

Middle East Technical University
Electrical and Electronics Engineering Department



EE414 – Introduction to Analog Integrated Circuits

Cadence Parametric Sweep Tutorial

In the first part of this tutorial, you will start making temperature sweep simulation and parametric sweep simulation of the components used at your design. The second part of the tutorial is about how to observe AC analysis of your design.

1.1. Parametric Sweep Simulation

First of all, for making parametric sweep simulation the parameter that is going to be swept has to be variable. Figure 1 shows a schematic example and it is used for observing the I_D vs. V_{DS} graph of an NMOS transistor. At the schematic you can see that supply voltage which is equal to drain-source voltage "vds" of the transistor and gate source voltage of the transistor "vgs" are assigned as variable. From the properties of the components, parameters can be set as variable. Figure 2 shows an example on setting DC voltage of a voltage source as a variable named "vgs".

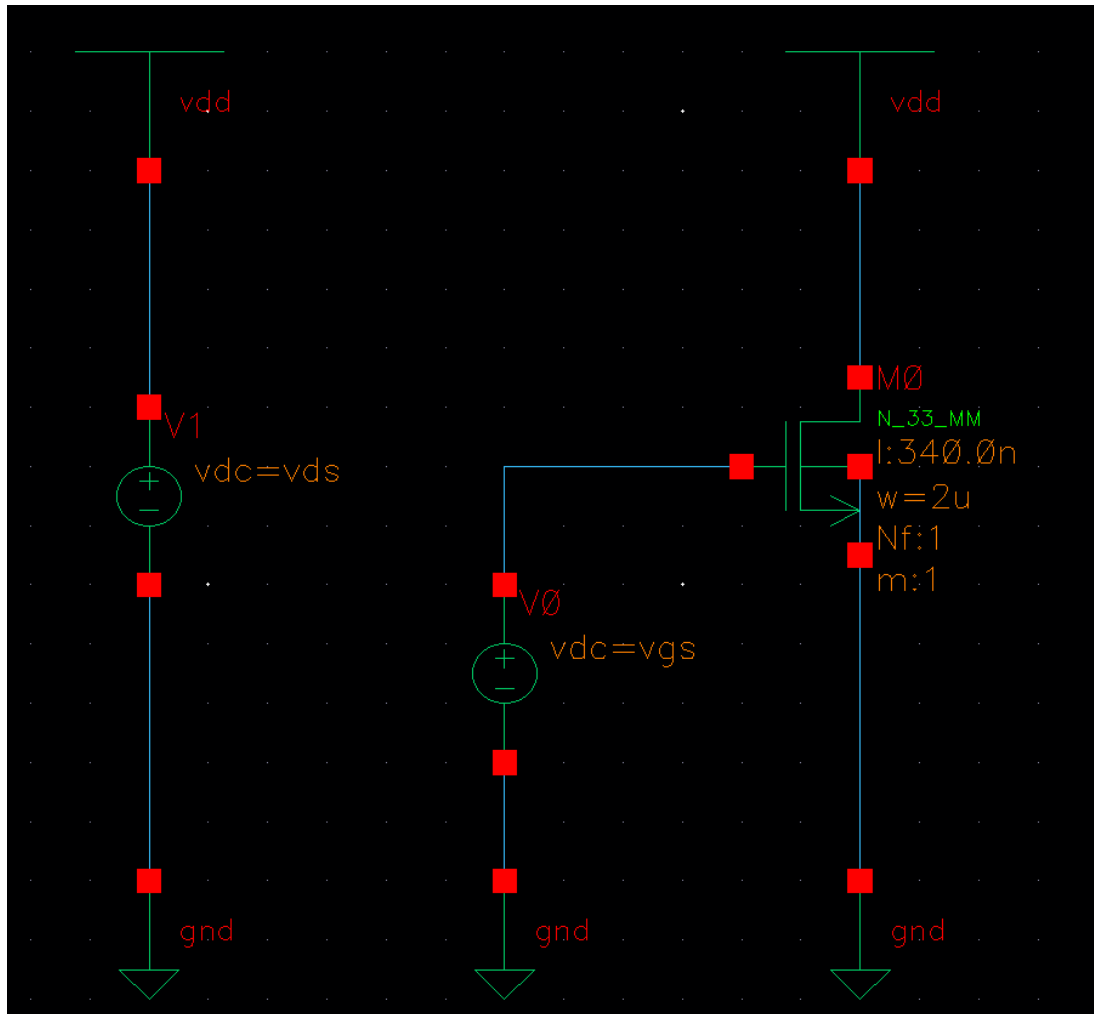


Figure 1: Schematic of the circuit used parametric sweep

Figure 2: Setting DC voltage of a voltage source as variable

After completing the schematic drawing of the circuit following steps are going to be followed for parametric sweep simulations.

1. Click on "Launch->ADE L" in order to run analog simulator.
2. Now, Click on "Variables->Edit" for setting values of the variables which was used at the schematic. Figure 3 shows an example of where values of variables "vgs" and "vds" are set to 2 V and 3.3 V, respectively.

Figure 3: Editing design variable screen

- Click on, "Analyses->Choose". For making DC sweep or DC simulation "dc" option must be chosen. The variable that is going to be swept is set as a design variable at previous step, so click on "Design Variable" under the "Sweep Variable" part and click on "Select Design Variable" button for selecting the variable that is going to be swept in DC analysis.

The screenshot shows the "DC Analysis" configuration window. In the "Analysis" section, the "dc" radio button is selected. Under the "Sweep Variable" section, the "Design Variable" checkbox is checked, and the "Variable Name" is set to "vds". The "Sweep Range" section shows "Start-Stop" selected, with "Start" at 0 and "Stop" at 3.3. The "Sweep Type" section shows "Linear" selected in the dropdown, and "Number of Steps" is set to 50. The "Enabled" checkbox is checked. At the bottom, there are buttons for "OK", "Cancel", "Defaults", "Apply", and "Help".

- Write 0 to "Start" and 3.3 to "Stop". You have set the range of sweep variable from 0 to 3.3 V. Set the "Sweep Type" to be "Linear" and write 50 to "Number of Steps" so, the variable "vds" are going to change linearly from 0 to 3.3 V at 50 steps.

- Now, you are ready to make "DC simulation" by selecting drain current of the transistor we can observe the $I_D - V_{DS}$ characteristic of the transistor at $v_{gs}=2V$. For observing $I_D - V_{DS}$ characteristic at different Gate-Source voltages you have to make Parametric Sweep.

- Click on "Tools->Parametric Analysis". Choose "vgs" as the variable and write from 0 to 5 as the range of the sweep. Finally, set the step control as "Linear Steps" and write 1 to "Step Size". You have programmed the simulator to change the variable "vgs" from 0 to 3.3 V with 0.66 V linear steps. Figure 5 shows Parametric Analysis options explained here.

- Click on "Analysis->Start All" to start the simulation. After the simulation is completed, a waveform window will be opened. Figure 6 shows the $I_D - V_{DS}$ characteristic of the NMOS transistor at different Gate Source voltages.

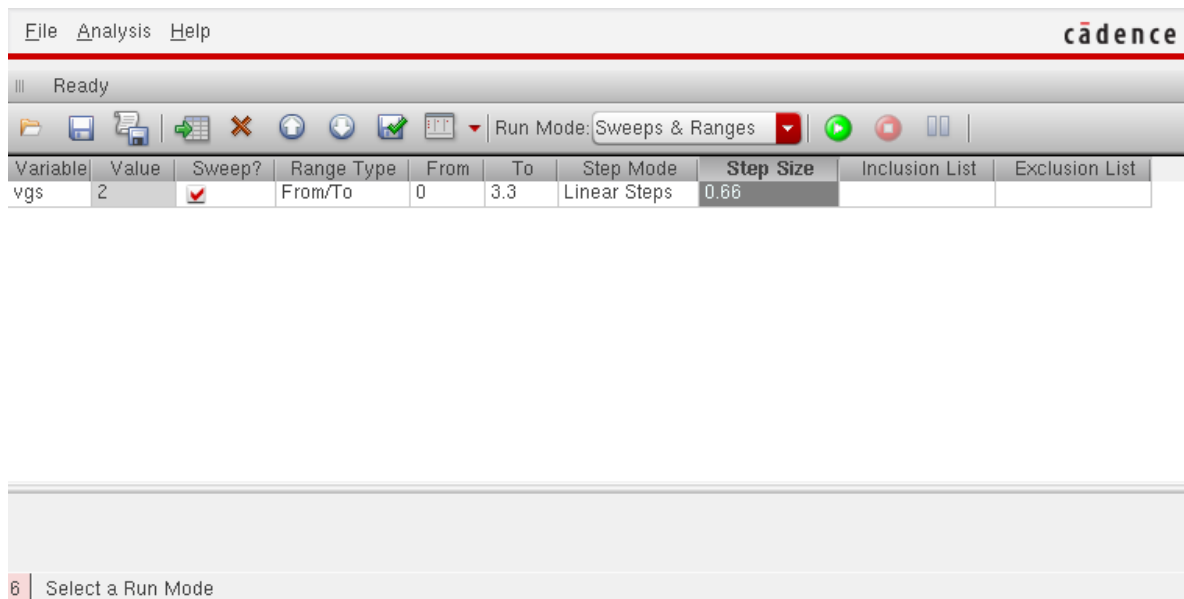


Figure 5: Parametric Sweep options

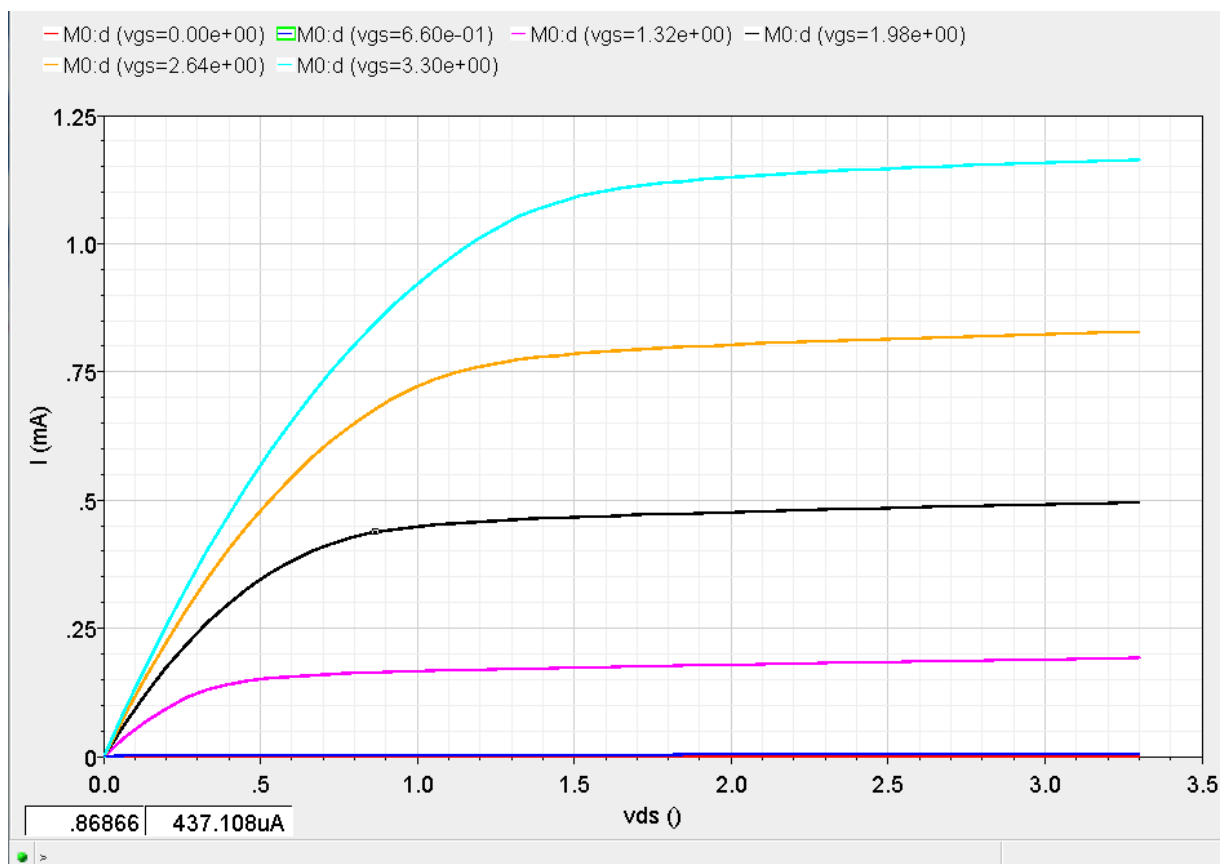


Figure 6: I_D - V_{DS} characteristic of the NMOS transistor

1.2. Temperature Sweep Simulation

For observing characteristic of the circuit at different temperatures, temperature that the circuit operates can be swept. Figure 7 shows the schematic of an inverter circuit. By sweeping the temperature, the output voltage of the circuit can be easily observed at different temperatures.

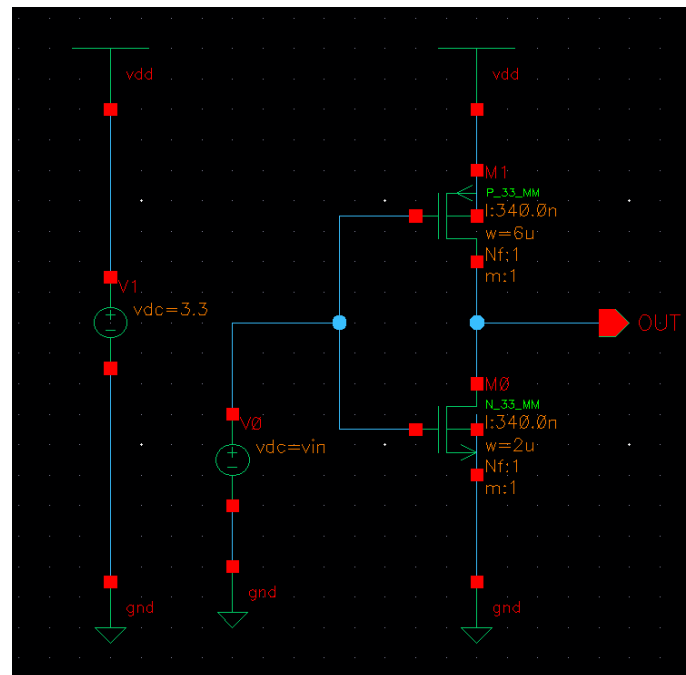


Figure 7: Schematic of an inverter circuit

For sweeping the temperature following steps can be applied.

1. After you run the analog simulator, click on "Analyses->Choose".
2. The temperature sweep is under DC analysis so mark the "dc" option under Analysis. The variable that is going to be swept is temperature so mark the "Temperature" under Sweep Variable. Write 0 and 75 to "Start" and "Stop", respectively. Choose "Linear" as the Sweep Type and determine number of steps (e.g. 50). Click "OK". You have adjusted the sweep range of Temperature from 0 to 75 and the temperature is going to change linearly with totally 50 steps. Figure 8 shows analyses screen that you adjust these settings.
3. After you choose the output to be observed you are ready to make simulation. Click on "Simulation->Run". Figure 9 shows the change of output voltage of the inverter circuit with respect to temperature for input voltage equal to 2 V.

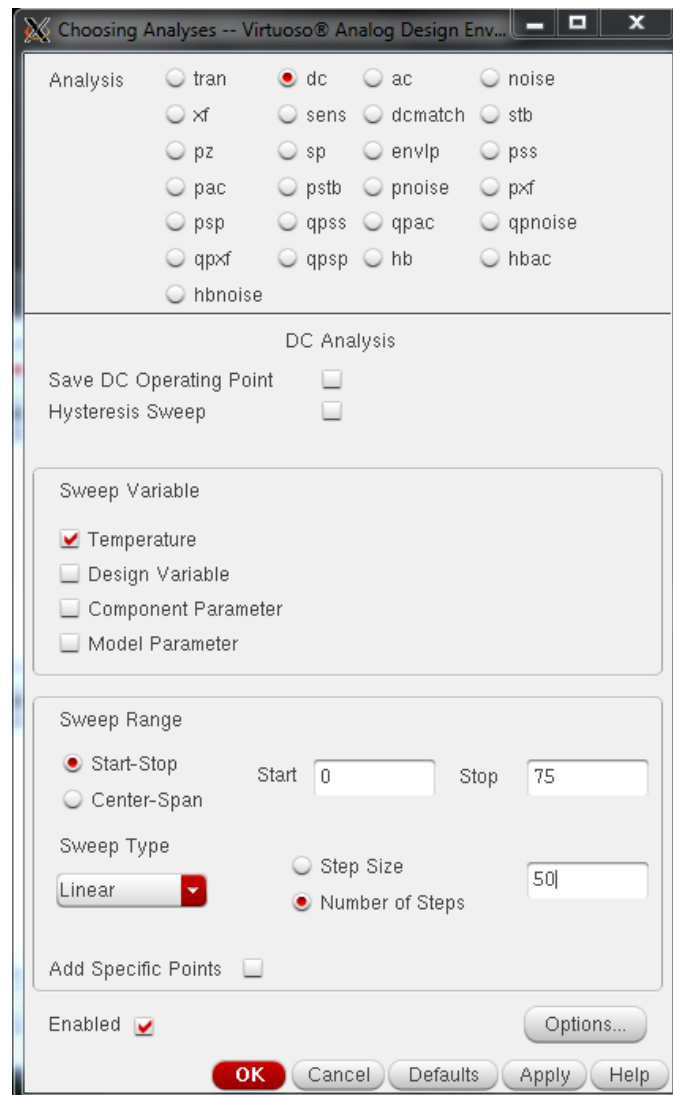


Figure 8: Choosing analysis to be observed

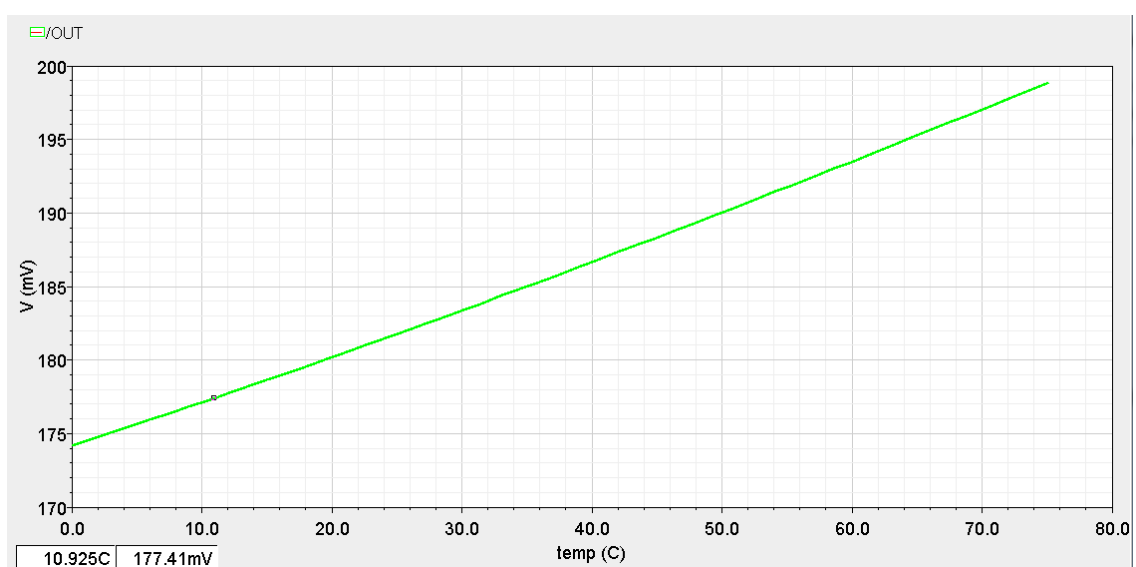


Figure 9: Change of output voltage with respect to Temperature

2. Observing AC Response

At this part of the tutorial, information about how you can observe the AC response of the circuit is going to be explained. Figure 10 shows the schematic of an inverter circuit. Let's step by step see how we can observe AC response of this circuit.

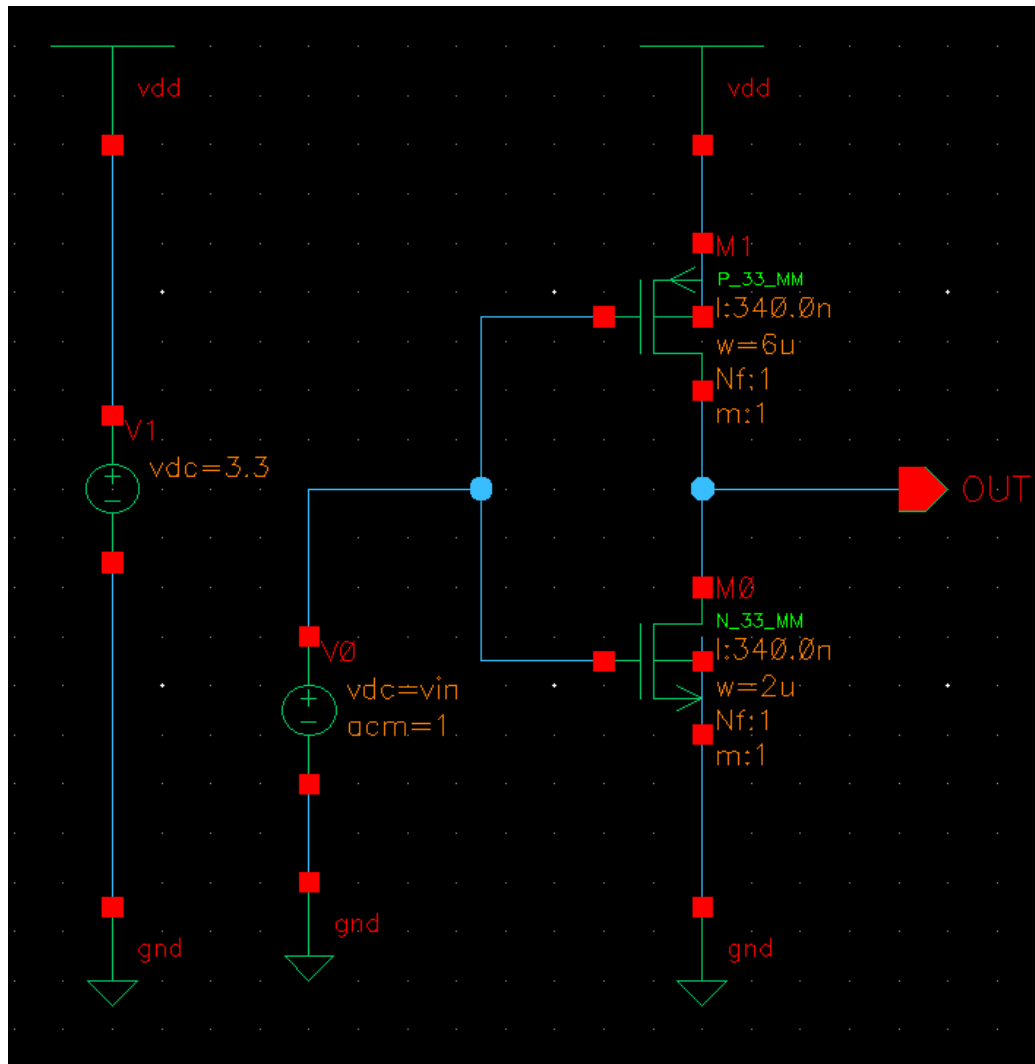


Figure 10: Schematic of an inverter circuit

1. After schematic drawing of the circuit completed, open properties of the DC source which was used as the input of the circuit. Write 1 to value of the "AC magnitude" AC phase of the source is automatically determined as 0 if you don't specify any value. You have added an AC signal with 1 V magnitude and 0° phase on your DC input source. Figure 11 shows properties of the input voltage source.
2. Click on "Launch->ADE L" in order to run analog simulator. Then, click on "Analyses->Choose".

3. Choose "ac" as the analysis type. Mark "Frequency" as the sweep variable type. Write 1 and 10G to "Start" and "Stop" of the range, respectively. Choose "Logarithmic" as the sweep type and set 50 points per decade measurement option. You have programmed the simulator to sweep the frequency from 1 to 10 GHz logarithmically and at each decade the simulator is going to take 50 measurement data. Figure 12 shows analysis type options.

Apply To		<input type="button" value="only current"/>	<input type="button" value="instance"/>
Show		<input type="checkbox"/> system	<input checked="" type="checkbox"/> user <input checked="" type="checkbox"/> CDF
<input type="button" value="Browse"/>		<input type="button" value="Reset Instance Labels Display"/>	
Property	Value	Display	
Library Name	analogLib	off <input type="button" value="v"/>	
Cell Name	vdc	off <input type="button" value="v"/>	
View Name	symbol	off <input type="button" value="v"/>	
Instance Name	V0	off <input type="button" value="v"/>	
<input type="button" value="Add"/>		<input type="button" value="Delete"/>	<input type="button" value="Modify"/>
User Property	Master Value	Local Value	Display
Ivsignore	TRUE		off <input type="button" value="v"/>
CDF Parameter	Value	Display	
Noise file name		off <input type="button" value="v"/>	
Number of noise/freq pairs	0	off <input type="button" value="v"/>	
DC voltage	vin V	off <input type="button" value="v"/>	
AC magnitude	1	off <input type="button" value="v"/>	
AC phase		off <input type="button" value="v"/>	
XF magnitude		off <input type="button" value="v"/>	
PAC magnitude		off <input type="button" value="v"/>	
PAC phase		off <input type="button" value="v"/>	
Temperature coefficient 1		off <input type="button" value="v"/>	
Temperature coefficient 2		off <input type="button" value="v"/>	
Nominal temperature		off <input type="button" value="v"/>	
<input type="button" value="OK"/> <input type="button" value="Cancel"/> <input type="button" value="Apply"/> <input type="button" value="Defaults"/> <input type="button" value="Previous"/> <input type="button" value="Next"/> <input type="button" value="Help"/>			

Figure 11: Properties of the input voltage source

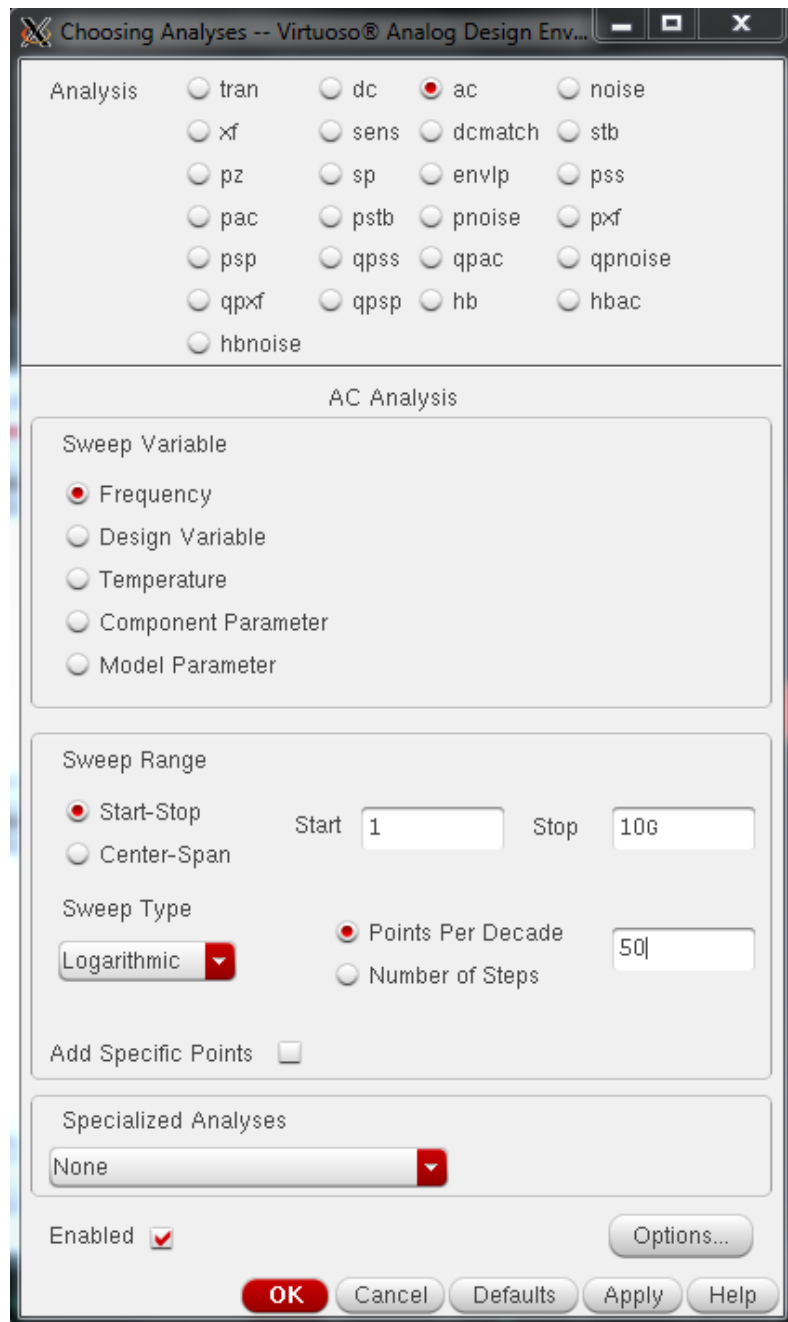


Figure 12: Choosing analysis type and its options

4. Now, you are ready to make AC analysis. Click on "Simulation->Run".
5. For observing the AC response of the circuit; click on "Results->Direct Plot-> AC Magnitude & Phase" after simulation completed. Now, you have to choose the point that you want observe the AC characteristic. Focus the schematics editor and click on the wire at the output of the inverter and then press "Esc" button.
6. You should see a waveform similar to the Figure 13 which is the ac response of the inverter circuit shown at Figure 10.

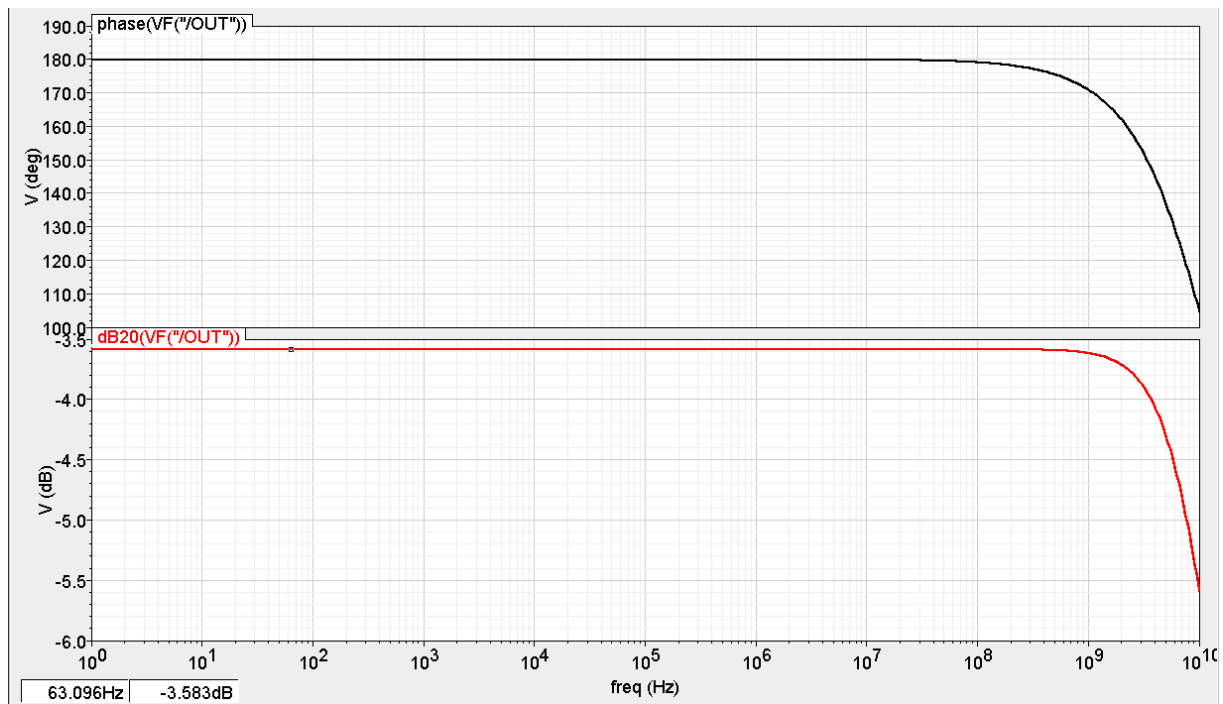


Figure 13: AC response of the inverter circuit