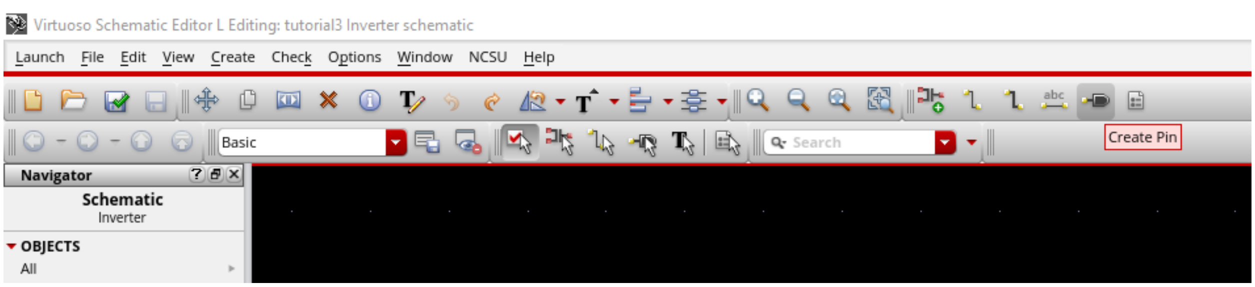
Lab 3: Symbol Editing and Transient Simulations

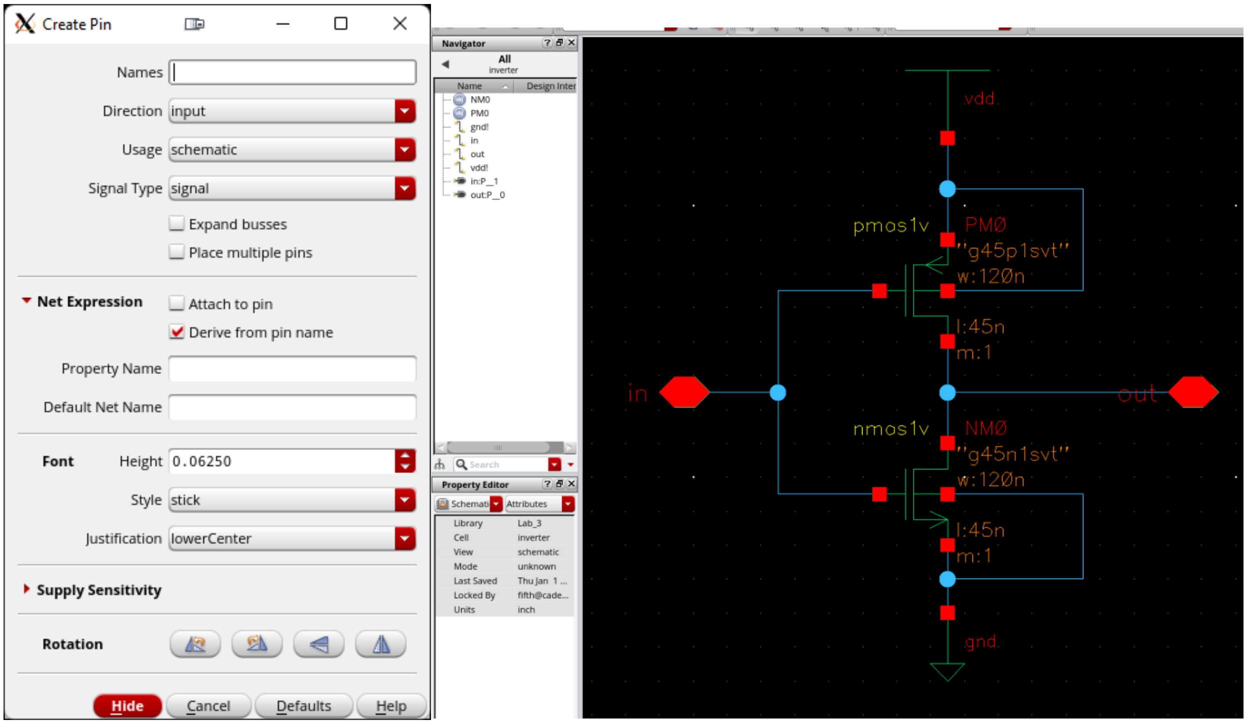
This lab will leverage Virtuoso Schematic Editor and Virtuoso Symbol Editor to have students design a CMOS ring oscillator composed of multiple inverter cells. Students will be introduced to hierarchical design by utilizing an abstracted inverter object multiple times in the ring oscillator design. Students will analyze delay properties of a CMOS inverter and a ring oscillator using the calculator to measure circuit delay via transient simulation of each circuit.

1. **Launch Cadence**
   - **a.** Connect to Cadence03
   - **b.** source cds.setup
   - **c.** Move to your cadence directory
   - **d.** run Cadence (Virtuoso &)

2. **Create a new library and copy your CMOS Inverter design from Lab 2**
   - **a.** File -> New -> Library
     - i. Attach to an Existing Technology Library – *gpdk045*
     - ii. *Create a new schematic cell (same as before). Verify you are using Schematic XL before clicking OK*.
   - **b.** Change your Lab2 Inverter by adding input and output pins to the design.
     - i. Select the pin tool on the tool bar as shown below. **(WAIT FOR THE INSTRUCTOR TO EXPLAIN THE PIN OPTIONS BEFORE PLACING THEM IN YOUR DESIGN)**
     - ii. Sometimes this tooltip is hidden, if you expand the toolbar that contains the “AddInstance” tooltip you will find the “CreatePin” tooltip



     - iii. Then enable “Derive from pin name” in this new window (under “Net Expression”)
     - iv. Add input and output pins to the gate input and drain output, respectively
     - v. Check and save!



3. Run Transient Simulation

a. Open the schematic view and launch *ADE Explorer* or launch from the *Library Manager* by double clicking the *maestro* view.

b. Design a test with the following parameters:

i. Conduct a Transient Analysis with a stop time of **4ns** on the inverter with the input signal set to the following parameters. (Part c has the instructions for entering this information.)

Input Pin			
Function	Pulse	**Type**	Voltage
Voltage 1	0 V	**Rise Time**	0.1ns
Voltage 1	5 V	**Fall Time**	0.1ns
		Pulse Width	1ns
		Period	2ns
Global Source Vdd			
Function	dc	**DC Voltage**	5 V

c. To set the input signal you must setup the Analog Stimuli for Vdd and your input pin. The Stimuli can be accessed via the Test Editor under the drop menu Setup. (This setup is a little tricky, please wait for the instructor to explain before proceeding.)

i. Setup -> Stimuli

ii. Authoring “On”

iii. Name the test setup in the right panel “Test_Pulse_1” or something helpful.

iv. Select the type of stimuli -> "Pulse" in this case but there are others that can be useful.
v. Select "V" for voltage (I is right next to it)
vi. Under are the parameters you need to fill out according to the table above.
vii. Select apply and you will see your function under the functions list. Create both "test_pulse" and "Vdd" before moving forward.
viii. Once complete, turn Authoring Off
ix. Under pin assignments click on pins. You should see both in and out. Enable in only
x. Select the function you want to assign and then right click on the pin you want to assign it to and select "assign stimuli to selected pins
xi. Now highlight the Vdd function, Select global, Right click on vdd! -> Assign
xii. Apply -> ok

d. Netlist -> Run the simulation
i. You can see both in and out on the same graph if you plot them both.
ii. Don't forget to select both to save for later calculator use.

4. Calculate the Inverter's Delay in Signal Propagation from Input to Output (Use the cadence built in calculator)

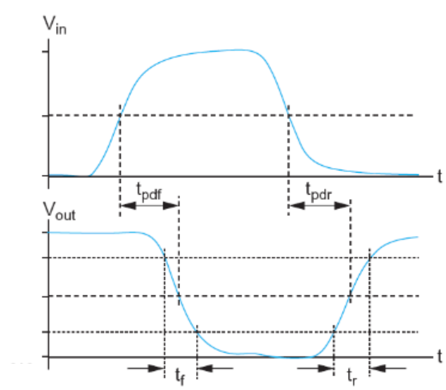
If you right click on the plot and select *Send to -> Calculator* it will send that plot name to the calculator for further analysis. Much easier than trying to select it later.

a. Propagation Delay Rise and Fall (PDR and PDF) times are how long the signal takes to propagate at 50% of the amplitude. Average Propagation time simply averages these two values

b. Rise Time is how long the signal takes to propagate from 20% of the amplitude to 80% while switching states.

c. Fall Time is how long the signal takes to propagate from 80% of the amplitude to 20% while switching states.

Rise Time (t_r)
Fall Time (t_f)
Propagation Delay Rising (t_{pdr})
Propagation Delay Falling (t_{pdf})
Avg. Prop. Delay (t_{pd}) $= \frac{(t_{pdf}+t_{pdf})}{2}$

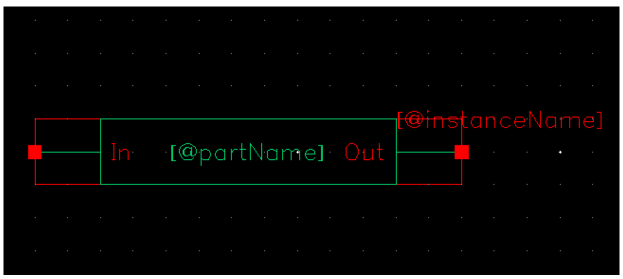


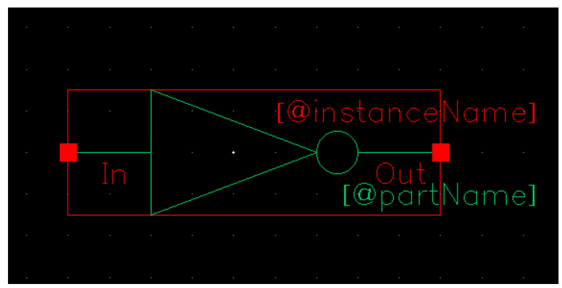
d. Use the calculator to compute the Propagation Delay times and the Rise and Fall Times
i. Use VT("/in") and VT("/out") to capture the Transient Voltage Signal
ii. Hint: The *Cross*, *Delay*, and *DelayMeasure* Functions will be useful here
iii. **Please neatly include your functions in your report**

e. <u>Verify</u> your results by manually computing the delay from the output plot
i. Hint: Markers can be placed on the output to reveal all intersecting points at a certain voltage by pressing **H** or selecting the **Markers drop-down** Menu

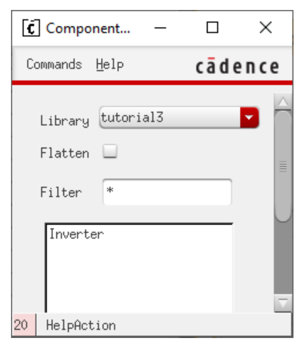
5. Creating a Symbol
Refer to this site for updated figures: https://www.acsu.buffalo.edu/~ajr33/cse-493_593/VirtuosoTutorial.html

- **a.** <u>In the Schematic View – There are additional steps if you do this from the library manager</u>
  - i. Create -> Cellview -> From Cell View
- **b.** Select *schematicSymbol* and click okay
- **c.** It will open an options window that allows you to specify pin locations.
  - i. (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.)
- **d.** This will open the *Symbol Editor* which should look like the image on the left
- **e.** Edit the symbol to resemble a logical inverter such as the image on the right, then check and save your symbol.

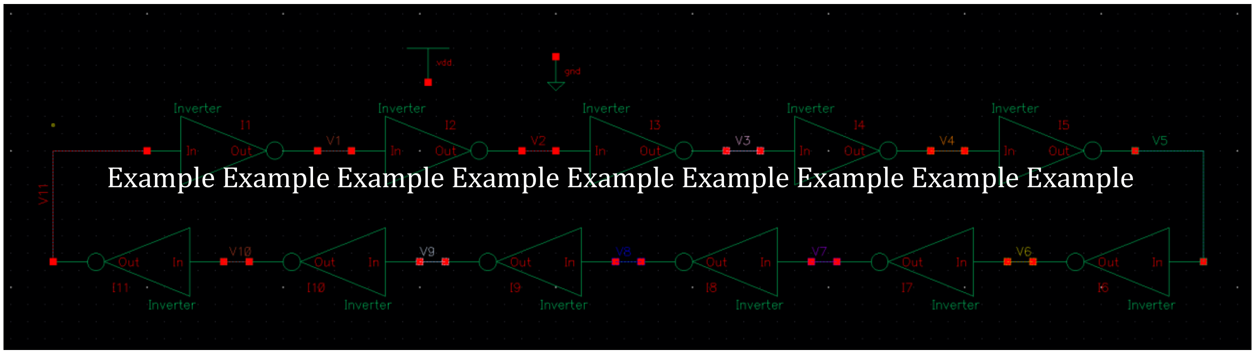




6. Create a Ring Oscillator using your new Symbol



- **a.** Create a **new schematic** called *osc_inv1* <u>in the same library as your schematic and symbol.</u>
- **b.** Place 11 copies of your inverter symbol (inv1) and connect them in a series loop.
  - i. Select your inverter symbol from **tutorial 3** library in the component browser
  - ii. A loop with an odd number of inverters forms an oscillator. Signal delay is introduced by each inverter.
  - iii. Name the nets in between each invert with v1:v11
- **c.** Place symbols of unconnected *vdd* and *gnd* from the gpdk Library.
  - i. Note this does nothing in the drawn circuit, but sets the global nets, vdd! and gnd!, that are used within each inverter symbol.



d. With 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 to ***osc_inv1***. In order to do that, from the ***osc_inv1*** schematic, you can click on an ***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.

i. **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

7. Transient Simulation of the Ring Oscillator

a. Launch ADE GXL and create a new view

b. Add a test under Tests and Open the Test Editor and choose your Oscillator schematic

c. Setup the Analog Stimuli

i. Setup -> Stimuli -> Global Sources

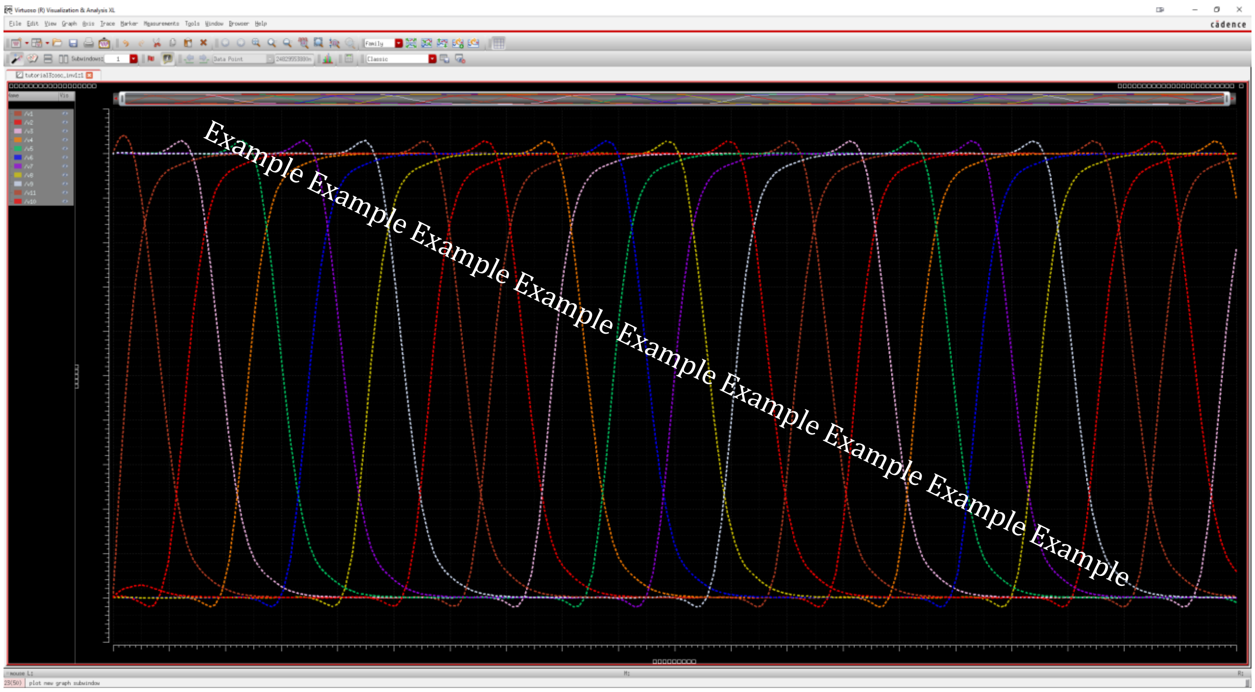
d. Set the analysis to a transient analysis with a stop time of **4ns**

Global Source Vdd	
Function	dc
DC Voltage	5 V

e. Set an initial condition from the ADE Editor, choose Simulation -> Convergence Aids -> Initial Condition.

i. 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, Condition, then click on a node in the schematic. A big 0 should appear on the net. ESC and click OK

f. Run the simulation and produce delays over every inverter as shown below. Use this combined output to further analyze the ring oscillator.



g. Re-measure the Propagation Delays, Rise Time, and Fall Time of an inverter in the Ring Oscillator and compare your results with the individual inverter delays.

h. Use the calculator to find the frequency (and the total delay) of the Ring Oscillator and compare this experimental value to the equation for Ring Oscillator frequency:

$$f = \frac{1}{2tn}$$

Where *t* is the delay of a single inverter and *n* is the number of inverters.