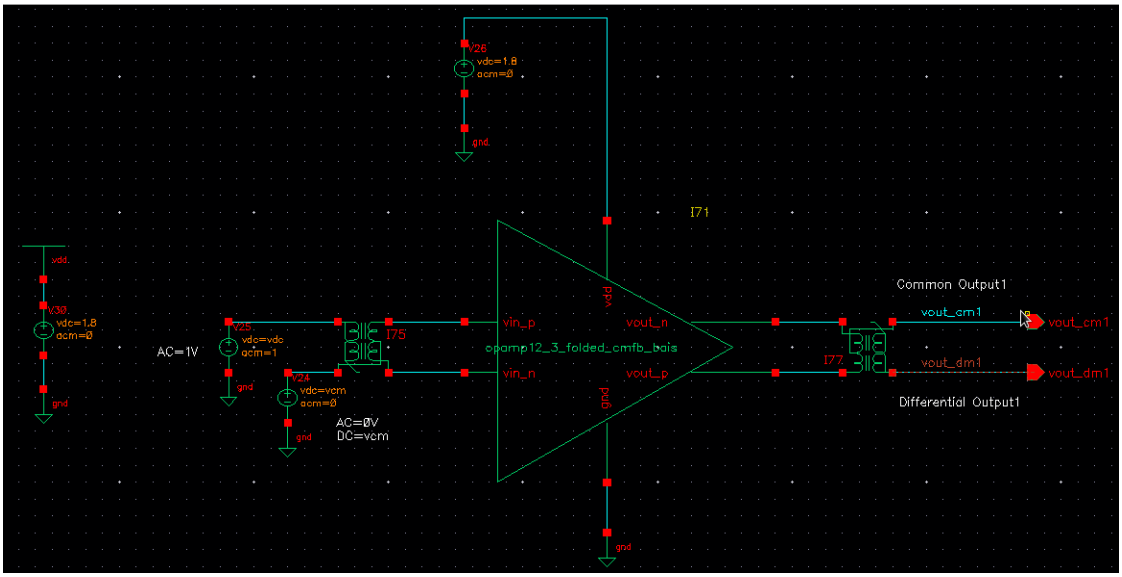
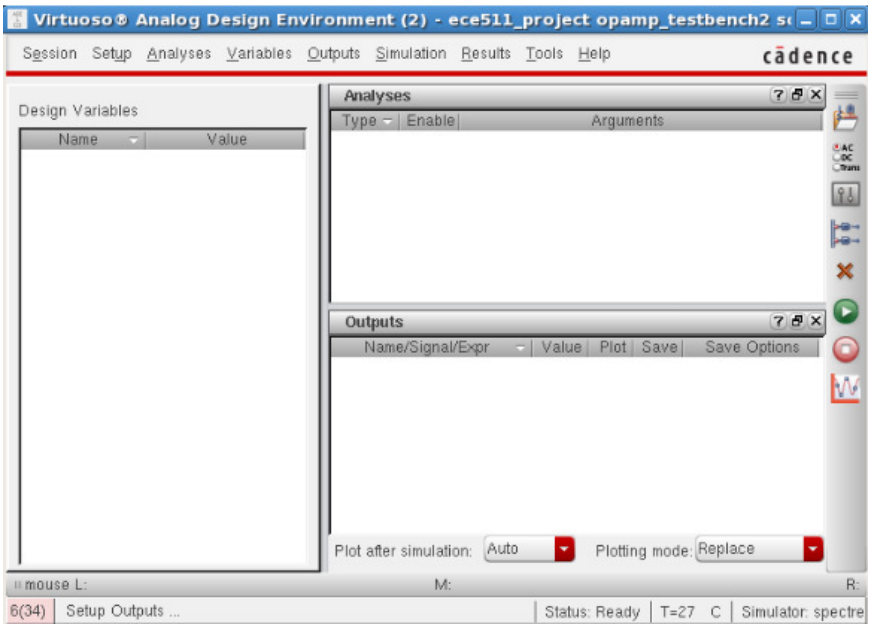
1. Open loop gain, Phase Margin, Unity Gain Frequency Simulation

- Testbench schematic: Differential Input AC =1V ; Common Input AC = 0V, DC = vcm

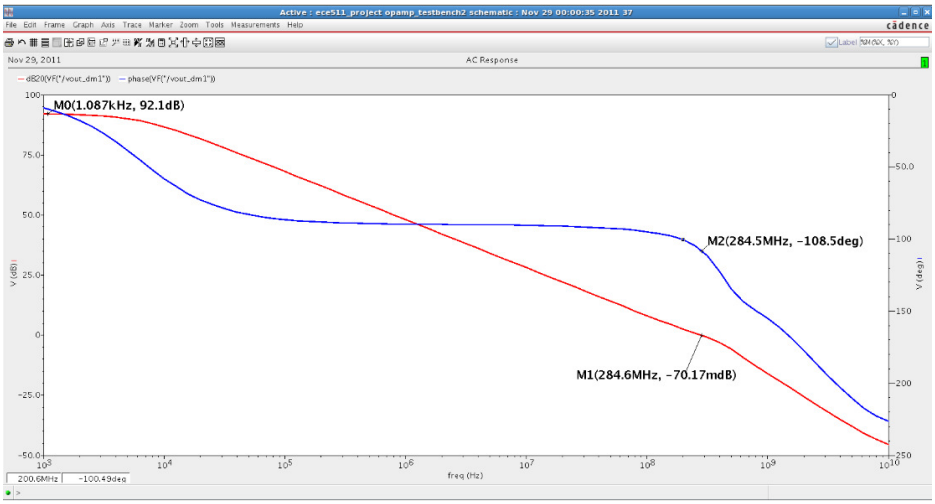


- Analog Design Environment



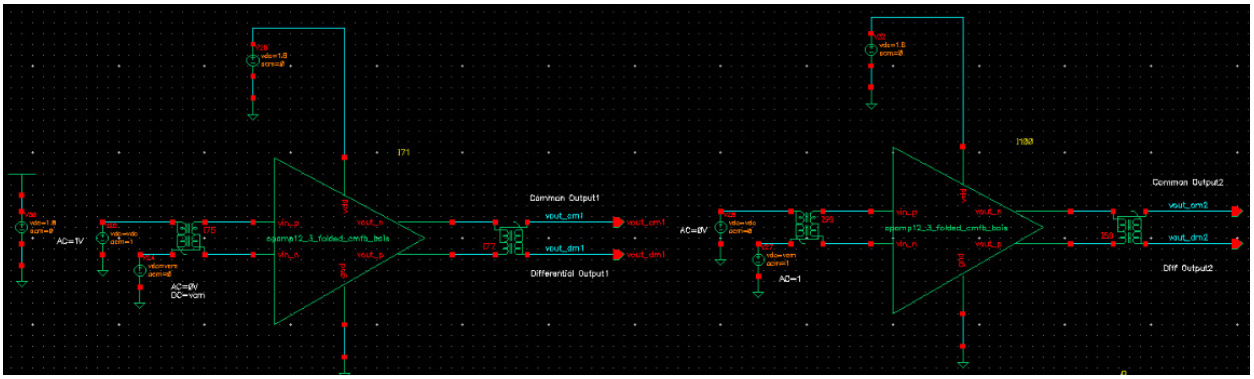
- Analyses->Choose->then set up the AC simulation as shown below

- Run AC simulation
- Choose Plotting Mode as “Append”
- Result -> Direct Plot -> AC dB20 and AC Phase -> Click Differential Output1-> Esc (Blue line is the gain and Red line is the phase)
- Session -> Save State (Please save the graphs for the project verification)

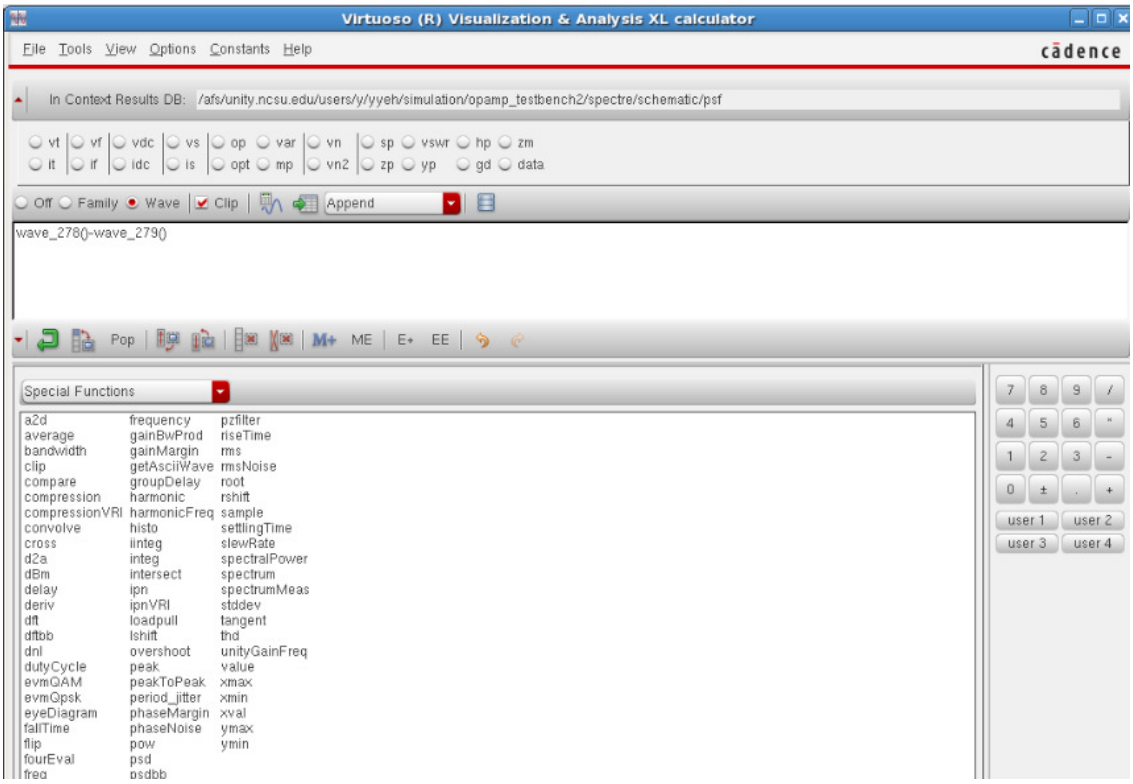


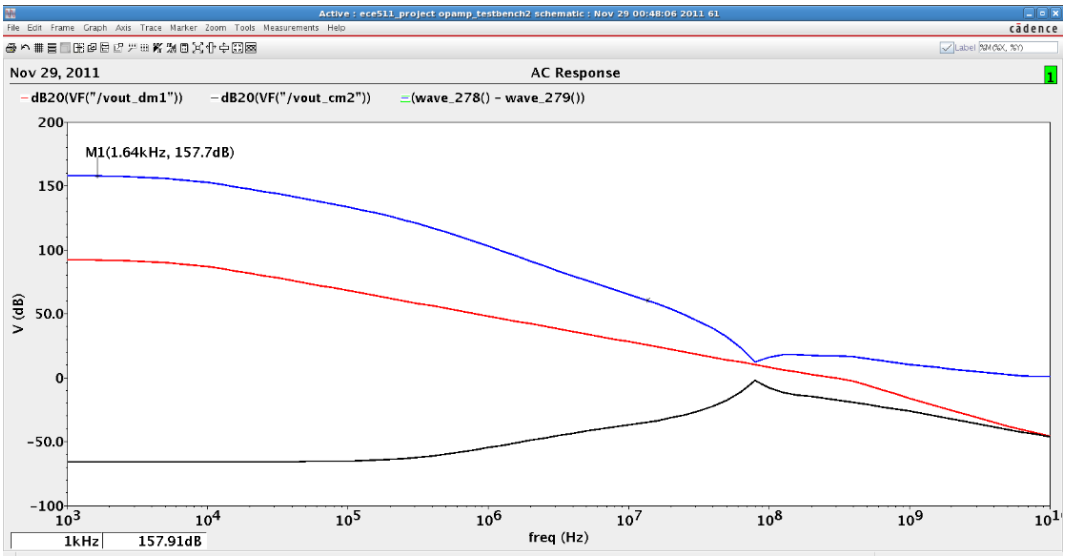
2. CMRR Simulation

- Testbench Schematic
  Left Opamp is for Diff gain simulation => Diff Input1 AC =1V; CM Input1 AC = 0V;
  Right Opamp is for Common gain simulation => Diff Input2 AC =0V ; CM Input2 AC = 1V;



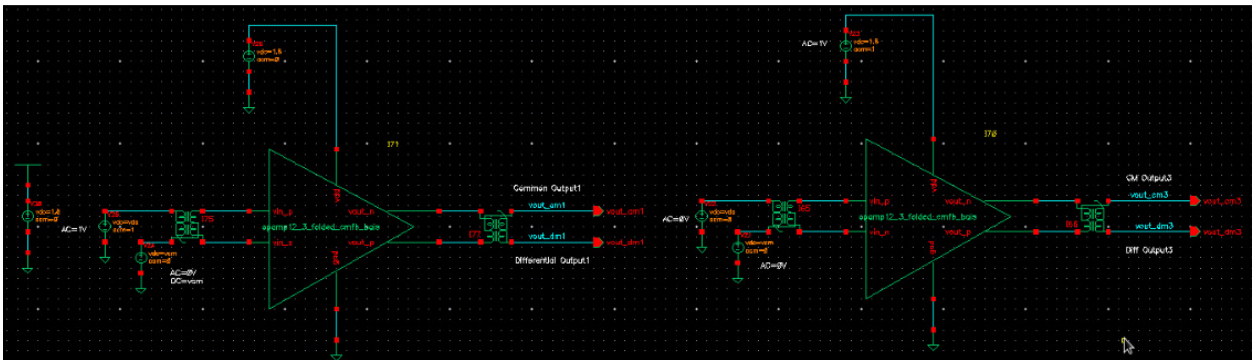
- Run AC Simulation
- Result -> Direct Plot -> AC dB20 -> Click Differential Output1 for Diff Gain and Common Output2 for CM Gain -> Esc
- Tool -> Calculator -> Click waveform -> CMRR = Diff Gain – CM Gain
  (Blue line is CMRR; Red line is Diff Gain; Black line is CM Gain)
- Session -> Save State



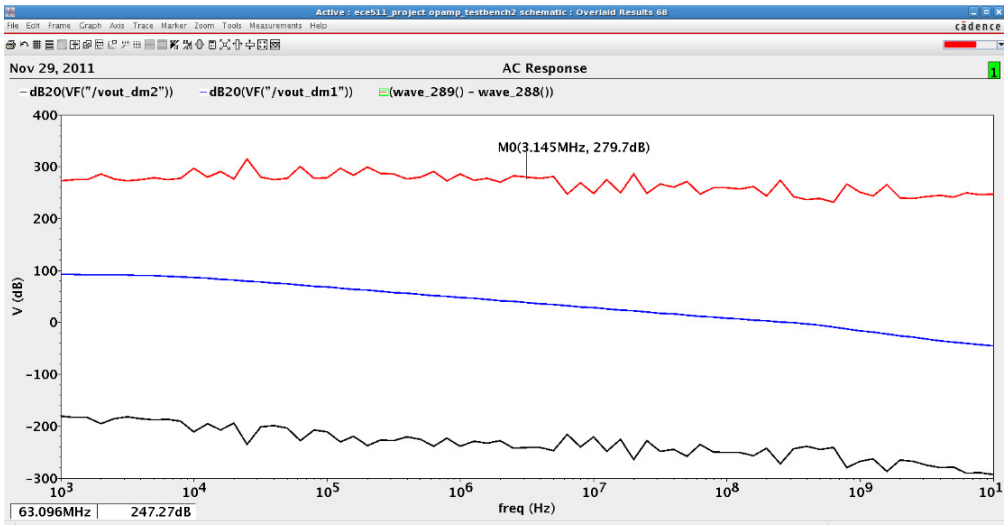


3. PSRR Simulation:

- Testbench Schematic
  Left Opamp is for Diff gain simulation => Diff Input1 AC =1V; CM Input1 AC = 0V; VDD AC=0V;
  Right Opamp is for Vdd gain simulation => Diff Input3 AC =0V ; CM Input3 AC = 0V; VDD AC=1V;

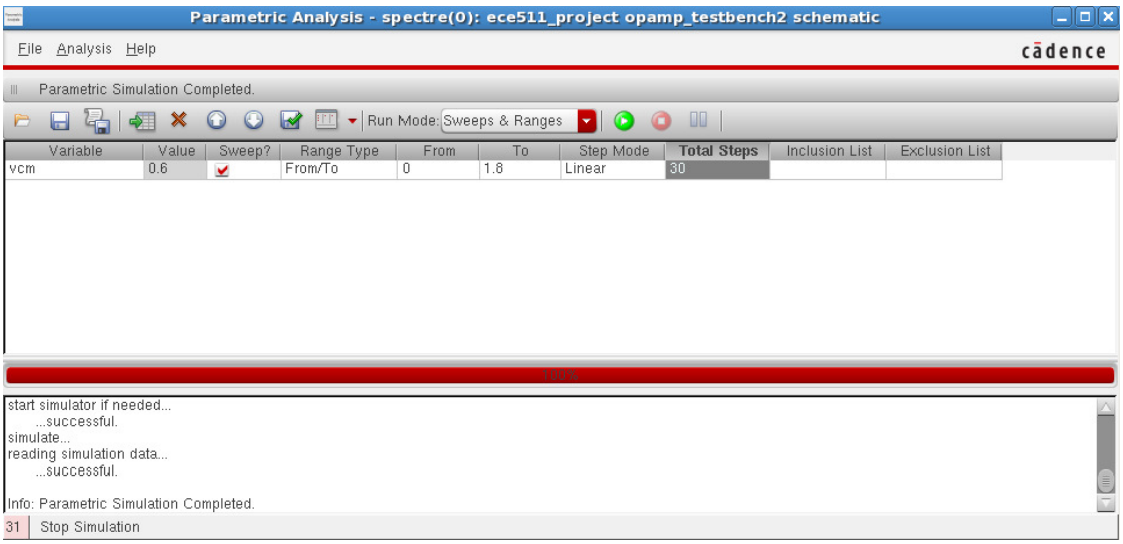


- Run AC Simulation
- Result -> Direct Plot -> AC dB20 -> Click Diff Output1 for Diff Gain and Diff Output3 for Vdd Gain -> Esc
- Tool -> Calculator -> Click waveform -> PSRR+ = Diff Gain – Vdd Gain
  (Red line is PSRR+; Blue line is Diff Gain; Black line is Vdd gain)
- Session -> Save State

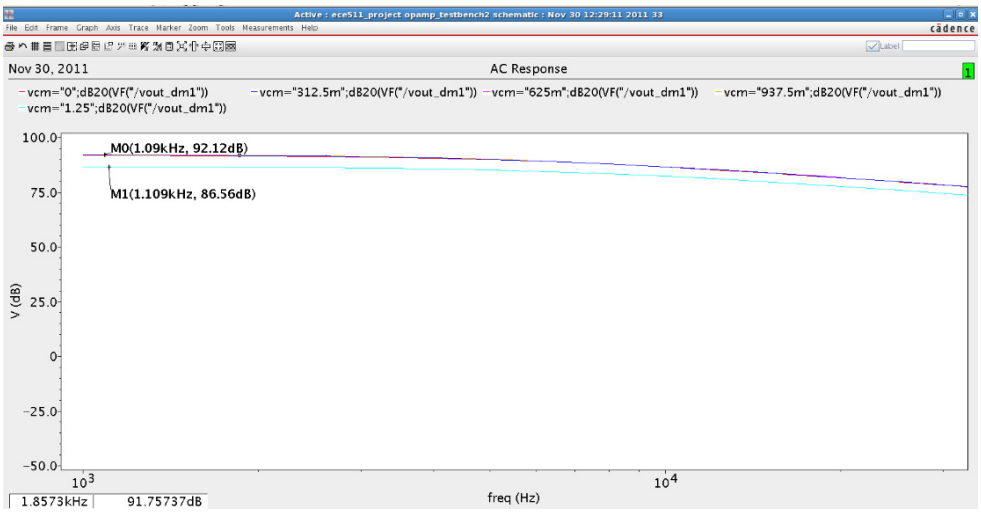


4.Input Common Mode Range:

- -Please give a DC voltage variable (“vcm”) in your common input voltage source.
- Setup AC analysis and run AC simulation -> Please also give AC = 1V to the differential input.
- -Tool -> Parametric Analysis -> choose the variable “vcm” as shown below

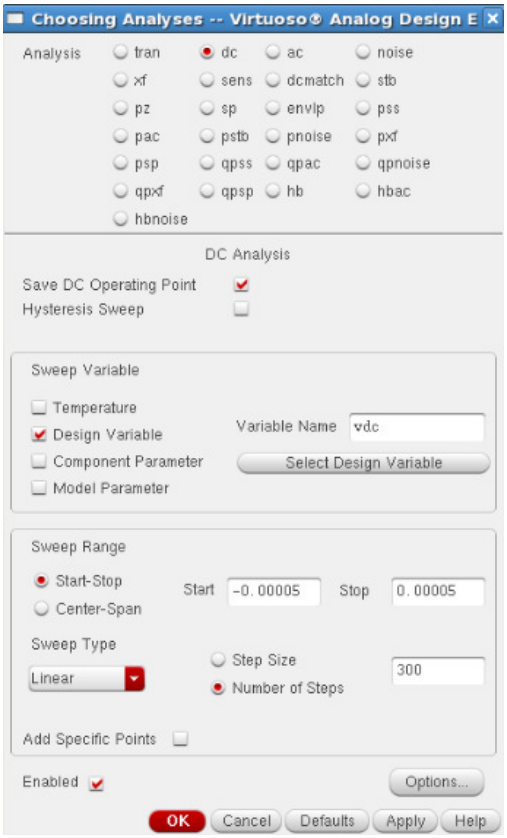


- Result -> Direct Plot -> AC dB20 -> Click Diff Output1 for Diff Gain v.s. vcm-> Esc
- Find the input common range where the low frequency gain is within +- 6dB of the nominal value
- Once you know some input common voltages are out of the range of +-6dB of your low frequency gain, you can narrow the vcm range and simulate it again until the vcm range is with the range of +- 6dB of the gain.
- Session -> Save State

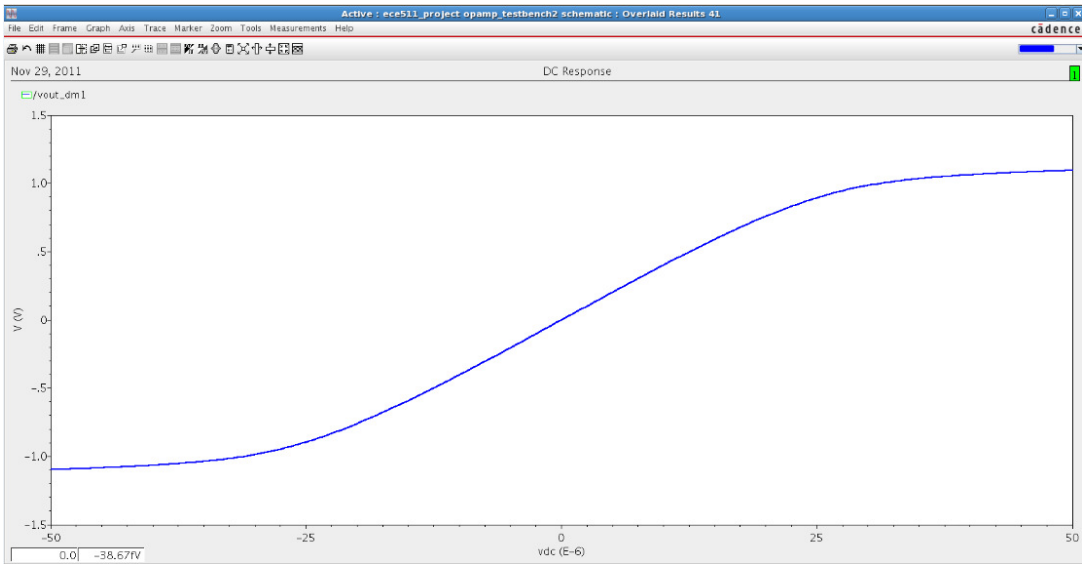


5.Output Swing Range:

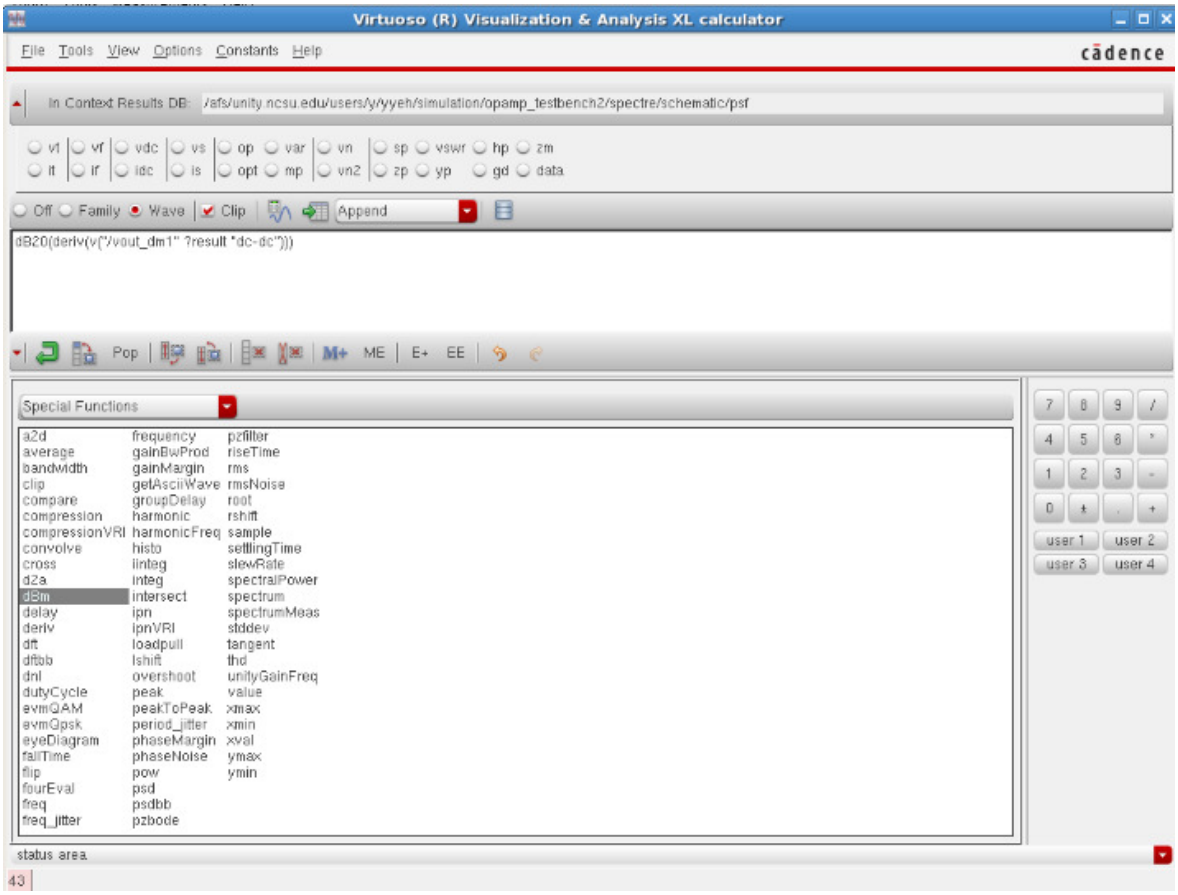
- Please give a DC voltage variable (“vdc”) in your differential input voltage source.
- Analyses->Choose->then set up the DC simulation as shown below.
- The numbers below are just for reference so you can give whatever numbers you want in the simulation.



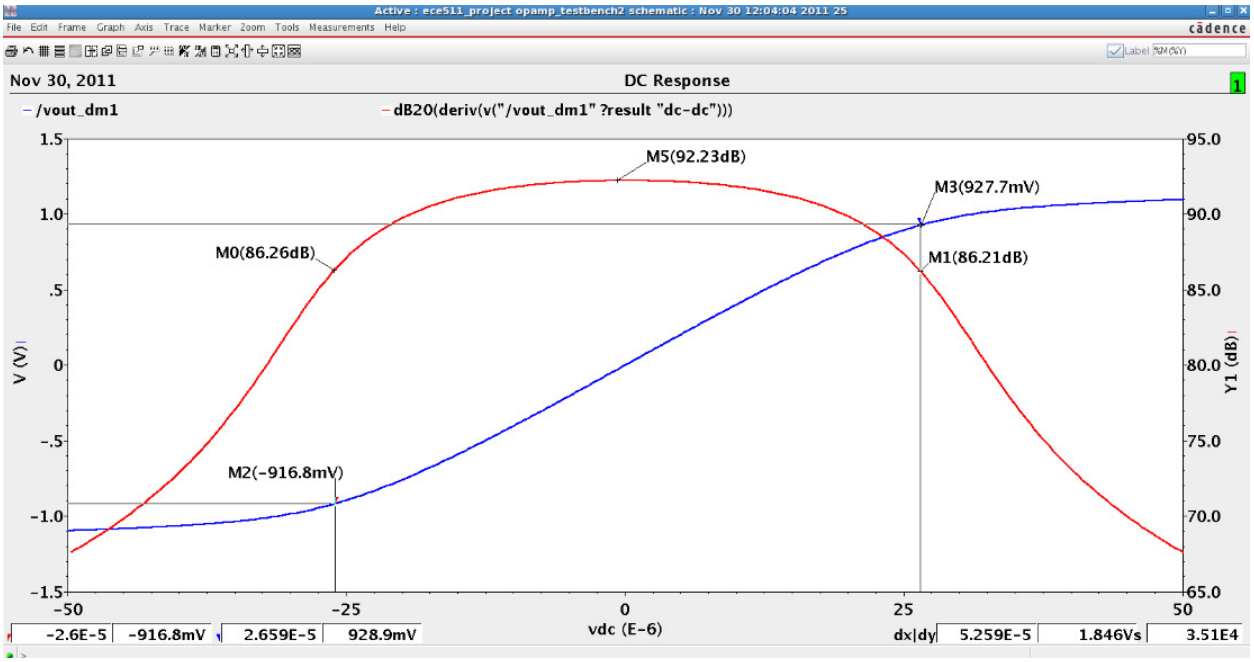
- Output -> To Be Plotted -> Select on Schematic for Differential Output 1 node
- Run DC simulation -> It will show the differential output1 voltage swing



- Tool -> Calculator -> Click waveform -> dB20(deriv(vout swing waveform))

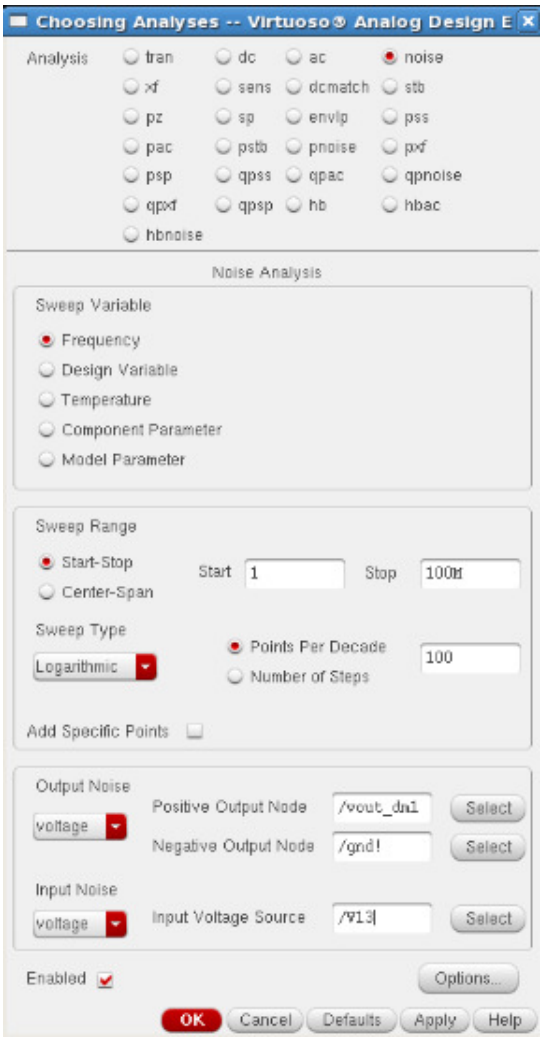


- Find the output swing points where the gain is 6dB less than the maximum gain
  (Blue line is the differential output and Red line is the 20dB of the slope)
- Session -> Save State (Please save the graphs for the project verification)

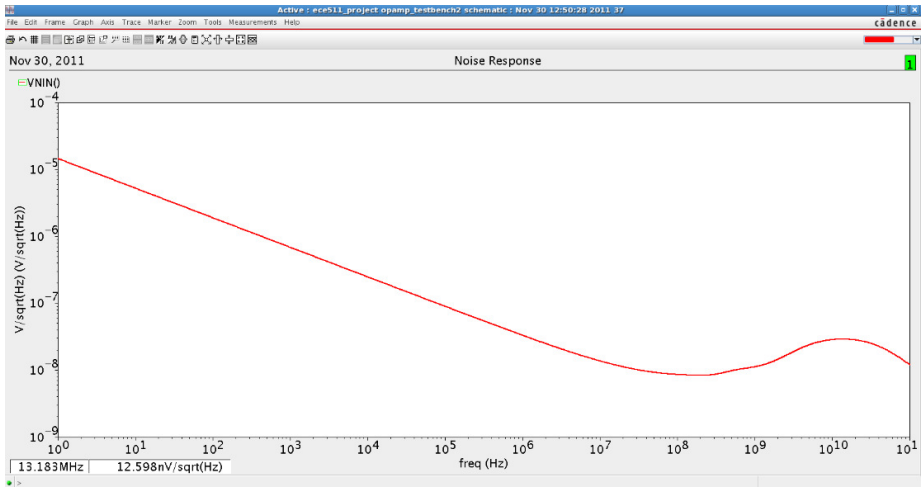


6. Noise simulation

- Please give Differential Input AC =1V ; Common Input AC = 0V
- Analyses->Choose->then set up the Noise simulation as shown below
  Please choose Output Noise as “voltage”; Please click the differential output on your schematic as Positive Output Node and click the ground on your schematic as Negative Output Node.
  Please choose Input Noise as “Voltage”; Please click the differential input voltage source on the schematic as Input Voltage Source.

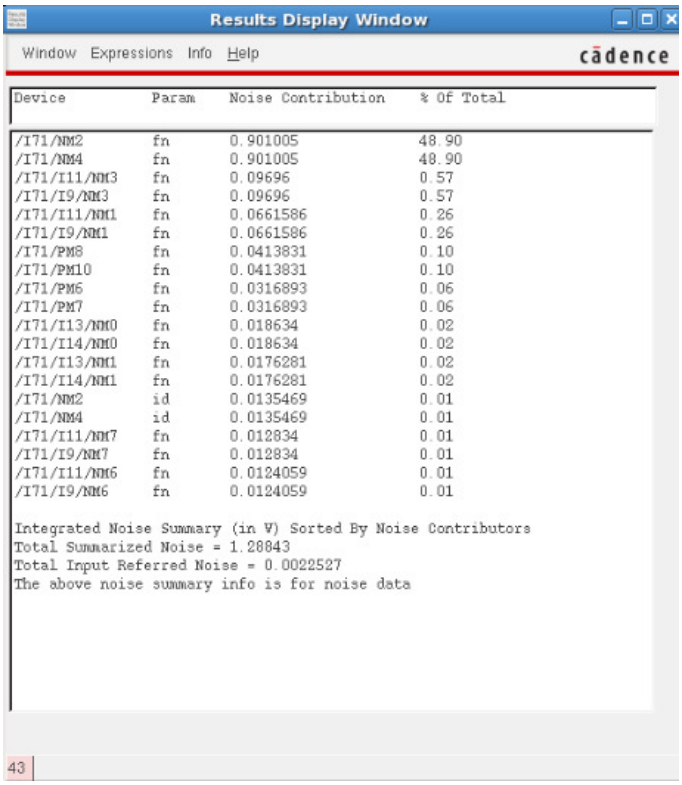


- Run Noise simulation
- Result -> Direct Plot -> Equivalent Input Noise
- Please click the vertical axis and choose “Log” on the graph screen



- For the integrated noise analysis, please go to Result -> Print -> Noise Summary

- Click OK, Then it will show you the total integrated output, input referred noise and the noise contributors from which transistors in your circuits.
- Session -> Save State

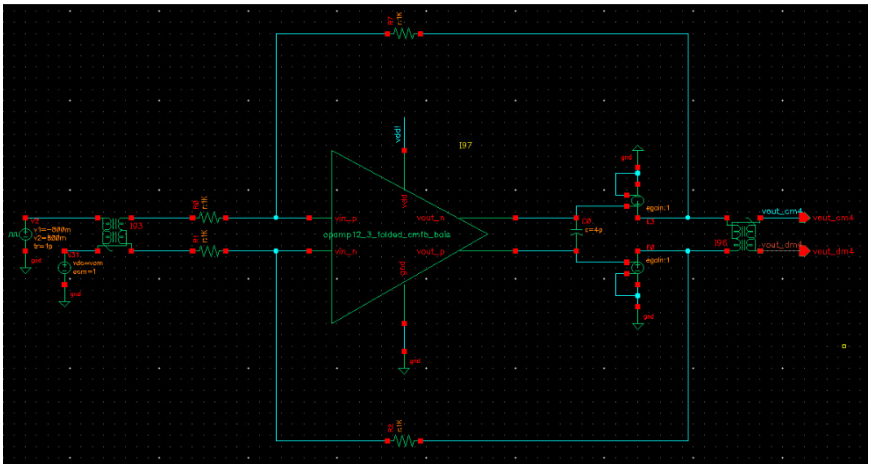


Device	Param	Noise Contribution	% Of Total
/I71/NM2	fn	0.901005	48.90
/I71/NM4	fn	0.901005	48.90
/I71/I11/NM3	fn	0.09696	0.57
/I71/I9/NM3	fn	0.09696	0.57
/I71/I11/NM1	fn	0.0661586	0.26
/I71/I9/NM1	fn	0.0661586	0.26
/I71/PM8	fn	0.0413831	0.10
/I71/PM10	fn	0.0413831	0.10
/I71/PM6	fn	0.0316893	0.06
/I71/PM7	fn	0.0316893	0.06
/I71/I13/NM0	fn	0.018634	0.02
/I71/I14/NM0	fn	0.018634	0.02
/I71/I13/NM1	fn	0.0176281	0.02
/I71/I14/NM1	fn	0.0176281	0.02
/I71/NM2	id	0.0135469	0.01
/I71/NM4	id	0.0135469	0.01
/I71/I11/NM7	fn	0.012834	0.01
/I71/I9/NM7	fn	0.012834	0.01
/I71/I11/NM6	fn	0.0124059	0.01
/I71/I9/NM6	fn	0.0124059	0.01

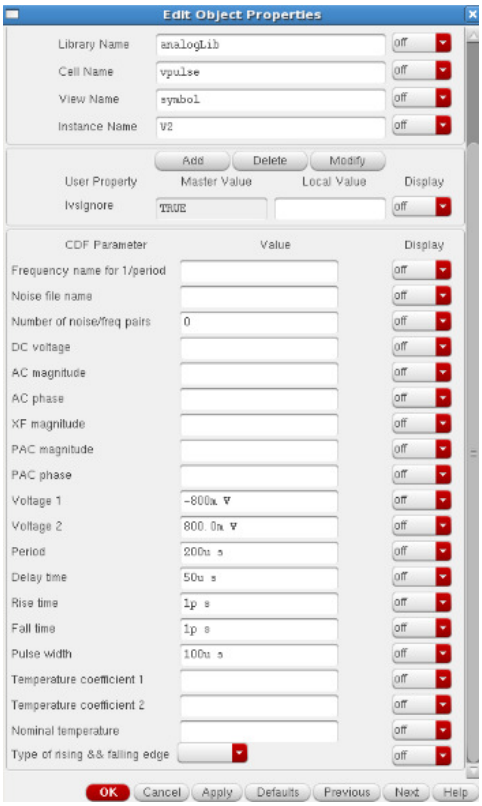
Integrated Noise Summary (in V) Sorted By Noise Contributors
Total Summarized Noise = 1.28843
Total Input Referred Noise = 0.0022527
The above noise summary info is for noise data

7.Settling time and Slew Rate

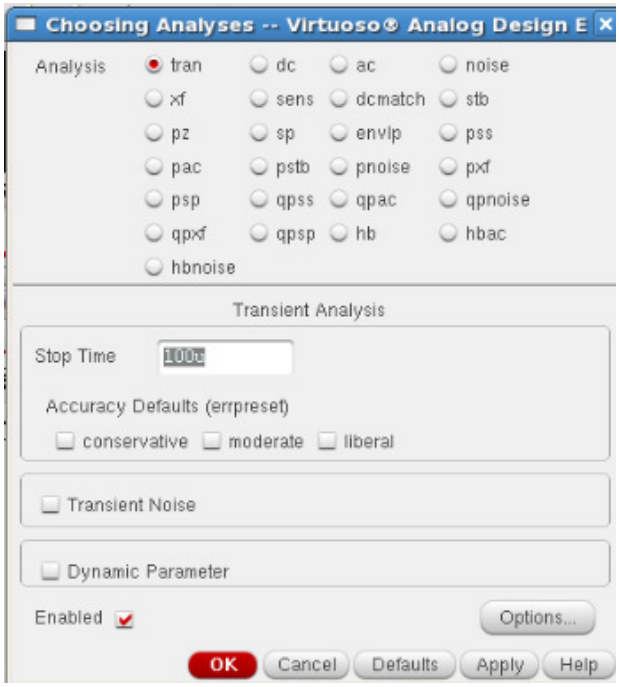
- Testbench schematic:
  Please add 3pF cap cross the two outputs; Please use vcvs sources as an ideal voltage buffers with the gain of 1 at the two outputs. Please give the same values for the 4 resistors in order to implement the unity-gain feedback;



- Please inject a vpulse source into the differential input as shown below.



- Analyses->Choose-> tran -> Setup the stop time



- Output -> To Be Plotted -> Select the differential output on your schematic.
- Run Tran Simulation

  The settling time is the time from the beginning of the step to the specified percentage of the step (1%); Slew rate is the slop of the ramp of the step.