
Harmonic balance can set the harmonics automatically for the signal that causes the most distortion. This is recommended in the general case. To enable this, select *Decide automatically* or *Yes* for the *Run Transient* selection in the *Transient-Aided Options* section. This single action will cause 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.

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

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**

The screenshot shows the 'Tones' section of the 'hb Choosing Analyses Form'. At the top, there are three tabs: 'Tones' (selected), 'Frequencies', and 'Names'. Below the tabs is a table with the following columns: '#', 'Name', 'Expr', 'Value', 'Mxham', 'Ovsap', 'Tstab', and 'SrcId'. The table contains two rows of data for LO tones. The first row is highlighted with a red border. Below the table, there are input fields for 'LO', 'flo', '1.9G', '7', '1', 'yes', and 'V0'. The '7' in the 'Mxham' field is highlighted with a red border. Below these fields are three buttons: 'Change' (highlighted with a red border), 'Delete', and 'Update From Hierarchy'. At the bottom, there is a label 'Freqdivide Ratio for tone with Tstab' followed by an empty input field.

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 1 | LO   | flo  | 1.9G  | 7     | 1     | yes   | V0    |
| 4 | LO   | flo  | 1.9G  | 7     | 1     | yes   | V7    |

LO flo 1.9G 7 1 yes V0

Change Delete Update From Hierarchy

Freqdivide Ratio for tone with Tstab

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

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

---

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**

Run transient? Decide automatically ▼

Detect Steady State ☒ Stop Time(tstab) auto

Save Initial Transient Results (saveinit) ☐ no ☒ yes

Dynamic Parameter ☐

Tones ☐ Frequencies ☒ Names

Tones

| # | Name | Expr | Value | Mxham | Ovsap | Tstab | SrcId |
|---|------|------|-------|-------|-------|-------|-------|
| 6 | IF2  | fif2 | 5M    | 5     | 1     | no    | rif1  |
| 8 | IF2  | fif2 | 5M    | 5     | 1     | no    | rif2  |
| 4 | L0   | flo  | 1.9G  | 7     | 1     | yes   | V7    |
| 2 | L0   | flo  | 1.9G  | 7     | 1     | yes   | V1    |

L0 flo 1.9G 7 1 yes ▼ V7

Change Delete Update From Hierarchy

Freqdivide Ratio for tone with Tstab

Harmonics Default ▼

Accuracy Defaults (errpreset)  
☒ conservative ☐ moderate ☐ liberal

Oscillator ☐

Sweep ☐

Loadpull ☐

LSSP ☐

Compression ☐

OK Cancel Defaults Apply Help

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**

hbac **hbnoise** hbasp

Harmonic Balance Noise Analysis

Multiple hbnoise ☐

Sweep type: default  Sweep is currently absolute

Output Frequency Sweep Range (Hz)

Start-Stop  Start  Stop

Sweep Type

Automatic

Add Specific Points ☐

Sidebands

Maximum sideband

When using hb engine, default value is harms of 1st tone.

Noise Figure ☐

Output

probe  Output Probe Instance

Do Noise ☒

Noise Type: timeaverage

Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis

Contribution Type:

☒ USB ☐ AM ☐ PM ☐ AM&PM ☐ ALL(AM,PM,USB,LSB)

Noise Separation ☐ yes ☐ no

Separate noise into source and gain

Enabled ☐



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

---

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**

☐ qpnoise ☐ qpxf ☐ qpdp ☐ hb  
☐ hbac ☒ hbnoise ☐ hbsp

Harmonic Balance Noise Analysis

Multiple hbnoise ☐

Sweep type **default** Sweep is currently absolute

Output Frequency Sweep Range (Hz)

Start-Stop Start 1.9G+1K Stop .9G+1K+10M

Sweep Type

Linear ☒ Step Size 9  
☐ Number of Steps

Add Specific Points ☐

Sidebands

Maximum sideband

When using hb engine, default value is harms of 1st tone.

Noise Figure ☒

Output

probe Output Probe Instance /PORT\_out Select

Input Source

none

Do Noise ☒

Noise Type **timeaverage**

Timeaverage: single-sided spectrum and harmonic-referred (modulated) noise analysis

Contribution Type:

☒ USB ☐ AM ☐ PM ☐ AM&PM ☐ ALL(AM,PM,USB,LSB)

Noise Separation ☒ yes ☐ no

Separate noise into source and gain

Enabled ☒

Options...

OK Cancel Defaults Apply Help

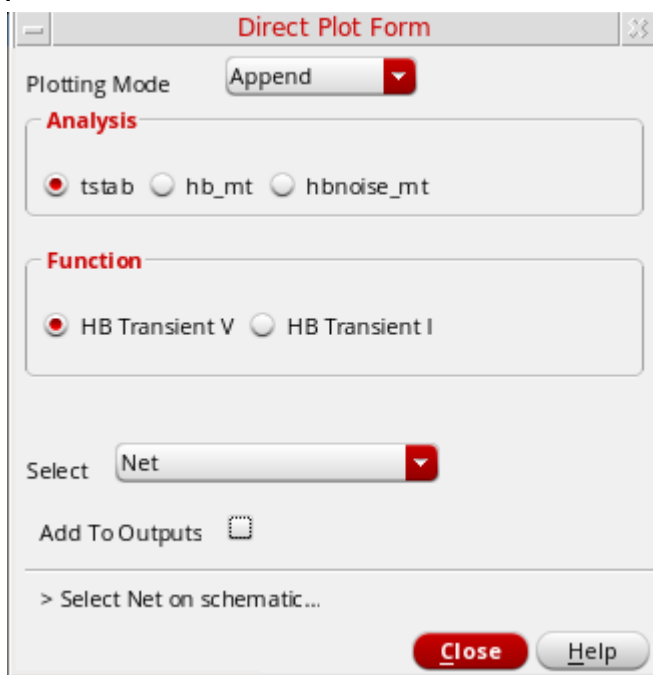
### Running the Simulation and Plotting the Results

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*.

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

---

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**

**Direct Plot Form**

Plotting Mode: Append

**Analysis**

☐ tstab ☒ hb\_mt ☐ hbnoise\_mt

**Function**

☒ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Power Added Eff. ☐ Power Gain Vs Pout  
☐ Comp. Vs Pout ☐ Node Complex Imp.

Select: Net

**Sweep**

☒ spectrum ☐ frequency ☐ ifft

Signal Level: ☐ peak ☒ rms

**Modifier**

☒ Magnitude ☐ Phase ☐ dB20  
☐ Real ☐ Imaginary

Add To Outputs: ☐

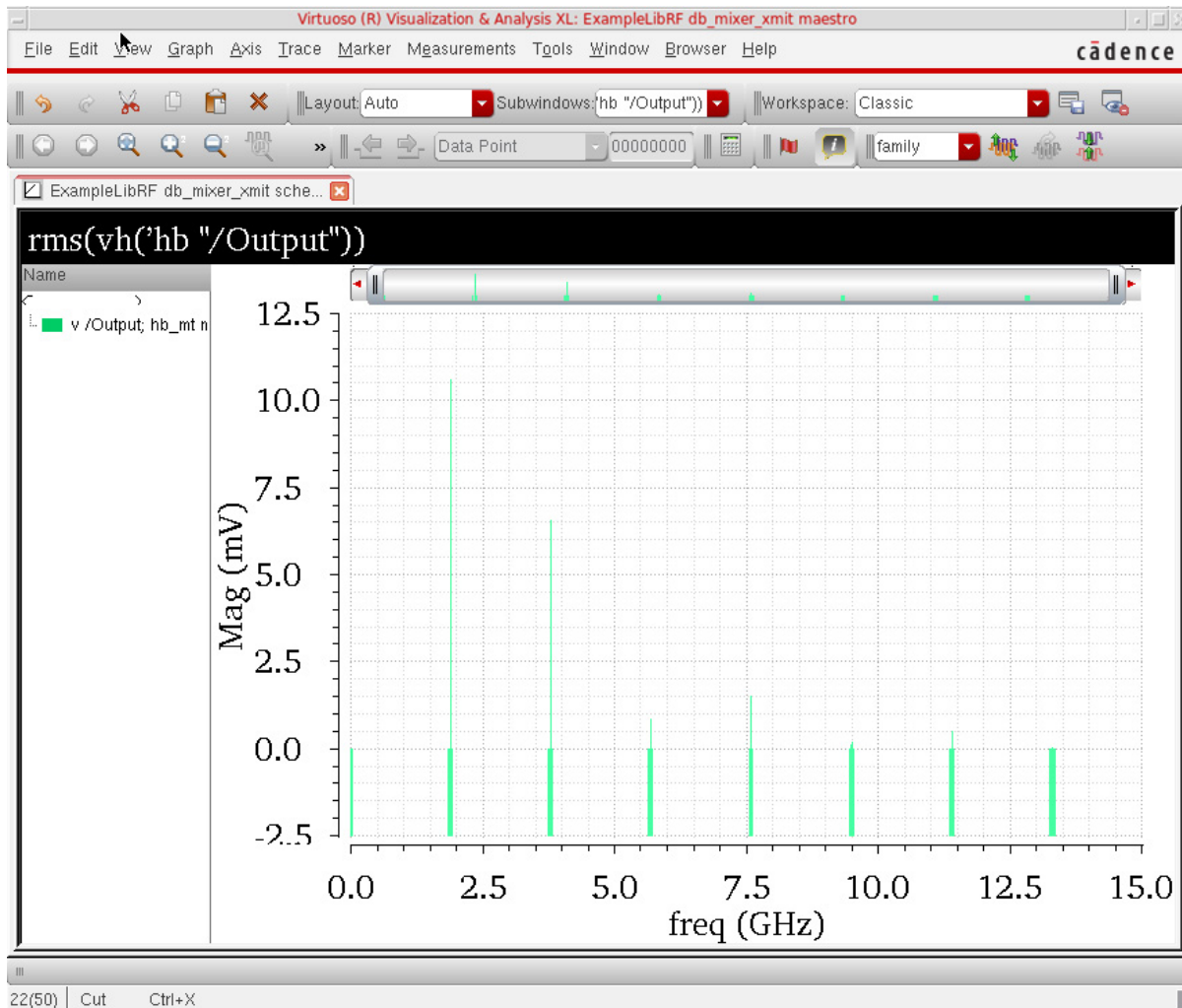
> Select Net on schematic...

Close Help

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

- Click on the net labeled *Output* on the right side of the schematic.
- The plot of Voltage Spectrum vs Frequency is displayed in the waveform window, as shown below.

**Figure 4-189 RMS Voltage vs Frequency**

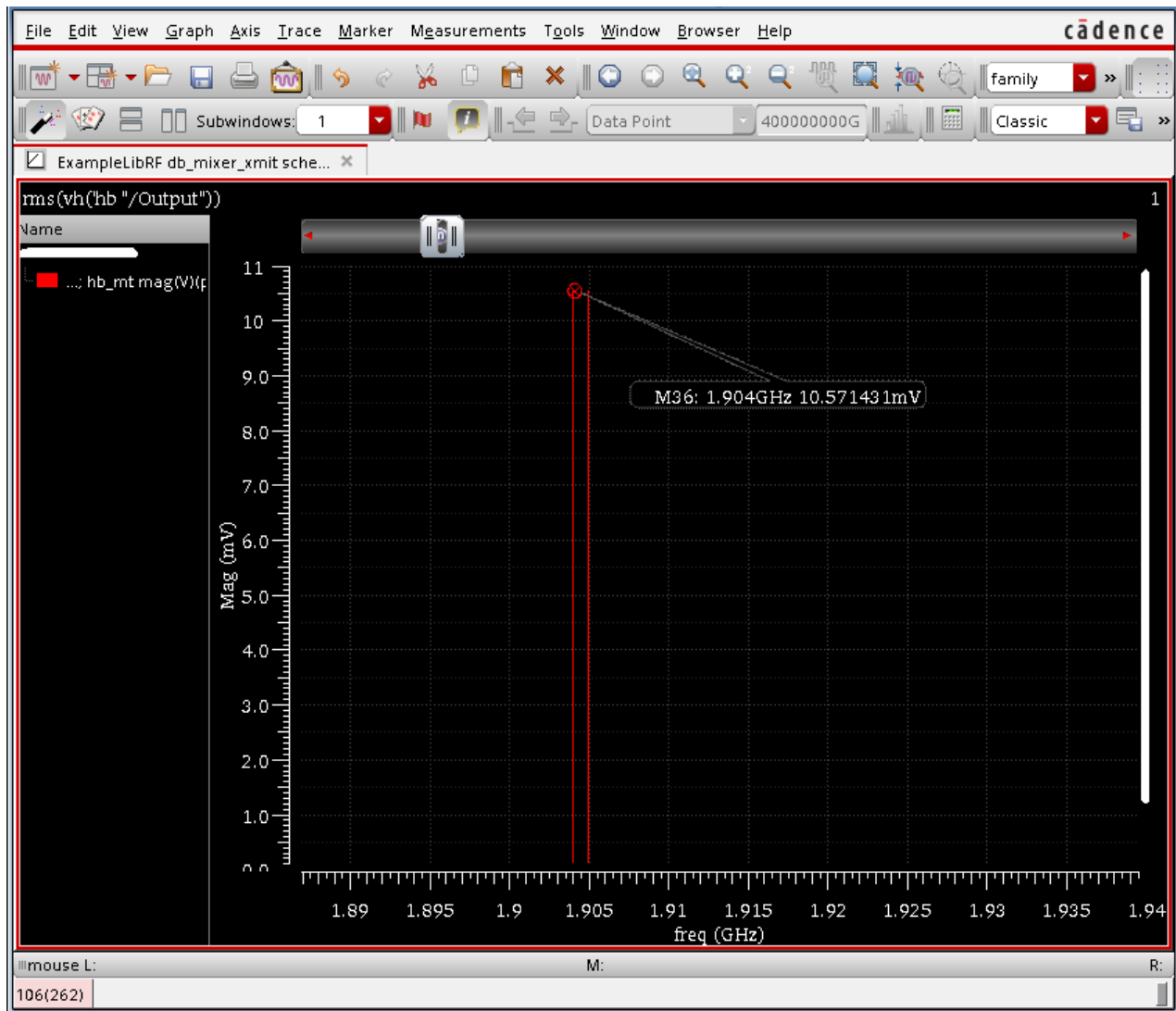


- Zoom into the region of the graph near 1.9GHz. Press and hold the right mouse button while moving the cursor to draw a box around the signals near 1.9GHz. When you release the mouse, you are zoomed in. You may need to do this several times to get zoomed in to just the area around 1.9GHz.
- Place a 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

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

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

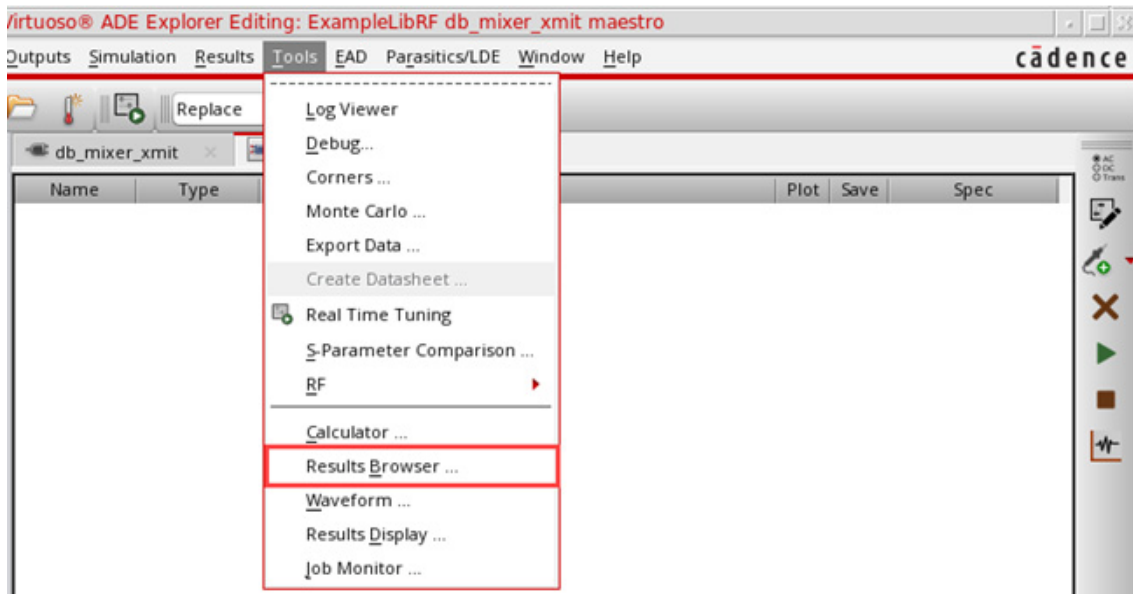
1. When the simulation has finished, plot the *Voltage Spectrum* in  $V/\sqrt{\text{Hz}}$  from the Results Browser.

ADE Explorer, select *Tools - Results Browser*.

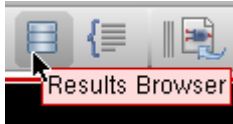
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

**Figure 4-191 Invoking the Results Browser After Simulation**

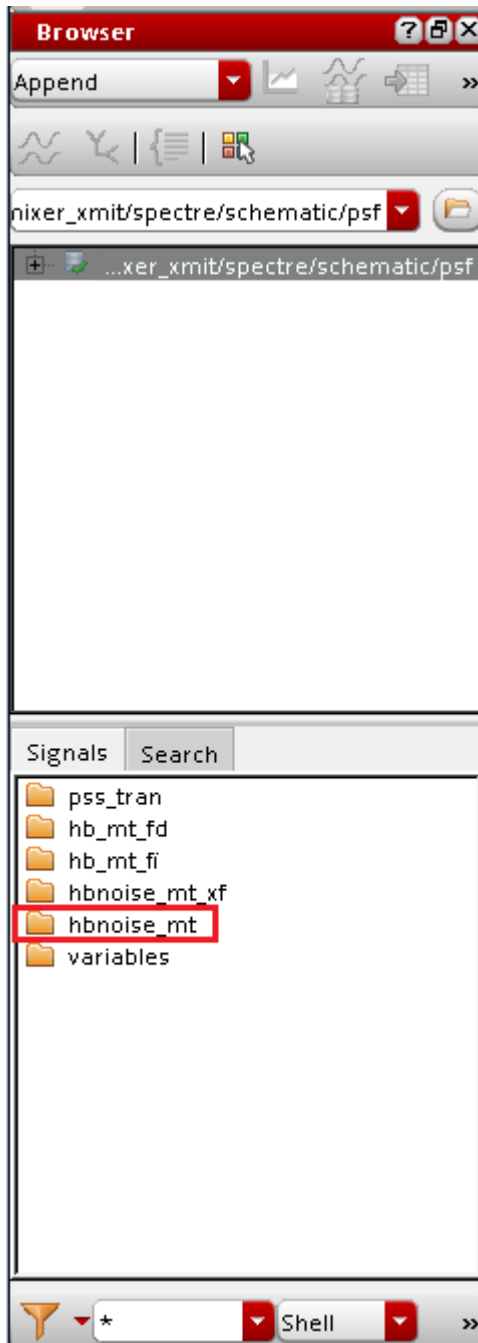


Alternately, you can click the Results Browser icon in the Schematic, as shown below.



2. In the Results Browser, select *hbnoise\_mt*.

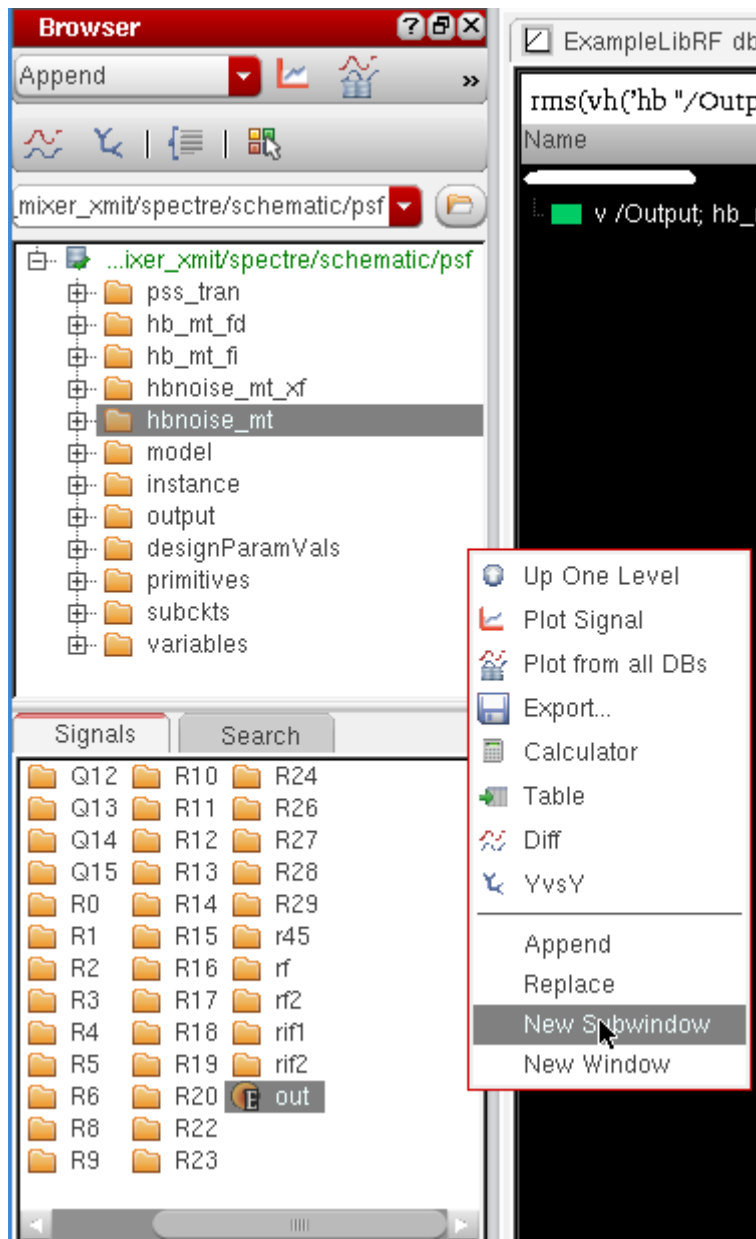
**Figure 4-192** Selecting hbnoise results using the Results Browser



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.



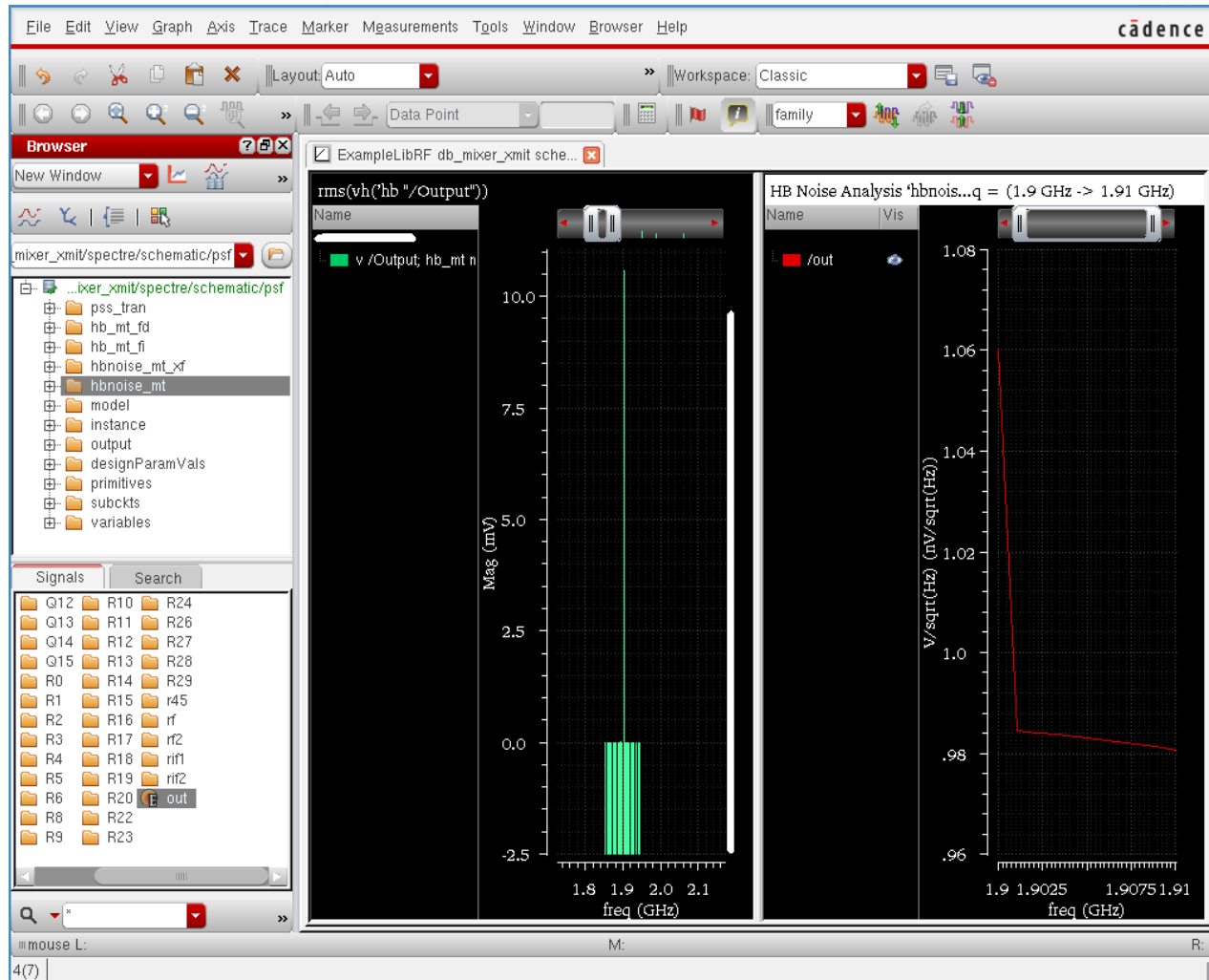
**Figure 4-193 Selecting out Net in the Results Browser**



4. Select and release the mouse over the *New Subwindow*. The Noise is plotted in  $V/\sqrt{\text{Hz}}$ , as shown below.

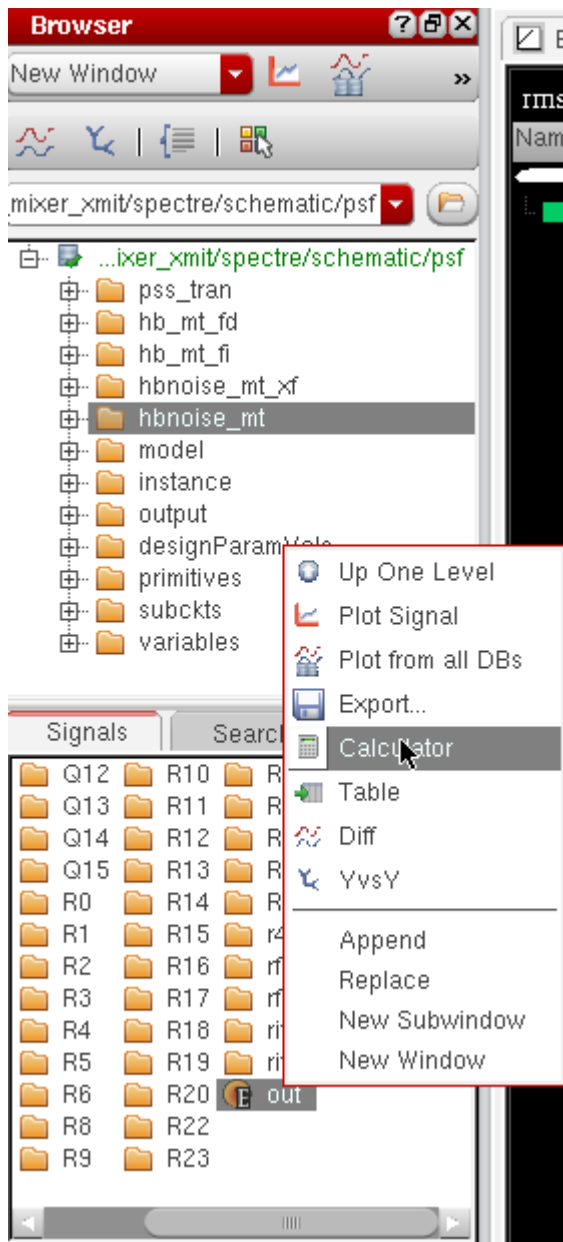
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

Figure 4-194 Noise Results



5. Note that the units are in  $V/\sqrt{\text{Hz}}$ . 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*.

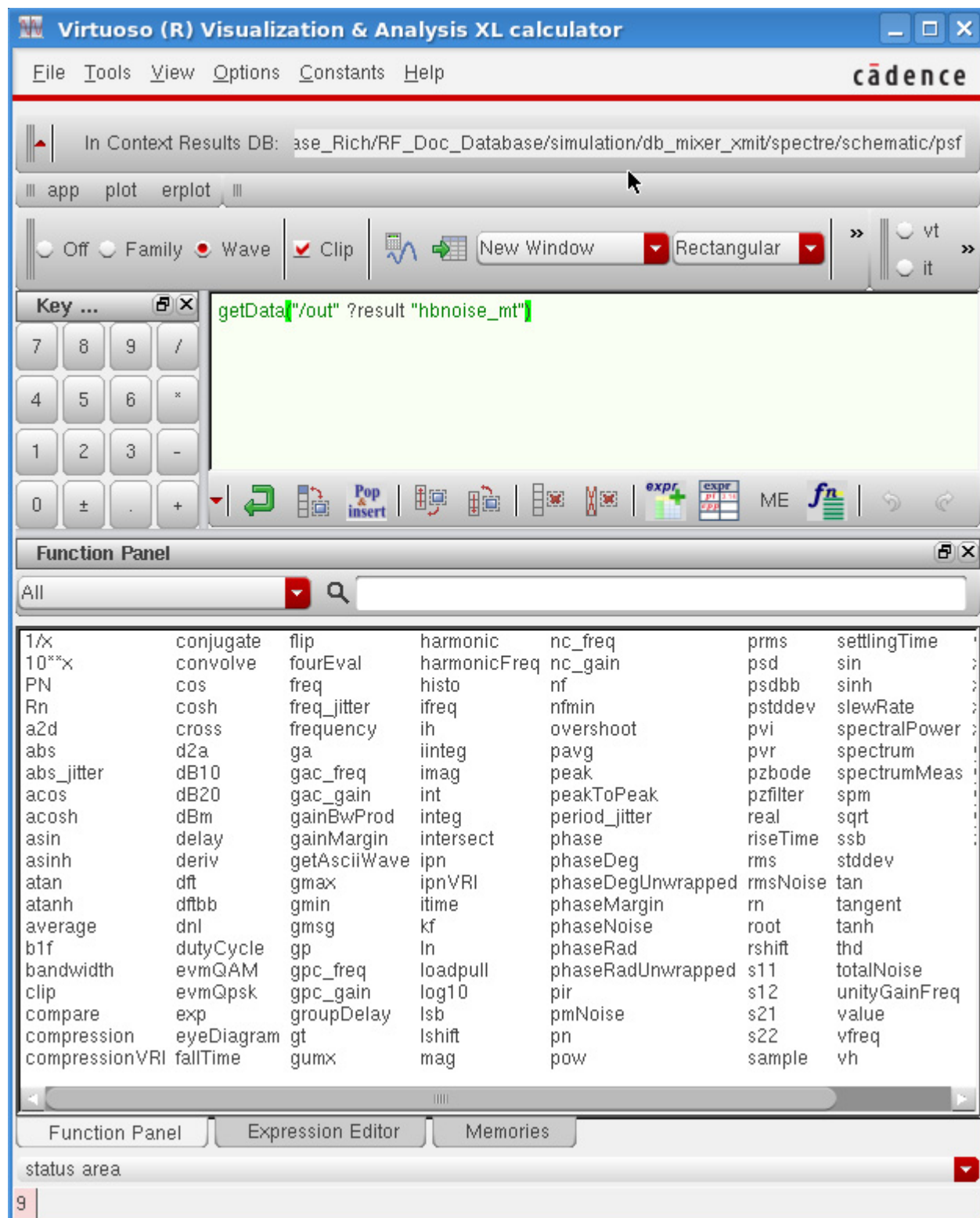
**Figure 4-195 Sending Output Waveform to Calculator**



The information is added in the Calculator Buffer, as shown below:

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

**Figure 4-196 Output Noise Data Entered into Calculator Buffer**



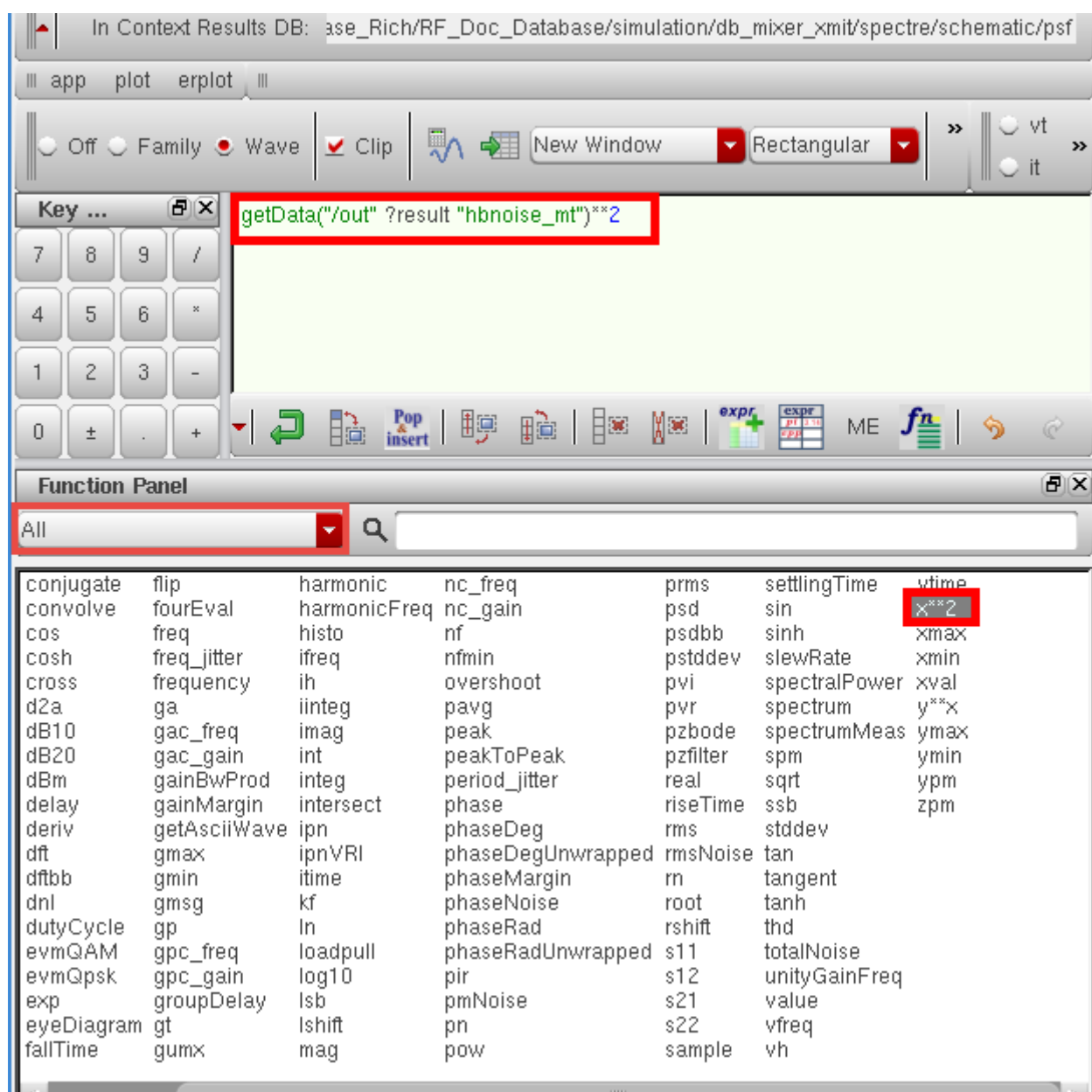
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

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**



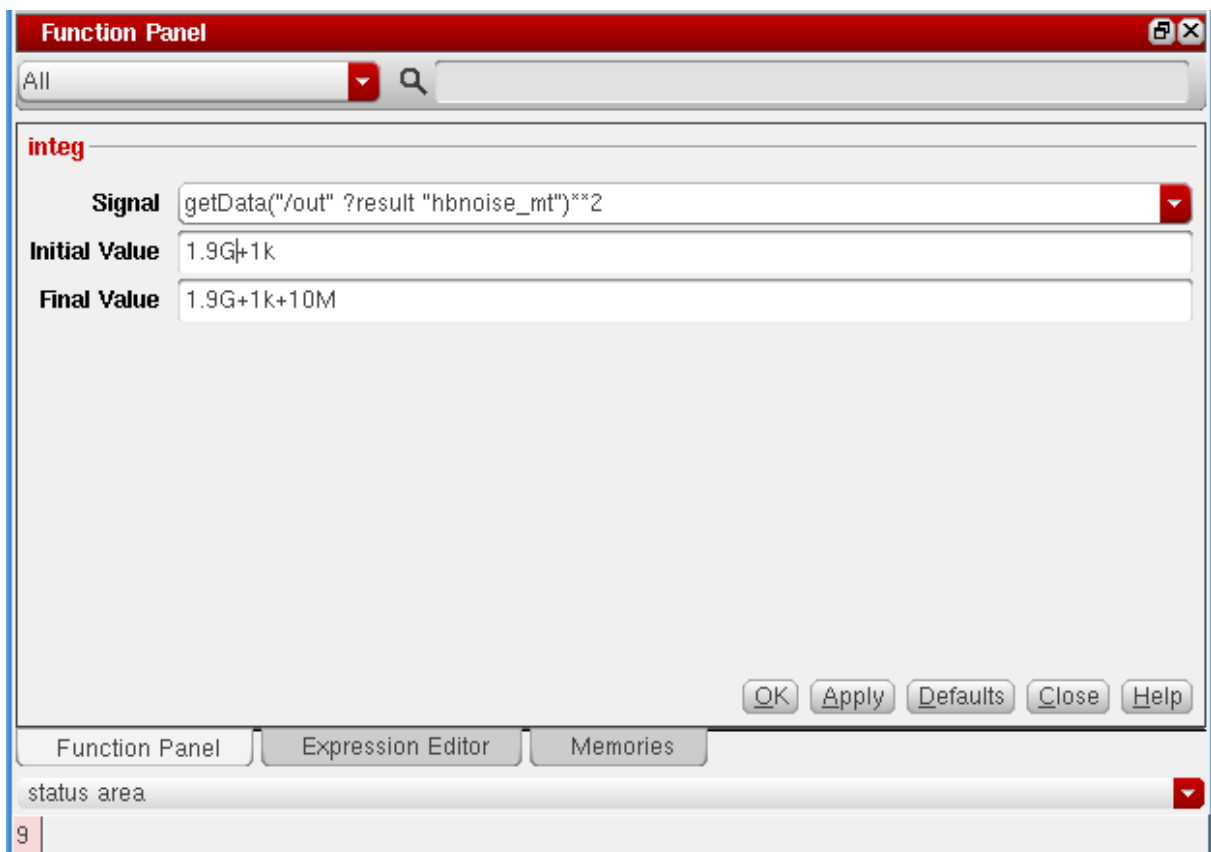
## Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer Workshop

---

- 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).
- 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*.

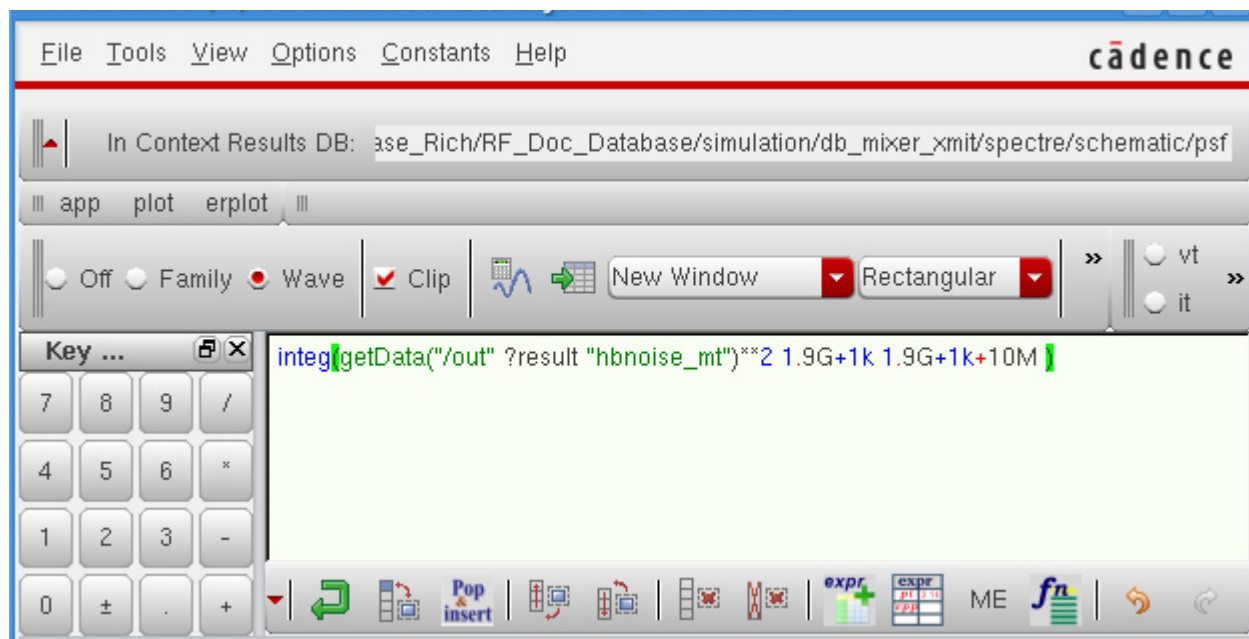
Note that this is the same range used in the hbnoise *Choosing Analyses* form.

**Figure 4-198 Integrating the Squared Noise Voltage over a 1Hz Bandwidth.**



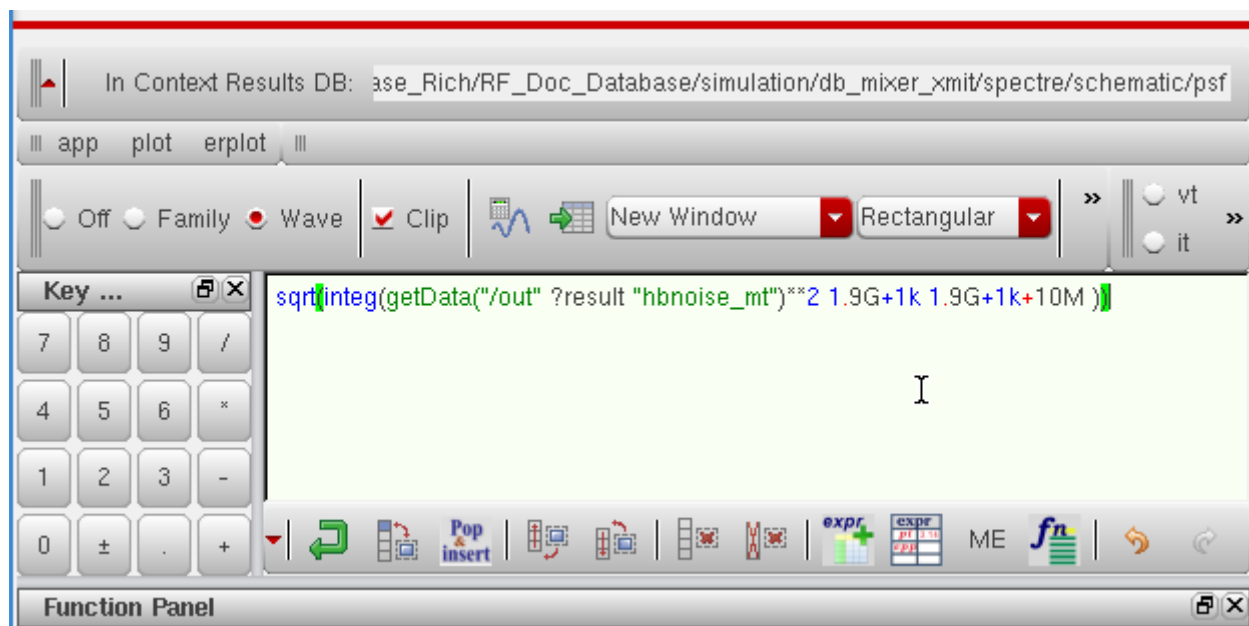
- Click *OK*. The Calculator buffer is updated, as shown in the figure below.

**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:

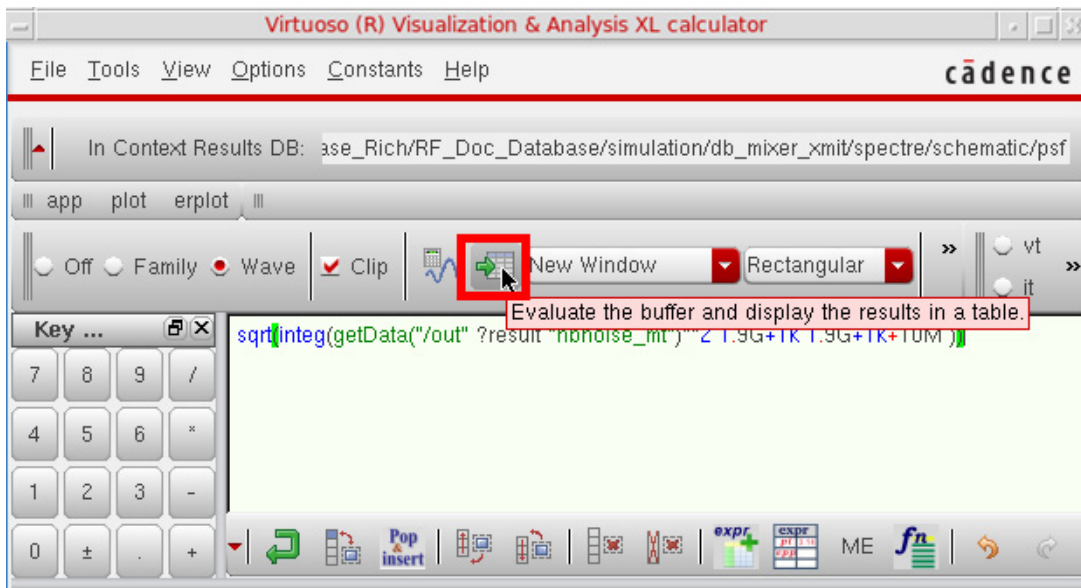
**Figure 4-200 Integrated Noise Voltage**



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

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.**



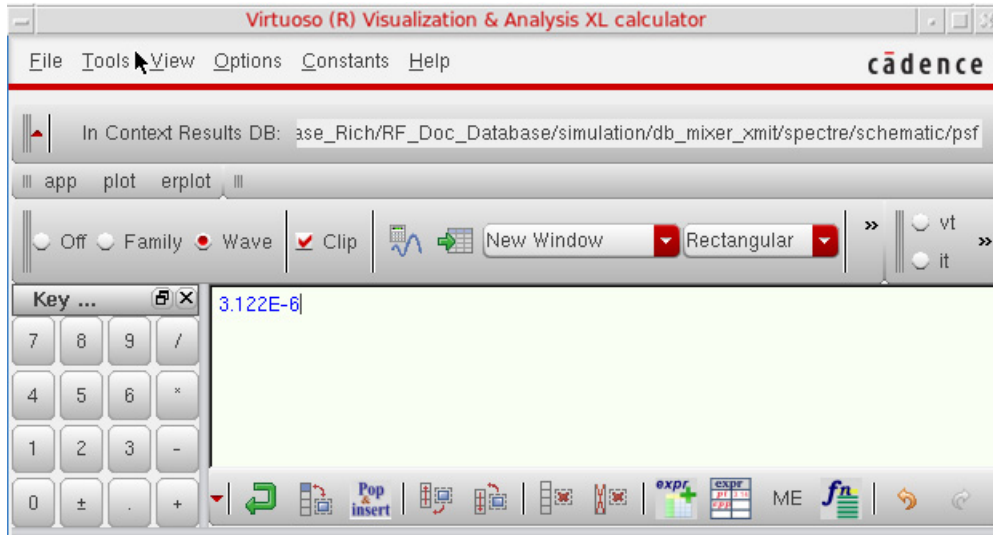
Alternately, if you click the *Plot* icon, the value for noise voltage appears in the Calculator buffer, as shown below.



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

---

**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.

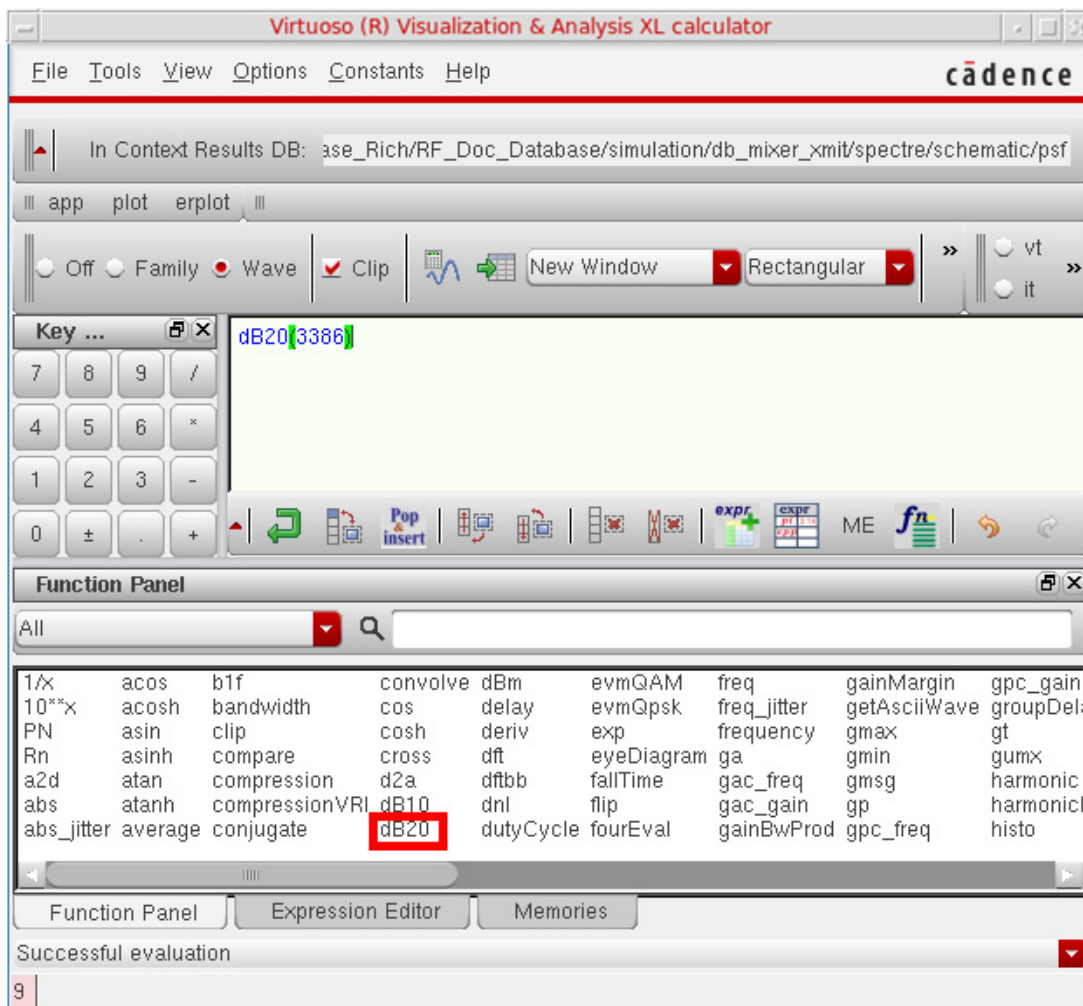
$$SNR = \frac{Signal}{Noise} = \frac{10.57134e^{-3}}{3.122e^{-6}} = 3386$$

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

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

---

- 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*.