

VCO Simulation in Cadence ICFB6

Last updated: 3/19/2015

Sudip Shekhar

Originally prepared by Parisa Behnamfar in 2014 for ICFB5
EECE 457/571F RFIC Design Course, University of British Columbia

I. Introduction

In this tutorial, various simulations for a Voltage Controlled Oscillator (VCO) are explained and illustrated. We will run transient, pss and pnoise analysis to check the output waveforms as well as characterize the tuning range and phase noise behavior of the VCO.

It is assumed that the reader is familiar with 1) Cadence schematic and components libraries, 2) Analog Design Environment (ADE) features, such as setting up the model libraries, working with variables, defining the outputs to be saved/plotted, using the calculator, and being able to check the results in the results → direct plot menu. Previous tutorials of this course have already covered the above-mentioned pre-requisite materials. You may refer to those tutorials for further information.

II. Setting up the VCO schematic in Cadence

VCO shown in Figure 1 is drawn in Cadence schematic environment using 0.13um CMOS technology and analogLib library components. This is a complimentary cross-coupled VCO (CCC-VCO) with an ideal current source [Your design project may be different and would NOT use an ideal current source]. The voltage-controlled capacitance is provided by two MOS transistors (D-S shorted) connected to the differential outputs voutp and voutn. Note that the bulk of the NMOS transistor in the circuit should be connected to the lowest voltage of the circuit (ground in our case).

Also, note that this circuit also has a fixed capacitive load. For your projects, you have to consider appropriate capacitive load, as the VCO is to be connected to the Mixer's LO port – with or without a buffer. (How does adding the capacitive load affect this circuit?)

As you have learned in the class, there is no ac input to the VCO circuit. Then, to provide a trigger, you can use set an initial voltage condition for one of the VCO nodes (Fig. 2(a)) using Simulation→Convergence Aids→Set initial condition → specifying a voltage and selecting the node on the schematic. Alternatively, a piecewise linear current source, I_{pwl} can be connected to one of the VCO nodes (NOT shown in Figure 1). If using the latter, the I_{pwl} setup is shown in Figure 2(b).

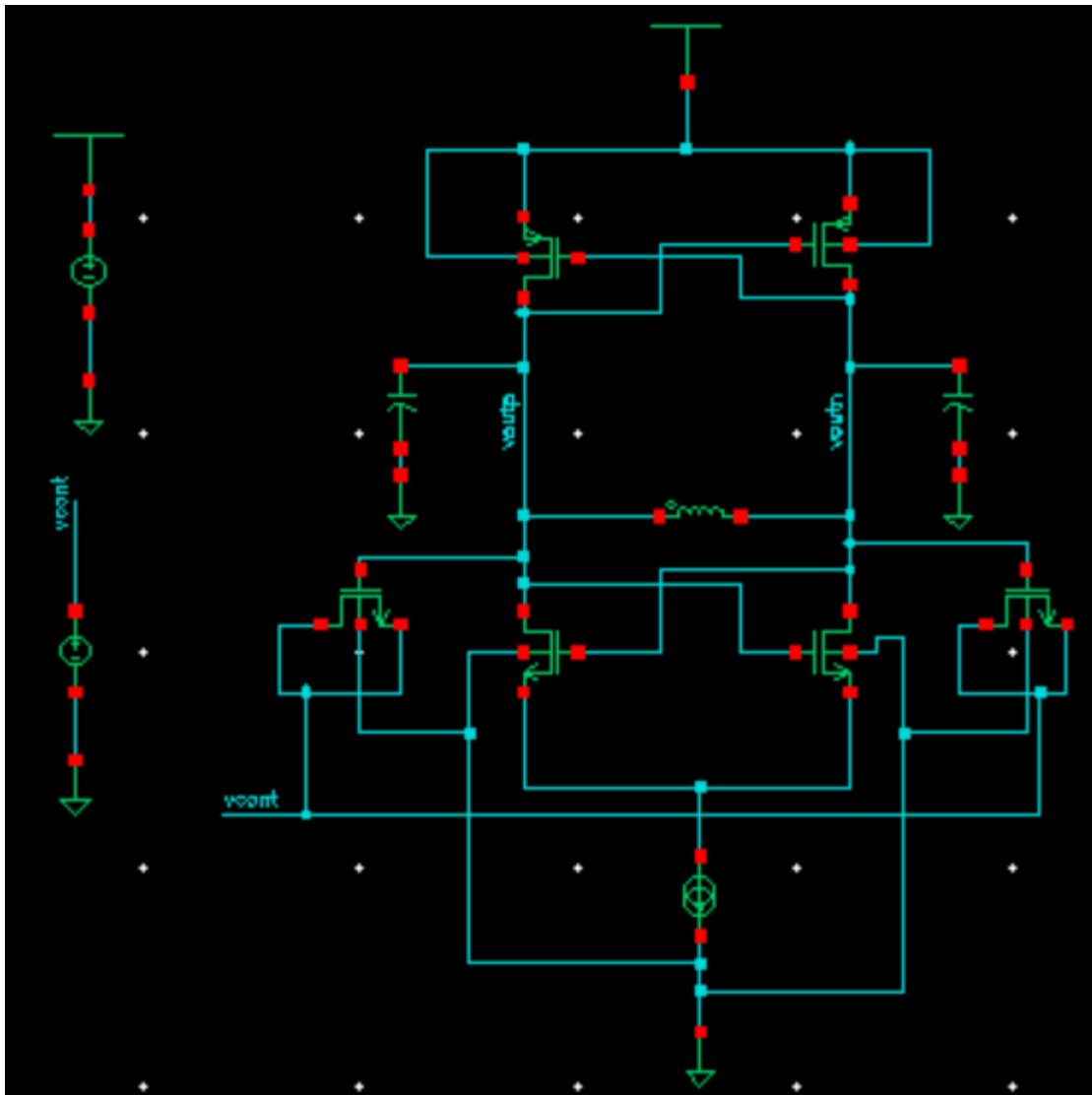


Fig. 1: VCO schematic

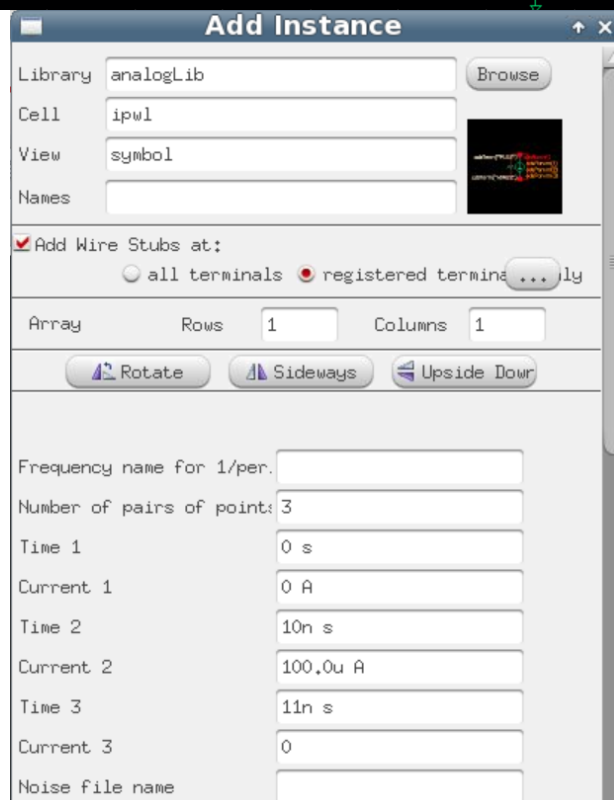
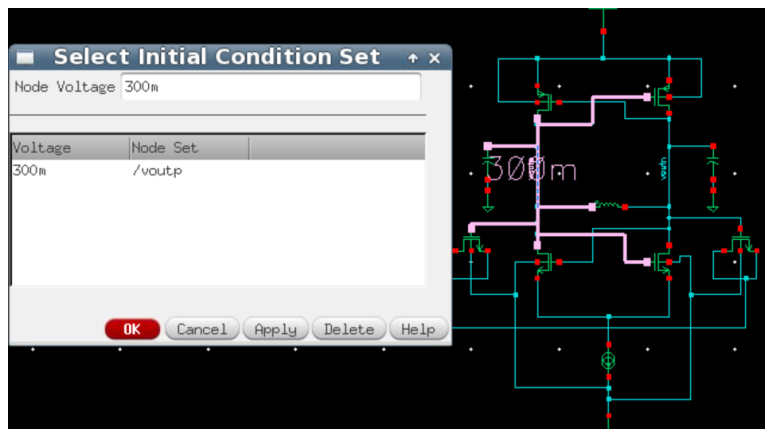
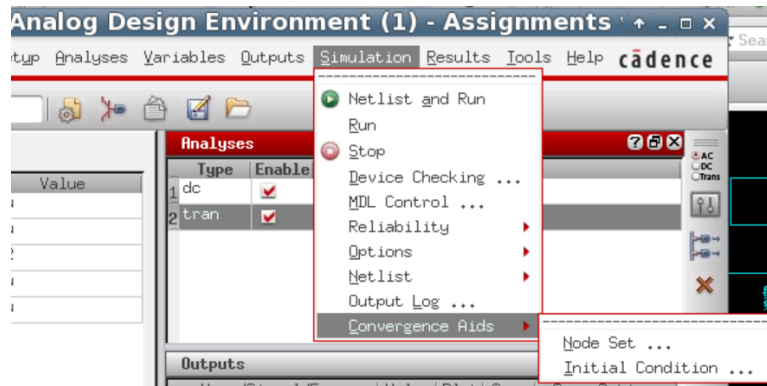


Fig. 2: Initial Node voltage, or I_{pwl} setup for VCO trigger

Questions:

VCO of Figure 1 is designed to provide 5GHz at the output. Nominal V_{dd} is 1.2V and the control voltage (v_{cont}) can have any value between 0 to 1.2 volts.

- 1) a) For $v_{cont}=0.5V$, provide the transient response showing both v_{outp} and v_{outn} output waveforms as well as the differential output of $(v_{outp})-(v_{outn})$. What is the approximate settling time and frequency of the output?
b) Measure the average current drawn from the power supply and calculate the power consumption.
- 2) Calculate the tuning range and K_{VCO} by plotting the frequency versus control voltage v_{cont} .
- 3) Assuming that a noise on the V_{dd} results in V_{dd} variation between 1V to 1.4V, plot the frequency pushing versus V_{dd} . How does this noise affect the target frequency?
- 4) Plot the phase noise for 10k to 20MHz relative frequency range and define the spot values at 100kHz and 1MHz offset from the center frequency.
- 5) Report the noise summary, including the top 5 noise contributors.
- 6) In transient analysis, plot the differential output waveform for swept control voltage of 0-1.2V, with linear steps of 0.1V.

III. VCO Characterization and Measurements

Typically, the frequency domain analysis of VCO requires PSS analysis. Other analysis such as Pnoise may be added to the PSS analysis to measure the circuit's noise performance. Before running the PSS analysis, it is recommended to run transient analysis to have an idea about the "beat frequency" and "settling time" (or t_{stab} = stabilization time) of the VCO as you will need these values for the PSS analysis setup. If the values you enter in the PSS analysis are too far from the real response of the circuit, PSS may not work properly.

- 1) a) To check the transient response, transient analysis is performed for 100ns (Fig. 3). We expect a few ns settling time for the VCO, which is the time required for the outputs to reach their final value. Figure 3 shows the transient response of the VCO at nodes v_{outp} and v_{outn} as well as the differential output. It can be seen that the output is almost settled at $t_{stab}=3ns$.

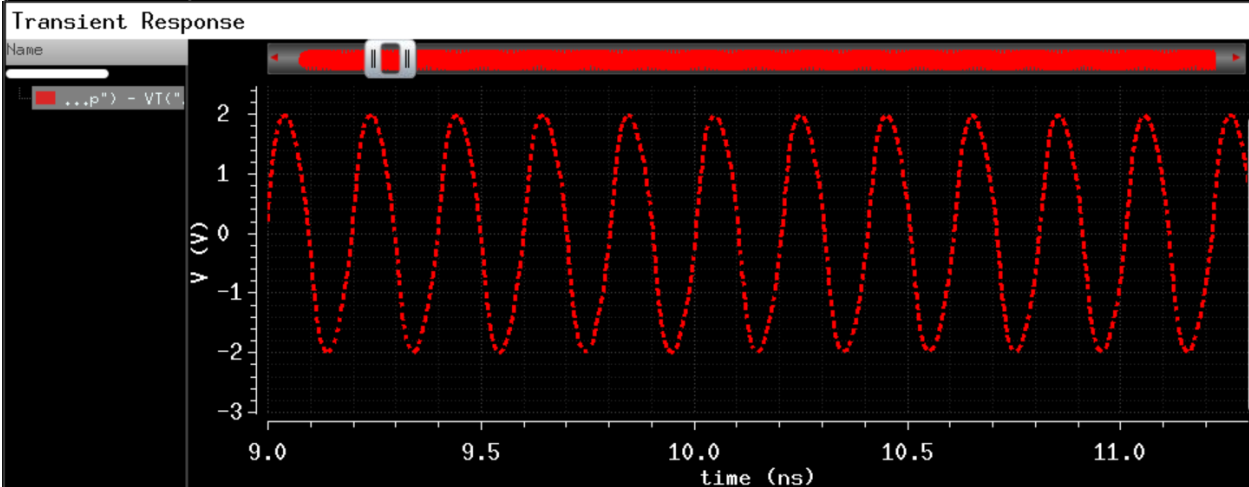
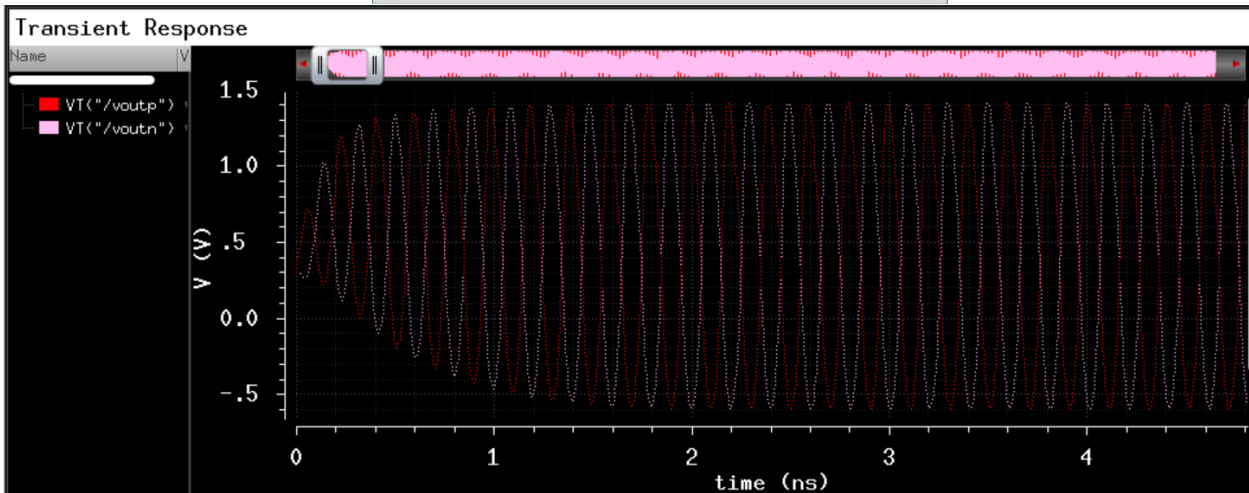
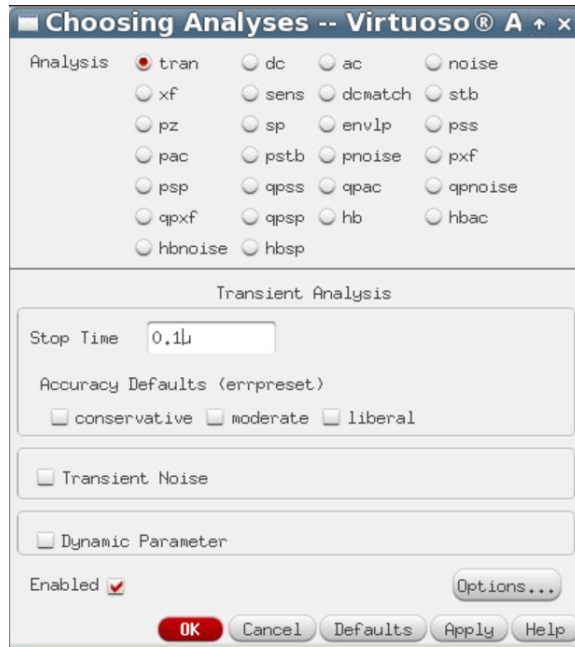


Fig. 3: VCO transient response (single ended and differential)

The frequency of the output can be measured using the calculator, shown in Figure 4. In the plot window, first zoom into a settled portion of the differential output signal e.g., between 30ns to 32ns. Then, in the graph window, go to tools → calculator and from the functions shown, select “freq”. In the window that appears inside the calculator window, first select “wave”, and then click inside the “signal” menu. After that, click on the zoomed output waveform to introduce that portion (clipped signal) as the source of the frequency measurement. Click ok. Again, from the functions in the calculator, click “Evaluate the buffer”. The frequency shows in the calculator, measured as 4.97GHz.

b) To measure the average current draw (Figure 5), use the calculator, select the current waveform and then select “average” function. After clicking on “Evaluate the buffer”, the average current value shows up in the calculator which is equal to 1.86mA for this circuit. This value is used to calculate the power consumption.

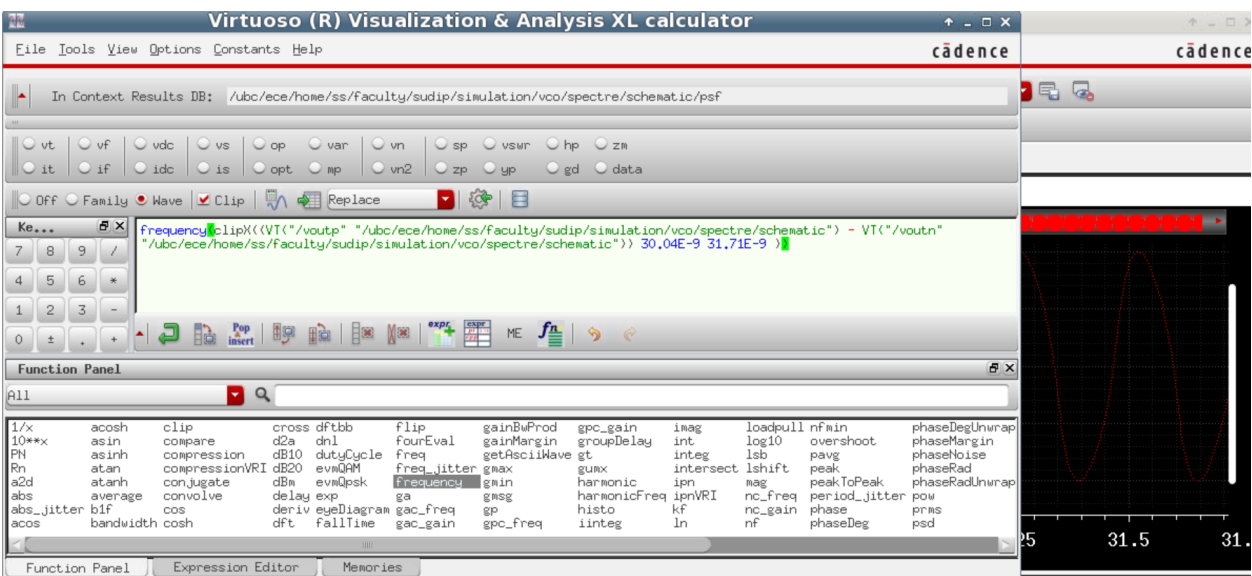


Fig. 4: Using calculator to measure the frequency

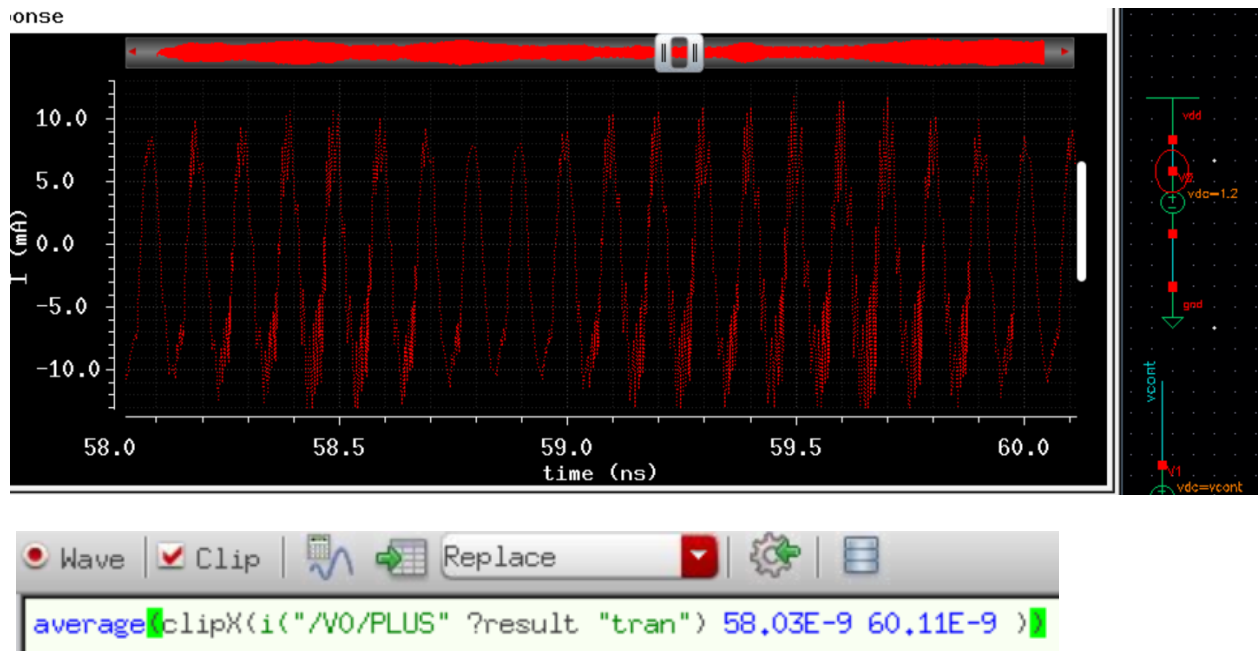


Fig. 5: Using calculator to measure the average current drawn from Vdd

- 2) We can now proceed to the PSS analysis. From the ADE window, go to Analysis → Choose → pss. Setup the pss analysis as in figure 6. Enter the measured value of beat frequency and tstab from the transient simulations performed in the previous step. Note that you should select “Oscillator” and choose voutp and voutn as the oscillator and reference nodes. Since we want to measure the tuning range and K_{VCO} , sweep the vcont in the desired range with linear sweep type. Select “Enable” and press ok on top of the “choosing analysis” window. Now, run the simulation and after simulation is performed, in the ADE window, go to results → direct plot → main form. As in figure 7, choose pss and under “Function”, choose “Harmonic Frequency”. Select the first harmonic frequency and plot it.

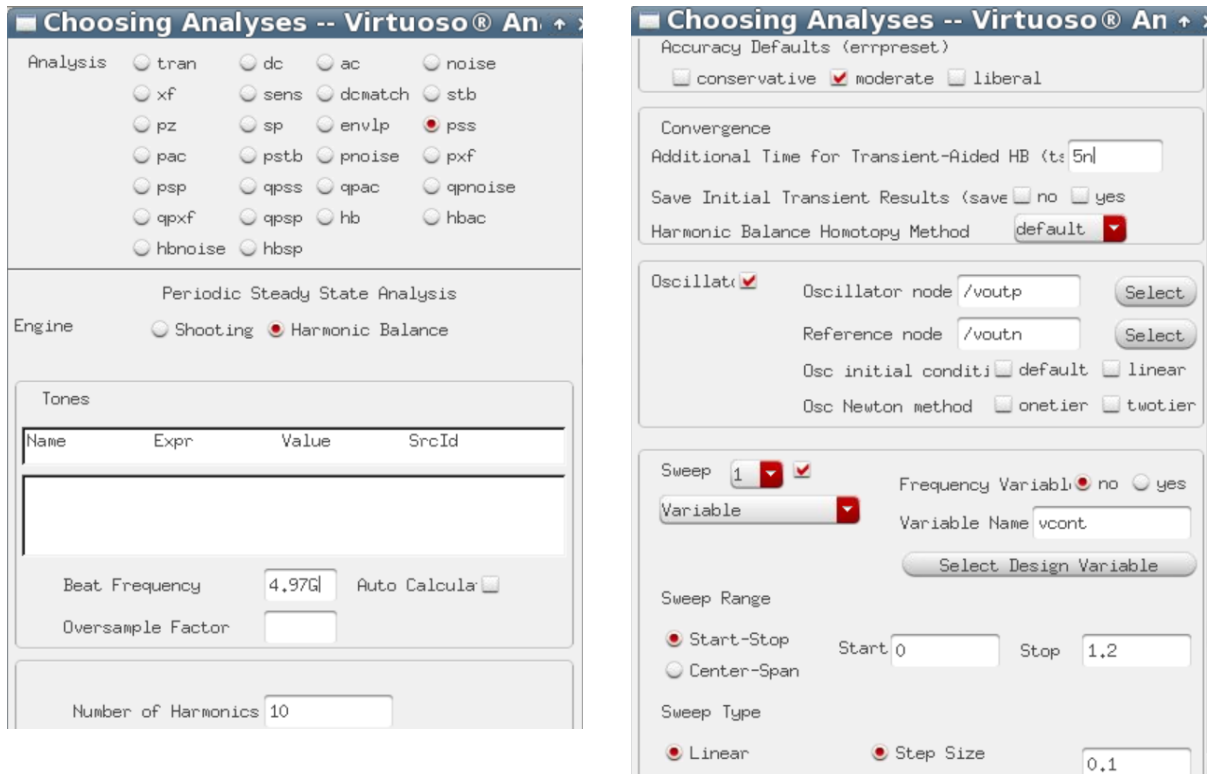


Fig. 6: Setting up the PSS analysis for measuring K_{vco}

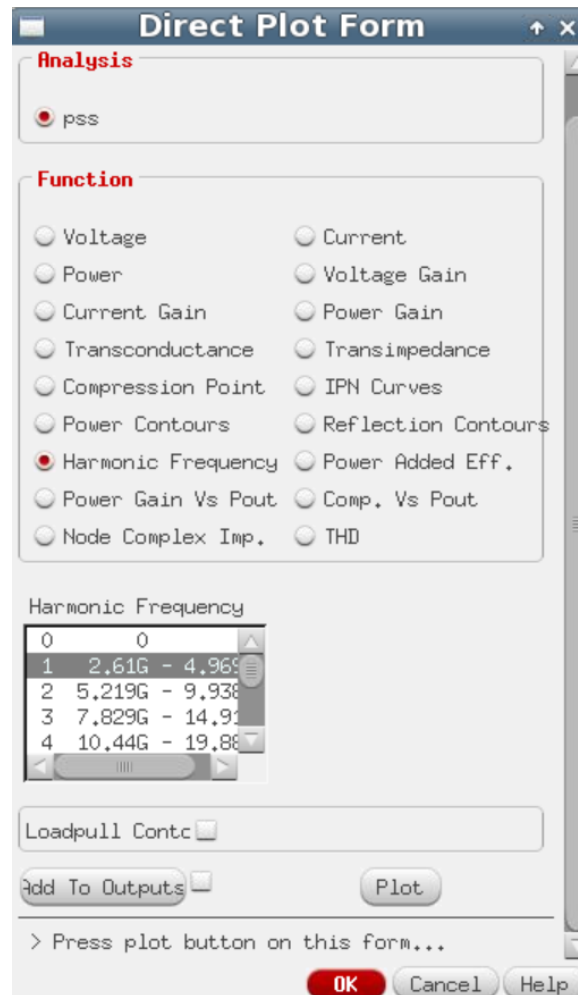


Fig. 7: PSS direct plot form

Figure 8 shows the frequency range versus v_{cont} . You can derive the K_{VCO} by measuring the slope of this curve. You can also calculate the tuning range of the VCO by finding the f_{max} and f_{min} (where else could you see the measured f_{min} and f_{max} ? Look at Figure 7 for the answer!). This VCO has a large tuning range!

- 3) To plot the frequency pushing, setup the PSS analysis for sweeping the V_{dd} value between 1.1 and 1.3 volts. At this step of the analysis, v_{cont} is fixed. Figure 9 shows the frequency pushing versus V_{dd} variations. As it is pointed on this figure, the frequency value at $V_{dd}=1.2V$ is 4.97GHz, the same as the result derived from the transient measurement.

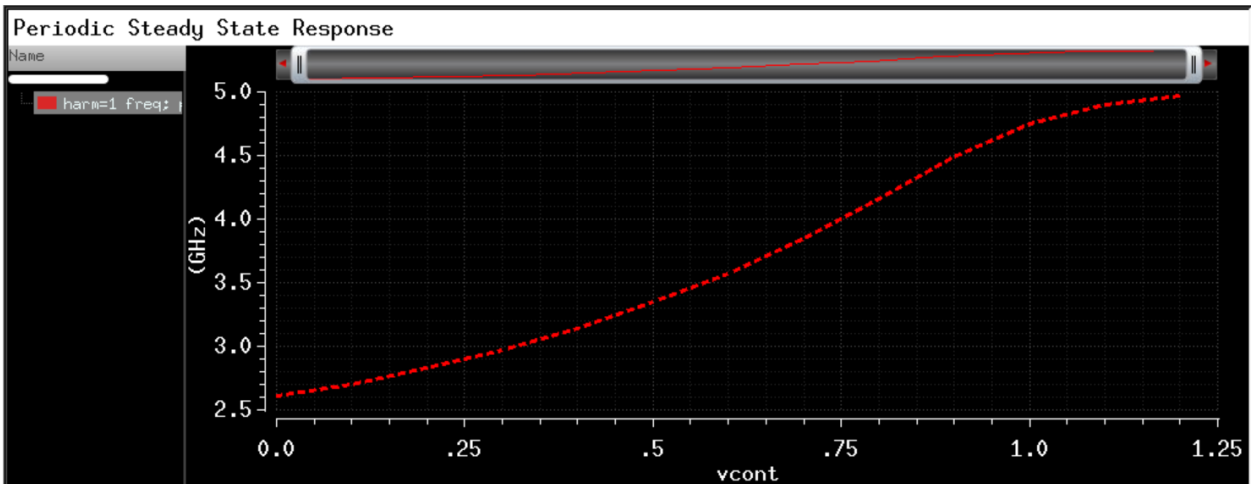


Fig. 8: Frequency range versus vcont

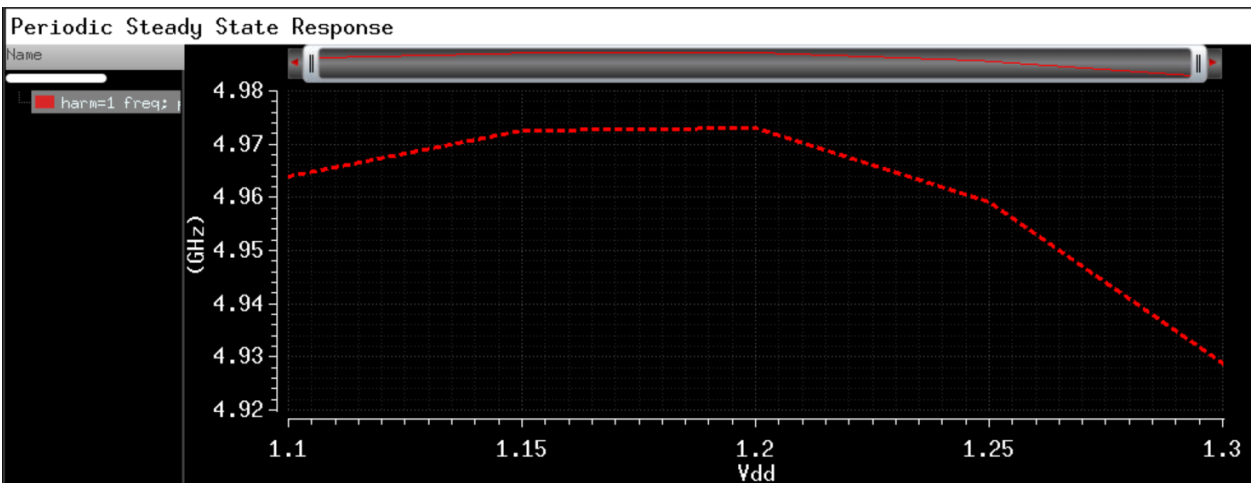


Fig. 9: Frequency pushing due to V_{dd} variations

- 4) To plot the phase noise, pss should be setup first and then pnoise. Setup noise analysis as shown in figure 10. Note that in this setup, pnoise is measured for a fixed vcont and V_{dd} value. (No sweeping is set in the pss).

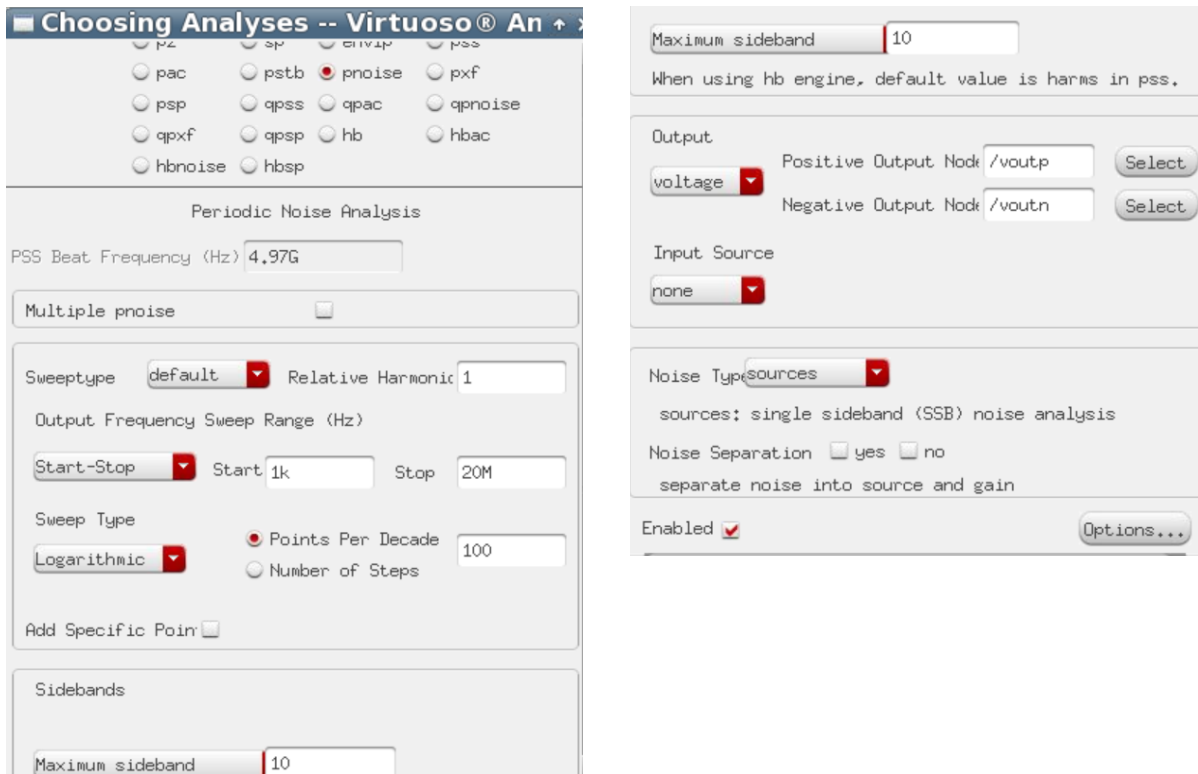


Fig. 10: Pnoise setup for plotting the phase noise

As shown in figure 10, in the pnoise setup, enter “1” for relative harmonic, as you want to measure the phase noise around the first harmonic of the output signal (which is around 4.97GHz). For the “output frequency sweep range”, enter the “relative frequency” start and stop points. These are the frequencies you are interested in measuring the phase noise at, relative to the center frequency. In the “output” section, make sure to choose “voltage”, and define the voutp and voutn as positive and negative output nodes. For the “input”, select “none”, as you have no input to the circuit. For noise type, select “sources”. Select enable and click ok. Run the simulation, and after the simulation is performed, go to results→direct plot→main form. Select pnoise as in figure 11, and under “function”, select “phase noise”. Plot the phase noise and measure the spot phase noise for 10kHz and 1MHz offsets, as shown in figure 12.



Fig. 11: Pnoise direct plot form

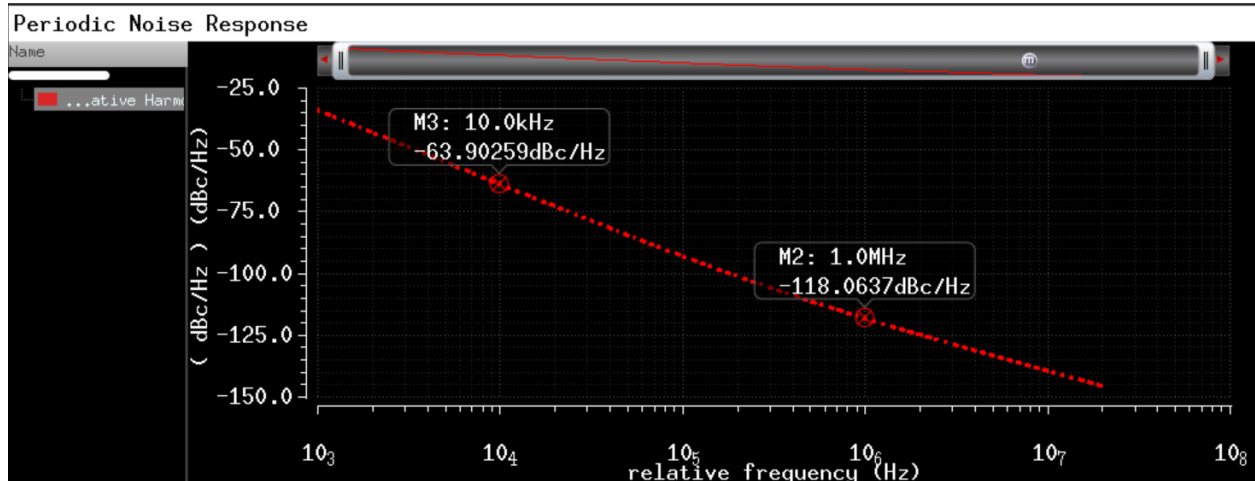


Fig. 12: Phase noise plot and spots at 10kHz and 1MHz offsets

- 5) To print the noise summary for 1MHz frequency spot, in the ADE go to results→print→noise summary, as shown in figure 13. In the pop-up window (figure 14), select “spot noise” for type, and enter 1M for frequency spot. Select your desired noise unit (V^2 or V) and click on “Include all types” under “FILTER”. For “truncate”, select “by number” and enter 10, to see the noise summary for top 5 contributors. For “sort by”, select “noise contributors” and click apply. Results will

appear in another window, as shown in figure 15. fn denotes flicker noise, and id denotes thermal noise.

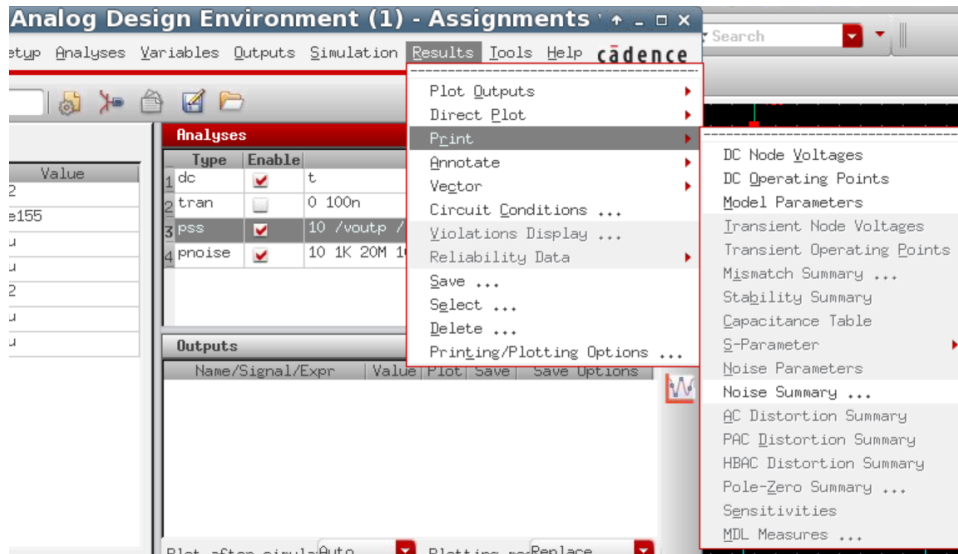


Fig. 13: Procedure to print the noise summary

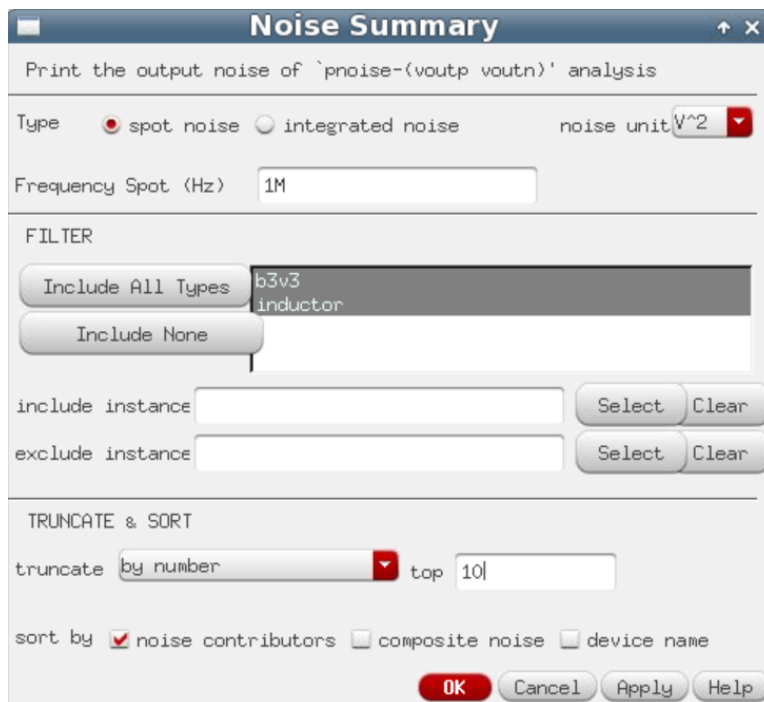


Fig. 14: Setting up the noise summary window

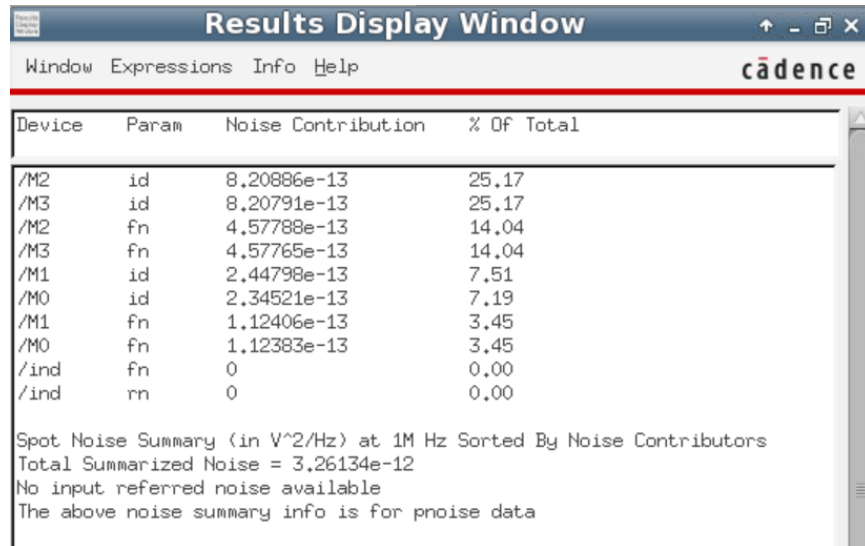


Fig. 15: Noise summary of top 10 contributors, in V²/Hz

- 6) To plot the transient response for differential output waveform for swept control voltage of 0-1.2V, with linear steps of 0.1V, enable the transient analysis and set it up for 20ns. Setup the parametric analysis in ADE→Tools→Parametric Analysis with vcont as the variable and run the simulation. Figure 16 shows the differential output signal in time domain (zoomed view).

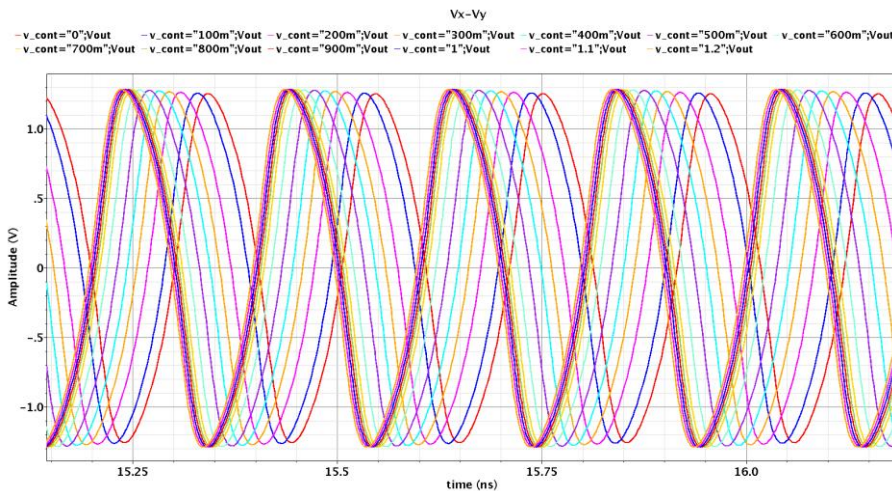


Fig. 16: Zoomed view of the differential output in time domain for vcont of 0-1.2V