# Design and Simulation of RF CMOS Oscillators in Advanced Design System (ADS)

**By**

**Amir Ebrahimi**

**School of Electrical and Electronic Engineering**

**The University of Adelaide**

June 2014

# Contents

# 1- Introduction

This tutorial is a quick guide for new ADS users to design CMOS RF oscillators and do the post processing on their simulation data in ADS.

Oscillators are fundamental parts of any RF communication systems. They are used as clock generators in microprocessors, local oscillators (LOs) for up/down conversion of the information bearing signal and carrier signal generators for modulations. Robust and high performance design of oscillators in CMOS technology poses interesting challenges.

Here, the design and simulation of CMOS ring oscillator in ADS will be presented and the simulation procedures for obtaining the different output parameters like: transient output waveform, output spectrum and phase noise are explained.

## 1-1- Objectives:

- Design a 3-stage ring oscillator based on common-source gain stages
- Performing the DC simulation for setting the gain stages DC bias point
- Performing the AC simulation for finding the individual and loop gain of the oscillator
- Performing transient analysis for displaying the oscillator output waveform in time domain
- Performing the "Harmonic Balance" simulation for finding the output spectrum in frequency domain
- Using "Harmonic Balance" simulation for "Phase Noise" analysis of the oscillator

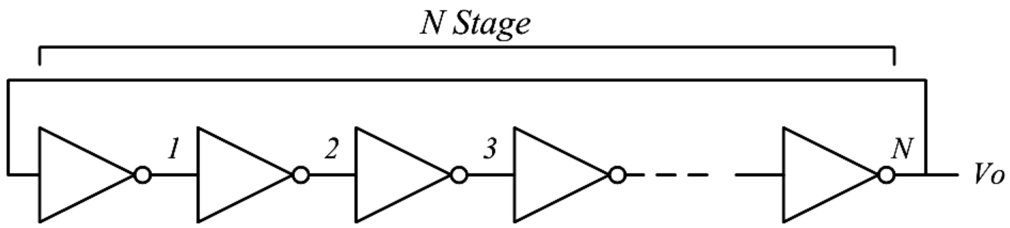



Fig. 1. General configuration of a ring oscillator.

## 2-Ring Oscillators

A ring oscillator consists of a number of gain stages in a loop as shown in Fig.1.

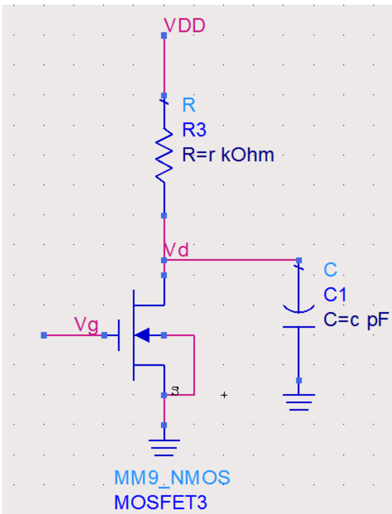



Fig. 2. A common-source amplifier (single gain stage for the ring oscillator).

By considering each stage as a simple common-source amplifier like Fig.2, each stage shows a voltage gain of:

$$G = -\frac{A_0}{(1+\frac{s}{\omega_0})}, \quad A_0 = g_m R, \quad \omega_0 = \frac{1}{RC} \tag{1}$$

where, $g_m$ is the small-signal trans-conductance of the CMOS transistor. Assuming the oscillator with three stages, the oscillator loop gain will be obtained as:

$$H(s) = -\frac{A_0^3}{(1+\frac{s}{\omega_0})^3} \tag{2}$$

According to the analysis in [1], the circuit oscillates if the frequency dependent phase shift equals 180°. This translates into a 60° phase shift in each stage so, the frequency at which oscillation happens is:

$$\tan^{-1}\frac{\omega_{osc}}{\omega_0} = 60^\circ \xrightarrow{yields} \omega_{osc} = \sqrt{3}\omega_0 \tag{3}$$

On the other hand, for oscillation, the minimum voltage gain per stage must be such that the magnitude of the loop gain at $\omega_{osc}$ is unity.

$$\frac{A_0^3}{\left[\sqrt{1+(\frac{\omega_{osc}}{\omega_0})^2}\right]^3} = 1 \xrightarrow{yields} A_0 = 2. \tag{4}$$

In circuit designs the gain should be chosen slightly larger than 2.

## 2-1- Oscillator Design

In order to design the oscillator loop, firstly, a single gain stage of Fig. 2 should be designed. It can be designed based on the following steps:

**S. 1-** It is known that in oscillation mode, the source and drain terminals of all three transistors will take the same DC voltage [2] so, for each gain stage we have:

$$V_{G(DC)} = V_{D(DC)} \tag{4}$$

**S. 2-** By considering the above condition, for a single gain stage of Fig. 2 in DC condition we have:

$$V_D = V_G = V_{DD} - KR(V_G - V_t)^2, \qquad K = \frac{1}{2}\mu_n C_{ox} {}^{W}\!/_{L} \tag{5}$$

**S. 3-** Based on the loop gain analysis (4), we assume $A_0 = 2.5$.

$$A_0 = g_m R = 2KR(V_G - V_t) = 2.5 \xrightarrow{yields} KR = \frac{1.25}{V_G - V_t} \tag{6}$$

**S. 4-** By substituting (6) into (5), the following relation is obtained giving us the appropriate value of $V_{G(DC)}$ for designing a single gain stage.

$$2.25V_G = V_{DD} - 1.25V_t \tag{7}$$

**S. 5-** By calculating $V_G$, the value of drain resistor ($R$) can be calculated from (6).

**S. 6-** Finally, the capacitor ($C$) value can be calculated by considering the $\omega_{osc}$ and (3).

Now, suppose that a ring oscillator of three common-source gain stages has to be designed for the oscillation frequency of 1 GHz with $V_{DD} = 5$ V. The considered transistor size is $({}^{W}\!/_{L} = {}^{50}\!/_{0.5})$ in 0.5 μm standard CMOS technology with the parameters given in [1], which are listed in Table 1.

**Table 1**: Level 1 SPICE Models for NMOS and PMOS Devices in 0.5 µm CMOC Technology.

| | | | |
|---|---|---|---|
| **NMOS Model** | | | |
| Level=1 | VTO=0.7 | GAMMA=0.45 | PHI=0.9 |
| NSUB=9e+14 | LD=0.08e-6 | UO=350 | LAMBDA=0.1 |
| TOX=9e-9 | PB=0.9 | CJ=0.56e-3 | CJSW=0.35e-11 |
| MJ=0.45 | MJSW=0.2 | CGDO=0.4e-9 | JS=1.0e-8 |
| **PMOS Model** | | | |
| Level=1 | VTO=-0.8 | GAMMA=0.4 | PHI=0.8 |
| NSUB=5e+14 | LD=0.09e-6 | UO=100 | LAMBDA=0.2 |
| TOX=9e-9 | PB=0.9 | CJ=0.94e-3 | CJSW=0.32e-11 |
| MJ=0.5 | MJSW=0.3 | CGDO=0.3e-9 | JS=0.5e-8 |

Based on the (S. 1- S. 6) and the data given in Table 1, the calculated values of $V_{G(DC)}$, $R$ and $C$ are 1.83 V, 0.2 kΩ and 1.29 pF respectively.

## 2-1-1- DC Simulation of Gain stages

For starting, create a new "Workspace" in ADS as shown below and name it "RingOsc".

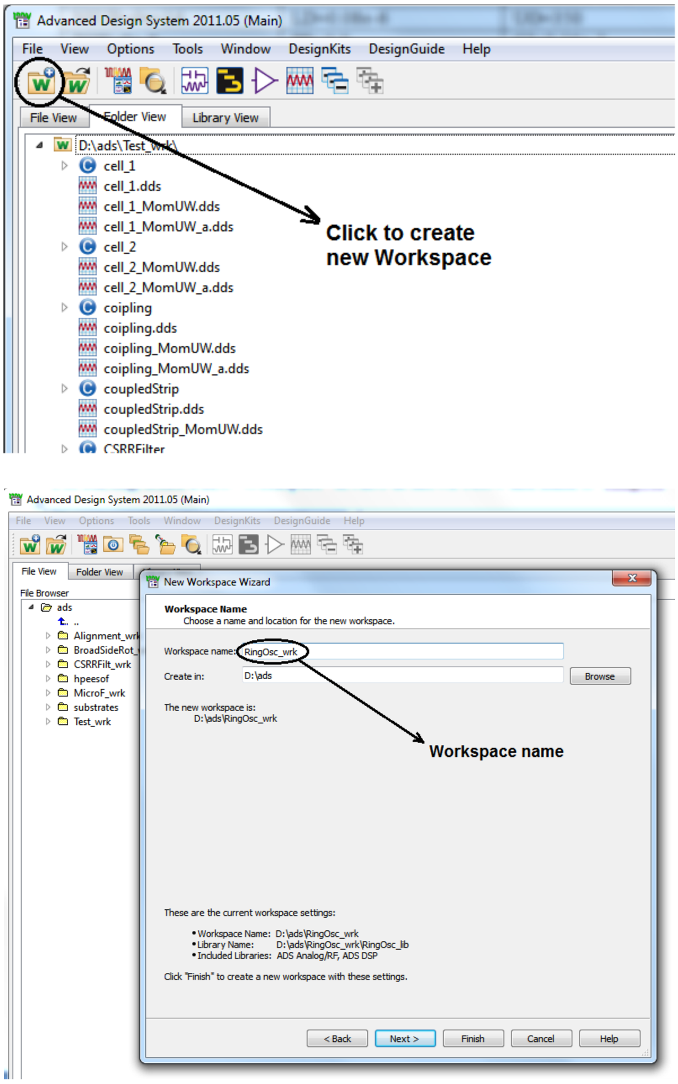



Fig. 3. Creating a new workspace in ADS.

After creating the new workspace, open a new schematic window as shown in Fig. 4 and name it “DCSim”.

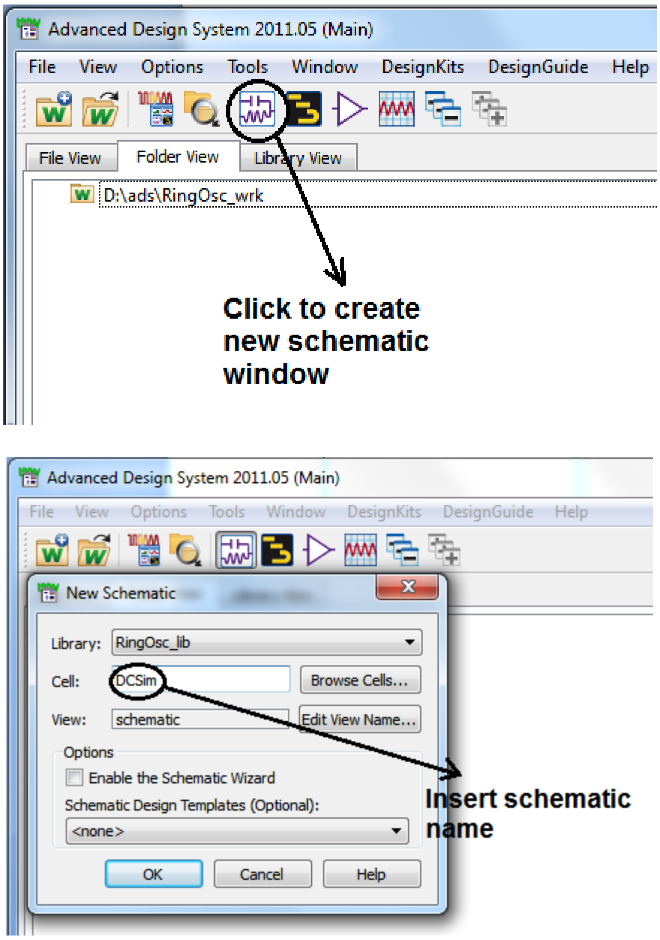



Fig. 4. Creating a new schematic window.

Now draw a common-source gain stage as shown in Fig. 2. You can choose the components needed for drawing using the component “Palette” at the left side of the schematic window as demonstrated in Fig. 5.

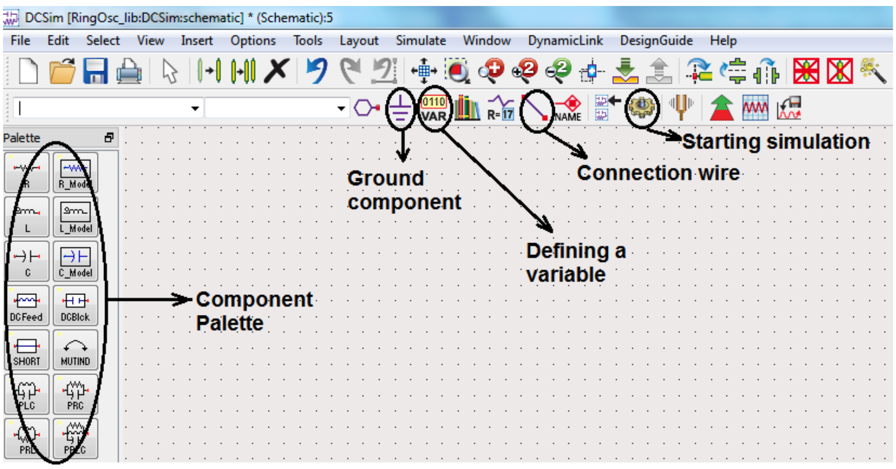



Fig. 5. Schematic window.

Select a NMOS transistor from the “Devices-MOS” menu of the component Palette. In order to simulate a CMOS device in ADS the technology parameters should be defined in schematic. For defining the transistor parameters choose a “LEVEL1_MODEL” component from the Devices-MOS menu and place it in the schematic. After that, the technology parameters can be defined by double clocking on the “LEVEL1_MODEL” component and inserting the parameters values given in Table 1. Also, the NMOS transistor dimensions can be defined by double clicking on the NMOS component. These steps are explained through Fig. 6.

After drawing the circuit of Fig. 2, define the resistor and capacitor values as a variable by placing a “VAR” component in schematic using the “VAR” tab shown in Fig. 5. Assign values to the resistor and the capacitor value by double clicking on the “VAR” component. Then, place a “V_DC” in the schematic for having a DC power supply and set its value to 5 V. Add another “V_DC” and connect it to gate of the transistor for the gate bias and set its value to 1.83 V. For DC simulation the “DC” component should be placed in the schematic from the “Simulation-DC” menu in the component palette. The final circuit is shown in Fig. 7.

Now the circuit is ready for simulation. It will be simulated by pressing the simulation start button (see Fig. 5). By the end of simulation, a window will pop up like Fig. 8 in which the simulation results can be displayed. You can choose different plot types (rectangular, Smith chart, stacked rectangular, data list) for displaying your simulated data. Also, there is an “Eqn” button which can be used for writing equations needed for data post processing. Here, because we just want to see the DC voltages at $V_g$ and $V_d$ nodes which are just single point data, the “Data list” is chosen for displaying them as shown in Fig. 8.

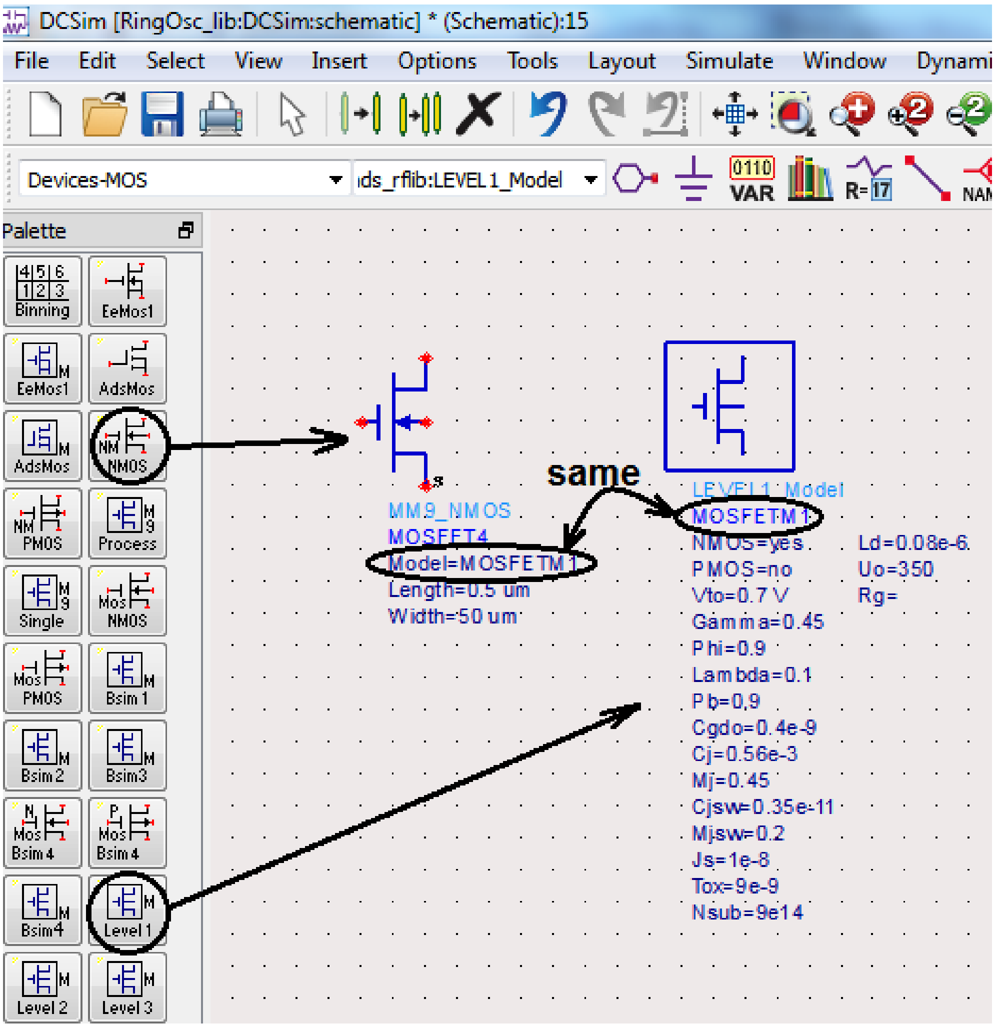



Fig. 6. Setting the NMOS dimensions and the technology parameters.

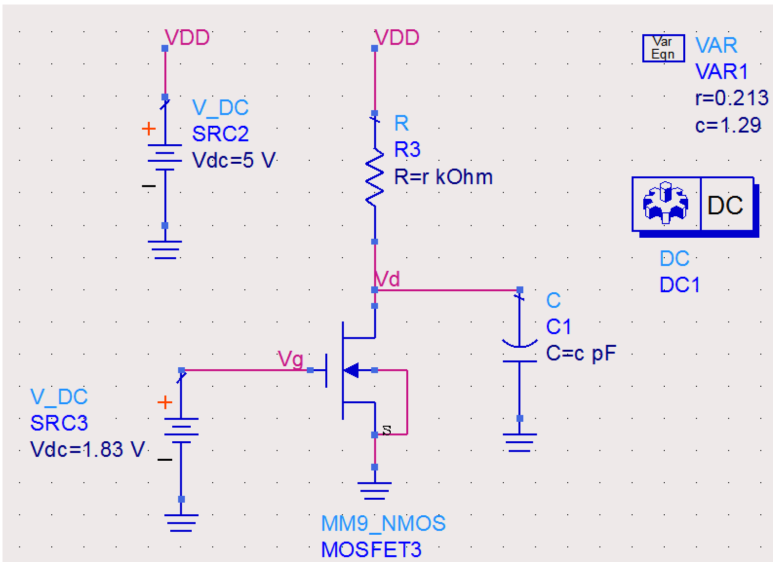



Fig. 7. The drawn gain stage for DC simulation.

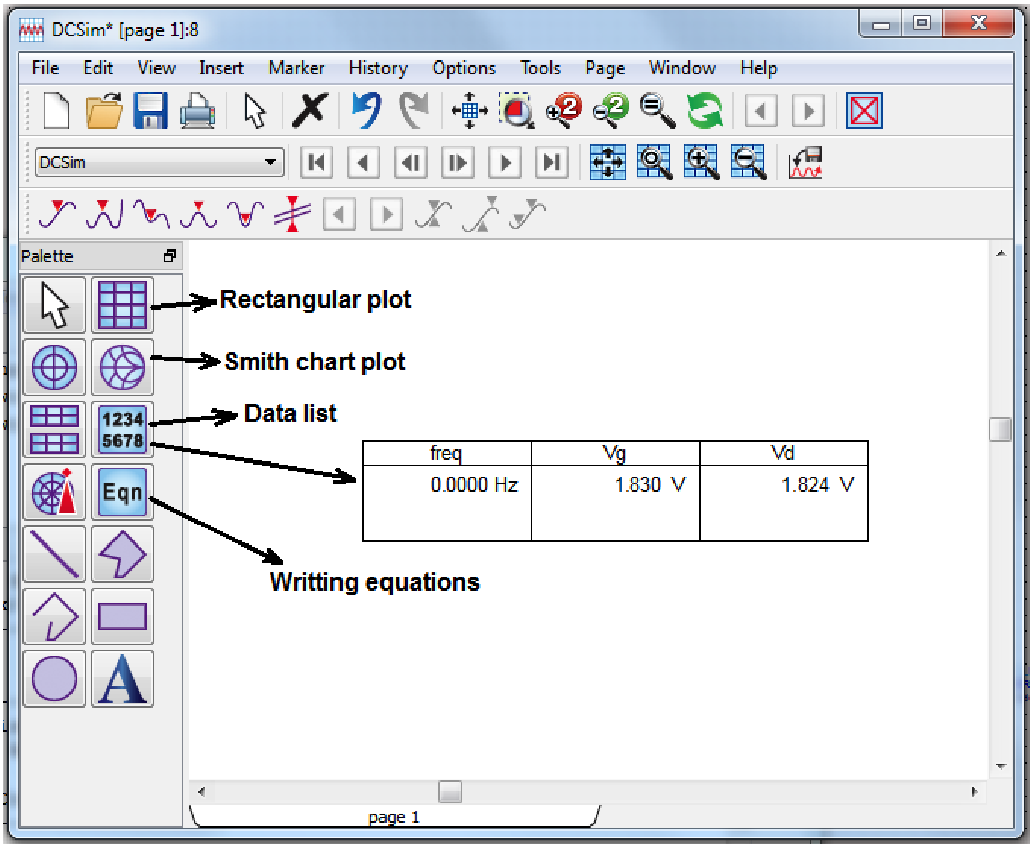



Fig. 8. Displaying the data obtained from simulation.

The simulation show 1.83 V and 1.824 V DC voltages for $V_g$ and $V_d$ respectively which are acceptable for the design (since it is assumed that $V_{d(DC)} = V_{g(DC)}$ in S.1-S.6 ).

## 2-1-2- AC Simulation of Gain stages

At this step, a single gain stage should be simulated to evaluate its gain performance at "$\omega_{osc}$". For measuring the voltage gain, the AC simulation should be done on a gain stage to see the voltage gain ($A_V = {}^{V_d}/_{V_g}$) at the oscillation frequency.

For AC simulation, the circuit should be drawn like Fig. 9. In this simulation:

- The component "V_AC" is connected to the gate of the transistor from the "Sources-Freq Domain" menu in components palette to provide the AC input signal and also the DC bias voltage for the transistor gate.
- The voltage gain is defined to be ($A = {}^{V_d}/_{V_g}$) by inserting a "Meas Eqn" component from the "Simulation-AC" menu of the component palette. The gain "$A$" is defined by double clicking on "Meas Eqn" component and inserting the gain describing equation ($A = {}^{V_d}/_{V_g}$).

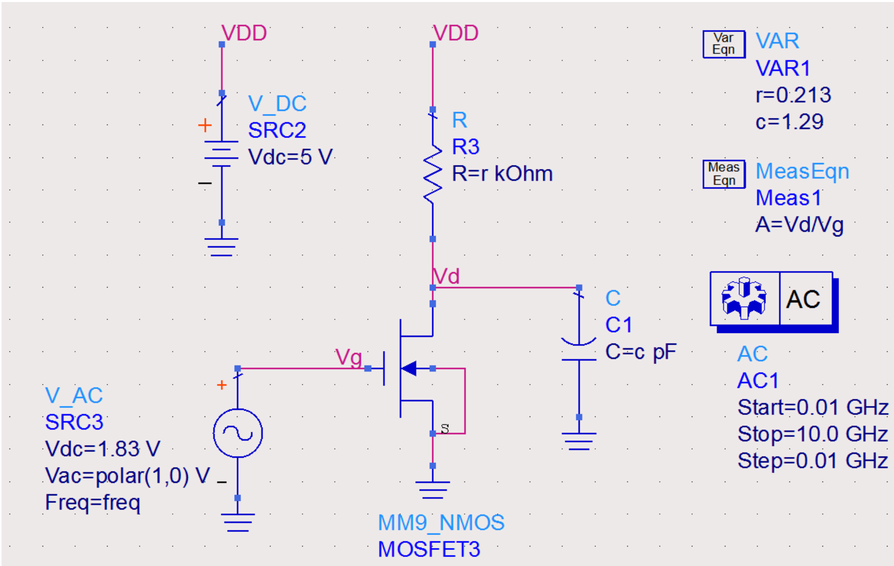



Fig. 9. The circuit setup for AC simulation.

- The AC simulation controller component "AC" is added to the schematic from the "Simulation-AC" menu of the component palette.
- The start/stop frequencies and the simulation step of the AC simulation can be set be double clicking on the "AC" component. A setup shown in Fig. 8 is appropriate for this simulation.

The circuit is ready for simulation now. Click on the simulation button to run the simulation. At the end of simulation, choose the "Rectangular plot" in the data display window and choose the gain "*A*" to be displayed. By doing that, a window will open as shown in Fig. 10. Since we-

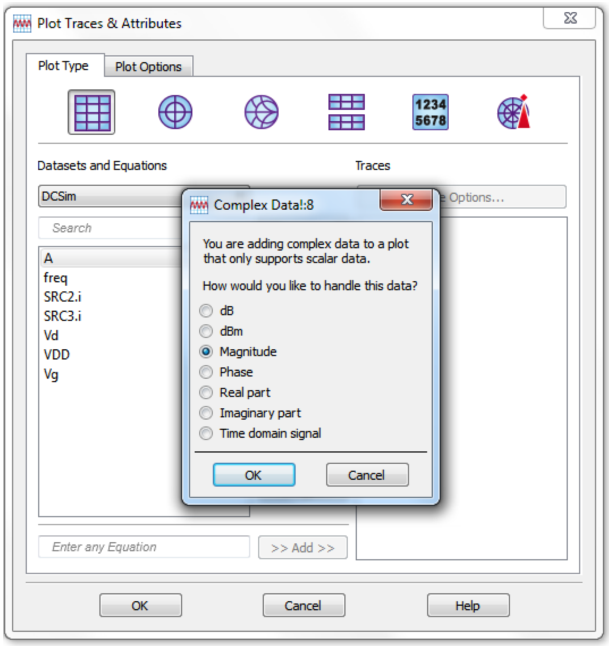
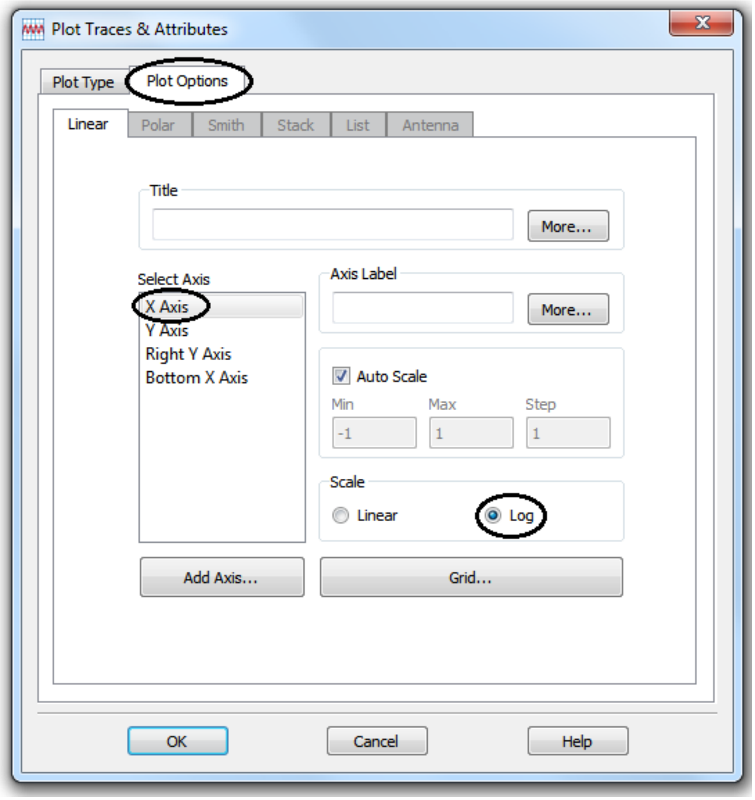
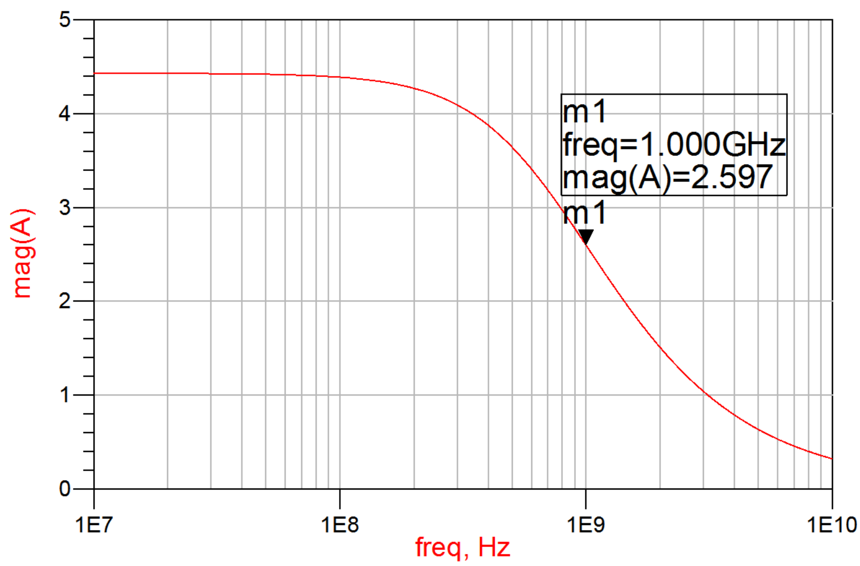



Fig. 10. Displaying the voltage gain "*A*" as a function of frequency.

want to measure the gain magnitude, choose "Magnitude" option and click on "OK". Then, choose the "Plot Options" tab on the top of the window (Fig. 11), choose "X Axis" and change its scale to "Log" as shown in Fig. 11(a) then, click "OK". The gain "A" will be shown as Fig. 11 (b).




(a)




(b)

Fig. 11. (a) Plotting the gain "*A*" as a function of frequency in logarithmic scale. (b) The displayed gain plot.

Based on the gain plot in Fig. 11(b), the voltage gain at the considered oscillation frequency "$f_{osc}$ = 1 GHz" is 2.59 which is in a good agreement with the assumption in (6).

If the simulated gain is smaller than 2 in AC simulation, the design procedure should be repeated again with a larger value of "$R$".

## 2-1-3- Transient Simulation of the Complete Oscillator Circuit

Based on the AC simulation in the last section, the gain stage shows a voltage gain of 2.5 at the considered oscillation frequency which satisfies the oscillation condition in (6). Thus, an oscillator composed of the three designed gain stages should oscillate with a frequency around 1 GHz. In order to see that, a transient simulation should be done on the complete oscillator circuit of Fig. 12.

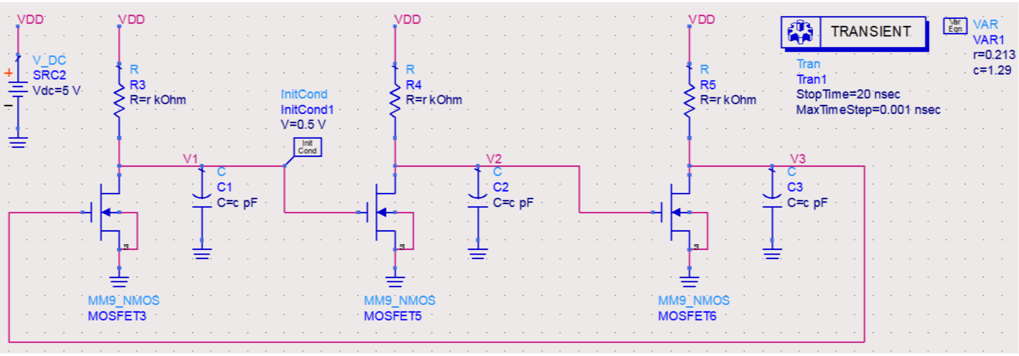



Fig. 12. The complete oscillator circuit setup for transient simulation.

As shown in Fig. 12, for preparing the oscillator for transient simulation:

- The three gain stages are connected to each other in a chain.
- The output (drain) terminals of the stages are named $V_1$, $V_2$ and $V_3$ respectively for displaying their waveform in the simulation results.
- A transient simulation controller "Tran" is added to the schematic from the "Simulation-Transient" menu of the components palette for running the transient simulation.
- The simulation start/stop time and also the time step can be set by double clicking on the simulation controller "Tran" and inserting them. They should be chosen based on the oscillation period. The start time of (0 ns), stop time of (12 ns) with the time step of (1 ps) are appropriate for the considered oscillator with ($f_{osc}$ = 1 GHz).
- An initial voltage is needed on one of the capacitors to start up the oscillation. This can be done by inserting a "InitCond" component from the "Simulation-Transient" menu of the components palette and connecting it to the one of the capacitors. The initial voltage can be set by double clicking on "InitCond" component and inserting the initial voltage. Here, this component is connected to the drain of the first transistor ($V_1$) and setting the initial voltage to 0.5 V as seen in Fig. 12.

Now, the circuit is ready for simulation. By pressing the simulation start button, the simulation will run. At the end of simulation, the output waveforms can be displayed on the data display window. The simulated waveform for "$V_1$" is demonstrated in Fig. 13.

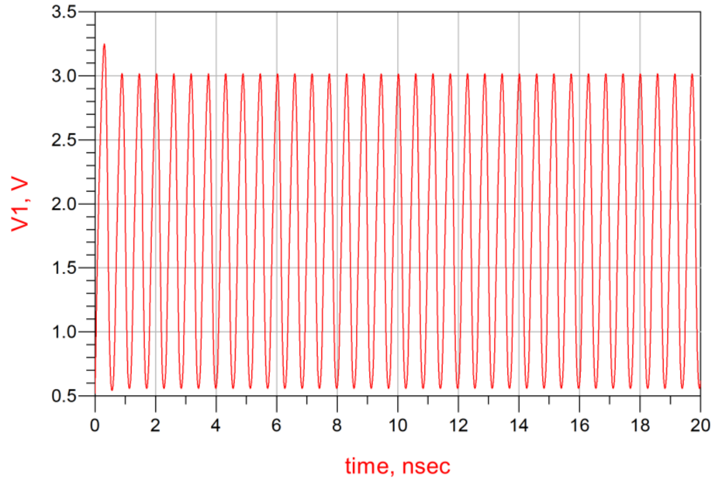
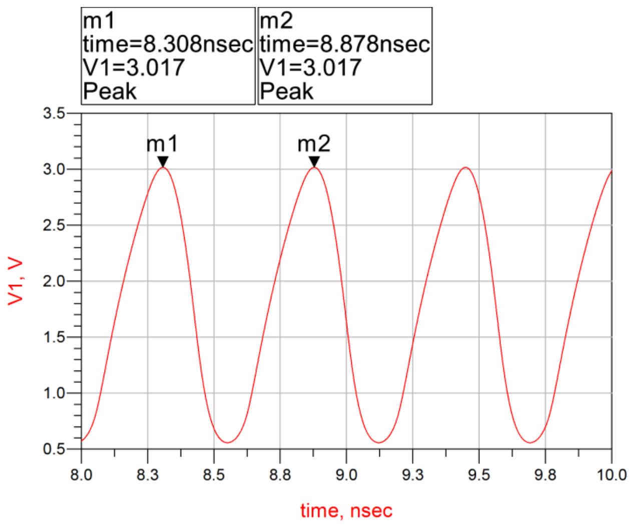



Fig. 13. The simulated waveform of "$V_1$" output.

For accurately measuring the oscillation period and frequency, the simulation should be run in a smaller time interval. A simulation time from 8 to 10 ns would be appropriate. The simulation result for 8 – 10 ns is demonstrated in Fig. 14.




Eqn T=indep(m2)-indep(m1)

Eqn f=1/T

| time | T | f |
|---|---|---|
| <invalid>sec | 5.700E-10 | 1.754E9 |

Fig. 14. Measuring the oscillation period and frequency.

For measuring the oscillation period and frequency, we can measure the time difference between the two adjacent max or min points of the waveform by inserting two data markers like Fig. 14. Then, the period and frequency can be obtained by inserting "Eqn" from the palette in data display window and writing the period (T) and frequency (f) equations in them like Fig. 14. The values of (T) and (f) can be displayed by inserting a data list plot. By inserting a data list plot a window will be opened as Fig. 15. In this window, insert the data that has been defined in "Eqn" (T or f) and click "Add" as shown in Fig. 15.

Based on the Fig. 14, the obtained oscillation period and frequency are 0.57 ns and 1.754 GHz respectively. As seen, the obtained oscillation frequency is larger than 1 GHz. This is because (1) and (3) equations for oscillation frequency are obtained based on small signal analysis that is only accurate for oscillation start up when the oscillation amplitude is very small. But after the start of oscillation and in stable oscillation, the dynamic of the circuit is completely nonlinear and the waveforms are all large signal and the small signal analysis is no longer valid for the oscillator. More accurate equations for describing the oscillation frequency of the ring oscillators can be found in [2].

All of the three output waveforms of the oscillator can be displayed in one plot to see the phase difference between them. This plot is demonstrated in Fig. 16 which shows approximately 120 degree phase shift between each couple of the outputs confirming the concept of 3-stage ring oscillator.

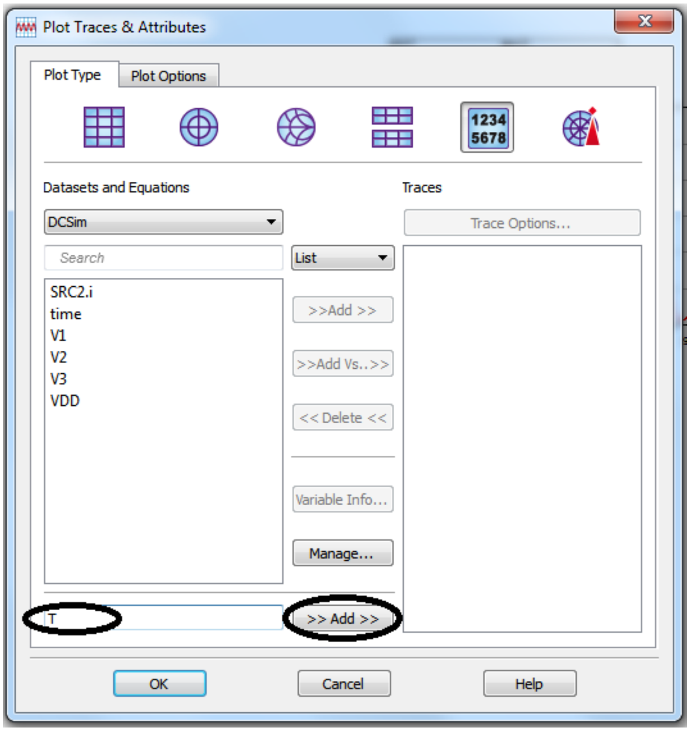



Fig. 15. Displaying the data defined in "Eqn".

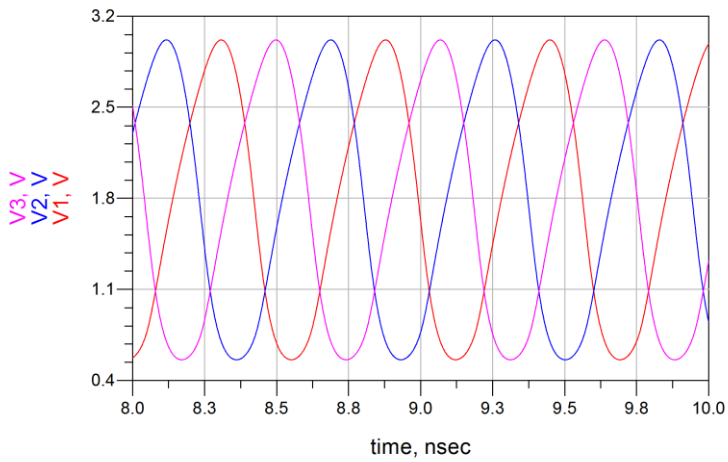



Fig. 16. Three outputs of the 3-stage ring oscillator shown in a single plot.

## 2-1-4- Harmonic Balance Simulation of the Complete Oscillator Circuit

Harmonic balance is a frequency-domain analysis technique for simulating distortion in nonlinear circuits and systems. It is usually the method of choice for simulating analog RF and microwave problems, since these are most naturally handled in the frequency domain. Within the context of high-frequency circuit and system simulation, harmonic balance offers several benefits over conventional time-domain transient analysis. Harmonic balance simulation obtains frequency-domain voltages and currents, directly calculating the steady-state spectral content of voltages or currents in the circuit. The frequency integration required for transient analysis is prohibitive in many practical cases. Many linear models are best represented in the frequency domain at high frequencies [3]. Use the HB simulation to:

- Determine the spectral content of voltages or currents.
- Compute quantities such as third-order intercept (TOI) points, total harmonic distortion (THD), and inter-modulation distortion components.
- Perform power amplifier load-pull contour analyses.
- Perform nonlinear noise analysis.

In this section, we will find the oscillator output voltage spectrum and the phase noise using the harmonic balance simulation.

### 2-1-4-1- Oscillator Output Voltage Spectrum

The main goal of simulating an oscillator using harmonic balance is to find the output spectrum of the oscillator. In order to find the output voltage spectrum of the designed ring oscillator, the simulation setup should be changed as shown in Fig. 17. In this setup:

- An "**OscPort**" component is inserted in the oscillator loop from the "Simulation-HB" menu of the component palette. The "**OscPort**" kept at the default setup.

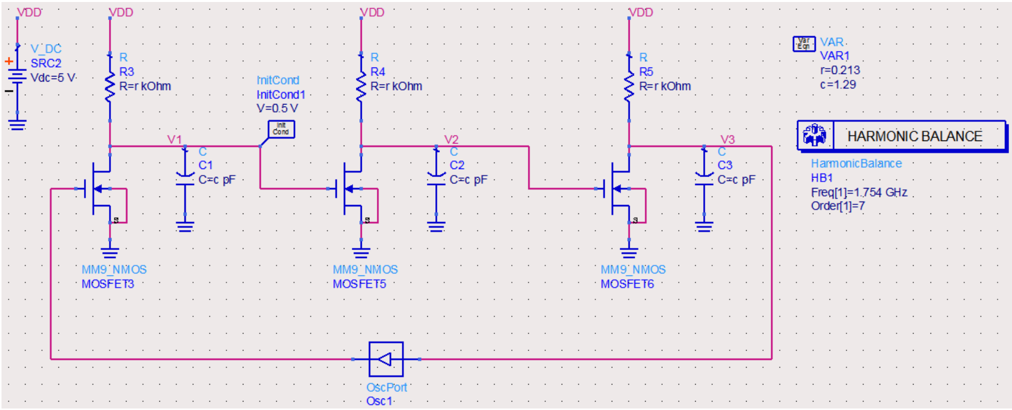



Fig. 17. The setup for harmonic balance simulation of the oscillator circuit.

- The harmonic balance simulation controller "**HarmonicBalance**" is added to the schematic from the "Simulation-HB" menu of the component palette.
- Double click on the harmonic balance simulation controller and on the top of the dialog box, select **Oscillator**. Click on "**Enable Oscillator Analysis**" option. Then change the method to "**Use Oscport**".
- Scroll to the "Freq" tab and enter the oscillation frequency (1.754 GHz) as the fundamental frequency (Freq[1]). Set the "**Order**" to the number of harmonics that you want to see in the output (we set to 7 here).
- Click "**Add**" button to at this setup to the frequency plan. Cut any other frequency setup.

Now, the circuit is ready for simulation. Press the simulation start button to run the simulation. At the end of simulation, in the data display window choose a rectangular plot to see the voltage spectrum of the output nodes. By choosing the rectangular plot, a window will be opened. In this window choose the output node ($V_1$, $V_2$ or $V_3$) in which you want to see the spectrum and then press the add button. In the opened dialog box choose the "**Spectrum in -**

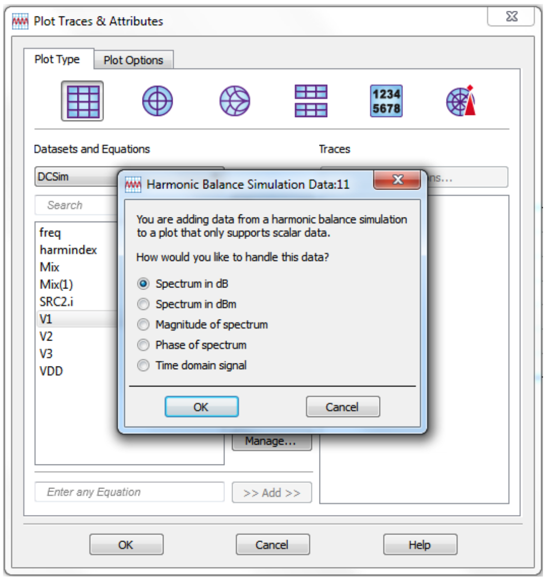



Fig. 18. Showing the voltage spectrum of output nodes.

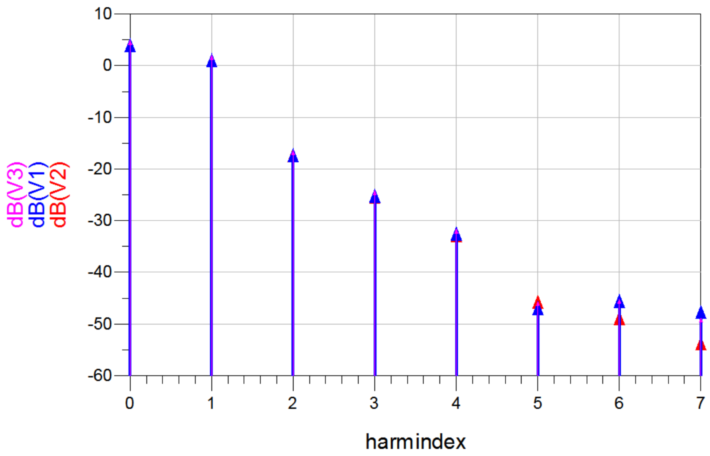



Fig. 19. Output voltage spectrum of the designed ring oscillator.

**dB**" option and click OK. The procedure is explained in Fig. 18. Now the spectrum will be displayed as Fig. 19.

## 2-1-4-2- Oscillator Phase Noise

The most important parameter of an oscillator is its phase noise performance. The phase noise of the oscillator can be simulated as follows in ADS software:

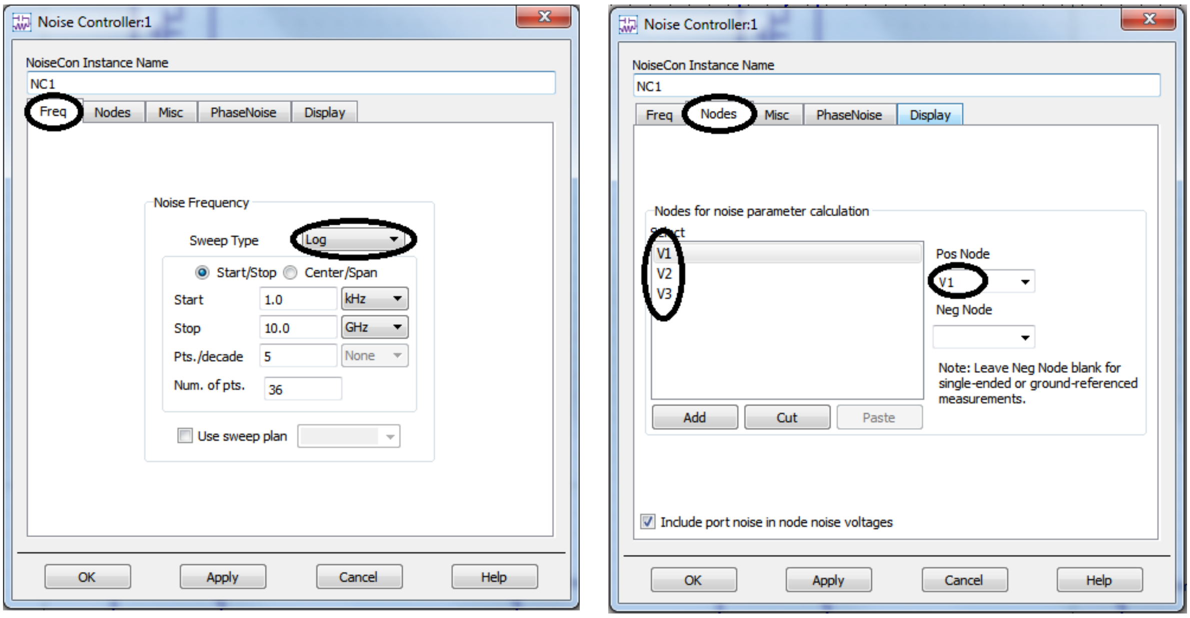
*Simulation Using* "**NoiseCons**"

In order to determine oscillator phase noise:

- Insert a harmonic balance noise controller "**NoiseCon**" component from the "**Simulation-HB**" menu of component palette in to the schematic.
- Double click on the "**NoiseCon**" component. Then choose the "**Freq**" tab on the dialog box, set the sweep type to "**Log**" and do the "**Noise Frequency**" settings. A setting shown in Fig. 20(a) should be ok for the simulation.
- In the same dialog box, choose the "Nodes" tab and from the "**Pos Node**" field choose the nodes in which you want to measure the phase noise ($V_1$, $V_2$ and $V_3$) and add them to the list. Leave the "**Neg Node**" field empty since it is used for the simulation of differential oscillators. This is shown in Fig. 20(b). Click on "**Apply**" and then "**OK**" buttons to confirm these settings.

Now the harmonic balance simulation controller "**HarmonicBalance**" should be edited as follows for the phase noise measurement.

- Double click on the harmonic balance simulator controller. Insert the oscillation frequency in the "**Fundamental Frequencies**" field and set the order of the harmonics that you want to simulate. Click "Add" button to add this setting to your simulation. Cut any other frequency settings. This is explained in Fig. 21(a).

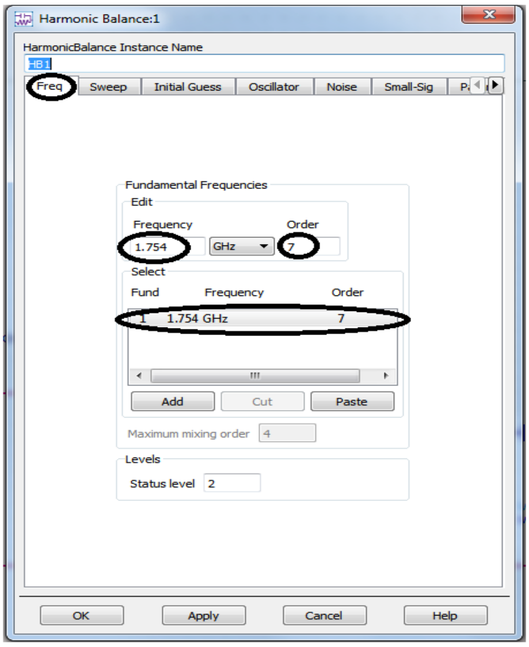



(a) (b)

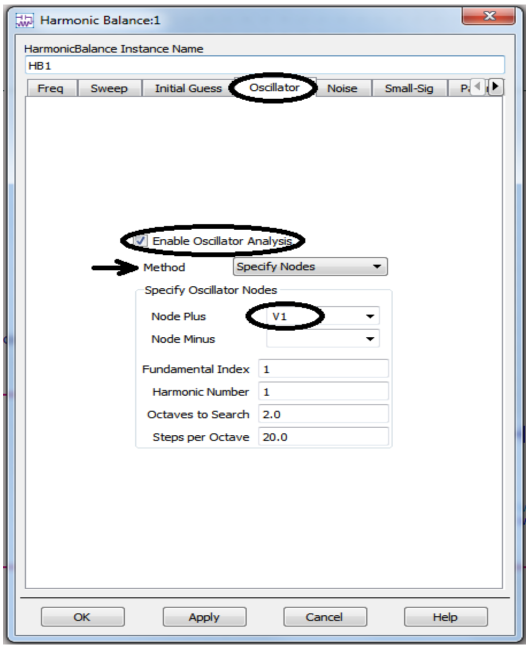
Fig. 20. Setting the "NoiseCon" parameters for the phase noise simulation.

- Choose the "**Oscillator**" tab on the top of the dialog box and "**Enable Oscillator Analysis**". Change the "**Mode**" to "**Specify Nodes**". Set the "**Node Plus**" to the node in which you want to measure the phase noise (we have chosen $V_1$ here but, $V_2$ and $V_3$ can also be chosen). Leave "**Node Minus**" empty since it is used for the differential oscillators. This is explained in Fig. 21(b).
- Choose the "**Noise**" tab on the top of the dialog box and enable "**NoiseCons**" option. Then on the "**Edit**" field choose "**NC1**" as the noise controller and click the "**Add**" button. Also enable the "**Nonlinear noise**" option. This is shown in Fig. 21(c).
- In the same window click on "**Noise (1)**" button. A window will open like Fig. 21(d) which asks you to set the noise frequency. Set these frequency fields exactly the same as the frequency setting in **HB Noise Controller** component then click "**OK**".
- Now click on "**Noise (2)**" button. In the opened window from the "**Edit**" field choose the oscillator output nodes ($V_1$, $V_2$ and $V_3$) and add them to the list of the nodes in which you want to measure the phase noise then click on "**OK**".
- Click on "**Apply**" and then "**OK**" buttons to confirm the Harmonic Balance simulation settings.

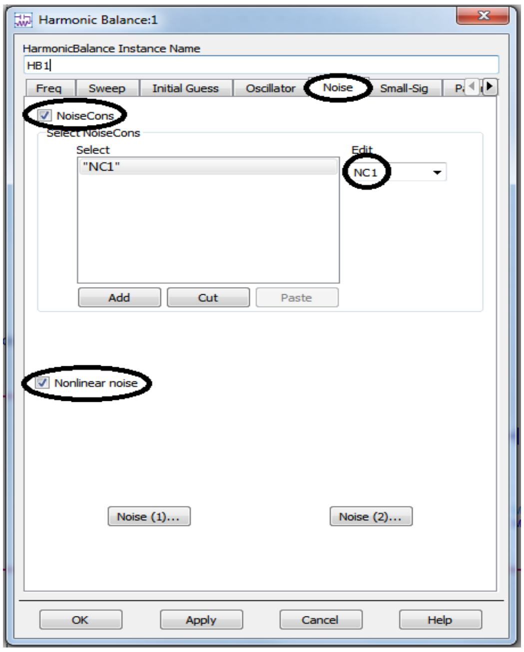
The circuit is ready for simulation now. Press the simulation start button to run the simulation. By the end of simulation, the results can be displayed on the data display window by inserting a rectangular plot showing "**V1.pmnx**" on "**dBc**" scale. The phase noise of the other output nodes can be measured by plotting "**V2.pmnx**" and "**V3.pmnx**". As an example the phase noise at $V_3$ "**V3.pmnx**" is plotted in Fig. 22.



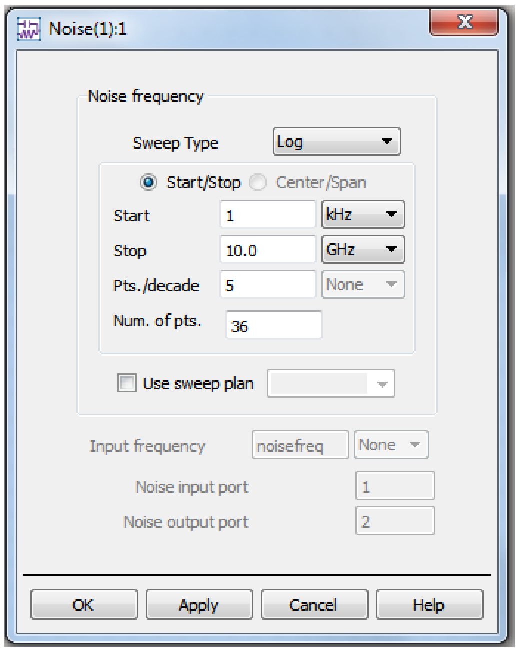

(a)




(b)




(c)




(d)

Fig. 21. Setting the harmonic balance simulation controller for phase noise simulation.

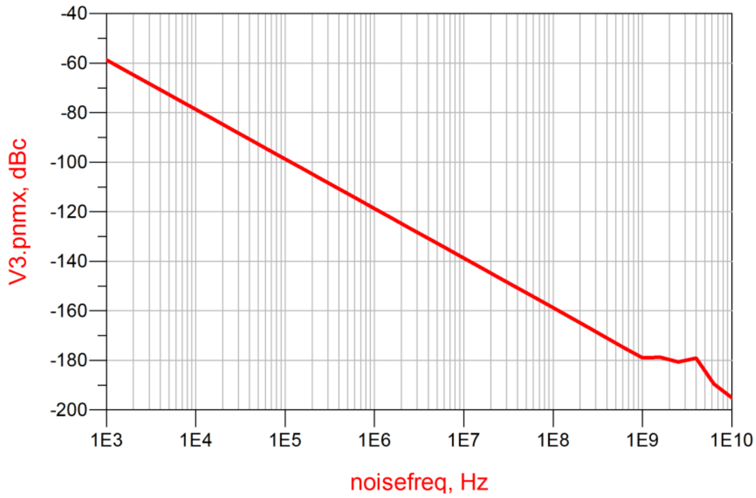



Fig. 22. The simulated phase noise of the oscillator at $V_3$ output node.

# References


[1] B. Razavi, Design of Analog CMOS Integrated Circuits, ch. 14, McGraw Hill, New York, 2001.

[2] Farahabadi, P.M.; Miar-Naimi, H.; Ebrahimzadeh, A, "Closed-Form Analytical Equations for Amplitude and Frequency of High-Frequency CMOS Ring Oscillators," *IEEE Transactions on Circuits and Systems I: Regular Papers*, vol.56, no.12, pp.2669,2677, Dec. 2009.

[3] A. Ebrahimi, H. MiarNaimi, "Analytical Equations for Amplitude Analysis of MOS Colpitts Oscillator," *International Journal of Electronics*, Vol.98, No. 7, pp. 883-900, July 2011.

[4] A. Ebrahimi, H. MiarNaimi and H. Adrang, "Remarks on Transient Amplitude Analysis of MOS Cross-Coupled Oscillators," *IEICE Transactions on Electronics*, Vol. E49, No.2, pp. 231-239, February 2011.

[5] A. Ebrahimi and P. Yaghmaee, "A New Enhanced CMOS Differential Colpitts Oscillator," *Journal of Circuits Systems and Computers*, Vol. 23, No. 1, 2013.

[6] Ebrahimi, A., Naimi, H.M., "An improvement on the analytical methods for amplitude analysis of the MOS Colpitts oscillator," Symbolic and Numerical Methods, Modeling and Applications to Circuit Design (SM2ACD), 2010 XIth International Workshop on , vol., no., pp.1,5, 4-6 Oct. 2010.

[3] http://eesof.tm.agilent.com/docs/adsdoc2004A/manuals.htm