

Analog IC Design

Lab 01 Mentor Pyxis Tutorial

LPF Simulation and MOSFET Characteristics

Part 1: Low Pass Filter Simulation (LPF)

1. Project Manager

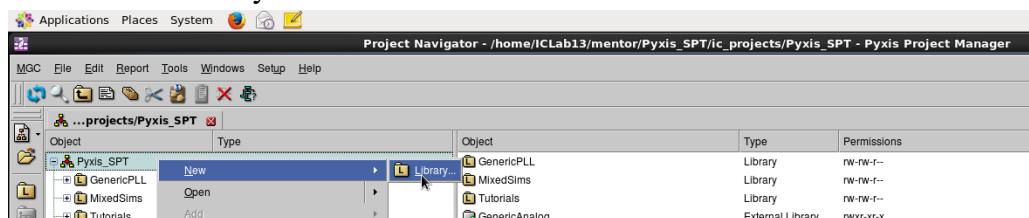
Open a new terminal and browse to the project directory.

```
>> cd ~/mentor/Pyxis_SPT_HEP_ASU
```

Run the “open_Pyxis_SPT” script to open the project navigator.

```
>> source open_Pyxis_SPT
```

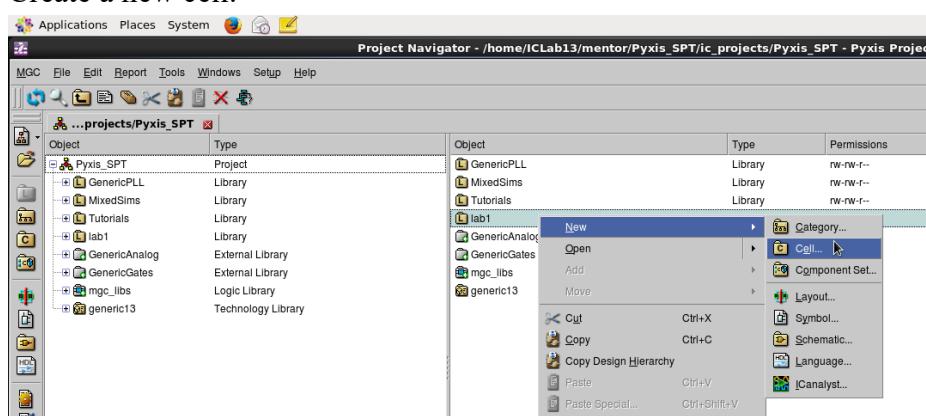
Create a new library.



Give your library a name.

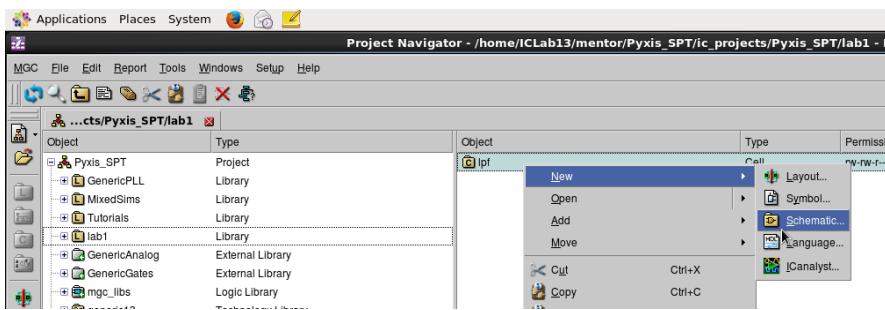


Create a new cell.



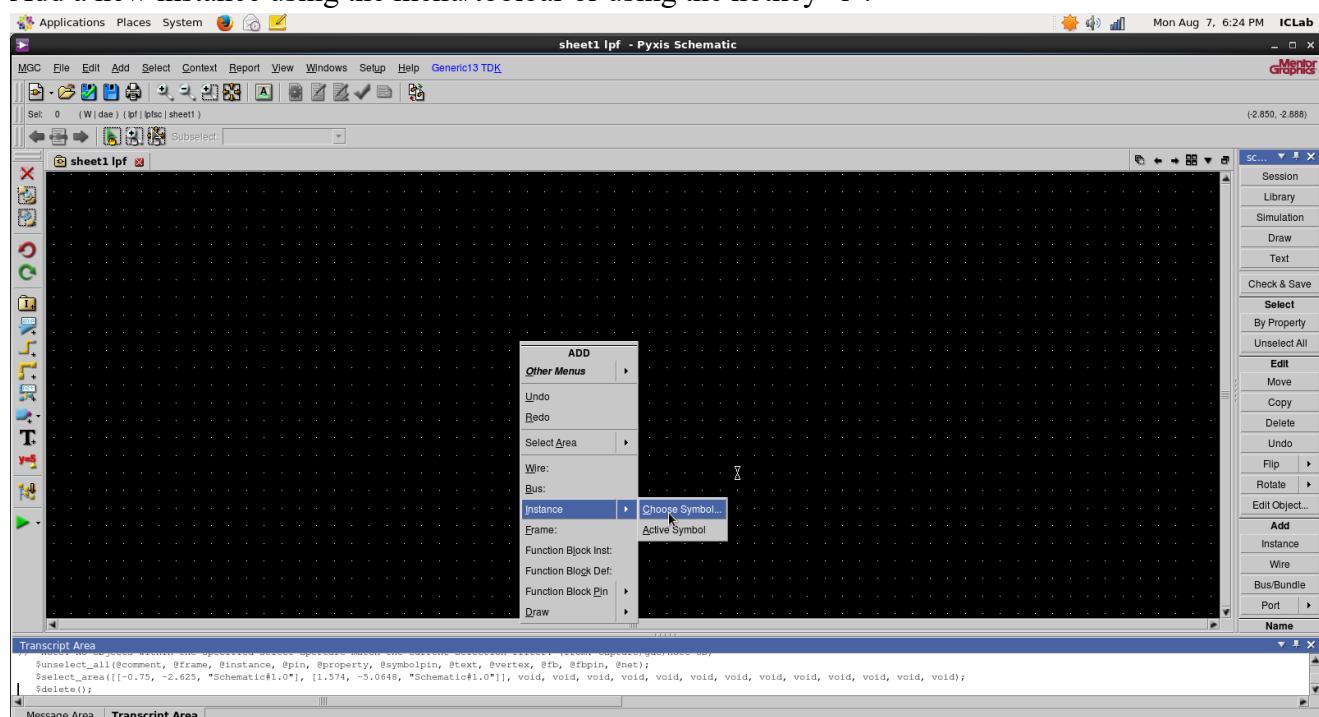
2. Design Entry (Schematic)

Create a new schematic.

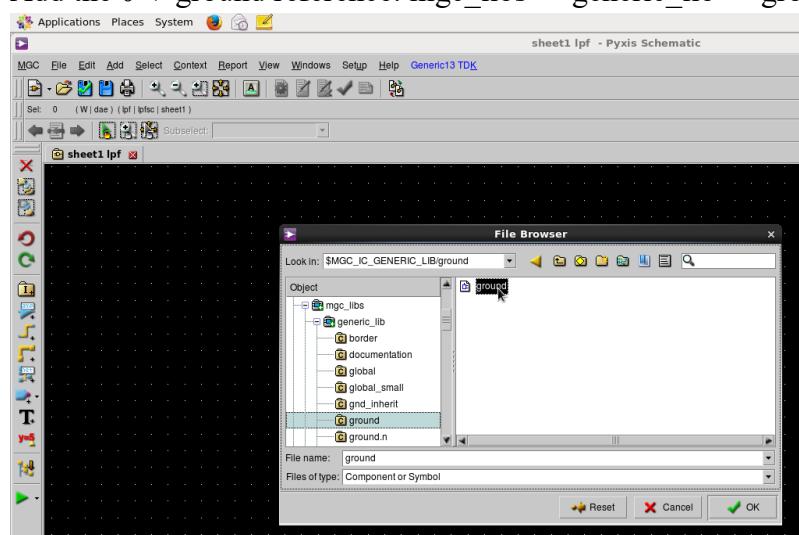


Pyxis schematic editor window will appear.

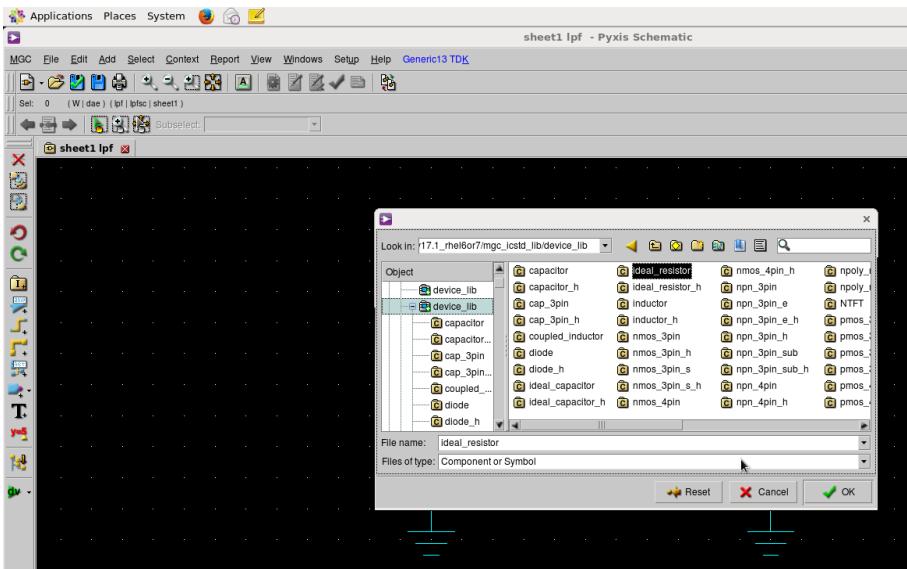
Add a new instance using the menu/toolbar or using the hotkey “i”.



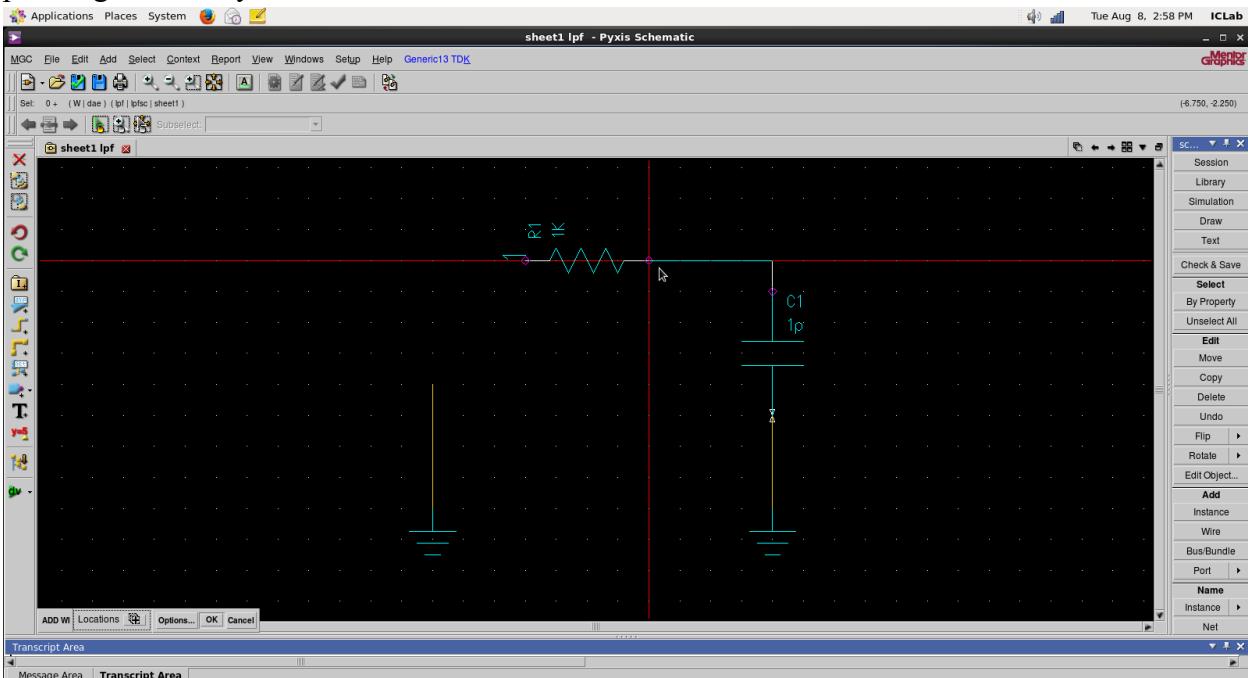
Add the 0 V ground reference: mgc_libs -> generic_lib -> ground.



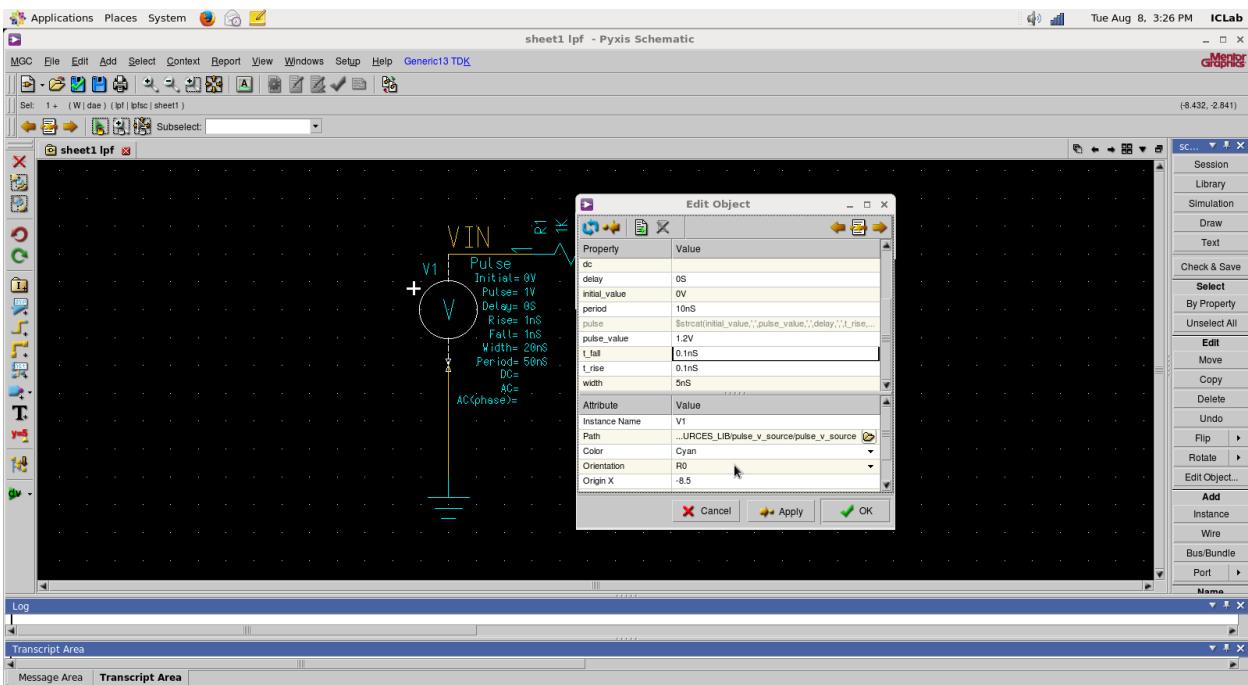
Add an ideal resistor from mgc_libs -> device_lib -> ideal_resistor.



Similarly add a capacitor. Connect them together using the wire tool from the menu/toolbar or by pressing the hotkey “w”.



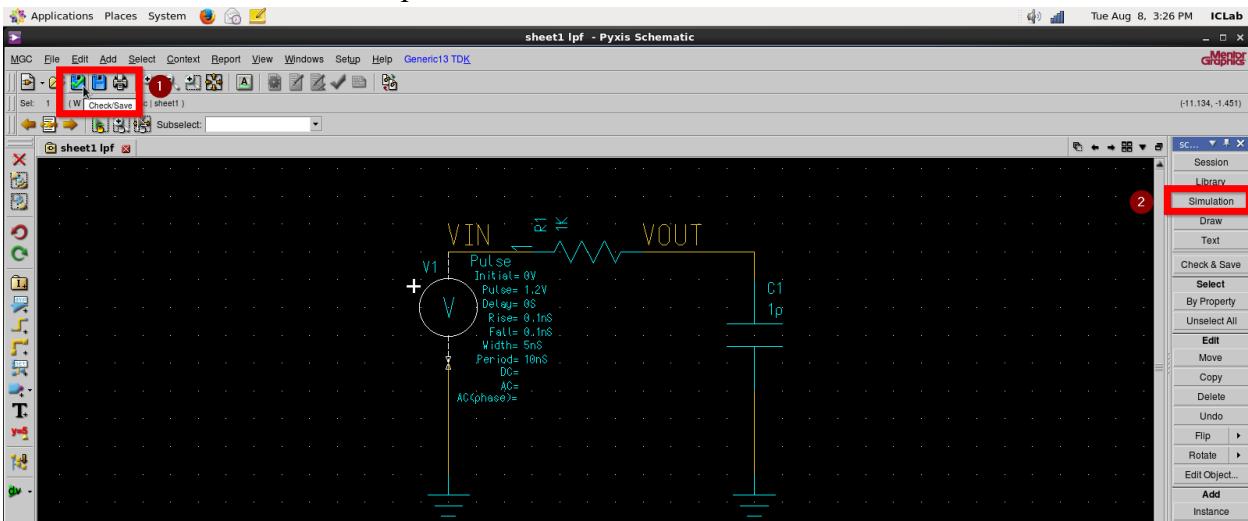
Add a pulse voltage source from mgc_libs -> sources -> pulse_voltage. Set the values of the source as shown below. You can open the properties window from the menu/toolbar or by using the hotkey “q” while selecting the device.



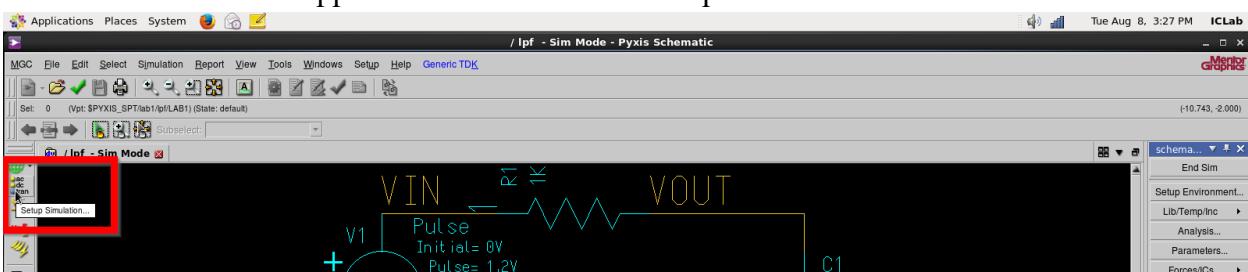
Add names (labels) for the nets by using the menu/toolbar or the hotkey “l”. Click check and save (or “shift + x”).

3. Transient Analysis

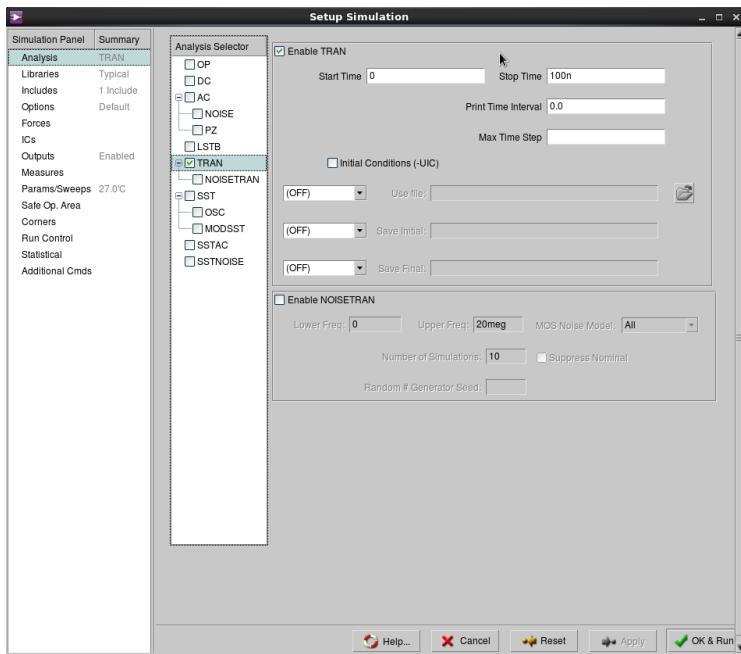
Choose “Simulation” from the palette.



The simulation toolbar appears on the left. Click “Setup Simulation”.

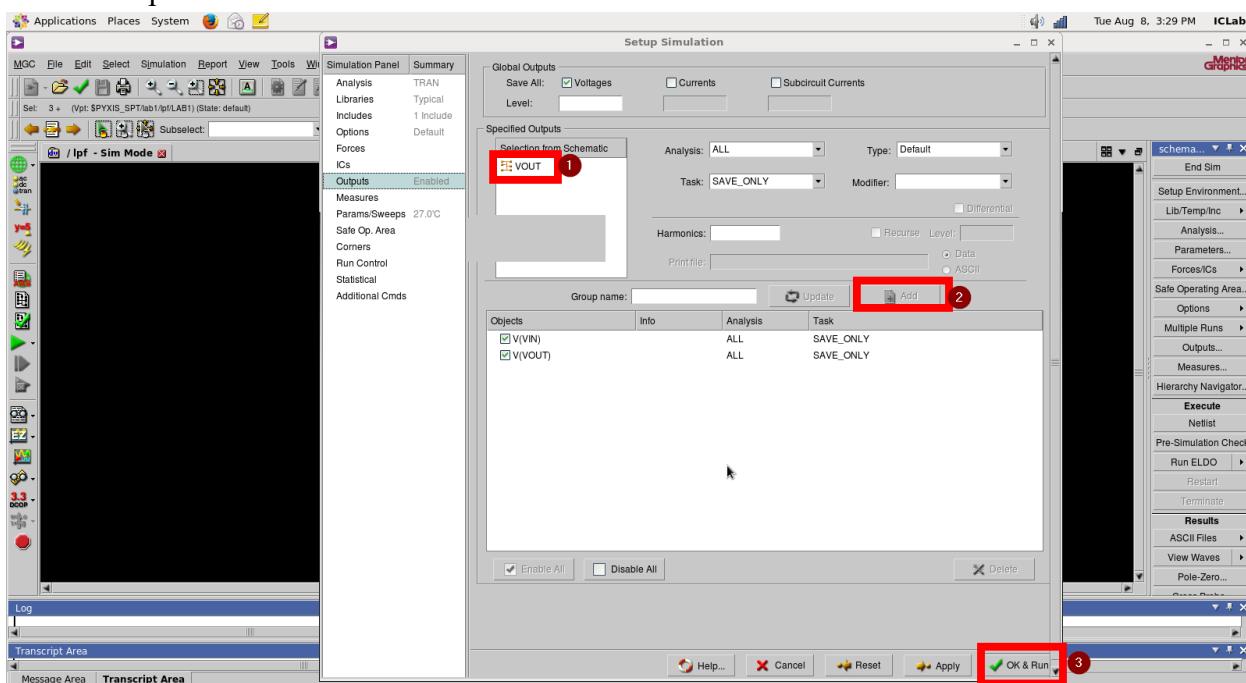


Choose TRAN (transient) simulation, then set the stop and start time. Choose the max time step to be equal to $T_{clk}/100$.

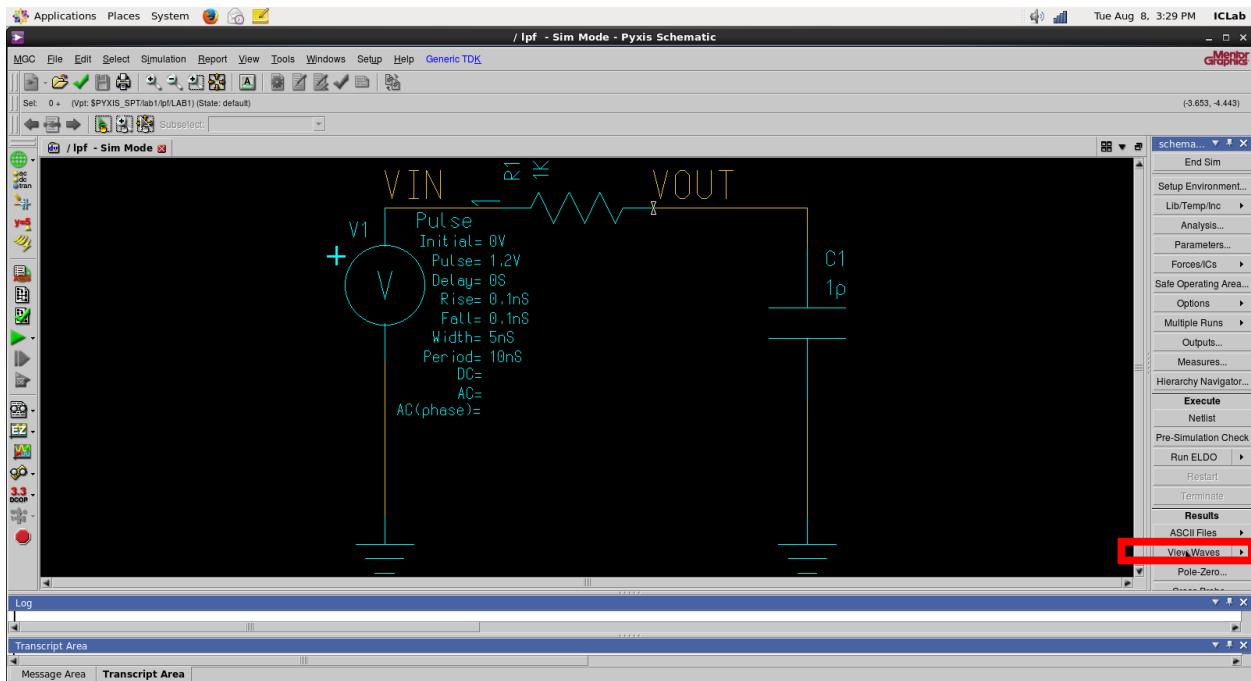


Setup the outputs to be drawn from the Outputs panel.

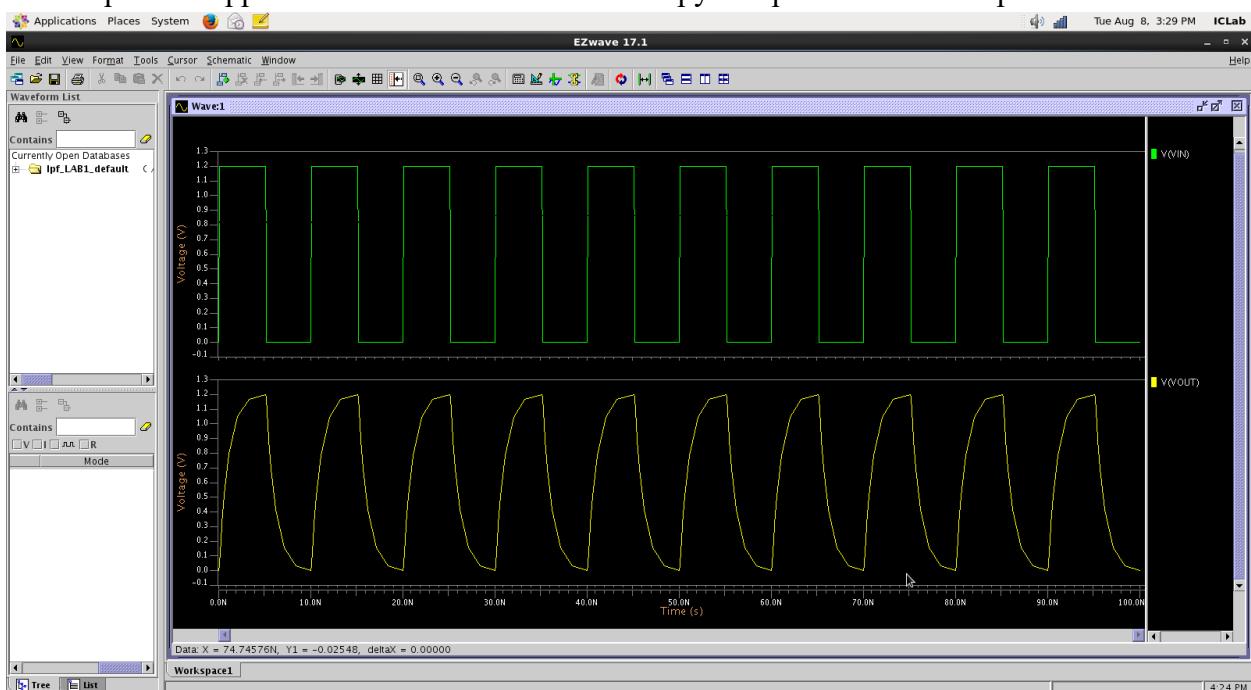
Click on “Selection from schematic” then press on the VOUT net in the schematic then click on “Add”. Repeat the same for VIN net. Then click on “OK & Run”.



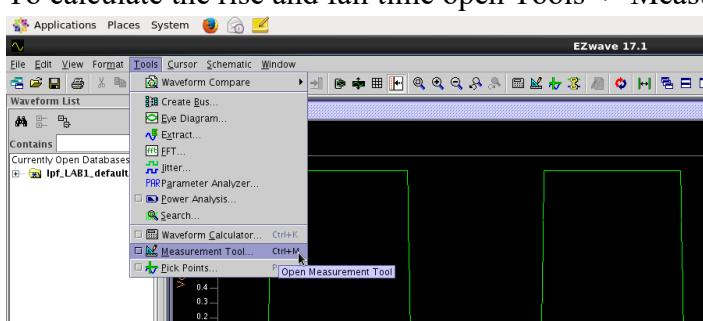
Click on “View Waves” in order to view the outputs.



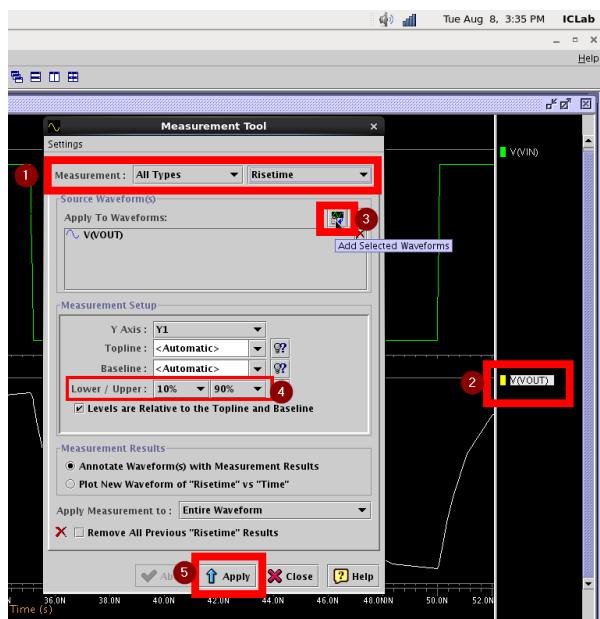
The output will appear as below. You can use cut/copy and paste to make outputs overlaid.



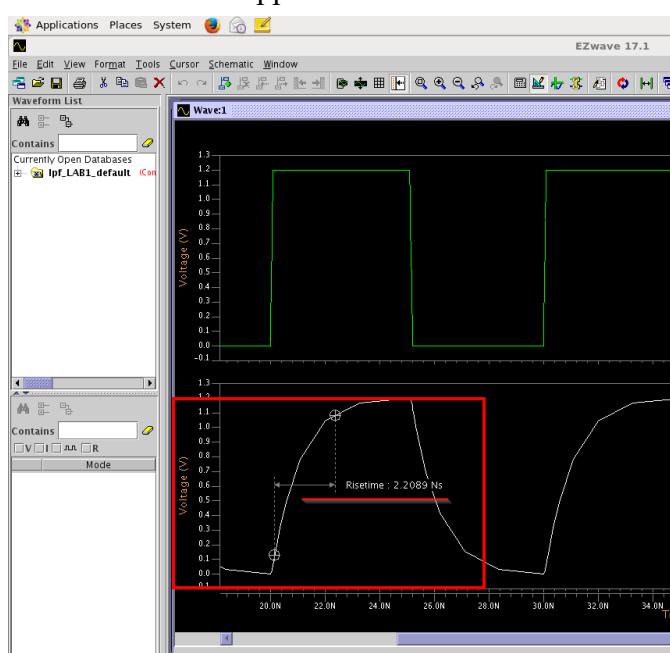
To calculate the rise and fall time open Tools -> Measurement Tool.



Add the waveform to the measurement tool as below.

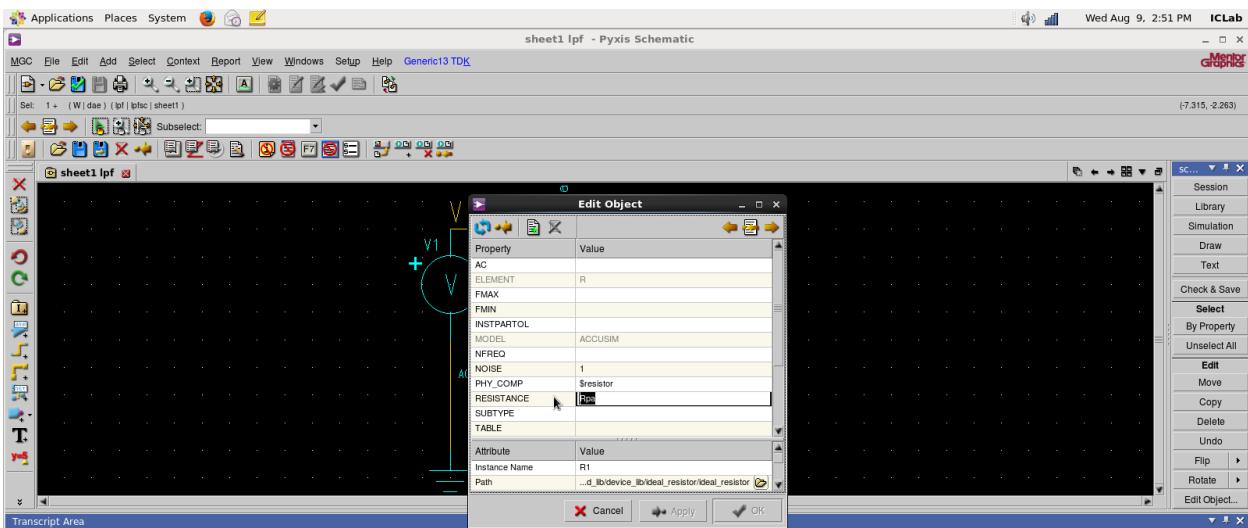


The rise time will appear on the wave.

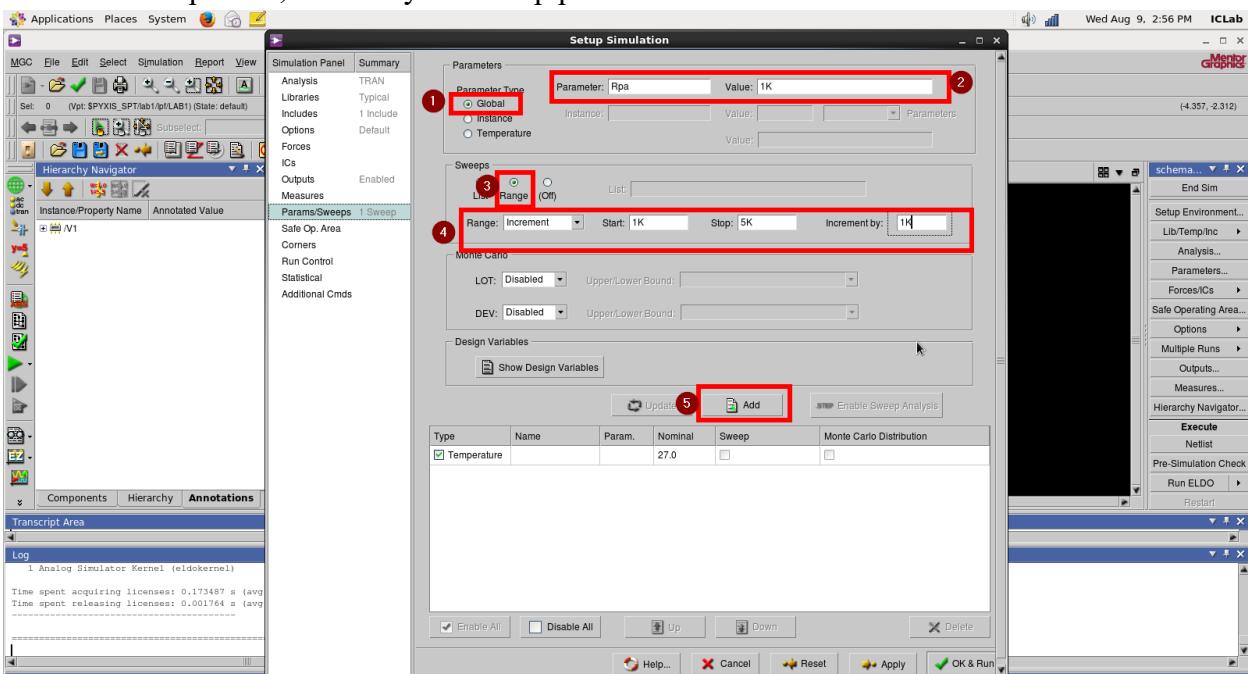


4. Parametric Analysis

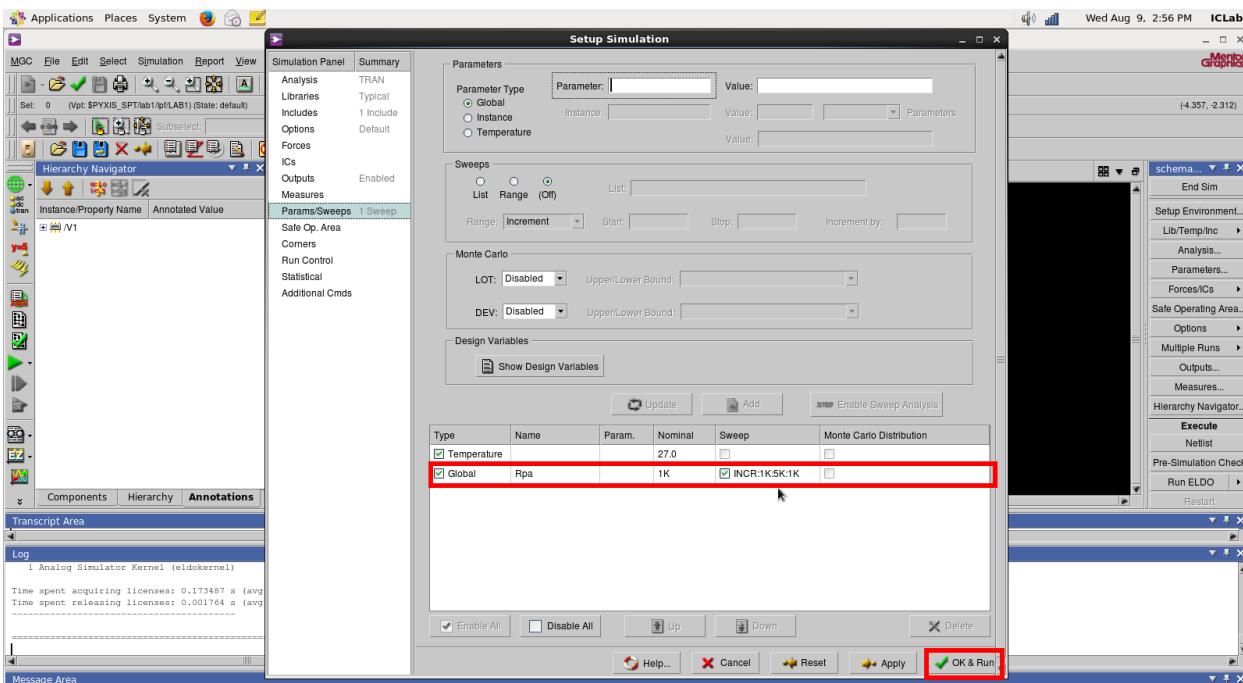
End the simulation mode, then click on the resistor, press “q”, and type Rpa in the resistance field as shown below (the resistance becomes a variable/parameter).



Enter the simulation mode again (click on “Simulation” from the palette). Go to the “Params/Sweeps” tab, and add your sweep parameter as shown below.



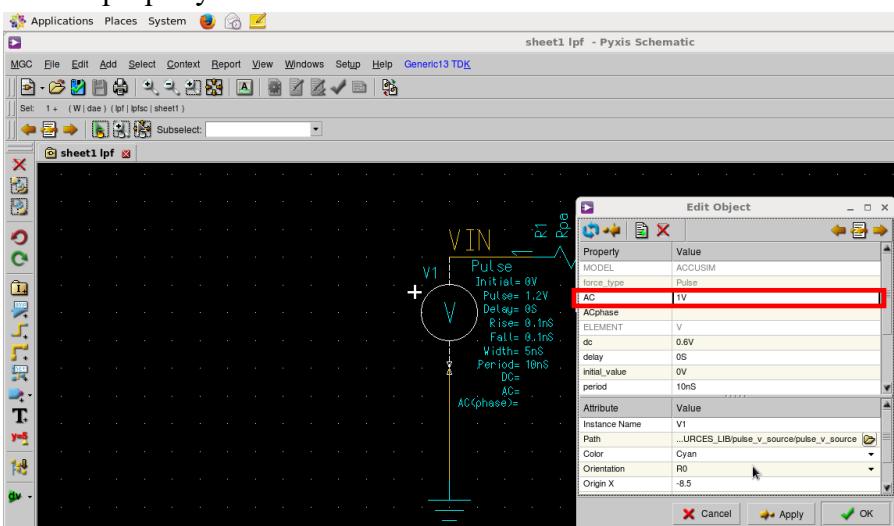
The parameter should appear as below. Click on “OK & Run”.



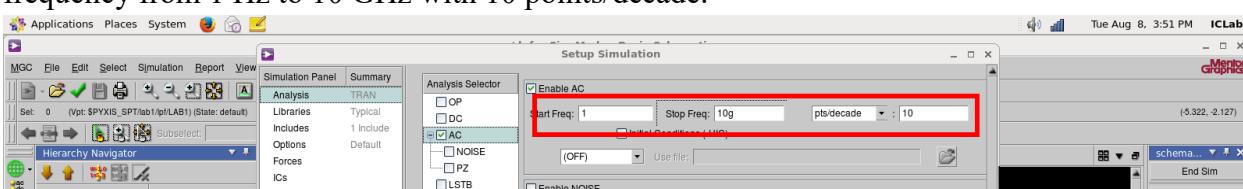
With the same outputs as before, click on “View Waves” in order to view the outputs.

5. AC Analysis

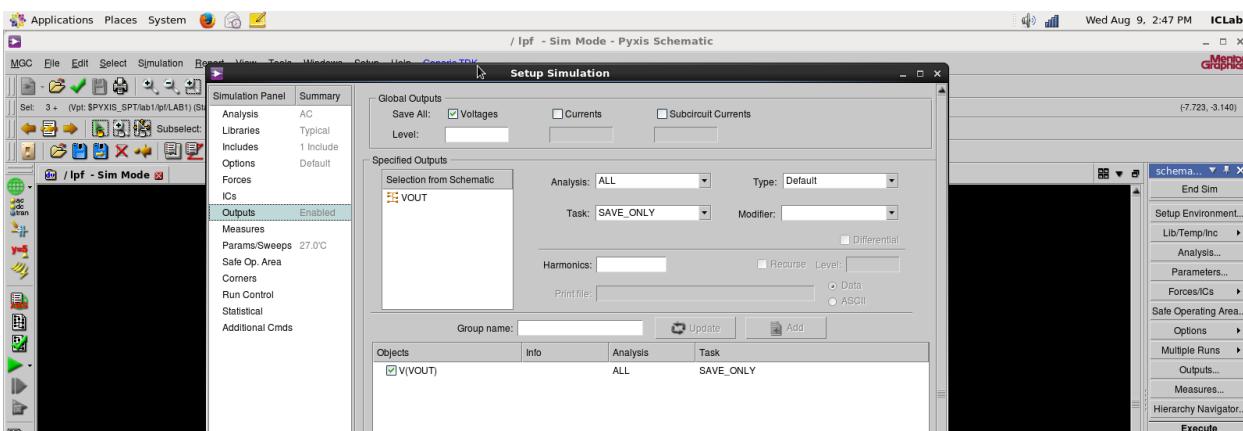
End the simulation mode. Set the resistance back to 1 k. In the voltage source properties enter 1V in the AC property as below.



Start the simulation mode. In the simulation setup choose AC. Make a logarithmic sweep of frequency from 1 Hz to 10 GHz with 10 points/decade.

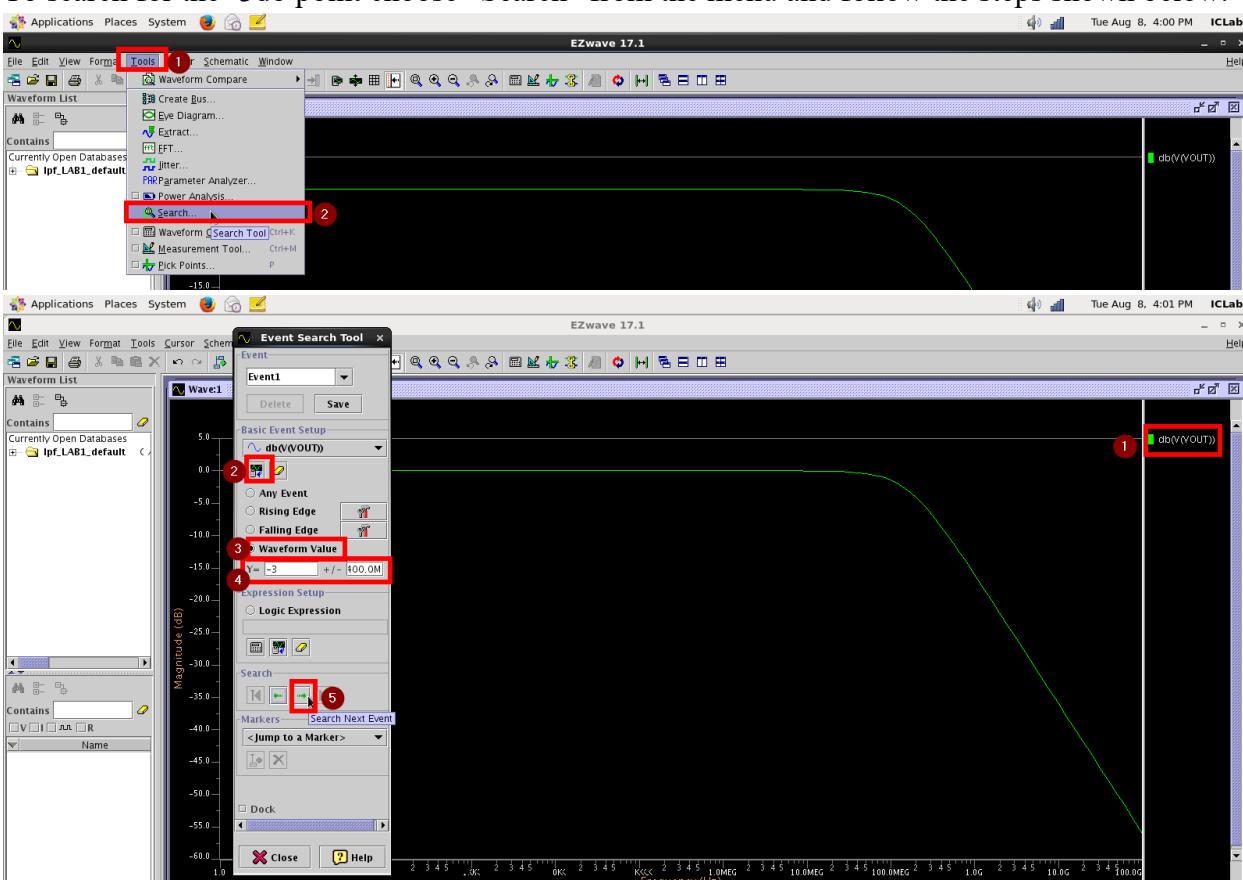


Add VOUT to the outputs as before.

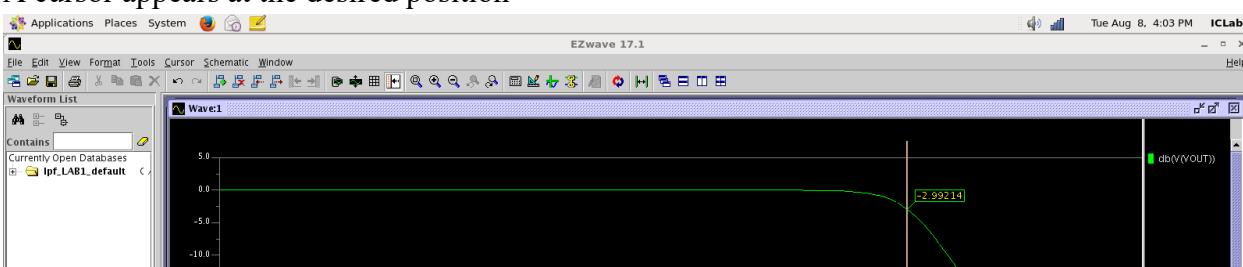


Click on “OK & Run” then click on “View Waves” in order to view the outputs.

To search for the -3db point choose “Search” from the menu and follow the steps shown below.

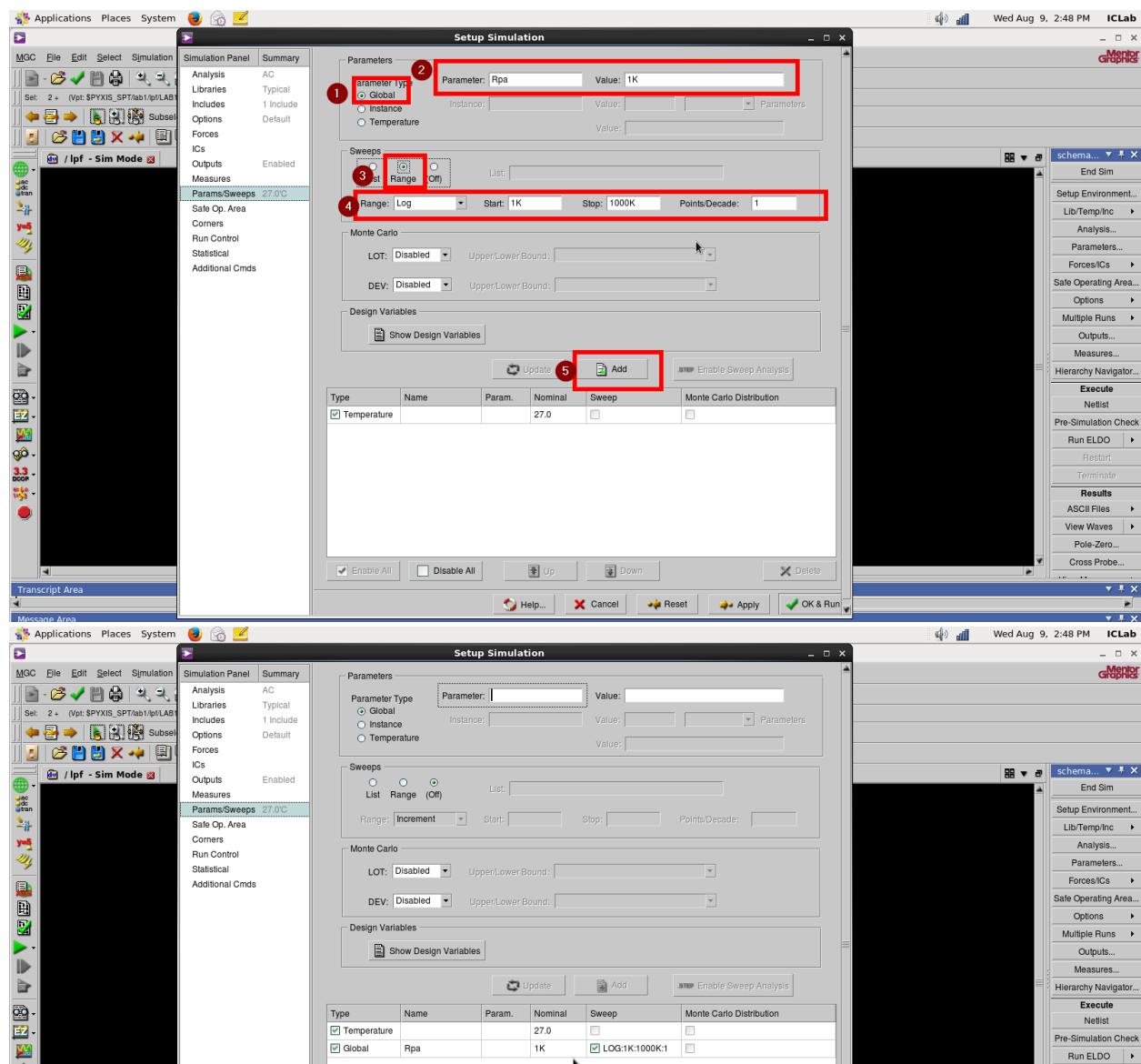


A cursor appears at the desired position



End the simulation mode and set the resistance as a parameter Rpa.

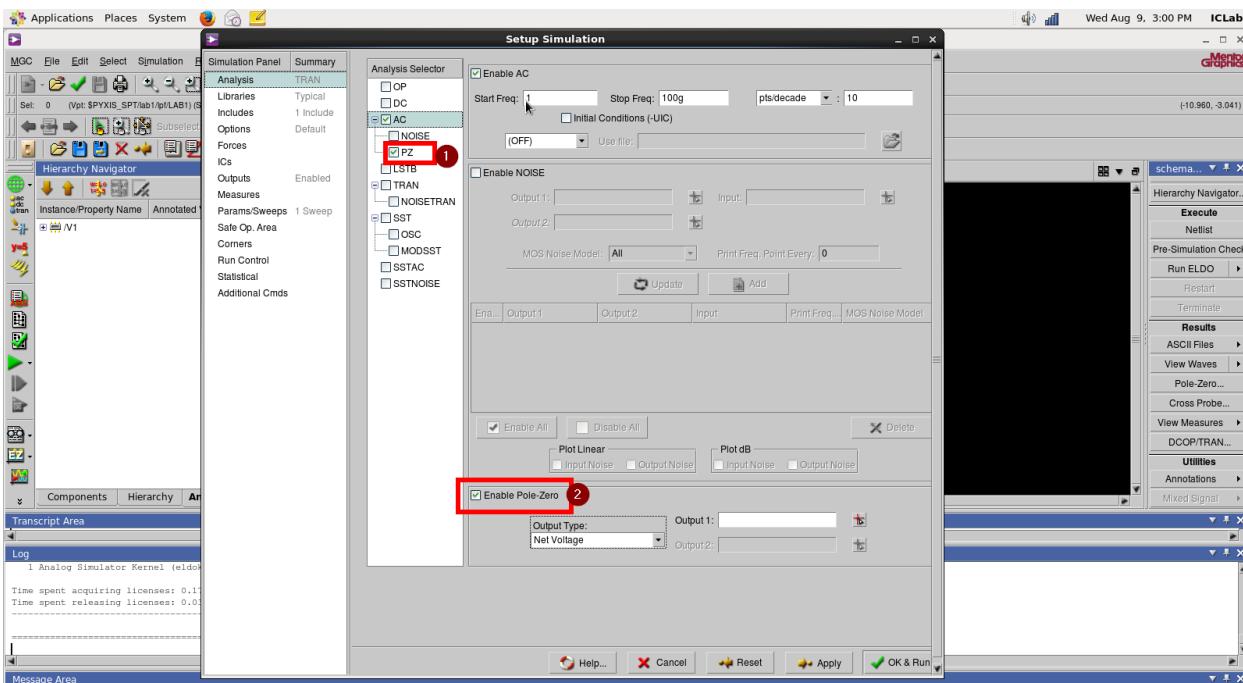
Enter the simulation mode. Go to the “Params/Sweeps” tab, and add your sweep parameter as shown below.



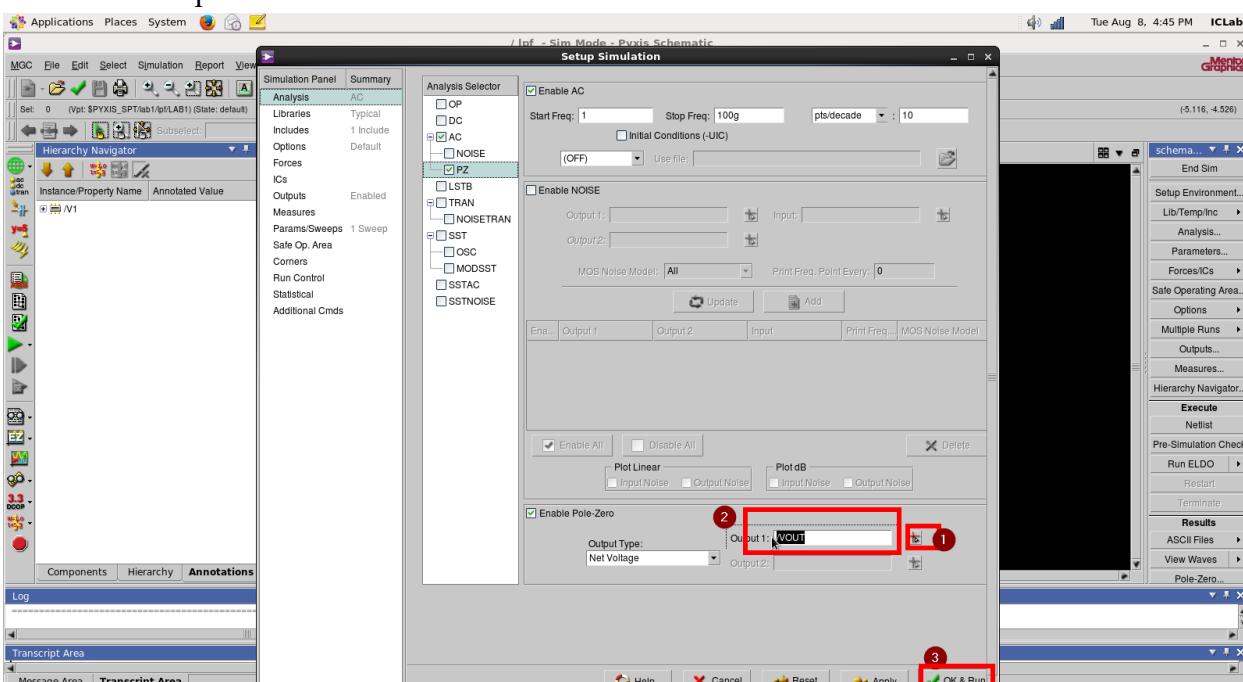
Click on “OK & Run” then click on “View Waves” in order to view the outputs.

6. Pole-Zero Analysis

End the simulation mode. Set the resistance value, then enter the simulation setup -> analysis -> AC -> enable PZ analysis.



Choose the output node then click on “OK & Run”.



Open a terminal and CD to the project directory. Use the “pz” command to open the .pz output file. Alternatively, open the AMS results browser tool using this command:

>> amsrb

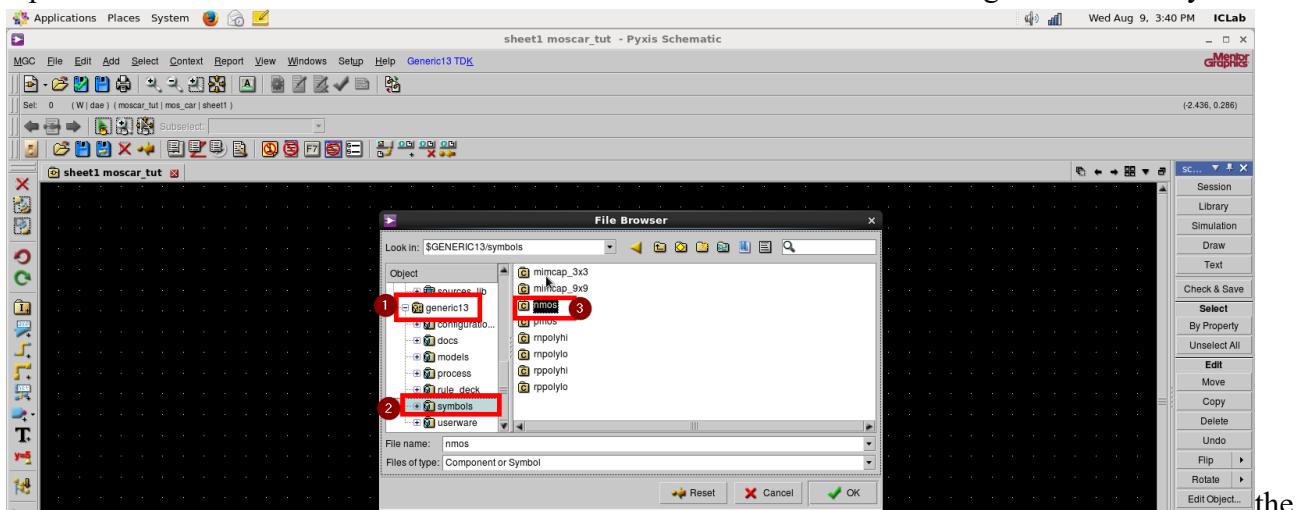
Then browse to the project directory and open the .pzs output file.

```
ICLab13@ICLab13:LAB1
File Edit View Search Terminal Help
[ICLab13@ICLab13 Pyxis_SPT]$ cd /home/ICLab13/mentor/Pyxis_SPT/ic_projects/Pyxis_SPT/lab1/lpf/LAB1
[ICLab13@ICLab13 LAB1]$ pz lpf_LAB1_default.pz
```

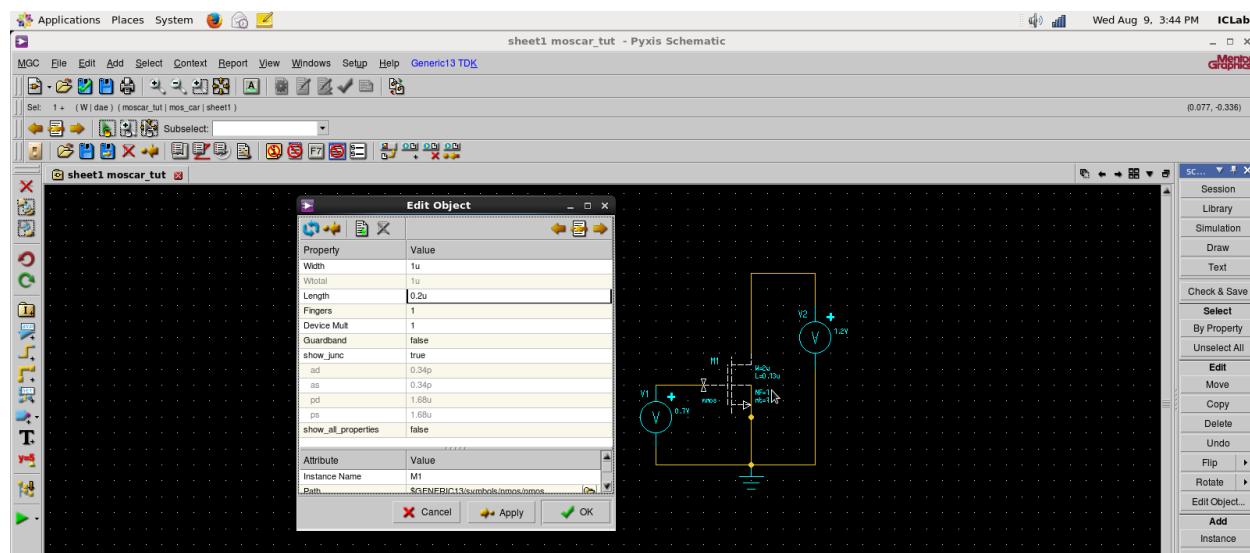
Part 2: MOSFET Characteristics

1. ID vs VGS

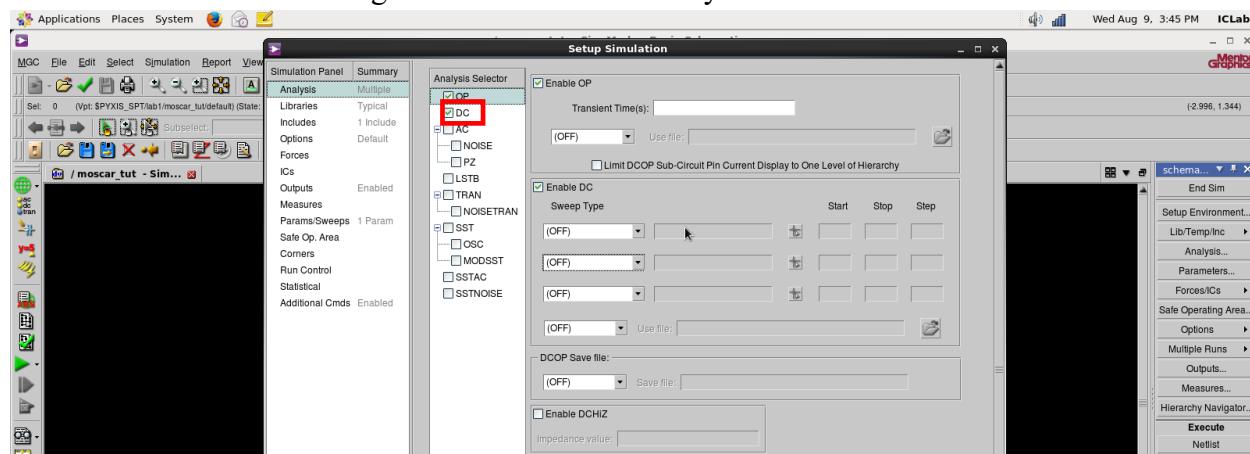
Open a new cell -> schematic then add an nmos transistor instance from the generic13 library.



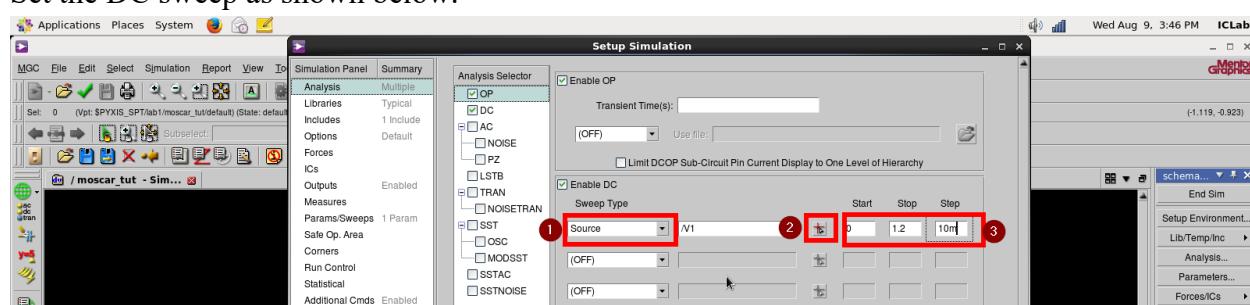
Add two DC voltage sources and wire the circuit as below. Set the properties of the transistor and the sources.



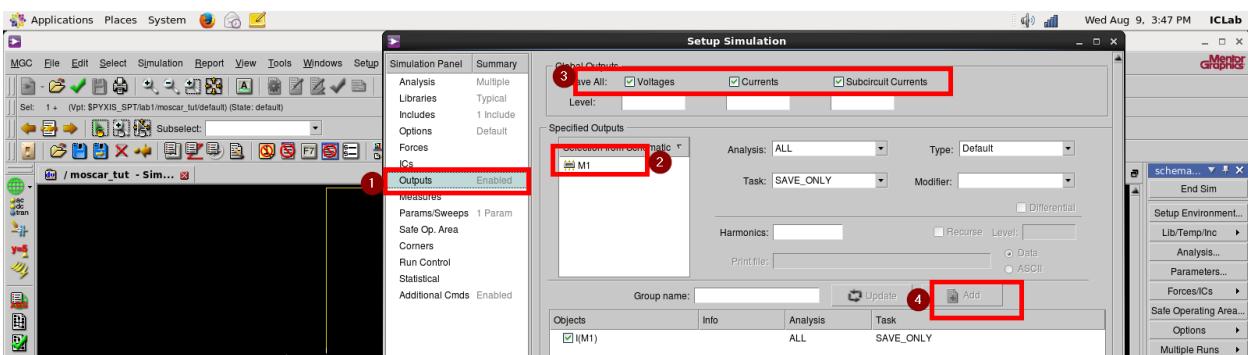
Start a new simulation configuration. Choose DC analysis.



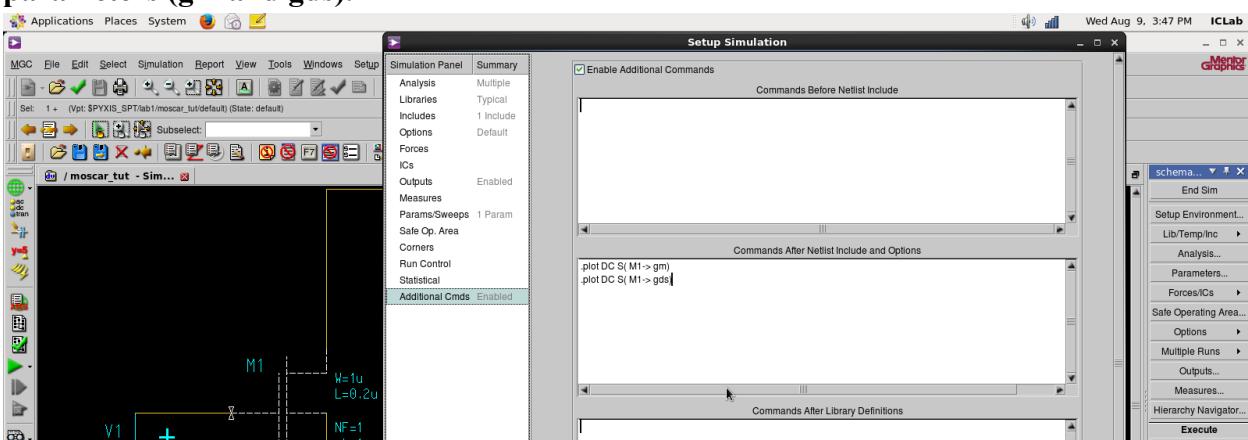
Set the DC sweep as shown below.



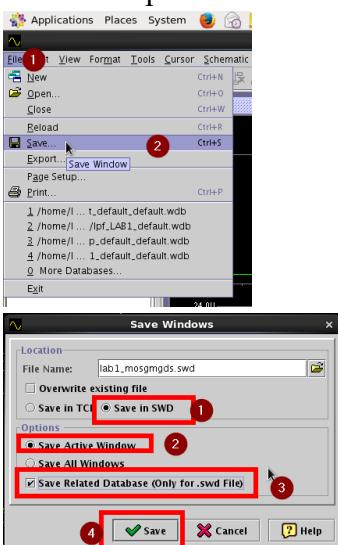
In the Outputs tab save all voltages and currents. You can also add a specific transistor, e.g., M1 to save its data. For a large circuit only save the outputs that you need to save disk space.



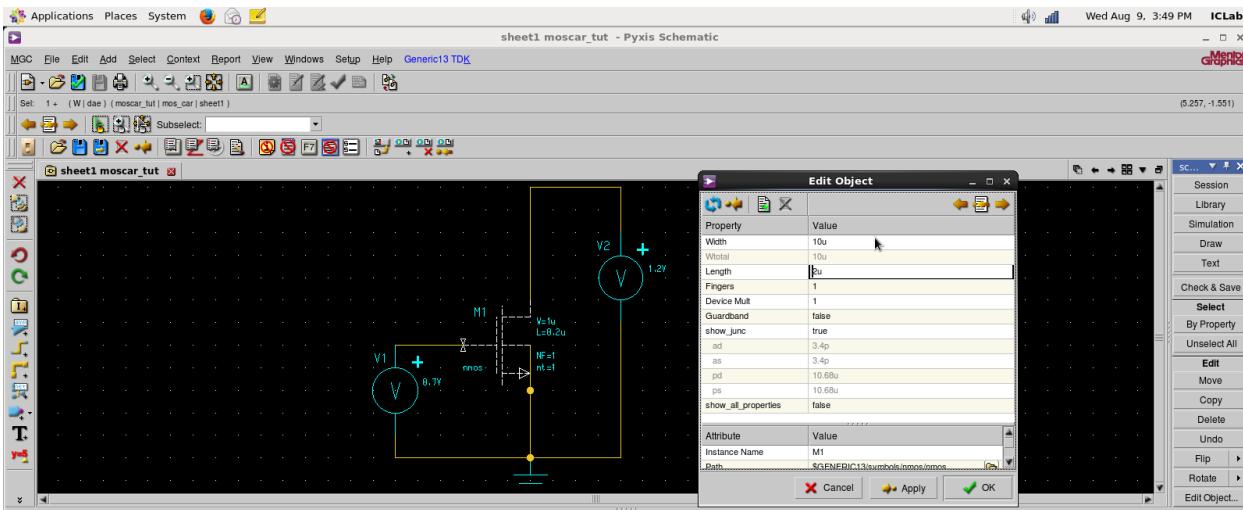
In the “Additional Cmds” panel type the commands shown below to draw the OP point parameters (gm and gds).



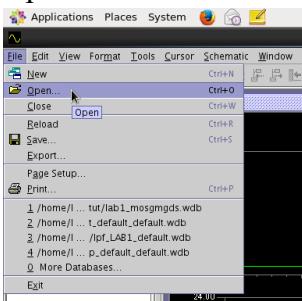
In order to plot overlaid results, we need to save the results as below.



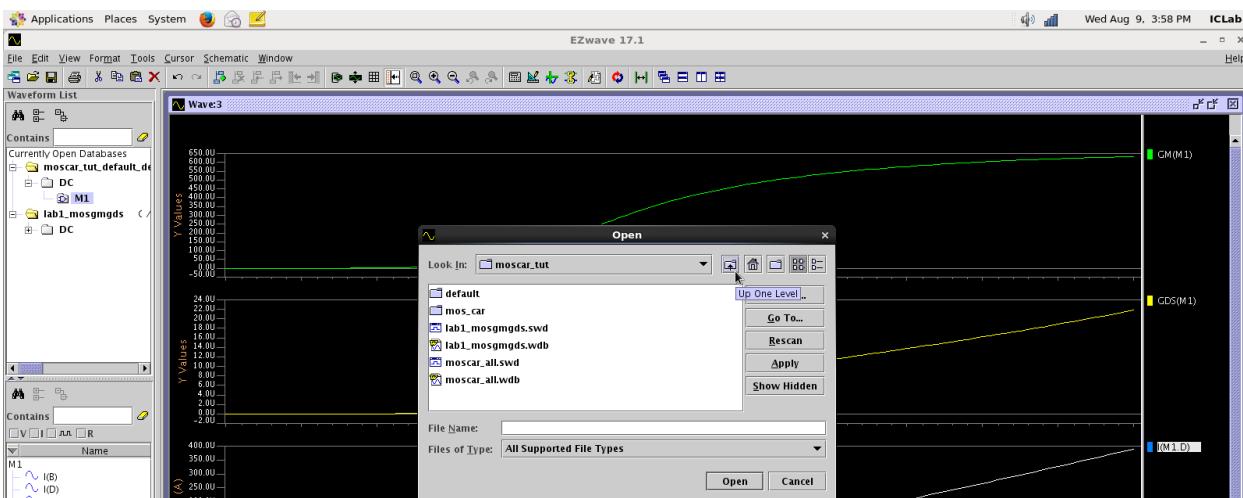
End the simulation mode, enter the new properties (W and L), and run the simulation again.



Open the old results.



Choose the .swd file.



The results will show in a new tab. Copy and paste the waveforms to show them overlaid.

2. ID vs VDS

We need to perform a nested sweep. From DC analysis choose the primary variable as Vds (V1) and the secondary variable as Vgs (V2) as shown below.

Ain Shams University – Faculty of Engineering – ECE Dept. – Integrated Circuits Lab.
Dr. Hesham Omran and Eng. Mohamed Fouad

