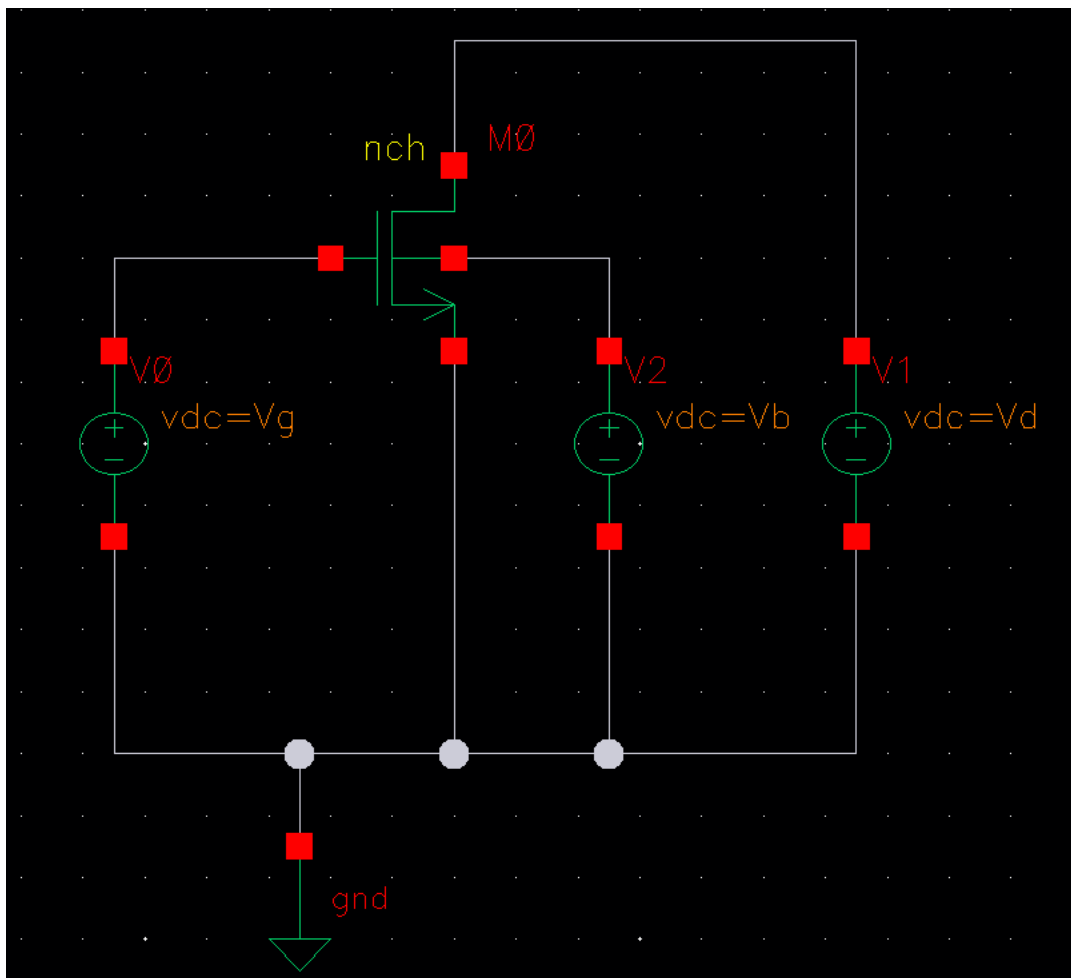


Homework 1 Tips

Problem 2:

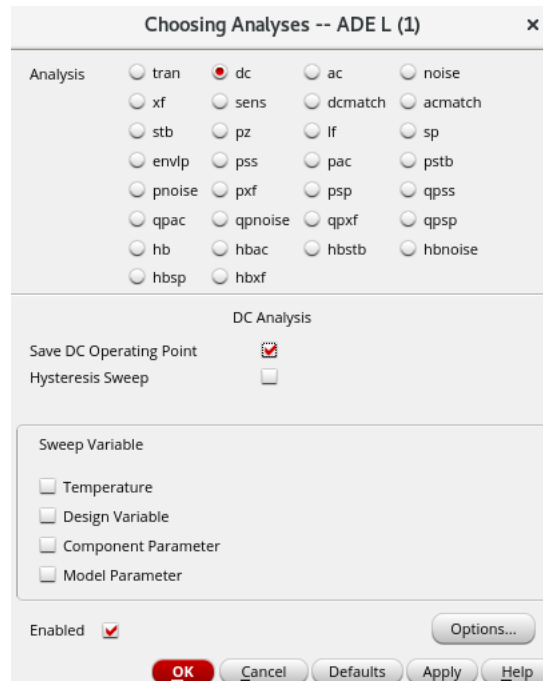
In this tutorial, we aim to teach the process of DC simulation and parameter sweep.

For this problem, you will be testing simple NMOS and PMOS devices, respectively. Here, we show an example to test the behavior of a NMOS device. Add a nmos device and DC voltage source into the schematic. The voltage of three voltage sources are set to V_g , V_b and V_d , respectively.

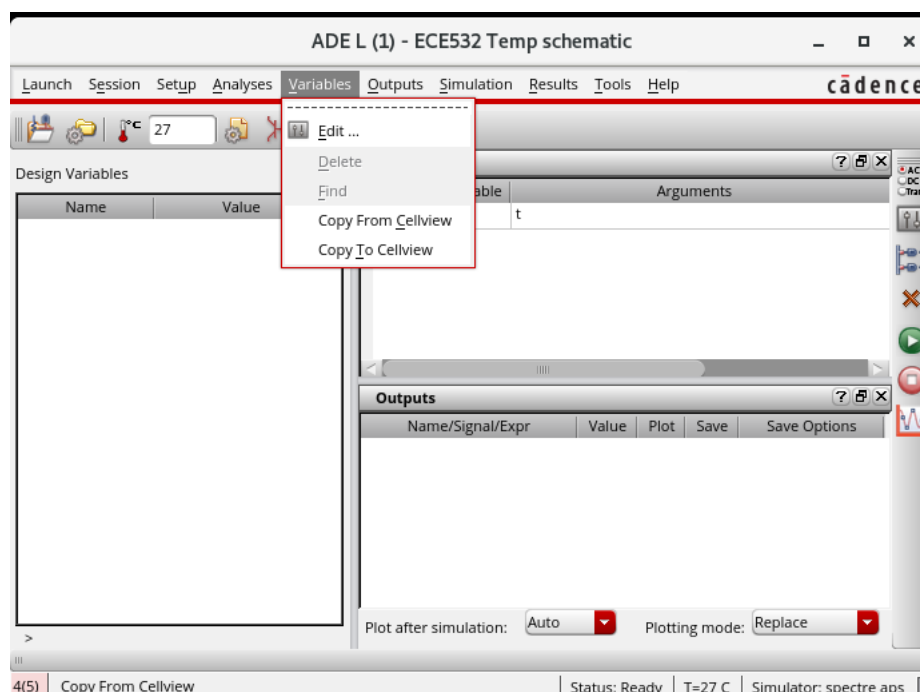


Next, you'll want to open your simulator as shown in the tutorial and sweep the various input voltages to get curves. You will still have to generate the NMOS and PMOS curves by yourself in this assignment.

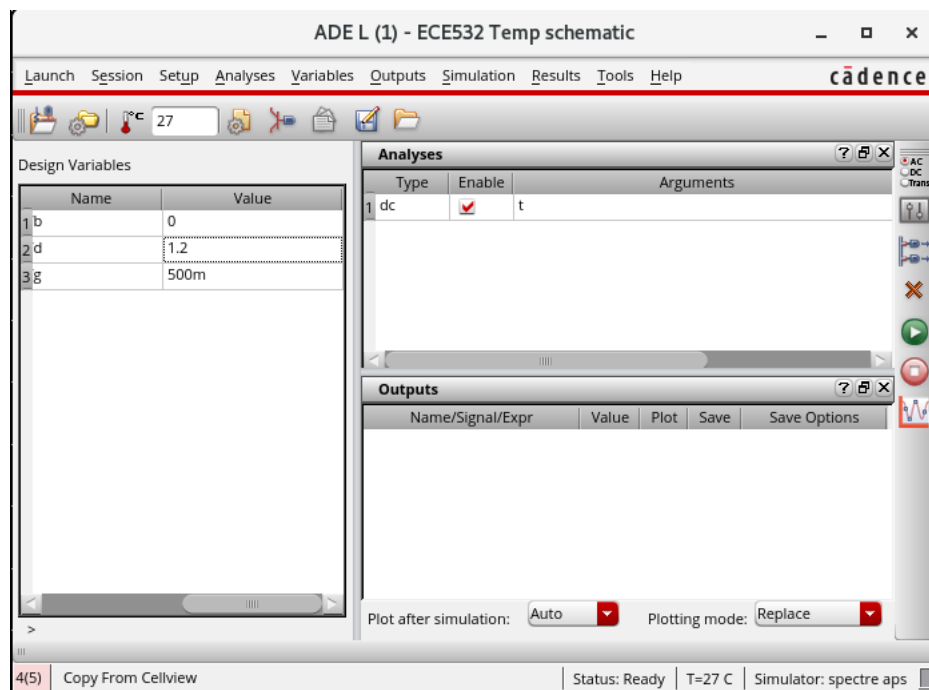
In order to run DC simulation and parameter sweep, you first launch the ADE L and add a DC simulation mode.



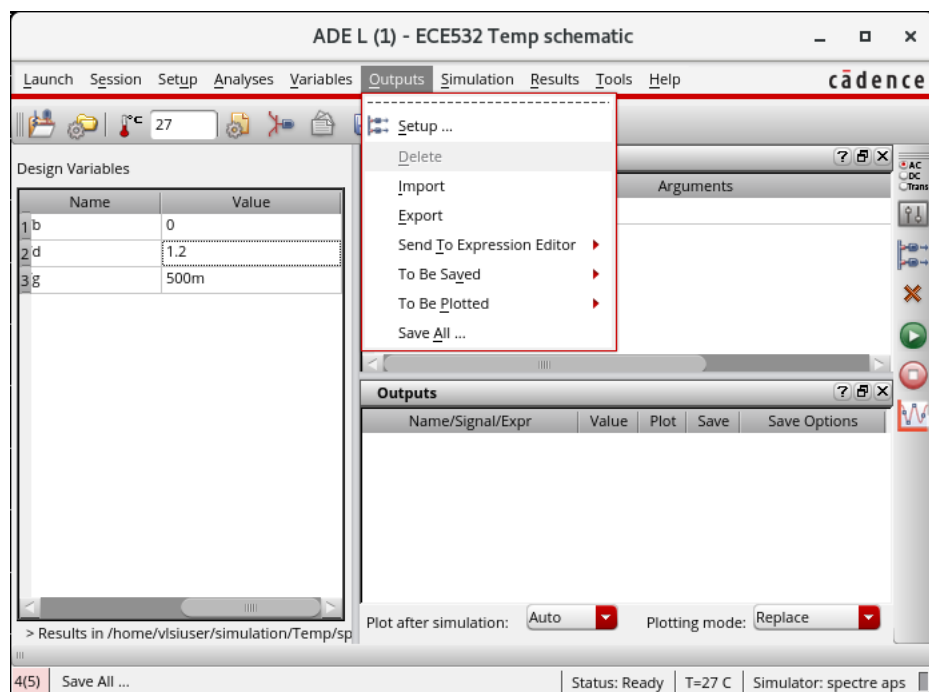
Now, you still cannot simulate your circuit because three voltages are not set. Click Variables and copy from cellview.



Then you can tell the simulator what parameter you want to use for this simulation. Here, we show an example that we set the voltage as a parameter. In practice, you can also set the width/length of the transistor as a parameter.



We are interested in the current flowing in the transistor. However, by default, the simulator will not save these data. So you need to save these data. Click outputs and save all.



Save Options

Basic

Save By Subckt

Save Options

Select signals to output (save)

☐ none
☐ selected
☐ lvpub
☐ lv
☐ allpub
☒ all

Select power signals to output (pwr)

☐ none
☐ total
☐ devices
☐ subckts
☒ all

Set level of subcircuit to output (nestlvl)

Select device currents (currents)

☐ selected
☐ nonlinear
☒ all
☐ none

Select AC terminal currents (useprobes)

☐ yes
☐ no

Subcircuit Probe Level (subcktprobelvl)

Select AHDL variables (saveahdlvars)

☐ selected
☐ all

Transient Time Window Options

☐ Transient time window save options

Time Setup

Save circuit information analysis

Name	What	Where	File	Extremes	Others	Enabled
modelParameter	models	rawfile				<input checked="" type="checkbox"/>
element	inst	rawfile				<input checked="" type="checkbox"/>
outputParameter	output	rawfile				<input checked="" type="checkbox"/>
designParamVals	parameters	rawfile				<input checked="" type="checkbox"/>
primitives	primitives	rawfile				<input checked="" type="checkbox"/>
subckts	subckts	rawfile				<input checked="" type="checkbox"/>
asserts	assert	rawfile				<input type="checkbox"/>
extremeinfo	all	logfile		yes		<input type="checkbox"/>
<Click_To_Add>	none	rawfile				<input type="checkbox"/>

Detail: node

Sort: name

Threshold:

Output Options

Output Format

☐ sst2
☐ psf
☐ psf with floats
☒ psfxl
☐ fsdb

OK

Cancel

Defaults

Apply

Help

Then you can run the simulation. Once the simulation is done, you can see the DC simulation output by clicking result-> print -> DC operating points. Then, you can click the nmos.

Results Display Window	
Window Expressions Info Help	
cadence	
signal	OP("/M0" "??")
self_gain	11.13
trise	0
ueff	19.32m
vbs	0
vdb	1.2
vds	1.2
vdsat	141.3m
vdsat_marg	NaN
vdss	141.3m
vearly	1.626
vgb	500m
vgd	-700m
vgs	500m
vgt	161.1m
vsat_marg	1.059
vsb	-0
vth	338.9m
vth_drive	NaN
signal	OP("/M0" "??")
beff	2.404m
betaeff	1.758m
cbb	12.78a
cbd	-816.8z
cbdbo	-816.8z
cbg	-25.68a
cbgbo	-25.68a
cbs	13.72a
cbsbo	13.72a
cdb	-5.572z
cdd	43.28a
cddbo	11.56z
cdg	-43.3a
cdgbo	36.35z

You will get this table of the DC simulation for this device. Now, we are interested in how the output changes when we change the design variables.

Choosing Analyses -- ADE L (1)

Analysis

<input type="radio"/> tran	<input checked="" type="radio"/> dc	<input type="radio"/> ac	<input type="radio"/> noise
<input type="radio"/> xf	<input type="radio"/> sens	<input type="radio"/> dcmatch	<input type="radio"/> acmatch
<input type="radio"/> stb	<input type="radio"/> pz	<input type="radio"/> lf	<input type="radio"/> sp
<input type="radio"/> envlp	<input type="radio"/> pss	<input type="radio"/> pac	<input type="radio"/> pstb
<input type="radio"/> pnoise	<input type="radio"/> pxf	<input type="radio"/> psp	<input type="radio"/> qpss
<input type="radio"/> qpac	<input type="radio"/> qpnoise	<input type="radio"/> qpxf	<input type="radio"/> qpss
<input type="radio"/> hb	<input type="radio"/> hbac	<input type="radio"/> hbstb	<input type="radio"/> hbnoise
<input type="radio"/> hbsp	<input type="radio"/> hbxf		

DC Analysis

Save DC Operating Point ☒

Hysteresis Sweep ☐

Sweep Variable

☐ Temperature

☒ Design Variable

☐ Component Parameter

☐ Model Parameter

Variable Name

Select Design Variable

Sweep Range

☒ Start-Stop

Start Stop

☐ Center-Span

Sweep Type

Linear

☐ Step Size

☒ Number of Steps

Add Specific Points ☐

Add Points By File ☐

Enabled ☒

Options...

OK Cancel Defaults Apply Help

Change the setup of DC simulation. In this time, we would like to see how the output changes when V_g changes. Re-run the simulation.

Direct Plot Form

Plotting Mode Append

Analysis

☒ dc

Function

☐ Voltage

☒ Current

☐ Power

☐ Transresistance

☐ Voltage Ratio

☐ Current Ratio

☐ Power Ratio

☐ Transconductance

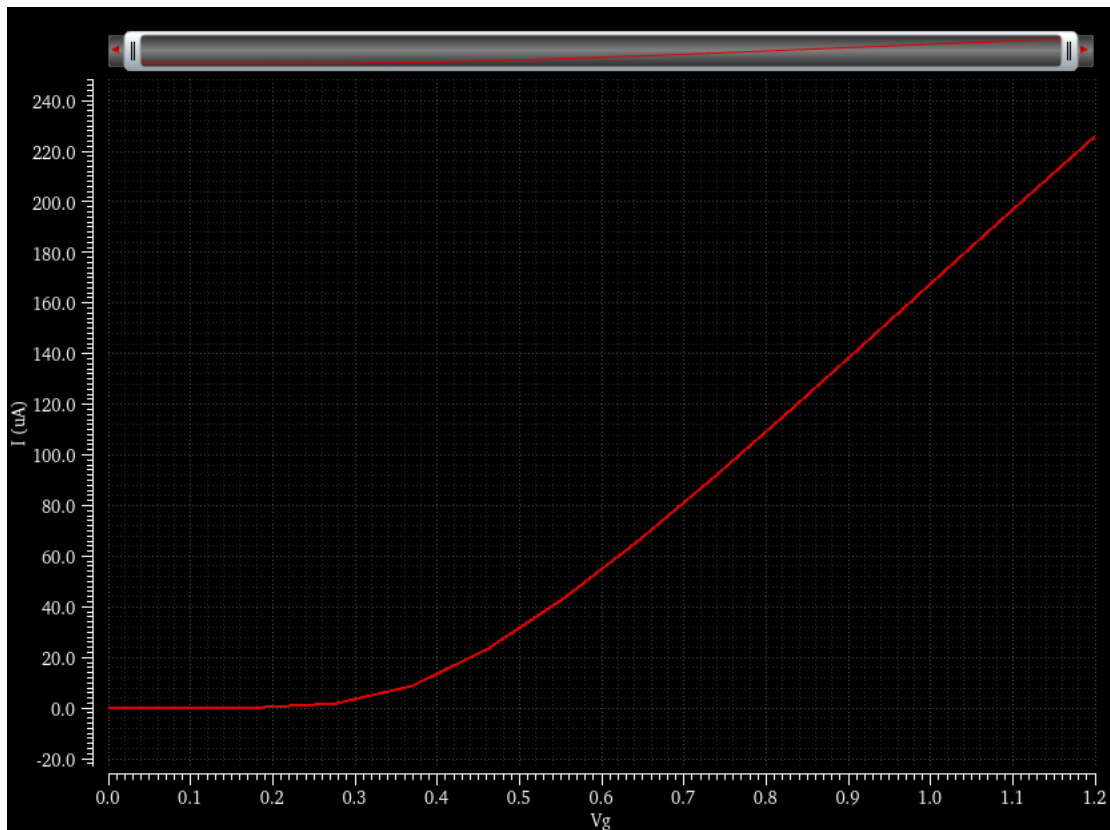
Select Terminal

Add To Outputs ☒

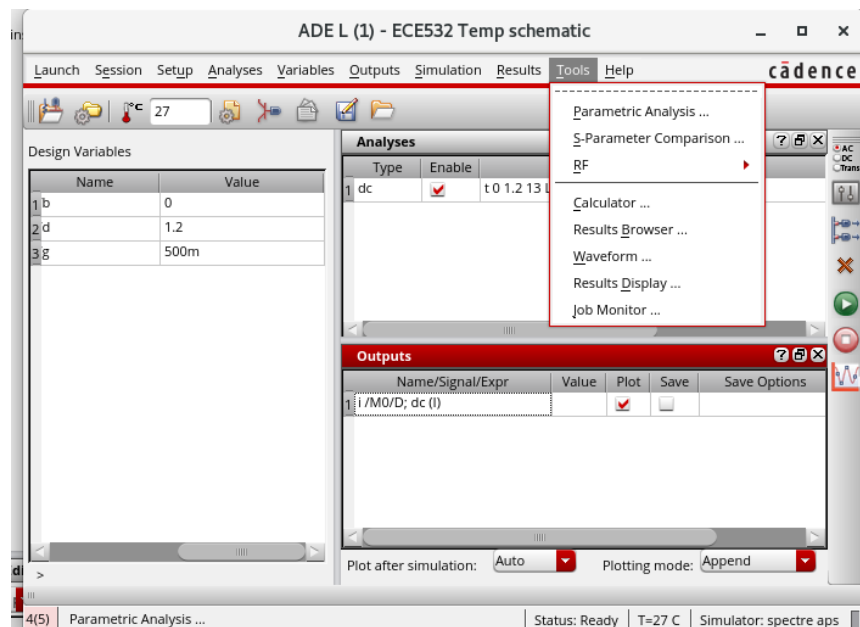
> Select Instance Terminal on schematic...

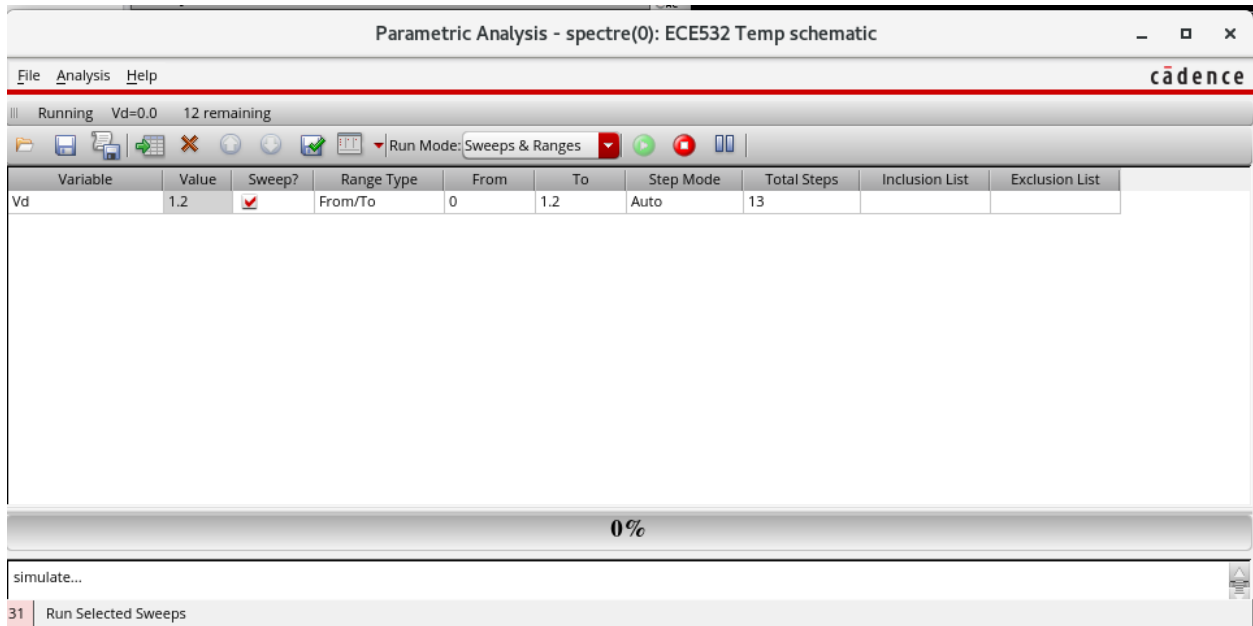
Close Help

After the simulation, use Direct plot, main form to print the current. Click the drain of the NMOS device.

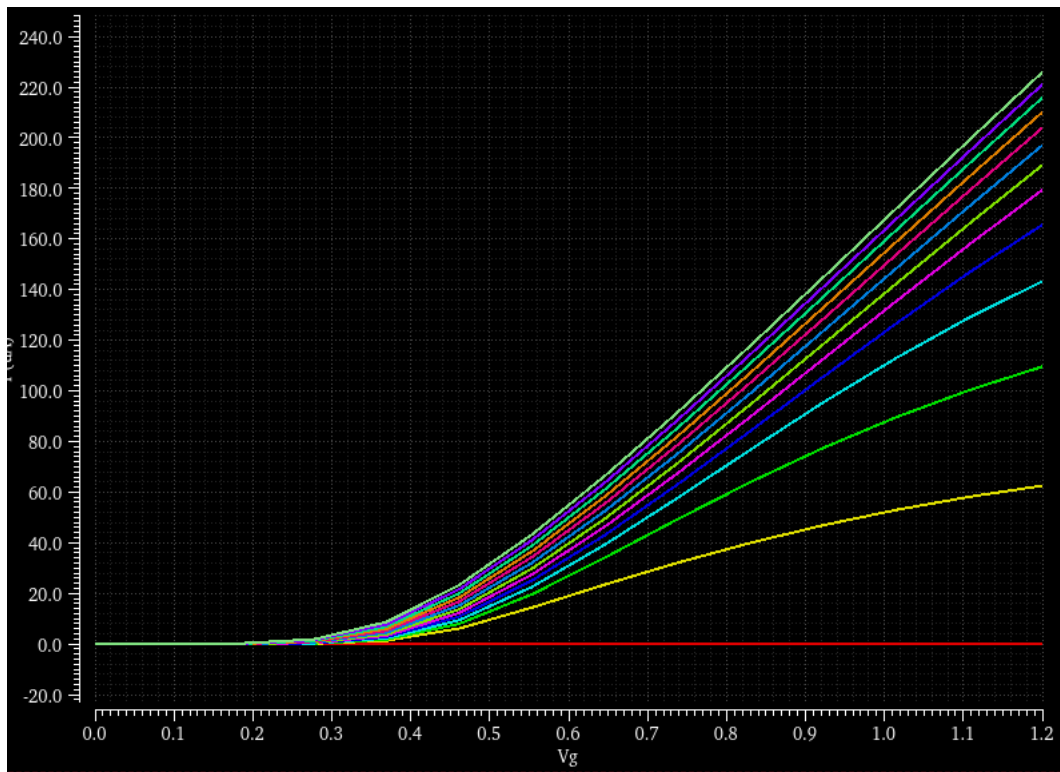


Here is the I_d - V_g waveform. Now we are interested in how to generate multiple I_d - V_g waveforms in one plot with different V_d . Click tool -> parameter analysis.

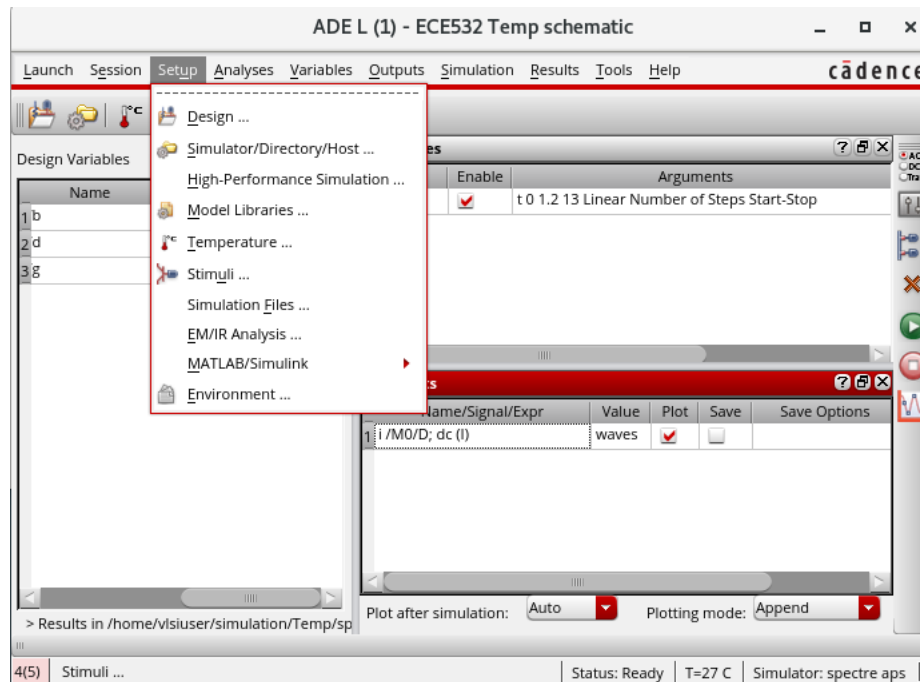




The simulation will give you multiple curve with different Vd.



Now we want to change the model of the device to see the difference between typical, best and worst case. Choose Setup -> Model Libraries.



Find the session of tt, and change the tt to ss, the re-do the simulation.

