

## Analog IC Design

### OTA Simulation Tutorial

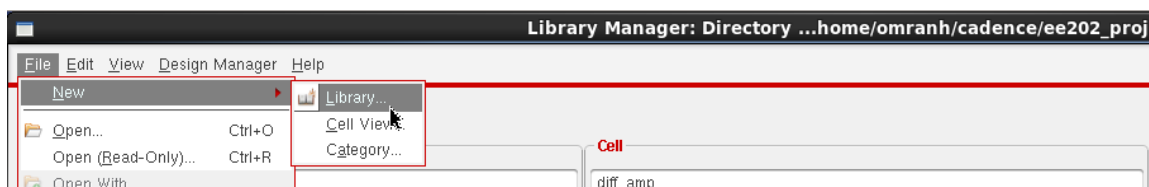
In this tutorial we will simulate an OTA that was designed using gm/ID methodology.

Note that this tutorial was done using AMS 0.18um technology. Your technology may be different, so library/instance/parameter names may be different.

## Part 1: Design Charts

### Creating Design Library

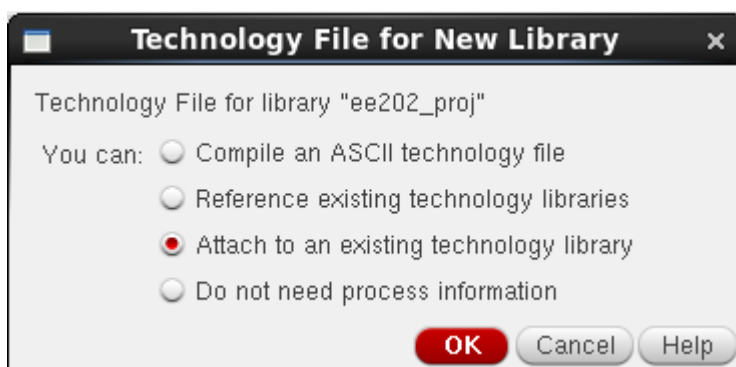
From library manager create new library.



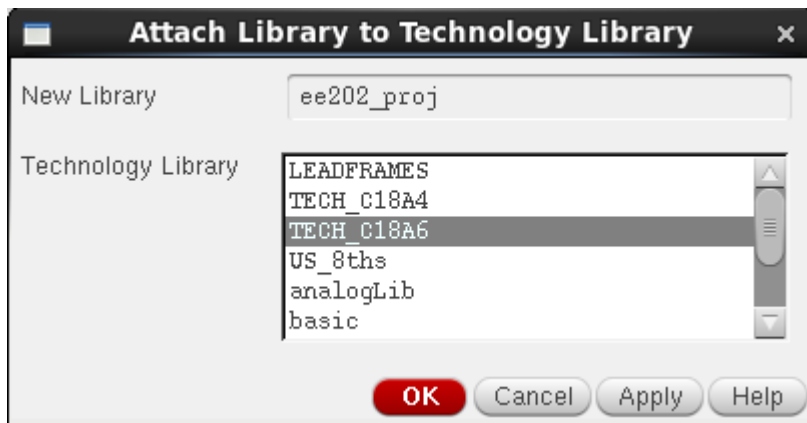
Give your lib a name.



Attach it to our technology (note that some pop-up windows may appear in the background).

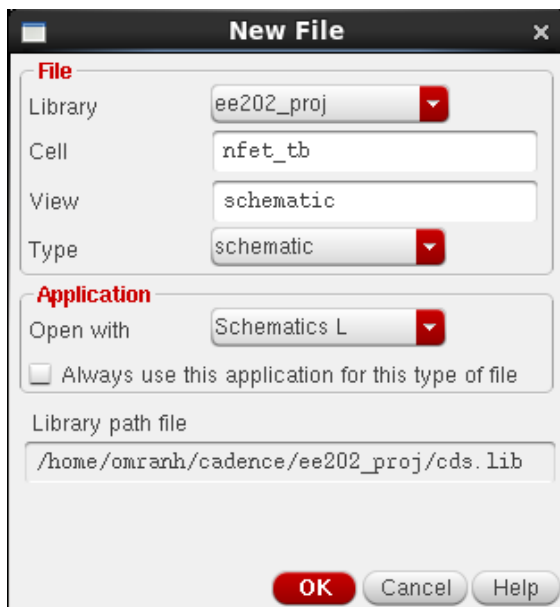
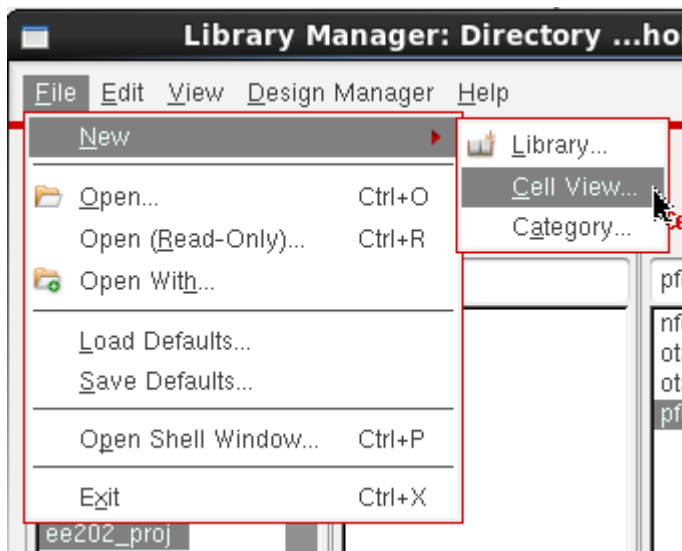


The below snapshot applies for AMS 0.18 technology. Your technology may be different. This step is important only if you will be doing layout.

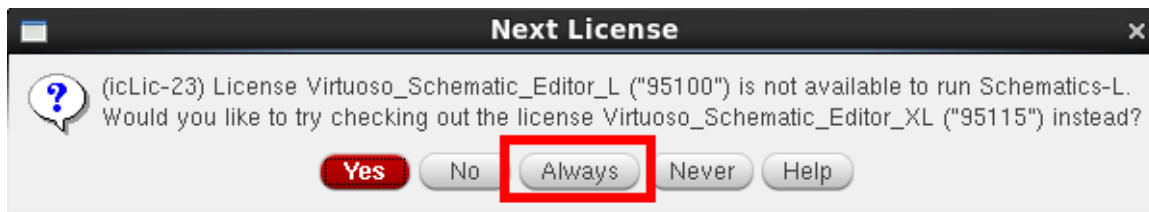


## Creating Design Charts

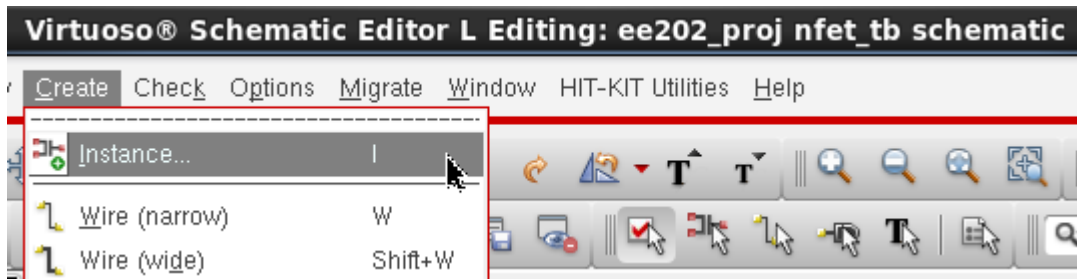
Create new schematic.



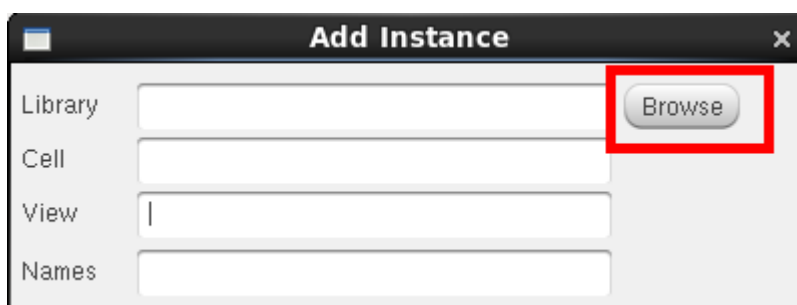
If you receive messages like the below just click always.



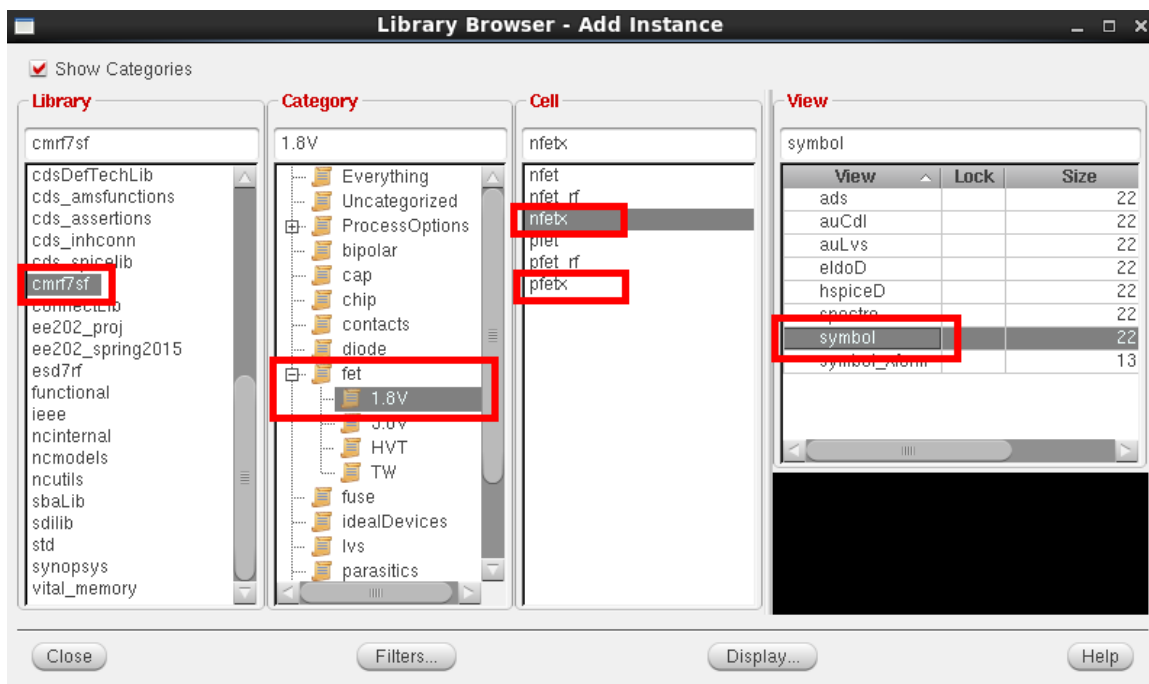
Add instance (you can use the hot-key 'i').



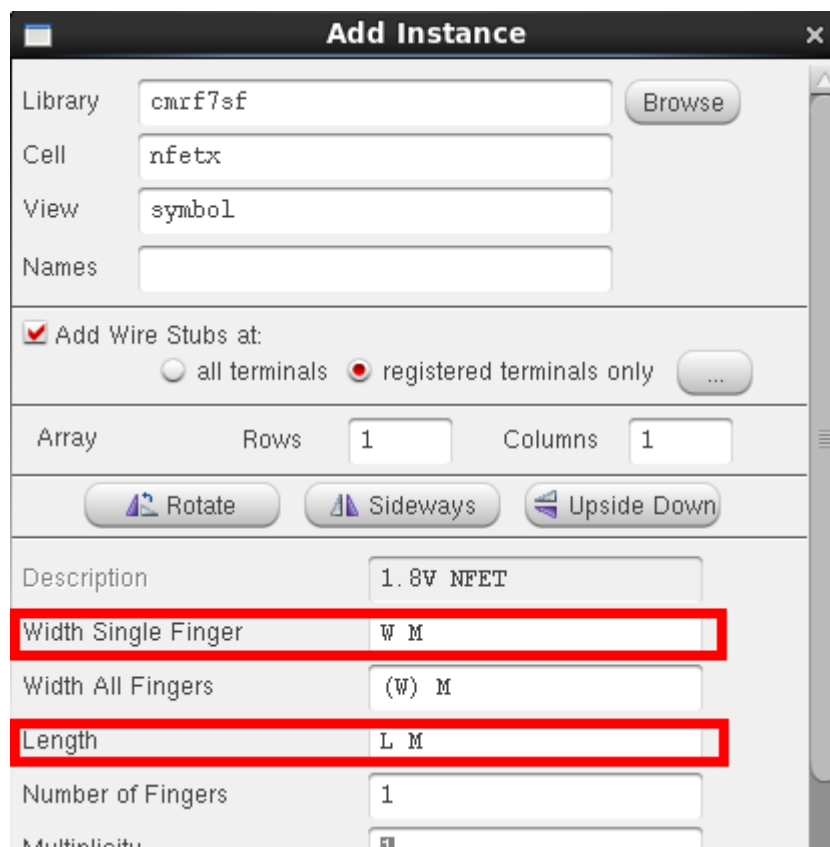
Browse.



The design kit components are in "cmrf7sf" library. These are the FETs that you will be always using.

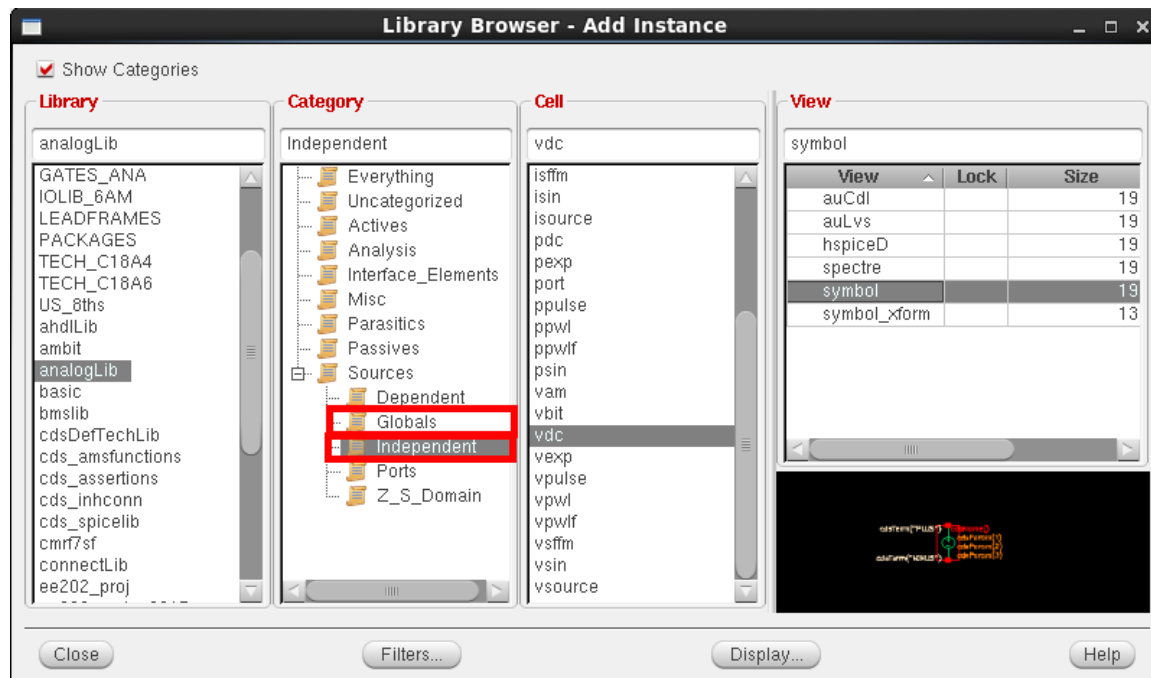


You can make properties as variables.



Sources, resistors, capacitors, gnd, etc., are in analogLib library.

Add DC voltage source.



**Edit Object Properties**

Apply To: only current instance

Show: ☐ system ☒ user ☒ CDF

Browse Reset Instance Labels Display

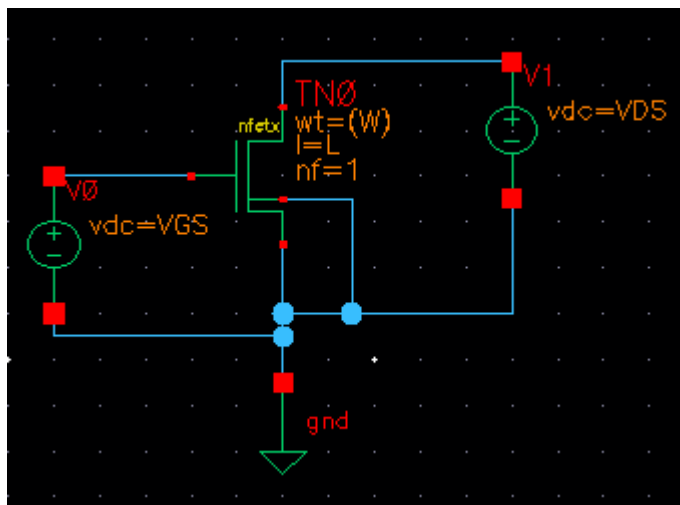
Property	Value	Display
Library Name	analogLib	off
Cell Name	vdc	off
View Name	symbol	off
Instance Name	V0	off

Add Delete Modify

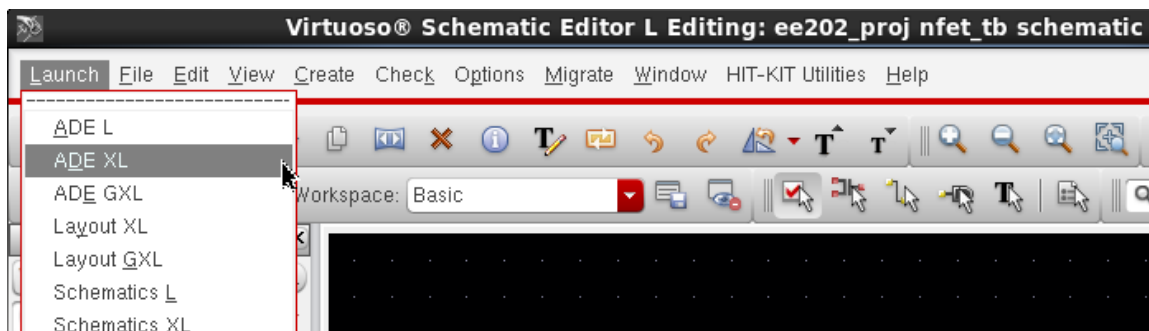
User Property	Master Value	Local Value	Display
Ivsignore	TRUE		off

CDF Parameter	Value	Display
Noise file name		off
Number of noise/freq pairs	0	off
DC voltage	VGS	off
AC magnitude		off
AC phase		off

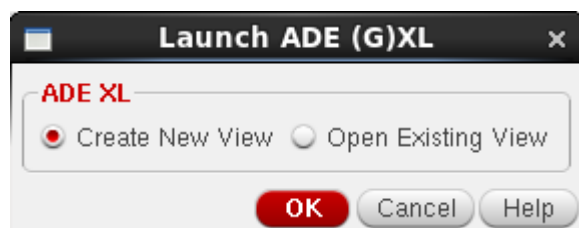
The schematic should look like this.



Launch adexl to start simulating.



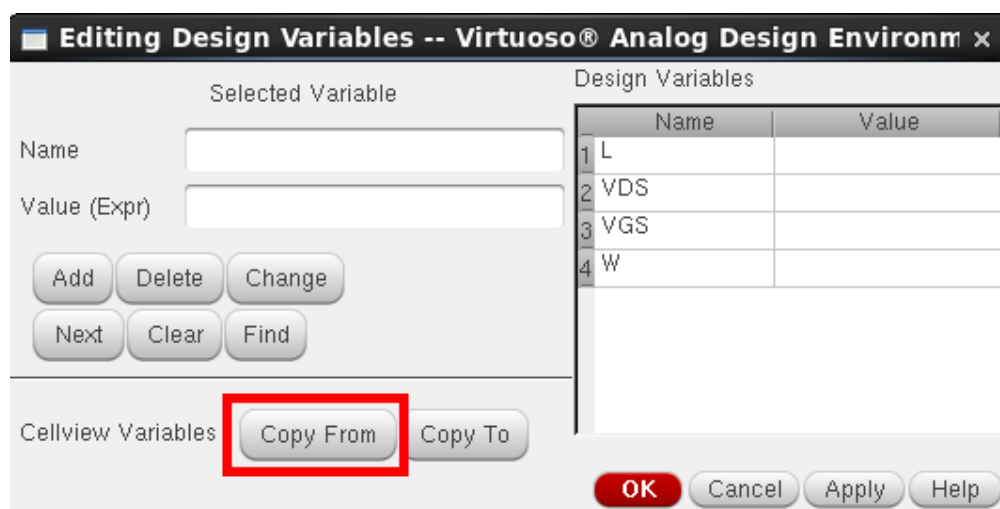
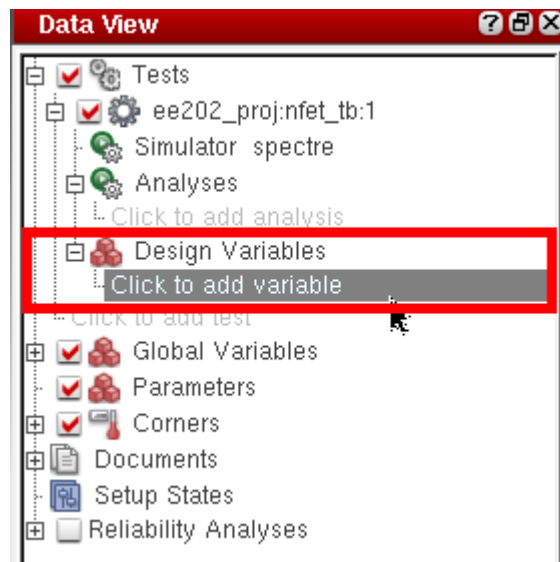
For the first time you create new view. After that you open the existing view.



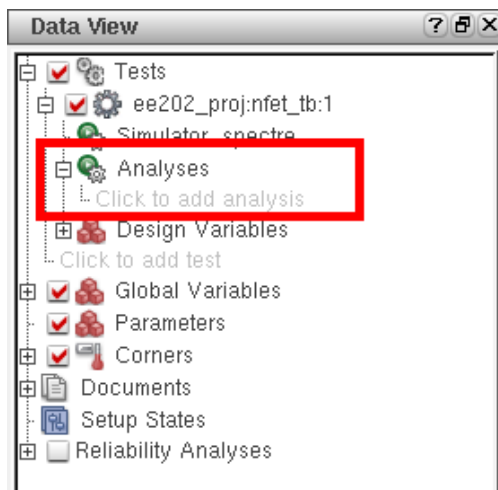
Add test.



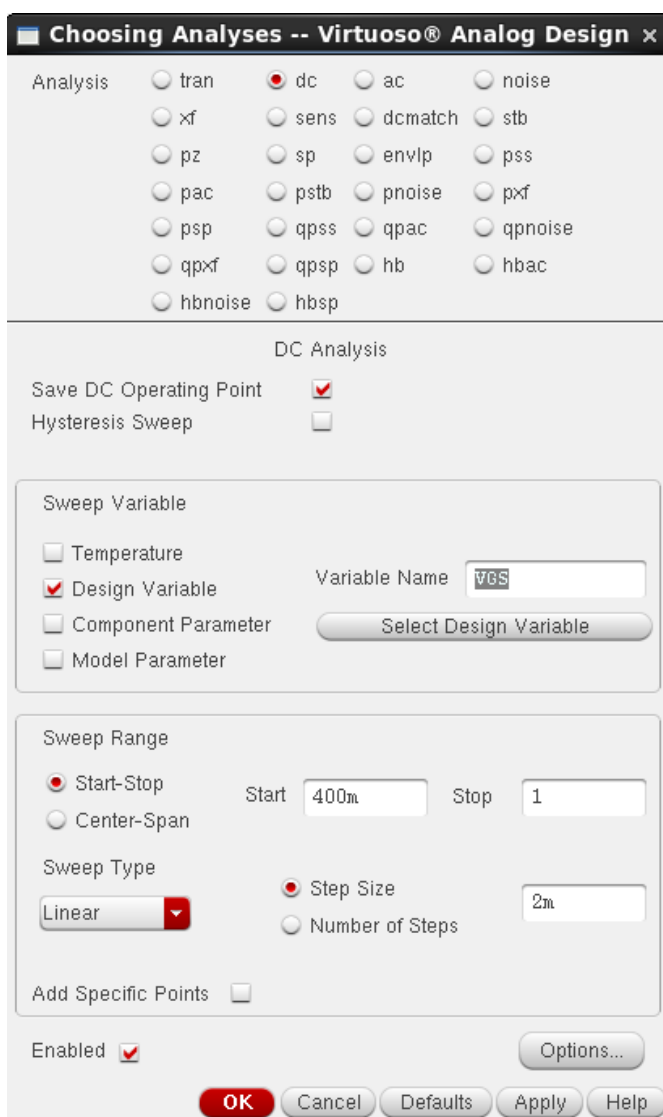
Copy variables from the schematic.



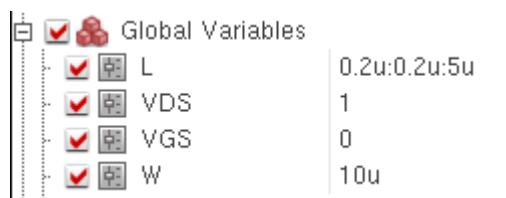
Add analysis.



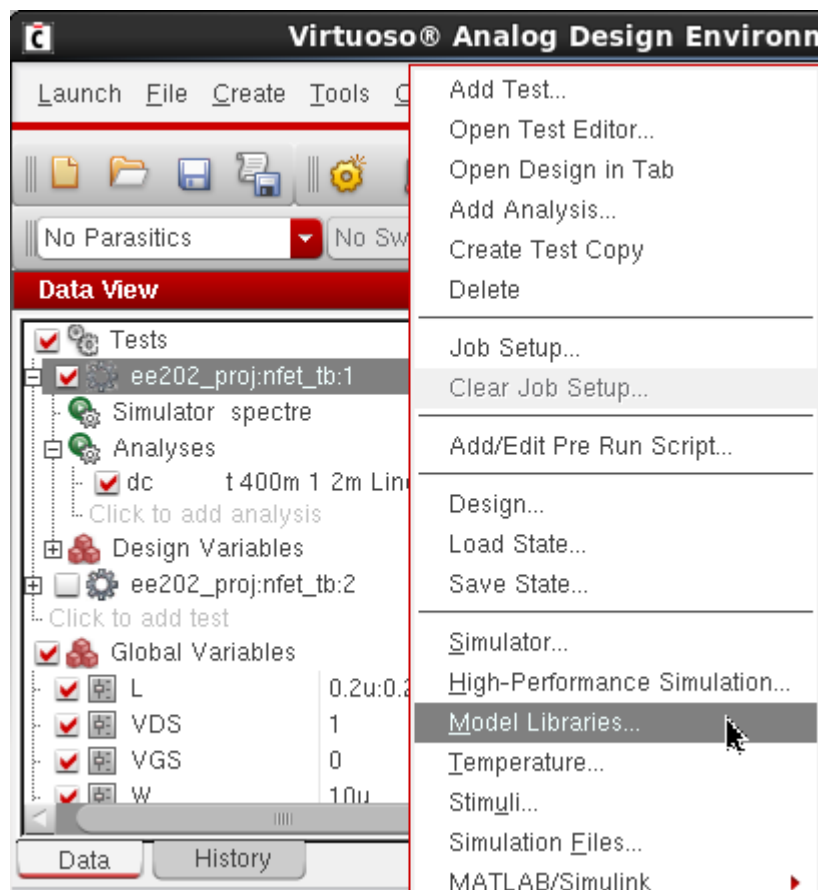
Setup DC analysis as below.



Setup the global variables as below.



You need to tell Cadence that you want to save some transistor parameters (gm, gds, etc.) So you need to add a text file to your model libraries.

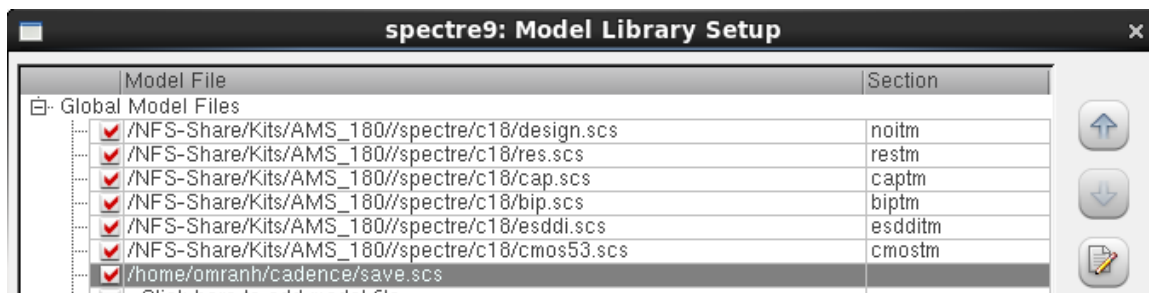


Create a text file, add the lines below in it, and then add it to model libraries as below.

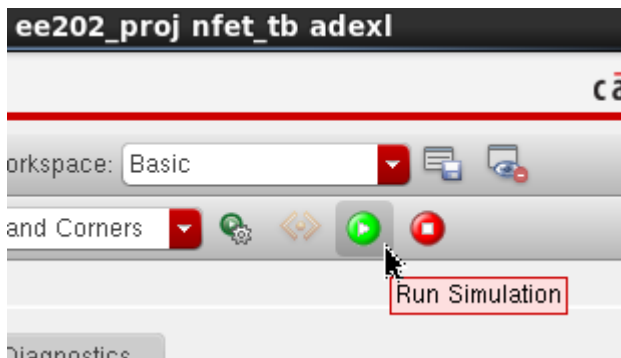
```
save *:gm sigtype=dev
save *:gds sigtype=dev
save *:id sigtype=dev
save *:cgg sigtype=dev
save *:cgs sigtype=dev
save *:gmoverid sigtype=dev
save *:vgs sigtype=dev
save *:vth sigtype=dev
```



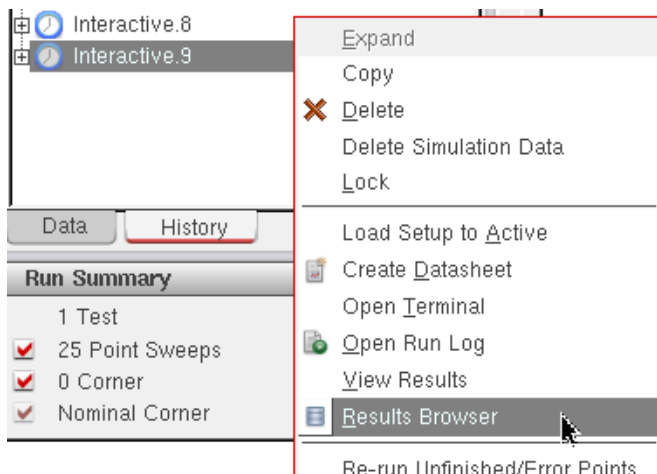
save \*.vdsat sigtype=dev



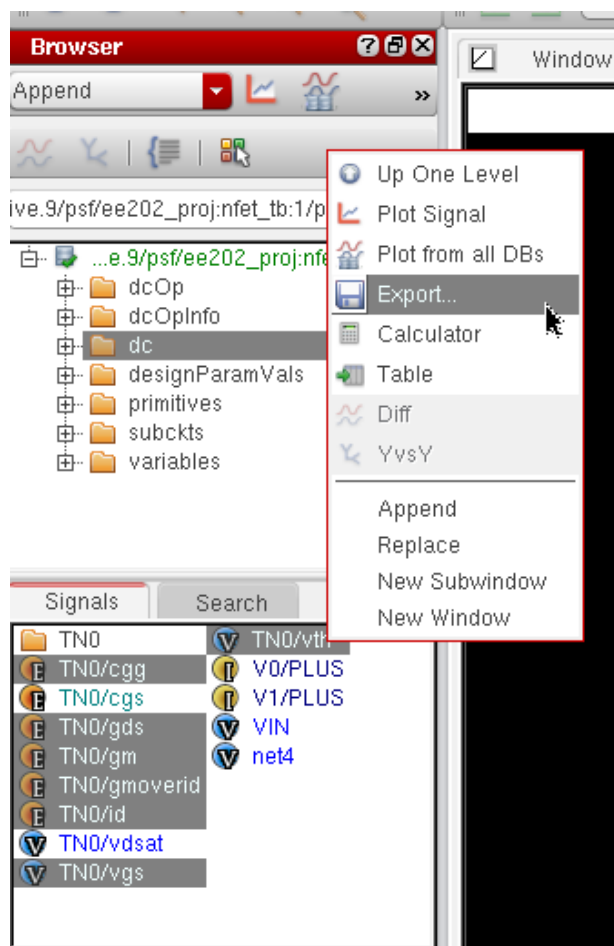
Run your simulation.



Browse the results.



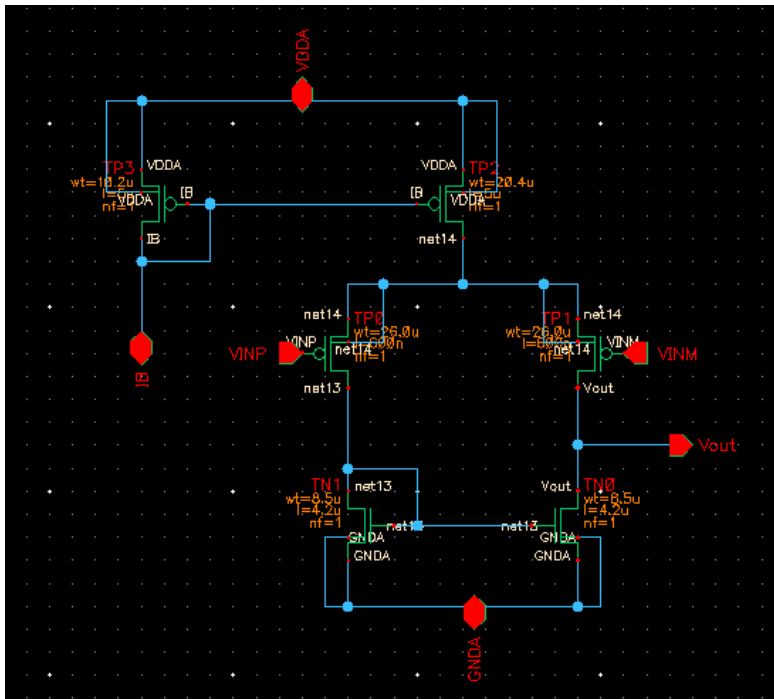
Export parameters to csv file.



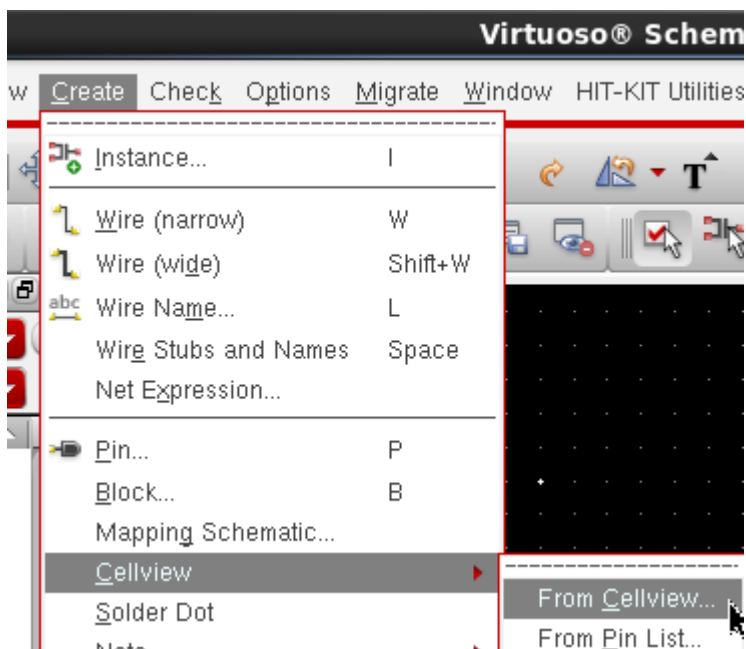
Use MATLAB to plot the design curves. A sample MATLAB code is available on Canvas.

## Part 2: OTA Simulation

Draw the schematic as below. Create pins for your design.



Create symbol view to use it in the test bench.



**Symbol Generation Options**

Library Name: ee202\_proj      Cell Name: ota\_simple      View Name: symbol

Pin Specifications      Attributes

Left Pins: VINM VINP     

Right Pins: Vout     

Top Pins: VDDA     

Bottom Pins: IB GND4 |     

Exclude Inherited Connection Pins:

☒ None    ☐ All    ☐ Only these:

Load/Save ☒    Edit Attributes ☐    Edit Labels ☐    Edit Properties ☐

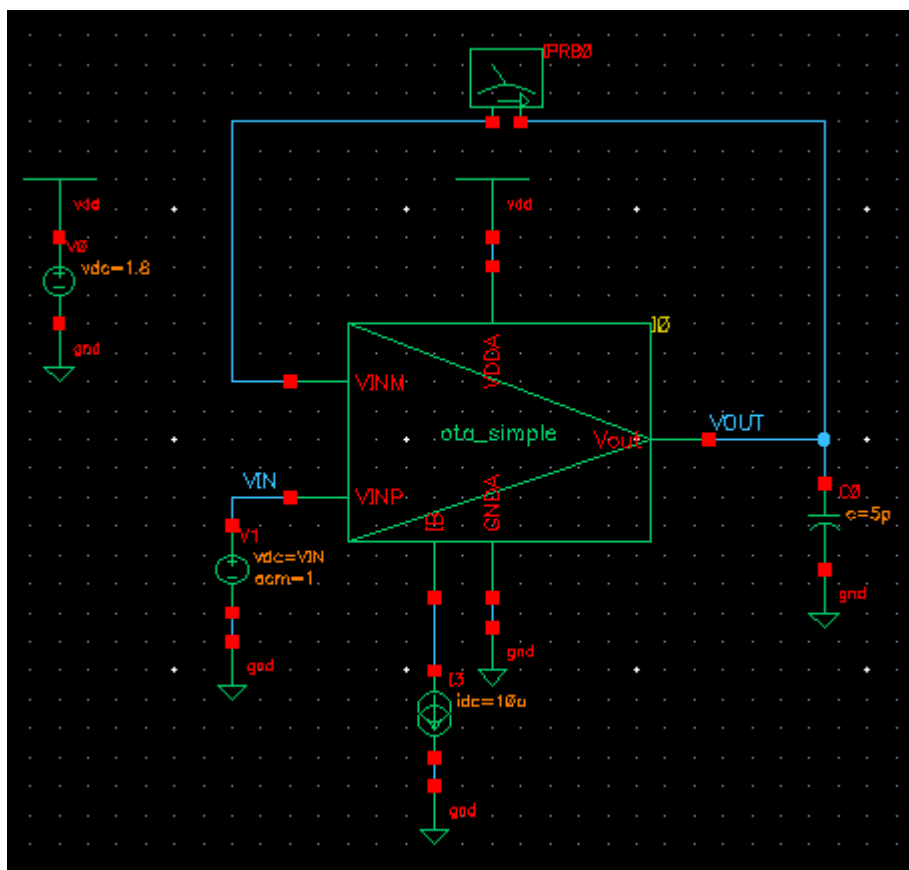
Load/Save Symbol Template Configuration

analog  /IC615/tools.lnx86/dfII/samples/symbolGen/artist.tsg

TSG File

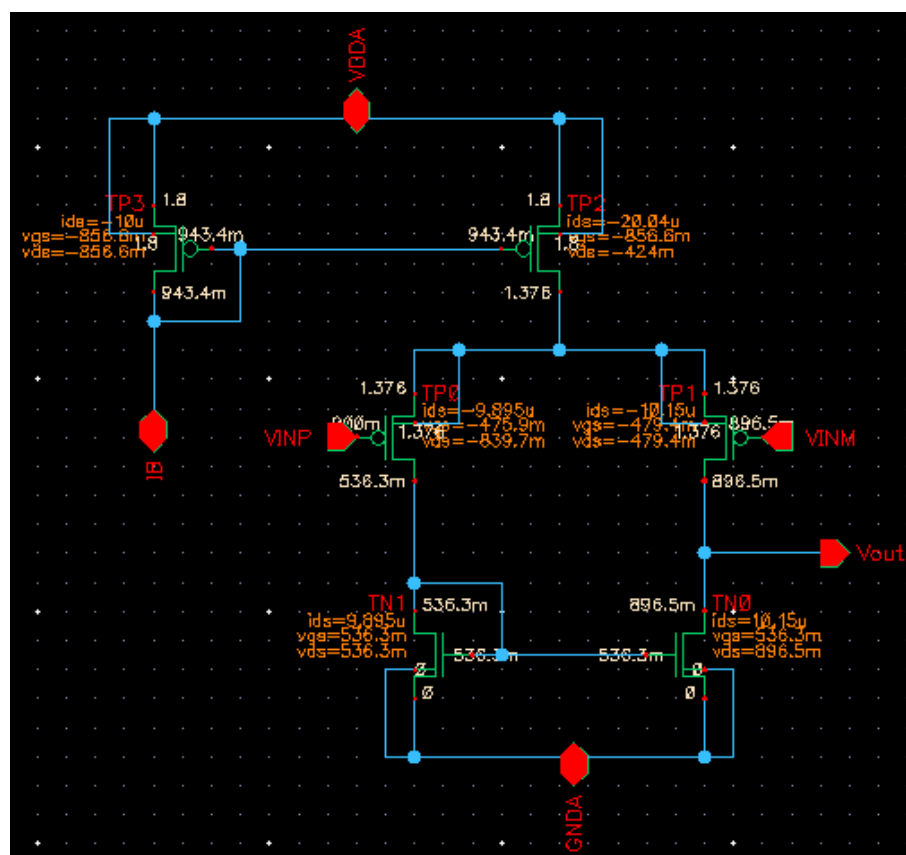
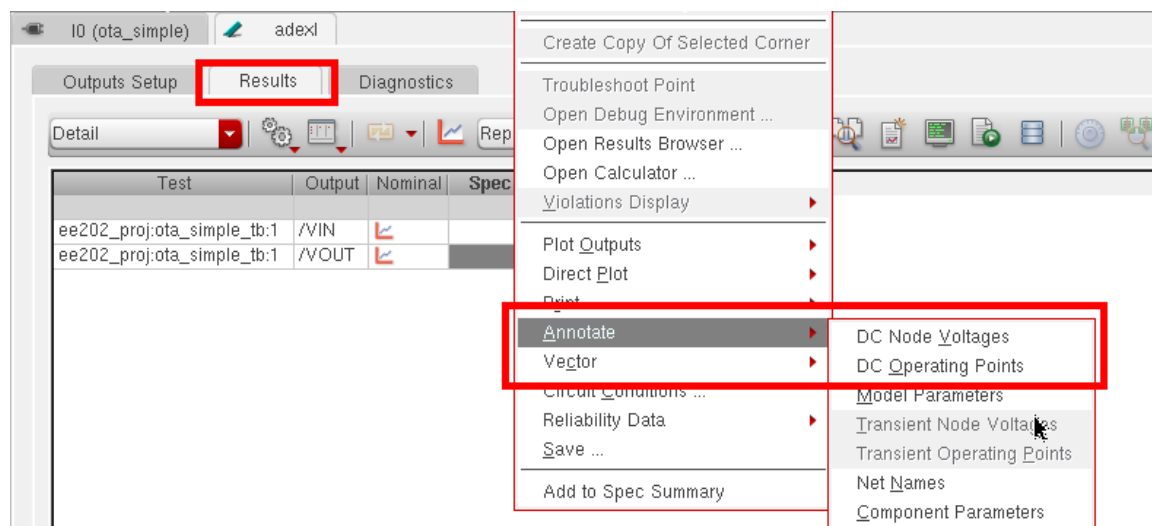
Library Name:       Cell Name:       View Name: symbol

Create new schematic for the test bench. Draw the test bench as below.



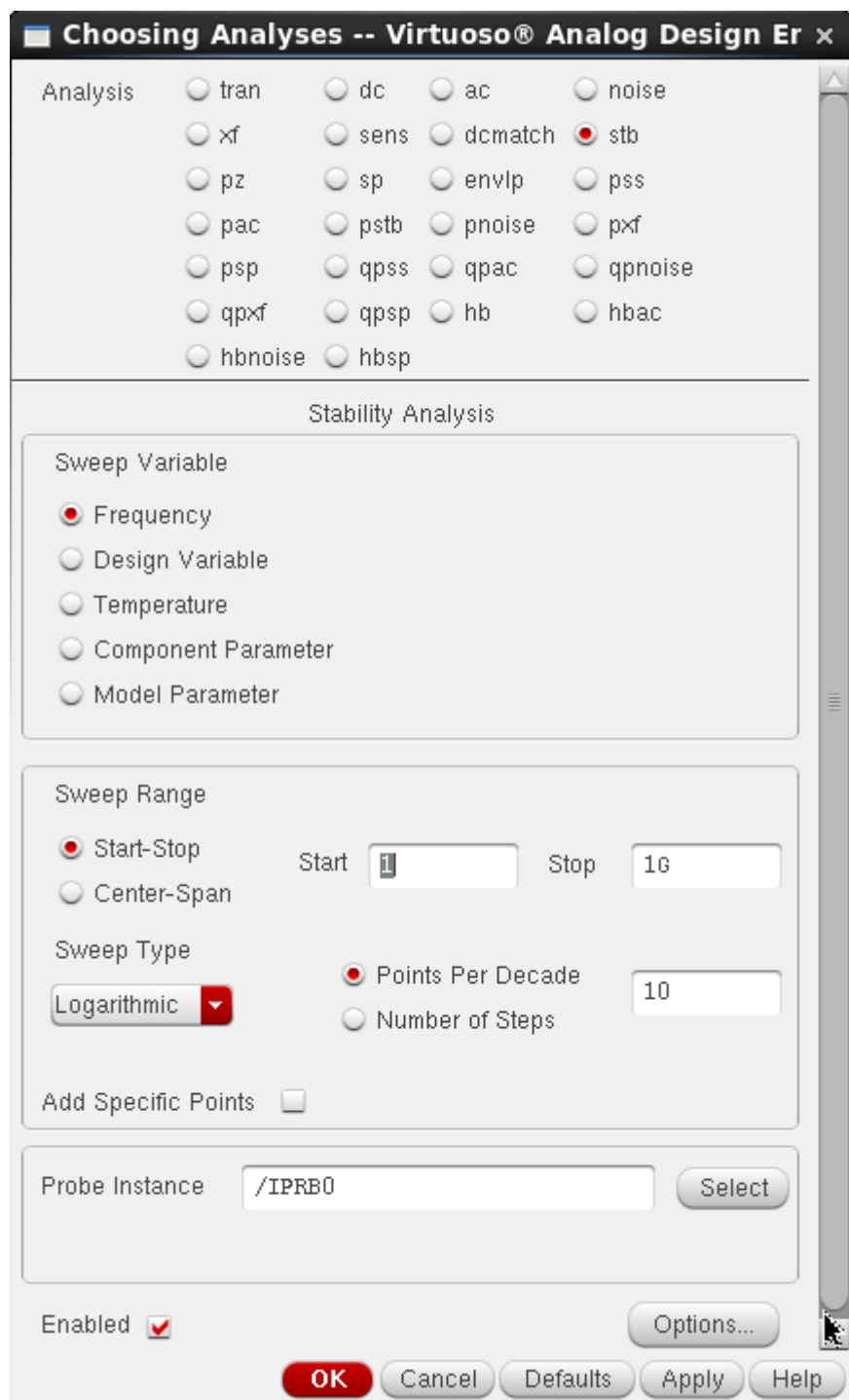
DC Simulation

Run DC simulation. You can annotate DC simulation results on the schematic.

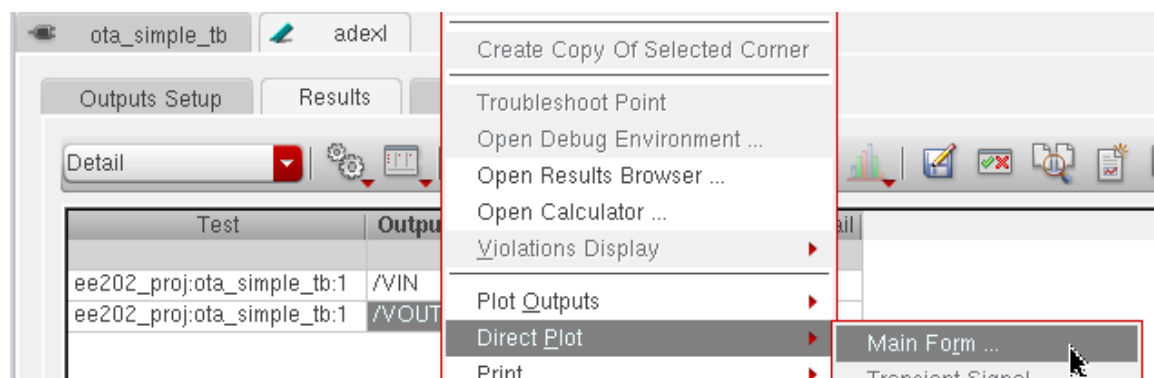


STB Simulation

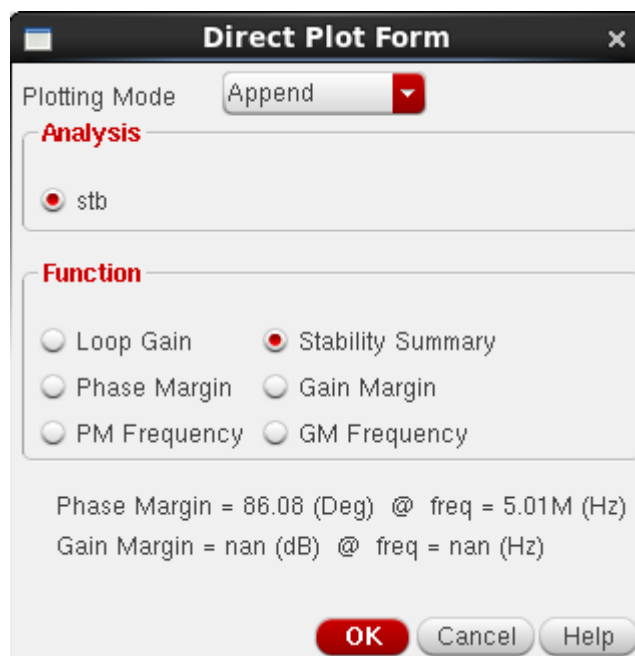
Run STB simulation to find loop gain and phase margin.



Use "Main Form" to plot STB simulation results.

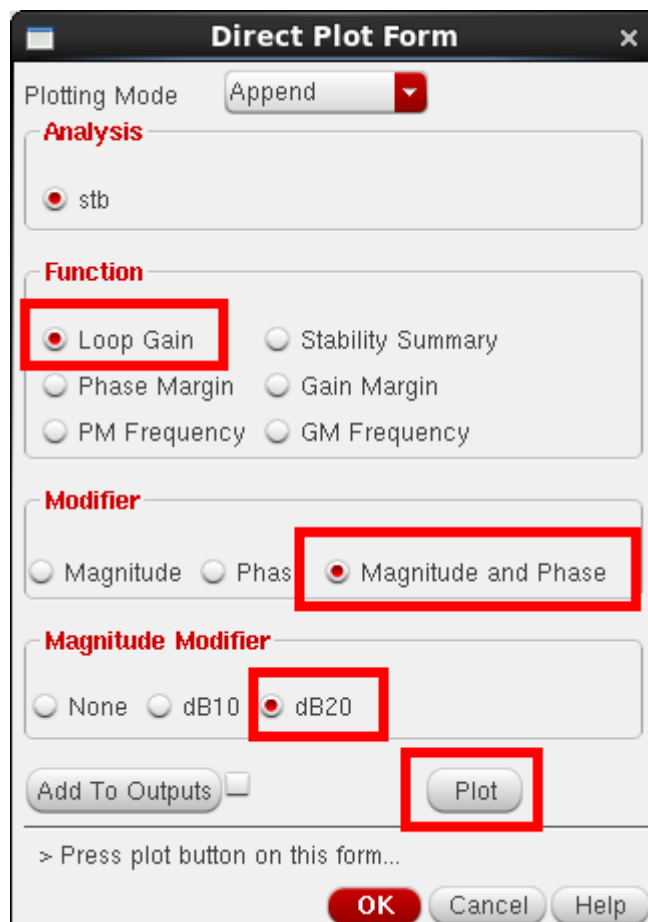


All important results can be plotted from here.



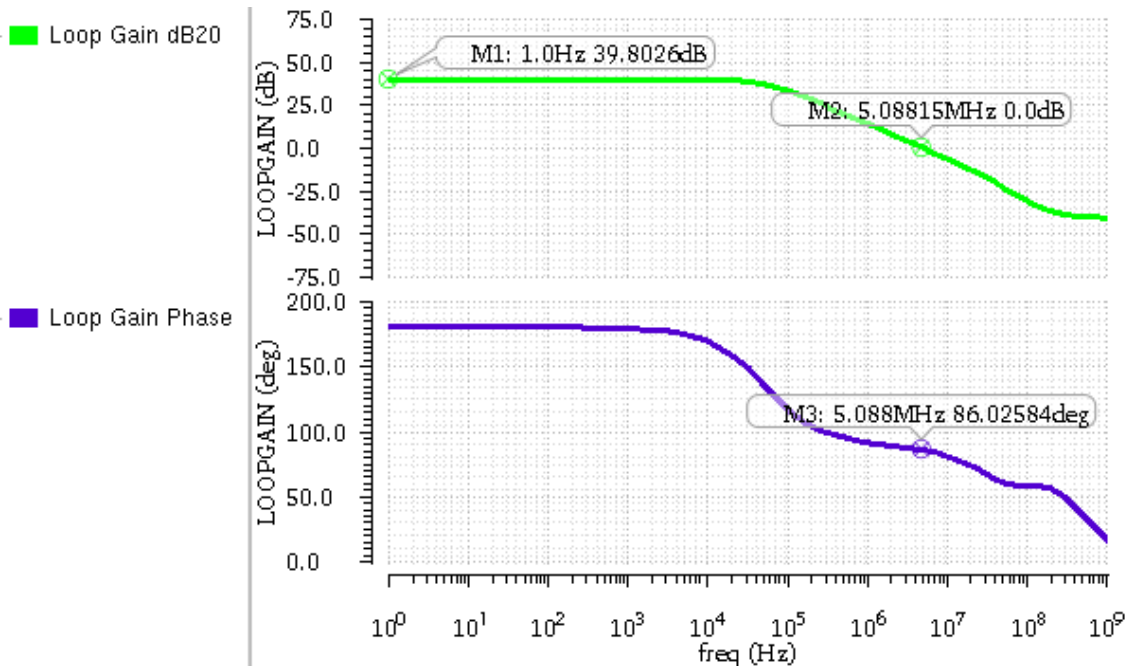
The image shows a 'Direct Plot Form' dialog box. At the top, there is a 'Plotting Mode' dropdown menu set to 'Append'. Below this, the 'Analysis' section has a radio button selected for 'stb'. The 'Function' section contains several radio buttons: 'Loop Gain', 'Stability Summary' (selected), 'Phase Margin', 'Gain Margin', 'PM Frequency', and 'GM Frequency'. Below the function buttons, the following text is displayed: 'Phase Margin = 86.08 (Deg) @ freq = 5.01M (Hz)' and 'Gain Margin = nan (dB) @ freq = nan (Hz)'. At the bottom, there are three buttons: 'OK' (highlighted in red), 'Cancel', and 'Help'.

You can plot gain and phase as below.

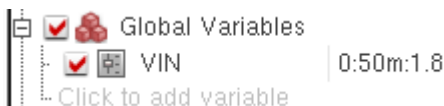


The image shows the 'Direct Plot Form' dialog box with additional sections highlighted by red boxes. In the 'Function' section, the 'Loop Gain' radio button is selected and boxed. In the 'Modifier' section, the 'Magnitude and Phase' radio button is selected and boxed. In the 'Magnitude Modifier' section, the 'dB20' radio button is selected and boxed. At the bottom, the 'Plot' button is boxed. Below the 'Plot' button, there is a text prompt: '> Press plot button on this form...'. At the very bottom, there are three buttons: 'OK' (highlighted in red), 'Cancel', and 'Help'.

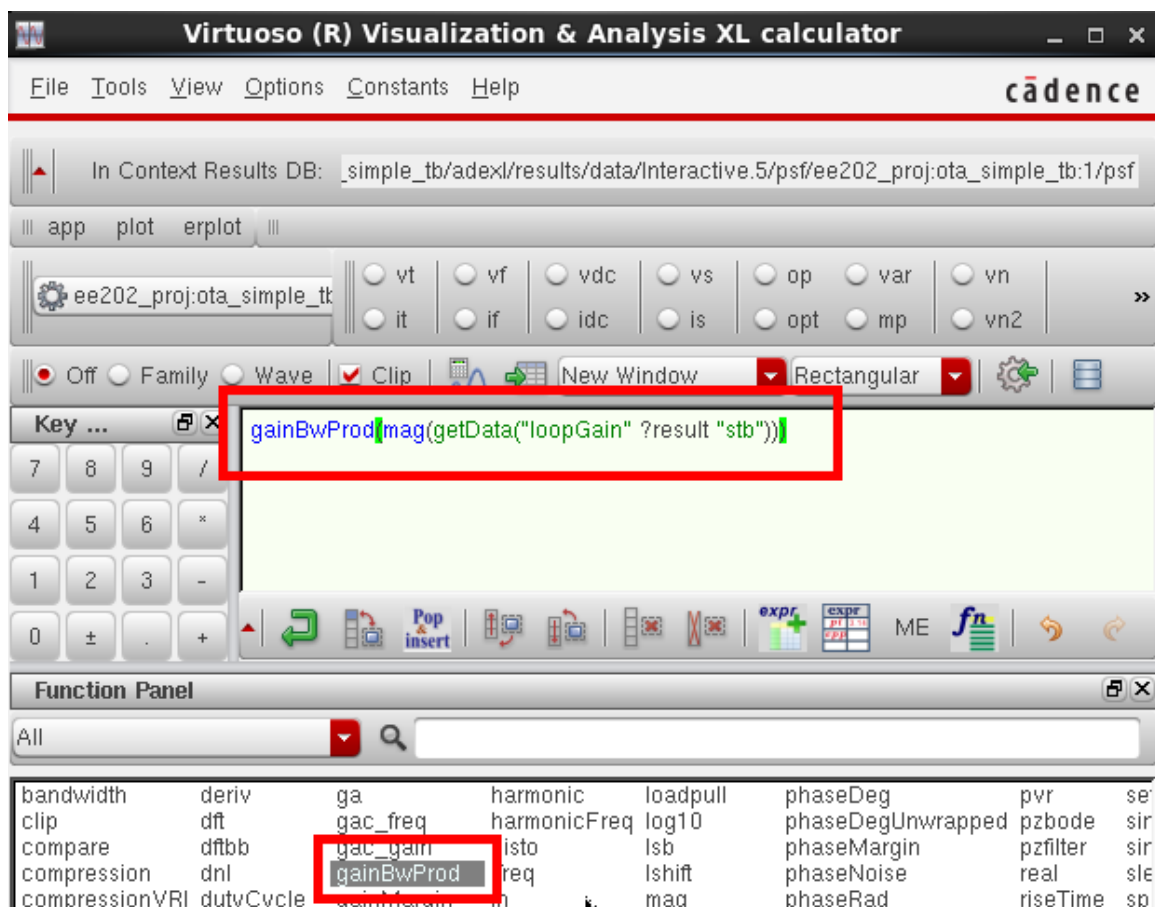
Make sure you annotate the plots using markers (m). Invert the colors in any image editor to avoid black background in printing.



To get input range you need to sweep VIN.



You can plot GBW vs VIN using calculator. You can also use the “Main Form” as before.





**Direct Plot Form**

Plotting Mode: New Win

**Analysis**

☒ stb

**Function**

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

Add To Outputs ☐ **Plot**

> Press plot button on this form...

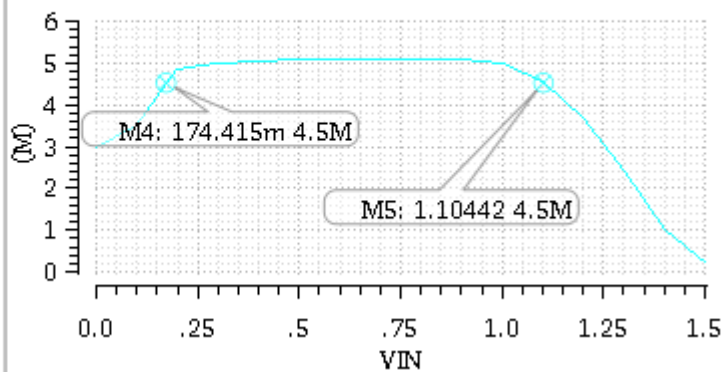
OK Cancel Help

This plot shows GBW vs VIN.

```
gainBwProd(mag(getData("loopGain" ?result "stb")))
```

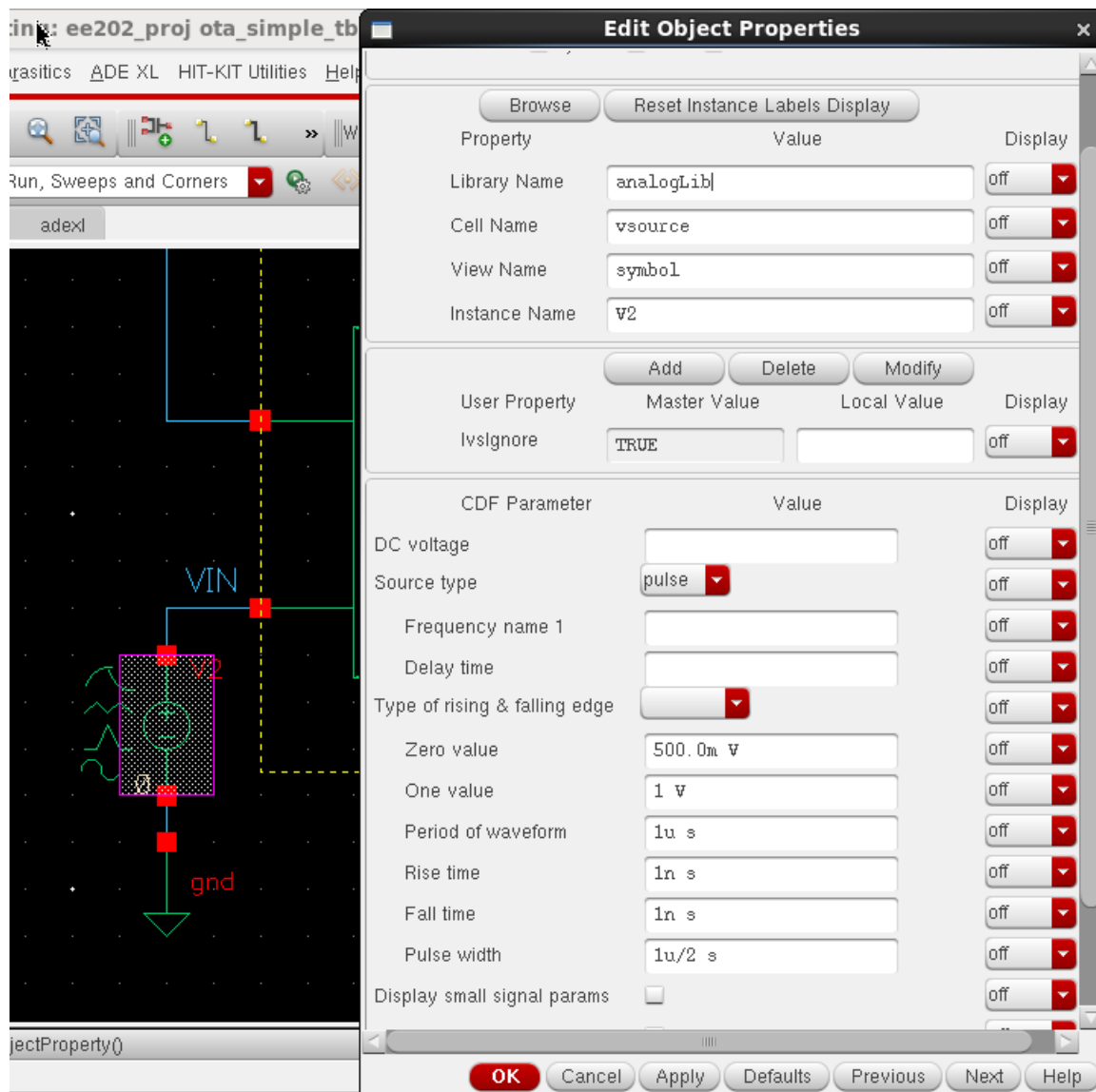
Name

...ain" ?result "st

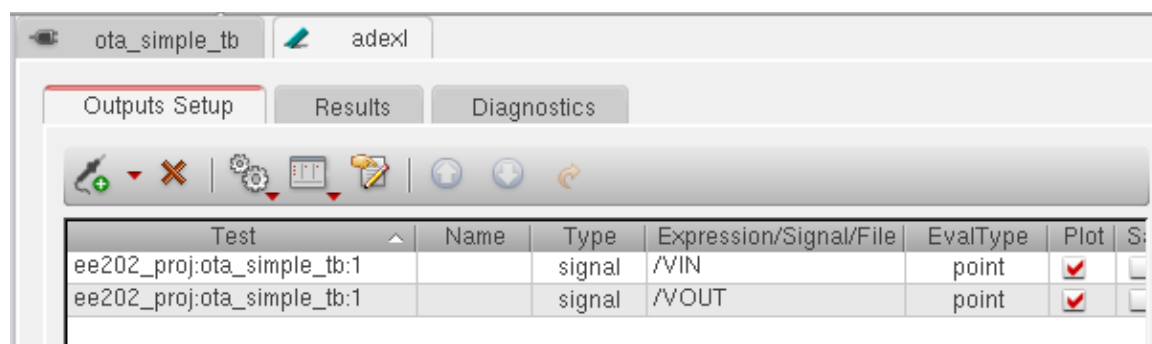


**Transient Simulation**

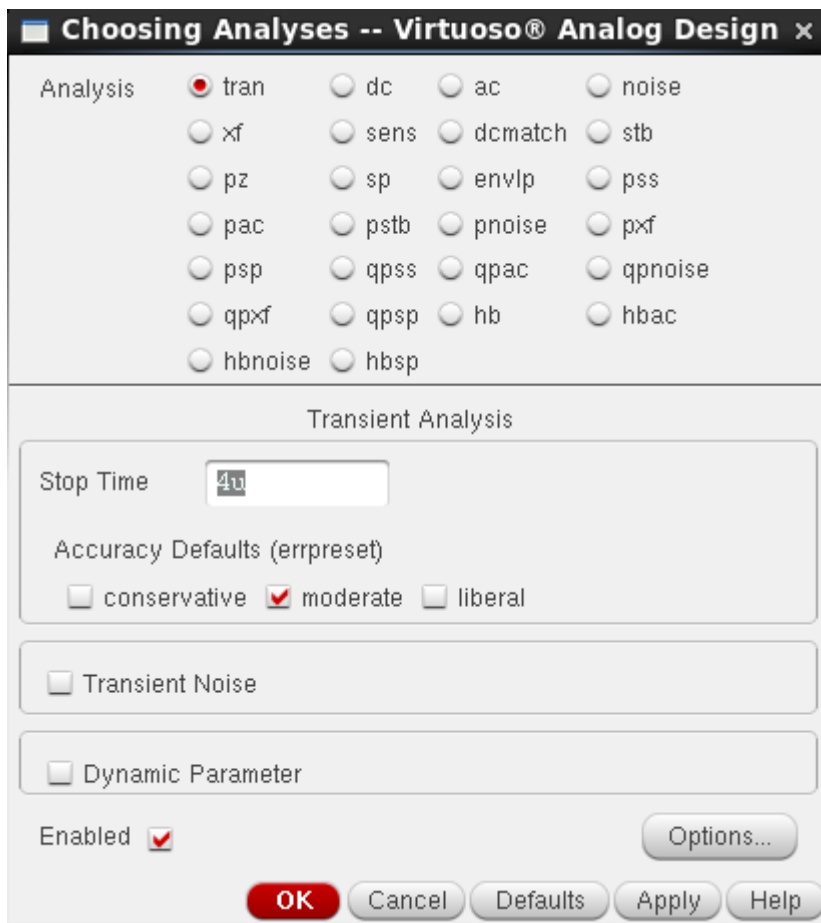
Add transient source as below.



Select the signals to be plotted.



Setup tran simulation.



Run the simulation and plot the results. Use (a,b) marker to calculate the slew rate.

