Simulation with Cadence Analog Design Environment

Analog Design Environment (ADE) is integrated on Cadence Custom IC Design software. You can simulate your design (schematic, extracted layout etc.) using the ADE.

This tutorial explains necessary steps required in preparing your design and using ADE to simulate the circuit. The tutorial assumes that you have the inverter cell with schematic and symbol views created as described in "Virtuoso Schematic Composer Tutorial" (available on class website).

Simulation Cell View Preparation

The first step is to prepare the simulation schematic view. In this tutorial, we will simulate the inverter using pulse voltage source connected at its input. First, create new schematic view, go to **File** -> **New** -> **Cell View**, we will call this cell **inv_sim** as shown in Figure 1.

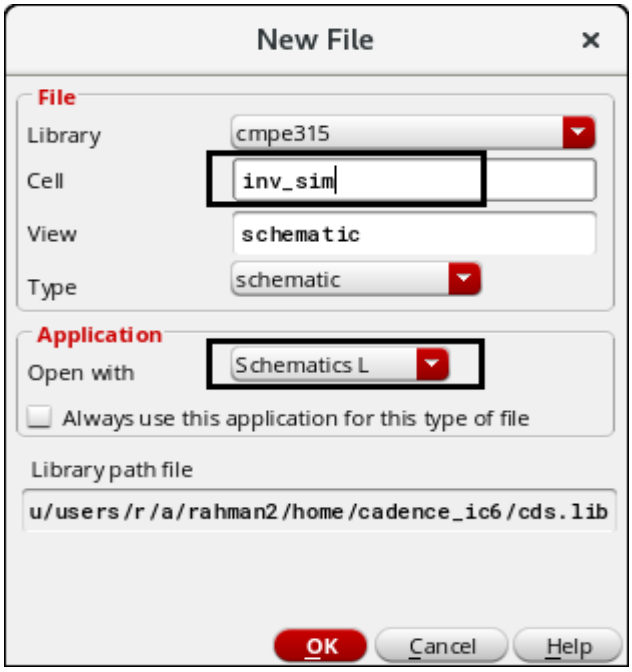


Figure 1: New simulation cell view

Next, we need to place the symbol of inverter on the schematic. Create new instance of the inverter by select "**inv**" from "**cmpe315**" library from the component browser as shown in Figure 2. At the Add Instance dialog, make sure you have **symbol** selected as shown in Figure 3, if not you can manually type in or click Browse.

Create instances of vdd, gnd, vdc and vpulse, create new OUTPUT pin (with "output" direction) and then make appropriate connections as shown in Figure 4. You can find vdc and vpulse from **Voltage_Sources -> vdc**, **Voltage_Sources -> vpulse** in **NCSU_Analog_Parts** library.

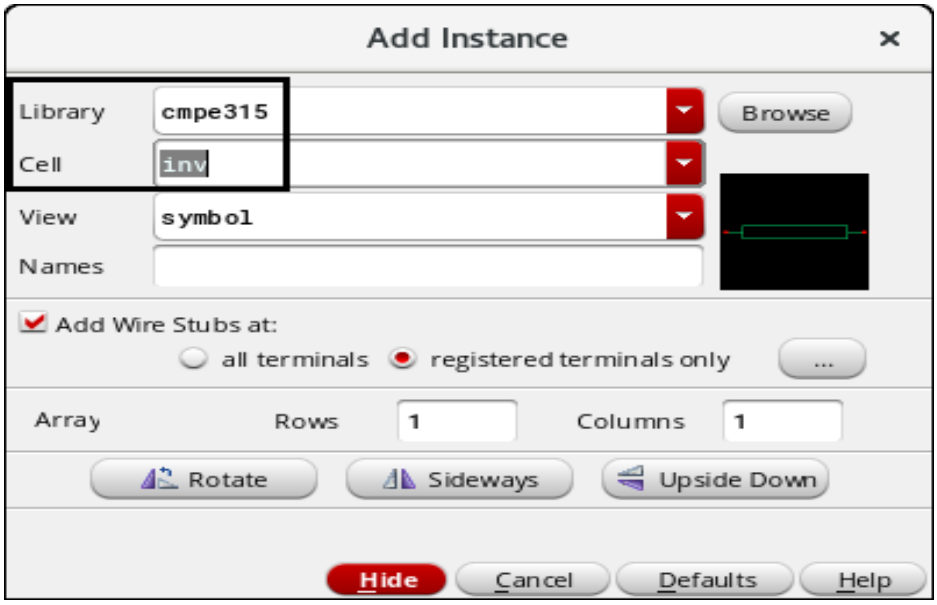


Figure 2: Select inv (inverter) cell from Component Browser

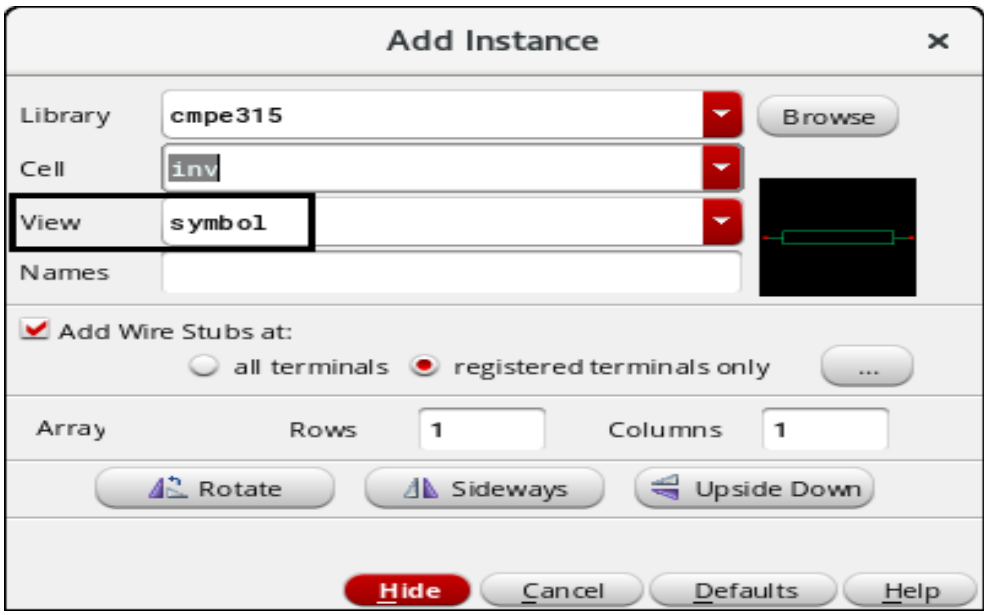


Figure 3: Select symbol view of inv cell

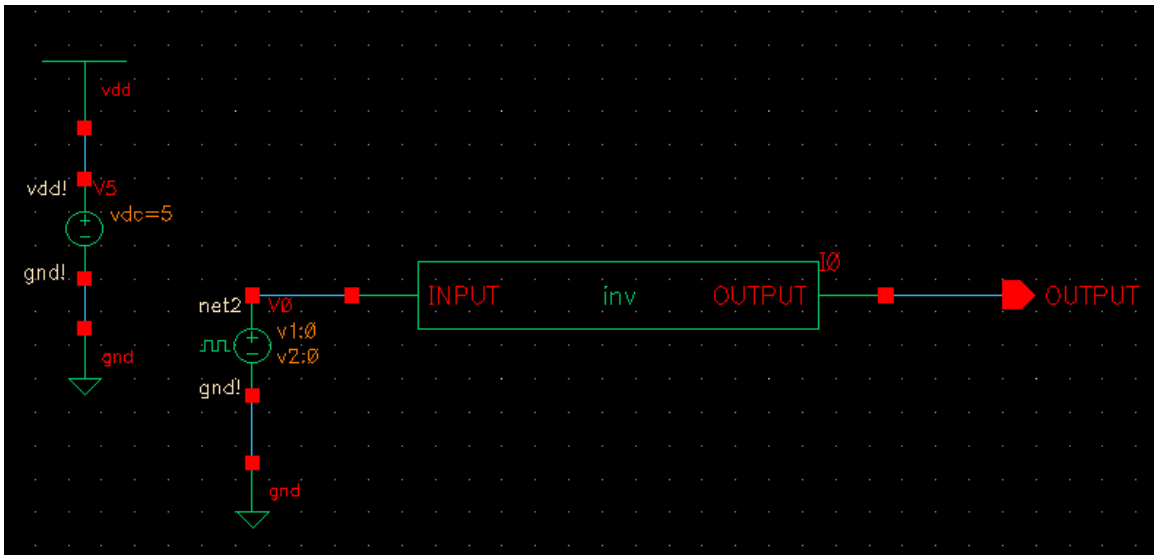


Figure 4: Schematic of inv_sim

Now, we need to change the properties of vdc and vpulse. To change the property, click on the component and go to **Edit -> Properties -> Objects**. For vdc, set DC voltage to 5V as shown in Figure 5. For vpulse, set Voltage1 to 0V, Voltage2 to 5V, Delay time to 1ns, rise and fall times to 500ps, pulse width to 5ns and period to 10.1ns as shown in Figure 6.

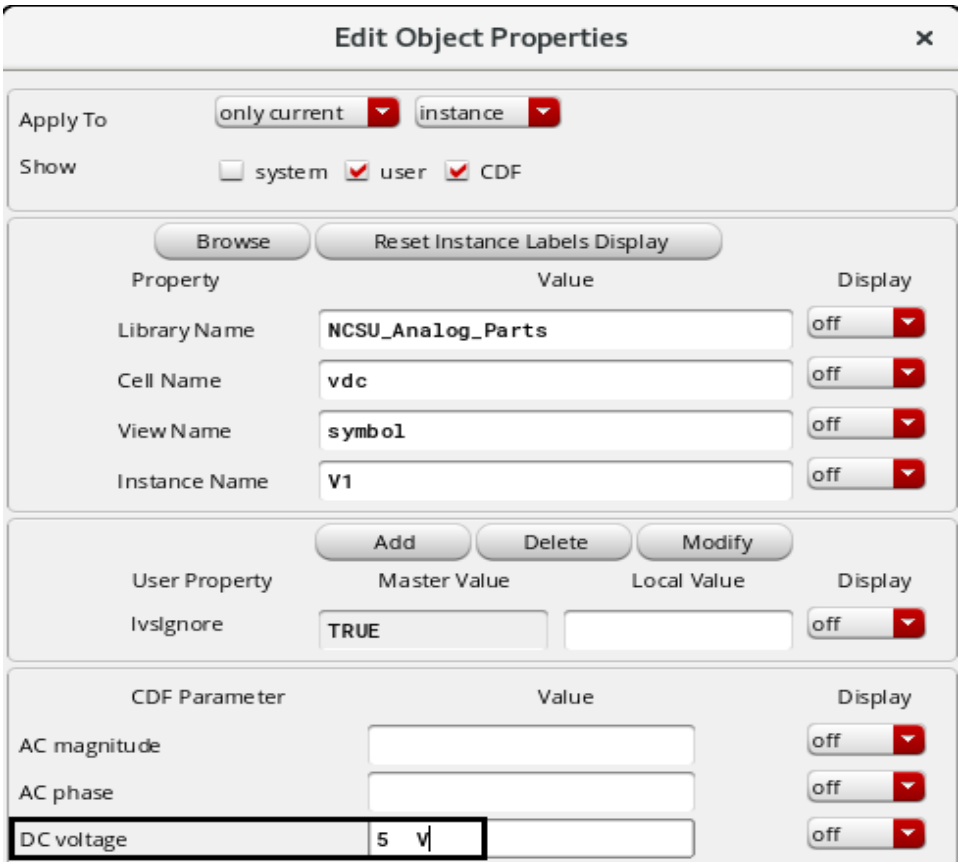


Figure 5: Property of vdc used for the simulation

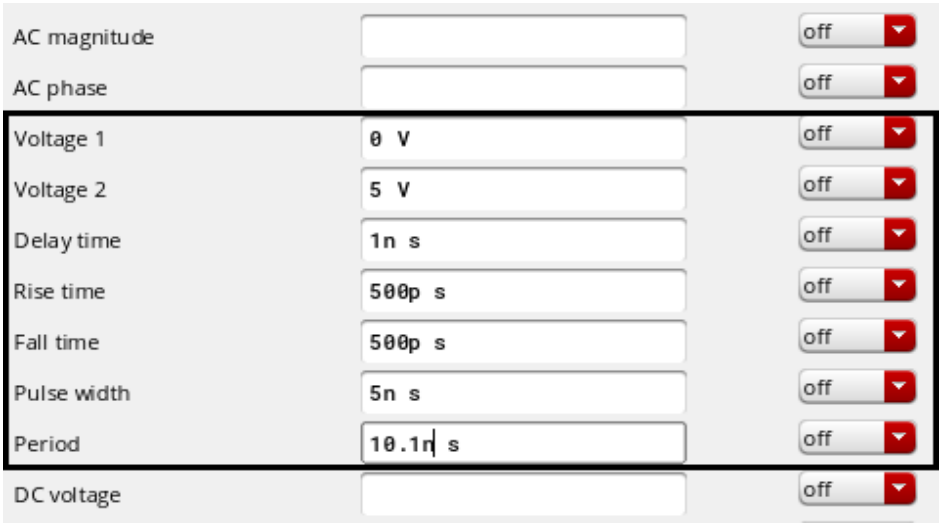


Figure 6: Property of vpulse used for the simulation

Hierarchy Editor

Hierarchy Editor lets you change views of the cell for simulation. For example, if you have made the schematic and layout for the inverter, you can create only one simulation view and then switch between schematic or extracted layout in the simulation using Hierarchy Editor. You can also mix multiple view types of cells in simulation using Hierarchy Editor (This mean you can simulate schematic of adder connects to layout of multiplier and/or vhdl of control logic).

To create new Hierarchy Editor view, highlight inv_sim cell in Library Manager, go to **File -> New -> Cell View** and select type as config as shown in Figure 7.

Figure 7: New Hierarchy Editor view

The New Configuration dialog will show up, click **view** and select schematic of inv_sim as the top cell as shown in Figure 8. Then click **Use Template**, another dialog will appear as shown in Figure 9, select **spectre** template and click OK. At View List, add "**extracted structural schematic**" separated by spaces exactly as shown in Figure 10. Click OK to close New Configuration dialog.

Now you will see Hierarchy Editor dialog as shown in Figure 11. From here, you can specify view you want to use in the simulation. For us, we will use schematic view of inverter in the simulation, so type in **schematic** in **View to Use** column of **inv** cell as shown in the figure. Click  icon to save the configuration and then close the Hierarchy Editor.

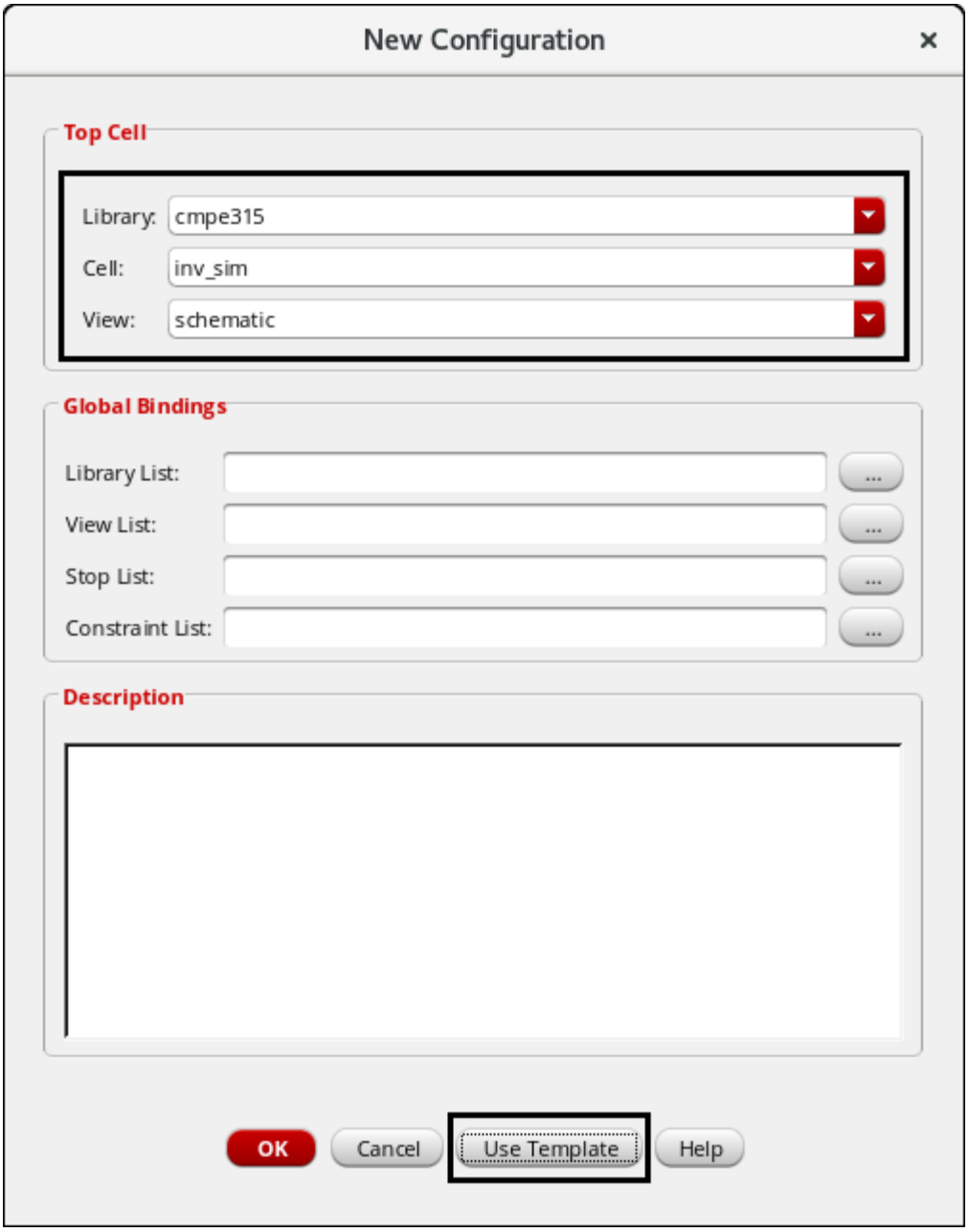


Figure 8: New Configuration

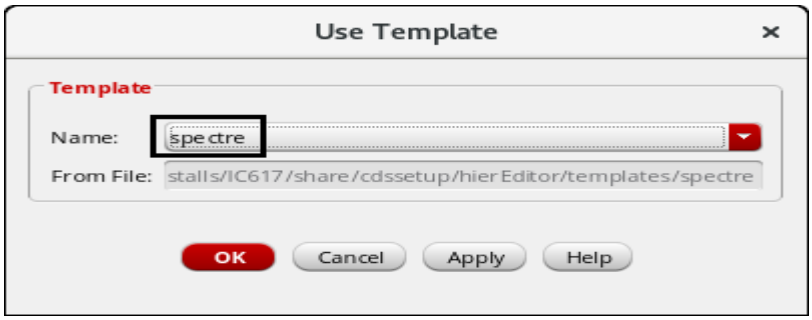


Figure 9: Use Template

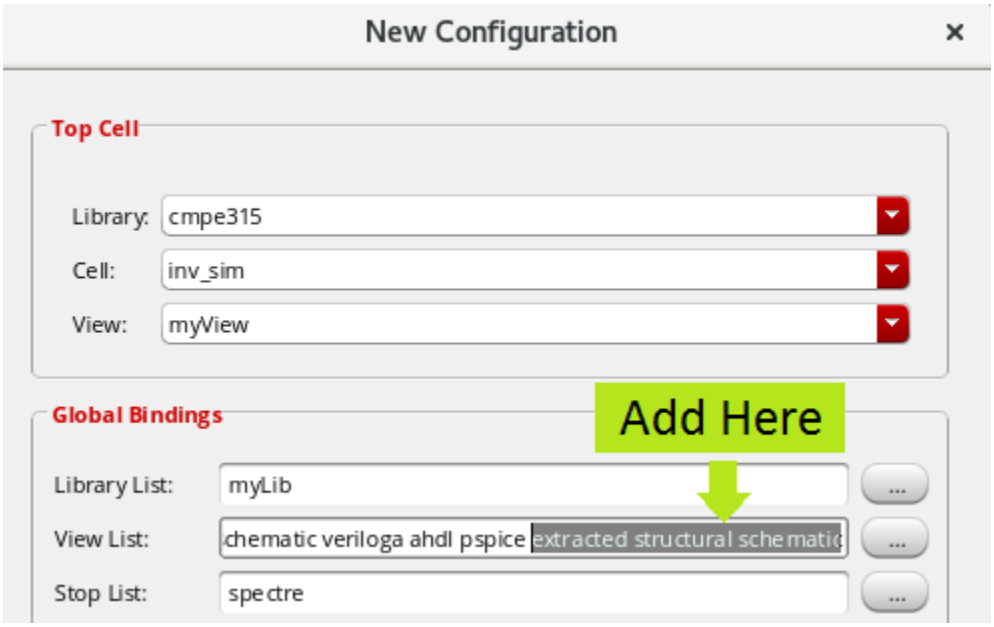


Figure 10: Add "extracted structural schematic" to View List

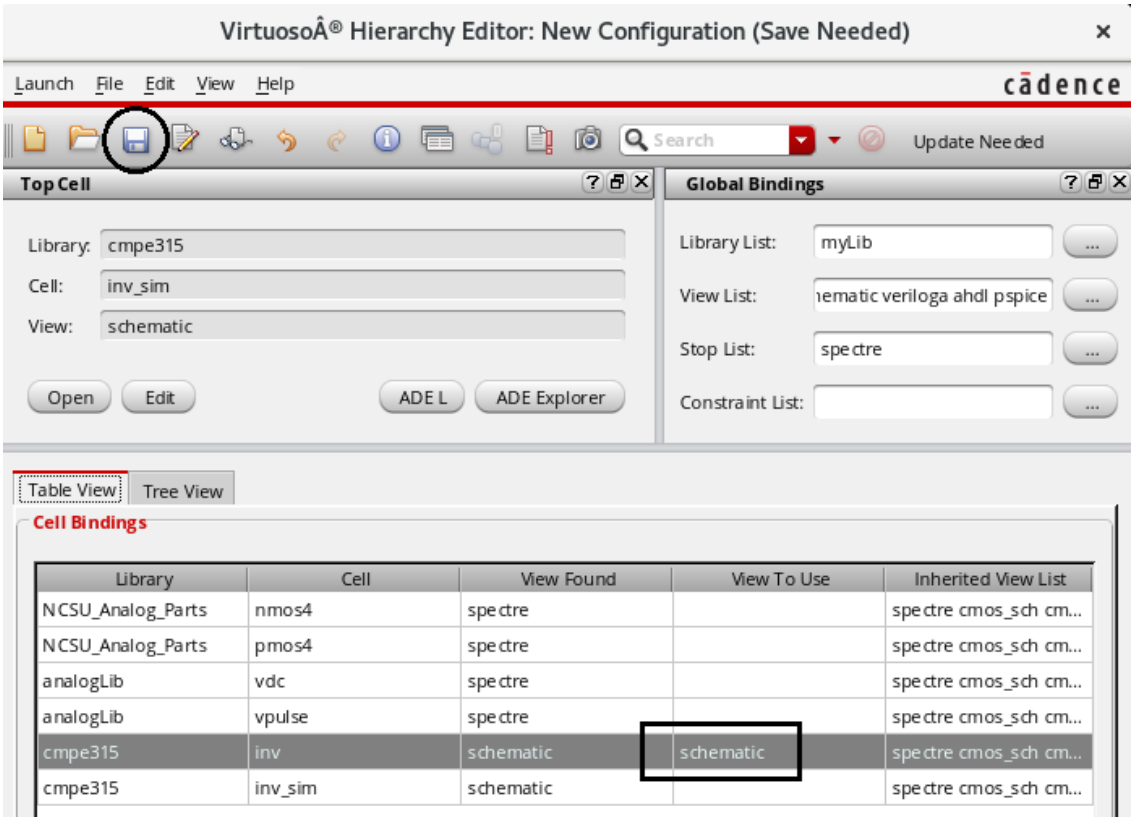


Figure 11: Hierarchy Editor

Open Analog Design Environment (ADE)

Open the config view of inv_sim cell (by double clicking) from Library manager, the dialog as shown in Figure 12 will show up, click OK. The schematic of inv_sim should be opened, go to **Launch -> ADE L** from the Hierarchy editor window. The ADE window should appear as shown in Figure 13.

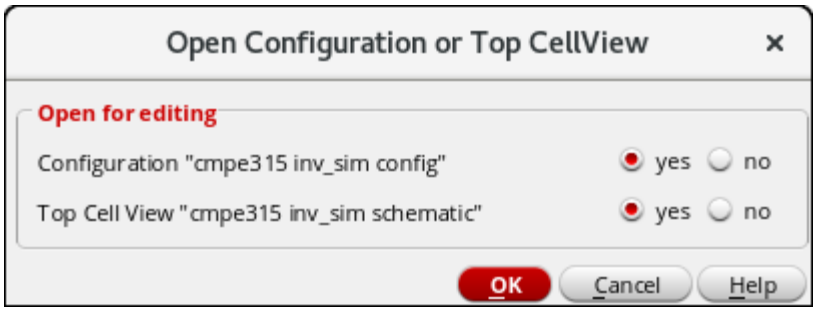


Figure 12: Open config view

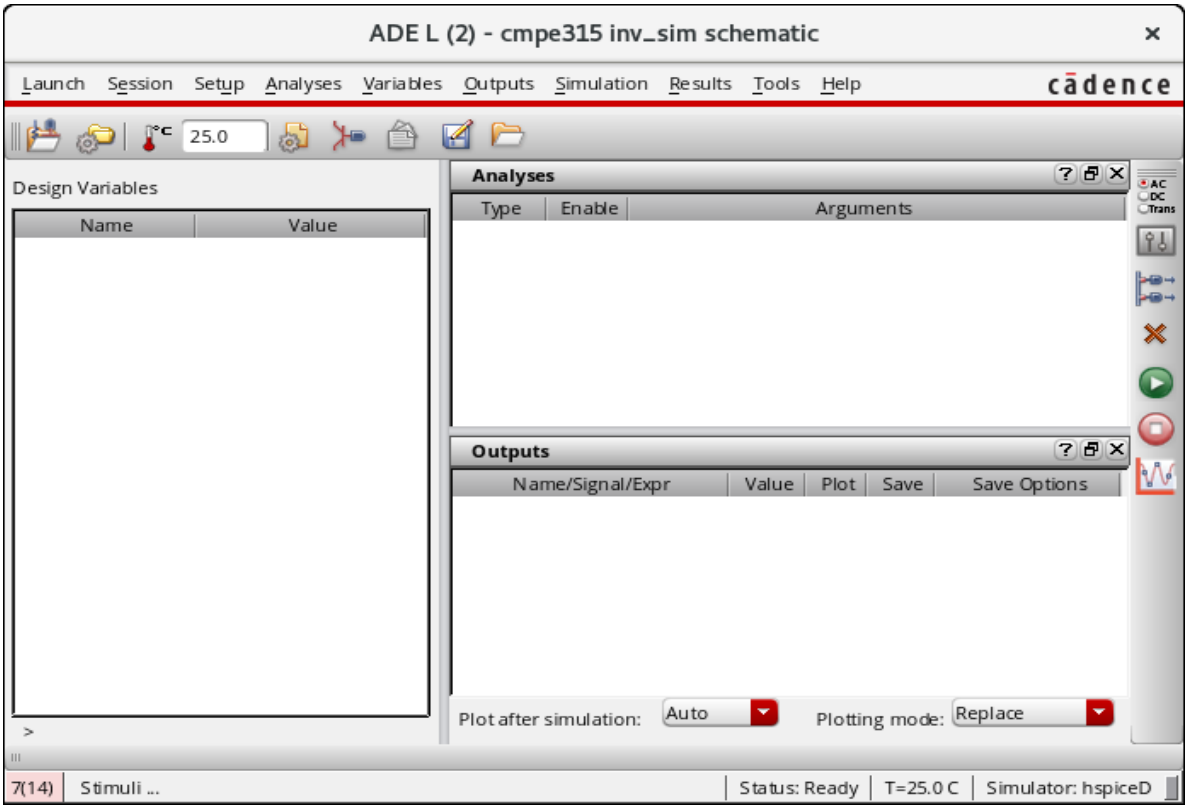


Figure13: Analog Design Environment

Simulation Setup

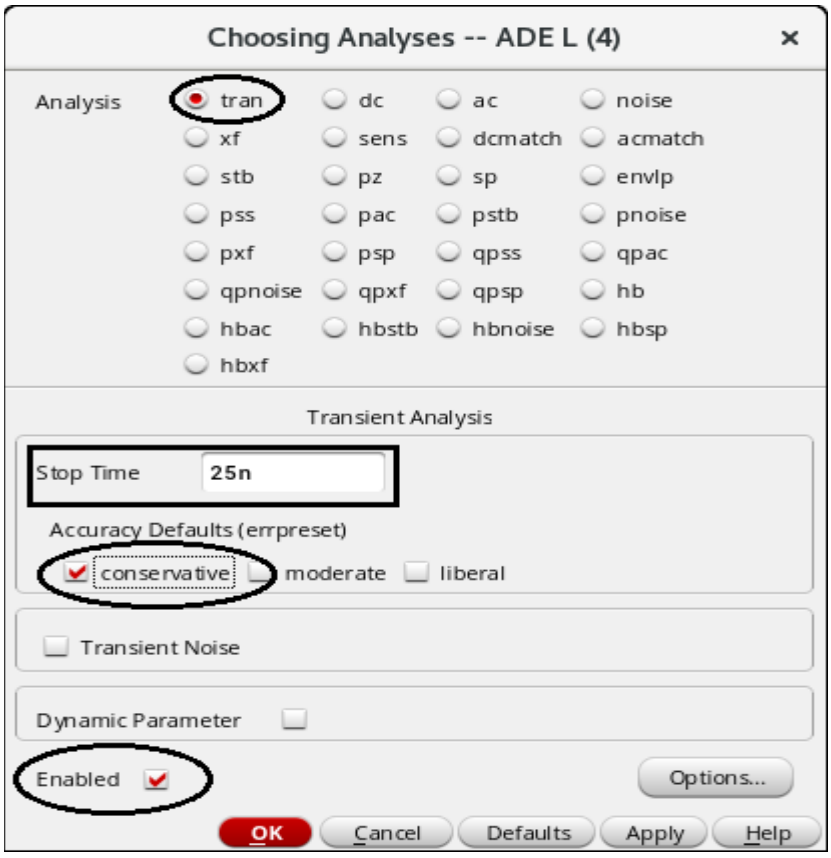
Go to **Simulator -> Directory -> Host**. Then select **Spectre** as a simulator as shown as Figure 14 (c).

Then, we have to set up the model libraries of the NMOS and PMOS . Go to **Setup -> Model library** then browse to these model files as shown in Figure 14 (b)

/afs/umbc.edu/software/cadence/design_kits/ncsu-cdk-1.6.0.beta/models/spectre/ /standalone/ami06N.m

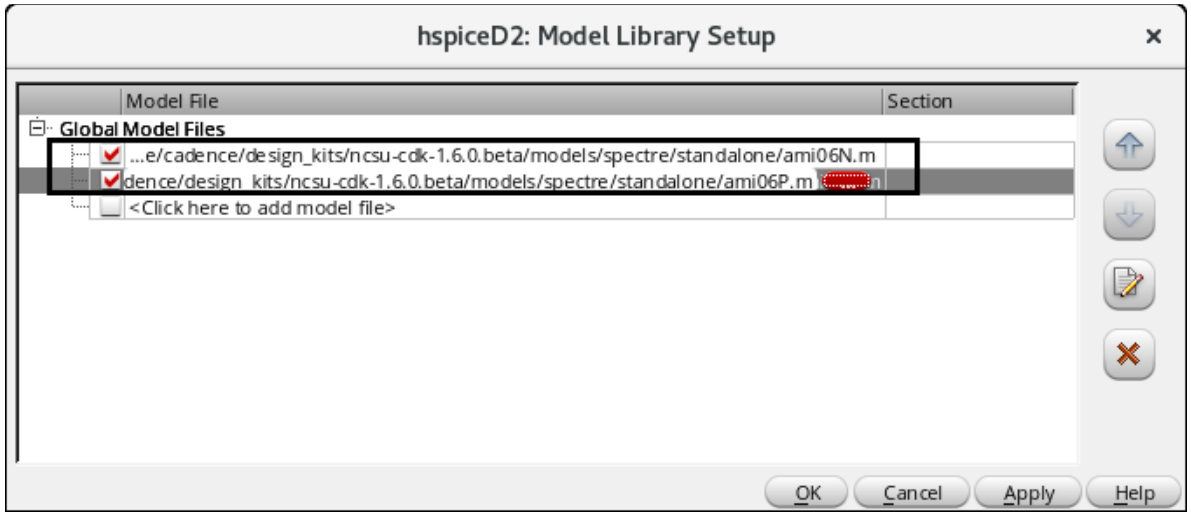
/afs/umbc.edu/software/cadence/design_kits/ncsu-cdk-1.6.0.beta/models/spectre/ standalone/ami06P.m

Next, we need to set the type of analysis, stop time and accuracy level for the simulation. Go to **Analyses -> Choose**, the Choosing Analyses dialog will appear. Select **tran** for analysis type, enter **25n** (run simulation for 25ns) for stop time, click on **conservative** for accuracy mode and **Enabled** to enable this simulation setup as shown in Figure 14 (a).



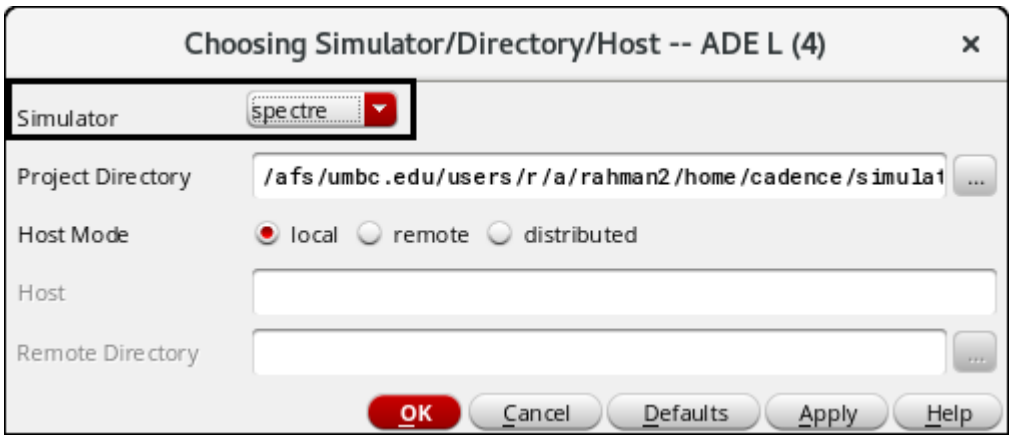
(a)

Figure 14 : Choosing Analyses



(b)

Figure 14 : Setting up the Model library.



(c)

Figure 14 : Selecting Spectre as the simulator.

Next, we will select signals we want to observe after run simulation. Go to **Outputs -> To Be Plotted -> Select On Design**. Click on wires/nets on schematic that you want to observe. In our case, click wire connected to INPUT and OUTPUT of the inverter. The ADE window after having analyses and outputs setup is shown in Figure 15.

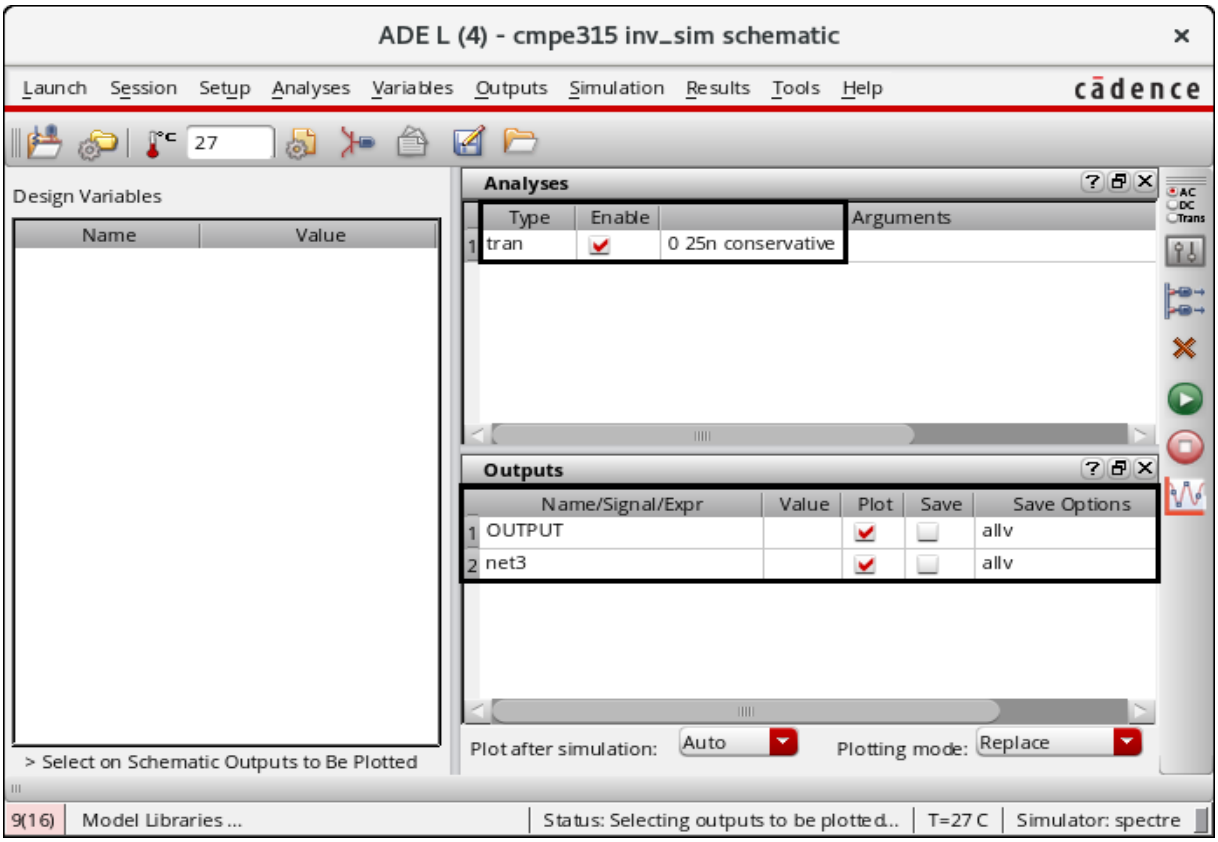


Figure 15: ADE after analyses and outputs setup

Go to **Simulation -> Netlist -> Create**, the final netlist will appear as shown in Figure 16. If the netlist does not show up, see if there are any errors on CIW window or go to **Simulation -> Output Log**. (Note that this step may be skipped but we recommend you to run if you are setting the new simulation for the new cell.)

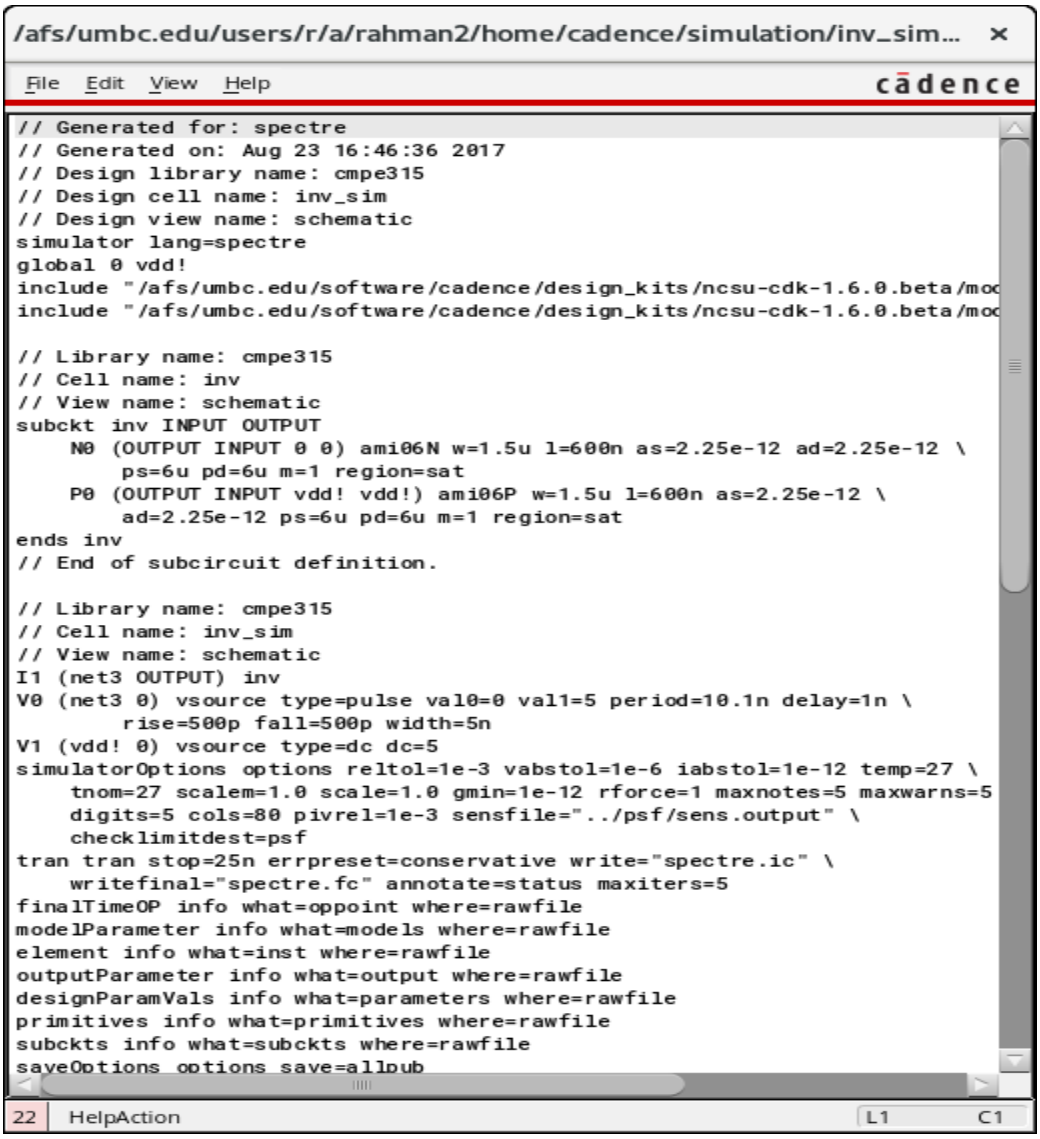


Figure 16: Created Final Netlist

Run Simulation

Now run the simulation, go to **Simulation** -> **Run**. If the circuit is large, it might take times for the simulation to finish (depends on the size of the circuit, might take from few minutes to many hours). The information on running simulation is displayed on CIW window.

Simulation Results on Waveform Window

After the simulation is finished, the waveform window will show up as shown in Figure 17. All signals selected will plotted on the same axes and difficult to look at some time. Go to **Graph -> Split current Strip** to have waveforms plotted on separate axes as shown in Figure 18.

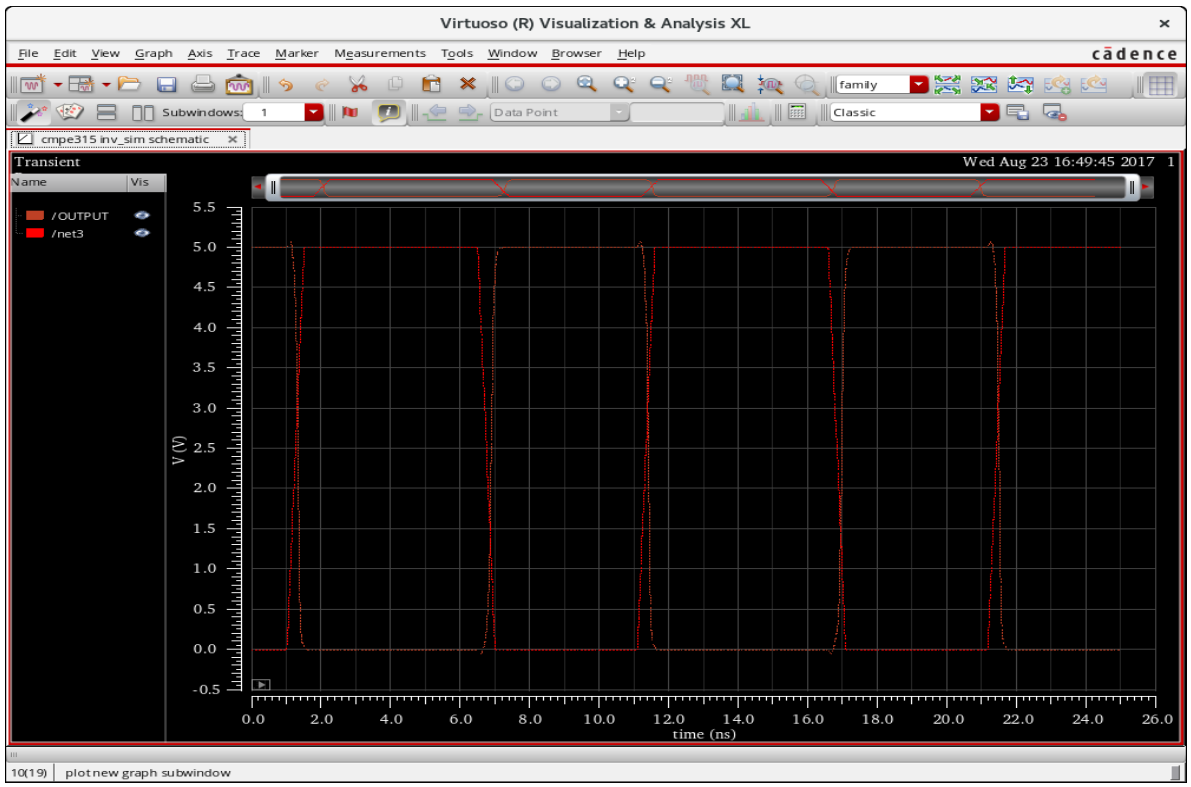


Figure 17: Waveform Window

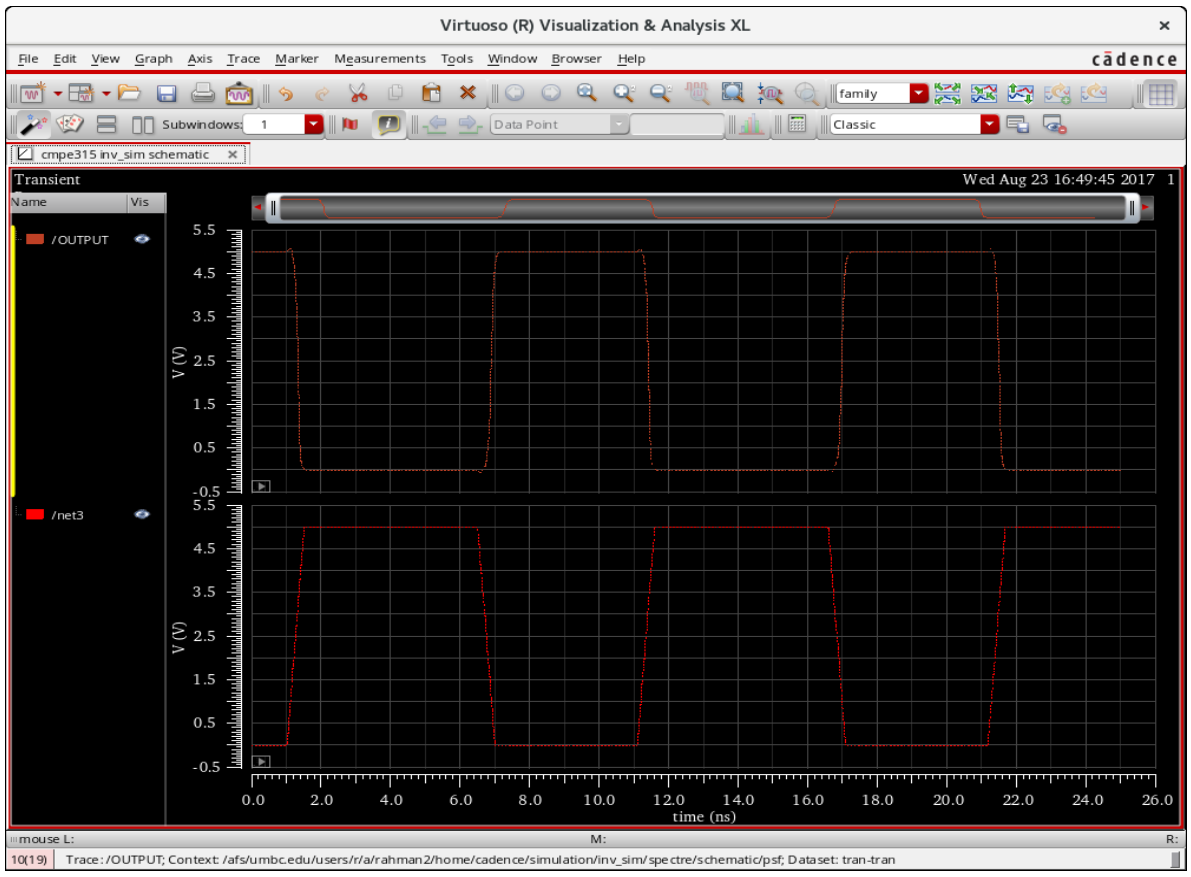


Figure 18: Waveform Window after Strip

Save State

You can save simulation setting so that next time you open ADE you do not need to setup Analyses and Output to be plotted (and any other settings) again. Go to **Session -> Save State**, enter state name as shown in Figure 19 and then click OK.

Figure 19: Save State

Load State

To load saved state, go to **Session -> Load State**, select the state you want to load from the list as shown in Figure 20.

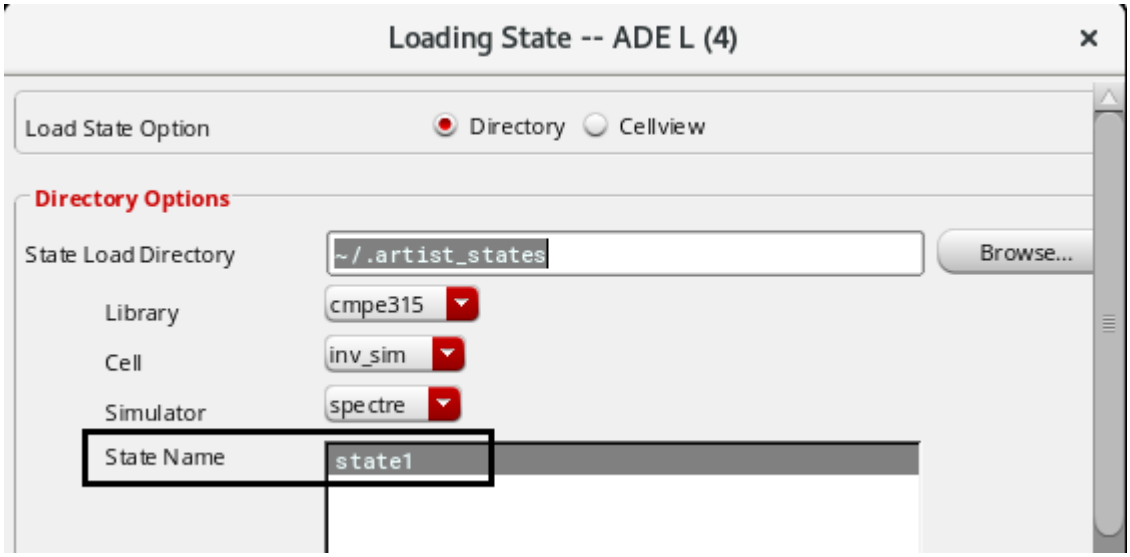


Figure 20: Load State

Save image from Waveform Window

You can save image of the waveform window by going to **File -> Save image**, the Save Image dialog as shown in Figure 21 will appear.Click on the **Directory bar** and set a directory to save the image file (You can set your save directory as per your liking.) Then put your desried directory in the Address bar, Type **plot** (You can give your desired name for the image file) in **File name**, select type as **PNG**. Finally click on **Save** in the window to save the image in your desired directory.

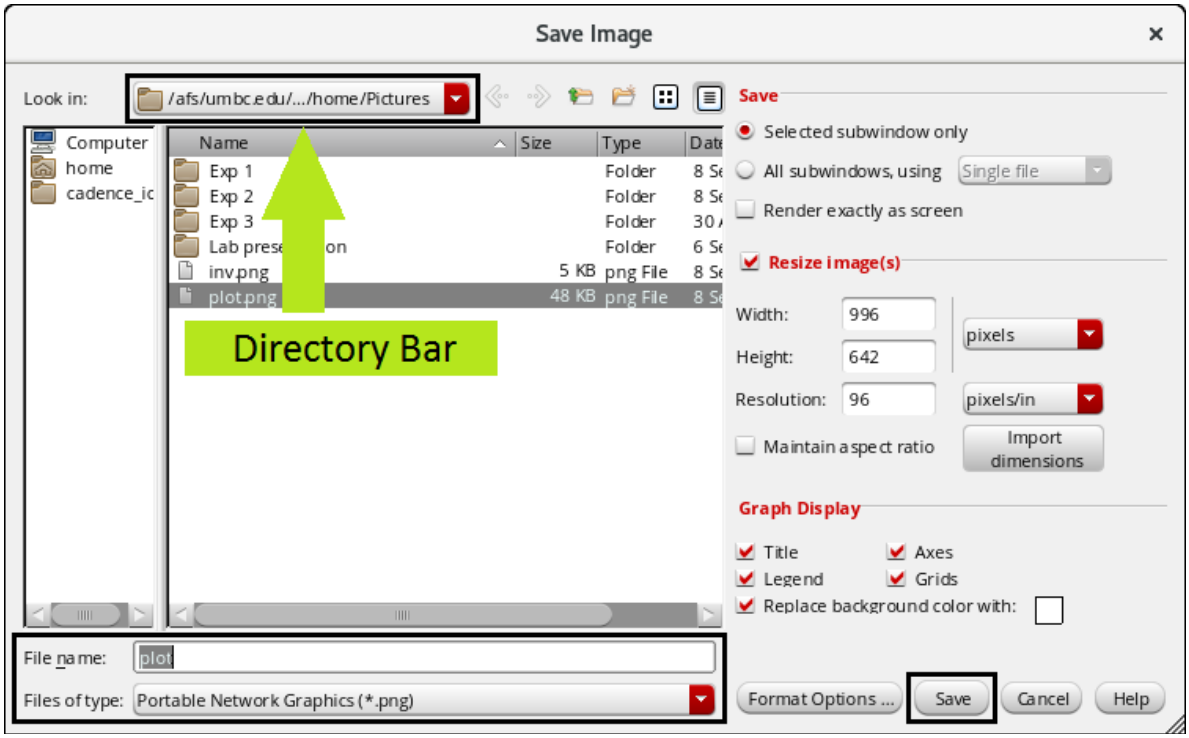


Figure 21: Save Image