

Cadence Virtuoso

Introductory Tutorial

ELECTGON
www.electgon.com
contact@electgon.com

24.05.2021



Abstract

For people who might be interested in having overview about Cadence Virtuoso and how it can be used for building and simulating electronics circuits, this document shows how to work with Virtuoso for very simple differential pair circuit.

1 Launching Virtuoso

To Open Virtuoso, the path to its binary or executable file shall be defined in your environment variables or ask your system administrator how to start it.

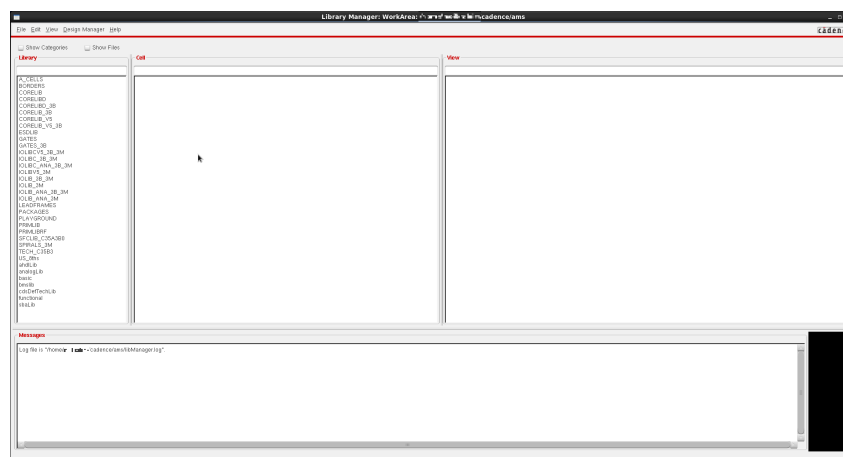
Virtuoso shall be opened as follows



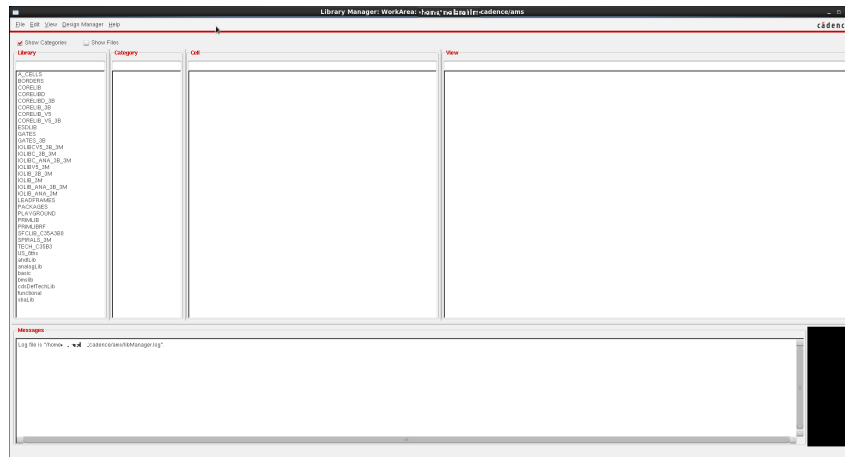
Log screen will appear also



Then Library Manager window will open also as follows

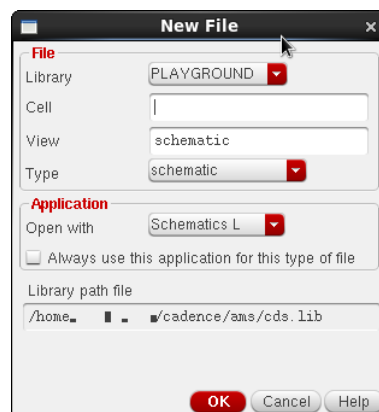


Initially, No categories pane is displayed. To display it, just check on 'Show Category' field in top of window

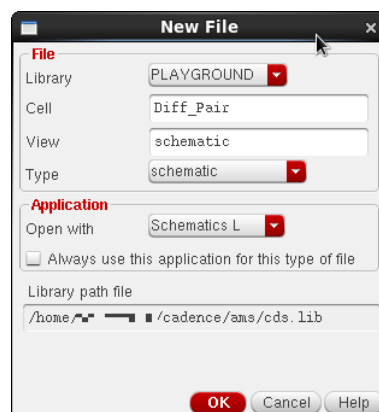


Now to start working on your project, you can create your own library. In this tutorial, our working library is 'Playground'.

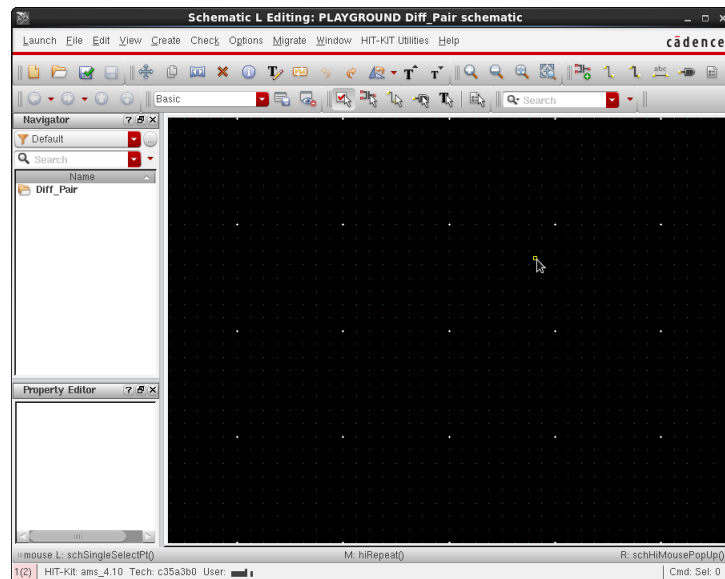
We have to create now design cells in the project. To create your design go to File>>New>>Cell View. The following dialog shall be opened.



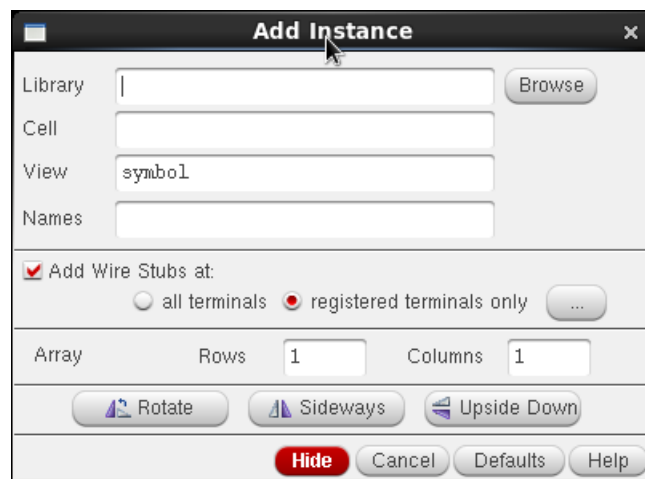
In this step we can name our cell as Diff_Pair



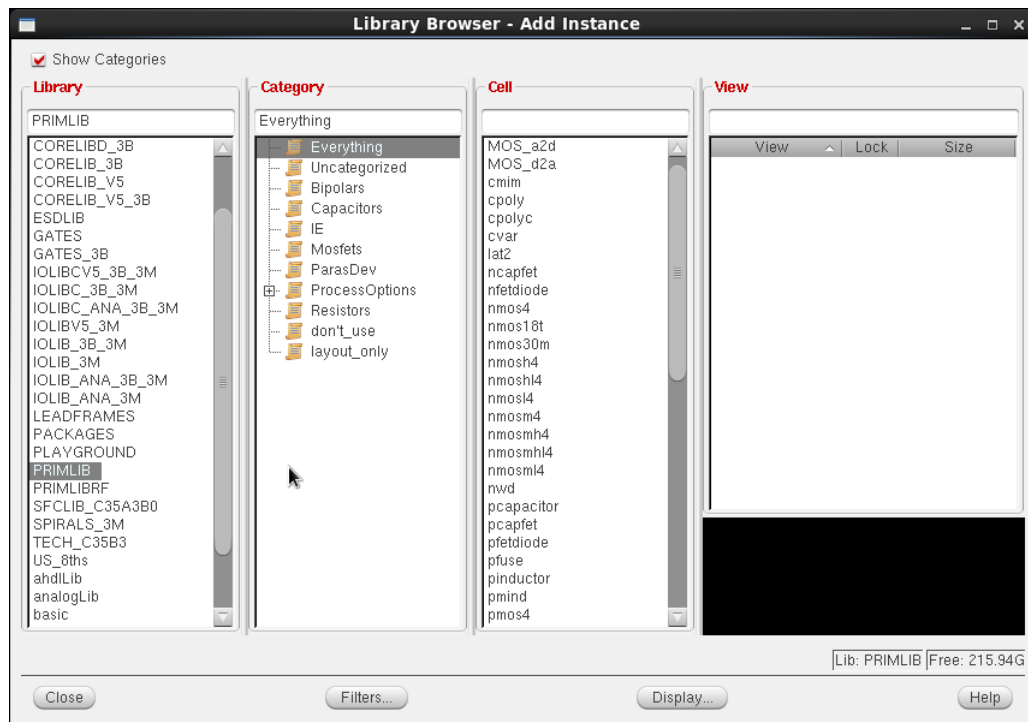
Click OK, the following schematic shall open



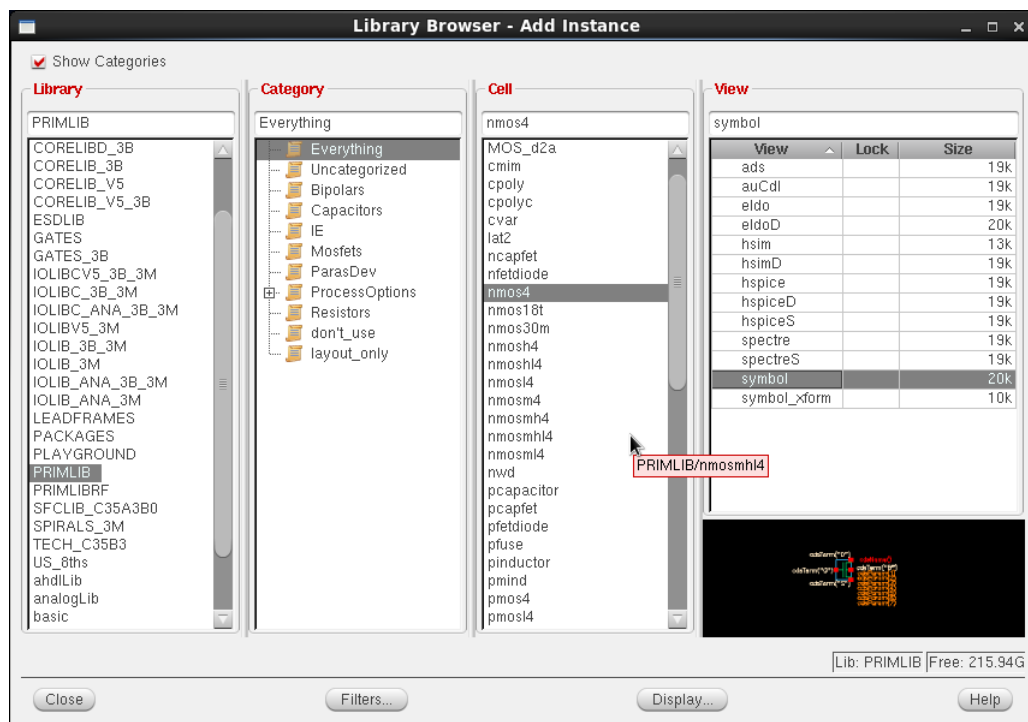
To start adding your components, on top bar click on Create>> Add Instance. The following dialog shall open



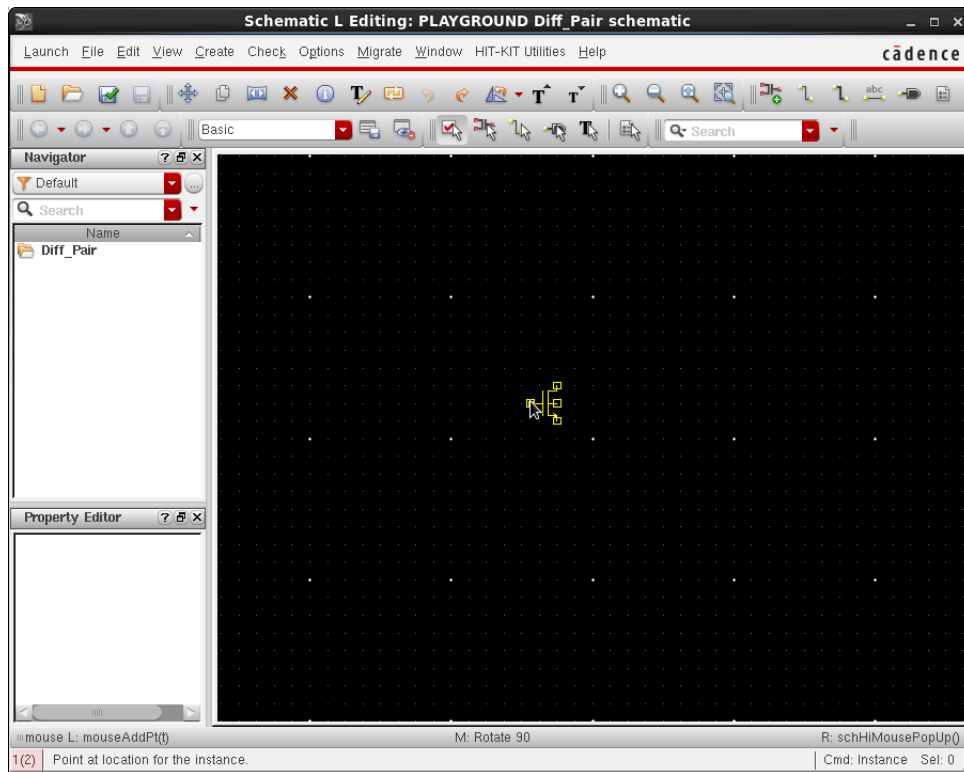
This dialog enables you getting ready components in Cadence libraries, to choose ready component click on Browse. The following dialog shall open



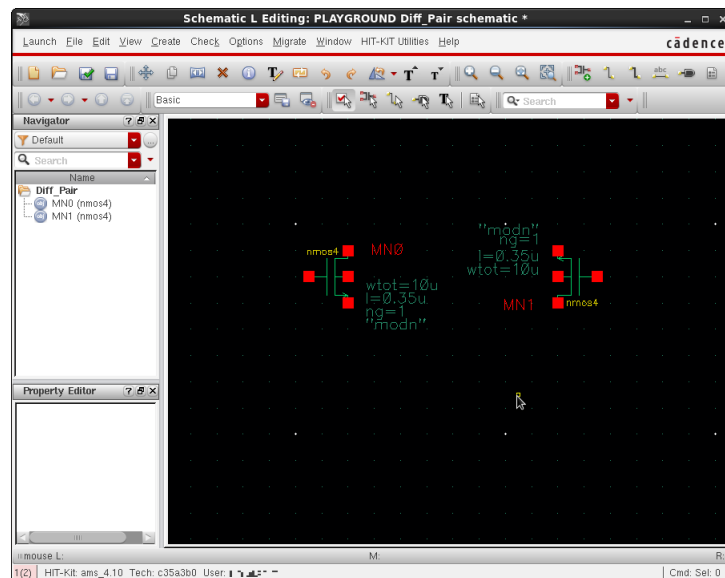
These are all libraries defined in Cadence. To pick up a MOSFET transistor, it is found in PRIMLIB We choose view as 'Symbol' to use it in our schematic



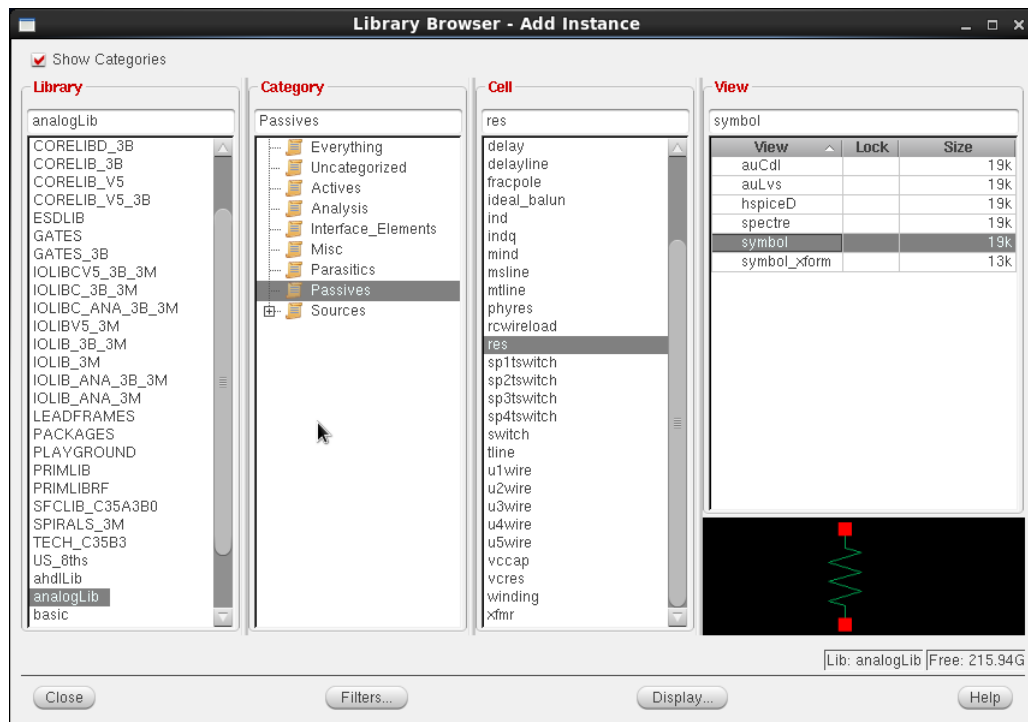
Then your schematic window is ready now for adding this instance



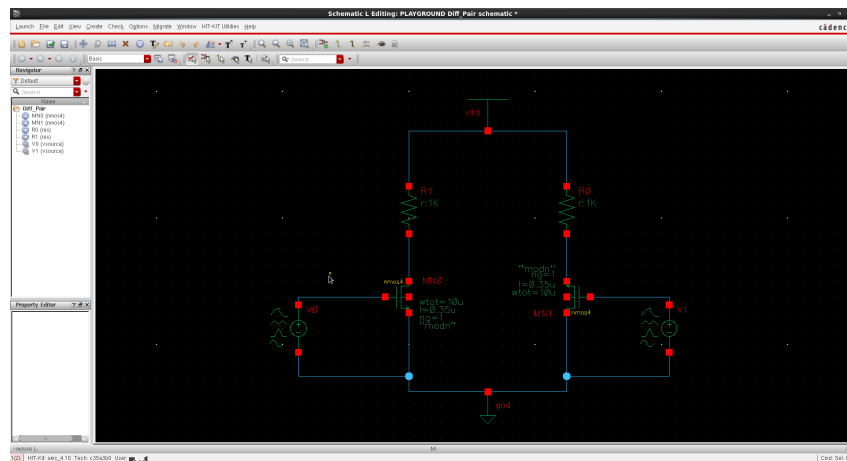
Simply add another transistor. To rotate it just hit the 'R' button in your keyboard.



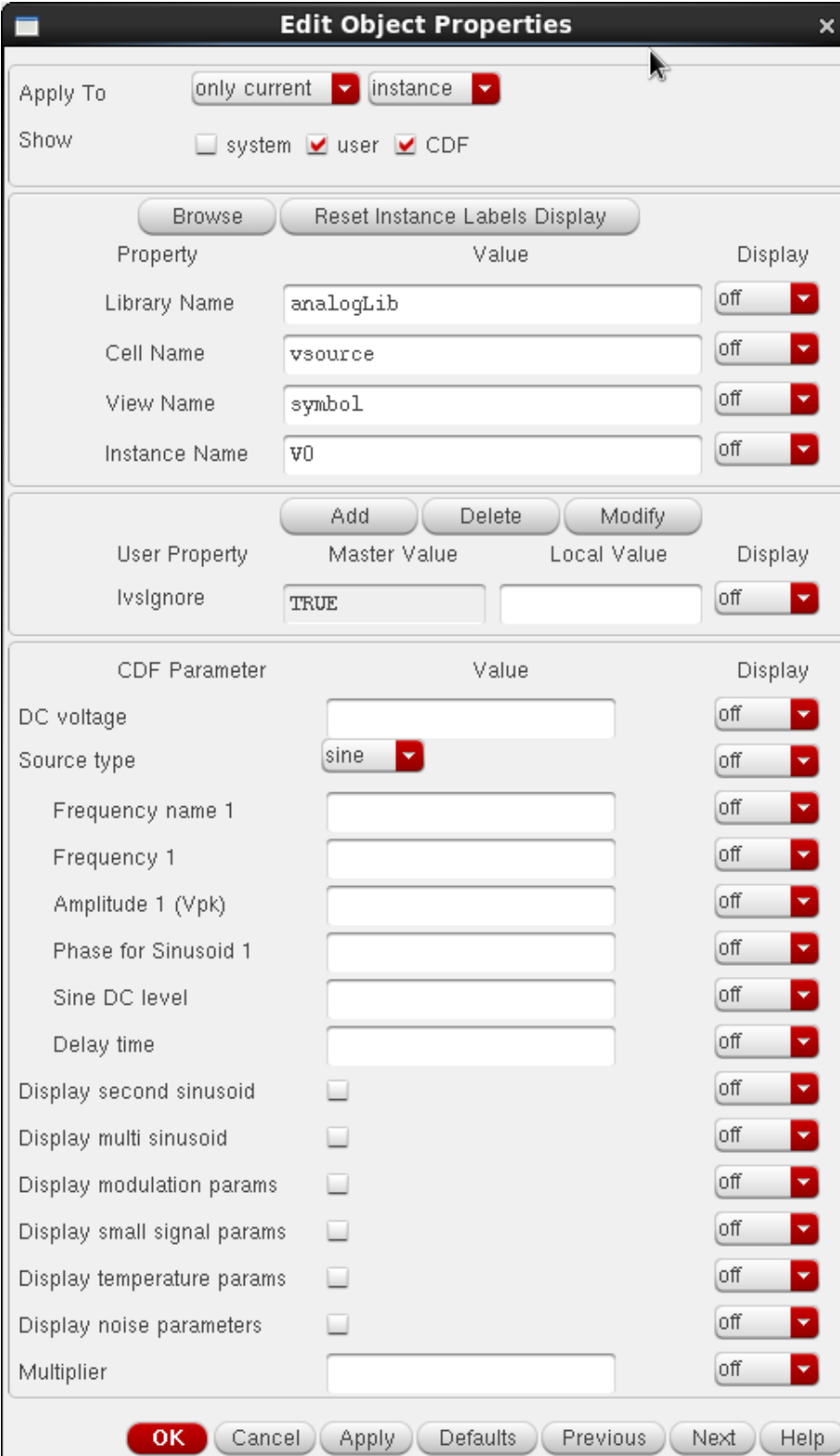
We need now to add a resistor. This can be found in library 'analog_lib' as shown below



You can find also other needed instances. You can complete your design now as follows



Now we need to define our components values. You can do that by highlighting the component then hit 'Q' button on your keyboard. Or right click on the component then choose Edit Object properties.



The **Edit Object Properties** dialog box is used to configure the properties of a selected object. It is organized into several sections:

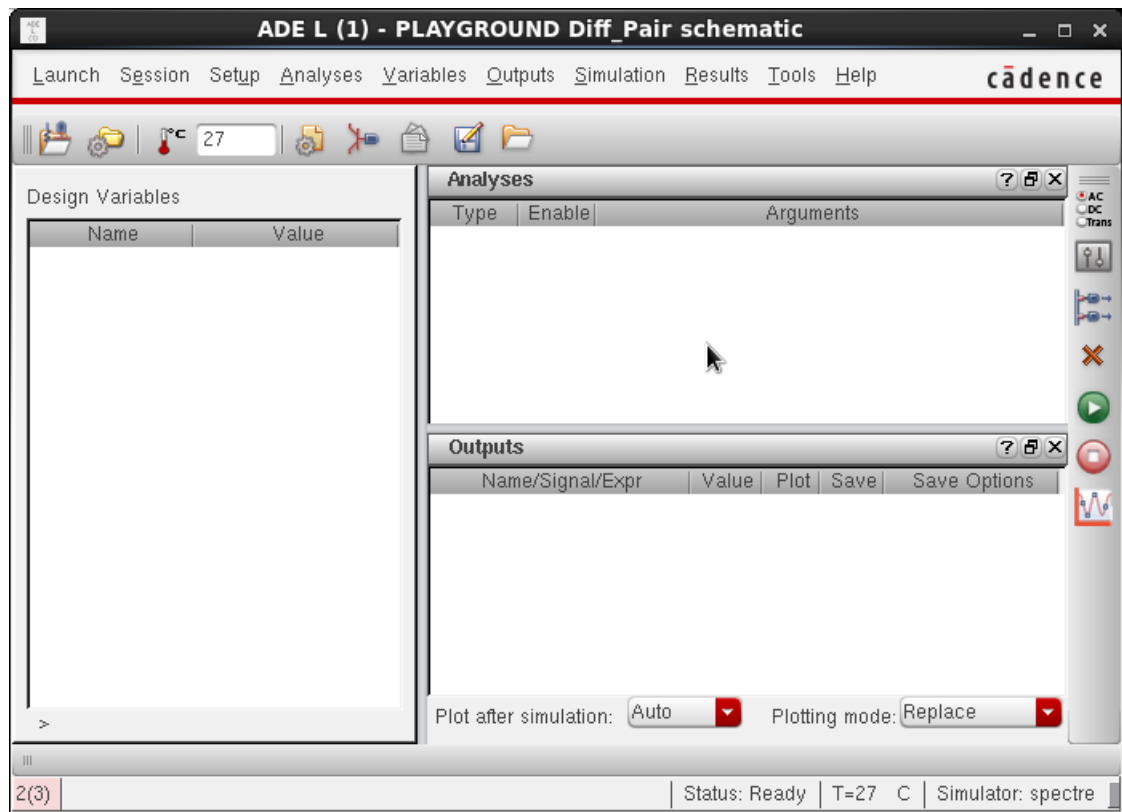
- Apply To:** ☒ only current ☒ instance
- Show:** ☐ system ☒ user ☒ CDF
- Buttons:** Browse, Reset Instance Labels Display
- Property Table:**

Property	Value	Display
Library Name	analogLib	off
Cell Name	vsource	off
View Name	symbol	off
Instance Name	V0	off
- User Property Table:**

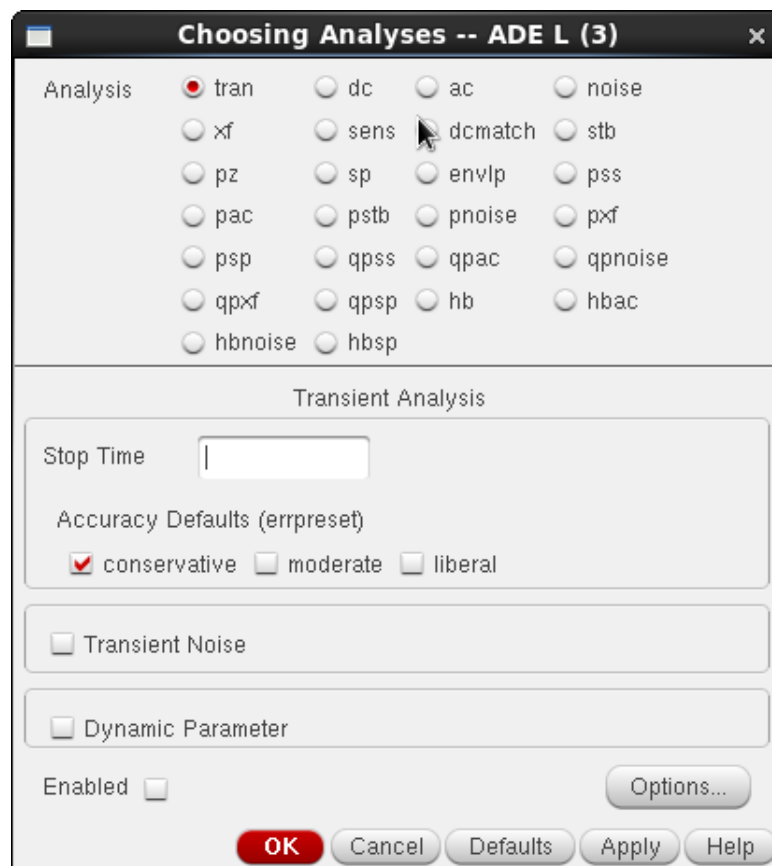
User Property	Master Value	Local Value	Display
IvsIgnore	TRUE		off
- CDF Parameter Table:**

CDF Parameter	Value	Display
DC voltage		off
Source type	sine	off
Frequency name 1		off
Frequency 1		off
Amplitude 1 (Vpk)		off
Phase for Sinusoid 1		off
Sine DC level		off
Delay time		off
Display second sinusoid	<input type="checkbox"/>	off
Display multi sinusoid	<input type="checkbox"/>	off
Display modulation params	<input type="checkbox"/>	off
Display small signal params	<input type="checkbox"/>	off
Display temperature params	<input type="checkbox"/>	off
Display noise parameters	<input type="checkbox"/>	off
Multiplier		off
- Buttons:** OK, Cancel, Apply, Defaults, Previous, Next, Help

Now, your design is ready for simulation. To start simulation, click in “Launch>>ADE L” in the top of your schematic window

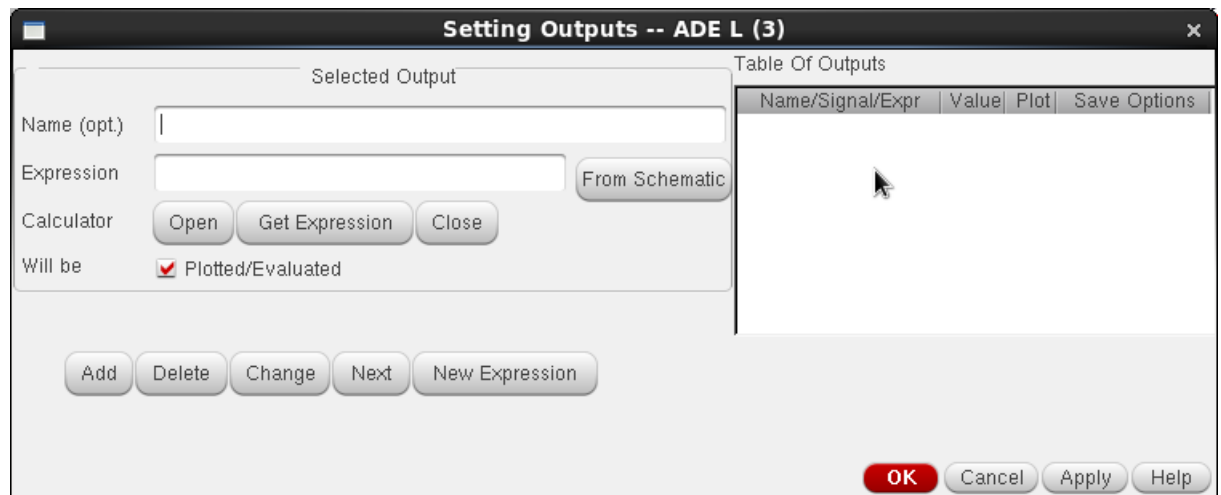


- To work on simulation, you have to define type of your analysis (DC, AC, Trans. Noise, ...)
- Click on Analysis>>Choose. This shall bring the following dialog

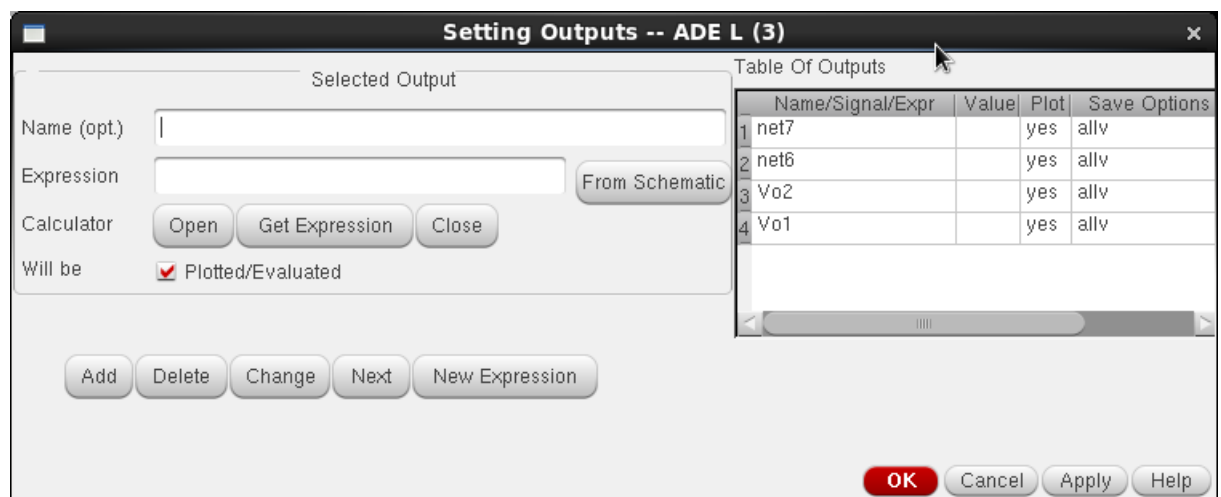


For our example here we will work on Transient Analysis, so provide '1u' in stop time field. Then Click OK

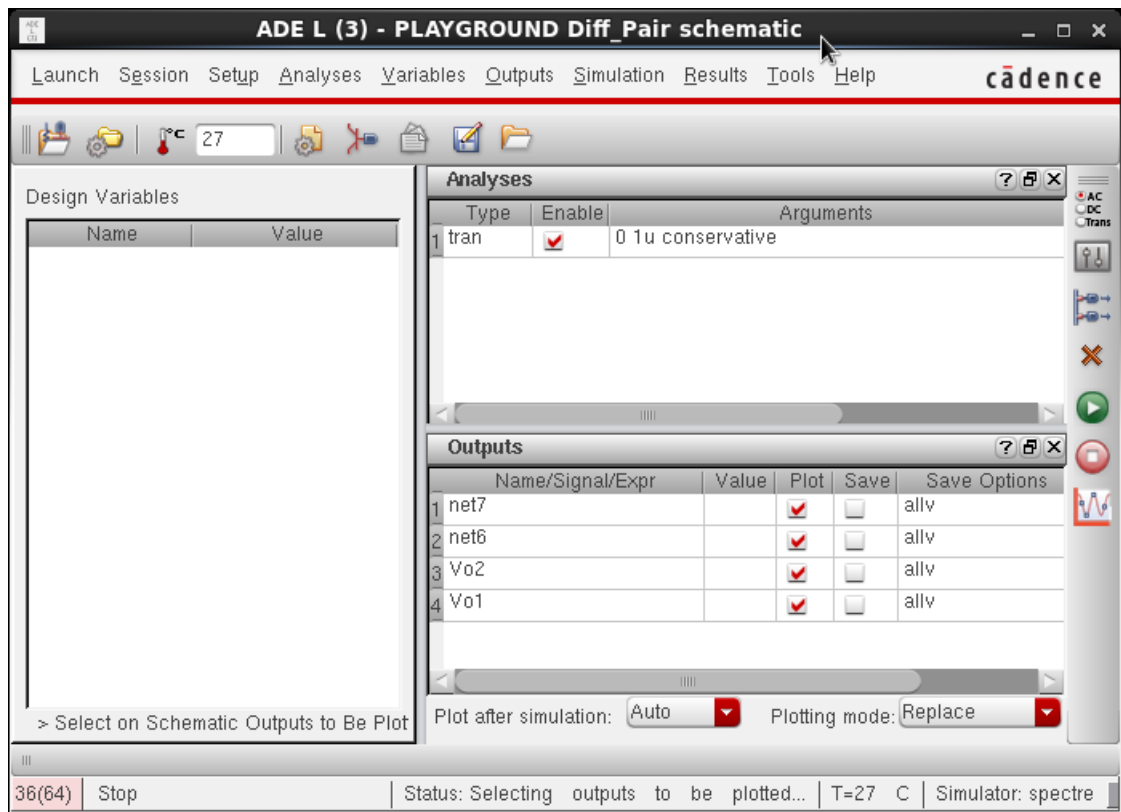
Now in your ADE L window, click on Outputs>>Setup. This step is needed to define which signals we need to observe. The following window shall be opened.



Instead of capturing signal names from your schematic, you can pick it by clicking on 'From Schematic' button. This shall activate your schematic window to choose it. Just single click by mouse on signals that you want to observe. Now your 'Setting Outputs' will be updated with these signals.



Also your ADE L window will be updated with these signals

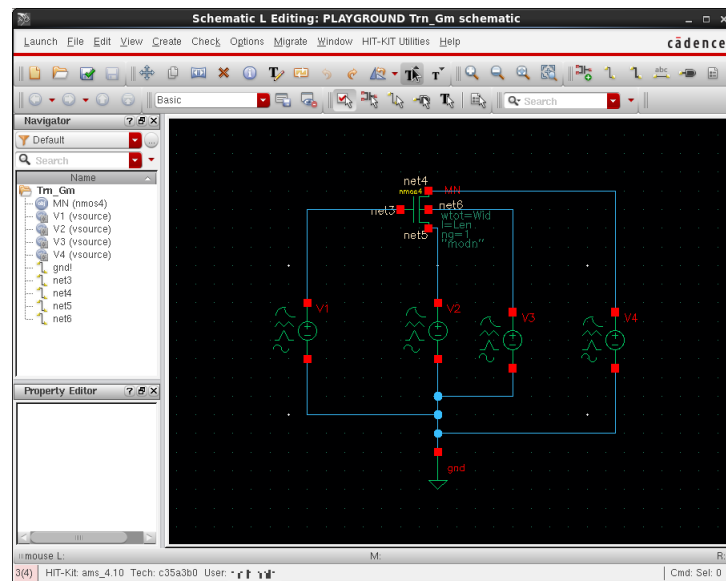


Now you can run your simulation by clicking green button run, Now you can see your results

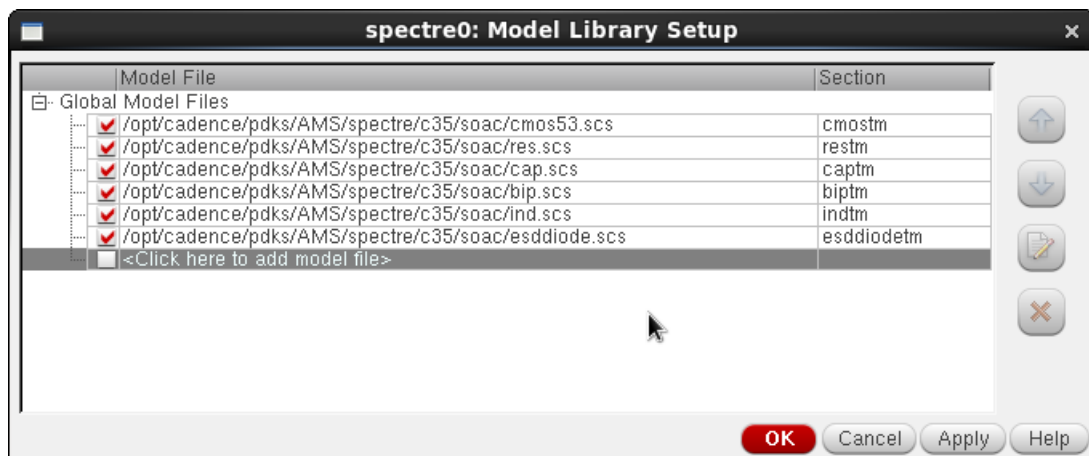


2 Adding Device Parameters

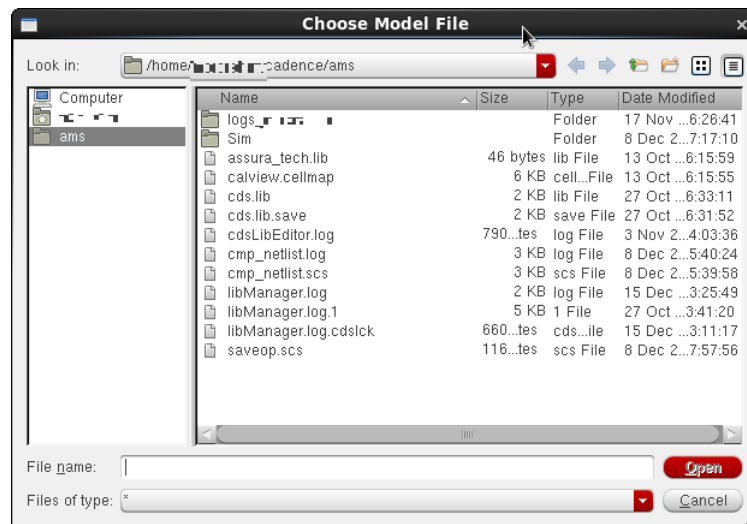
Now assume that we have a design like the following:



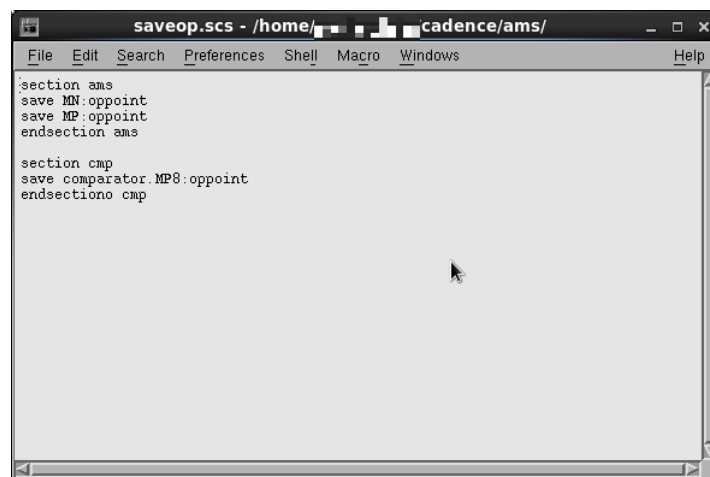
We aim currently at adding parameters of used MOSFET in our simulation results. Open ADE L window (Launch>>ADE L), from ADE L window click on toolbar 'Setup>> Model Libraries ..' the following window shall appear



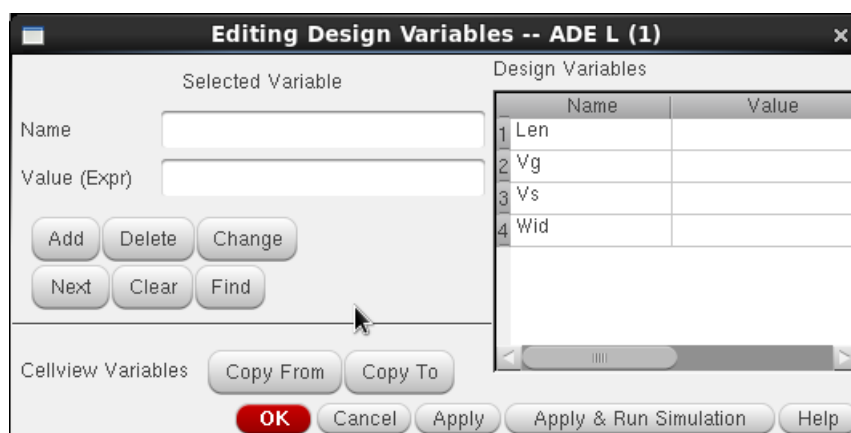
In this window, tick on 'click here to add Model file', then browse to your file location



your file should be written like that



Then in Model Library Setup window, click on section to choose ams section from this file
Now start your simulation in ADE L, add simulation paramters (click on copy from button to add these paramteres from schematic)



Add simulation type

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"/> stb
<input type="radio"/> pz	<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"/> qqsp	<input type="radio"/> hb	<input type="radio"/> hbac
<input type="radio"/> hbnoise	<input type="radio"/> hbsp		

DC Analysis

Save DC Operating Point ☒

Hysteresis Sweep ☐

Sweep Variable

☐ Temperature

☒ Design Variable

Variable Name

☐ Component Parameter

☐ Model Parameter

Select Design Variable

Sweep Range

☒ Start-Stop

Start Stop

☐ Center-Span

Sweep Type

Automatic ☒

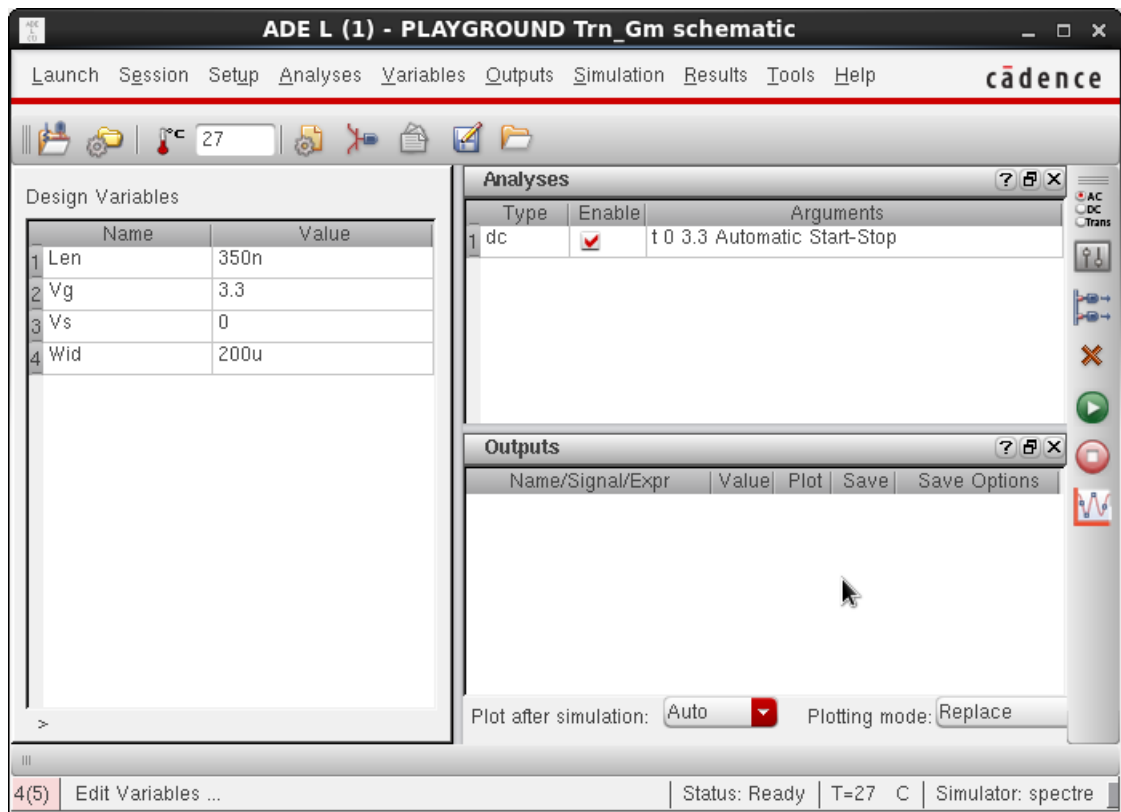
Add Specific Points ☐

Enabled ☒

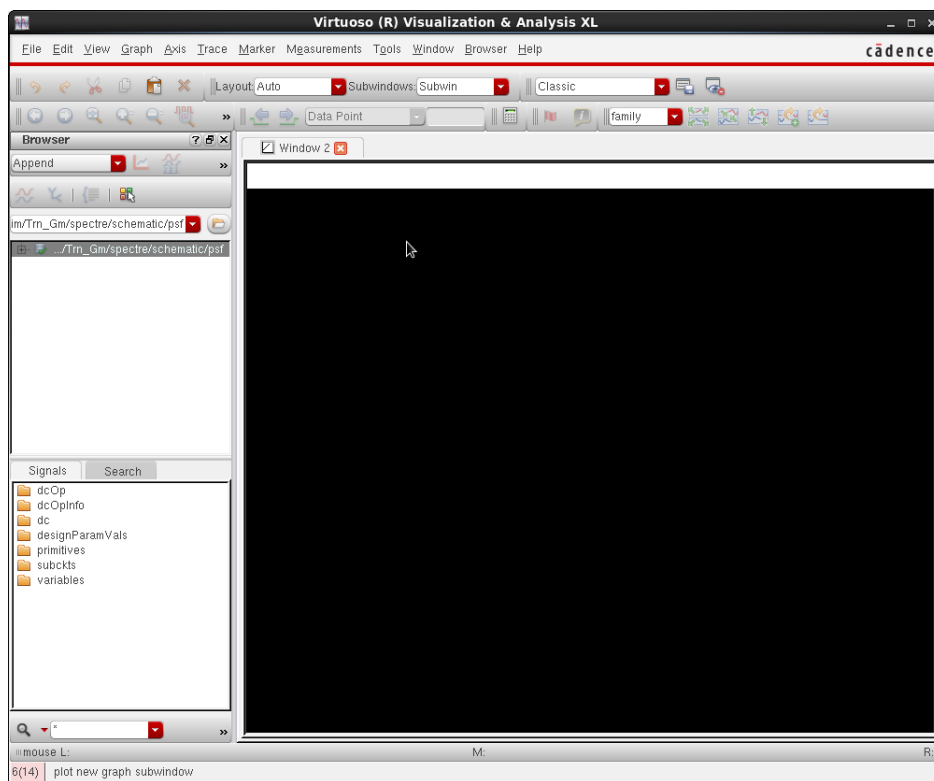
Options...

OK Cancel Defaults Apply Help

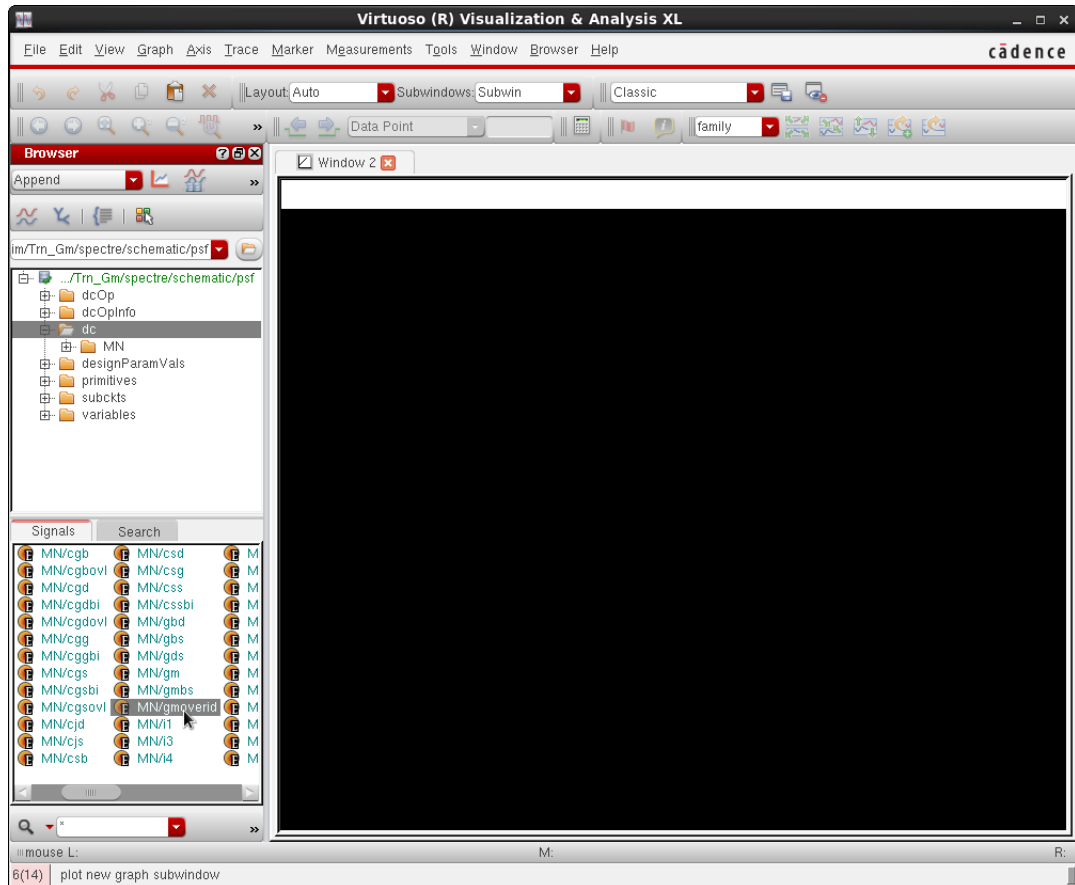
Run



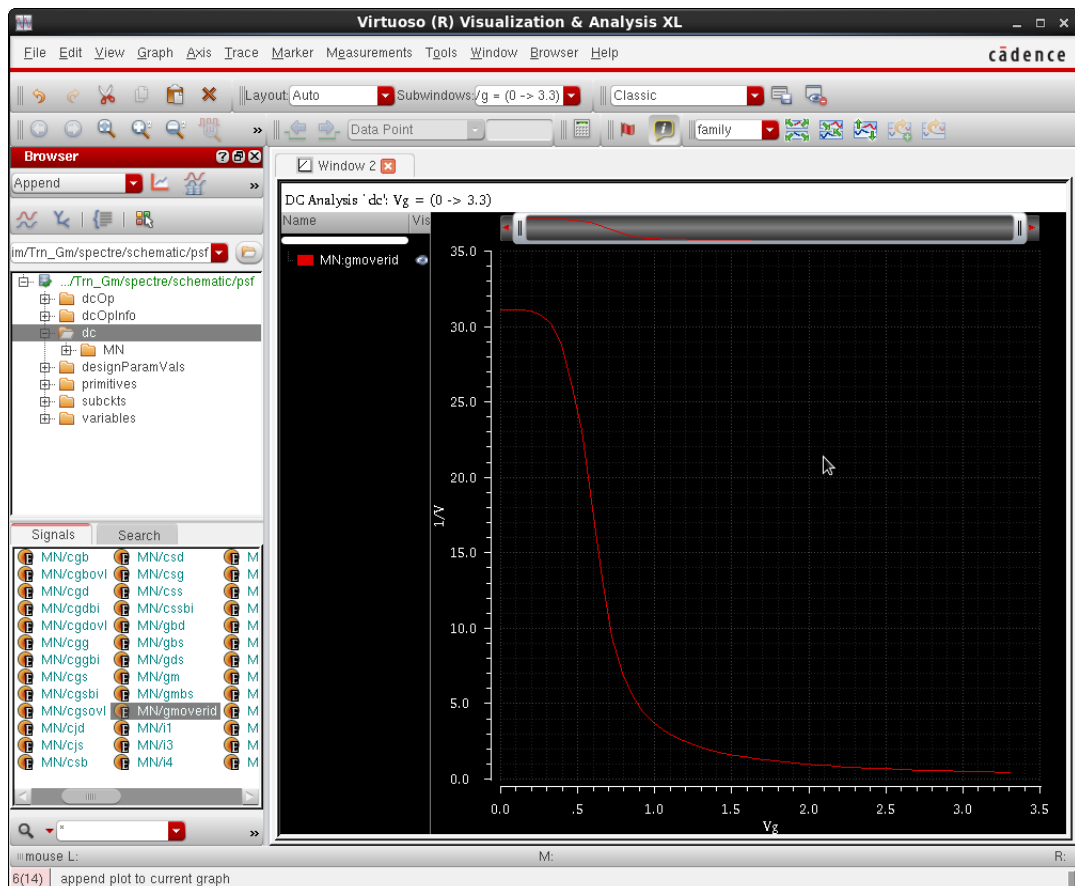
Now in ADE L window choose 'Tools>Result Browser' it shall open like the following



In dc folder find your parameter that you want to check in simulation, in our case here we need to check Gm/Id

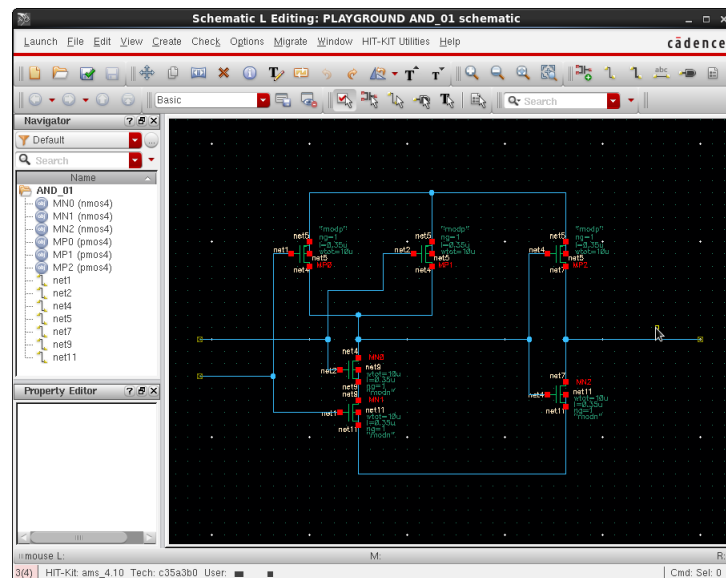


Drag and drop

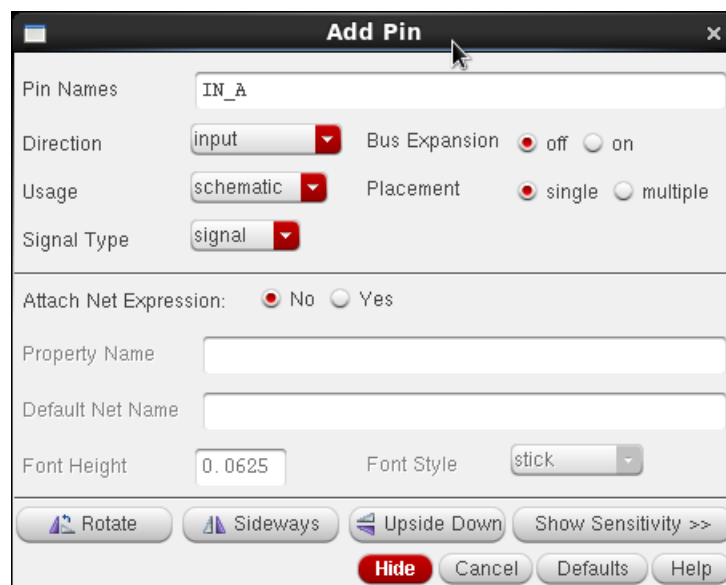


3 Creating Symbols of a design

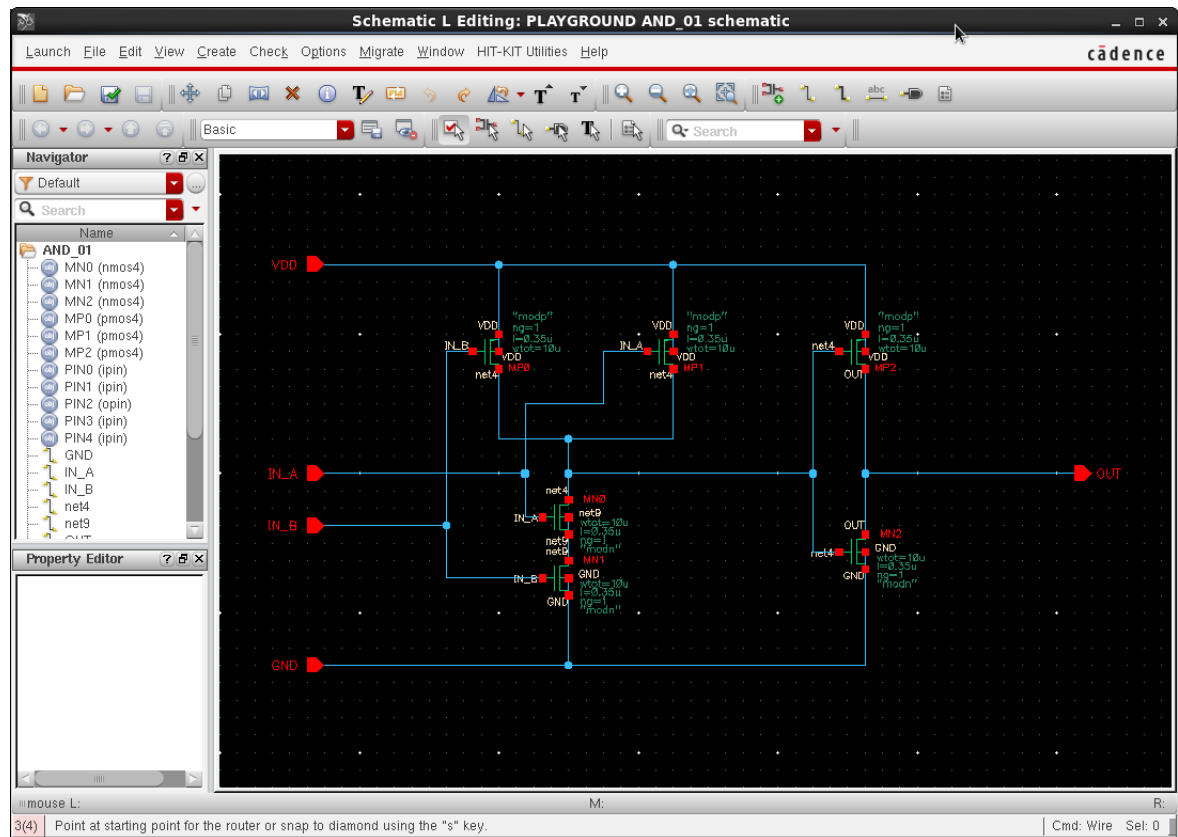
Assume we have the following design. It represents an AND gate.



We need to use this gate many times in our design, we need to create a symbol of it that can be used then several times. In schematic window, click on Create>Pin. The following dialog shall appear. Write a name for the pin and in the schematic place it.



Do the same for another input pin 'IN_B' and input pin 'VDD' and input pin 'GND' and finally output pin 'OUT'. Now your schematic should be like the following



Now in schematic window click on Create>Cellview>From Cellview

Cellview From Cellview

Library Name:

Cell Name:

From View Name:

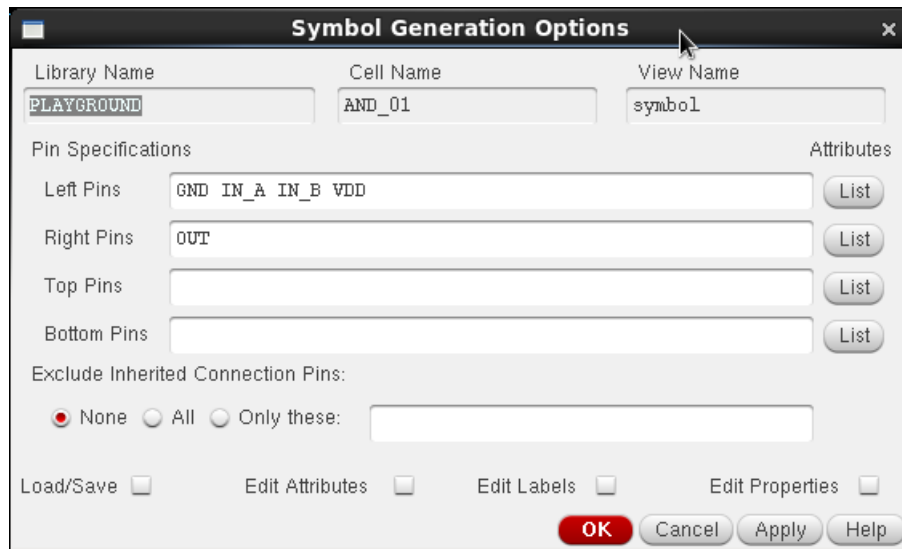
To View Name:

Tool / Data Type:

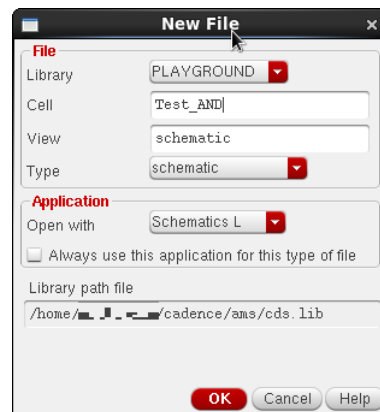
Display Cellview: ☒

Edit Options: ☒

Click OK



Click OK Now we can work with this Symbol. Create New cell view, lets call it Test_AND



In schematic Create>Instance then browse to library in which you saved your symbol. Then create your schematic to be like the following

