Power Supply Rejection Ratio (PSRR) Characterization Using SpectreRF Driven and Autonomous Circuits

Rapid Adoption Kit (RAK)

Product Version: IC 6.1.8 ISR22, SPECTRE 20.1 ISR12
January, 2022

Copyright Statement

Contents

Purpose

This RAK illustrates how to use the SpectreRF tool within the ADE Explorer/Assembler environment to measure PSRR for driven and autonomous circuits.

Audience

This document is intended for users of SpectreRF within the ADE Explorer/Assembler environment.

Overview

The measurement of PSRR is important for improvements of immunity to noise sources such as power supply ripples due an interference or to the digital part of the system. The same methodology is used to measure the effect of the substrate noise coupled through its deep portion.

Tool Versions

This RAK has been verified using IC.6.1.8. ISR22 and Spectre20.1. ISR12 with Spectre/SpectreX mode.

Setting Up RAK Database

Unzip and untar the `PSRR_RAK.tar.gz` database.

```
Unix> tar -zxvf PSRR_RAK.tar.gz
```

Navigate to the `OPAMP_RAK` directory.

```
Unix> cd PSRR_RAK
```

Lab 1: PSRR Measurement of Driven Circuits Using PSS and PXF Analyses

This lab uses divider as a driven circuit example. Nominal input frequency is around 2GHz. The ratio is 16 with the output at 125MHz.

To measure PSRR of the divider, you set up the testbench using dc voltage sources as the power supply and the substrate, while pulse voltage source represents the input.

Action 1: Invoke Virtuoso.

```
Unix> virtuoso &
```

Action 2: Open the Library Manager from the CIW window by navigating to **Tools > Library Manager**.

Action 3: From the Library Manager, open **PSRR_sim > Driven_testbench > schematic** view.

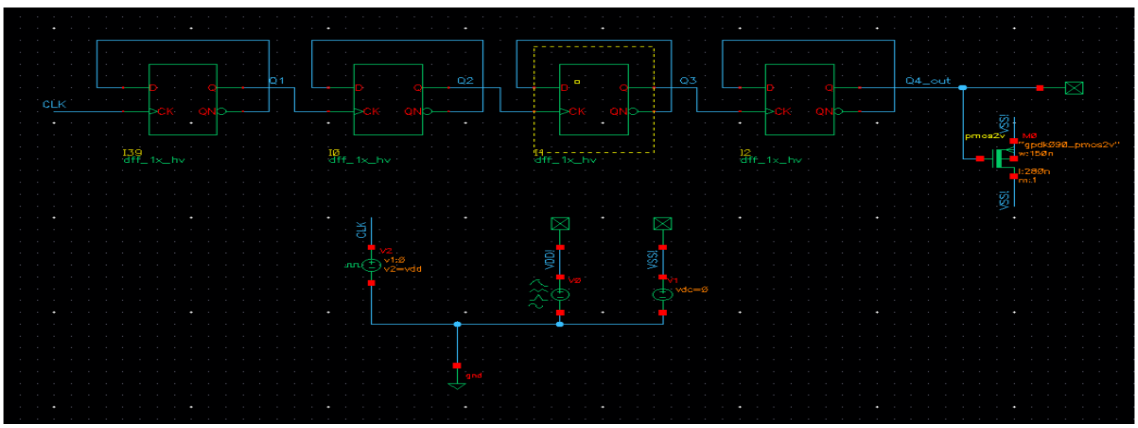

Figure 1: Driven circuit testbench

Action 4: From the schematic, select **Launch > ADE Explorer**.

Action 5: In the **Launch ADE Explorer** window that opens, select **Open Existing View** and click **OK**.

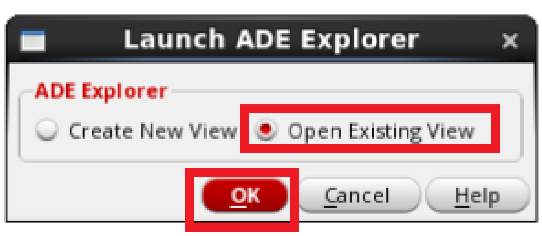

Figure 2: Launching ADE Explorer from schematic window

Action 6: In the **Open ADE Explorer View** window that opens, select the view as **maestro** and click **OK**.

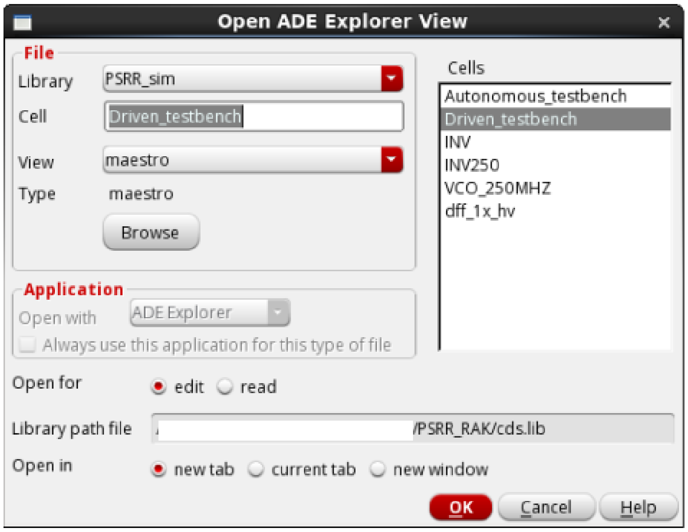

Figure 3: Opening maestro view of Driven_testbench cell

The **ADE Explorer** window opens as shown in Figure 4.

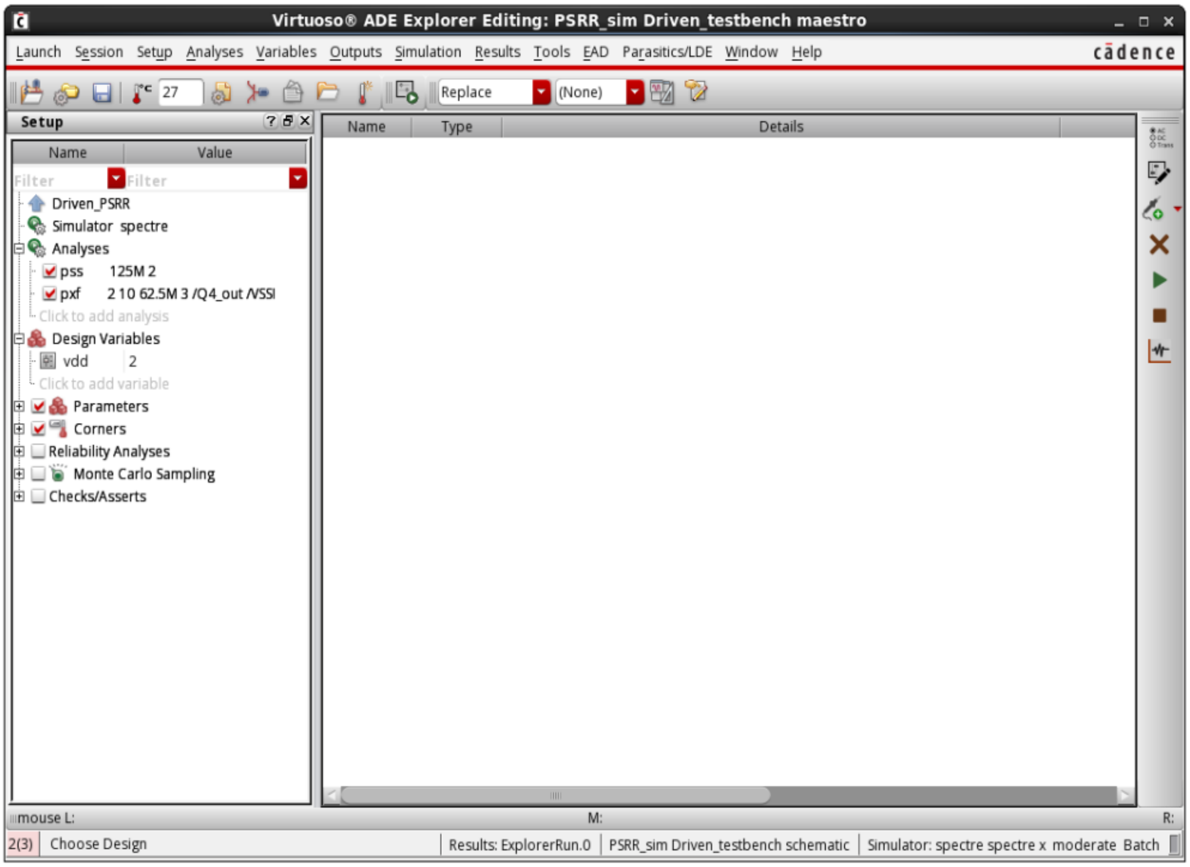

Figure 4: ADE Explorer window showing maestro view of Driven_testbench cell

Action 7: In the **ADE Explorer** setup view, double-click on the **PSS** analysis or right-click and select **Edit** to review the **PSS** analysis setup.

- **Beat Frequency** is set to **125MHz** for **2GHz** input frequency, corresponding to the expected low frequency output of **125MHz**.
- Use **moderate** accuracy settings.
- **Number of harmonics** is set to **2**.
- **Stop Time (tstab)** is set to **8ns**; a single period of **Beat Frequency** is often enough.
- Click **OK**.

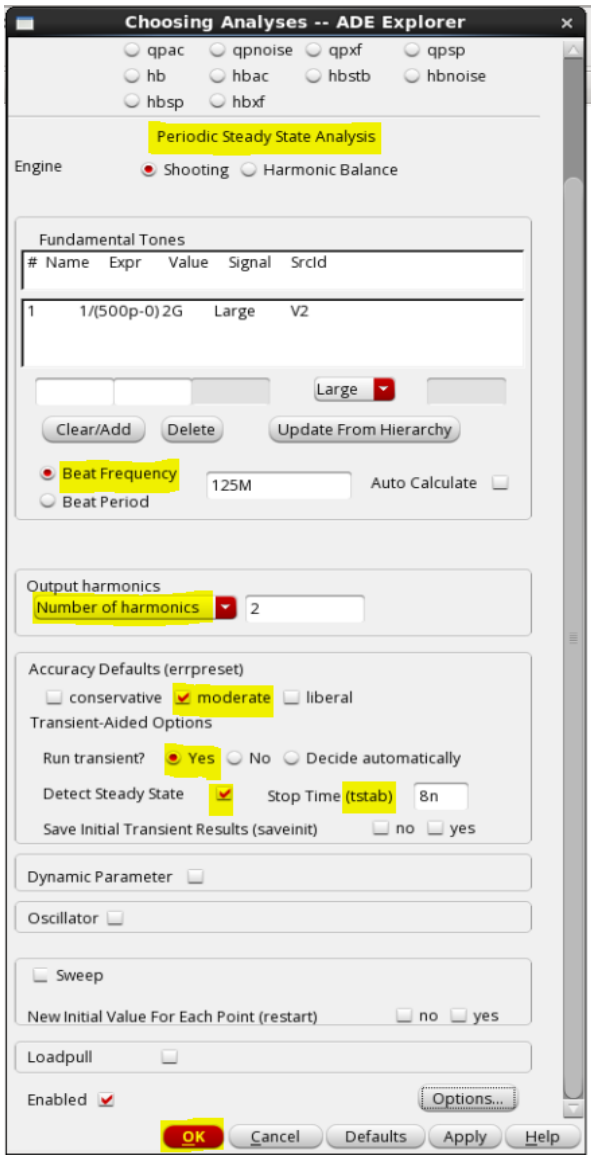

Figure 5: Setup of PSS analysis

Action 8: In the **ADE Explorer** setup view, double-click on the **PXF** analysis or right-click and select **Edit** to review the **PXF** analysis setup.

- **sweeptype** is set to **absolute** from the drop-down list, but it could also be "**relative**" to zero harmonic for the Baseband output. The default for driven circuits is **absolute**.
- **Output Frequency Sweep Range** is set from **10** to **62.5MHz**.
- **Sweep Type** is set to **Logarithmic** and **Points Per Decade** to **3**.
- **Maximum sideband** is set to **2**.
- **voltage** is enabled in the **Output** section, **Positive Output Node** > **/Q4_out** and **Negative Output Node > /VSS!** are selected from the schematic view.
- **Specialized Analyses** is set to **Sampled** from the drop-down list.
- **Threshold** is set to **1**. The triggering event will be the crossing of the threshold by the output signal. Half amplitude **1V** is used for the threshold.
- **Crossing Direction** is set to **all** from the drop-down list. The PXF runtime will be proportional to the number of crossings to analyze, so you need to be careful while selecting the fast switching output. In this case, **all** is used since you will have one of each kind only.
- Click **OK**.

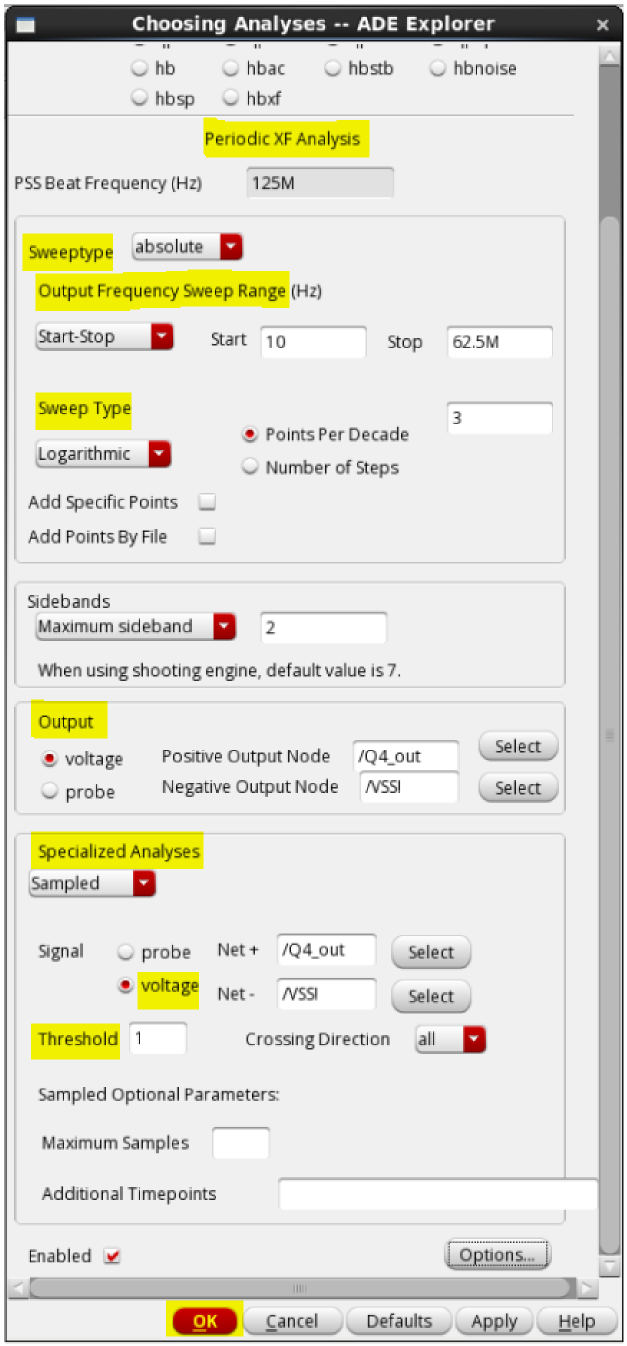

Figure 6: PXF setup for driven circuit

Note: In the **PXF** setup form, **Maximum Samples** will limit the number of events to compute if you expect a lot of events during simulation. This happens for signals with a much shorter period than the period of PSS fundamental.

If you want to measure specific pss times, specify a list of pss simulation times in the **Additional Timepoints** field. Separate the entries with a space.

Action 9: In the **ADE Explorer** window, click on the green arrow button  to **netlist and run** the simulation.

Action 10: Once the simulation finishes, click on the **Results > Direct Plot > Main Form** in the **ADE Explorer** window as shown in Figure 7 to open **Direct Plot Form**.

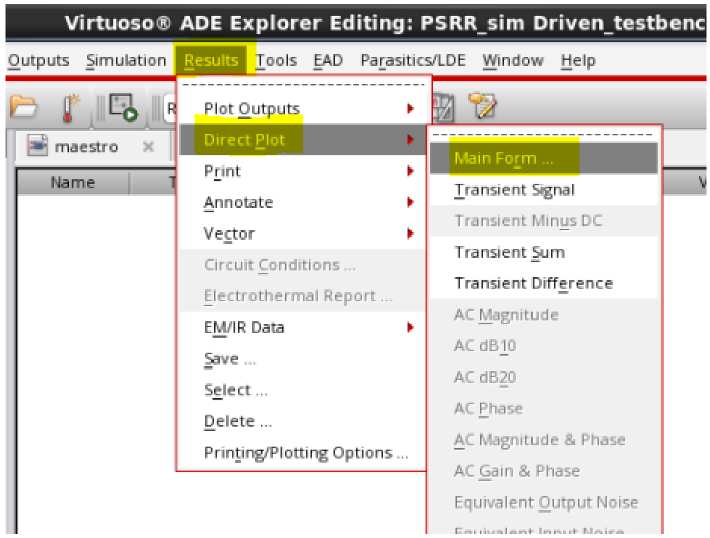

Figure 7: Opening Direct Plot Form from ADE Explorer window

Action 11: **Direct Plot Form** appears as shown in Figure 8. Do the following setup:

- Select **pxf** as **Analysis**.
- Select **Voltage Gain** as **Function**.
- Select suitable **Modifier**; use **Magnitude** or **dB20**.
- Select the crossing **Variable Value** (**eventtime**). In this case, select **4.999n** as **eventtime**.

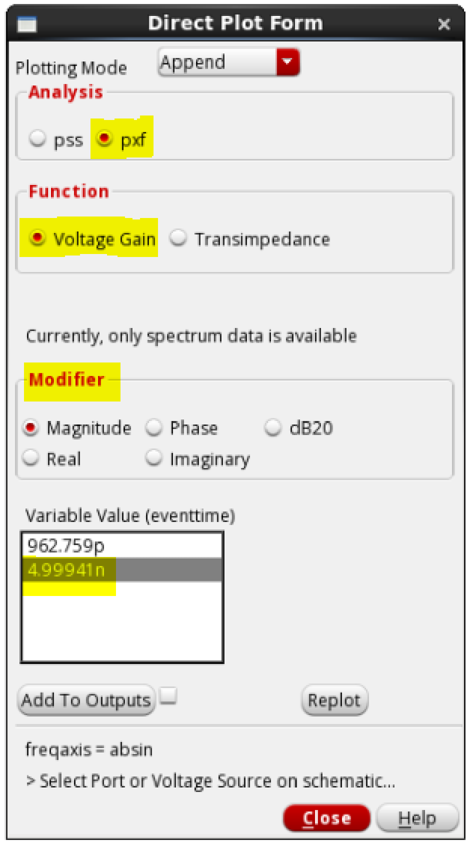

Figure 8: Direct Plot Form setup

- To plot the power supply to the output transfer function, select the VDD supply source in the schematic as shown in Figure 9.

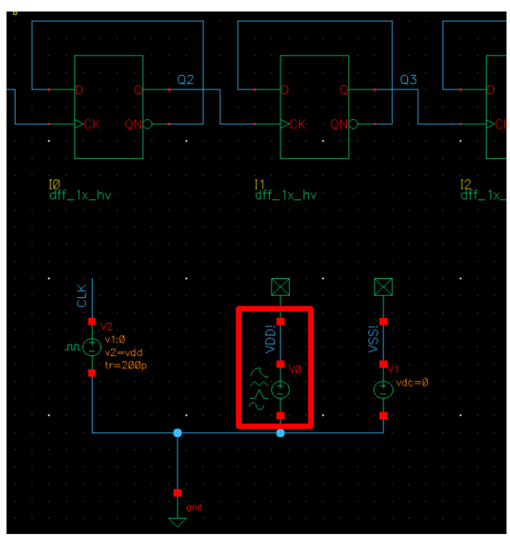

Figure 9: Select voltage source from schematic

Action 12: The **Voltage Gain** plot (in V/V) will appear as shown in Figure 10.

The frequency translation is indicated on the plot by **harmonic=0** or **harmonic=-1** and so on. The **0** case corresponds to the input being at the same frequency as the output, while **-1** to the input frequency being one fundamental of the PSS below the output frequency. That is, if the output is near the PSS fundamental tone, in case of **-1**, the input is at baseband.

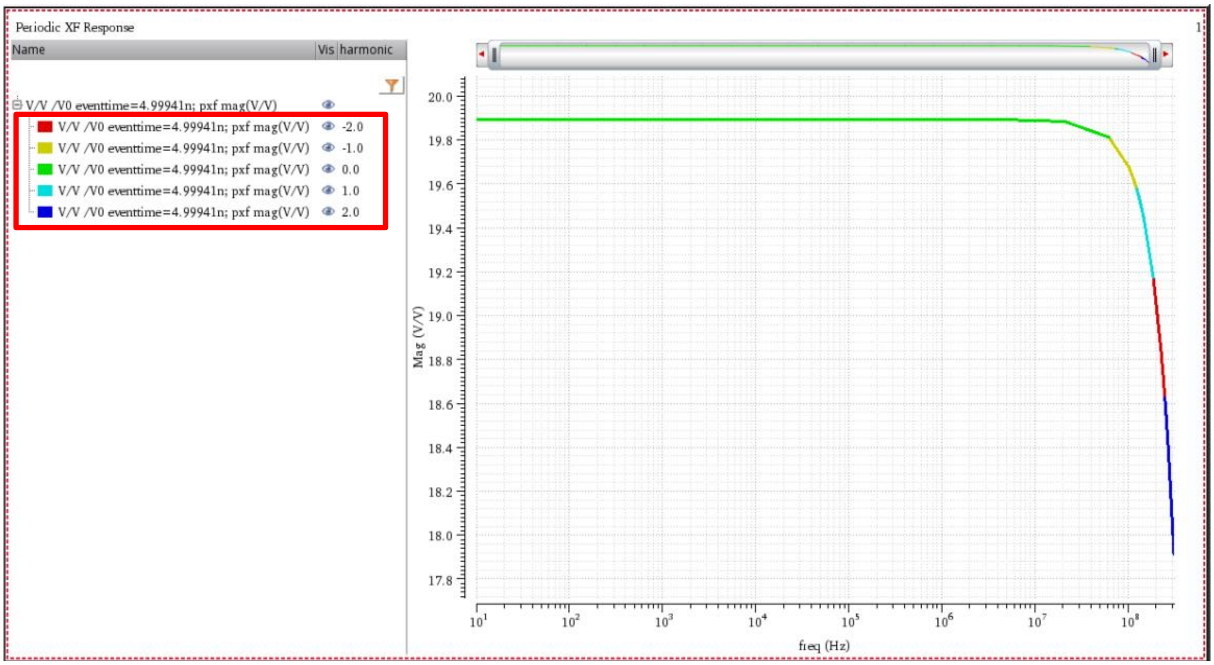

Figure 10: Voltage Gain plot in ViVA XL window

Action 13: Close all windows but leave the **Virtuoso** session open for the next lab.

Lab 2: PSRR Measurement of Autonomous Circuits Using PSS and PXF Analyses

This lab uses VCO as an autonomous circuit example. Nominal frequency is around 250MHz.

To measure PSRR of the oscillator, you set up the testbench using dc voltage source as the control voltage input. The power and substrate voltages are supplied by dc voltage sources. The dc voltage values are parametrized using the device property in **Schematic Editor**, vcntl for VCO input and VDD for the power supply. The nominal values for the oscillator are 1.**25V** and **2.5V** correspondingly.

Action 1: From the Library Manager, open **PSRR_sim > Autonomous_testbench > schematic** view. The schematic window appears as shown in Figure 11.

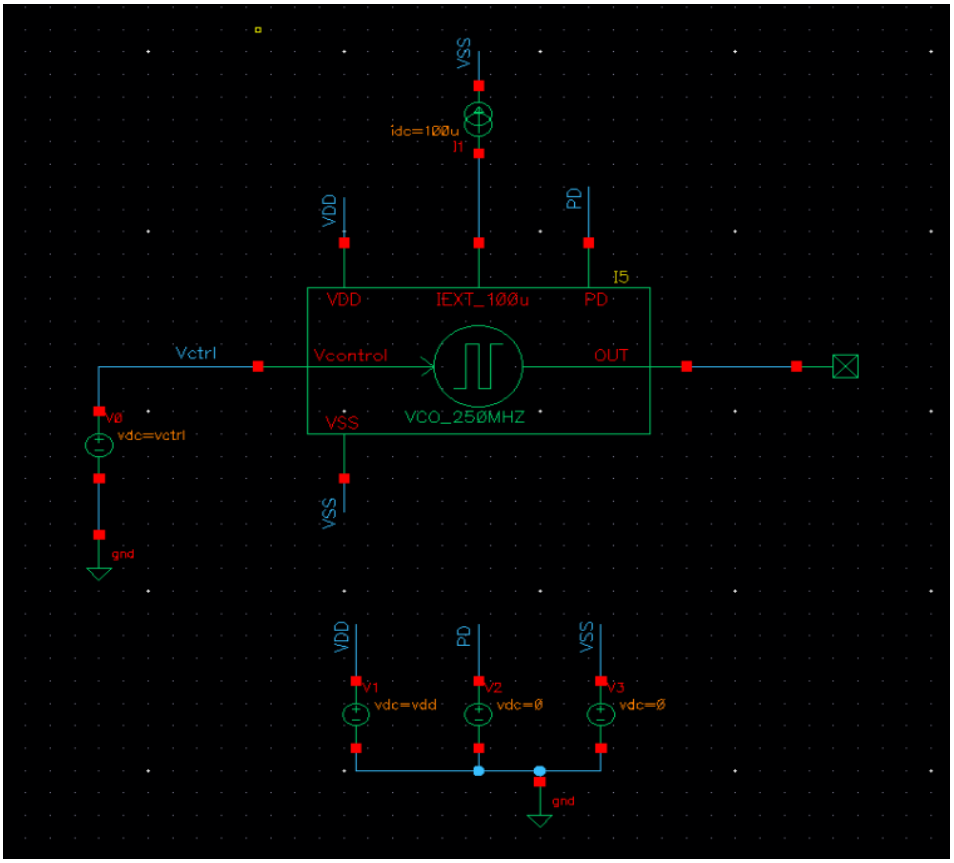

Figure 11: Schematic of the autonomous circuit testbench

Action 2: From the schematic, select **Launch > ADE Explorer**.

Action 3: In the **Launch ADE Explorer** window that opens, select **Open Existing View** and click **OK**.

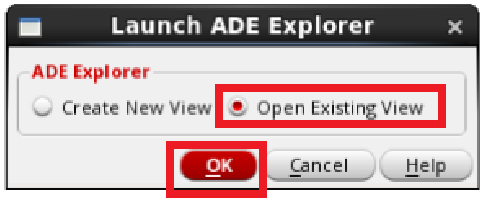

Figure 12: Launching ADE Explorer from schematic window

Action 4: In the **Open ADE Explorer View** window that opens, select the view as **maestro** and click **OK**.

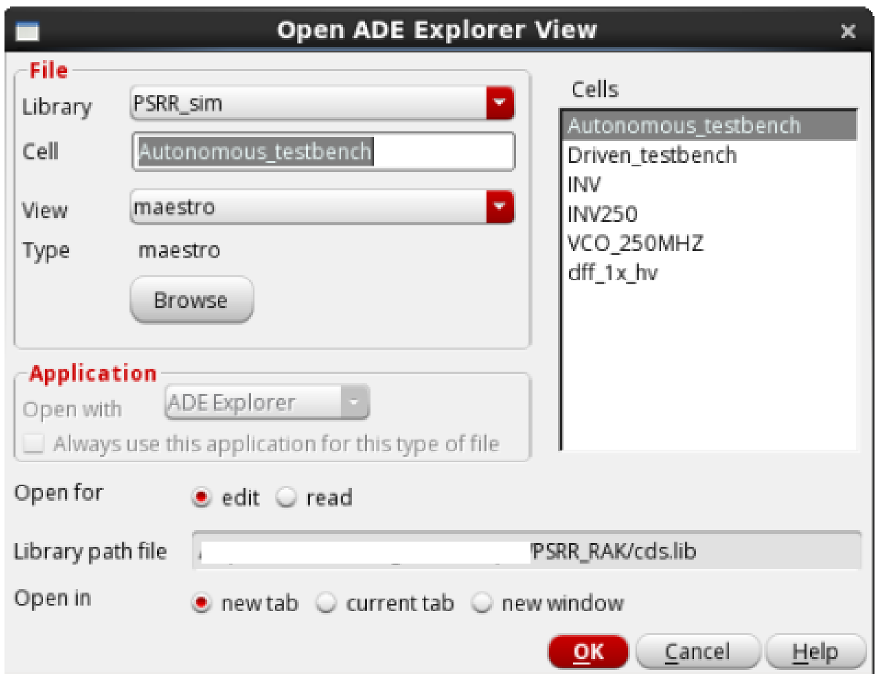

Figure 13: Opening maestro view of Autonomous_testbench cell

The **ADE Explorer** window appears as shown in Figure 14.

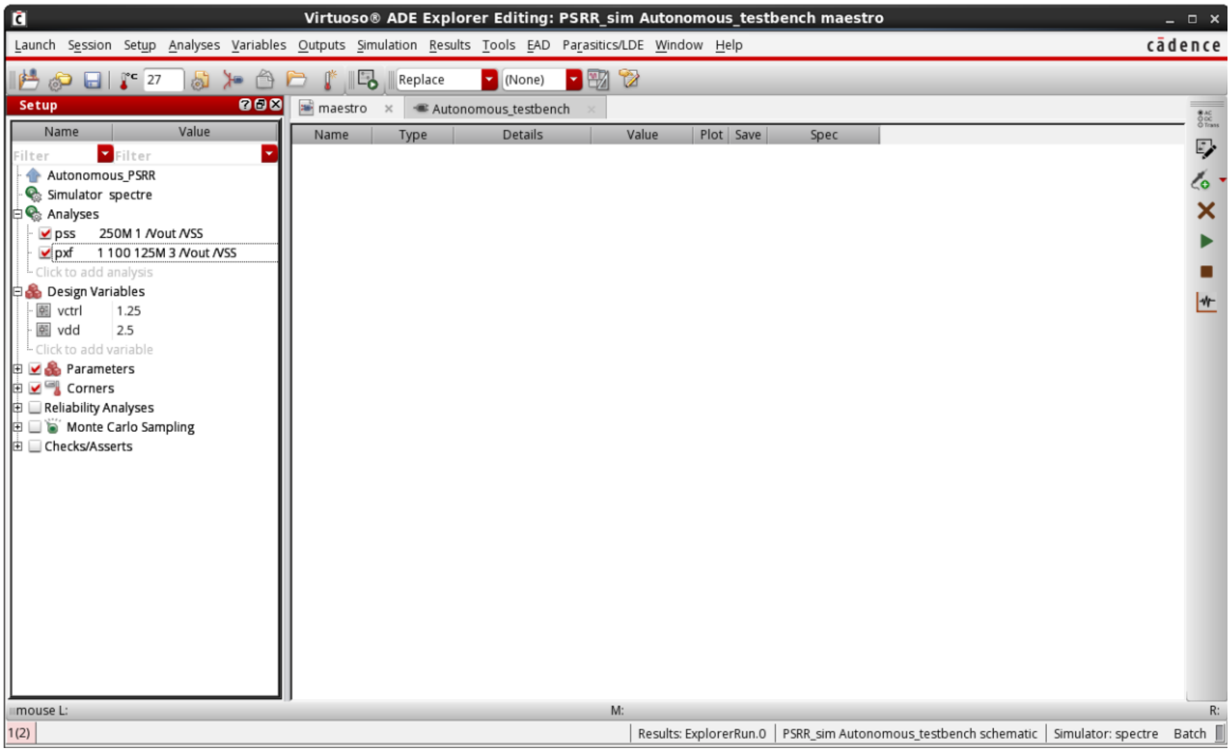

Figure 14: ADE Explorer window showing maestro view of Autonomous_testbench cell

Action 5: In the **ADE Explorer** setup view, click on **Simulation > Convergence Aids > Initial Conditions** as shown in Figure 15.

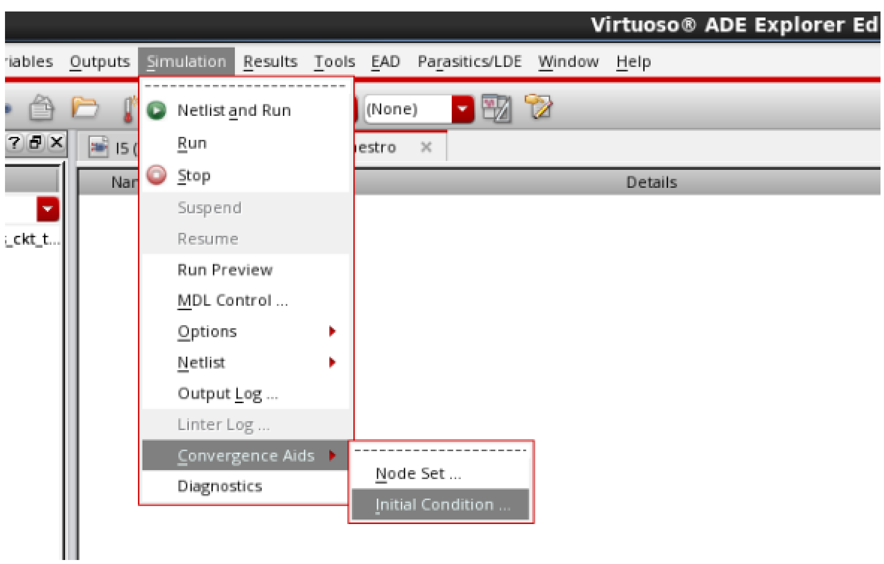

Figure 15: Opening Select Initial Condition Set window from ADE Explorer window

The **Select Initial Condition Set** window appears as shown in Figure 16.

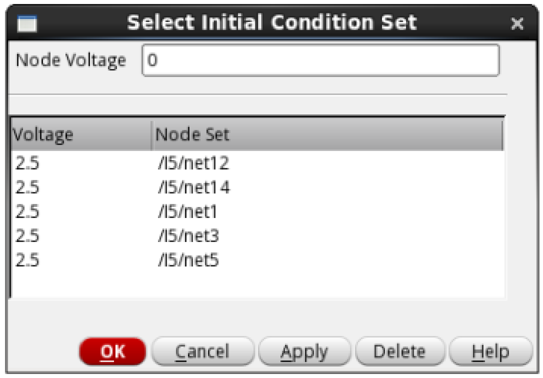

Figure 16: Select Initial Condition Set window

Action 6: In the **Select Initial Condition Set** window, descend into the schematic and review the **Node Voltage** 2.5V (high) setup on nodes between the stages of the ring oscillator circuit. Click **OK** as shown in Figure 17.

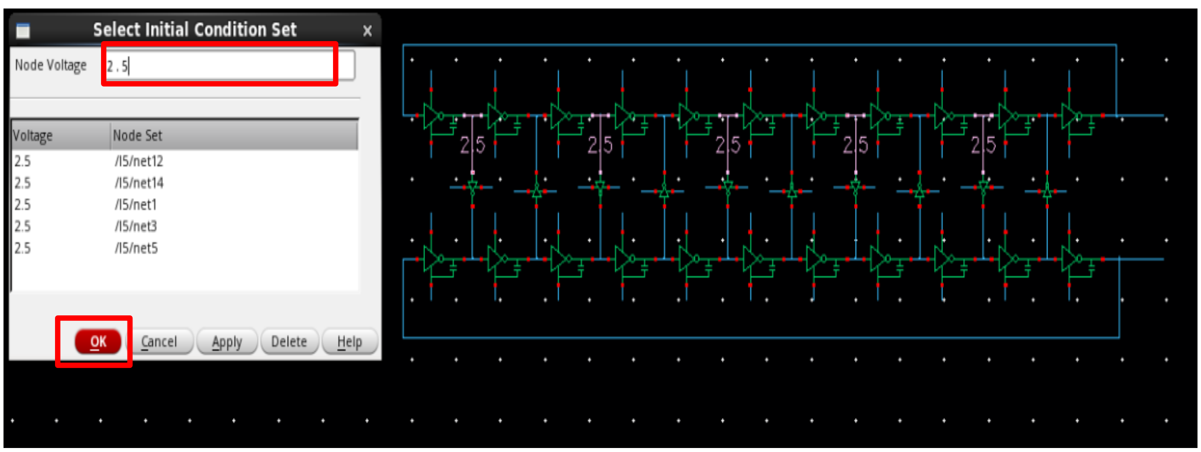

Figure 17: Set Node Voltage in Select Initial Condition Set window

Action 7: In the **ADE Explorer** setup view, double-click on the **PSS** analysis or right-click and select **Edit** to review the **PSS** analysis setup.

- **Beat Frequency** is set to **250MHz**.
- **Accuracy Default (errpreset)** is set to **moderate**.
- **Stop Time (tstab)** is set to **40ns**. Use several estimated periods.
- The **Oscillator** option is selected; the net pair **Oscillator node+** "/Vout" and the **Oscillator node-** "/VSS" are used.

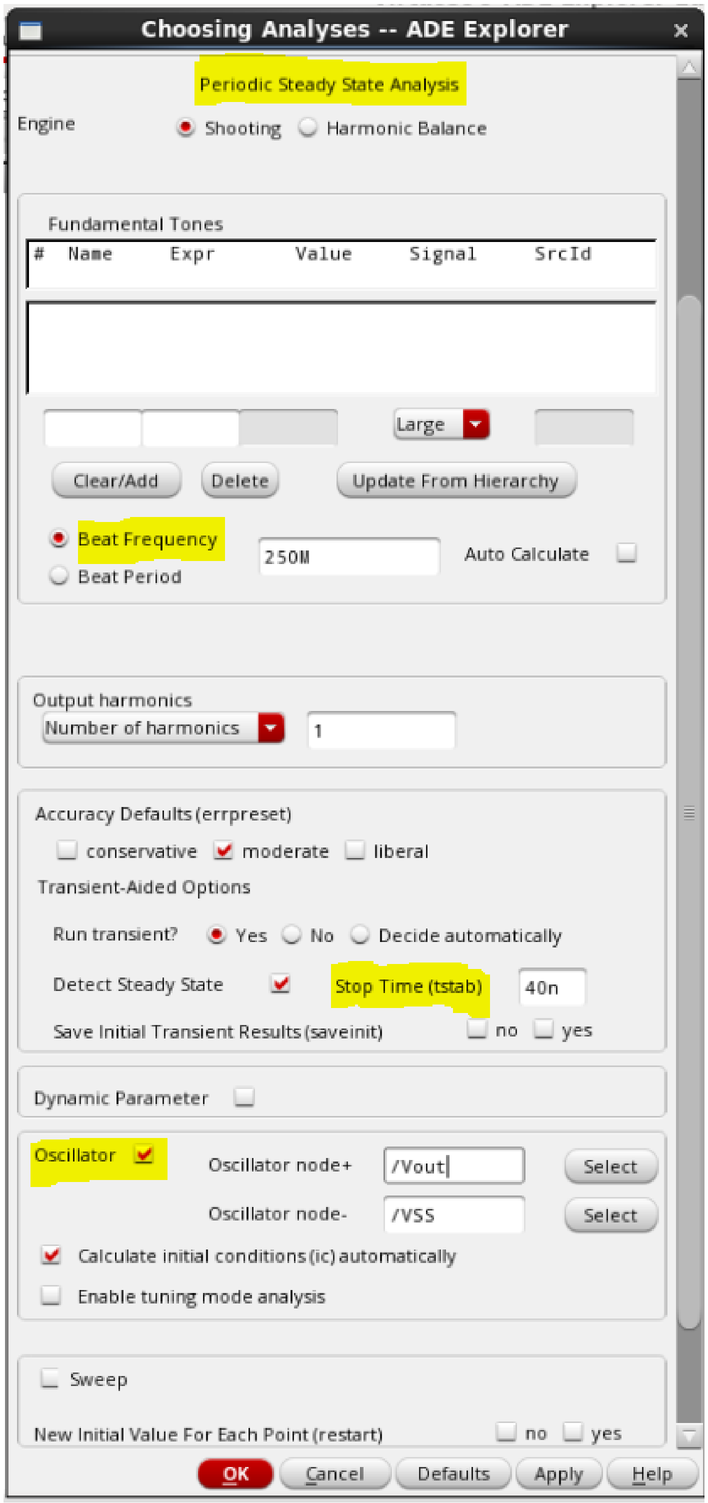

Figure 18: Setup of PSS analysis

Action 8: In the **ADE Explorer** setup view, double-click on the **PXF** analysis or right-click and select **Edit** to review the **PXF** analysis setup.

- **sweeptype** is set to **relative** from the drop-down list.
- **Relative Harmonic** is set to **1**.
- **Output Frequency Sweep Range** is set from **100** to **125MHz**.
- **Sweep Type** is set to **Logarithmic** and set **3** points per decade.
- **Maximum sideband** is set to 1.
- In the **Output** section, the voltage signal pair of **Positive Output Node** > **/Q4_out** and **Negative Output Node > /VSS!** are used.
- **Specialized Analyses** is set to **Modulated** from the drop-down list.
- **Output Type** is set to **SSB/AM/PM** from the drop-down list. Since you are interested in phase component at the output of the oscillator, this will provide you with the complete set of small signal transfer functions.
- In **Input Modulated Harmonic List**, click the **Choose** button to the right of the field. The Harmonic is set to **0** from the list of available frequencies, click **OK**.

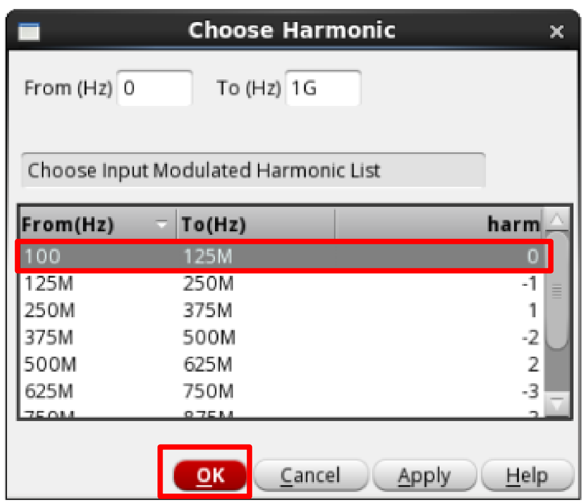

Figure 19: Choose Input Modulated Harmonic List form

- In **Output Modulated Harmonic**, click the **Choose** button to the right of the field. The Harmonic is set to **1** from the list of available frequencies, click **OK**.

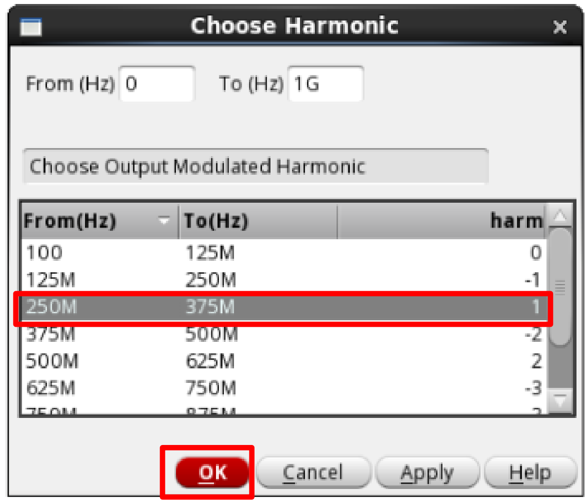

Figure 20: Choose Output Modulated Harmonic form

Note: Here input ripples or noise of interest is at baseband. So, we choose **0** in the **Input Modulated Harmonic List**. The output harmonics, we choose **1** for **Output Modulated Harmonic**.

- Click **OK**.

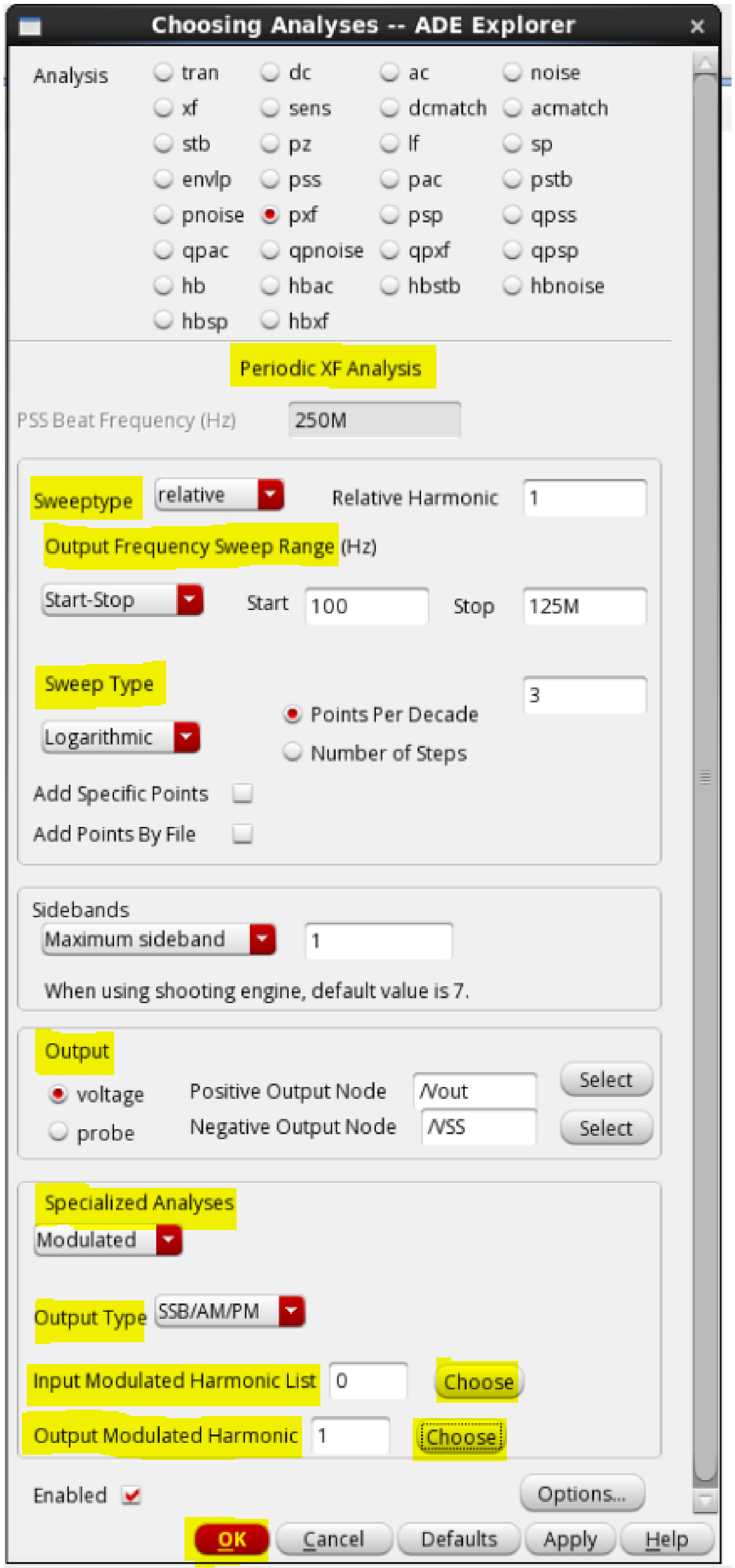

Figure 21: Setup of Modulated PXF analysis

Action 9: In the **ADE Explorer** window, click on the green arrow button  to run the simulation.

Action 10: Once the simulation finishes, click on the **Results > Direct Plot > Main Form** in the **ADE Explorer** window as shown in Figure 22 to open **Direct Plot Form**.

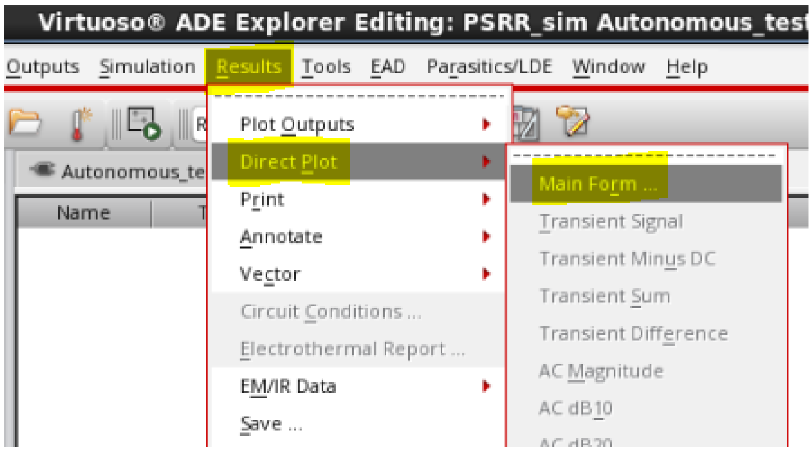

Figure 22: Opening Direct Plot Form from ADE Explorer window

Action 11: In **Direct Plot Form**, do the following setup:

- Select **pxf modulated** as **Analysis**.
- Select **SSB** for the non-modulated **input**; harmonic **0** indicates the baseband frequency range. **SSB** measures the gain from the signal on one side of the carrier to the PM component. This is appropriate for a PSRR measurement where the ripple frequency is near zero.
- The output is **PM** and the modulated output harmonic of **1** is selected.
- Select **Voltage Gain** as **Function**.
- Select **dB20** as **Modifier**.

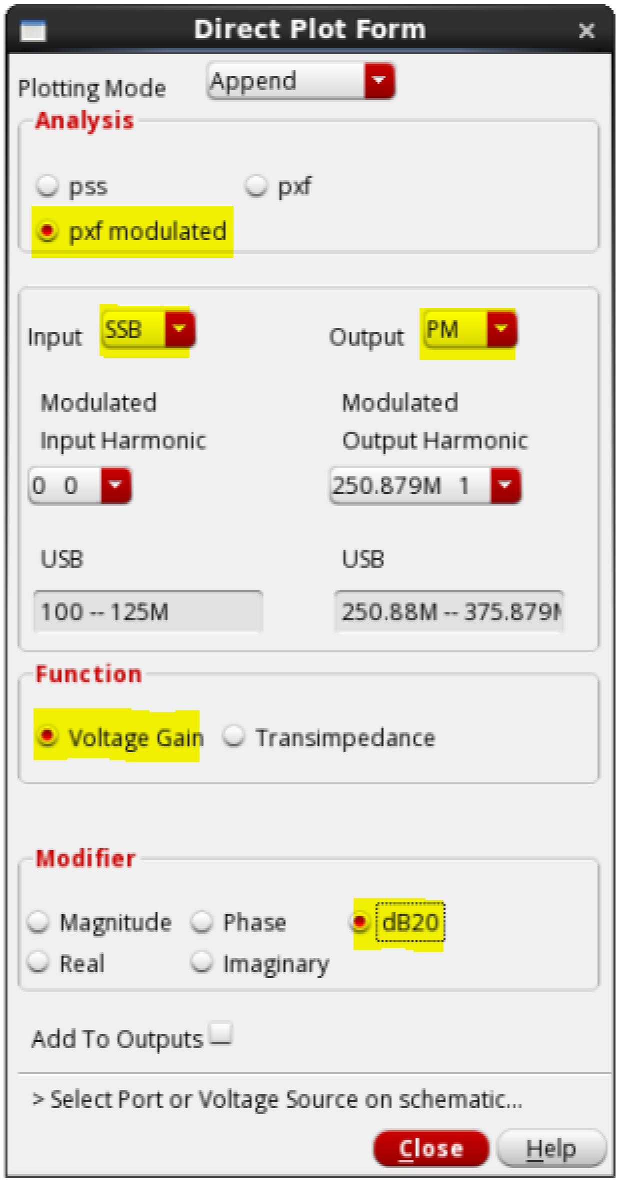

Figure 23: Direct Plot form setup

Action 12: Select the power supply voltage source on the **schematic** for the sensitivity to the power supply variation as shown in Figure 24. This will provide you the ratio of the output phase variation of the signal to the amplitude variation on the power supply pin (VDD sensitivity).

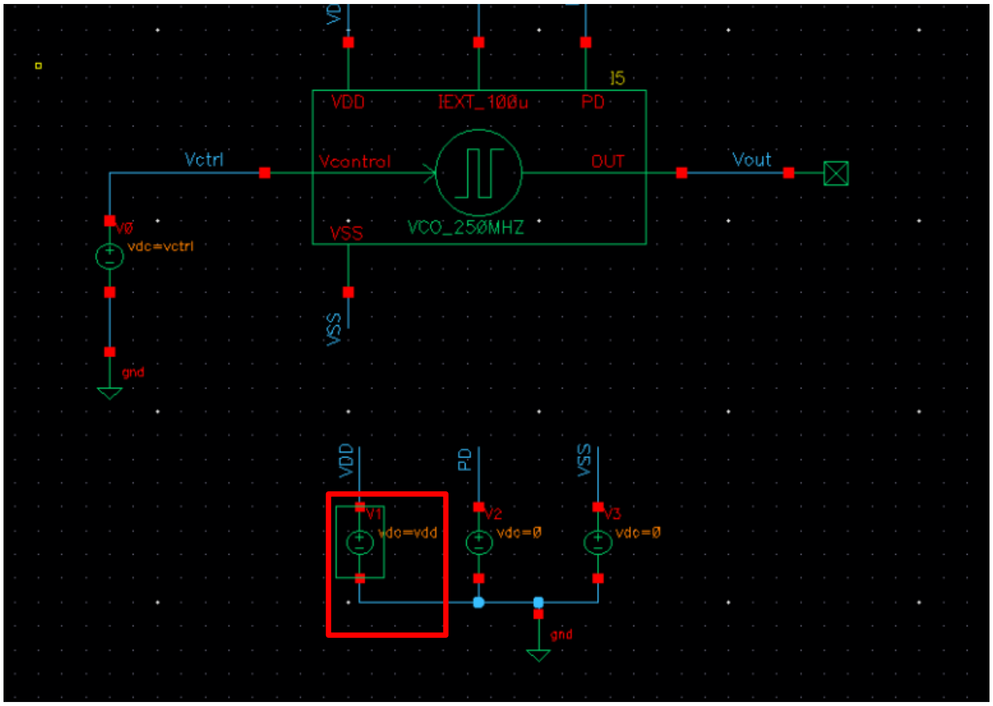

Figure 24: Select the Supply Voltage source from schematic

Action 13: The ratio of the output phase variation of the signal to the amplitude variation on the power supply pin result plot will appear as shown in Figure 25.

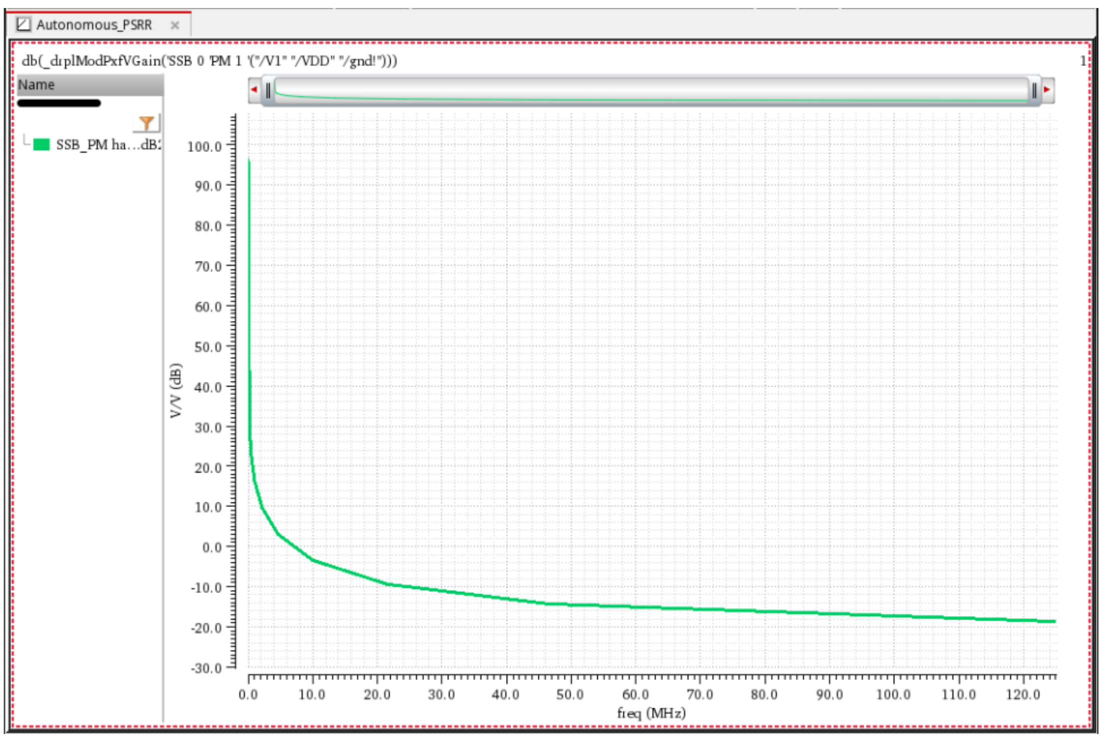

Figure 25: Modulated PXF result (VDD sensitivity)

Action 14: In this plot, right-click on X-axis and select **Log Scale**.

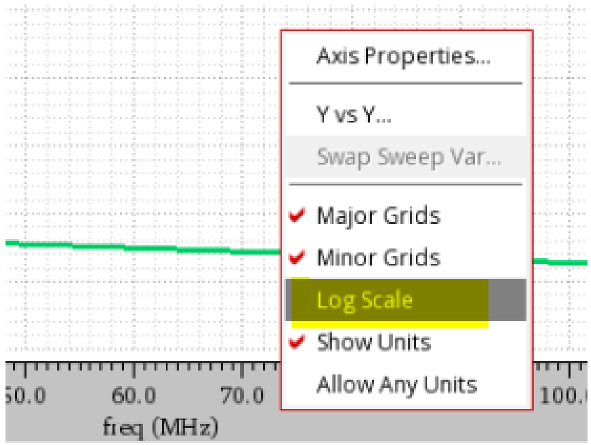

Figure 26: Change the X-Axis scale to log scale

The waveform with **Log Scale** looks like the following:

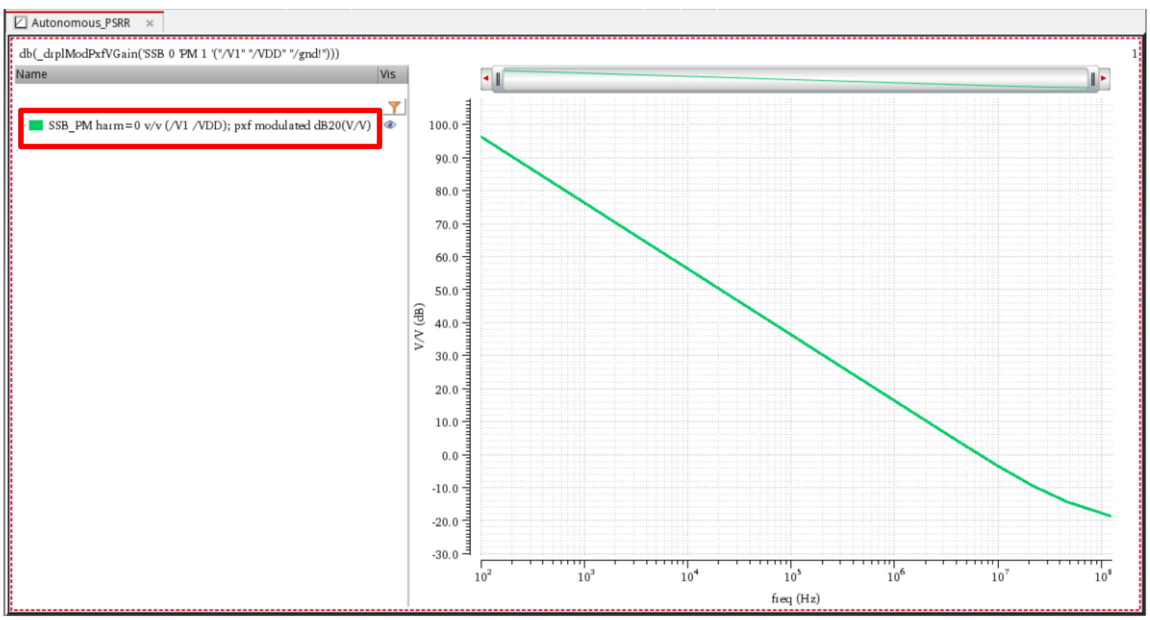

Figure 27: Modulated PXF result (VDD sensitivity) in log scale

Note: There are different definitions of PSRR.

- When it is defined as the ratio of the output amplitude variation of the signal to the amplitude variation on the power supply pin, the dB plot (VDD sensitivity) as shown in Figure 27 will be the answer.
- The other definition involves the ratio of two sensitivities: the output to the input signal voltage variation and the output to the power supply variation. The ratio of two sensitivities measure the PSRR.

Action 15: In **Direct Plot Form**, select **Magnitude** as **Modifier** and keep other settings same as described in Action 11.

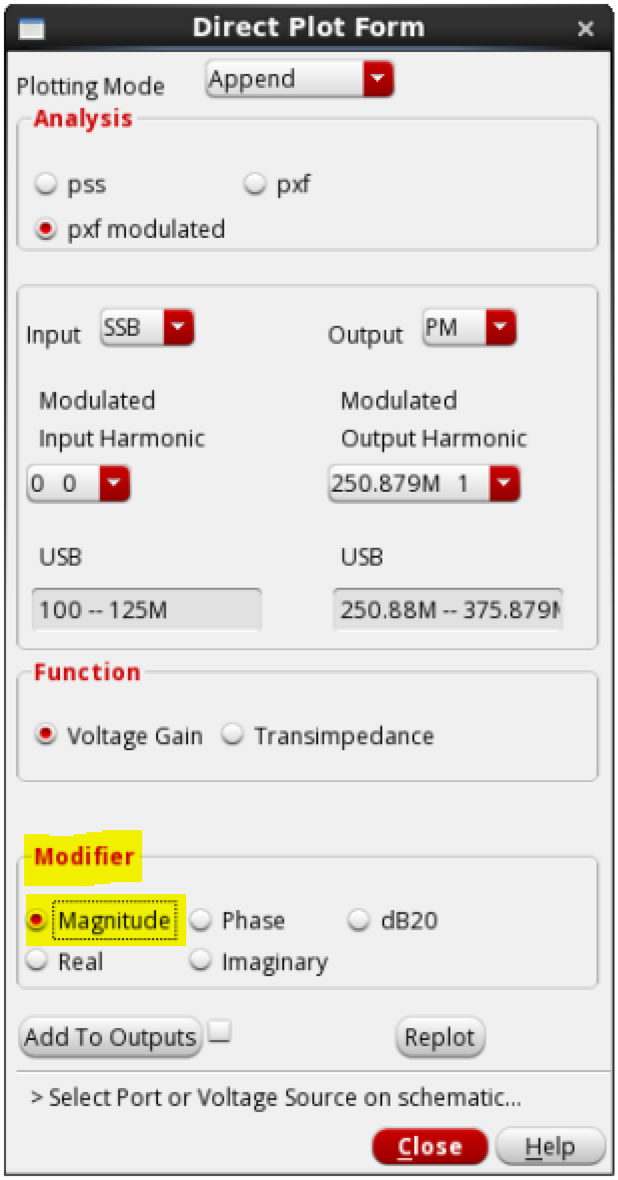

Figure 28: Direct Plot Form

Action 16: Select the following on the **schematic** as shown in Figure 29.

- Select the input voltage source on the **schematic** for the sensitivity of the output to the variations at the control voltage input.
- Select the power supply voltage source on the **schematic** for the sensitivity of the output to the power supply variations.

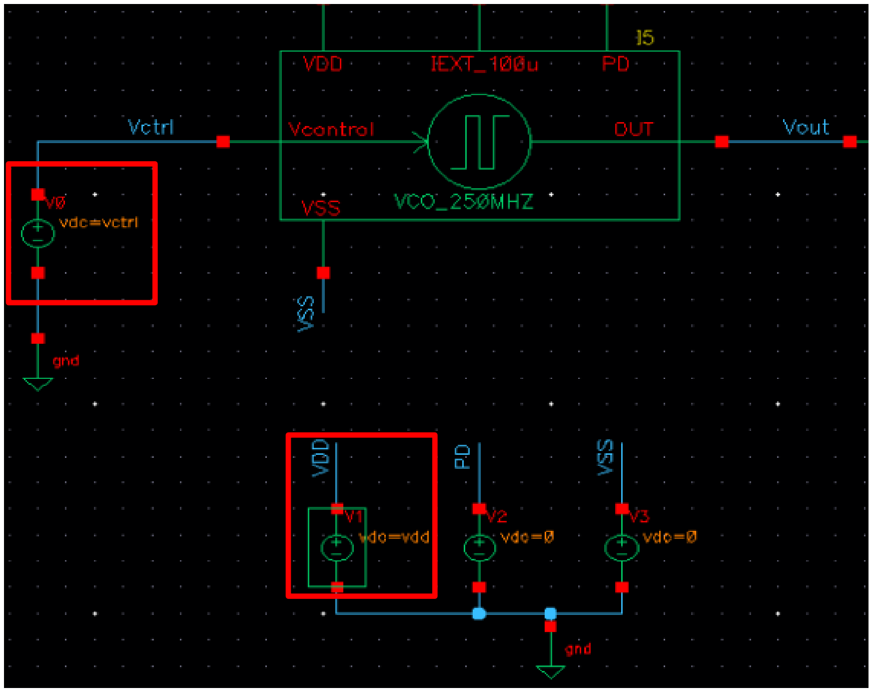

Figure 29: Select supply voltage and input voltage sources from schematic

Action 17: For the sensitivity of the output to the variations at the control voltage input and sensitivity of the output to the power supply variations, the results' plot will appear as shown in Figure 30 (in the strip mode).

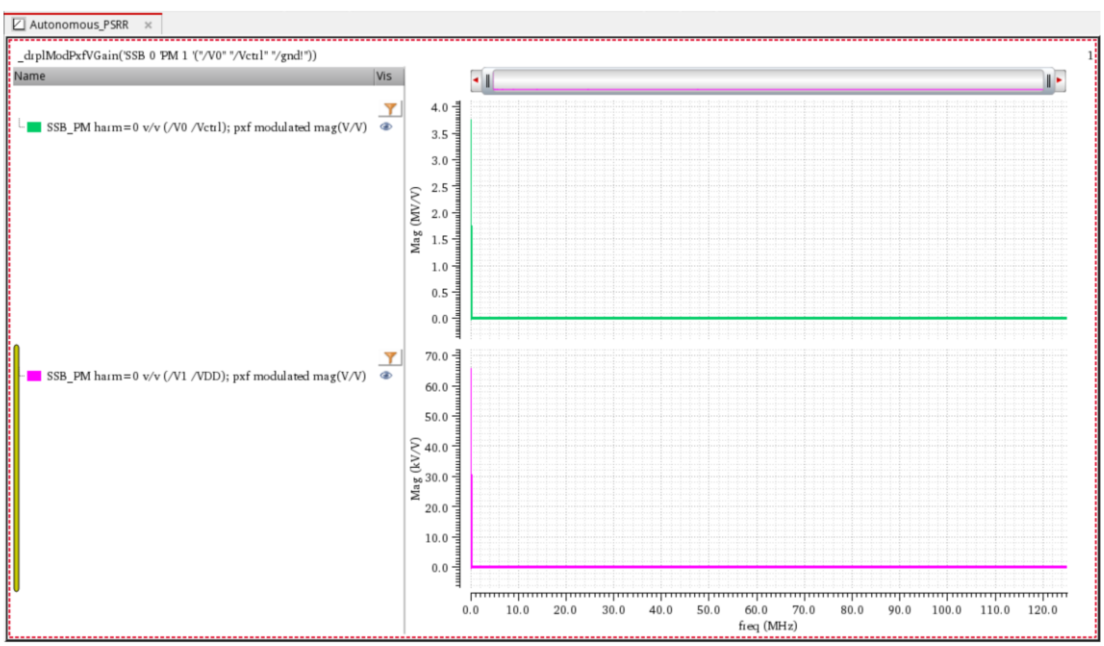

Figure 30: Modulated PXF results for sensitivity of output to power supply and input variations

Action 18: In this plot window, right-click on X-axis and select **Log Scale**.

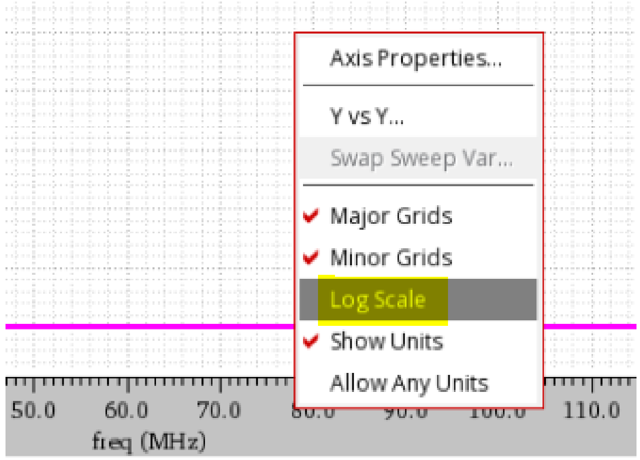

Figure 31: Change the X-Axis scale to log scale

Waveforms with **Log Scale** look like the following (in the strip mode):

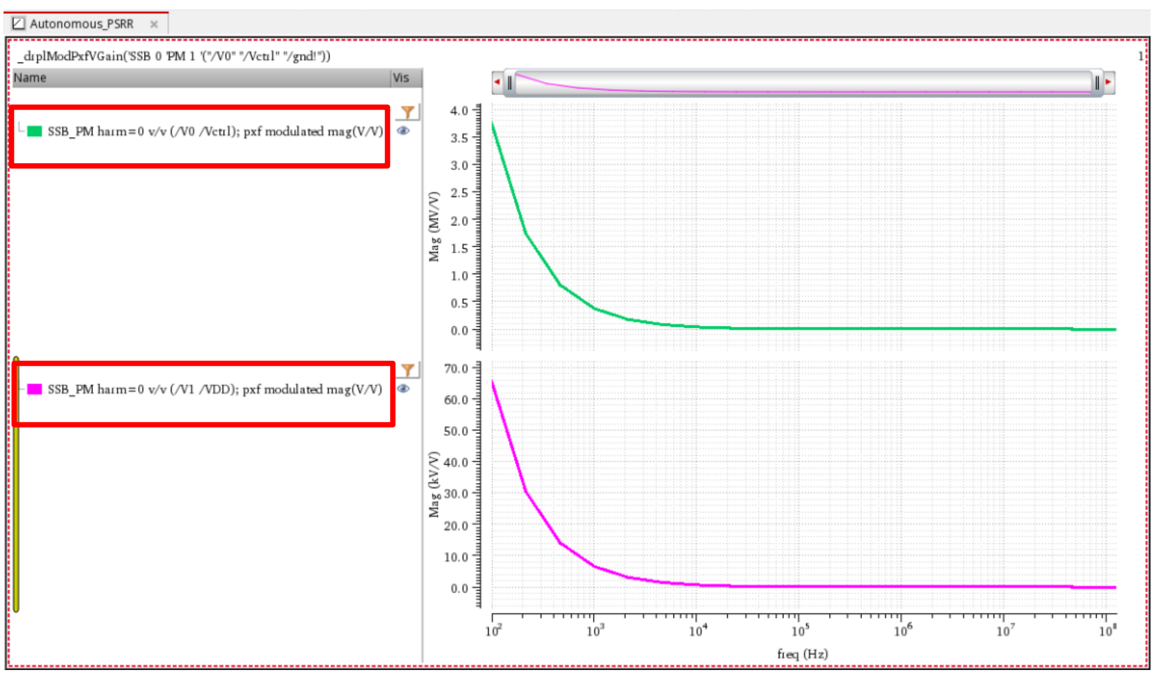

Figure 32: Modulated PXF results in log scale

Action 19: The PSRR will be measured by the ratio of the two sensitivities, the output of the input signal voltage variations and the output of the power supply variations.

To compute the ratio of two sensitivities (PSRR), you use the **ViVA XL Calculator** tool where you take the ratio of two curves from PXF and use the dB20 function.

- Right-click on the signal and select **Send To > Calculator**.

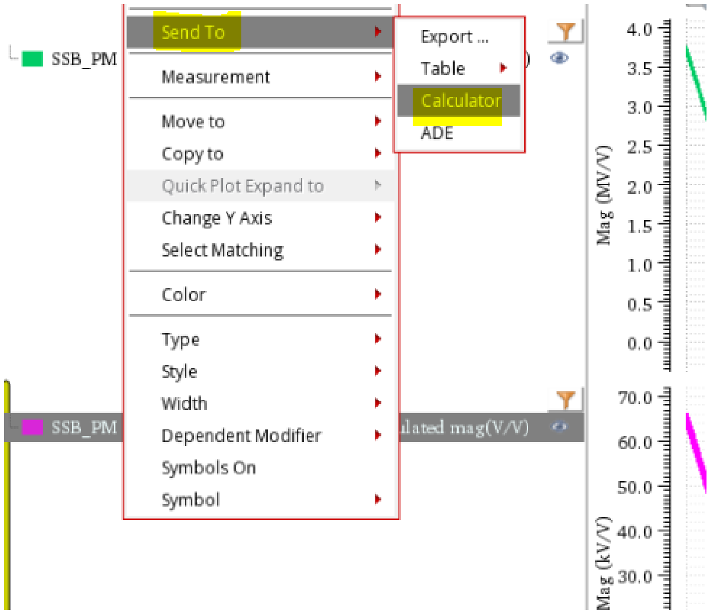

Figure 33: Sending signal to calculator

- This signal will be sent to the **ViVA XL Calculator** window and buffer the expression for the signal.
- Repeat the action for the other signal and divide by the previous expression.

The following expression for PSRR is created in the calculator:

```
dB20(_drplModPxfVGain('SSB 0 'PM 1 '("/V0" "/Vctrl"
"/gnd!"))/_drplModPxfVGain('SSB 0 'PM 1 '("/V1" "/VDD"
"/gnd!")))
```

Figure 34: Expression for PSRR in calculator

Action 20: Click on the **Evaluate the buffer** icon in the ViVA XL calculator window to evaluate this PSRR expression.

Waveforms with **Log Scale** look like the following (in the strip mode):

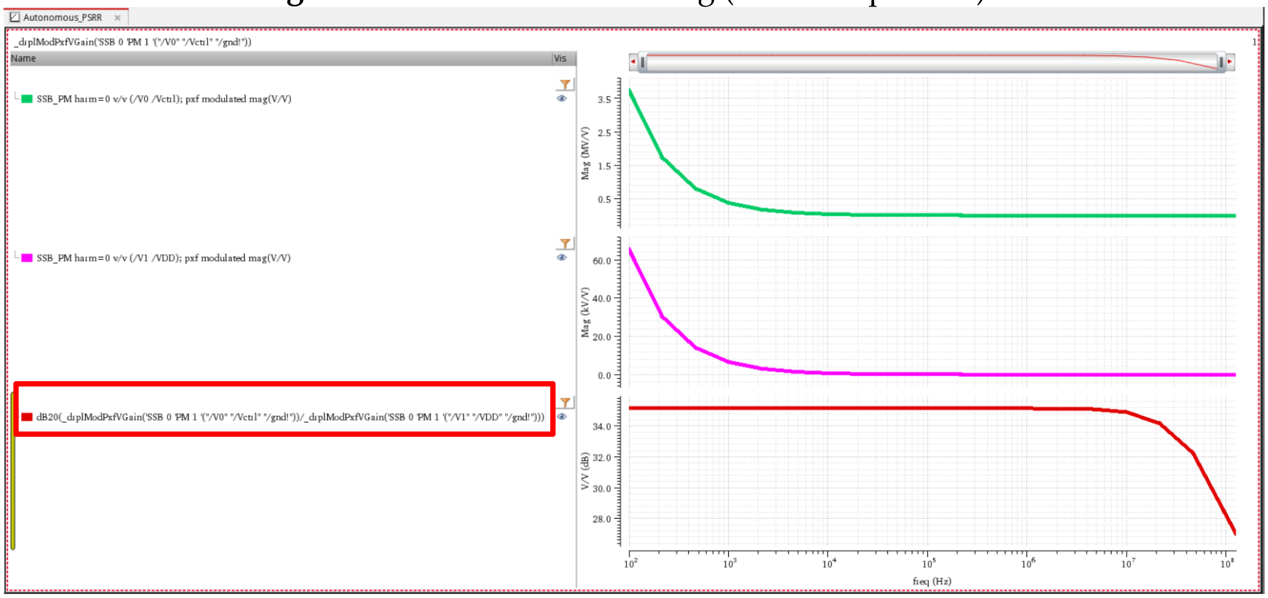

Figure 35: Plot ratio of two sensitivities (PSRR)

Note: You can send this expression in ADE also by clicking on the **Send buffer expression to ADE Outputs** icon in the calculator:

Figure 36: Send expression to ADE

In the **ADE Explorer** window, you can click the **Plot Outputs** button and plot the expression.

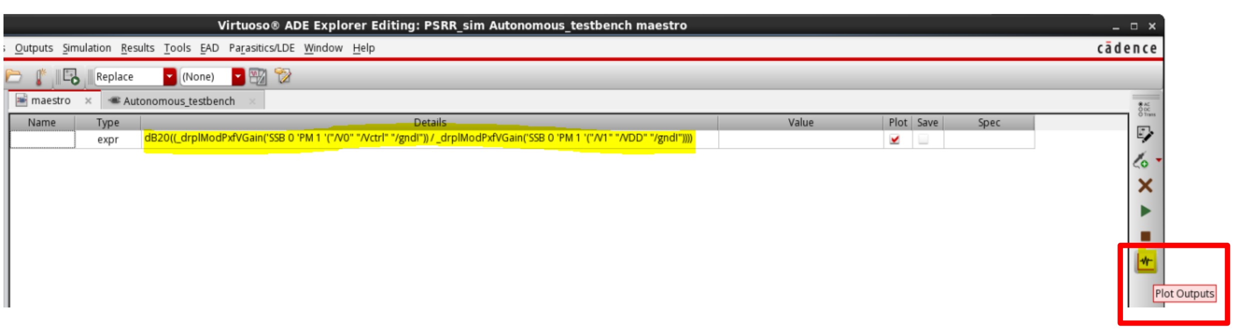

Figure 37: Plot the expression from ADE

The waveform with **Log Scale** looks like the following:

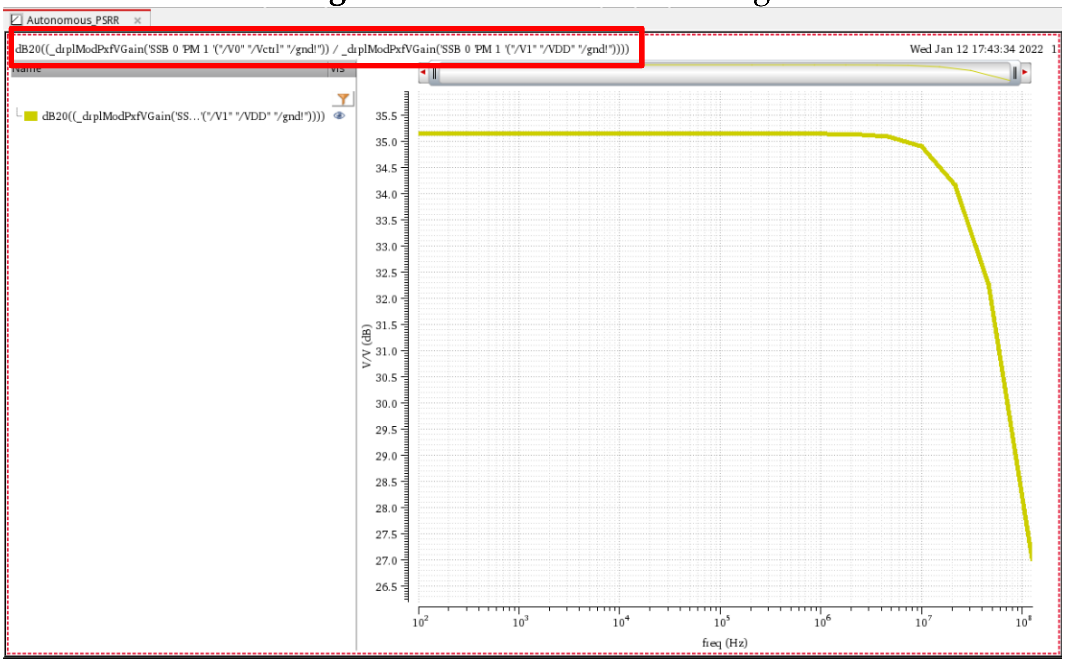

Figure 38: Plot ratio of two sensitivities (PSRR)

Action 21: Close all the windows and the **Virtuoso** session.

This completes this lab and the RAK.

References

- *Spectre Classic Simulator, Spectre Accelerated Parallel Simulator (APS), and Spectre Extensive Partitioning Simulator (XPS) User Guide*
- *Virtuoso ADE Explorer User Guide*
- *Virtuoso Visualization and Analysis XL User Guide*

Support

Cadence Learning and Support Portal provides access to support resources, including an extensive knowledge base, access to software updates for Cadence products, and the ability to interact with Cadence Customer Support. Visit https://support.cadence.com.

Feedback

Email comments, questions, and suggestions to content_feedback@cadence.com