

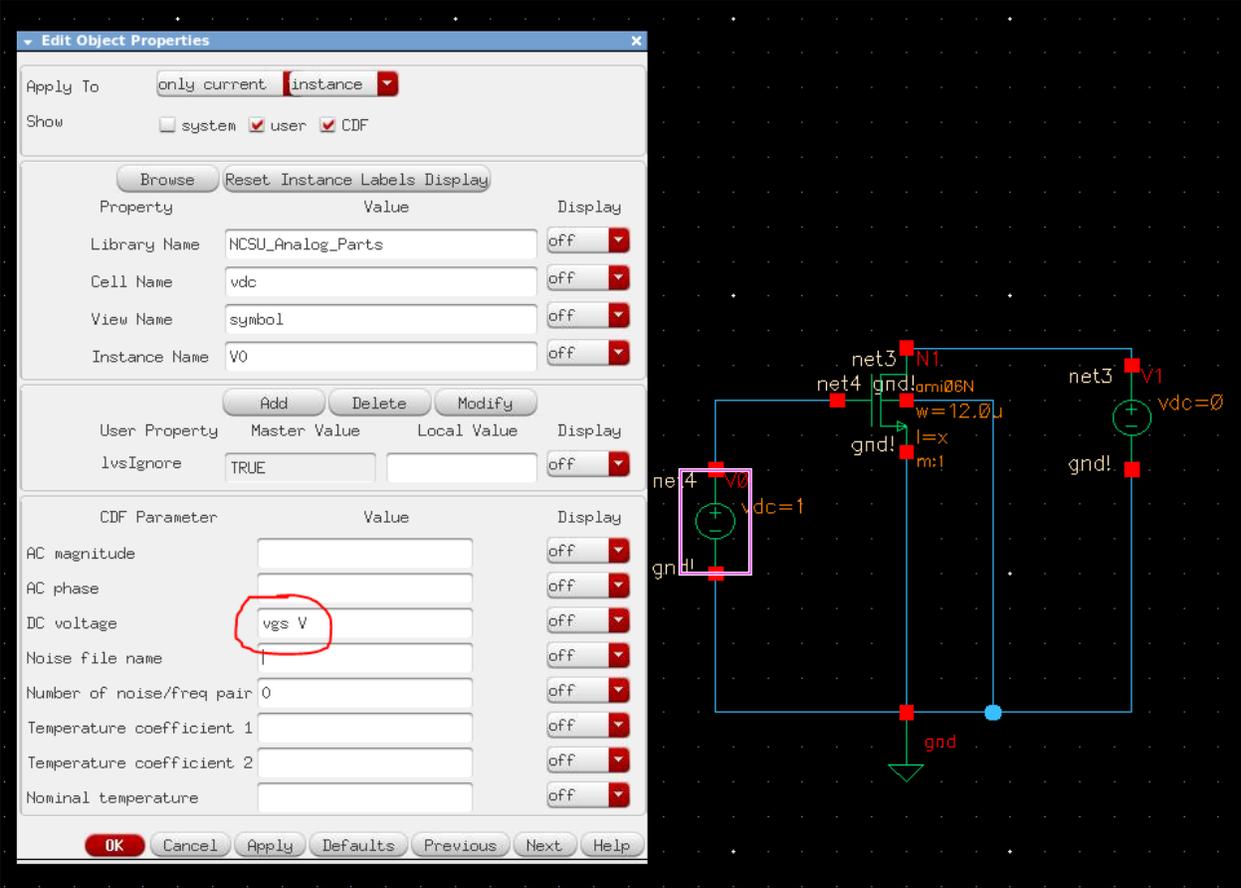
# EE330 lab 6 Supplemental Instruction

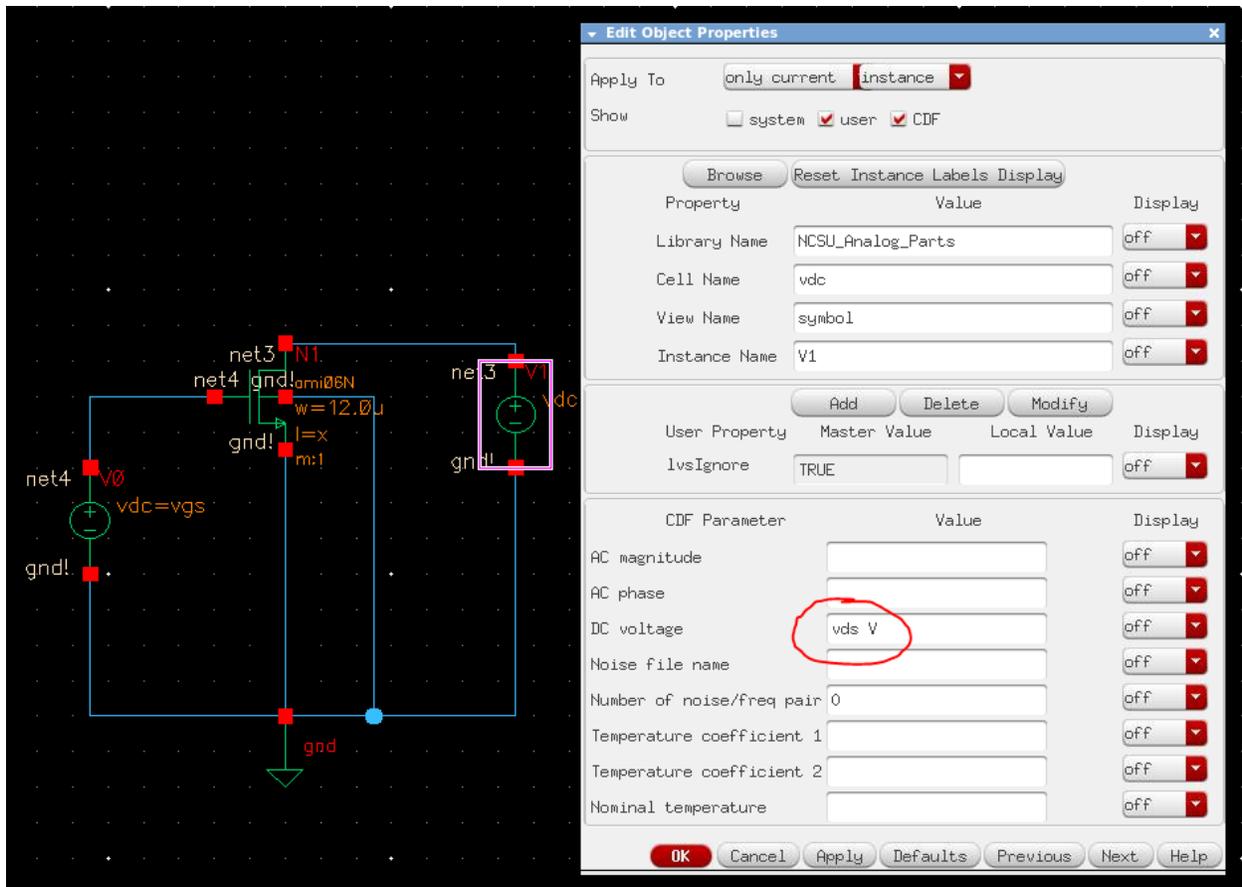
## How to setup parametric analysis

Author: Xilu Wang 2016.2

### Step 1

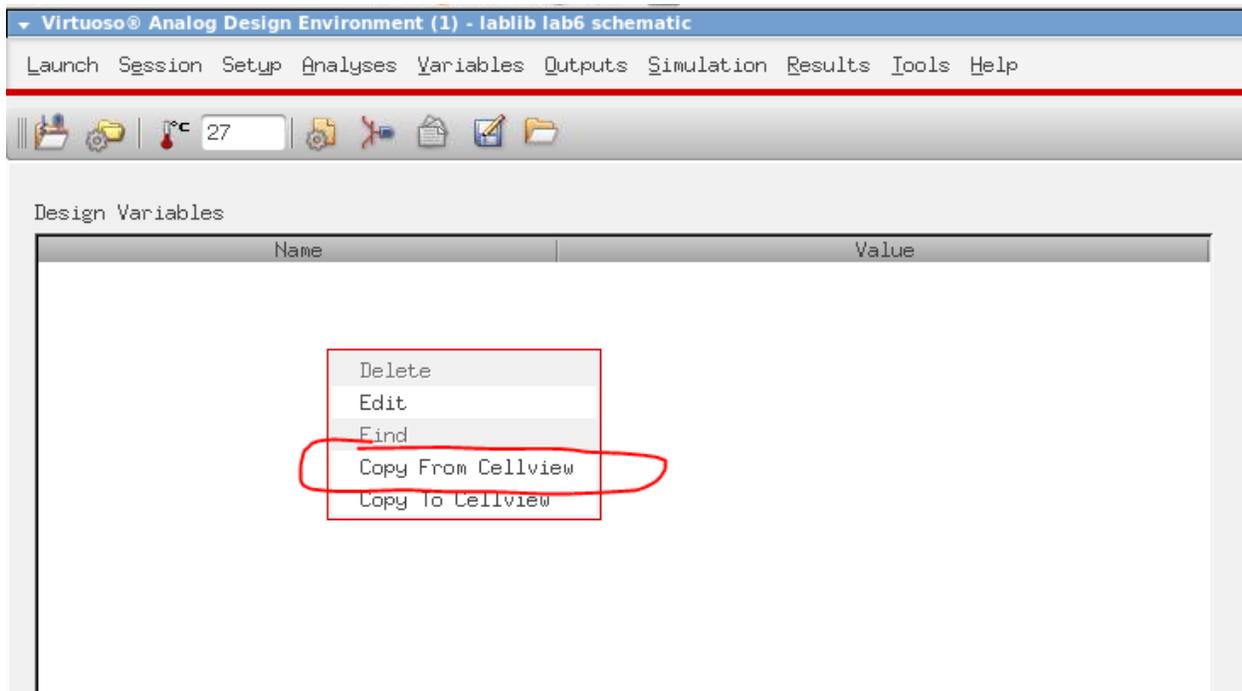
set up variables for “vgs” and “vds”





## Step 2

Open **ADE L** by in the dropdown menu of **Launch**. Right click in the **Design Variables** window and then choose **Copy from cellview** to call all the variables you just created in the schematic.



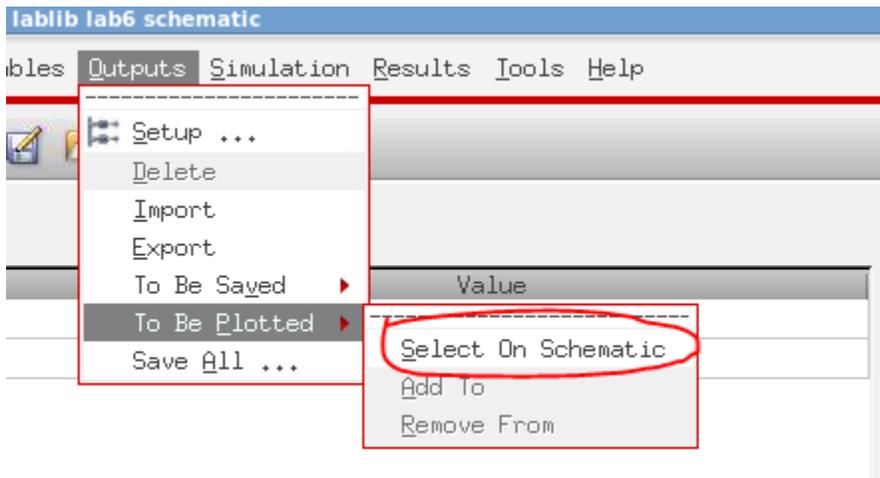
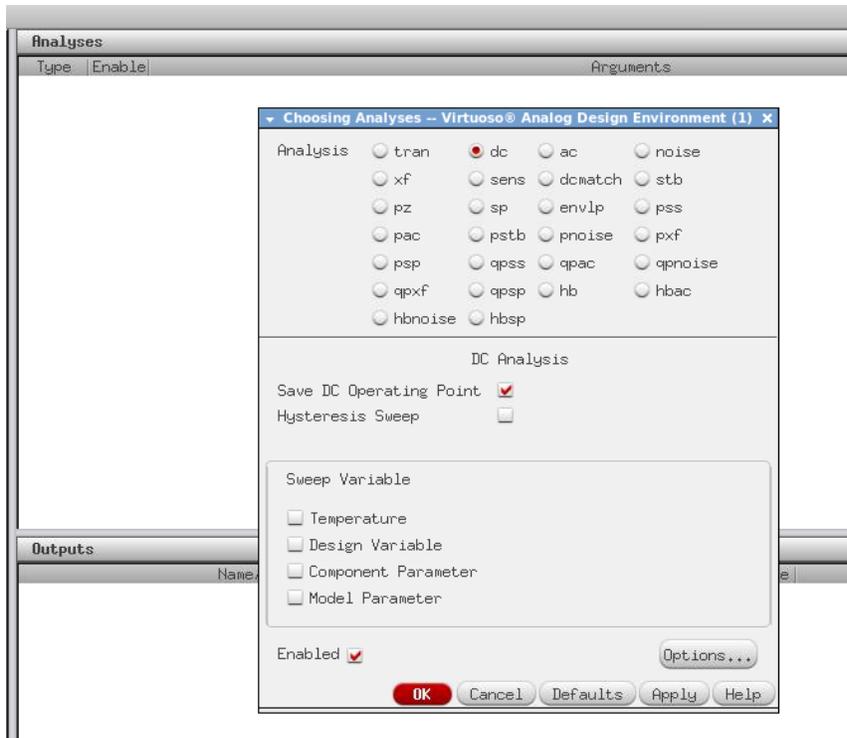
Design Variables

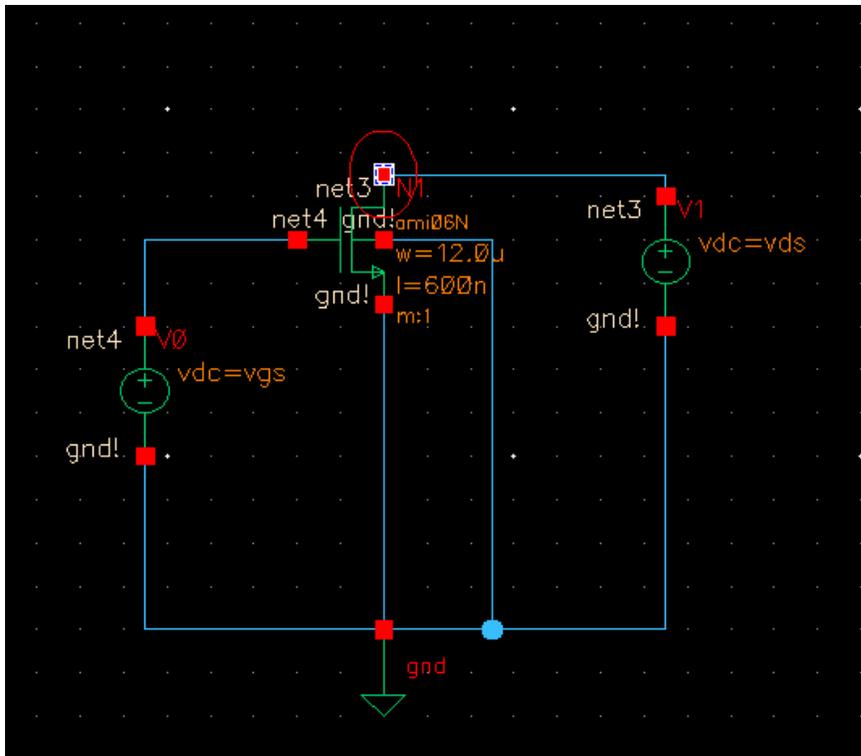
	Name	Value
1	vds	0
2	vgs	0

Two variables you just created will show up, then set up values for them. (These values can be used for one time simulation)

### Step 3

set up **DC analysis** and also **outputs** you want to plot. In this case we want to plot the current going through the NMOS.

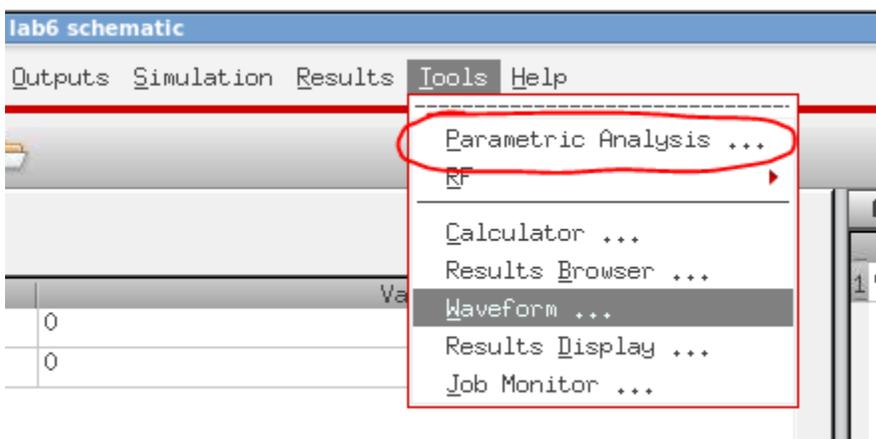


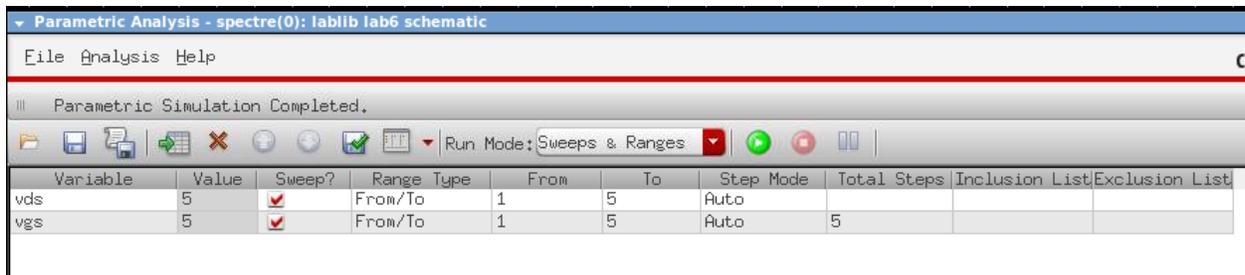


Since we want to plot the current, click on the node to choose current as output (don't click on the net, this is important), then press ESC.

#### Step 4

Set up parametric analysis



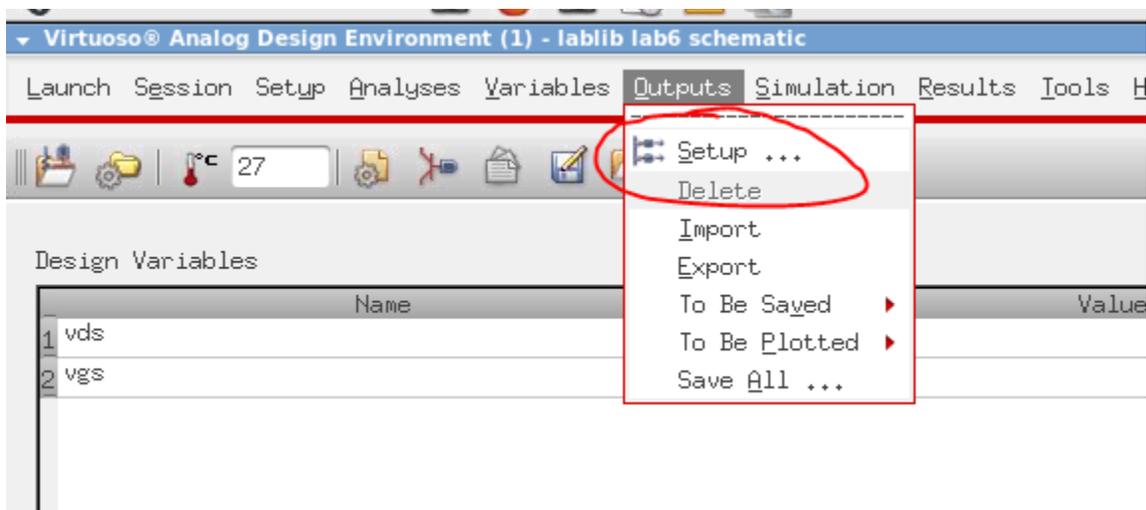


This is my voltage range setting, you may set up as you wish, also play around with step sizes and step mode to see how they affect your simulation. After setup the parametric analysis, click on green play button to run.

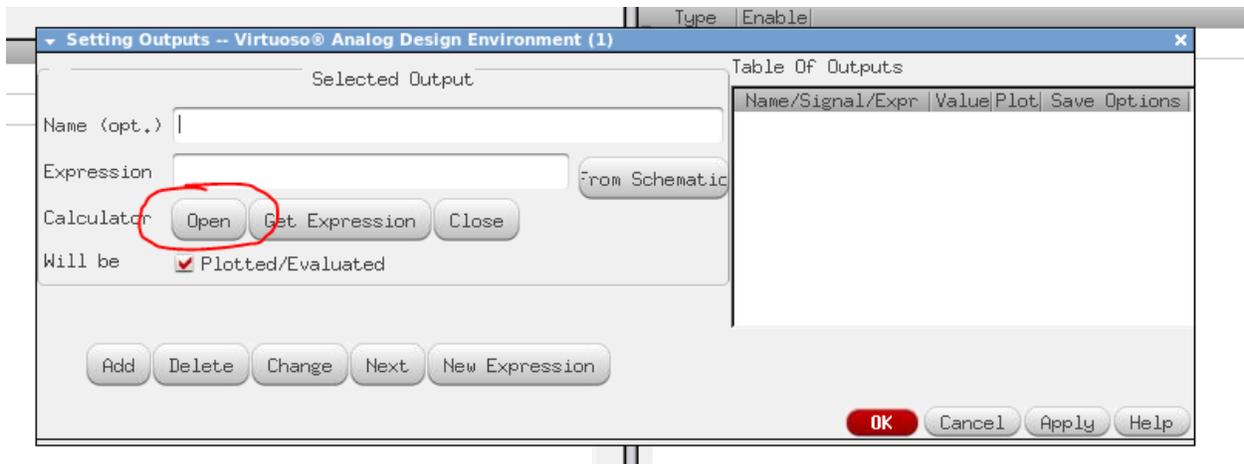
After the simulation window pops up, some of you may have vgs on your X-axis instead of vds, you can fix this by right click the X-axis and name and choose “swap sweep variabl...”.

**Alternative way to add outputs if the simulation result window doesn't pop up**

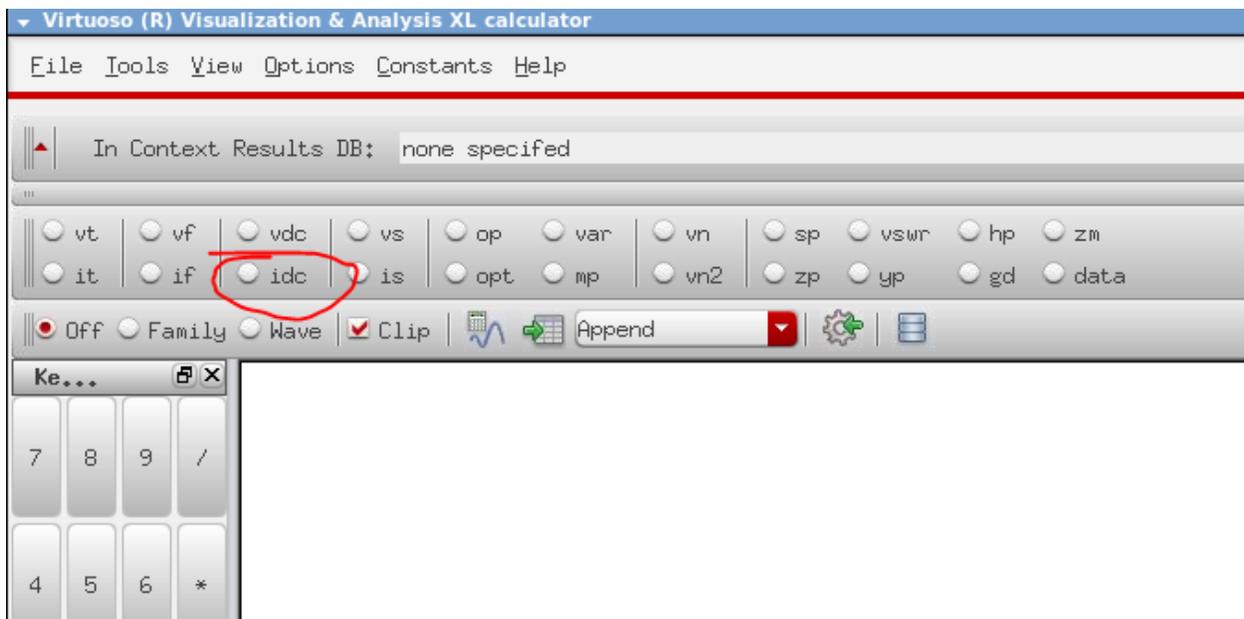
Choose **setup**.



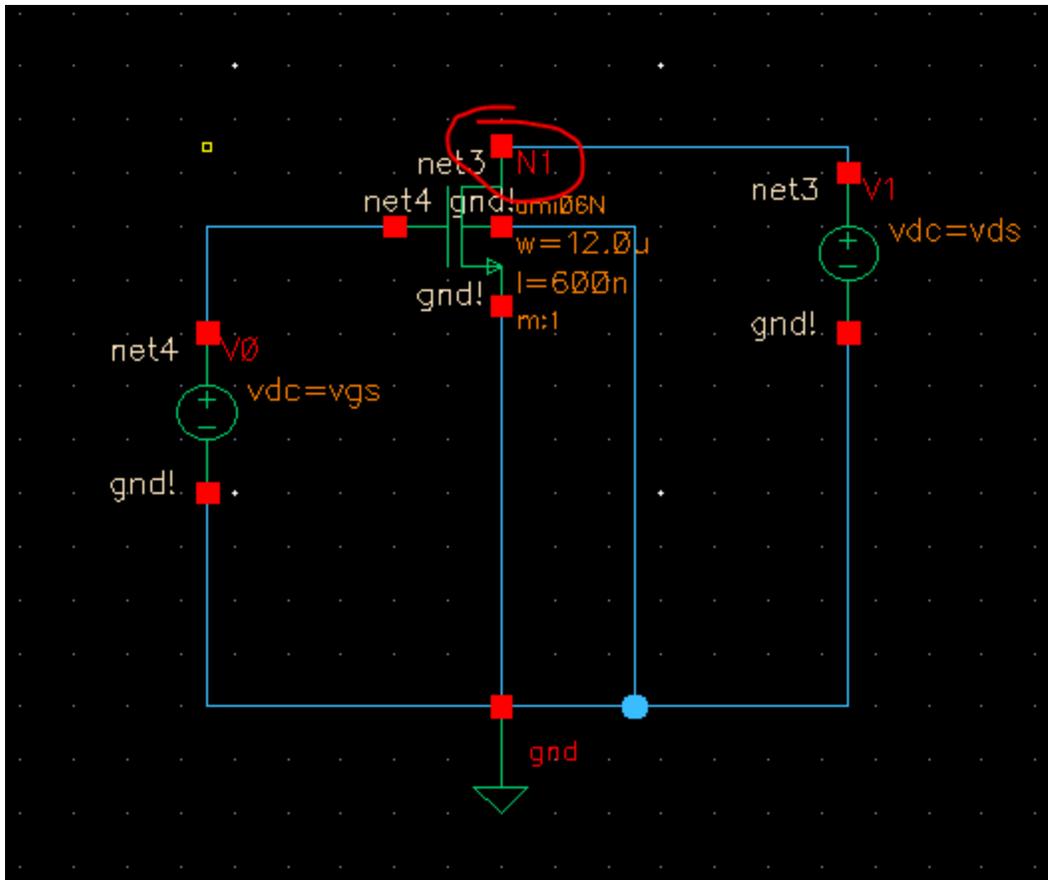
Click on **Open**



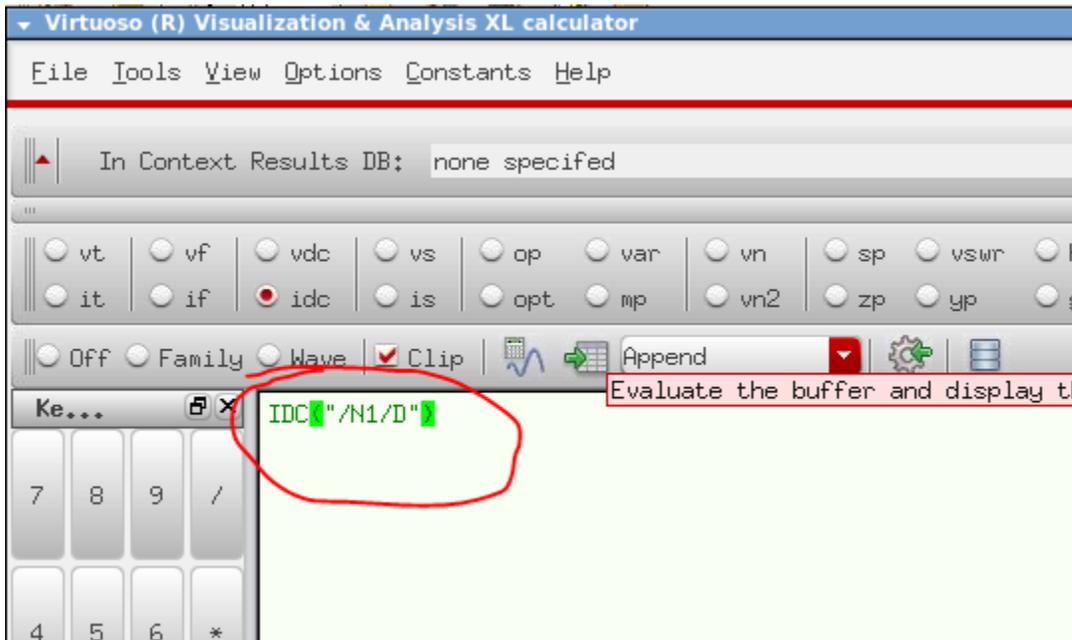
In the new window choose **Idc**



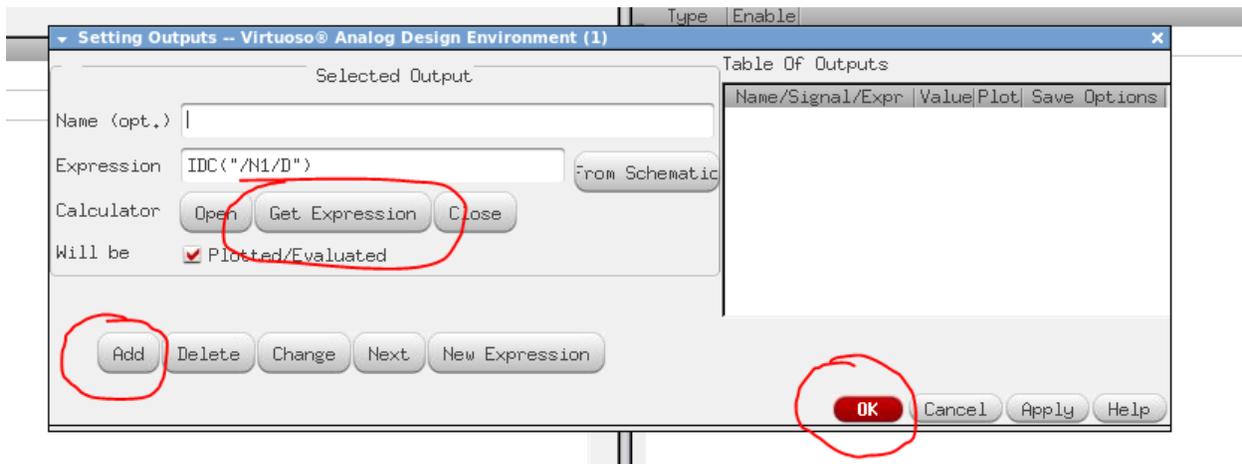
Click **on the node**



Then the current equation will be automatically inserted in the window.



Go back to the setup window. Click “get expression” and click “add”, then click “ok”



Now go back to parametric window and rerun the simulation.