

Symbol Editing and Transient Simulations

This lab will deal with Virtuoso Schematic Editor and Virtuoso Symbol Editor, to design a CMOS ring oscillator which is composed of multiple inverter cells. Through transient simulations with the ring oscillator, basic transient characteristics of an inverter will be discussed.

Objectives of this lab are to:

- Create/edit symbols and hierarchical design
- Set and run transient simulations
- Understand delay of an inverter

1) Launch Cadence:

- a. Connect to the cadence server (cadence1.rowan.edu), and source cds.setup from your home directory.

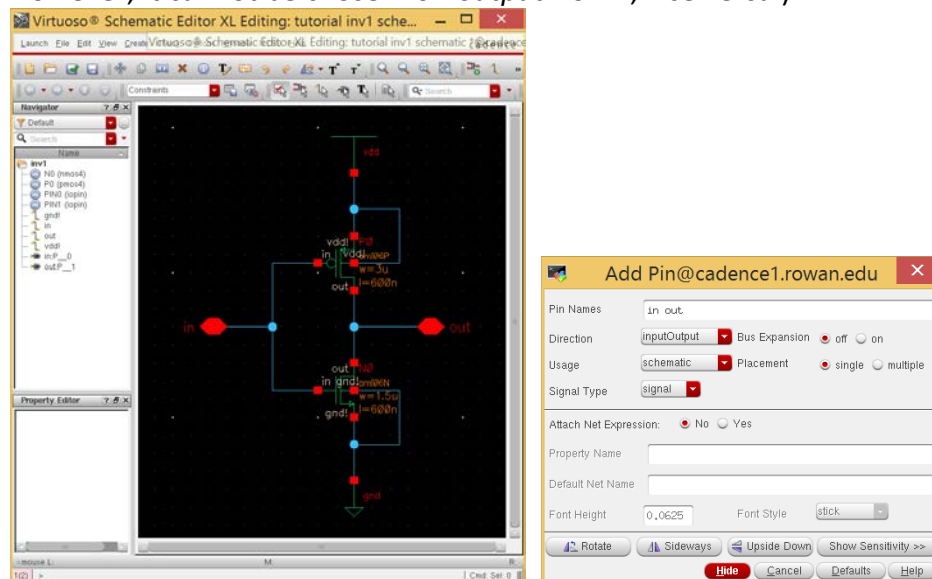
>> source cds.setup

- b. Then move to your Cadence project directory, and run the Cadence:

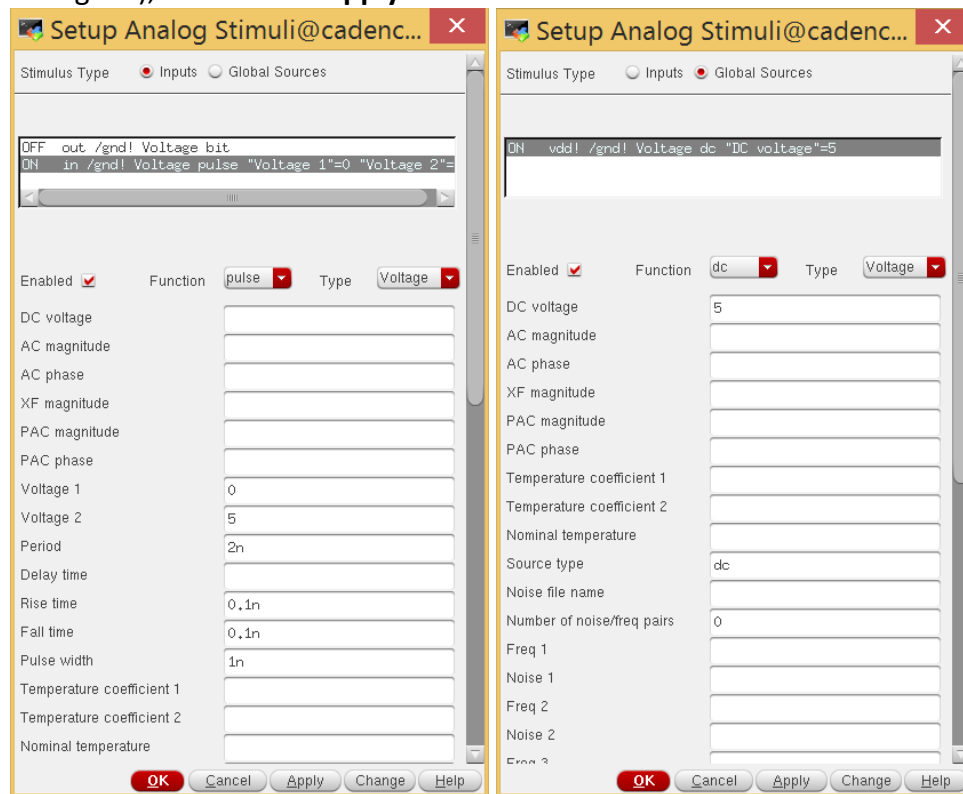
>> virtuoso &

2) Create a schematic for a CMOS inverter:

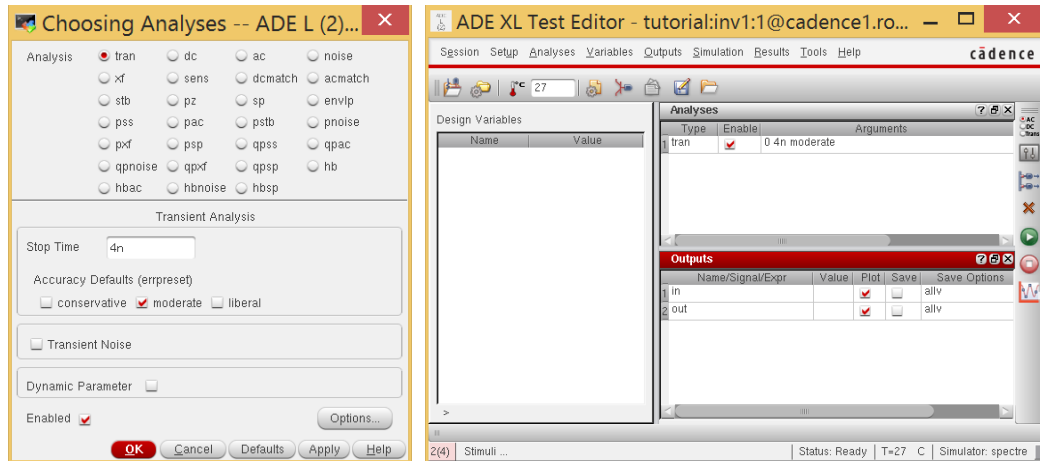
- a. From the **Library Manager**, create a new Schematic view, with a name **inv1**, under your work library.
- b. Draw an inverter with transistors **nmos4** and **pmos4** from the library **NCSU_Analog_parts**, and add **PINs** for **in** and **out**. (For the type of pins, it can be chosen for 'input' for **in** and 'output' for **out**, or for 'inputoutput' for **both**. However, it cannot be chosen for 'output' for **in**, vice versa.)



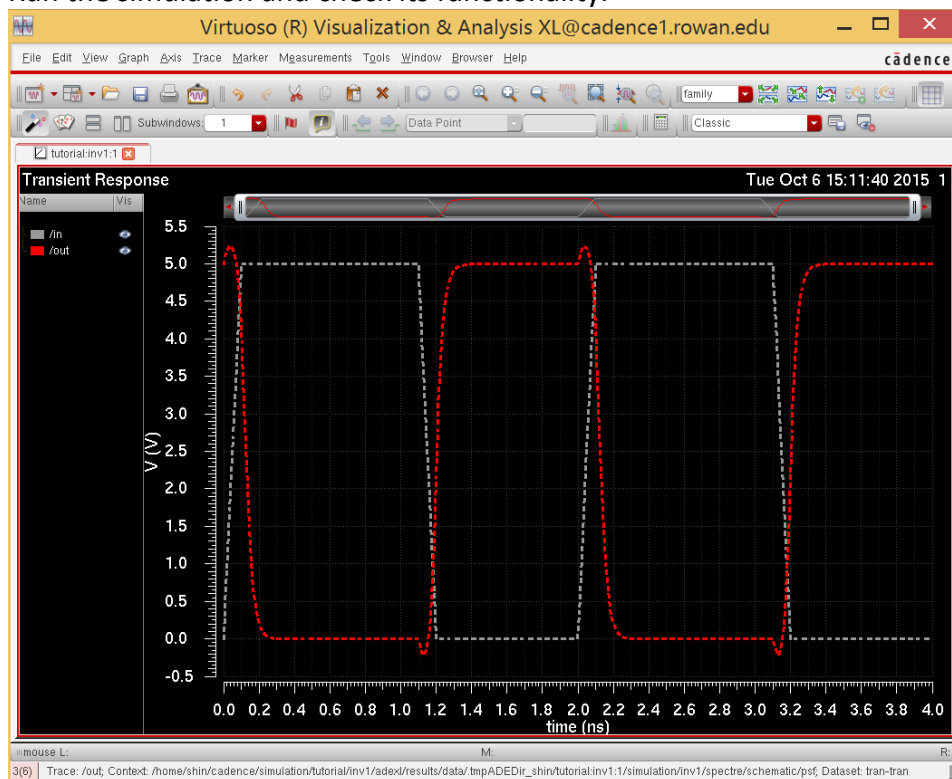
- c. **Check and Save** the schematic view, and make sure there is no error.
- 3) Run **Transient Simulation** and check its operation:
- Launch **Spectre** to set it up (Launch **ADE GXL**, then open **ADE XL Test Editor**.)
 - Set the simulator as **Spectre** and add the **NCSU models** to the model libraries.
(You may load a setting previously saved for other design, and modify details.)
 - From the ADE Editor, choose Setup -> Stimuli. Stimuli for inputs and global source (vdd!) can be set in the pop up **Stimuli** window.
 - With the chosen **Inputs** for *Stimulus Type*, click on the input pin (e.g., **in /gnd!**), then check for **Enabled**, choose *Function* as **pulse**, *Type* as **voltage**. Now specify the detailed pulse pattern (e.g., Voltage1=0, Voltage2=5, Period=2n, Rise time=0.1n, Fall time=0.1n, Pulse width=1n), and click on **Apply**.
 - With the chosen **Global Sources** for *Stimulus Type*, check for **Enabled**, choose *Function* as **dc**, *Type* as **voltage**. Then specify the supply voltage (e.g., DC voltage=5), and click on **Apply**.



- f. Set the transient simulation and outputs to display. Setting the transient simulation with Stop Time of 4ns will run the simulation for two input periods.



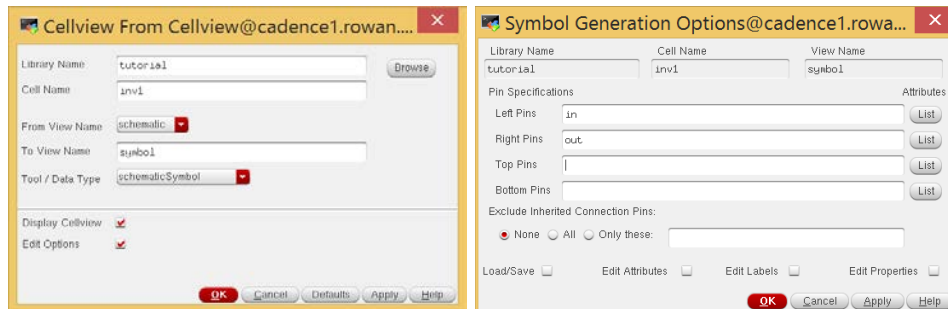
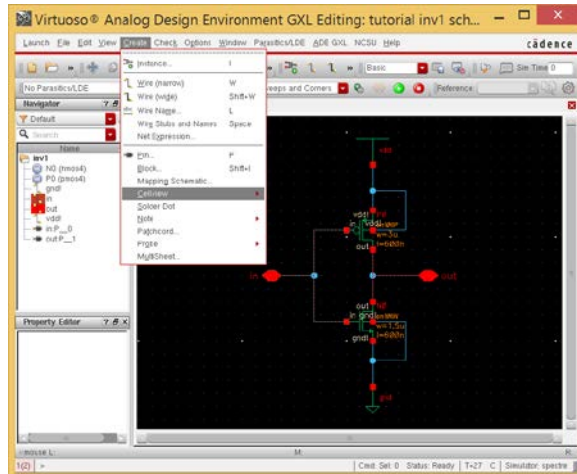
g. Run the simulation and check its functionality.



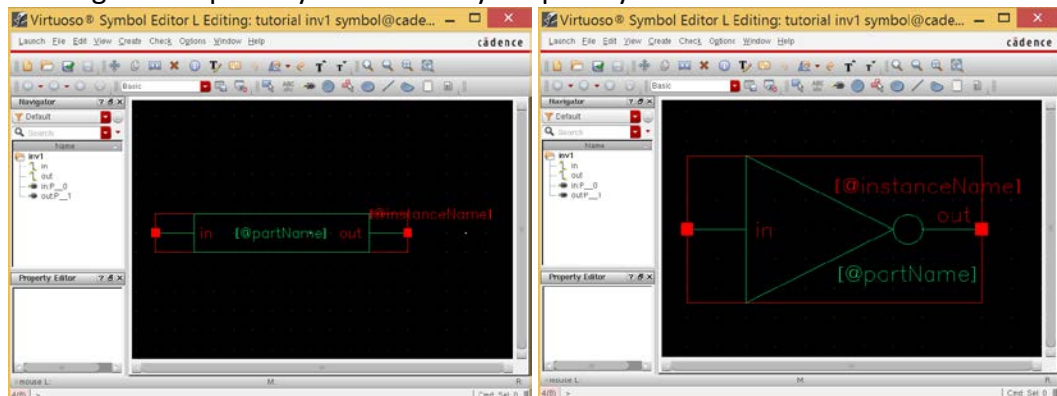
h. Find the rise time (t_r), fall time (t_f), propagation delay for falling (t_{pdf}), and propagation delay for rising (t_{pdr}). What's the average propagation delay (t_{pd})?

4) **Create a Symbol** for the inverter:

- a. From the **Schematic Editor**, choose *Create->Cellview->From Cellview*. Then choose **schematicSymbol** for the *Tool/Data Type*, then click on **OK**. It will open an options window that allows you to specify pin locations. (By default, 'input' pins are to be in left side, 'output' pins are to be in right, 'inputoutput' pins are to be on top of the symbol.)



- b. You should get a pop up Symbol editing window where two pins (*in* and *out*) are placed in the location specified in above step. Now you can modify the default rectangular shape of symbol into any shape as you wish.

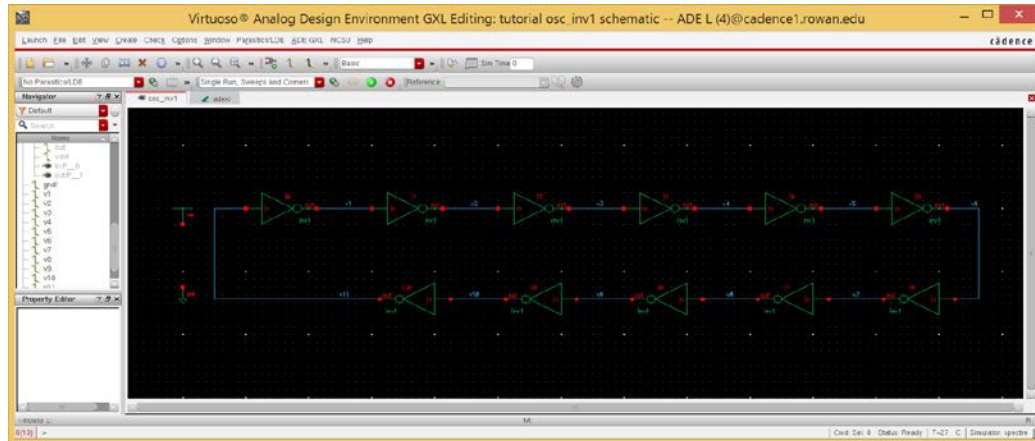


- c. Save your symbol and close the editor. Then go to the Schematic window and try Check and Save again. That is to double check if the pins specified in the symbol you created exactly match to those defined in the schematic. You should get no errors or warnings.

5) Design a ring oscillator:

- a. Create a new schematic, named **osc_inv1**, under your work library.

- b. Place 11 copies of your inverter symbol (**inv1**), and connect them in a series loop. (A loop with an odd number of inverters forms an oscillator.) Give nodes net names to the nodes, e.g., v1 ~ v11. Place symbols of unconnected **vdd** and **gnd**. (This does nothing on the simulation, but is just to let simulator know of the global nets, *vdd!* and *gnd!*, used in the inverter cells.)



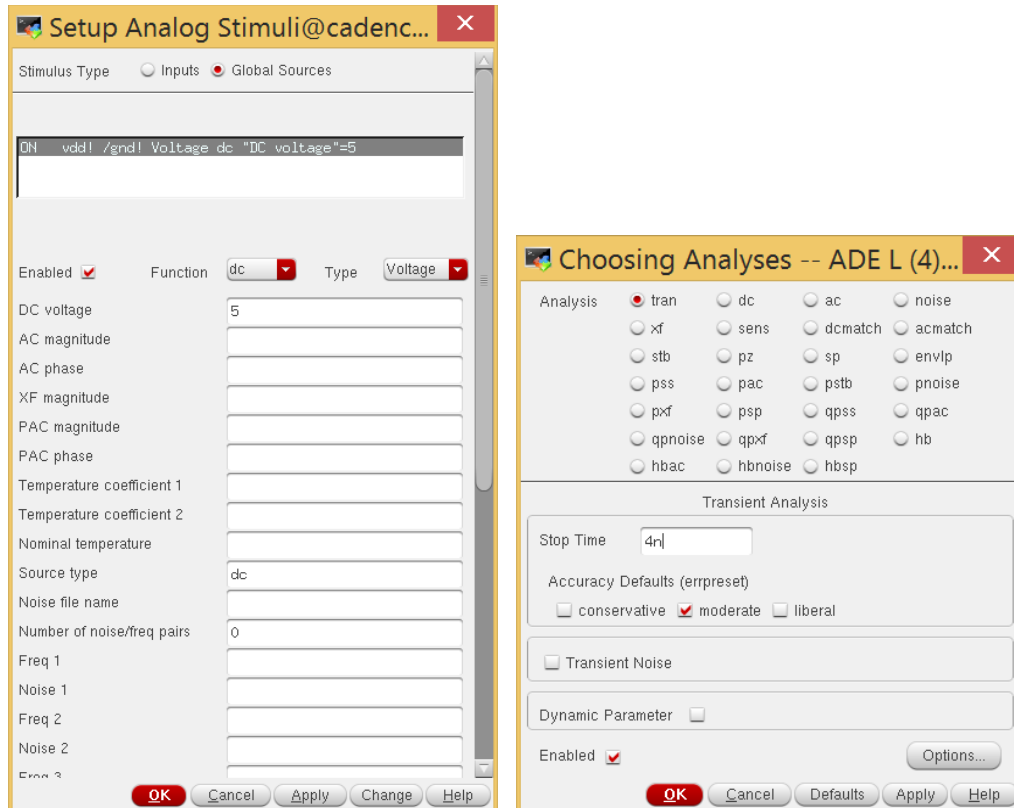
Many times when you have a complex hierarchical schematic you may want to edit cells without having to close and open different windows. You can do that by traversing the hierarchy. For example, you can go down from **osc_inv1** to **inv1** and modify the inverter cell, then return back to **osc_inv1**. In order to do that, from the **osc_inv1** schematic, you can click on a **inv1** symbol that you want to descend to (or edit in place), and choose *Design -> Hierarchy -> Descend Edit (or Edit in Place)*, then choose the view (schematic or symbol) you want to edit and click OK.

Tips: Hotkeys for traversing across hierarchy:

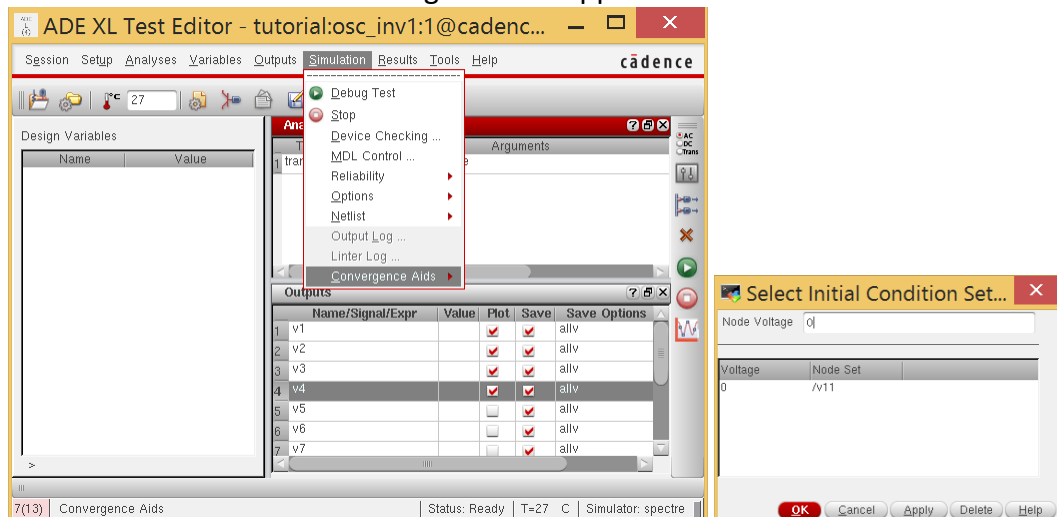
- '**Shift+X**': descend to the selected cell (then you can choose a view)
- '**Ctrl+x**': get into the selected cell in current place
- '**Shift+B**': go up to the parent cell

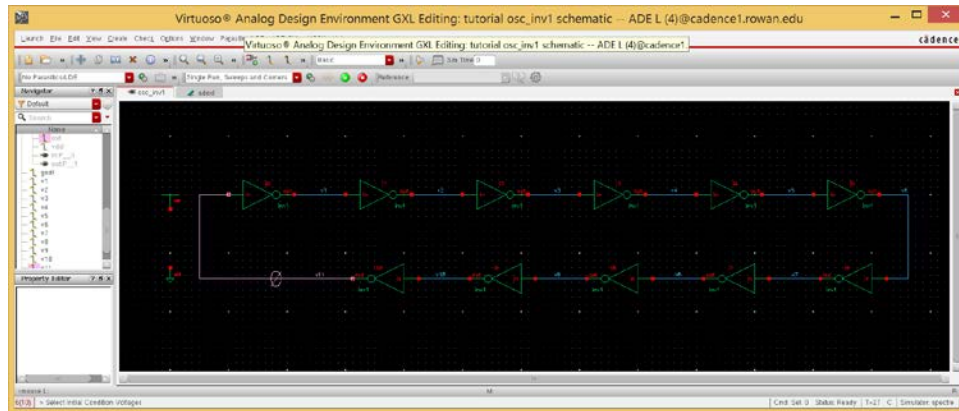
6) Run **Transient Simulation**:

- a. Launch **ADE Test Editor** and set it up (Refer the section (3) above.)
Check if the simulator is set for **Spectre** and **NCSU model** libraries are loaded, and set the **stimulus** for **vdd!** (5V DC). Select outputs you want to save and/or to display, and choose **tran** for the transient analysis.

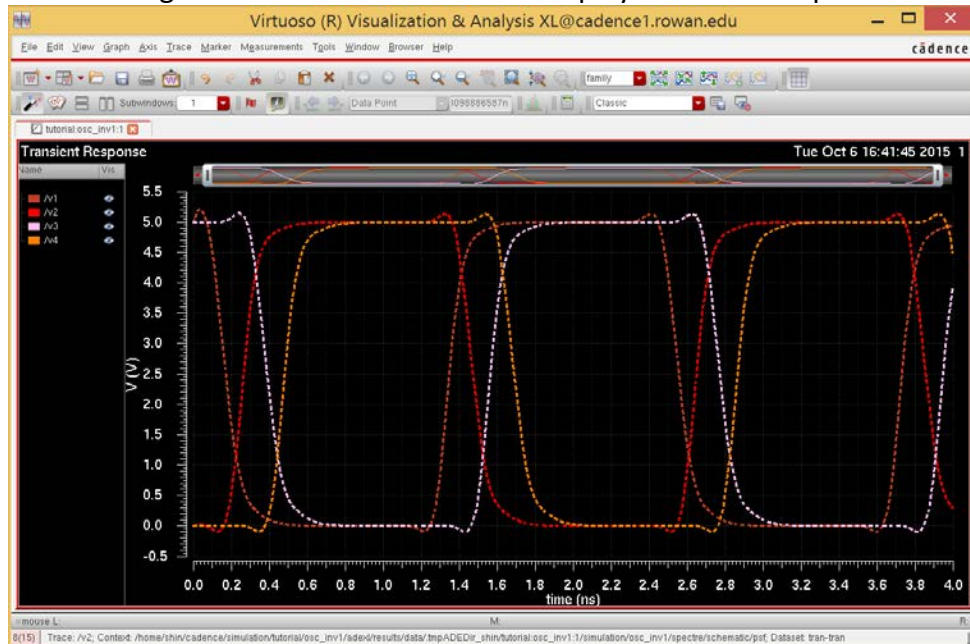


- b. Set for **initial condition**. In many cases, defining clear initial conditions (either in voltage or current) helps simulator to converge its computation. With the ring oscillator, we may define an initial voltage for any node. To do that, from the **ADE Editor**, choose *Simulation -> Convergence Aids -> Initial Condition*, then click on a node in the schematic. A big 0 should appear on the net. **ESC** and click **OK**.





- c. **Run the simulation.** (Make sure to Check and Save everytime you edit cells.)
 You should get a waveform window that displays selected output waveforms.



7) Output Analysis:

- Find the rise time (t_r), fall time (t_f), propagation delay for falling (t_{pdf}), and propagation delay for rising (t_{pdr}). What's the average propagation delay (t_{pd})? (You may compute using Calculator, or extract those from the waveforms.)
- Compare** the results with those you got in the **section 3.h**, and **discuss** if the two results meet '*qualitatively*' with theoretical expectations.