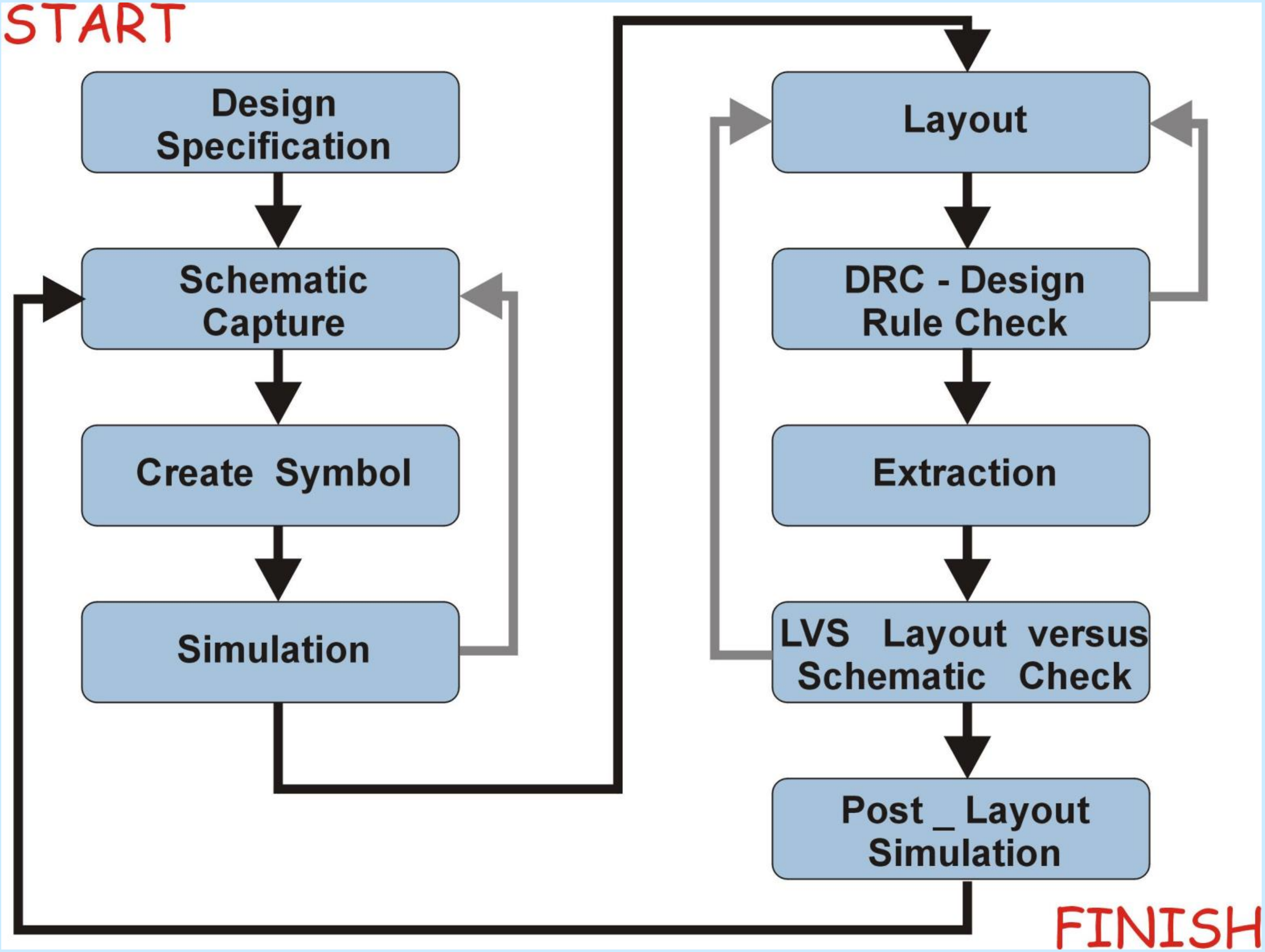
Micromagic

# ~~CADENCE~~ Tools

**Lecture bases on CADENCE Design Tools Tutorial**

**http://vlsi.wpi.edu/cds**

# Analog design flow




7hAdder

# CADENCE Tools for IC Design

Cadence is a set of different design tools used at different stages of the design process.

**Which tools we exactly need ?**

Sue ← ~~Composer~~ -> *Schematic editor*

Max ← ~~Virtuoso~~ -> *Layout editor*

~~Analog Artist -> preparing simulation~~ → ciw LibraryMgr or other single TCL file

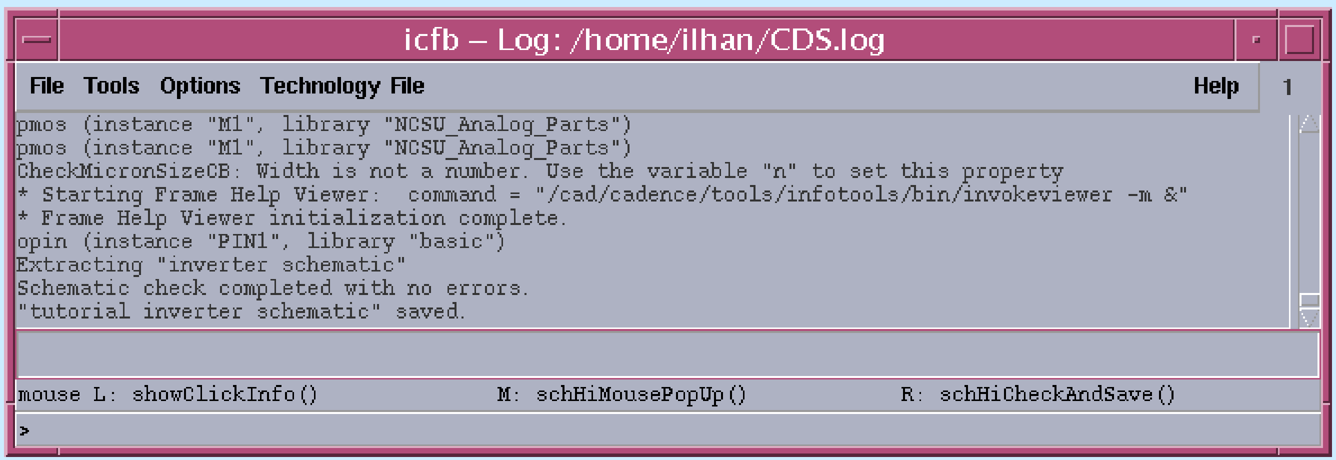
*(SpectreS in this tutorial)*

~~DIVA~~ -> *Design Rule Check (DRC) , Layout Versus Schematic Check (LVS) , Extraction*

micromagic built-in gemini tool.
It is not Google Gemini !!

# First steps with Cadence

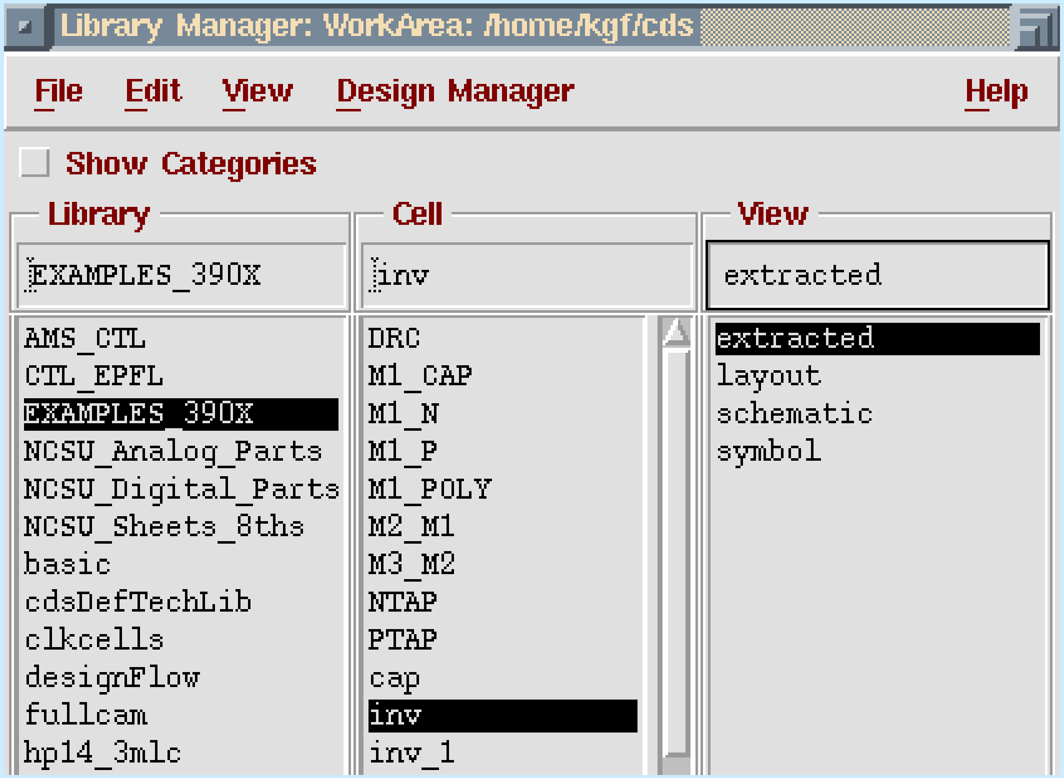
- To start Cadence software enter: ~~icfb~~ & CIW
- The cadence window after start up:




- To see your cadence libraries open Library Manager from „Tools menu"

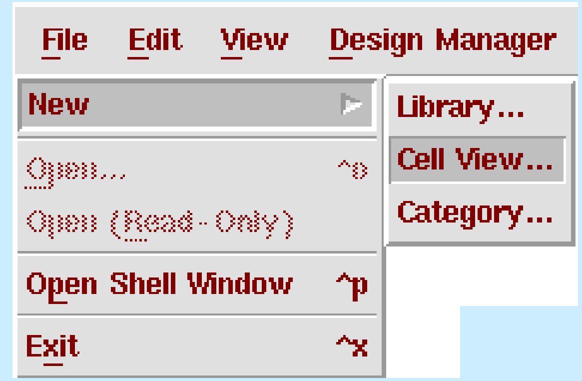
# First steps with Cadence

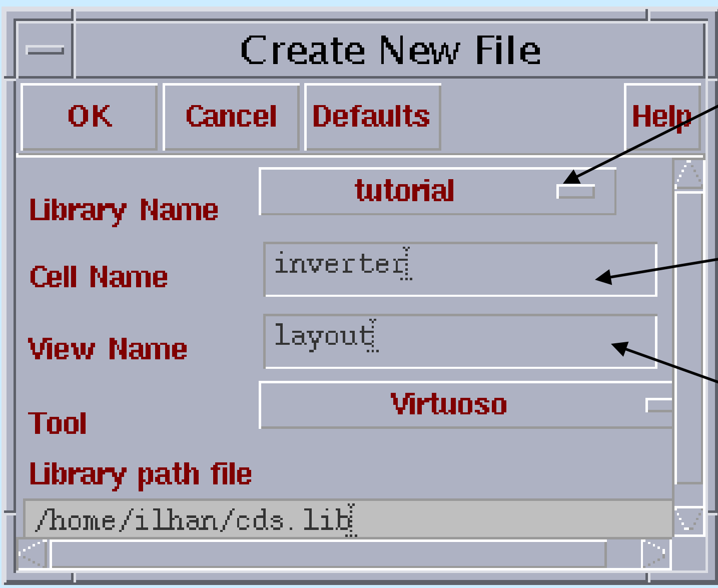
- Cadence libraries:
  - Technological Libraries (usually names written with upper cases)
  - User Libraries
- Libraries contain cells – basic elements of your design
- Every cell may have different views – different views created at different stages of the design and edited with different tools



# Creating new libraries and cells

1. In *File* on the menu of *Library Manager* choose *New* -> *Library* or *Cellview*.
2. Examplarty „Creat New File window"




**Library Name**
Choose your working directory and library name

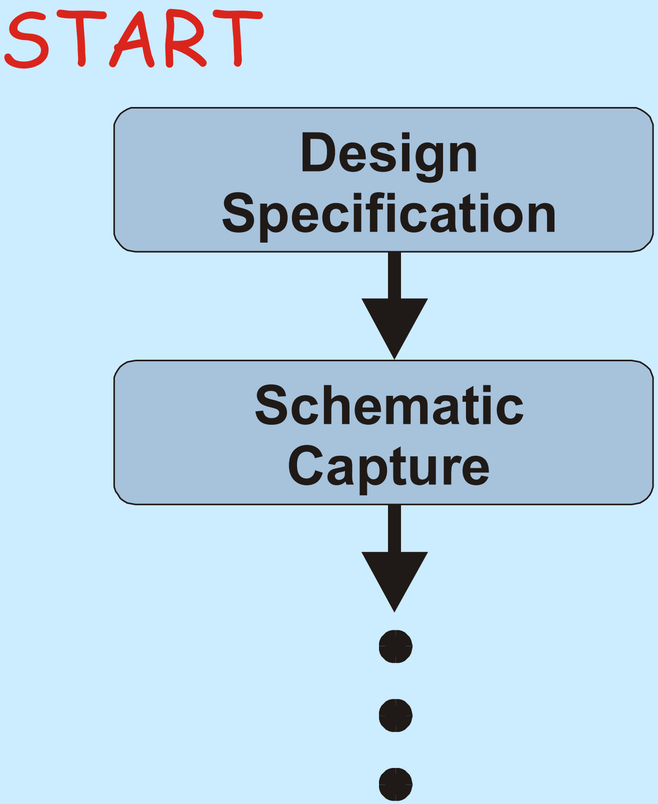
**Cell Name**
Enter the cell name

**View Name**
Enter view name depending onthe level of the design hierarchy.
Eg. schematic or layout

# Design Example

## CMOS - Inverter

# Creating Schematics

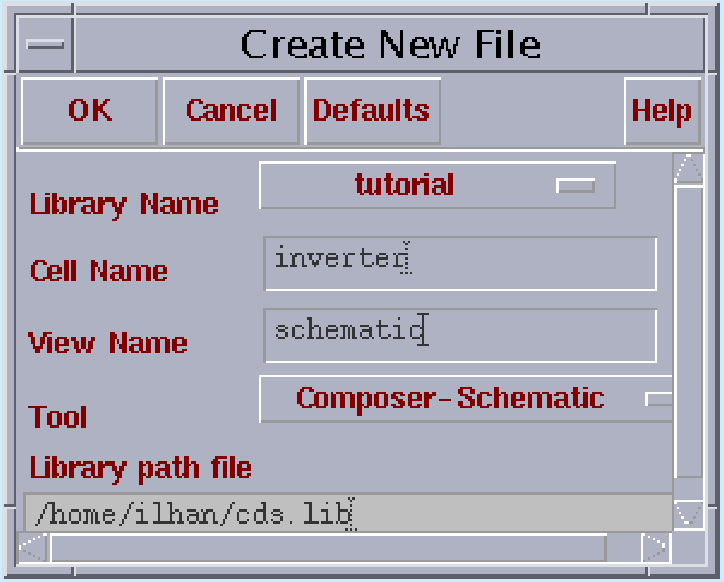



- The traditional method for capturing (i.e. describing) your transistor-level or gate-level design is via the schematic editor.
- Schematic editors provide simple, intuitive means to draw, to place and to connect individual components that make up your design.
- The resulting schematic drawing must describe the main electrical properties of all components and their interconnections.

# Open a new schematic window

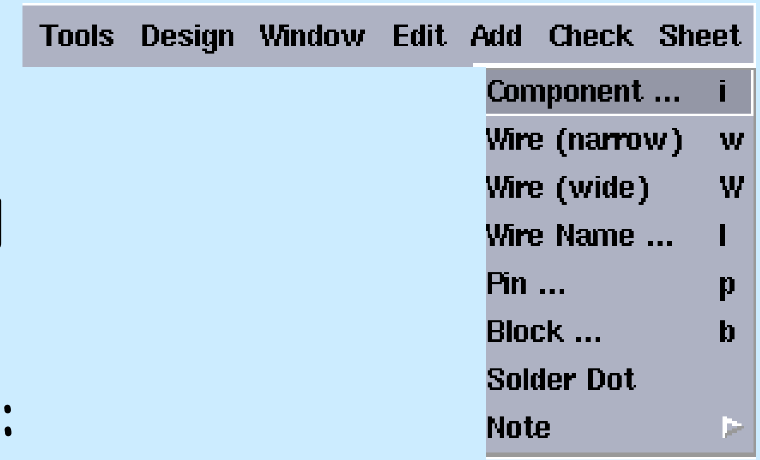
- **View Name**

Indicates the level of the design hierarchy.The correct view name choice is "schematic" for our example.




*Click OK to finish*

# Add components

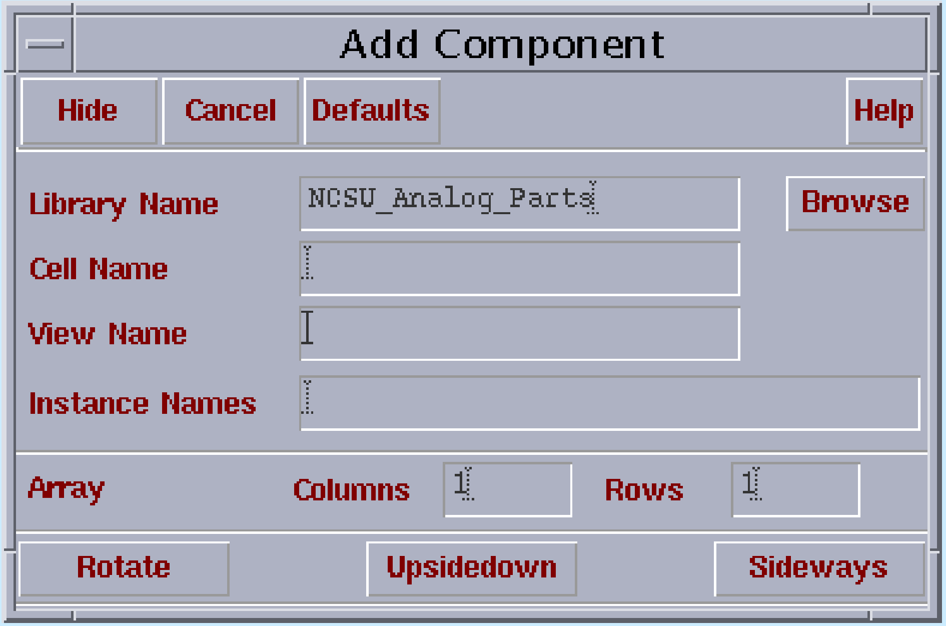


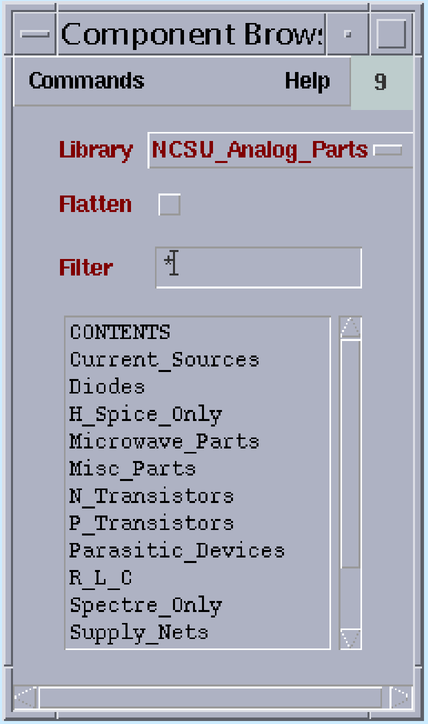

**The first thing to do is to add and place components which will be used in the schematic.**

We need the componets as folow:

- PMOS : p-type MOSFET
- NMOS : n-type MOSFET
- VDD : vdd! global net marker
- GND : gnd! global net marker

# Add components




- **Add Component Window**
  Enter the *Library Name*, *Cell Name* and the *View Name* of the component

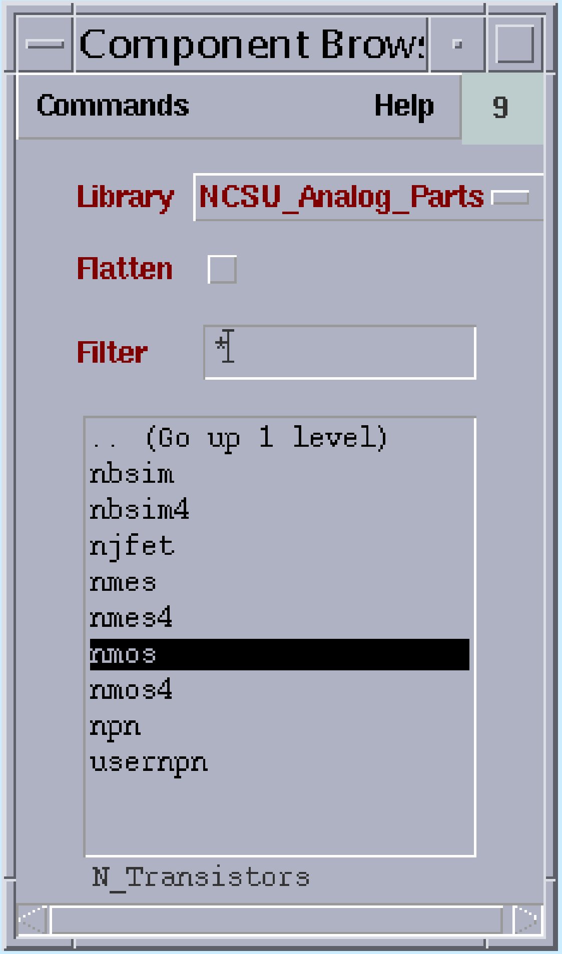
- **Component Browser** enables the designer to browse easily through the available libraries and select the desired components.

# Add components

**Pick up the MOS transistors from the *Component Browser* window.**

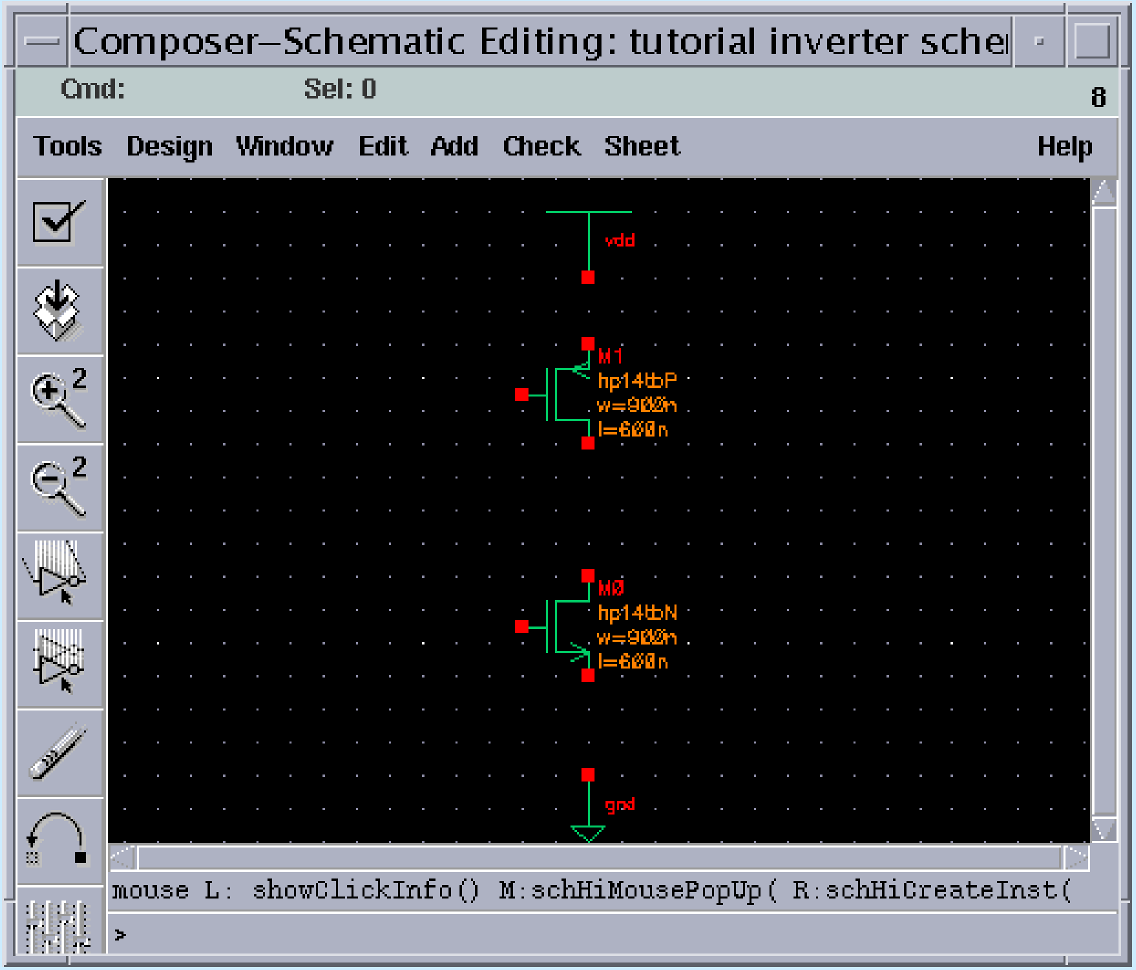
•Open the "N_Transistors" folder by clicking once on it.

• Pick up the NMOS transistor by clicking once on "nmos", which is a model for a three terminal n-type MOSFET.



# Add components

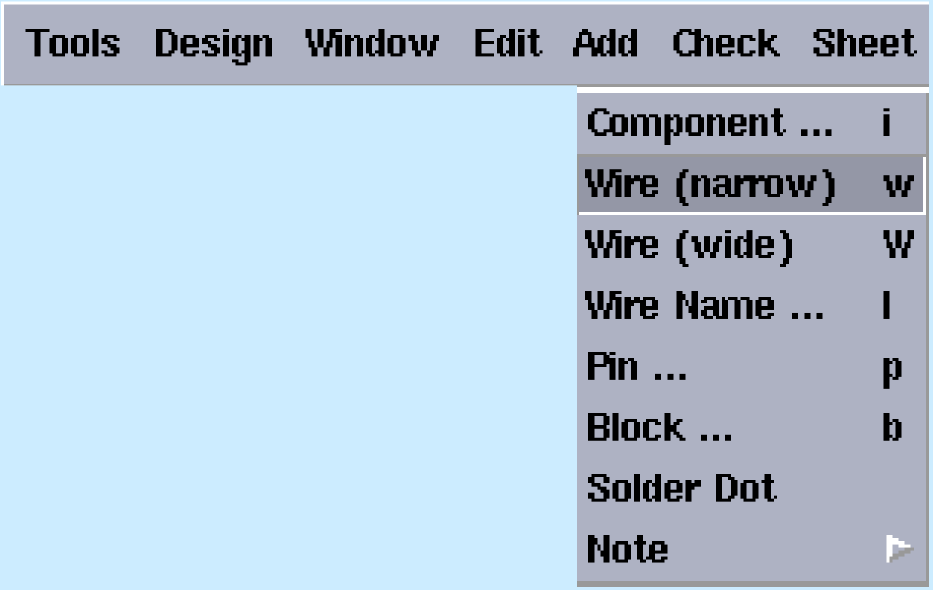
- Click on a location in the schematic window, where you want to put the transistor.
- Use the same procedure to select and to place the PMOS transistor.




***Picking up the supply voltage components involves the same steps as in adding transistors to the schematic.***

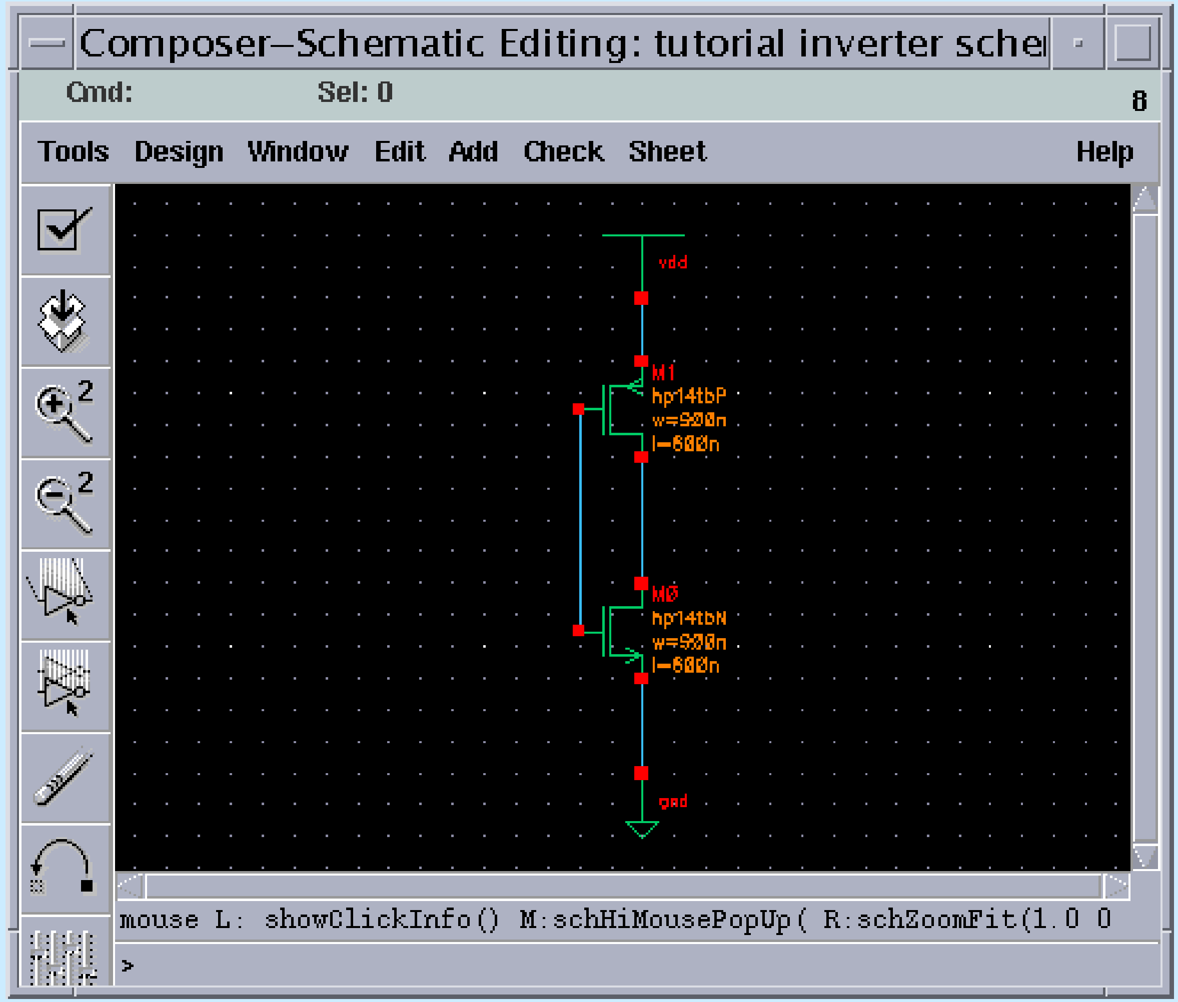
# Conecting components

To connect the components in a schematic, we use wires by choosing *Add* and then *Wire (narrow)* on the menu banner.



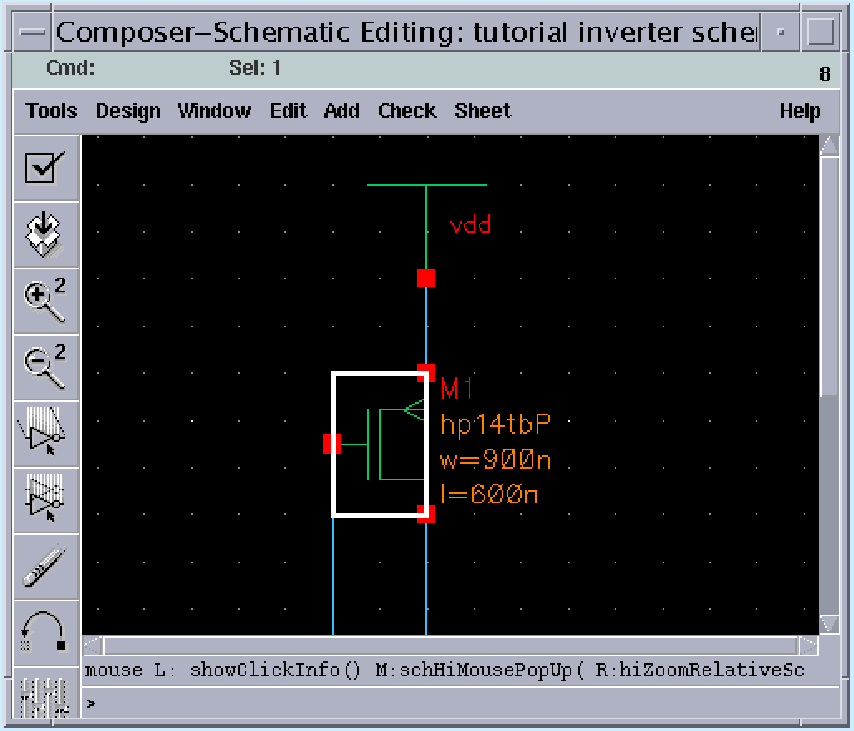
# Conecting components

Connecting any two nets in the schematic is done by first clicking at one of the nets and then at the other one.




Press ESC key to leave the wiring mode.

# Edit properties of components

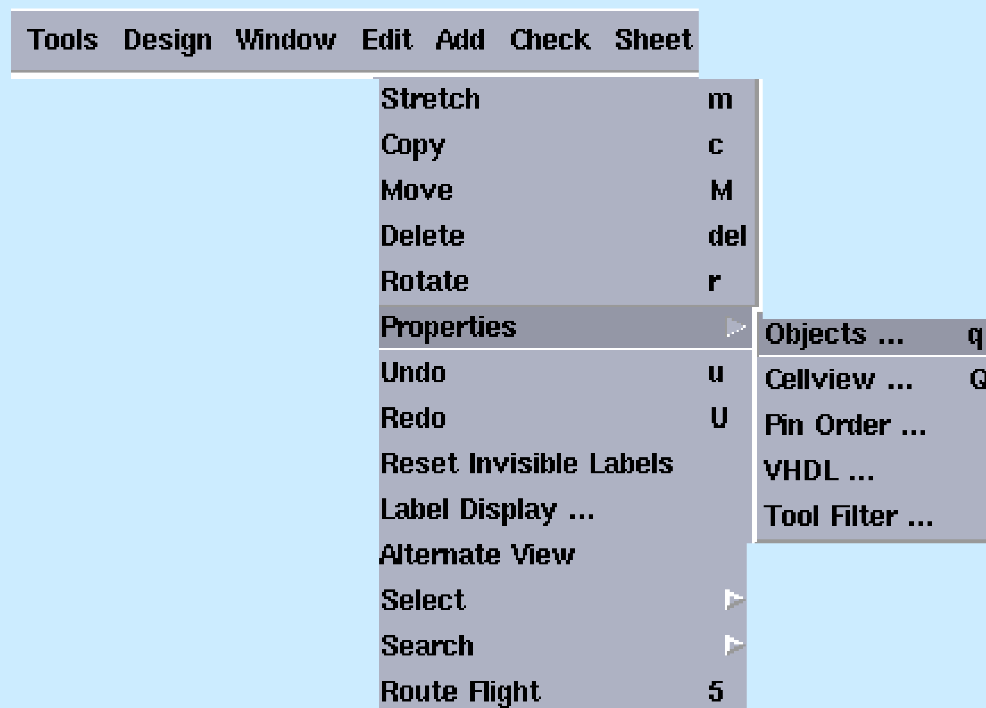



<= 1. ***Select component***

by clicking on it

2. ***Choose Properties =>***

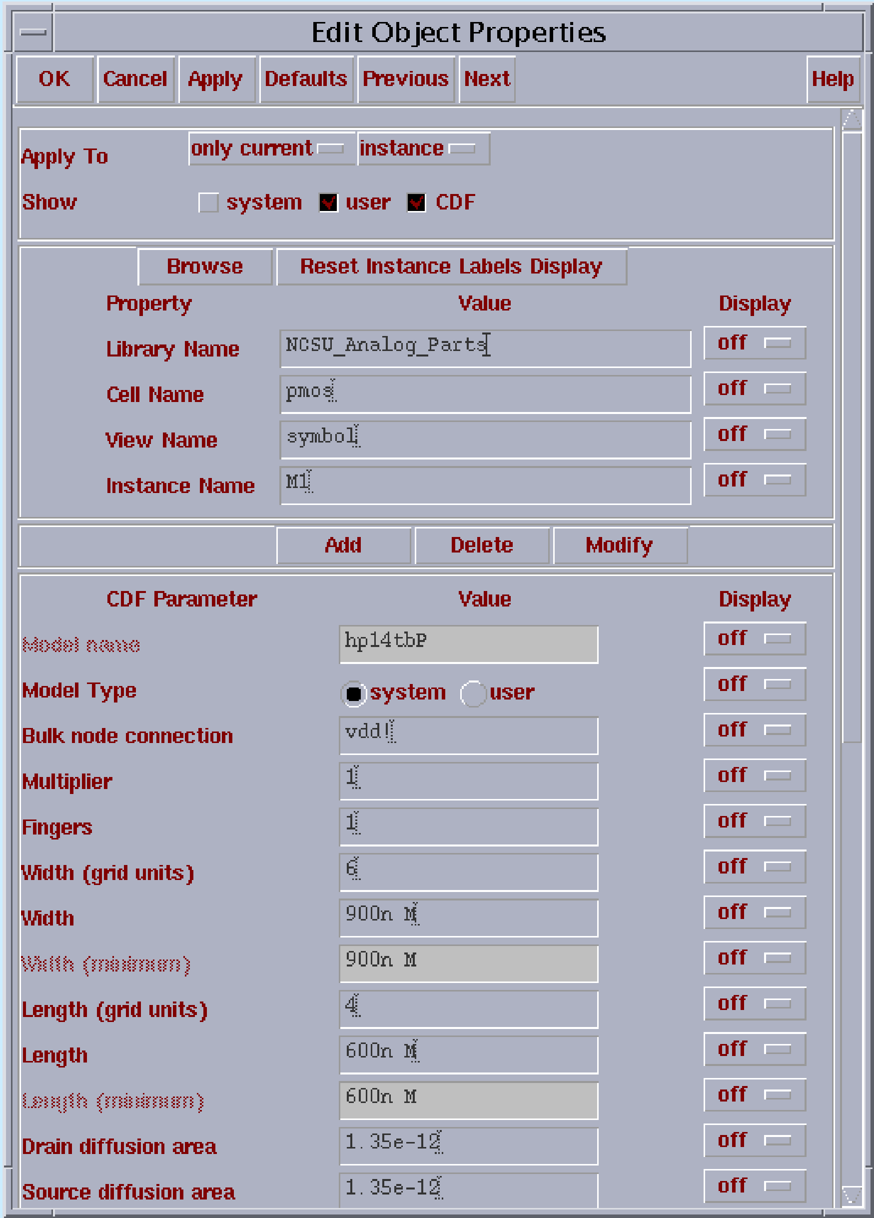
and then *Object* from the *Edit* menu.



# Edit properties of components

3. Edit the properties =>

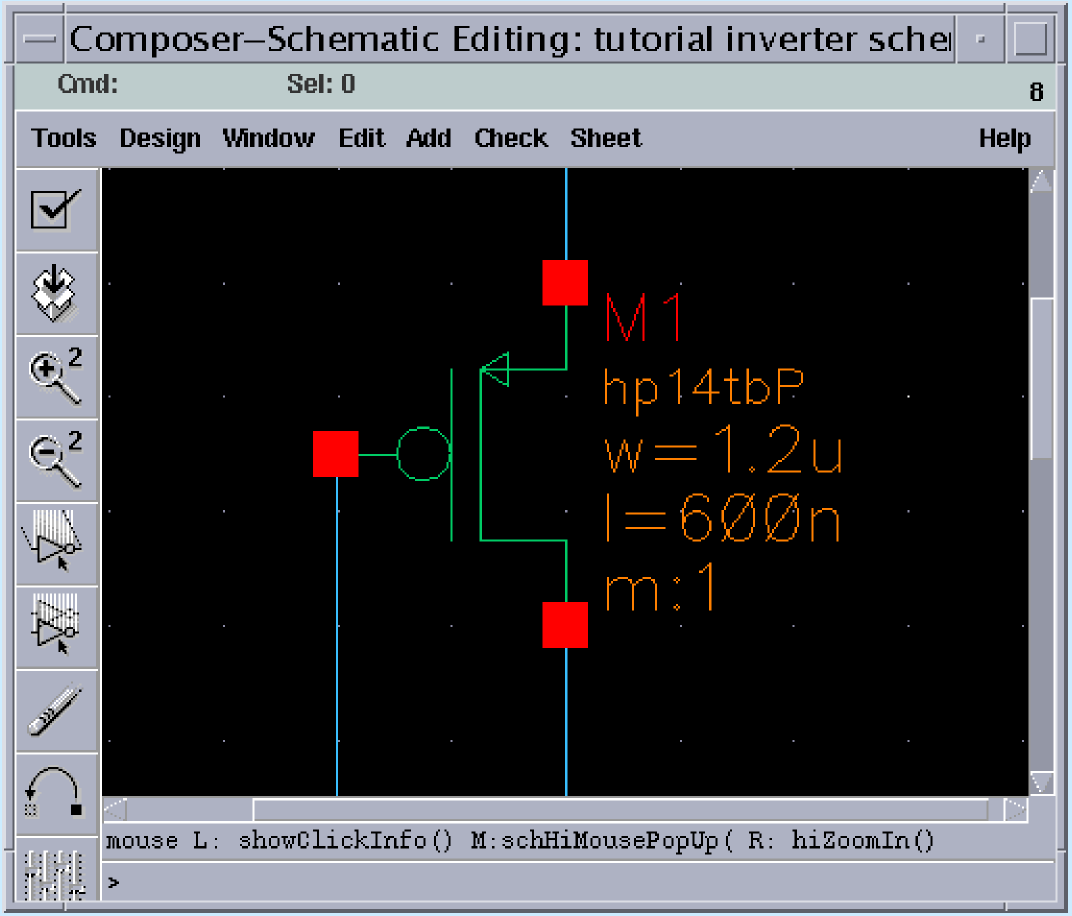
   by clicking on the corresponding field. You may change the values for *Width* or *Length* depending on your design specifications.



# Edit properties of components

**4. Click OK after => editing the properties**

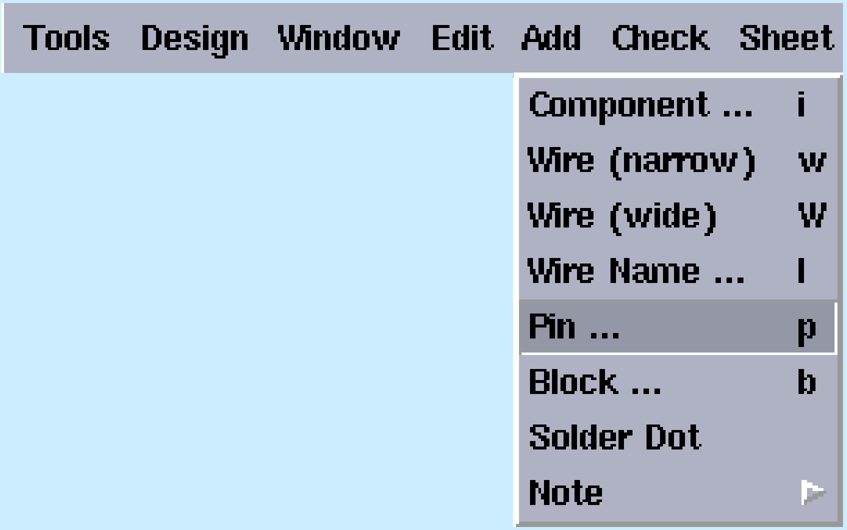
in the *Edit Object Properties Window.*




**The most important parameters always appear in the schematic window.**

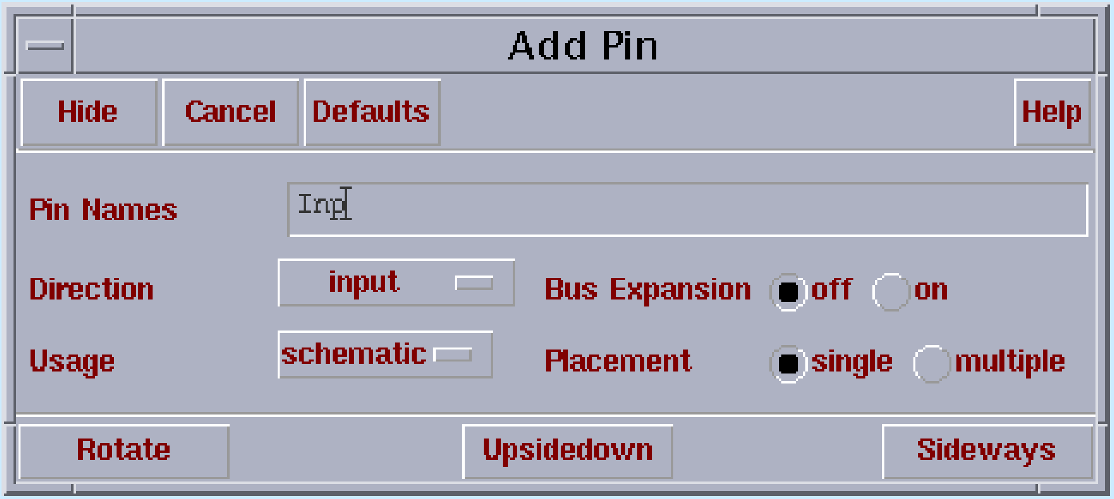
# Placing the pins

**You must place I/O pins in your schematic to identify the inputs and the outputs.**

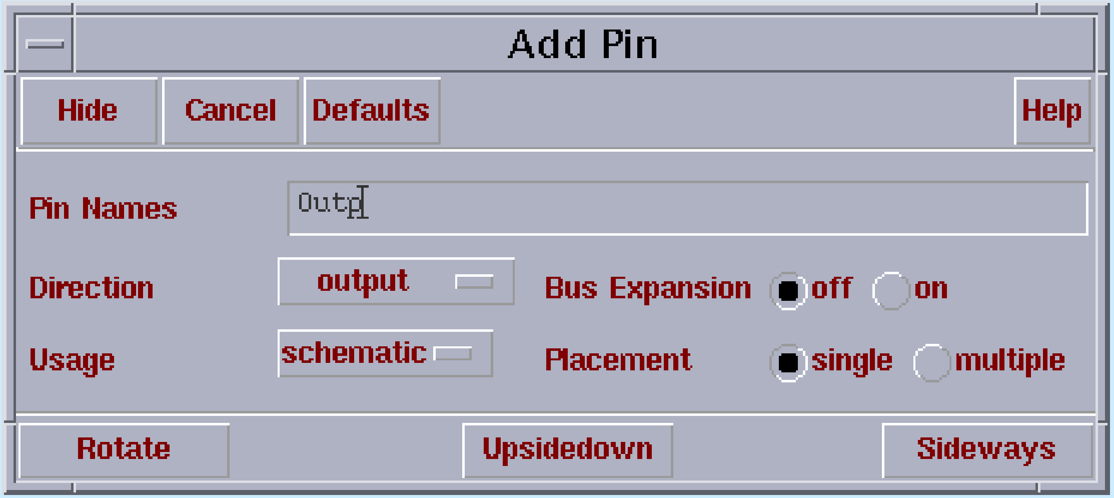



1. Click *Add* on the => menu and then select *Pin* on the pull-down menu.

# Placing the pins



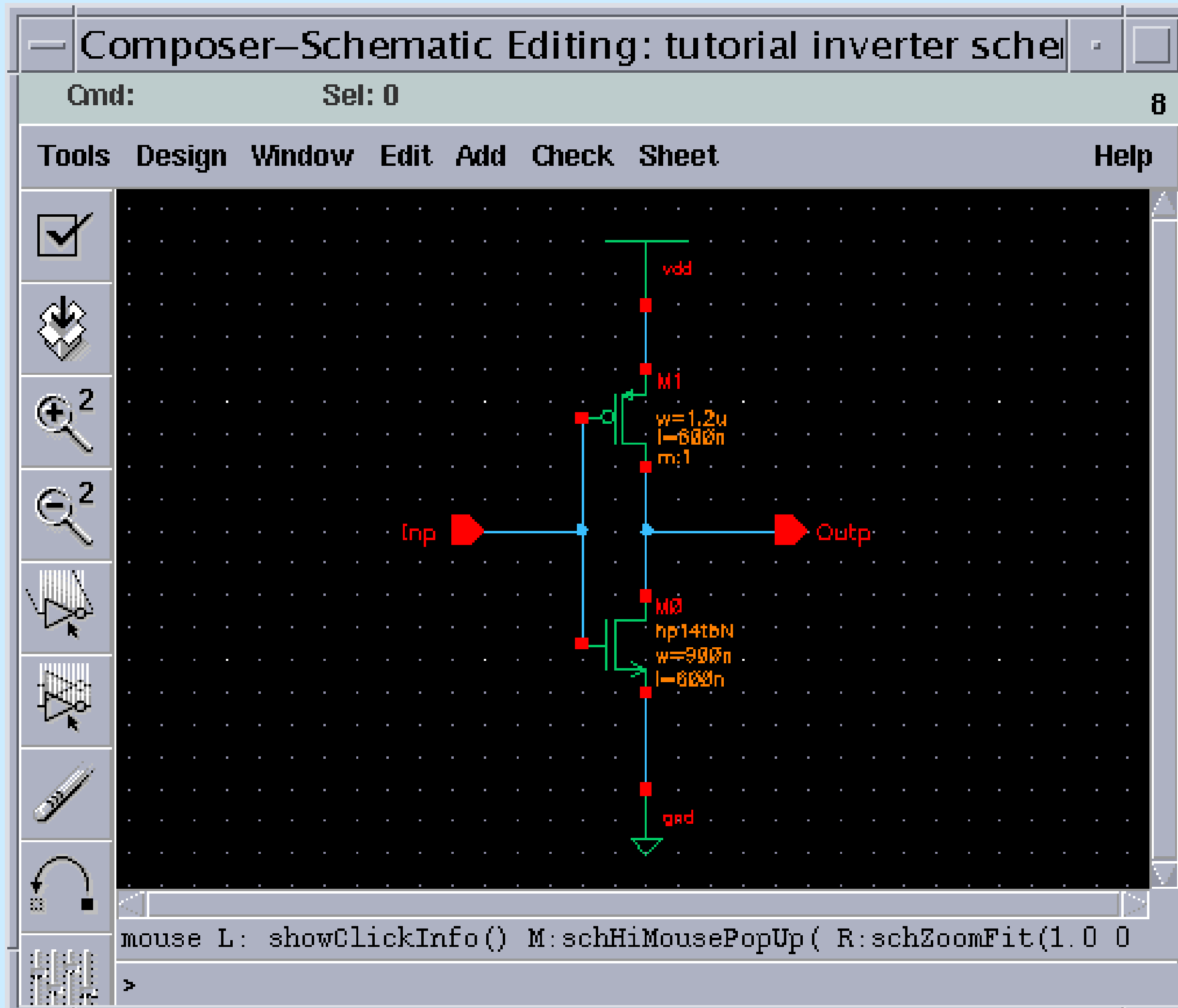

2. Enter the name of your pins in the *Pin Names* field. Choose the direction.




Place pins by clicking on a lcoation in the schematic window.

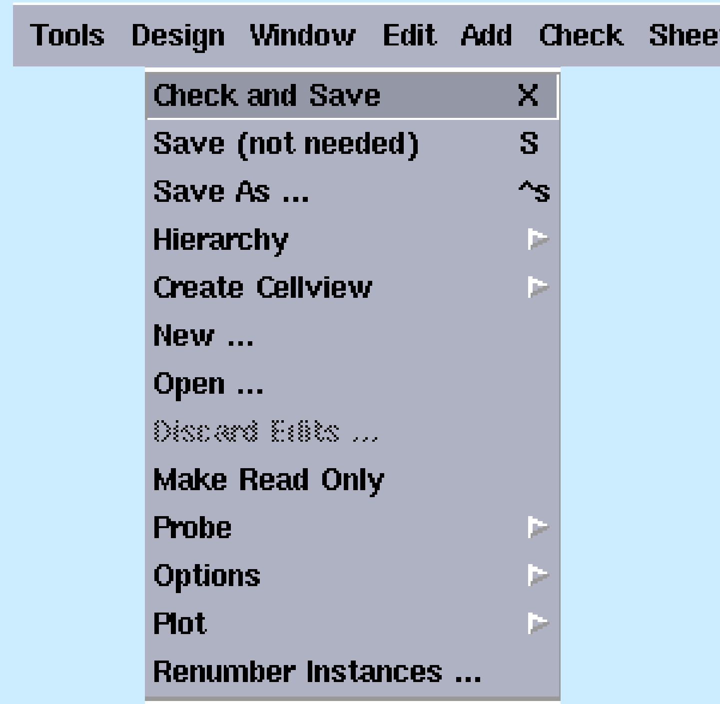
# Placing the pins

**Connect the => pins** to the corresponding nodes using wires.

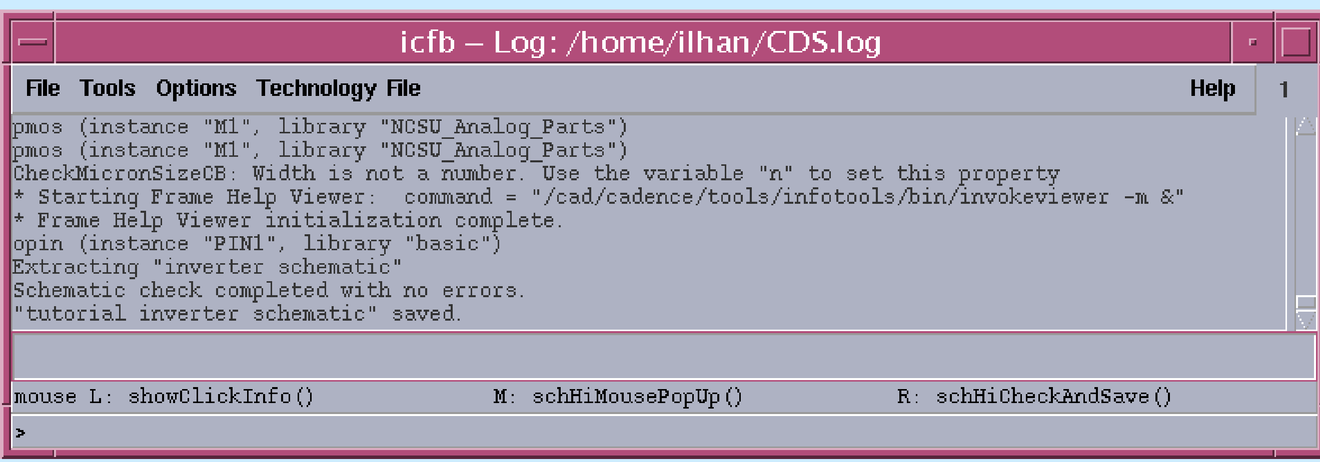



The wiring procedure is the same as described in the previous steps.

# Check and Save



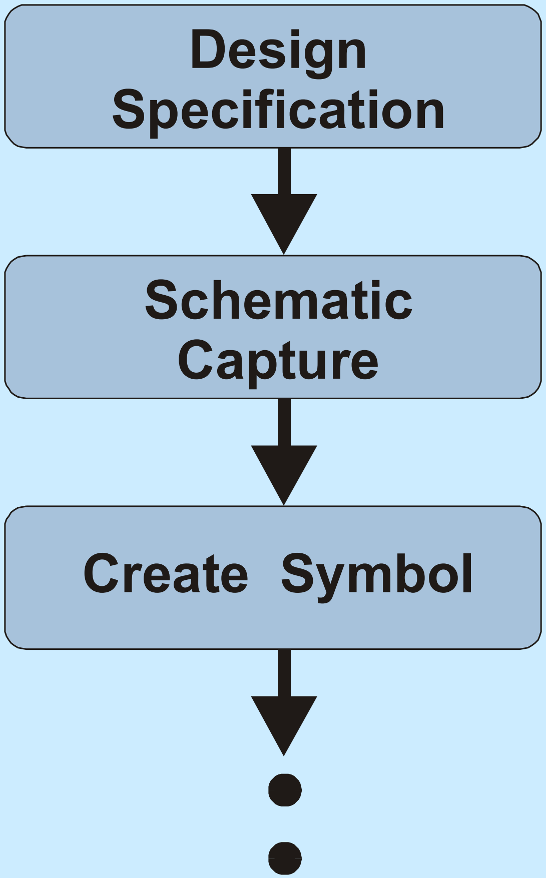

Click *Design* on the menu banner =>
and then select ***Check and Save***.




**<= Check the message** field every time you save a design.

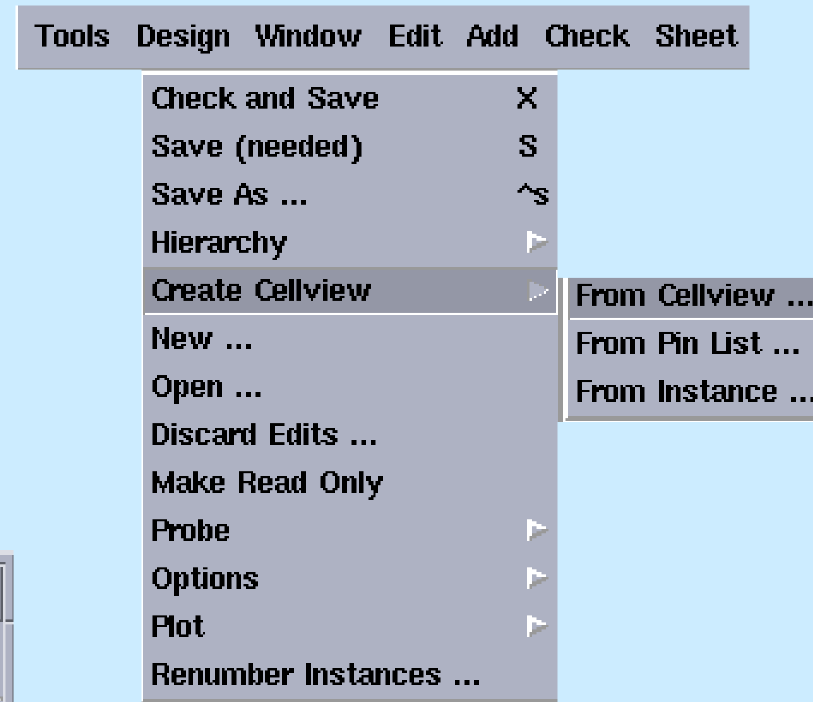
# Creating cellview

START




If a certain circuit design consists of smaller hierarchical components (or modules), it is usually very beneficial to identify such modules early in the design process and to assign each such module a corresponding symbol (or icon) to represent that circuit module.

# Creating cellview

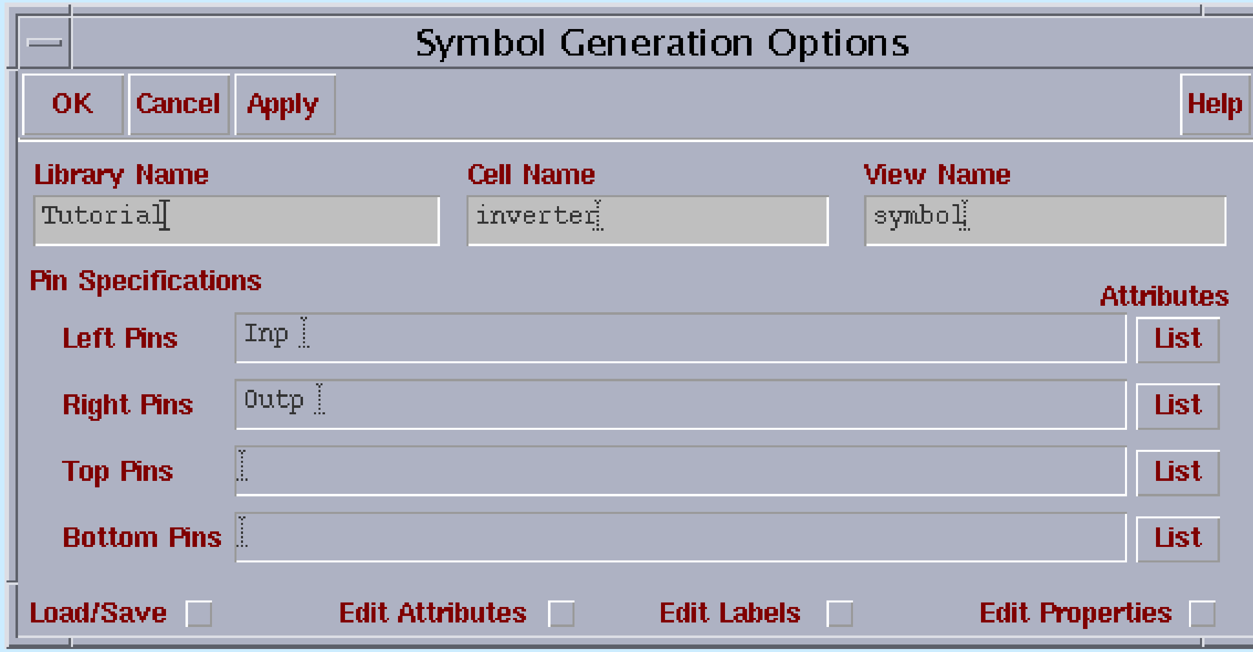



From the *Design* menu, select ***Create Cellview*** and then ***From => Cellview***


**<= Check the view names** You have to ensure that the *target view* name is *symbol.*

# Locating the pins

*After clicking OK in the Cellview From Cellview,* window the following window pops up :



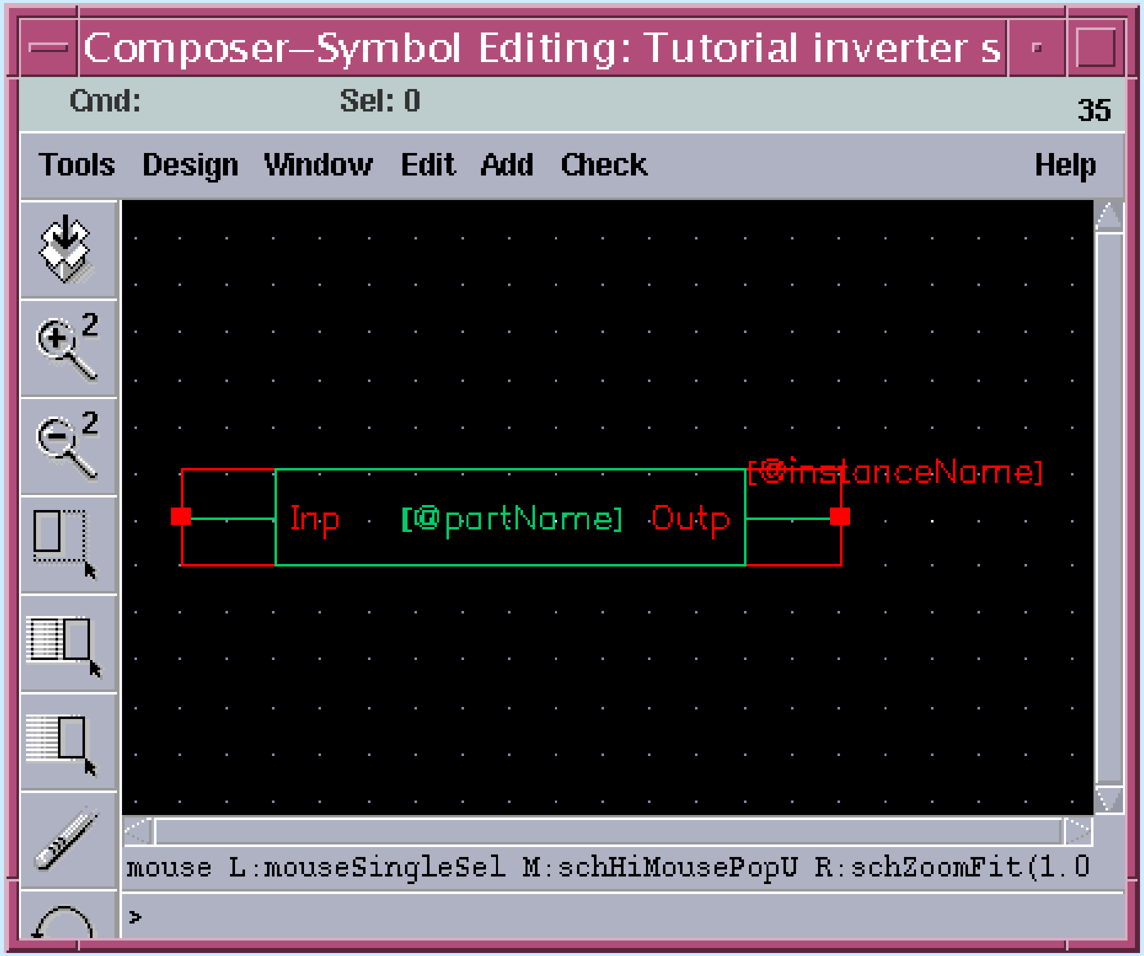

**Edit your pin attributes and locations.** In the default case, you will have your

- input(s) on the left of the symbol
- output(s) on the right of the symbol.

Change pin locations by putting the pin name in the corresponding pin location field

# Editing the shape of the symbol icon

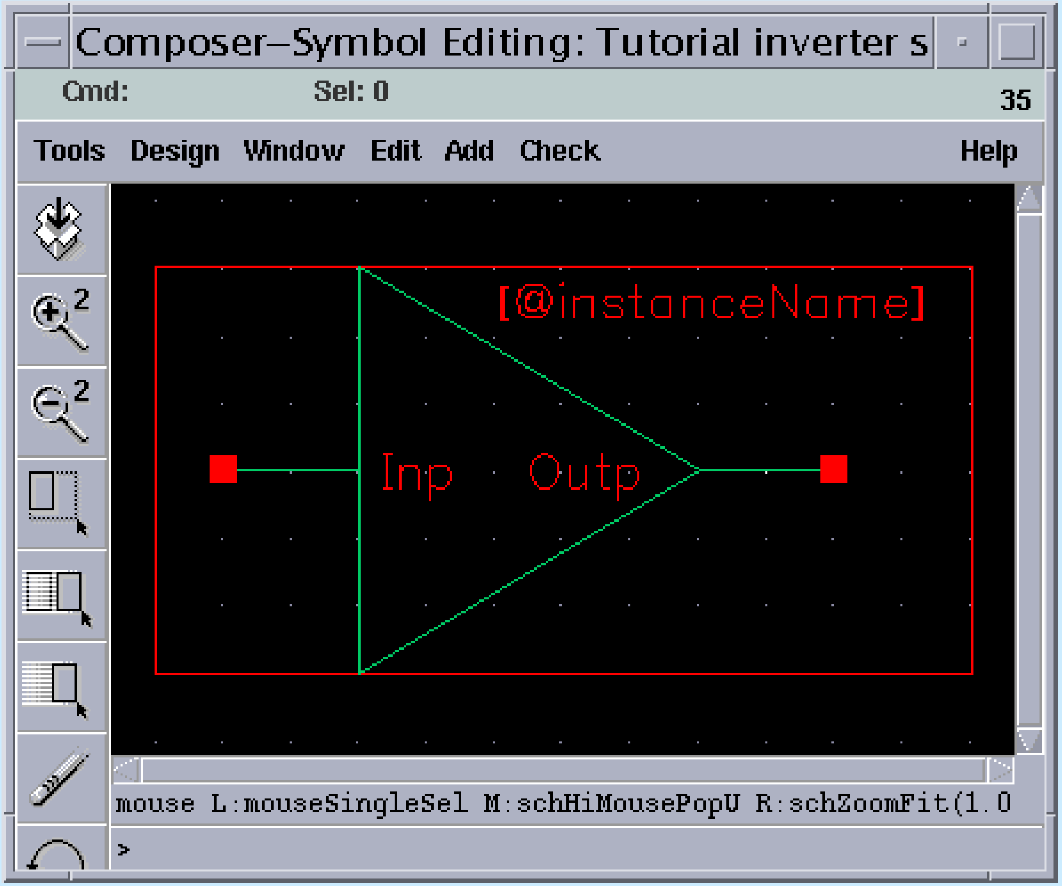
In the new window, the automatically generated symbol is shown.



# Editing the shape of the symbol icon

**You can do the following operations on your symbol**

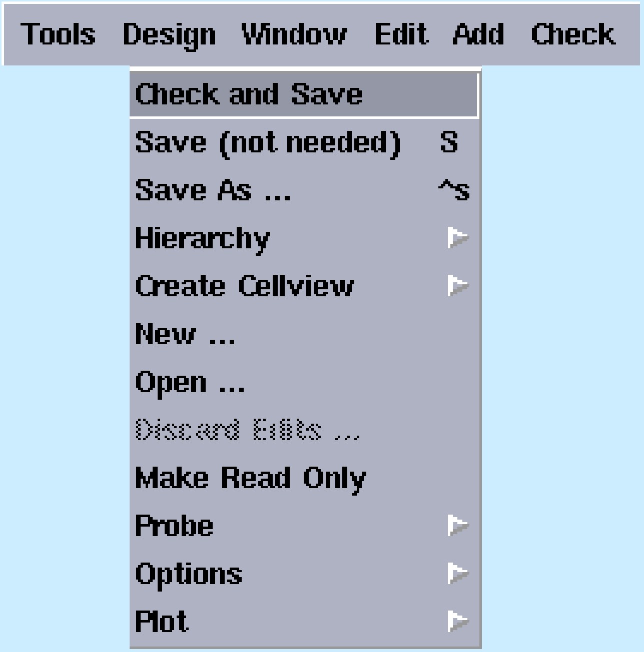
- Deleting/replacing some existing parts
- Adding new geometric shapes
- Changing the locations for pins and instance name
- Adding new labels



# Check and Save – Once Again

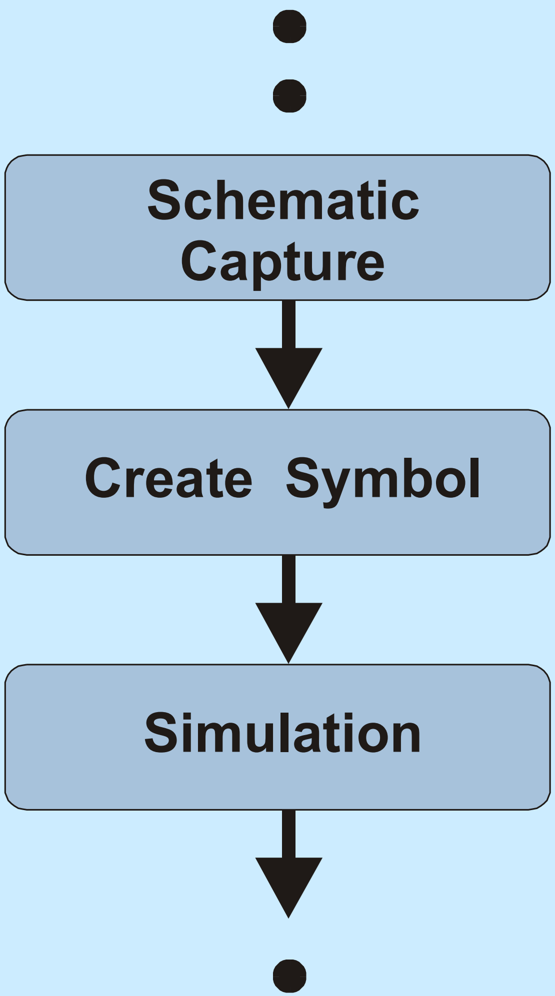
• ***Save*** – doesn't check anything.

•***Check****ing* a symbol means comparing the symbol view with the corresponding schematic view, by matching all of the pin names.




To **check and save the symbol**, choose *Check and Save* from the *Design* menu

# Simulation




- The electrical performance and the functionality of the circuit must be verified using a Simulation tool.

- Based on simulation results, the designer usually modifies some of the device properties.

# Simulation -New Schematic Design

1. **Open a new schematic.**

• Follow the same procedure described in „Open a new schematic" to create a new schematic where you will put your simulation schematic for the inverter.

• Give a name to your new schematic which makes it clear that the new schematic is to *simulate* the inverter.

•**Note** : You should first create the *symbol* of the circuit schematic which you want to simulate

# Simulation - Select and place components

**The first step is to add and to place the components which will be used to simulate the inverter.**

The components we need for the simulation of the inverter are the following :

- Inverter - Symbol created for the inverter
- VDD - Power supply voltage
- GND - Ground line
- vdc - DC voltage source
- vpulse - Pulse waveform generator
- C - Capacitor

# Simulation - Select and place components

How to **pick up a *symbol*** from library, and to place it in the schematic ?

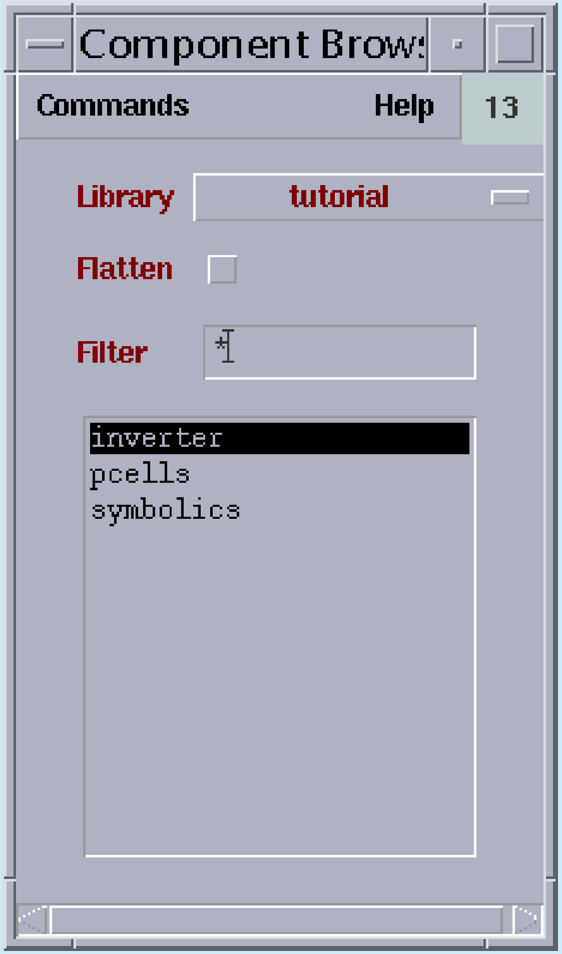

<= To pick up the inverter symbol, change the library of the *Component Browser* to library, "tutorial".

# Simulation - Select and place components

After the library "tutorial" is selected, there will be a new list of

• components which are included in this library

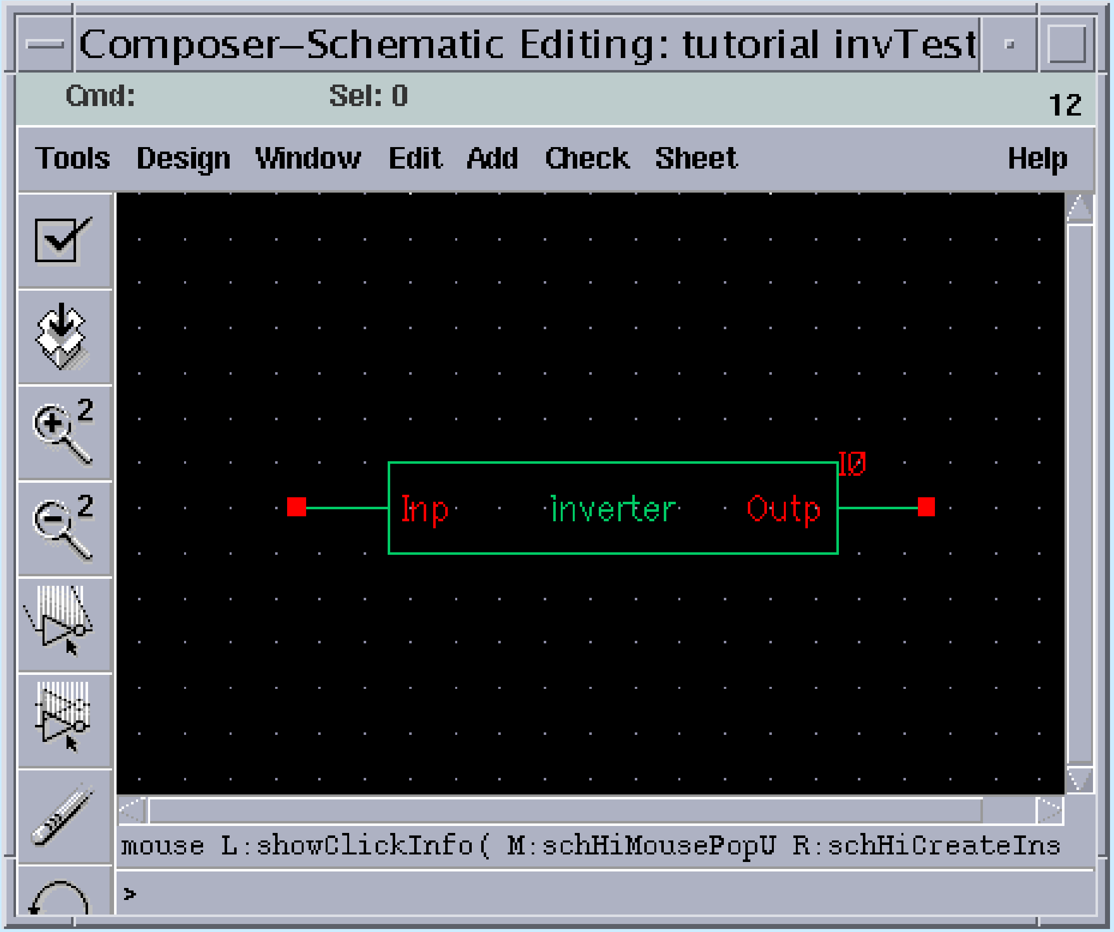
• every symbol that you created within this library will show up here.




By clicking on "inverter" in the component list in the *Component Browser*, you can pick up the symbol you created for the inverter

# Simulation - Select and place components

You can go to the schematic window and place the symbol of the inverter to a point by clicking on it



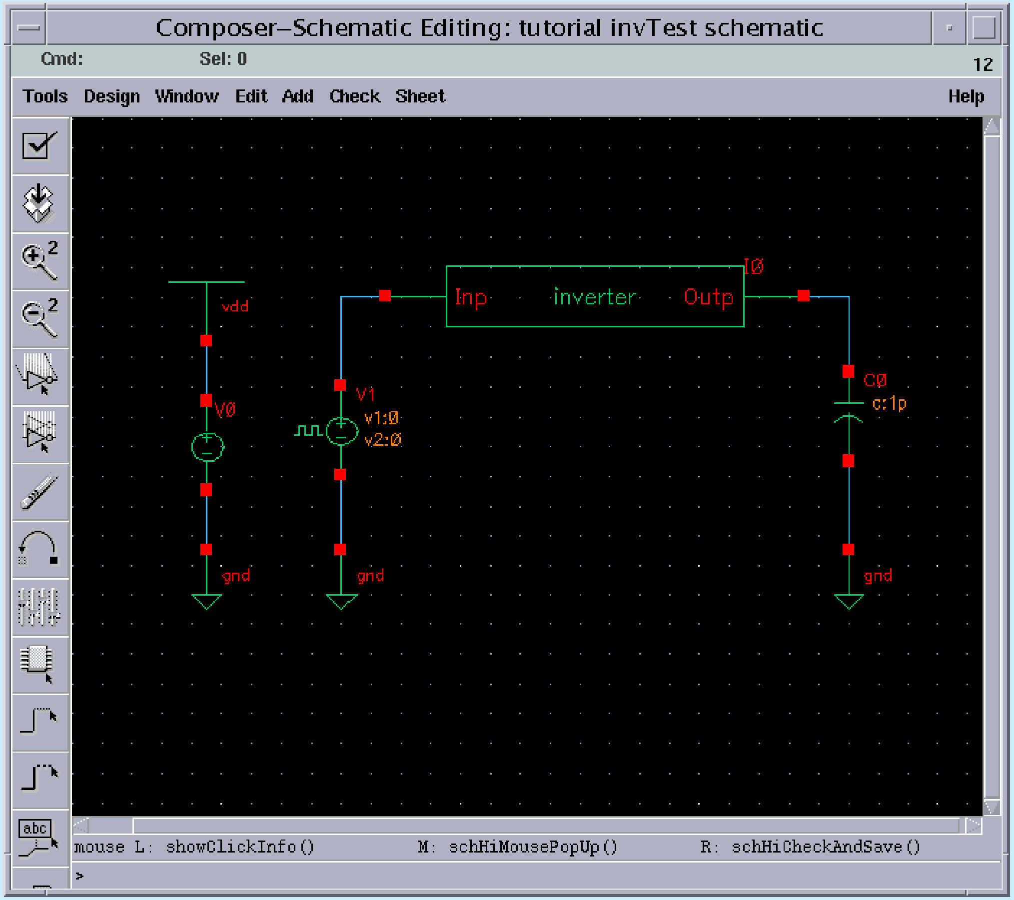
# Simulation - Select and place components

Pick up and place the rest of the components required for the simulation.

•Place the supply nets, "vdd" and "gnd".

•Place the voltage sources, "vdc" and "vpulse".

•Place the capacitance which will be the output load, "cap".



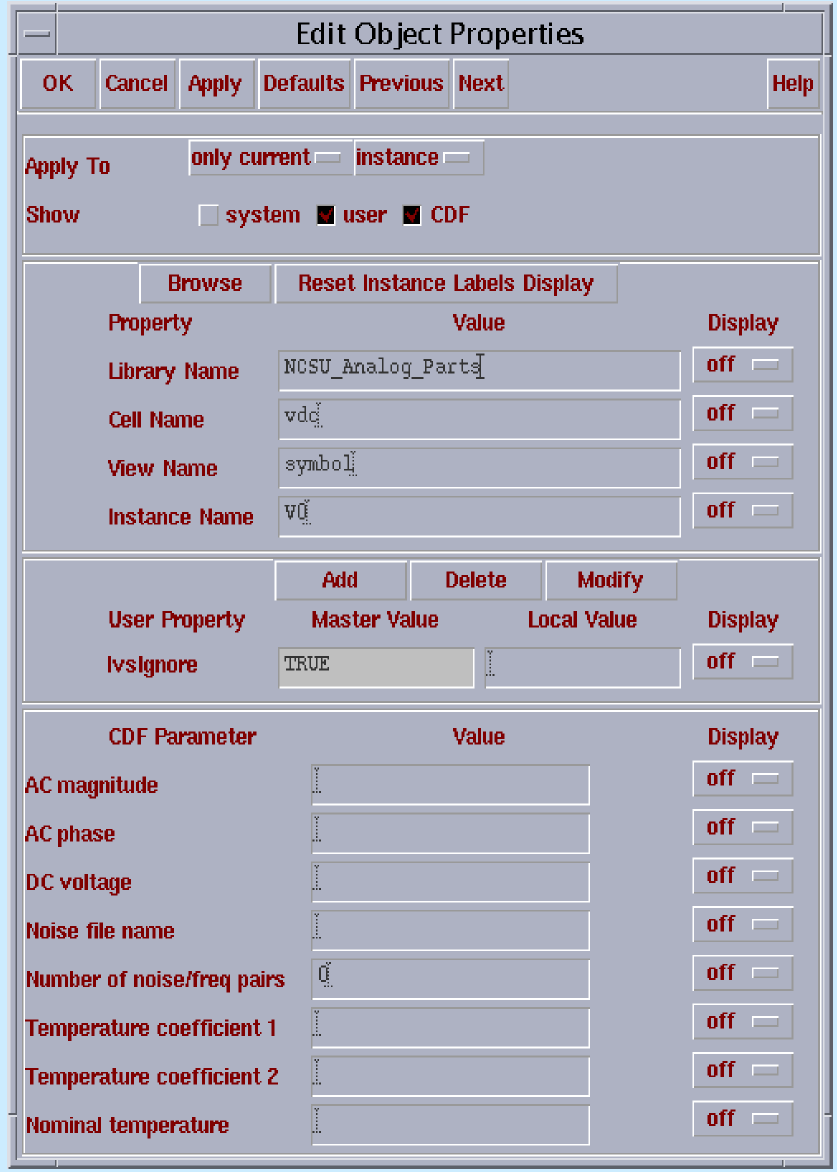

Match placed componets as was showed on the picture. Use the same method as previously.

# Simulation - Define the voltage source VDC

A DC-voltage source called "vdd" is required as the power supply voltage in all digital circuits.

The value of this voltage usually depends on the technology used.

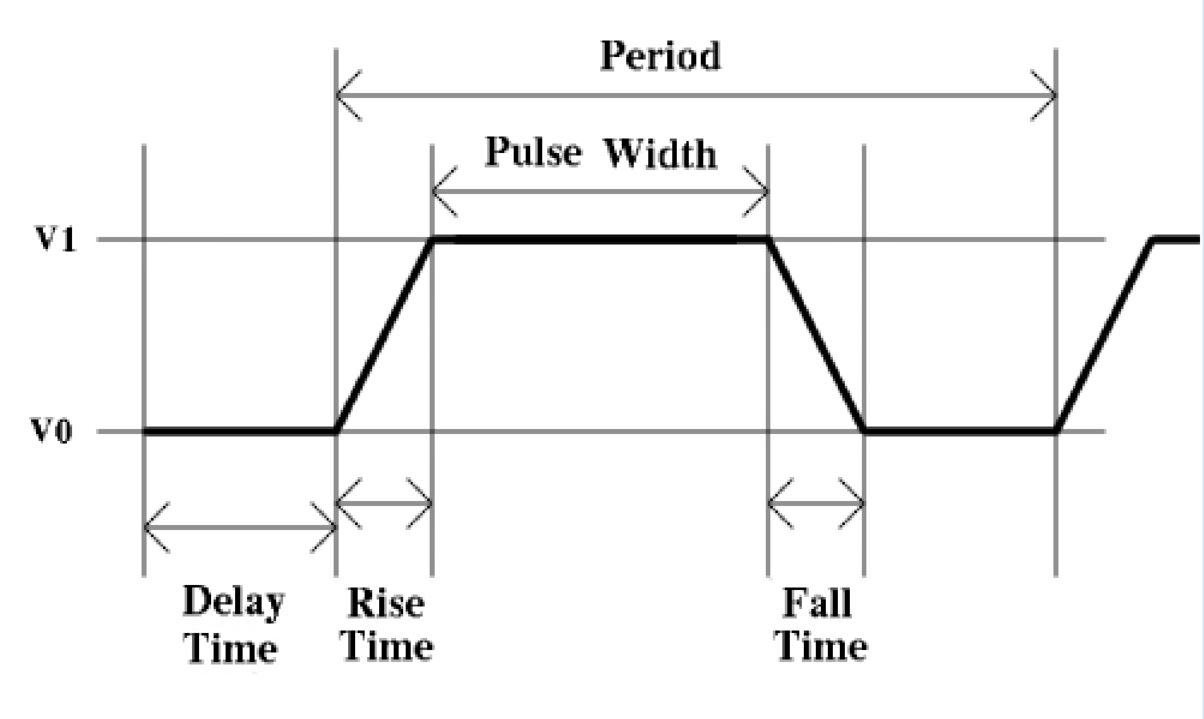
**Edit** the ***DC voltage*** field in=> the *Edit Object Properties* window and type the VDD value which is 3.3V



# Simulation - Define the voltage source Vpulse

- The pulse generator is a voltage source which can produce pulses of any duration, period and voltage levels.
- This source will be used to generate the input data

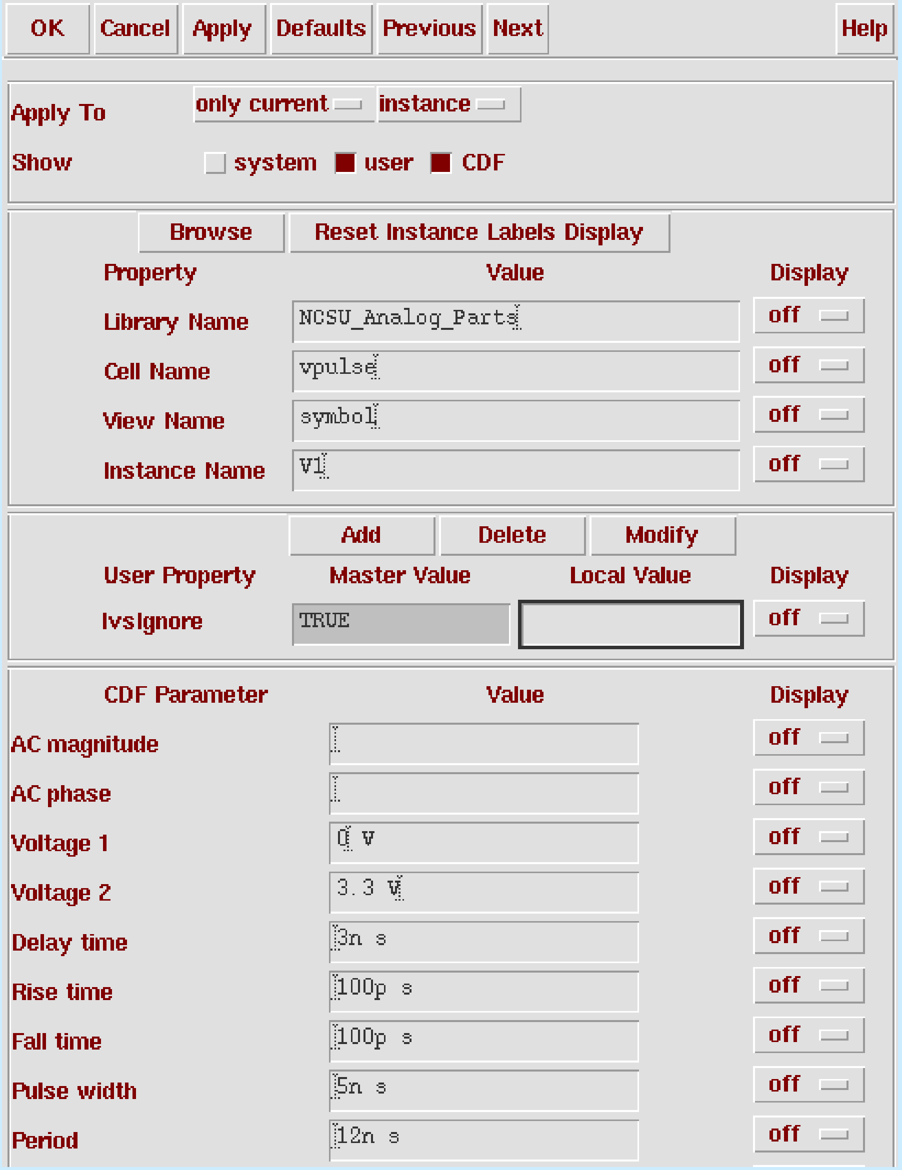
How to use parameters to define the input pulse waveform ?=>



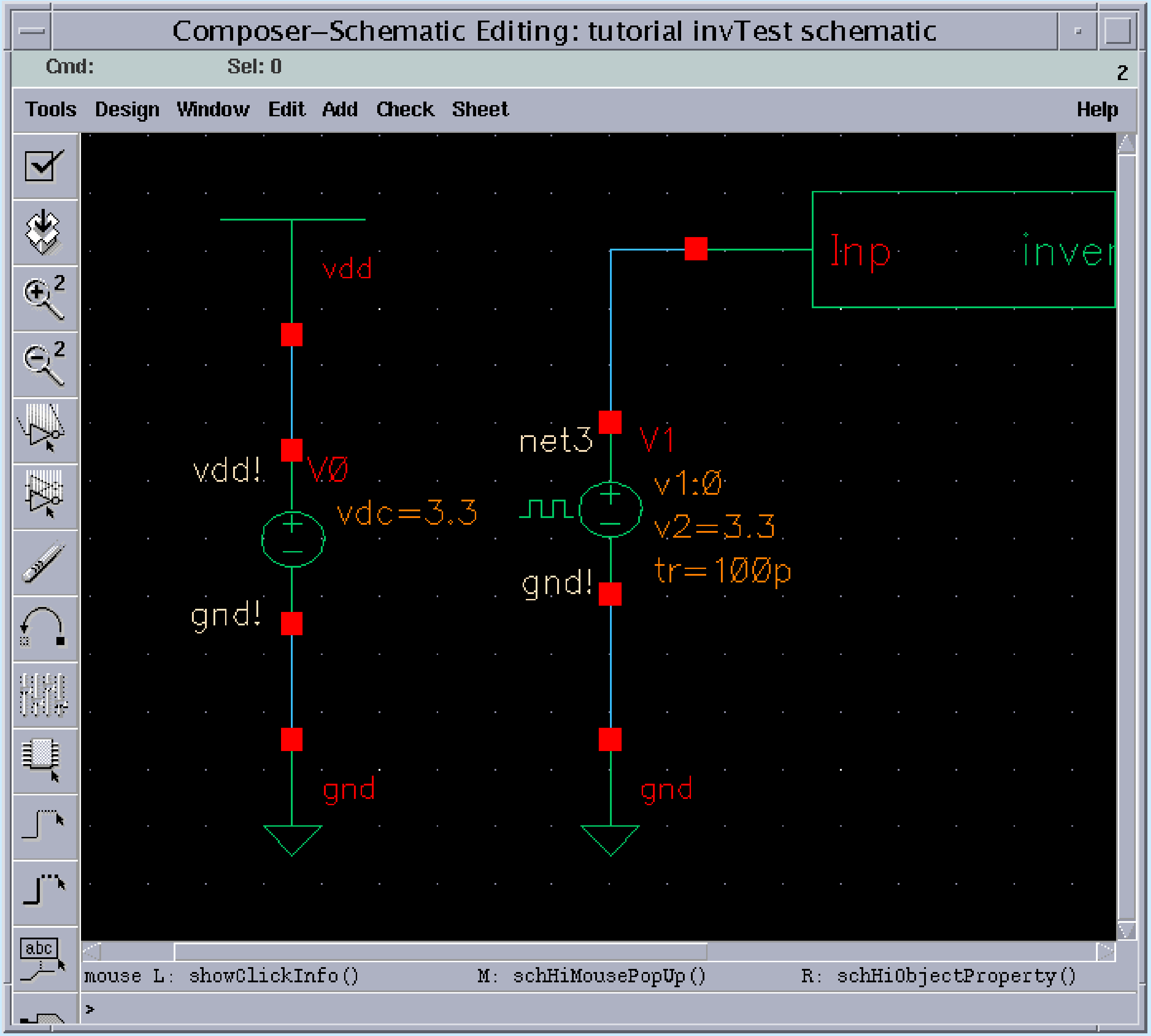
# Simulation - Define the voltage source Vpulse

The values for the=> pulse generator parameters which are used to define the input waveform.

Change them using the method as previously.

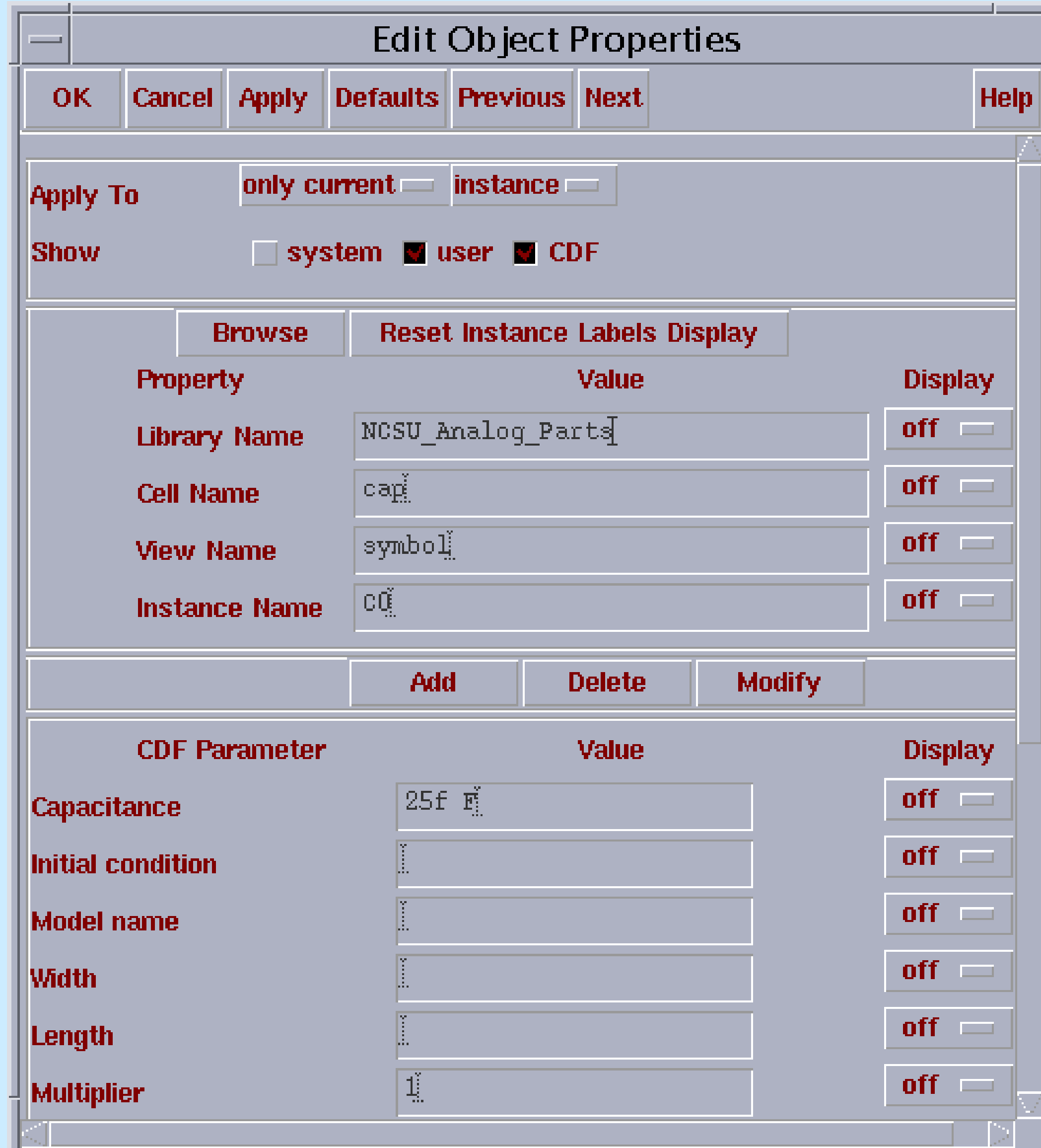


# Simulation - Define the voltage source - results



# Simulation - Determine the output load

Edit the properties of the capacitor which is the output load of the inverter.




Change capacitance => from the default value

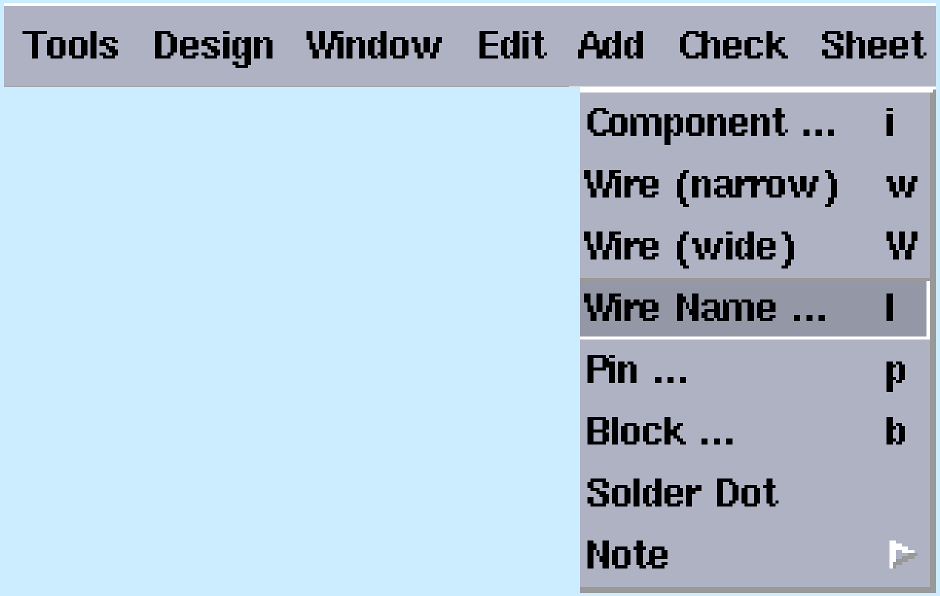
(1 pF) to 25 fF

# Simulation - Adding labels

- labeling a node = adding names to the wires.
- it allow to observe important nodes (or wires) during simulations.
- adding pins (during drawing the schematic) ≠ labeling a node

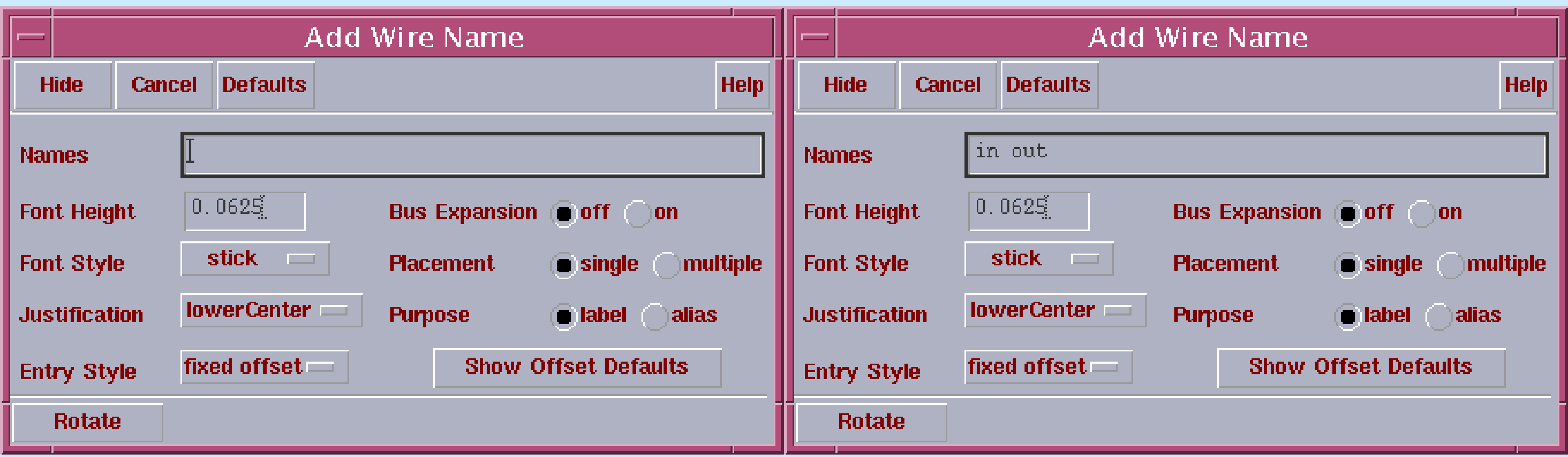
*How can I do it ?*

First, select *Wire Name* in the *Add* command list. =>



# Simulation - Adding labels

- type all the label names one after the other in the *Names* field
- there isn't any information related to the direction of the nodes - only the pins are defined with a direction.



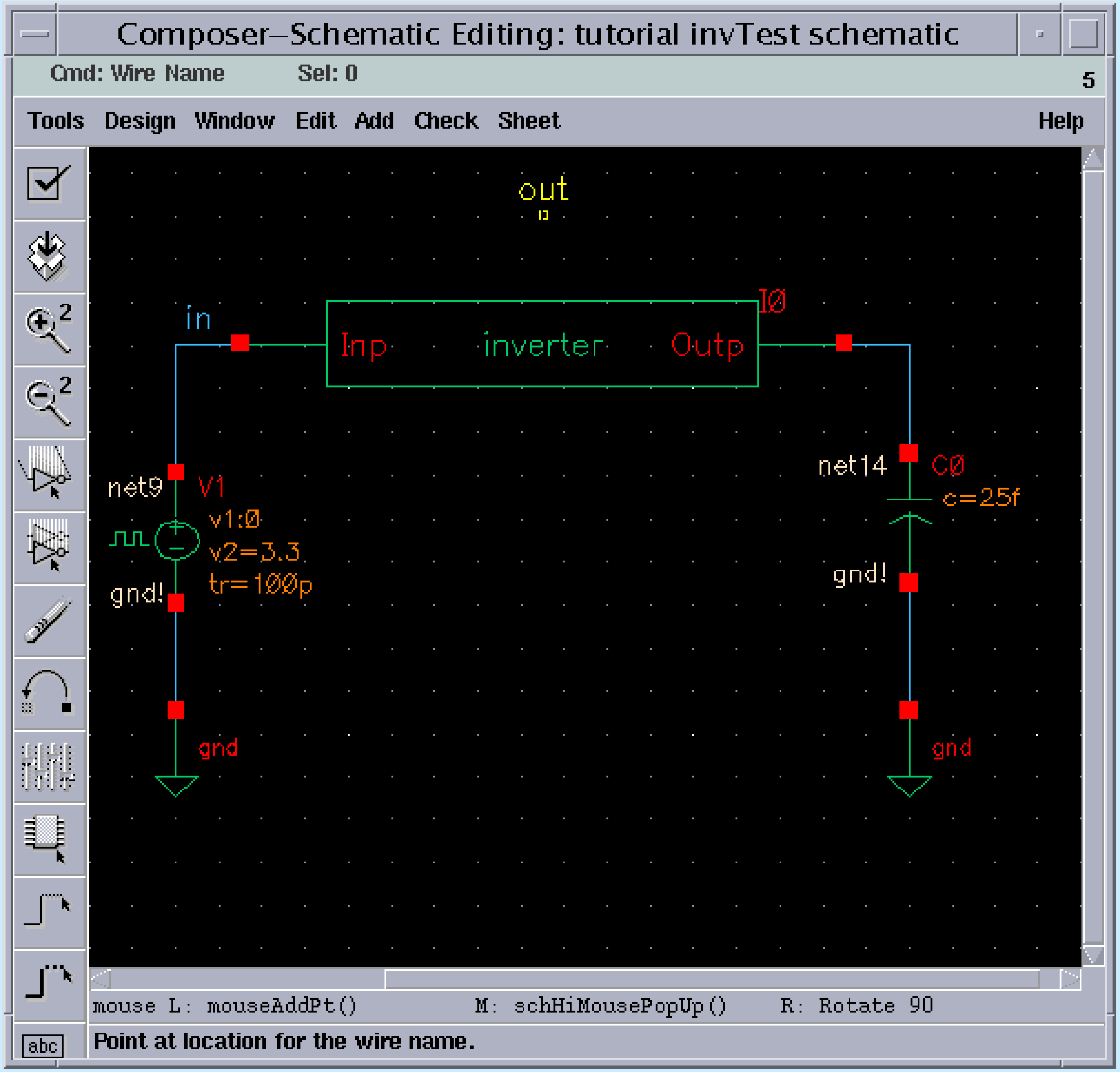

We will label the two wires as "in" and "out".

# Simulation – Adding labels

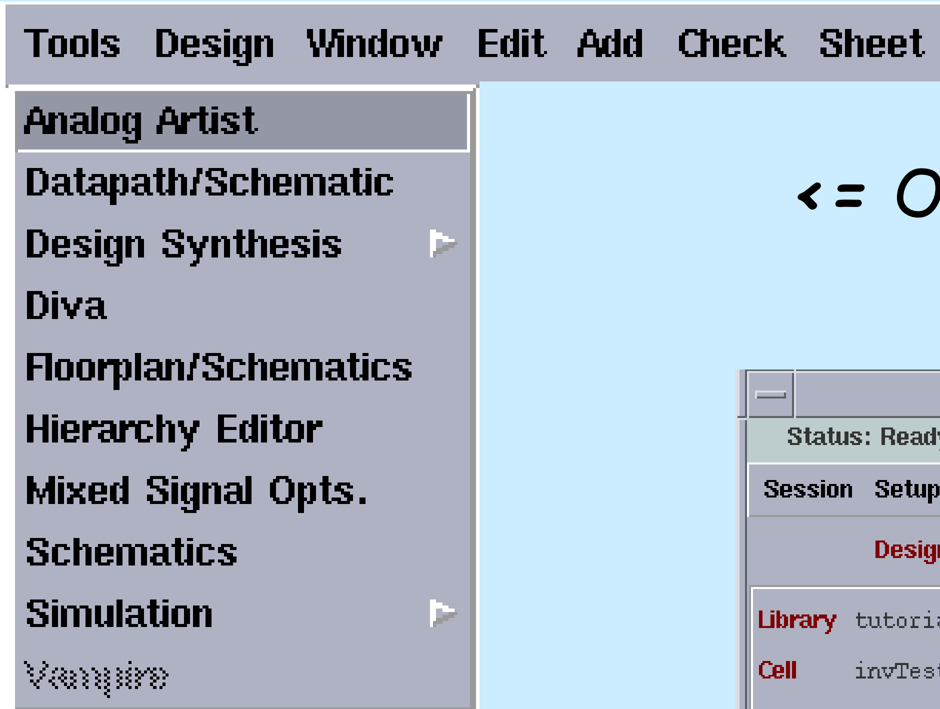
- After all the labels are typed, move the mouse cursor on the schematic
- You will see the first label floating with the mouse cursor. Click on the corresponding net to name the net with this label.
- As soon as you put the first label, the second label will appear on the mouse cursor.
- This procedure is repeated until you are finished putting all label names you entered in the *Add Label* window.

# Simulation - Adding labels results

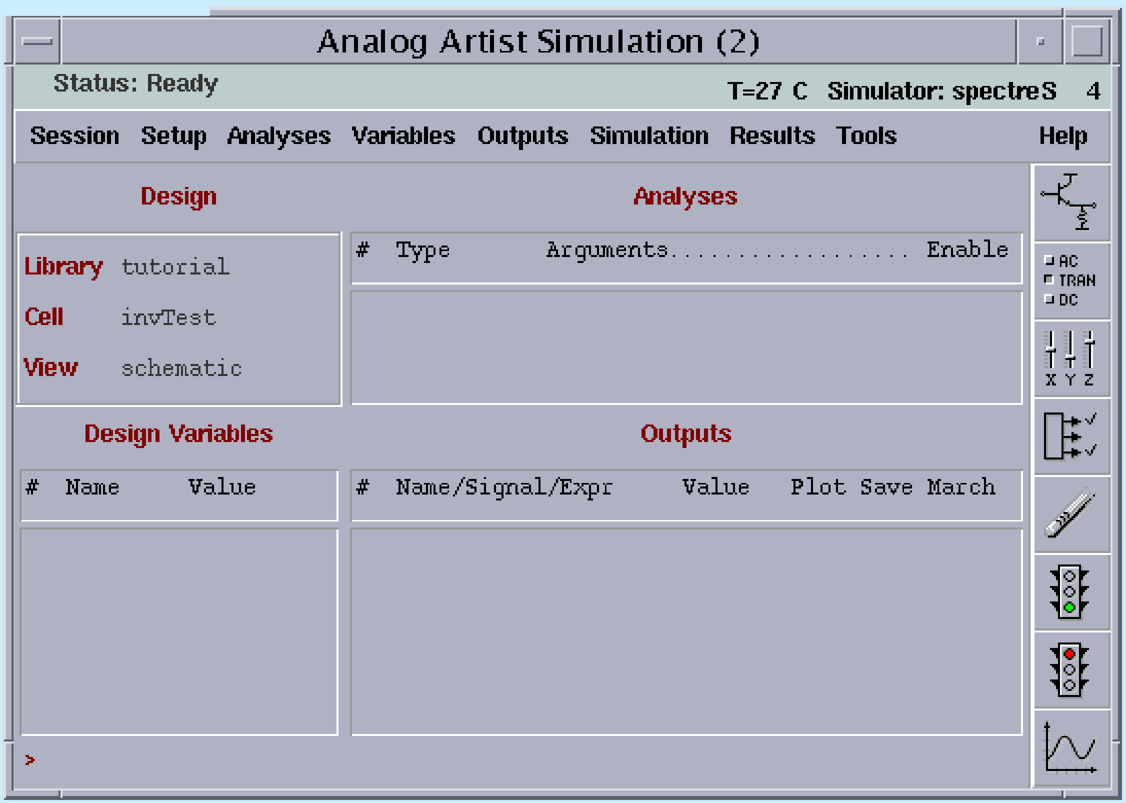
***Note:*** Save your design by using *Check and Save* in the *Design* command list. Be sure that the CIW doesn't report any errors or any warnings.



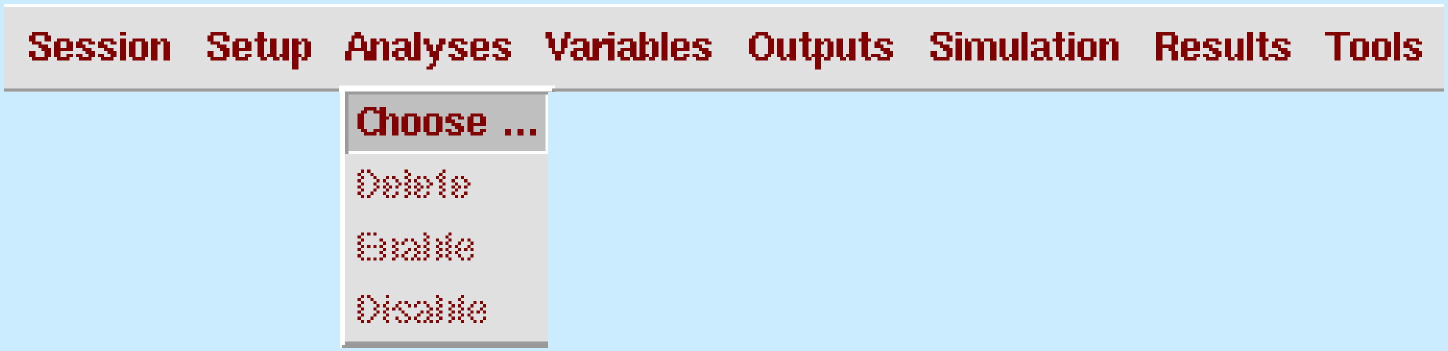
# Simulation - Open the simulator window




<= Open the *Analog Artist* window.



# Simulation - Edit the Simulation Parameters



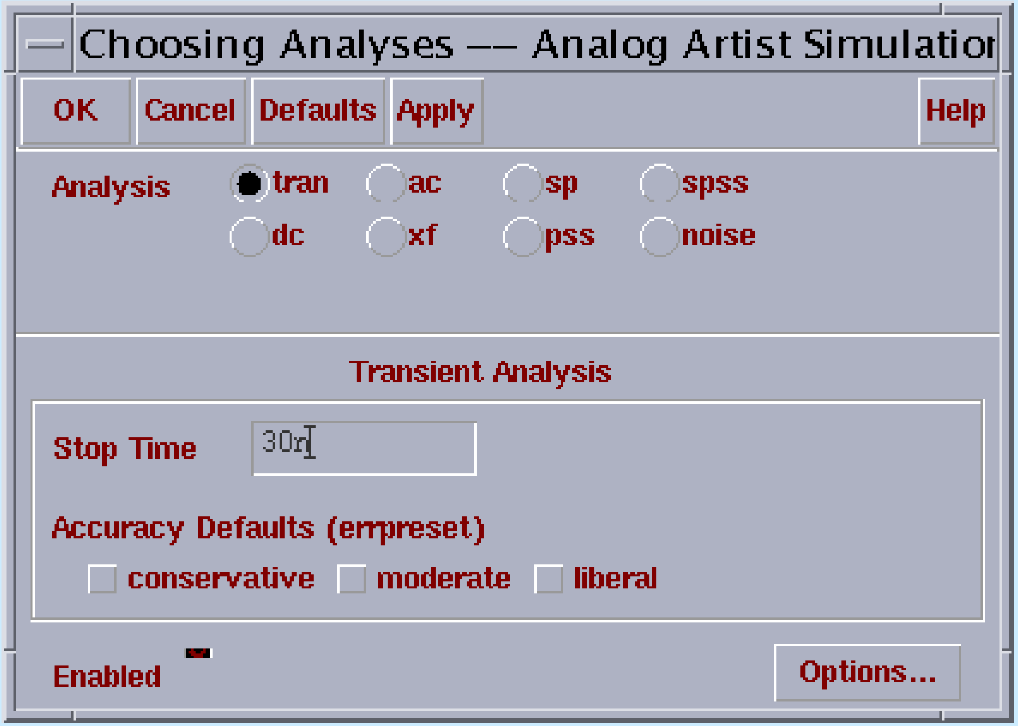

- there are many available analysis options you can choose.

- each of these options provides a specific sub-region within the *Choosing Analysis* window.

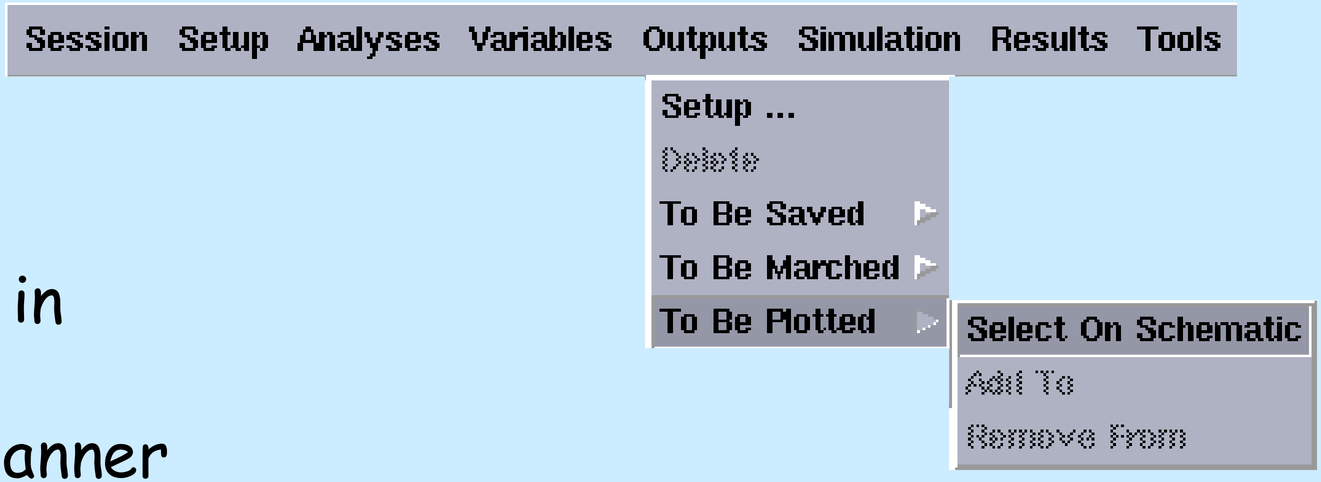
We want to obtain the delay information for the inverter, we choose the transient simulation type, so that the output can be traced in time domain.

# Simulation - Edit the Simulation Parameters

In the *Transient Analysis* region, type a value in the *Stop Time* field to determine how long the simulation will take place.



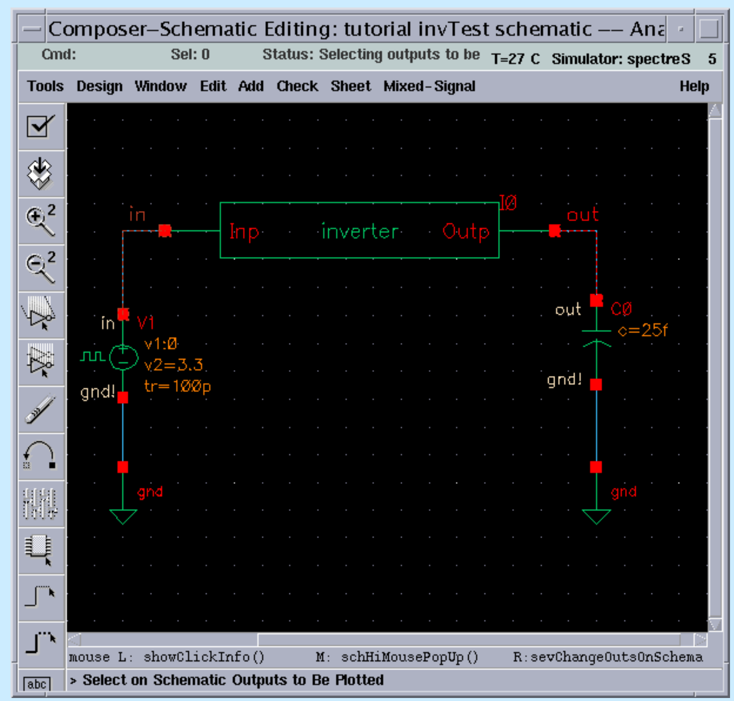
# Simulation - Run the Simulation



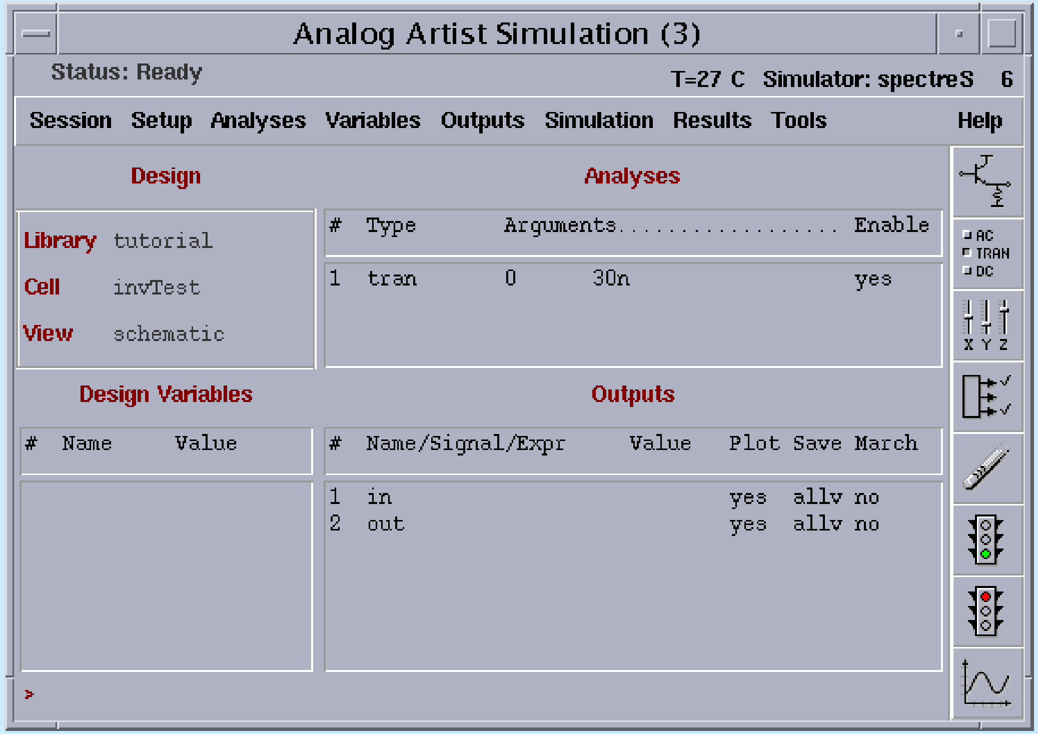

- click on *Outputs* in the *Analog Artist Simulation* menu banner

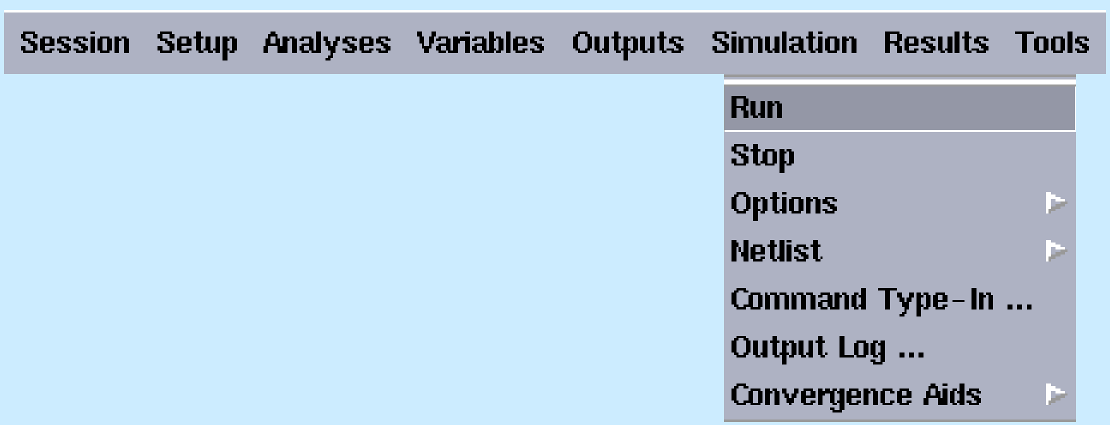
- select *To Be Plotted* and then *Select on Schematic.*

- when the schematic window becomes automatically active, **select the nodes to be observed**



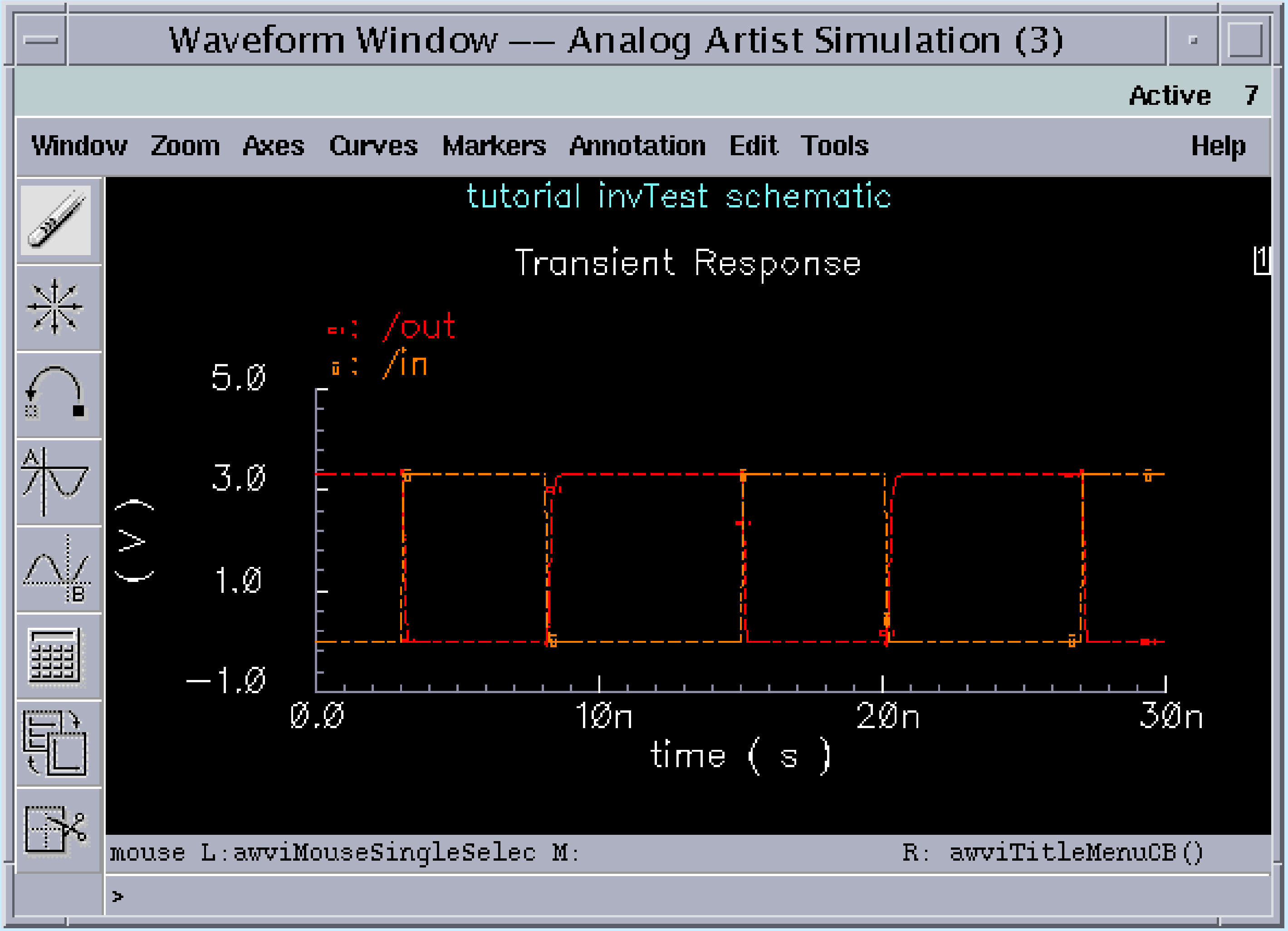
# Simulation - Run the Simulation




<= **Start the simulation** by clicking *Simulation* and then selecting *Run.*

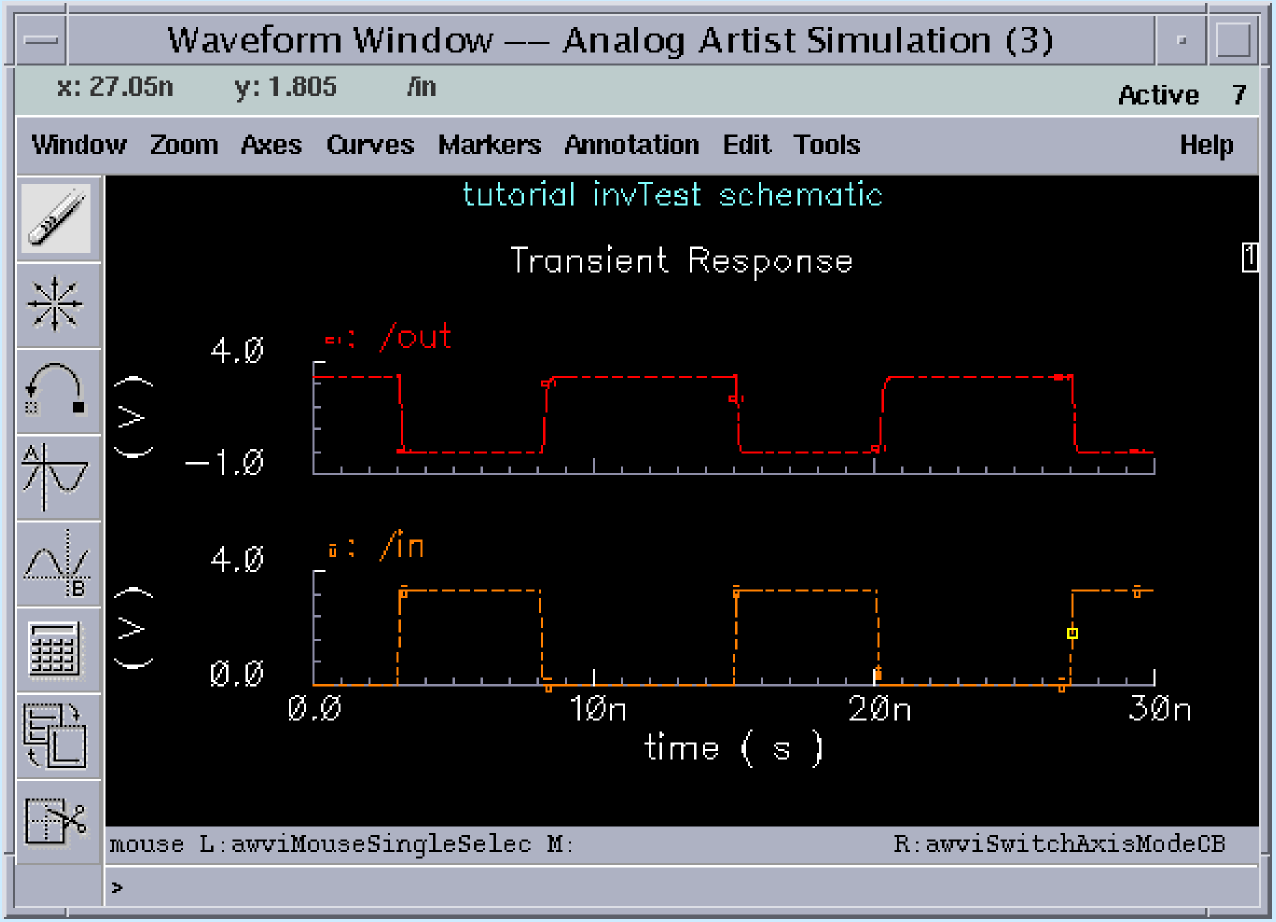
# Simulation - Run the Simulation




The waveform window appears after the simulation is completed.

# Simulation - Run the Simulation

To separate the waveforms, from the menu *Axes* select option *To Strip*..



# Simulation - Re-run the Simulation

*If you are not satisfied with the simulation results, there are two different aspects that can be modified :*

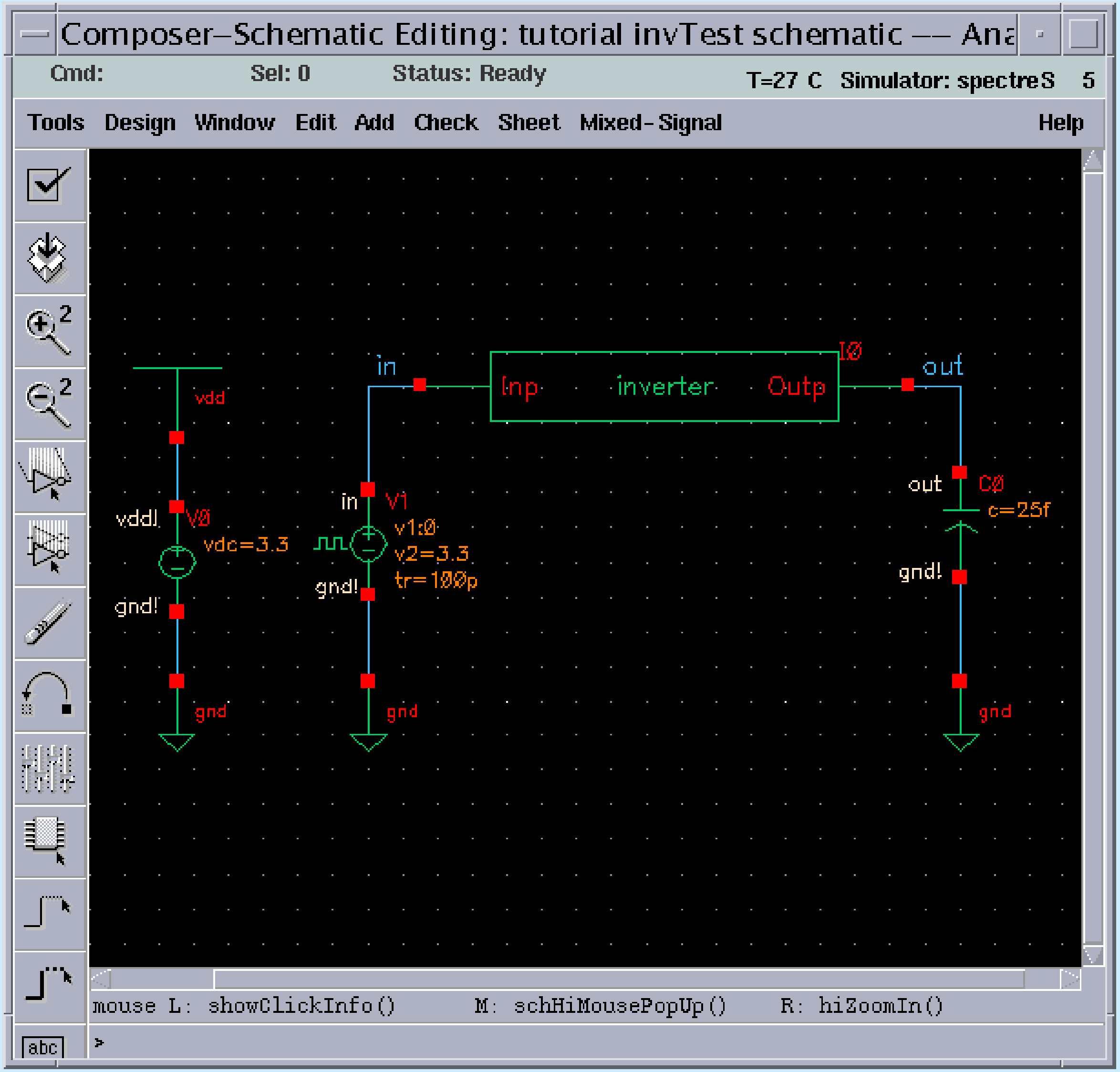
- **The simulation environment is not satisfactory.**
  This means that the setup to simulate your design should be modified.Make sure that the power supply voltages are connected properly.

- **You have to modify your circuit design.**
  Usually, you will need to change the W/L ratios of the transistors to meet your design specifications.

# Simulation - Re-run the Simulation

**How to re-run** the simulation after editing ?

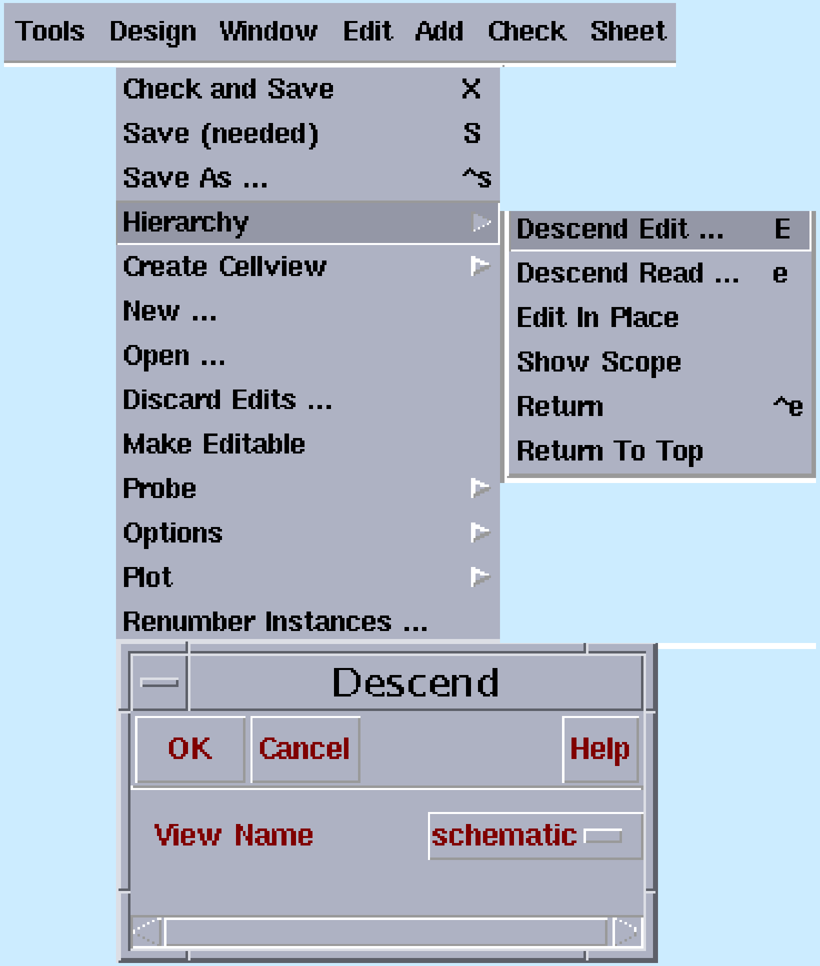
Go back to the => schematic window and select the symbol of your design.



# Simulation - Re-run the Simulation

Click on *Design* in the => menu banner, select *Hierarchy* and then *Descend Edit*.

Click on OK in the *Descend* window which asks the designer which view of the design is to be edited. =>



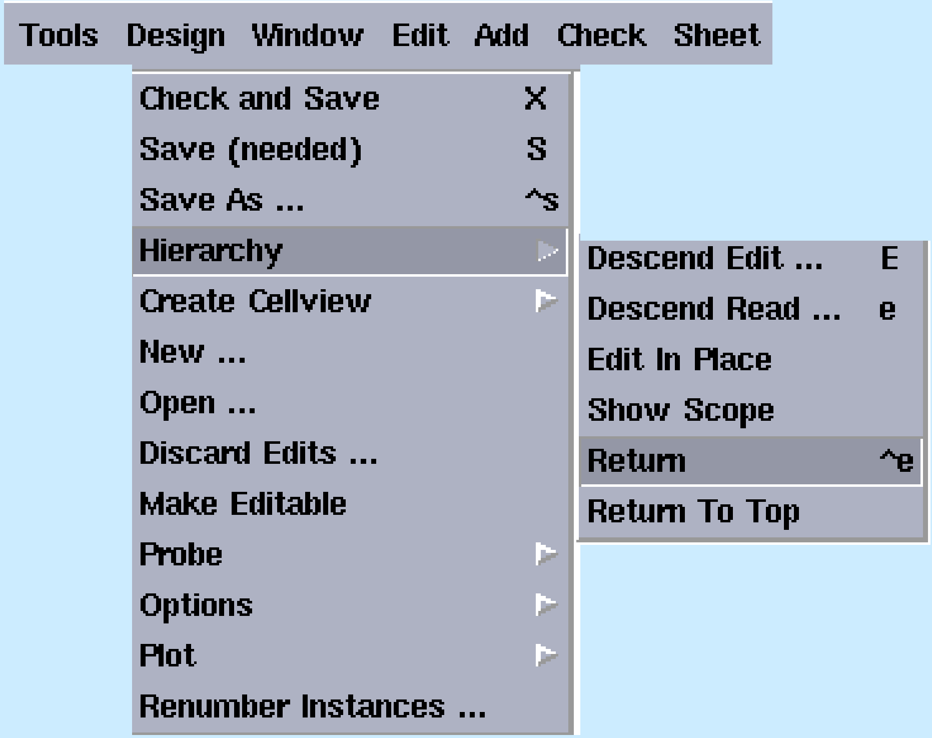

**The existing schematic window now displays the schematic view for the inverter, by going one level down through the design hierarchy.**

# Simulation - Re-run the Simulation

• Make the appropriate changes in the editable schematic of the design. *To change the existing W/L ratio for a specific transistor, you have to edit its object properties.*

• Check and save your new schematic.

• Click on Design in the menu banner, select Hierarchy and then Return.



# Simulation - Re-run the Simulation

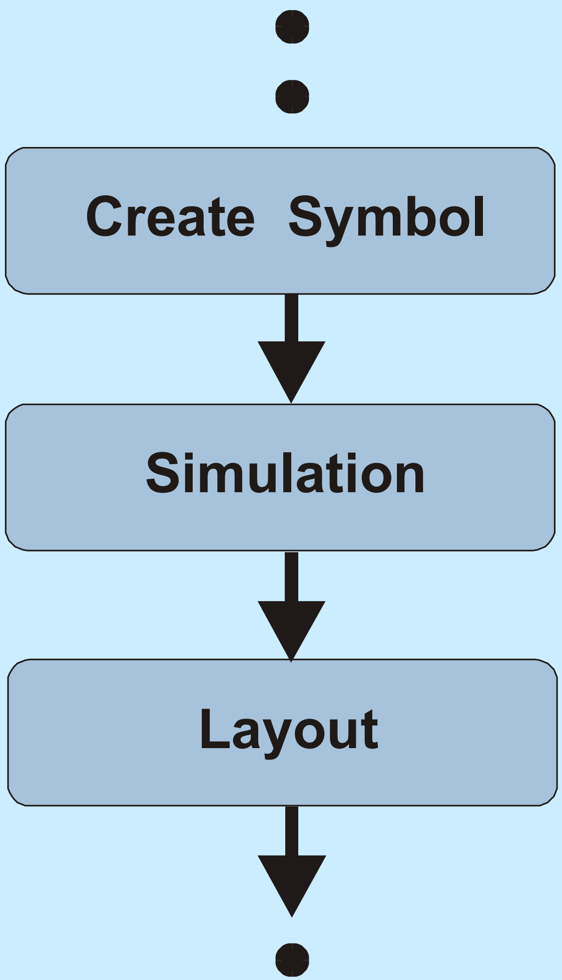
**Go to the *Analog Artist* window and run the simulation again.**

•As the simulation runs, you can switch to the waveform window, because the waveforms will be updated after the simulation is finished.

•You can iterate on your design as described in this section of the tutorial.

•When you want to end the simulation, quit the *Analog Artist* simulator. This will automatically close the *Waveform* window, too.

# Mask Layout




- The mask layout is one of the most important steps in the full-custom (bottom-up) design flow.
- It describes the detailed geometries and the relative positioning of each mask layer to be used in actual fabrication.
- Physical layout design is very tightly linked to overall circuit performance.
- The detailed mask layout of logic gates requires a very intensive and time-consuming design effort.

# CMOS Inverter Layout

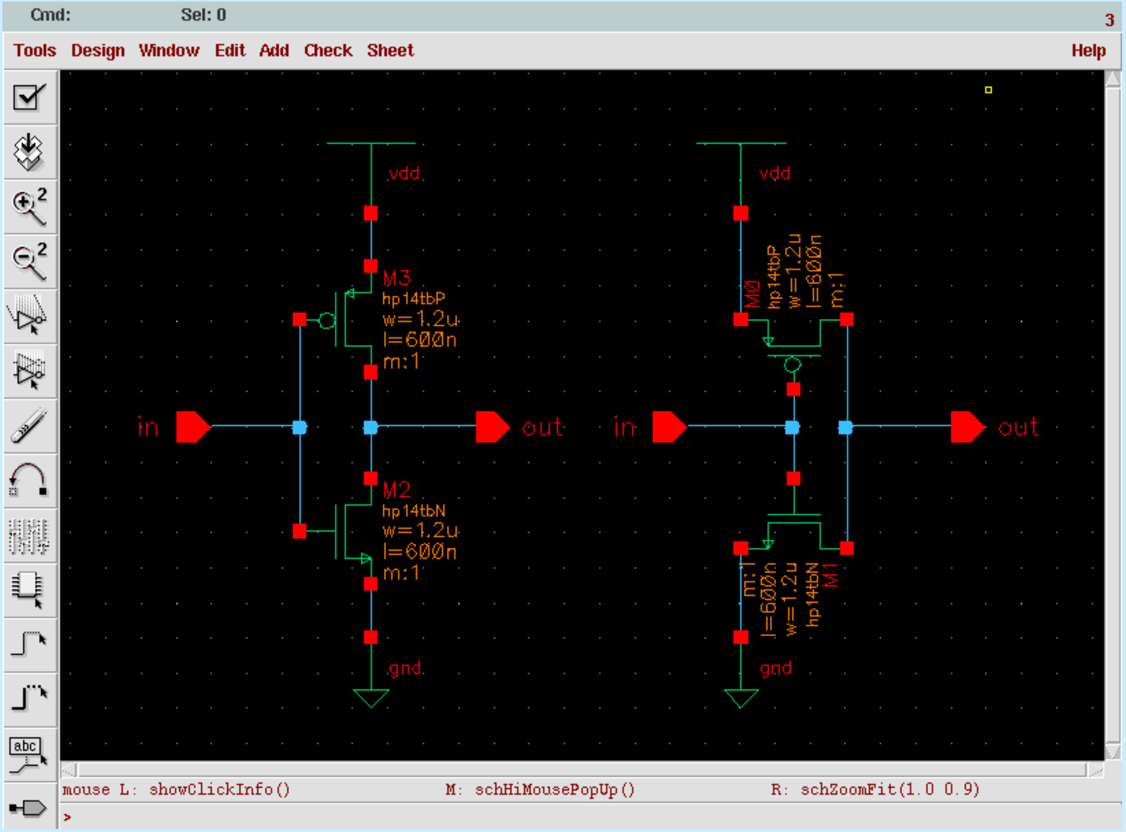
## Design Idea

To draw the mask layout of a circuit, two main items are necessary at the beginning:

1. A circuit schematic
2. A signal flow diagram

# Circuit Schematic

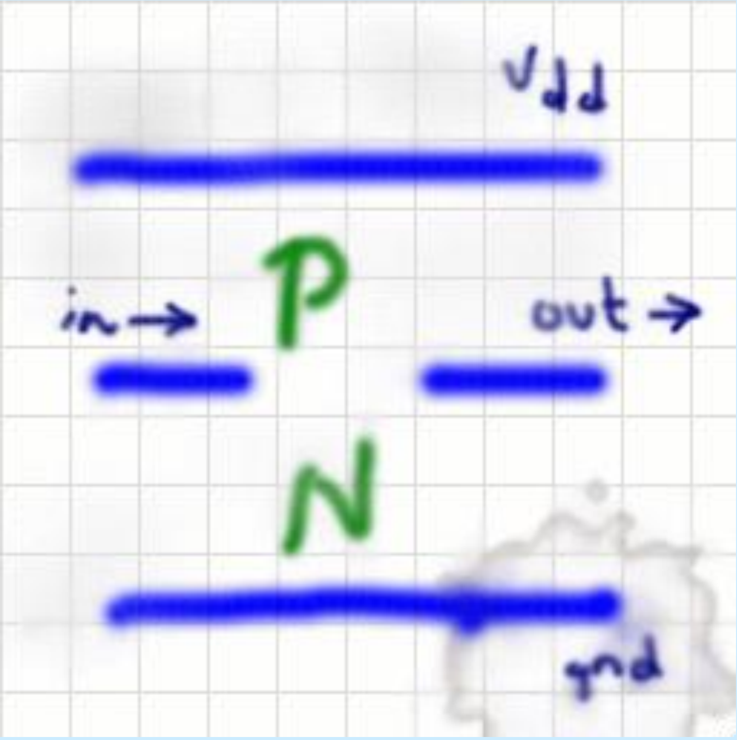
The layout is drawn according to the schematic (and not the other way around).




While both schematics are identical, the one on the right is drawn in a way to resemble the final layout.

# Signal Flow Diagram

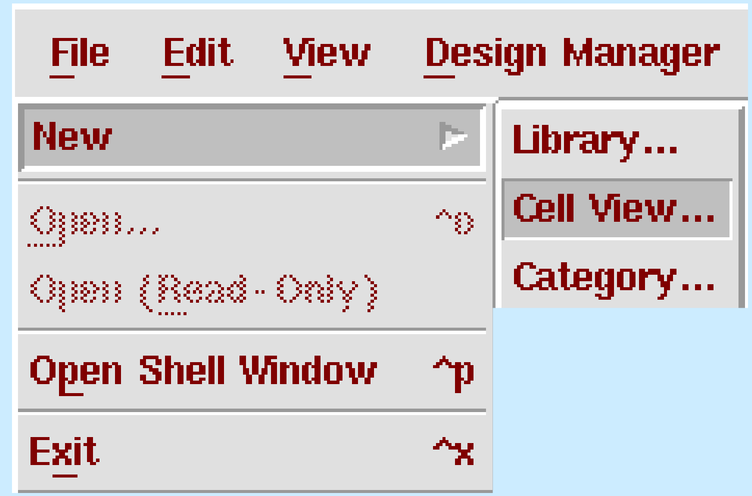
The most important factor determining the actual layout is the signal flow.



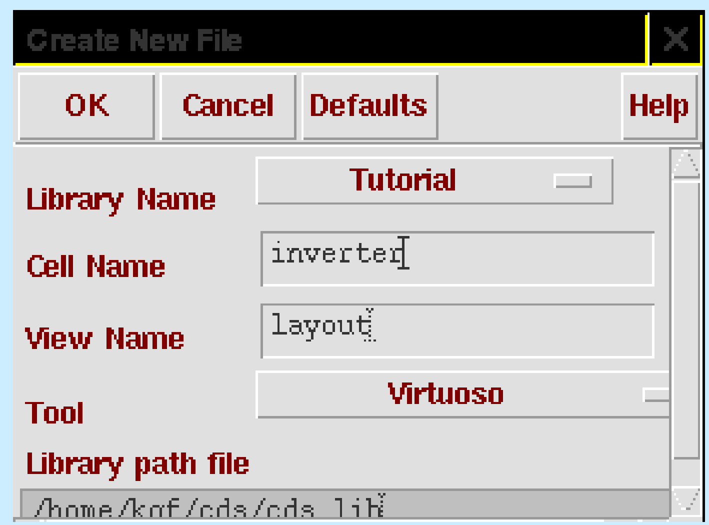

The signal flow diagram is just a concept that you can visualize for a particular circuit.

# Create Layout Cellview

1. From the Library Manager : File --> New --> Cellview



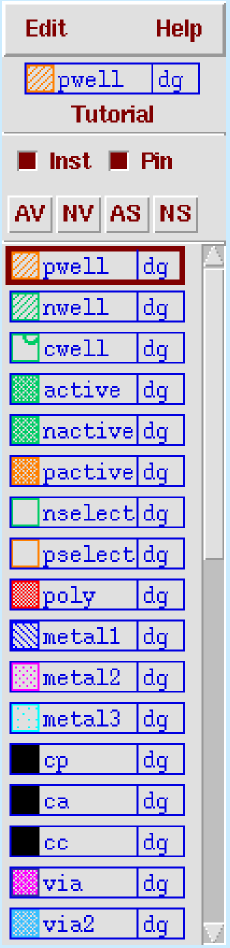

**2.** Enter cellname and choose *layout cellview*

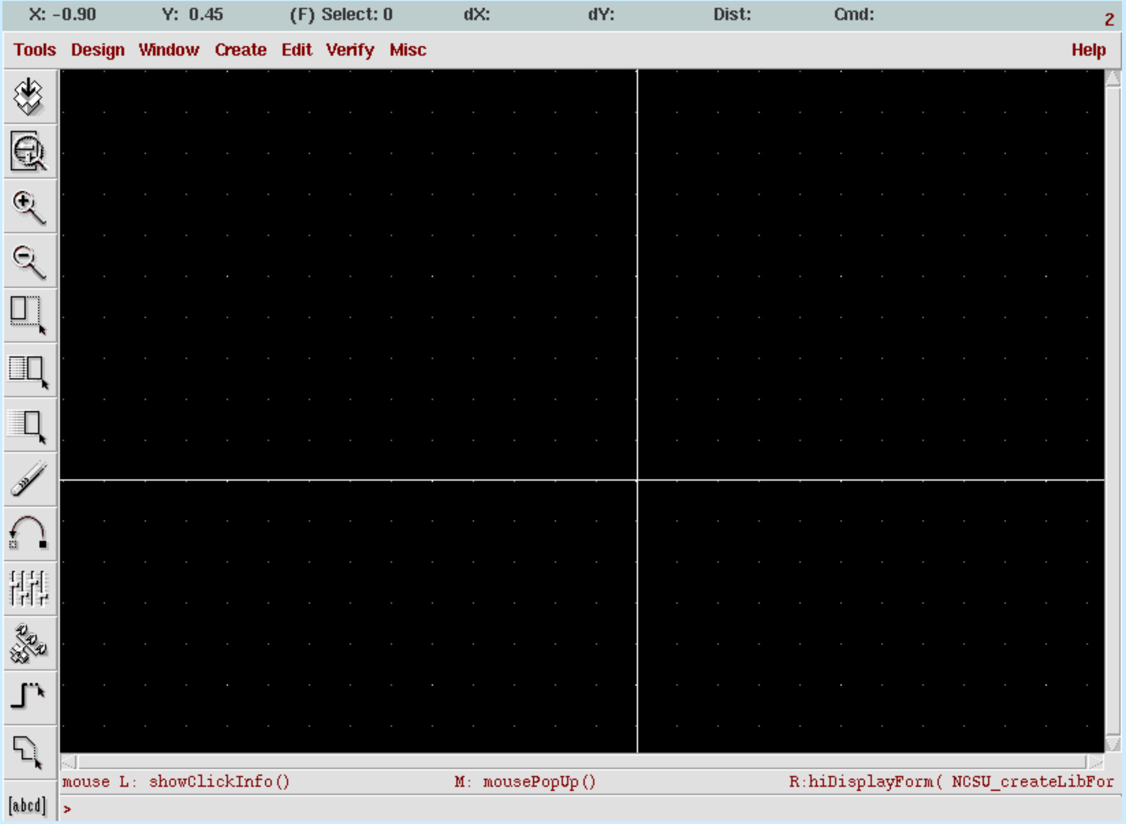


# Virtuoso and LSW

Two design windows will pop-up after you have entered the design name.

LSW

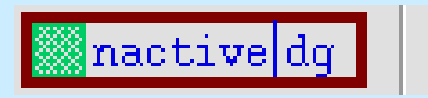


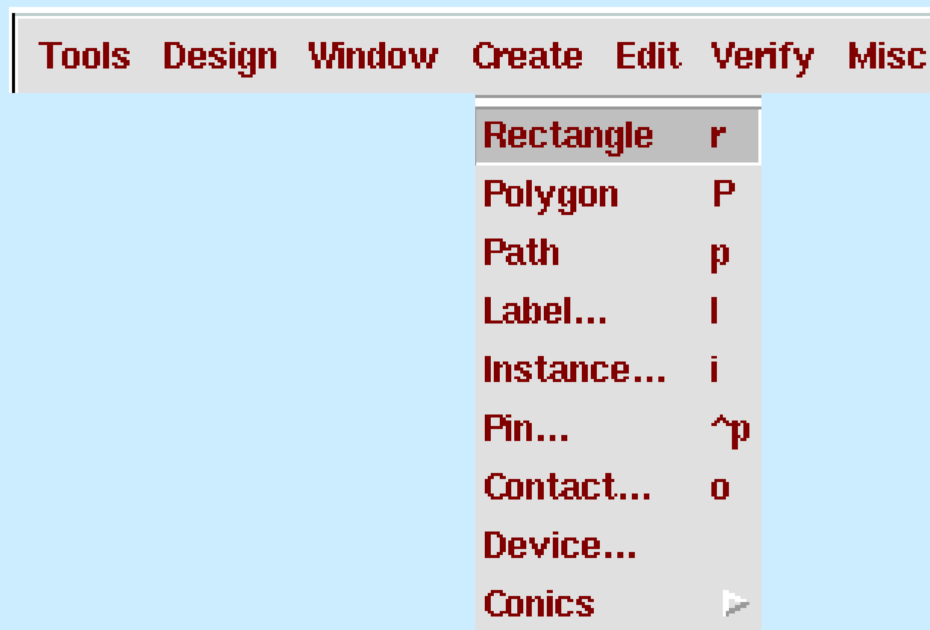

Virtuoso

# Drawing the N-Diffusion (Active)

1. Select *nactive* layer from the LSW

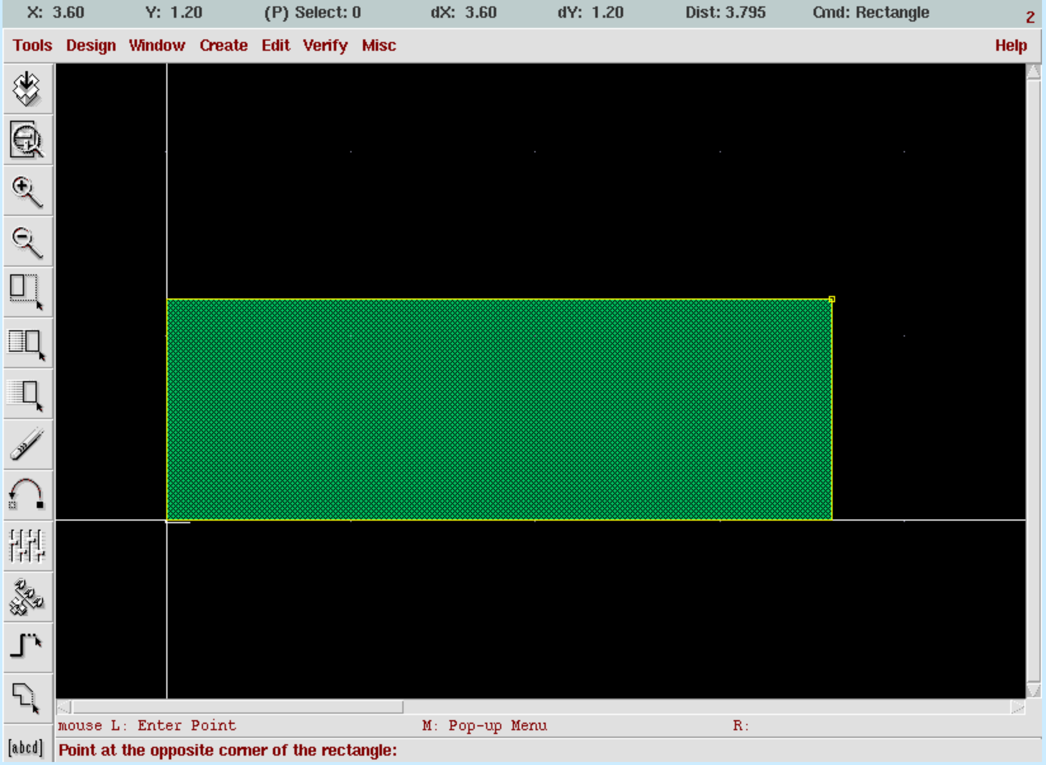



2. From the *Create* menu in Virtuoso select Rectangle
( Create --> Rectangle )



# Drawing the N-Diffusion (Active)

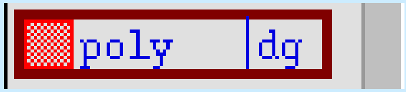
3. Draw the box
Select the first corner of rectangle in the layout window, click once, and then move the mouse cursor to the opposite corner.




- A grid of half a lambda is used

# The Gate Poly

1. Select *poly* layer from the LSW



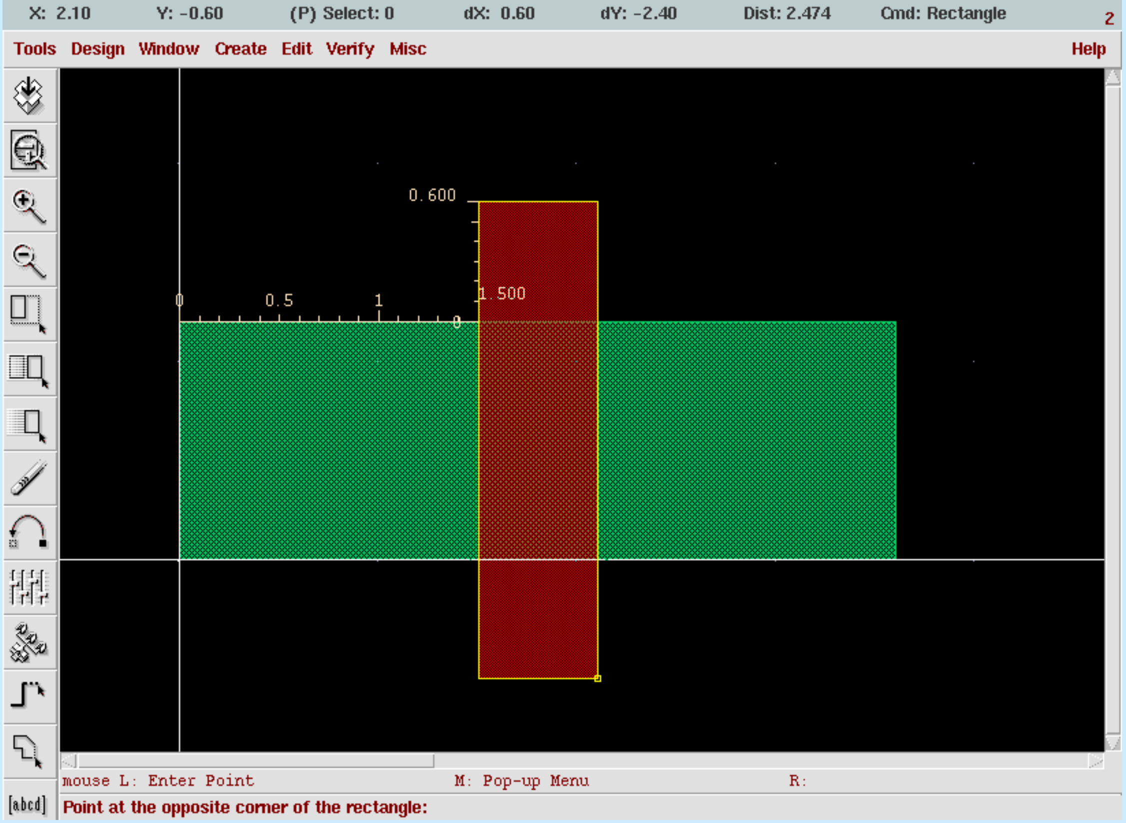

2. From the menu *Misc* choose *Ruler*
( Misc --> Ruler )


*! The ruler is a very handy function.*

# The Gate Poly

3. Draw poly rectangle

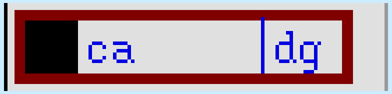
Design rules tell us that poly must extend at least by 0.6$\mu$ (2 Lambda) from edge of diffusion



# Making Active Contacts

Contacts will provide access to the drain and source regions of the NMOS transistor.

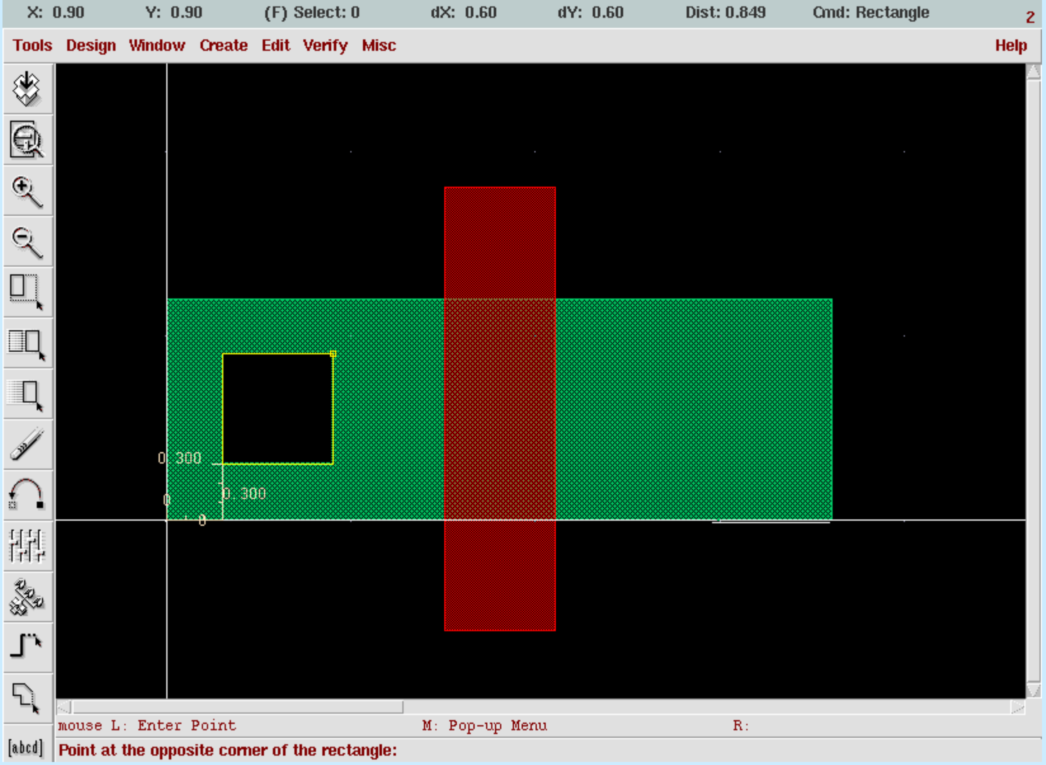
1. Select the *ca* (Active Contact) layer from the LSW.



# Making Active Contacts

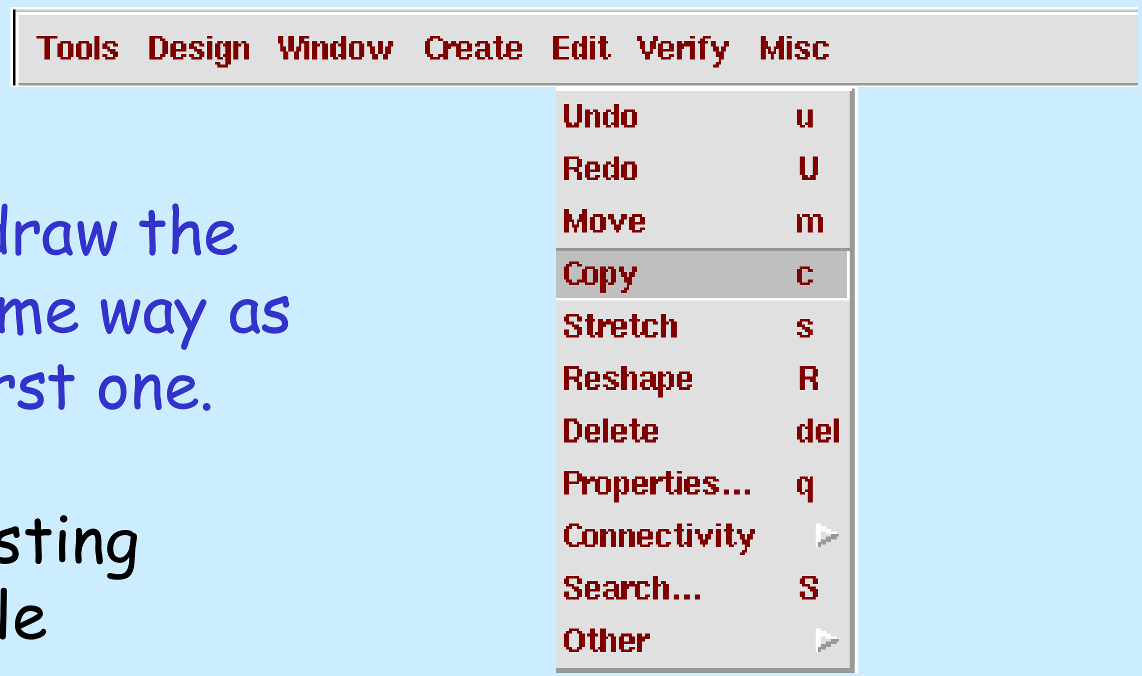
2. Use the ruler to pinpoint a location 0.30u from the edges of diffusion.

3. Create a square with a width and hight of 0.6u within the active area. ►



# Making Active Contacts

**4.** From the *Edit* menu choose *Copy*
( Edit --> Copy )

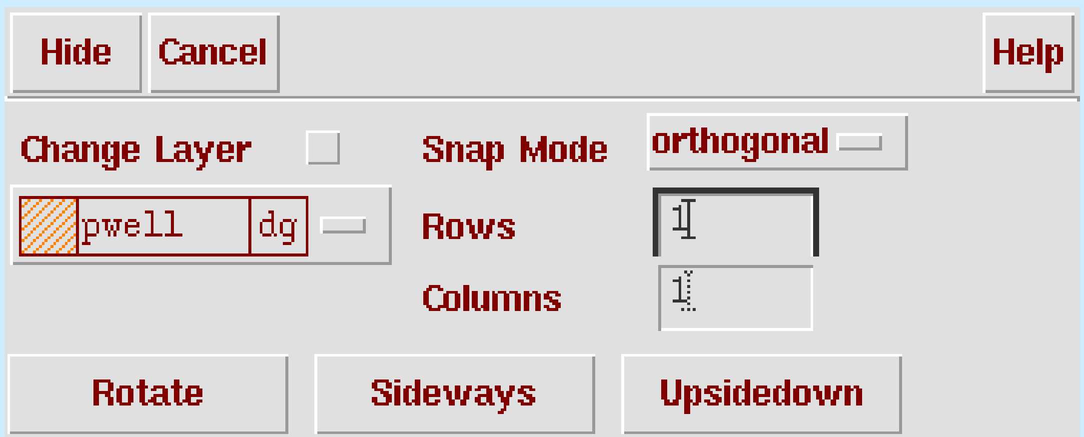



•You could choose to draw the second contact the same way as you have drawn the first one.

•However, copying existing features is also a viable alternative.

# Making Active Contacts

When the *Snap Mode* is in *orthogonal* setting the copied objects will move only along one axis.

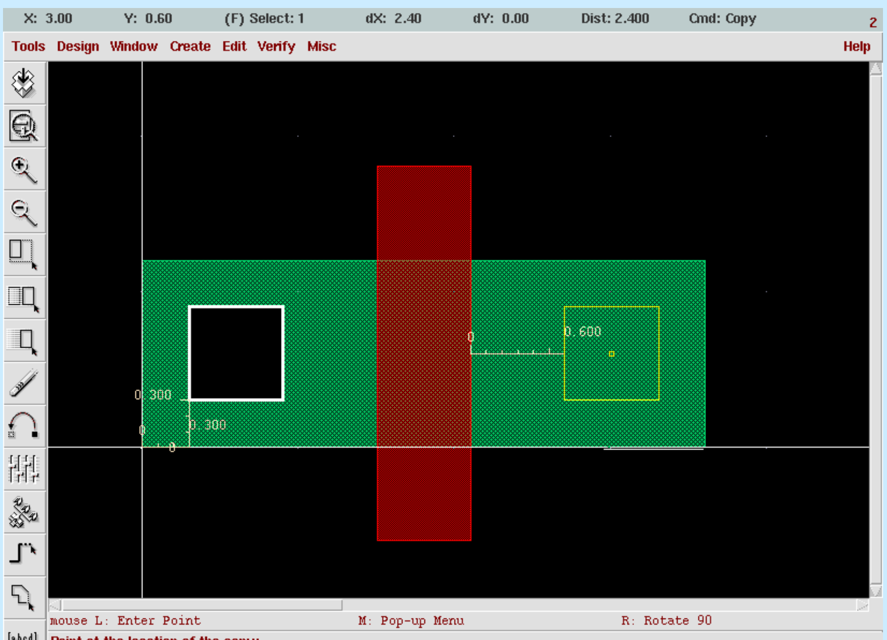


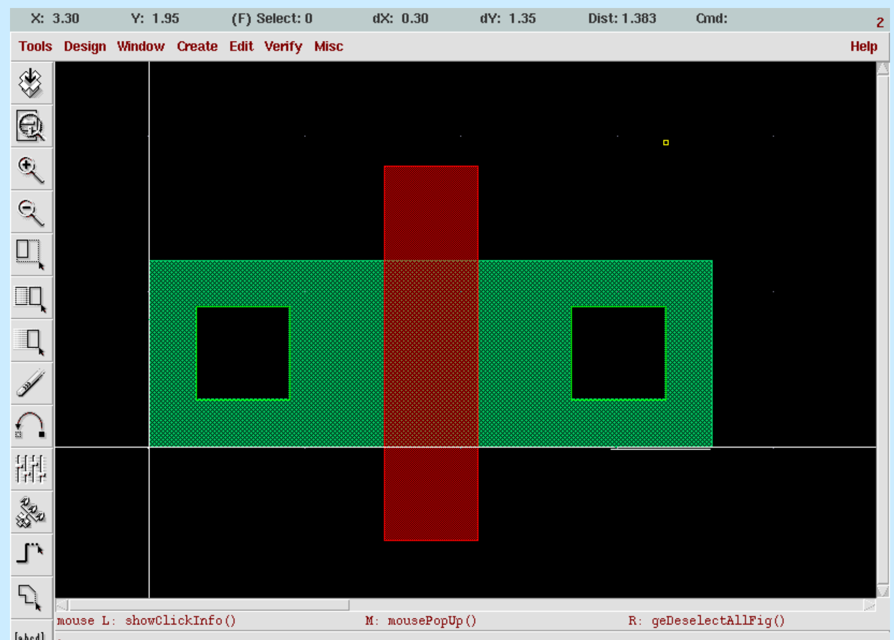

Copy dialog box.

# Making Active Contacts

5. Copy the contact

- Select the object (click in the contact- the outline of contact will attach to your cursor.
- Move the object and click at the final location.



# Covering Contacts with Metal-1

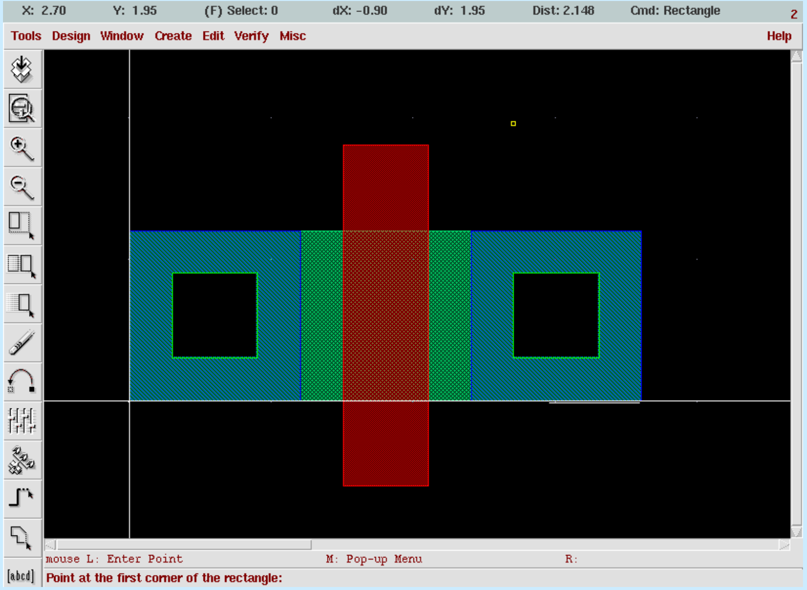
•Active contacts define holes in the oxide (connection terminals).

•The actual connection to the corresponding diffusion region is made by the Metal layer.

# Covering Contacts with Metal-1

1. Select layer *Metal-1* from the LSW


2. Draw two rectangles 1.2u wide to cover the contacts

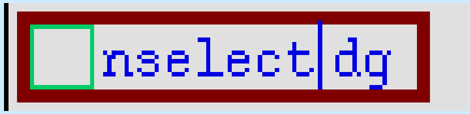



Note that Metal-1 has to extend over the contact in all directions by at least 0.3 u (1 lambda).

# The N-Select Layer

Each diffusion area of each transistor must be selected as being of n-type or p-type.

1. Select *nselect* layer from the LSW.

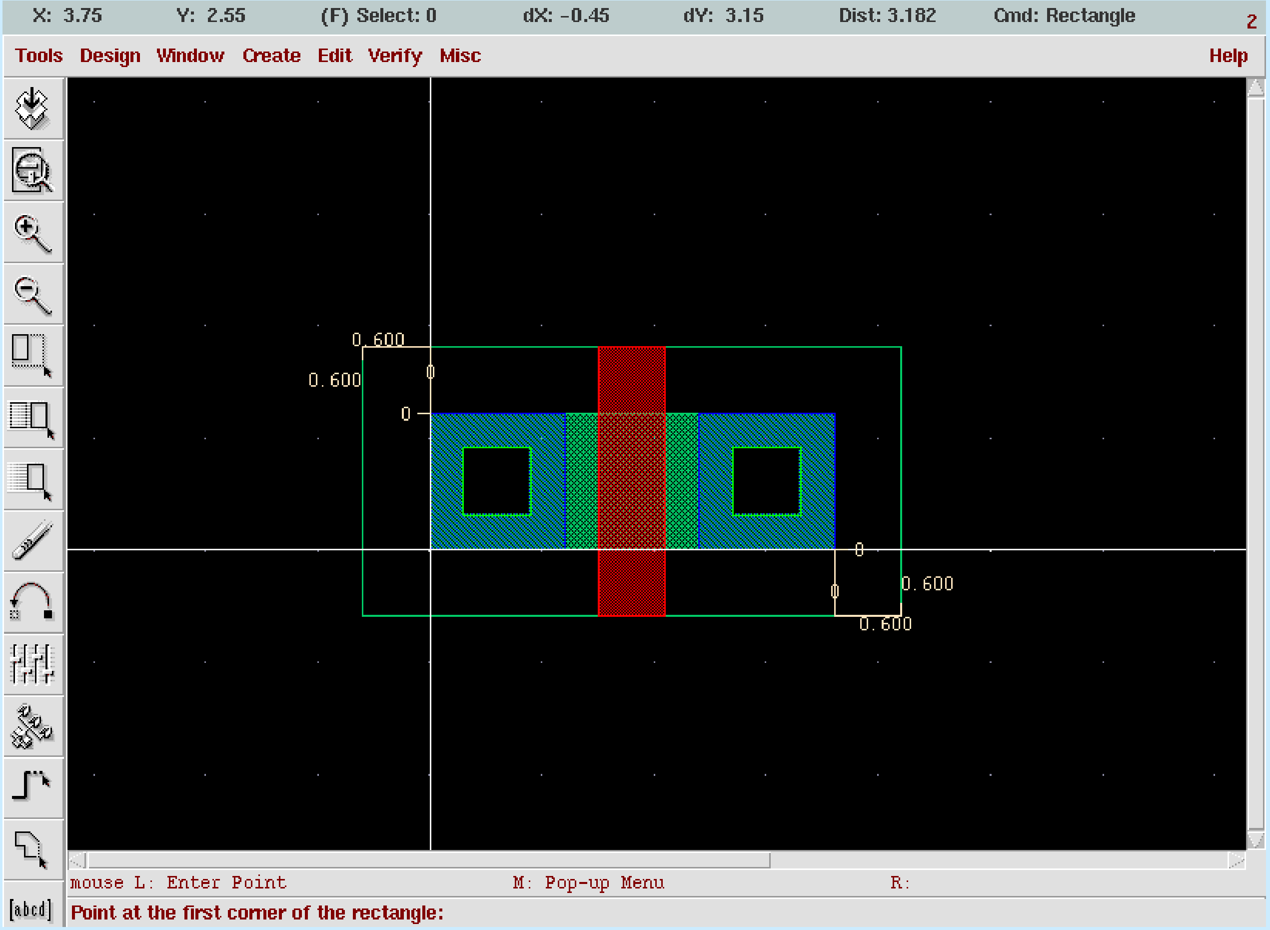


# The N-Select Layer

2. Draw a rectangle extending over the active area by 0.6u (2 lambda) in all directions.

This is it ! Our first transistor is finished, now let us make a few million more of the same

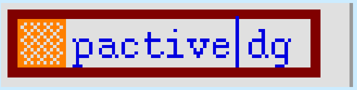
:-)



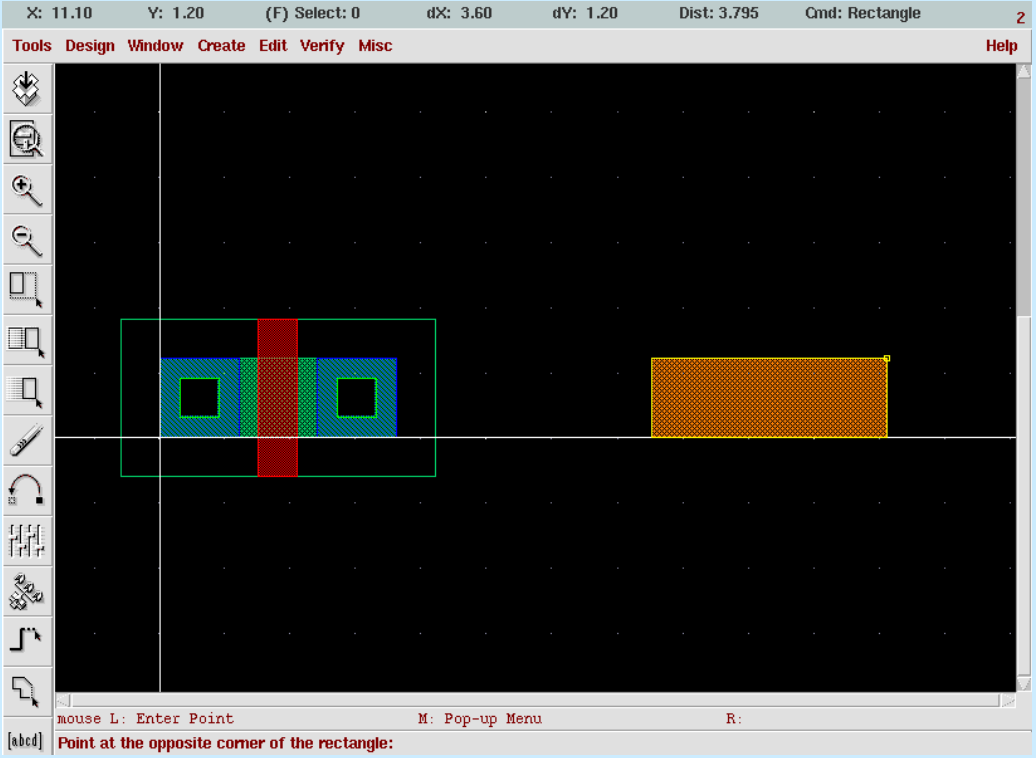
# Drawing the P-Diffusion (Active)

The basic steps invloved in drawing the PMOS are the same.

1. Select *pactive* layer from the LSW




2. Draw a rectangle ► 3.6μ by 1.2μ

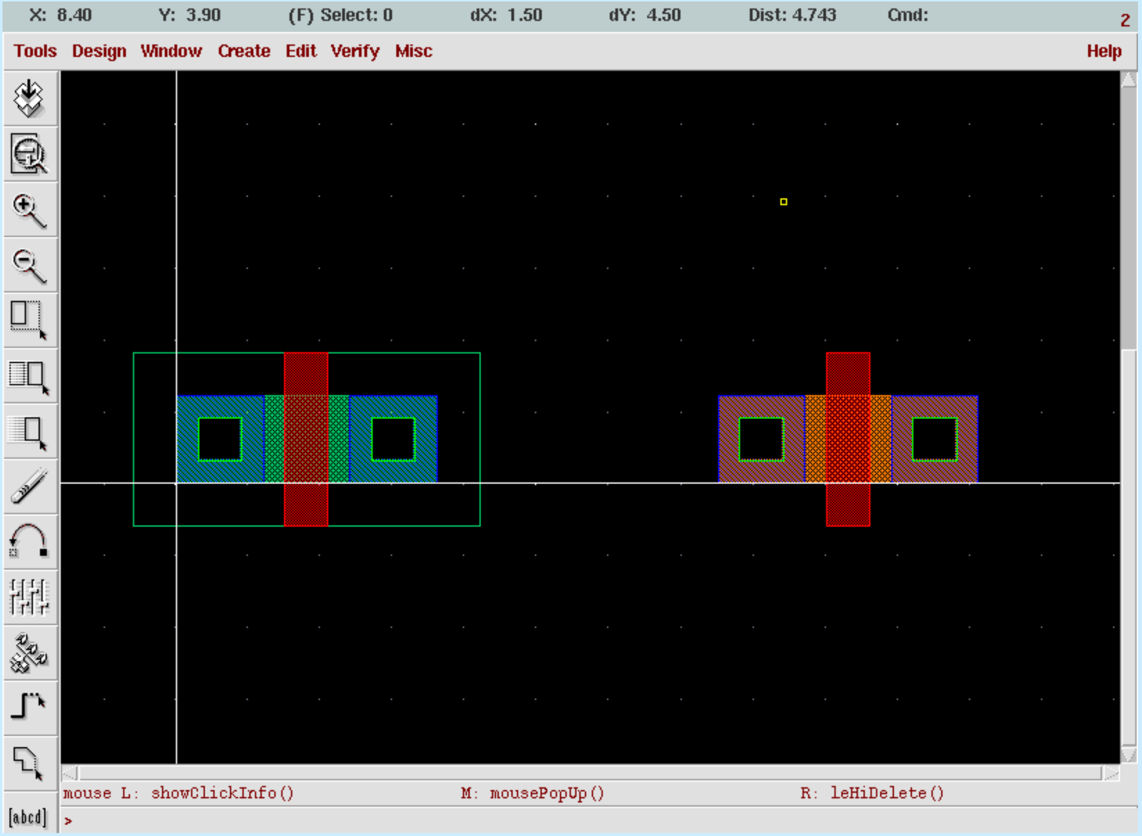



**Note that the PMOS transistor will also be sorrounded by the N-well region.**

# Transistor Features

These three steps are identical to the ones done for the NMOS.

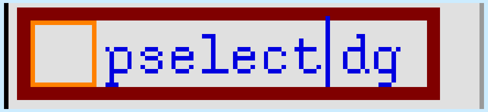
1. Draw the gate poly
2. Place the contacts
3. Cover contacts with Metal-1



# The P-Select Layer

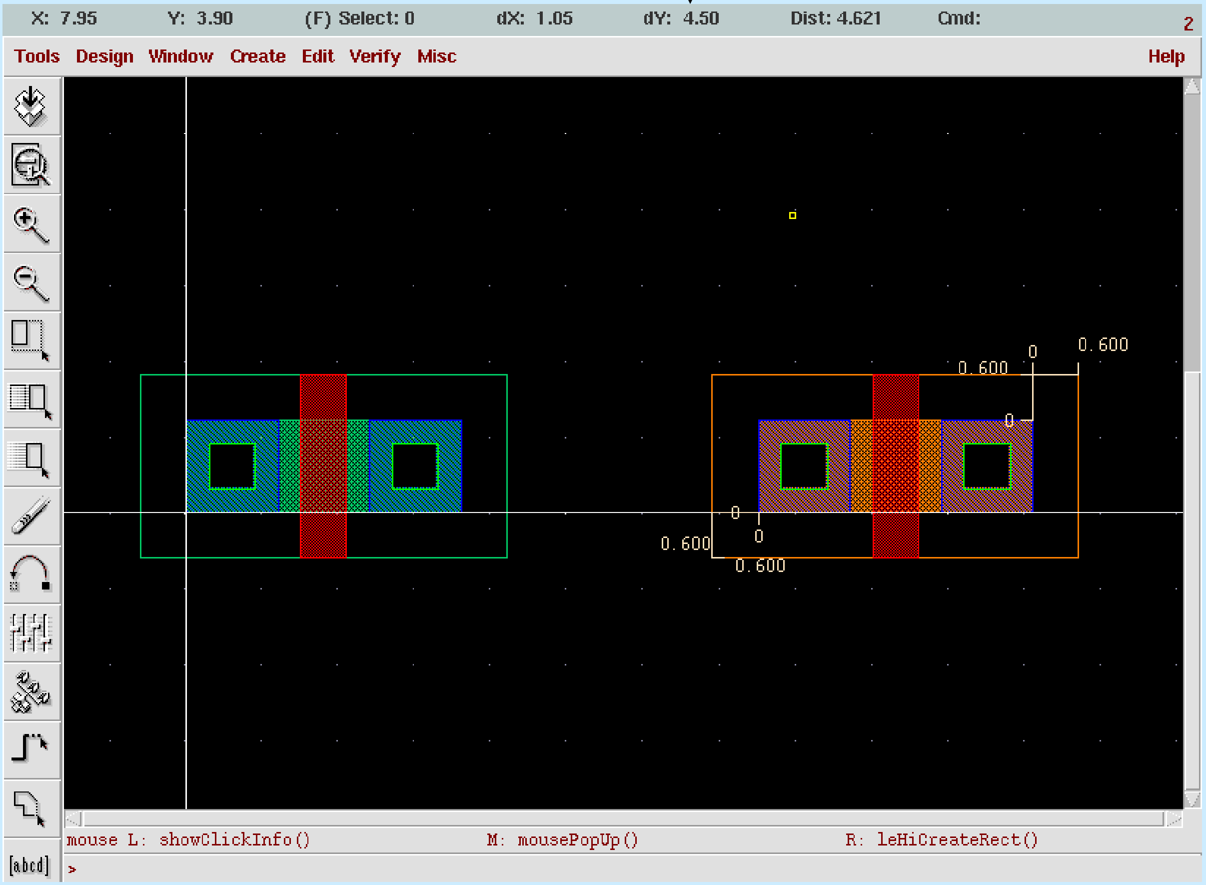
The p-type doping (implantation) window over the active area must be defined using the n-pelect layer.

1. Select *pselect* layer from the LSW



# The P-Select Layer

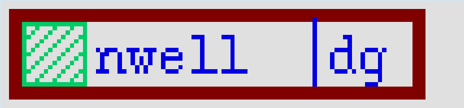
2. Draw a rectangle that extends over the active area by 0.6u (2 lambda) in all directions.



# Drawing the N-Well

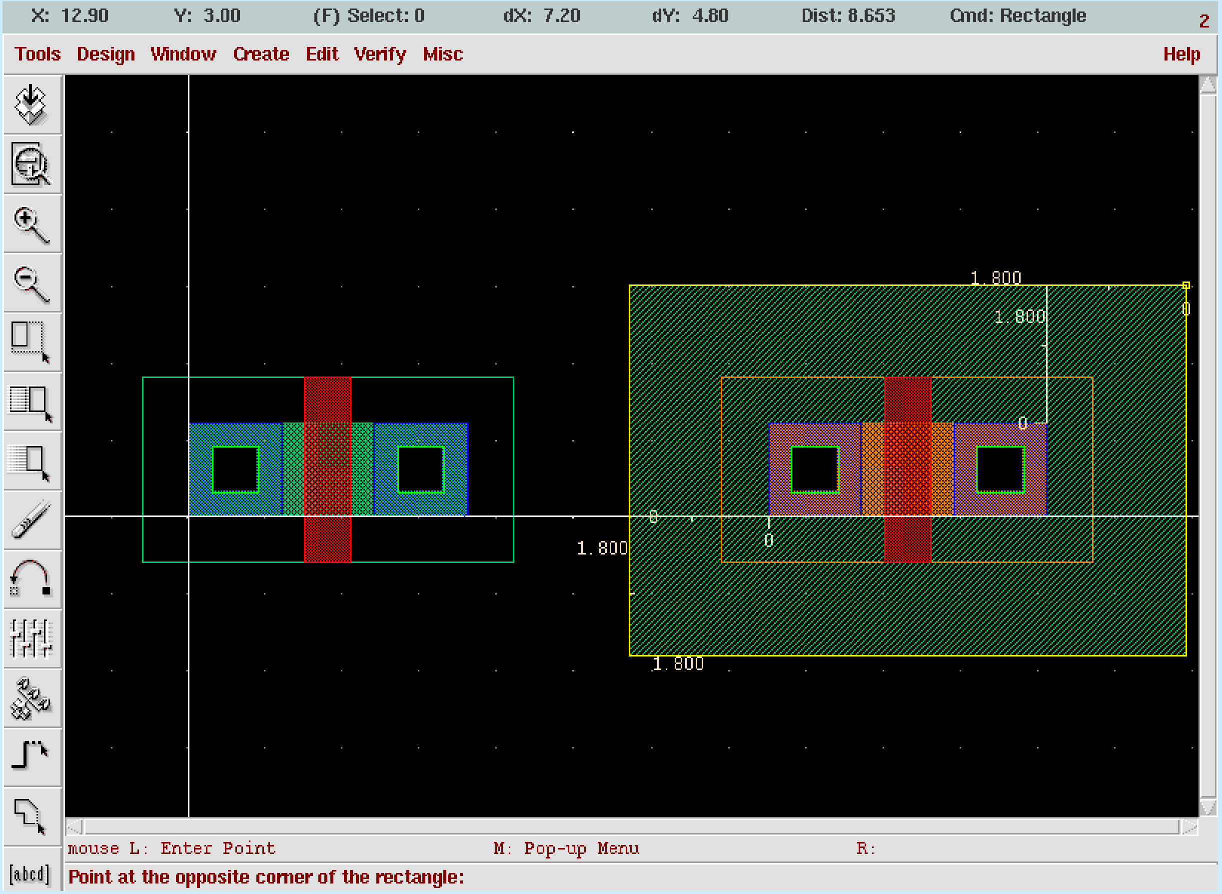
Note that the drawing sequence of different layers in a mask layout is completely arbitrary, it does not have to follow the actual fabrication sequence.

1. Select the *nwell* layer from the LSW



# Drawing the N-Well

2. Draw a large n-well rectangle extending over the P-Diffusion



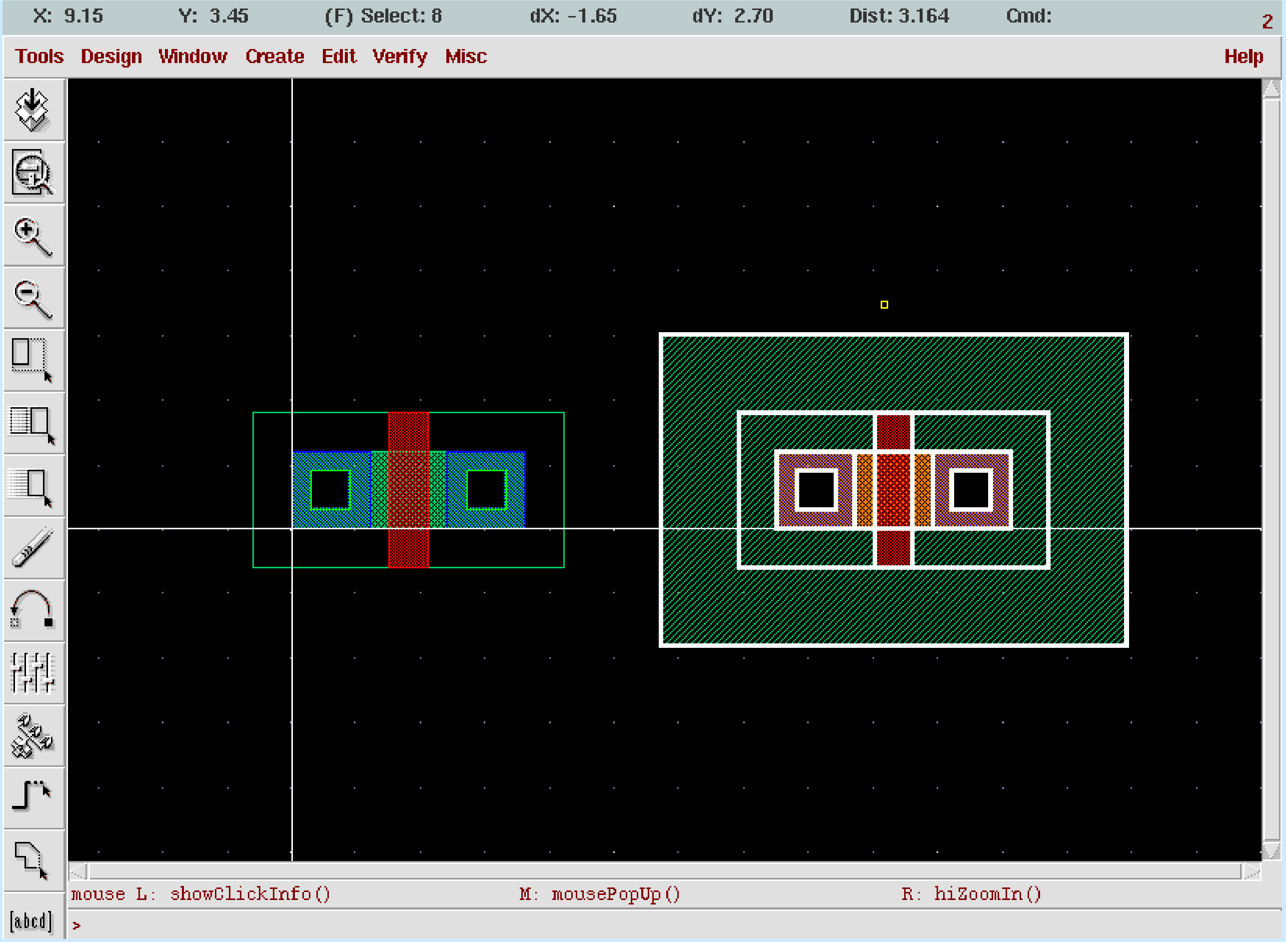

The n-well must extend over the PMOS active area by a large margin, at least 1.8u

# Placing the PMOS and NMOS transistors

Based on our original signal flow diagram, it is more desirable to place the PMOS transistor directly on top of the NMOS transitor- for a more compact layout.

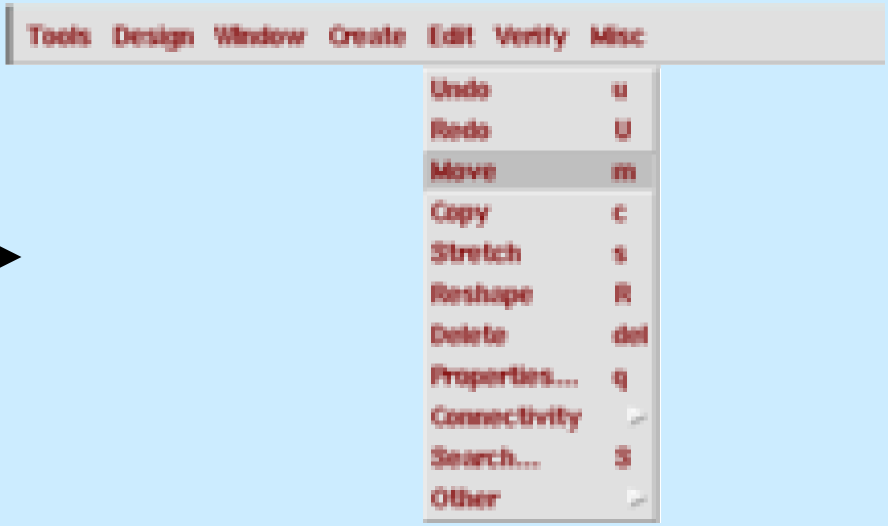
# Placing the PMOS and NMOS transistors

1. Select the PMOS transistor



# Placing the PMOS and NMOS transistors

2. From the menu *Edit* select the option *Move*



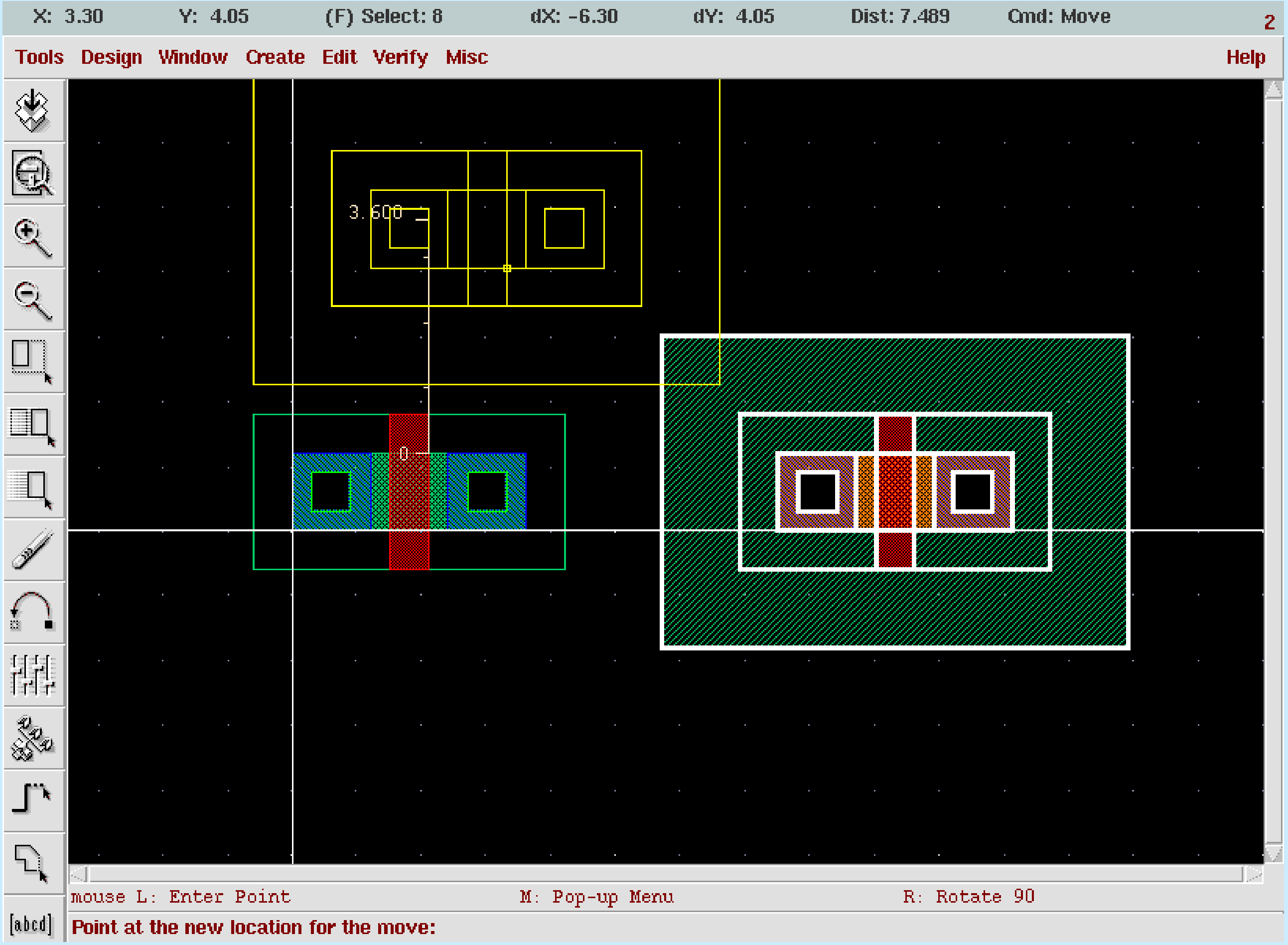

A window will pop-up:


We have to change the *Snap Mode* option to *Anyangle* so that we can move the transistor freely.

# Placing the PMOS and NMOS transistors

## 3. The reference point Pick

After we have picked the reference point, the outline of the shape will appear attached to the cursor and we will be able to move the shape around.

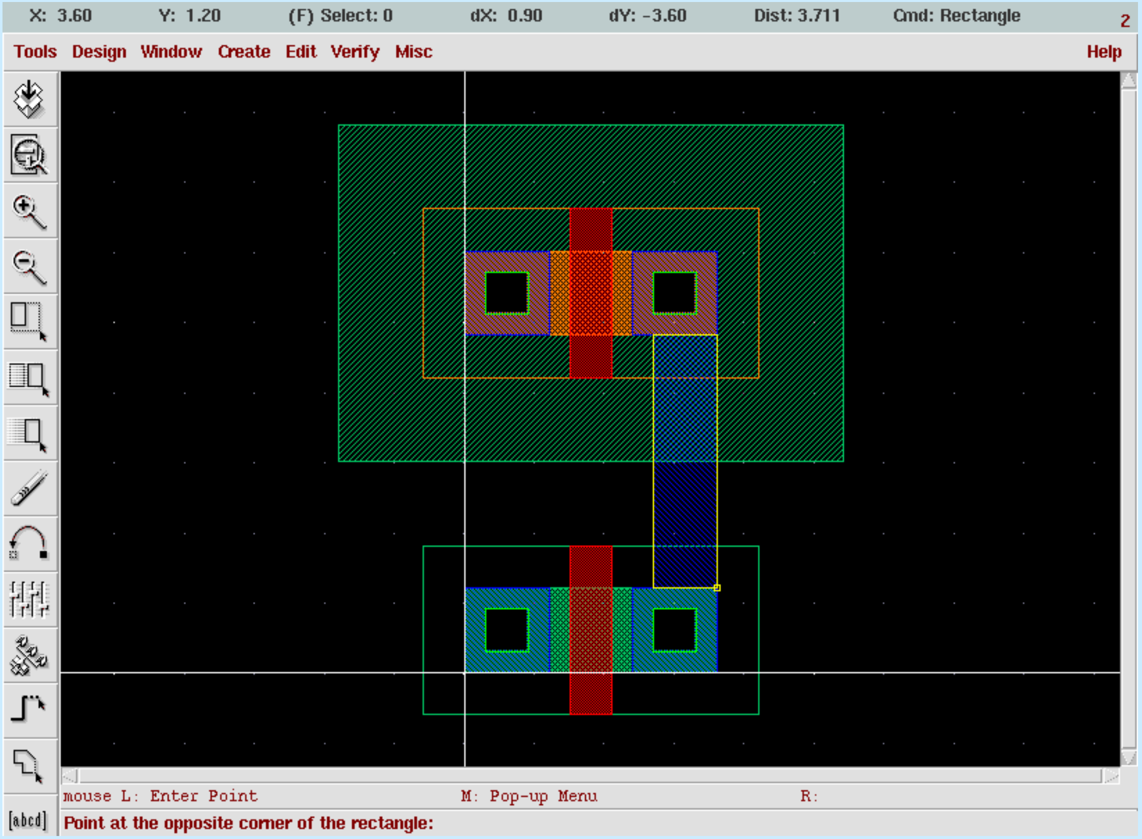


# Placing the PMOS and NMOS transistors

4. Place the transistor

# Connecting the Output

1. Draw a Metal-1 rectangle between NMOS and PMOS drain region contacts



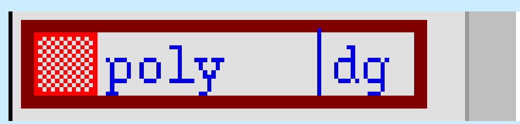

- The minimum Metal-1 width is 0.9u (3 lambda), thus narrower than the Metal-1 covering the contacts.

- The transistors are completely symmetric, the source and drain regions are interchangeable.

# Connecting the Input

Connect the gates of both transistors to form the input.

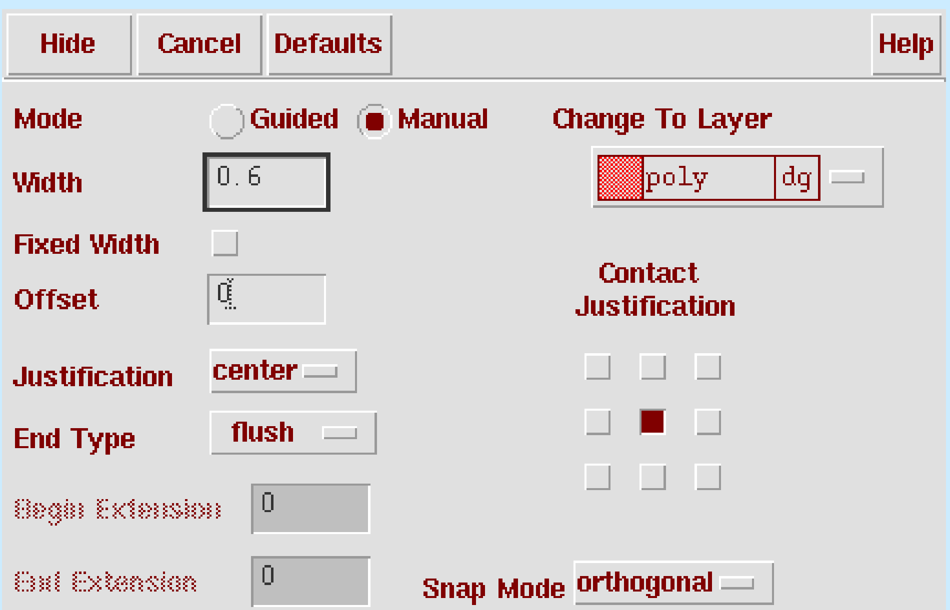
1. Select *poly* layer from the LSW



# Connecting the Input

2. From the *Create* menu select *Path* ▶


The path options box will pop up:



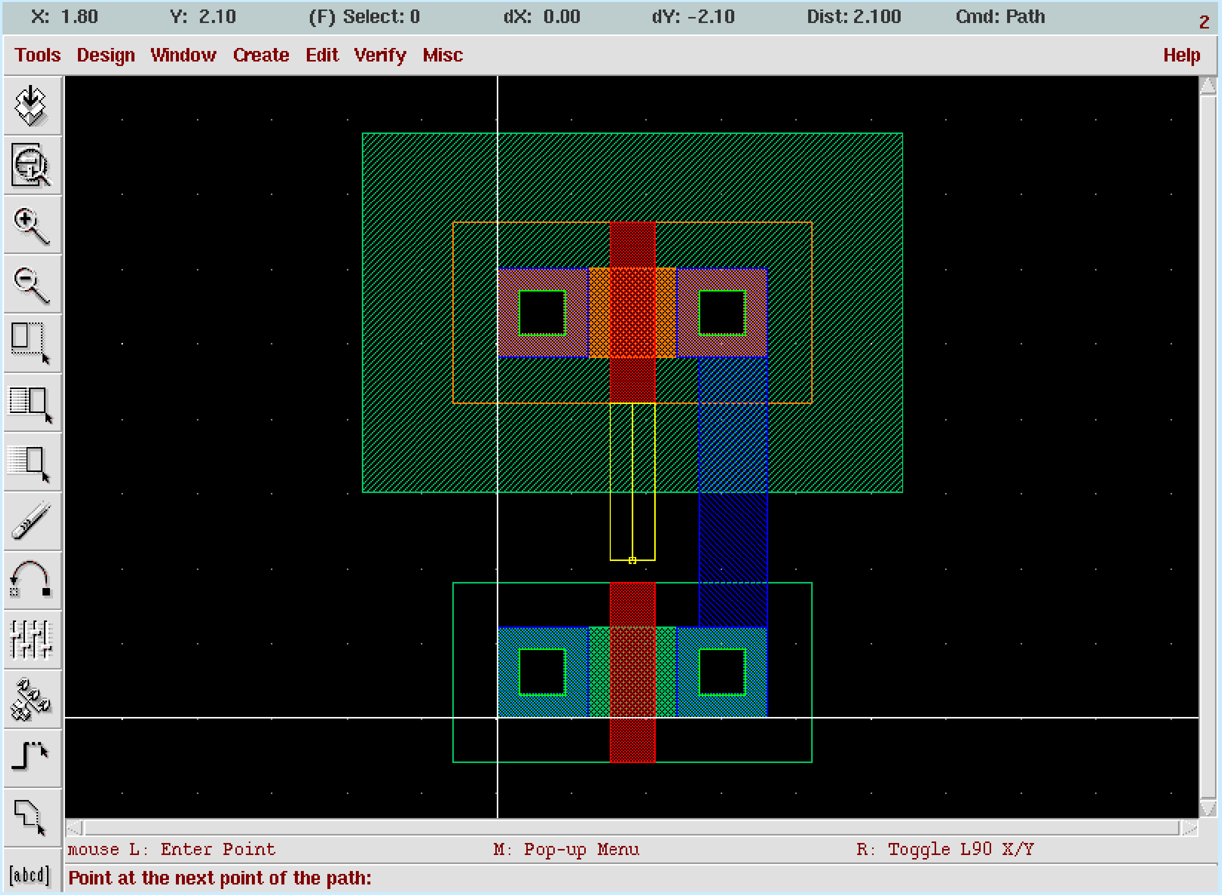

In the *path* mode you can draw lines with the selected layer. The width of the drawn line can be adjusted, the default is the minimum width of the selected layer.

# Connecting the Input

3. Start path

- Click on the middle of the PMOS poly extension a ghost line appear

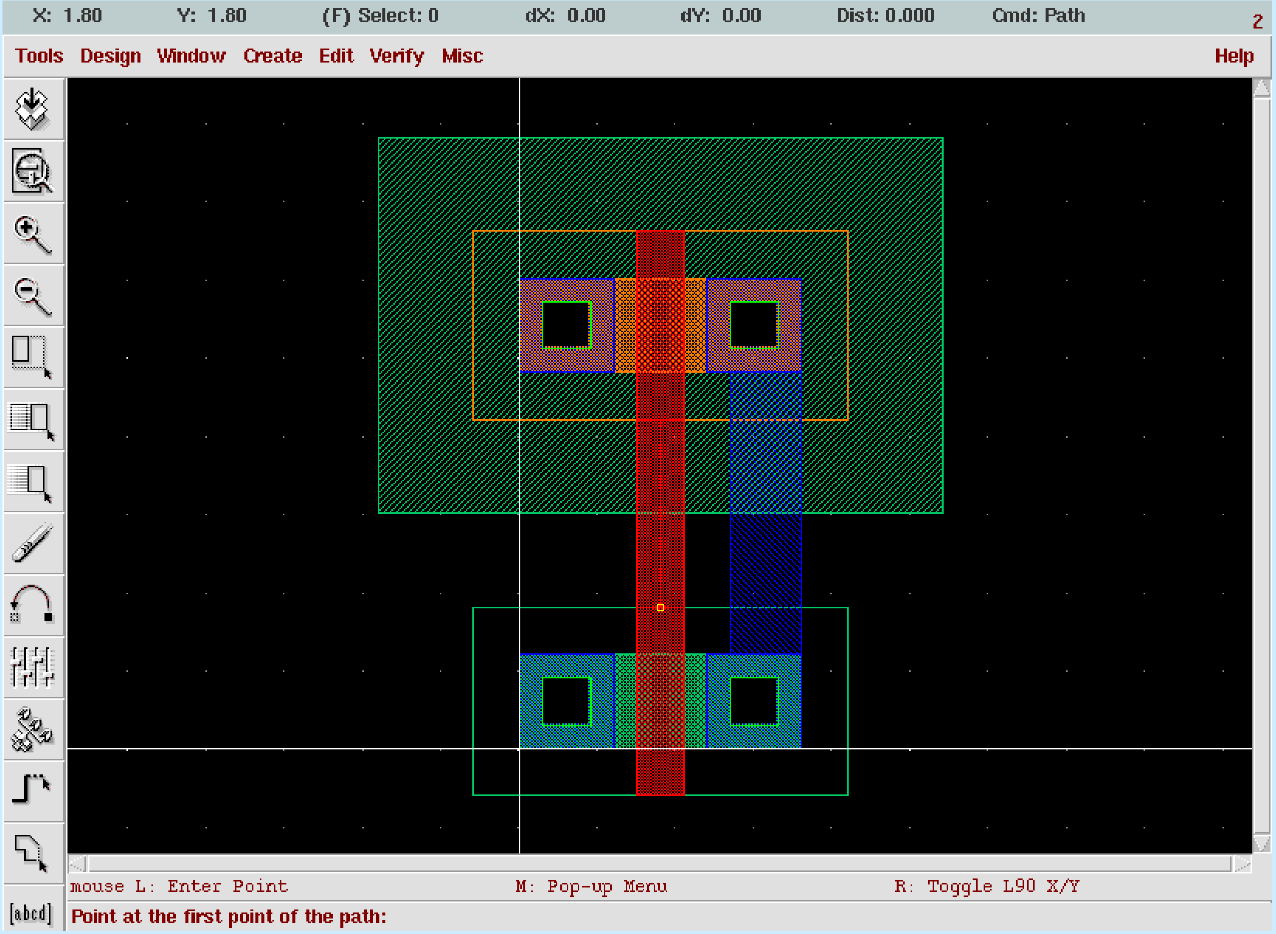
- Move this ghost line to the NMOS poly extension.



# Connecting the Input

4. Double click to finish path

A single click will finish a line segment and let you continue drawing, a double click will finish the path.



# Making a Metal-1 Connection for the Input

Now we have to make a connection from the poly layer to the Metal-1 layer.

This connection can be done:

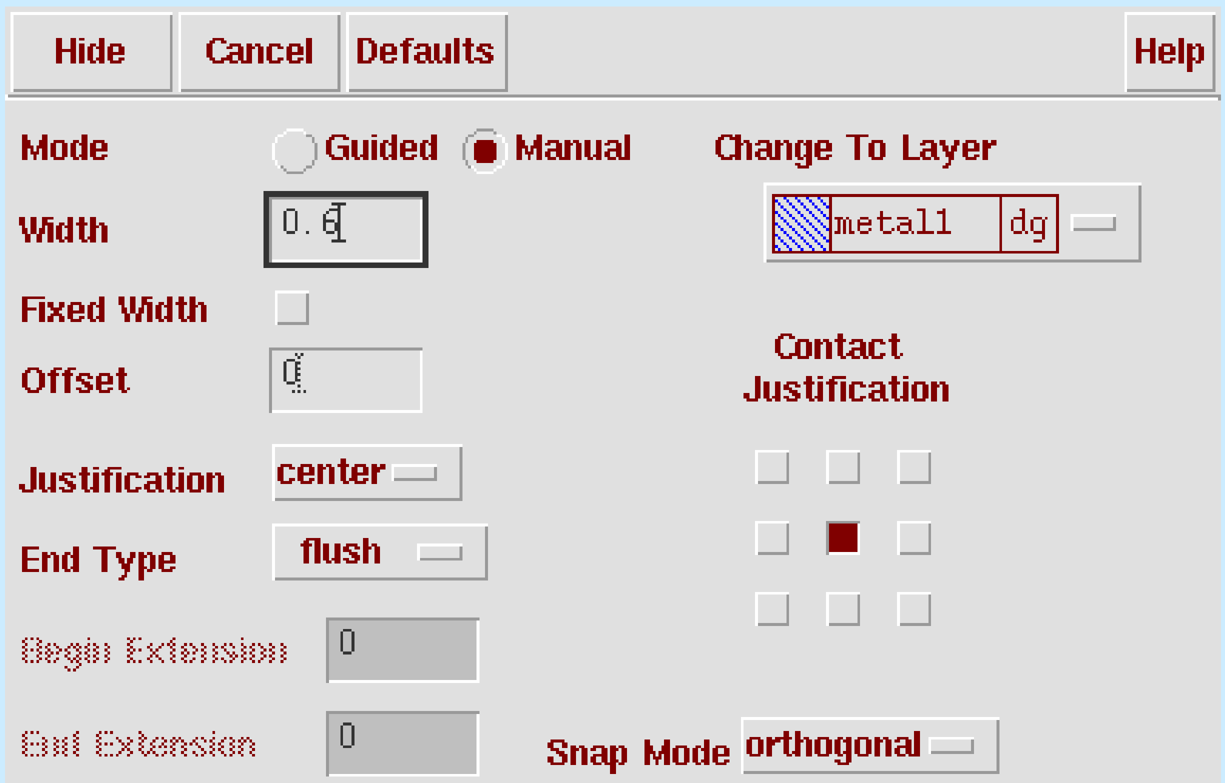
- manually by drawing a poly contact layer between Metal-1 and poly,

- using the *path* command to automatically add the contacts.

# Making a Metal-1 Connection for the Input

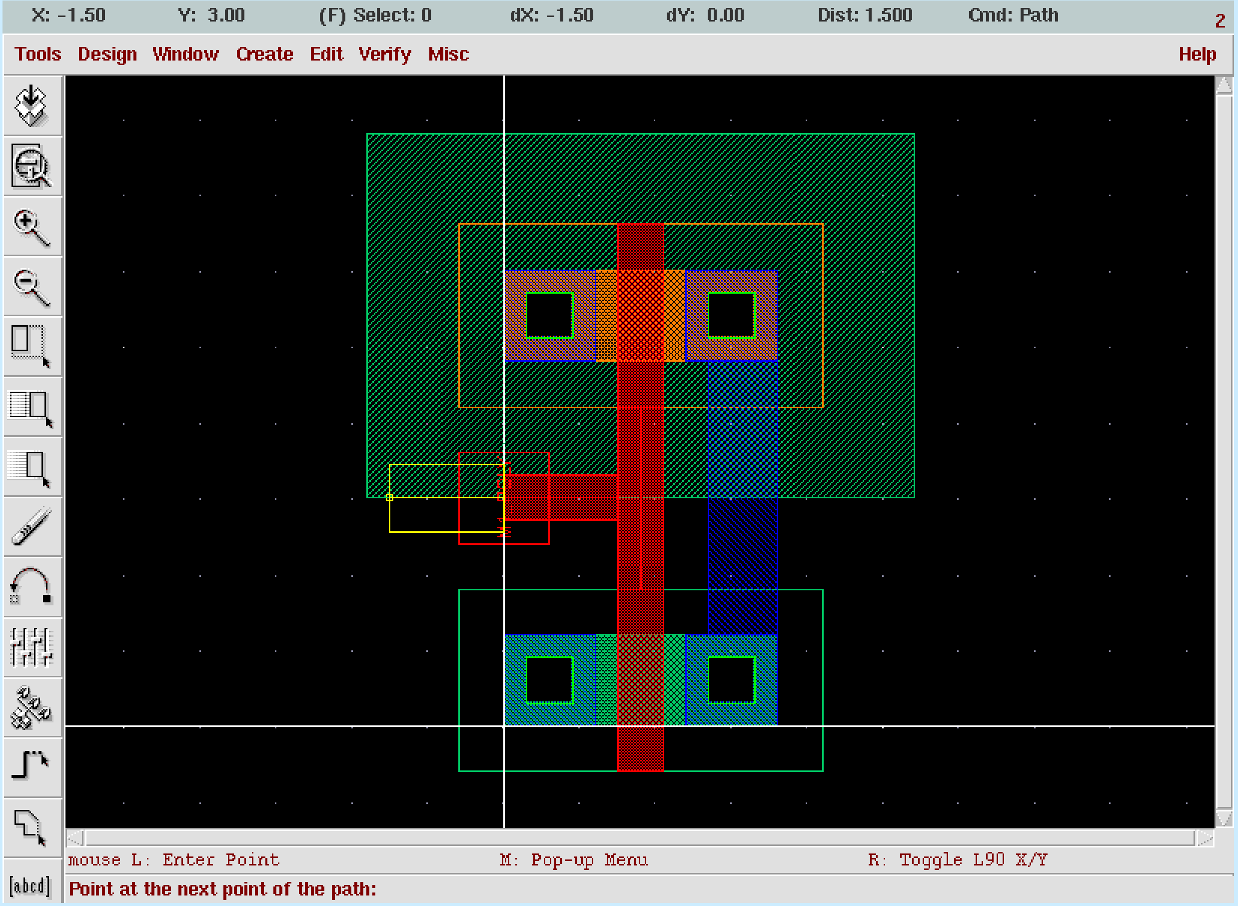
1. Starting from the poly line connecting the gates, start drawing a horizontal poly path

2. On the *Path Options* dialog box, click on *Change To Layer* and switch to *Metal1* ►



# Making a Metal-1 Connection for the Input

This will automatically add a contact to the end of the current path.

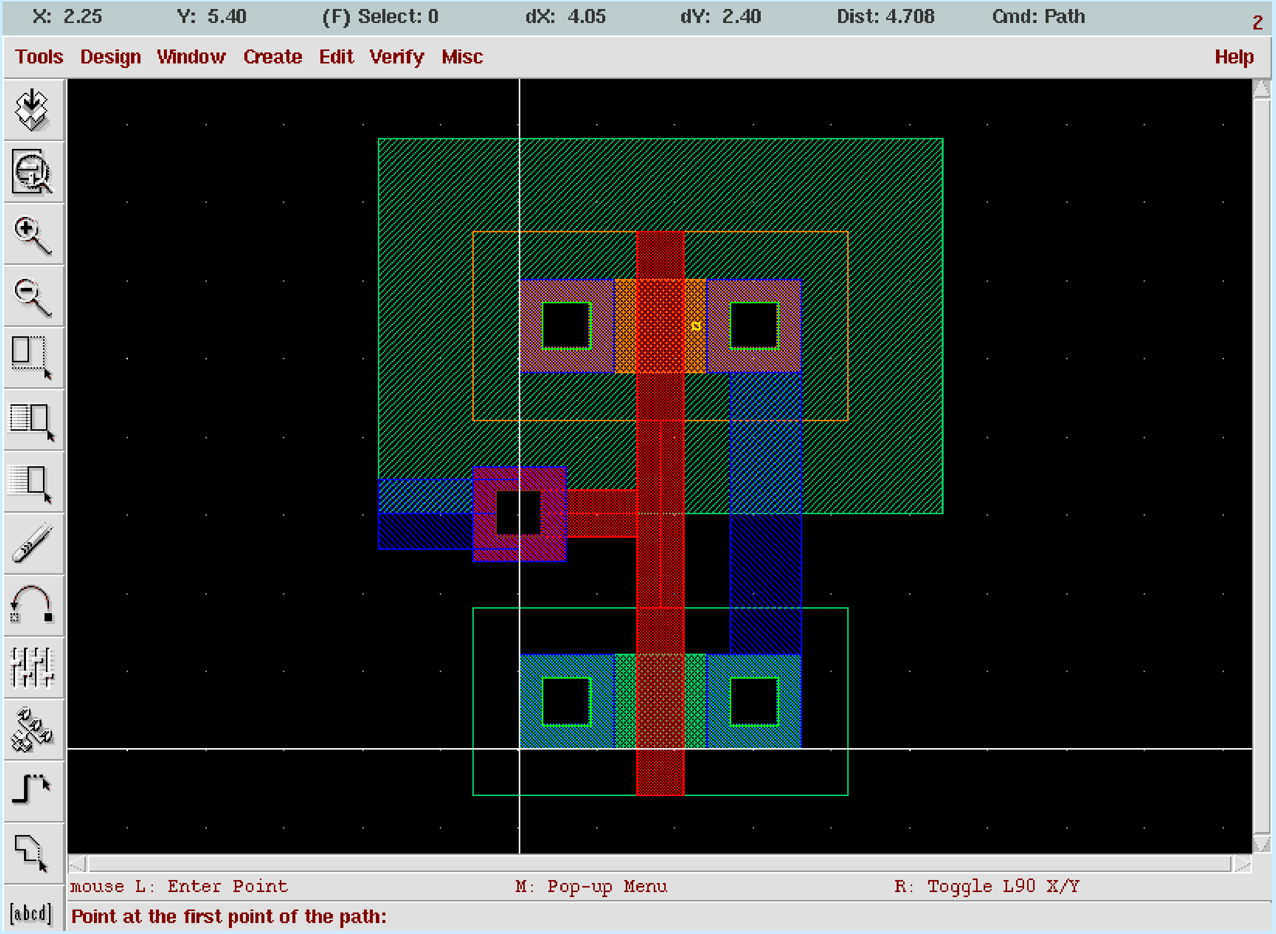


# Making a Metal-1 Connection for the Input

3. Finish the path (by double clicking)

**SHIFT-F** to see all levels of hierarchy.

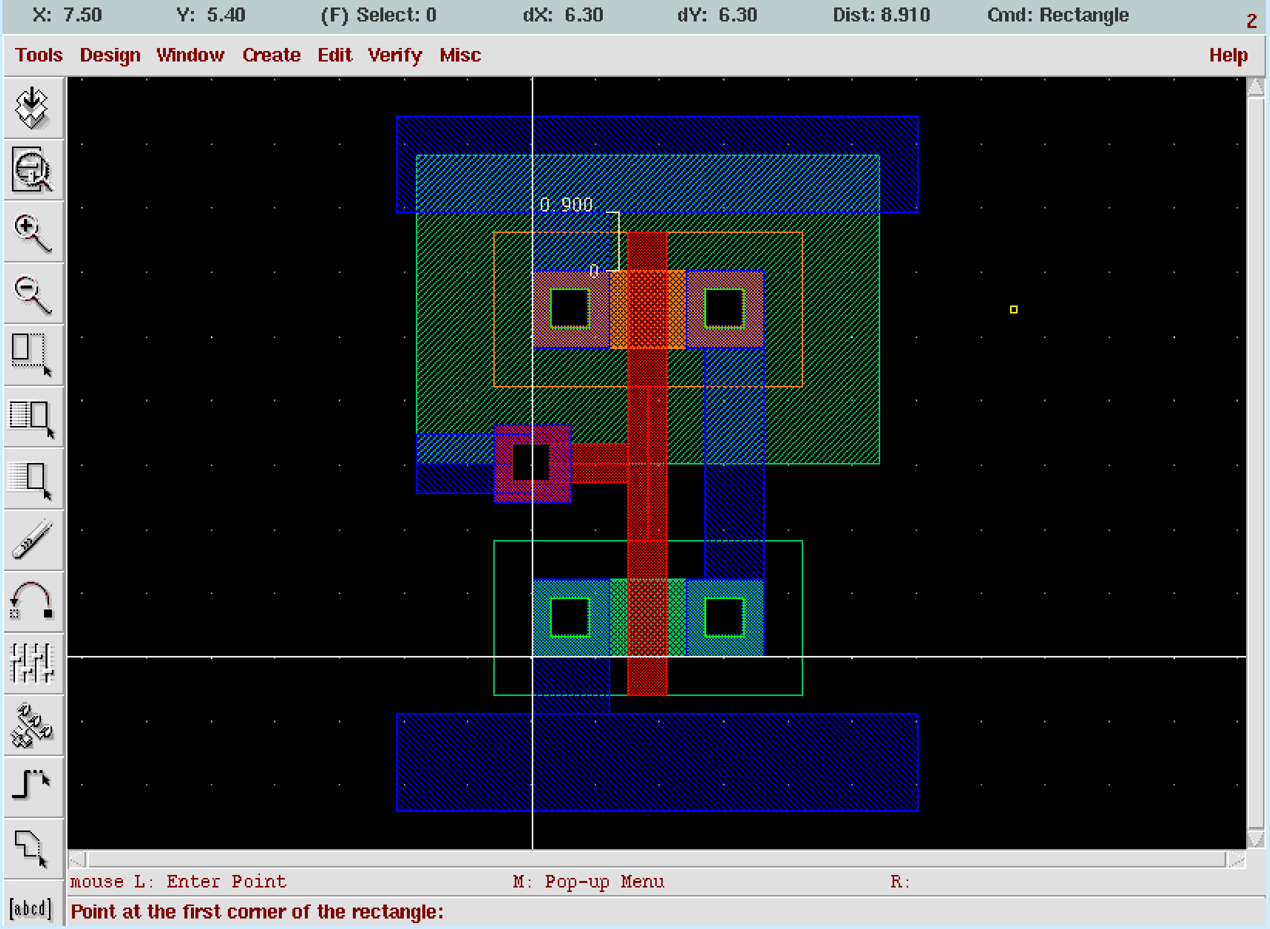
**CTRL-F** to see a single layer of hierarchy.



# Power Rails

Our Signal Flow Graph suggests horizontal power and ground lines in Metal-1.

1. Draw the Power Rail in Metal-1 above the PMOS
2. Draw the Ground Rail in Metal-1 below the NMOS

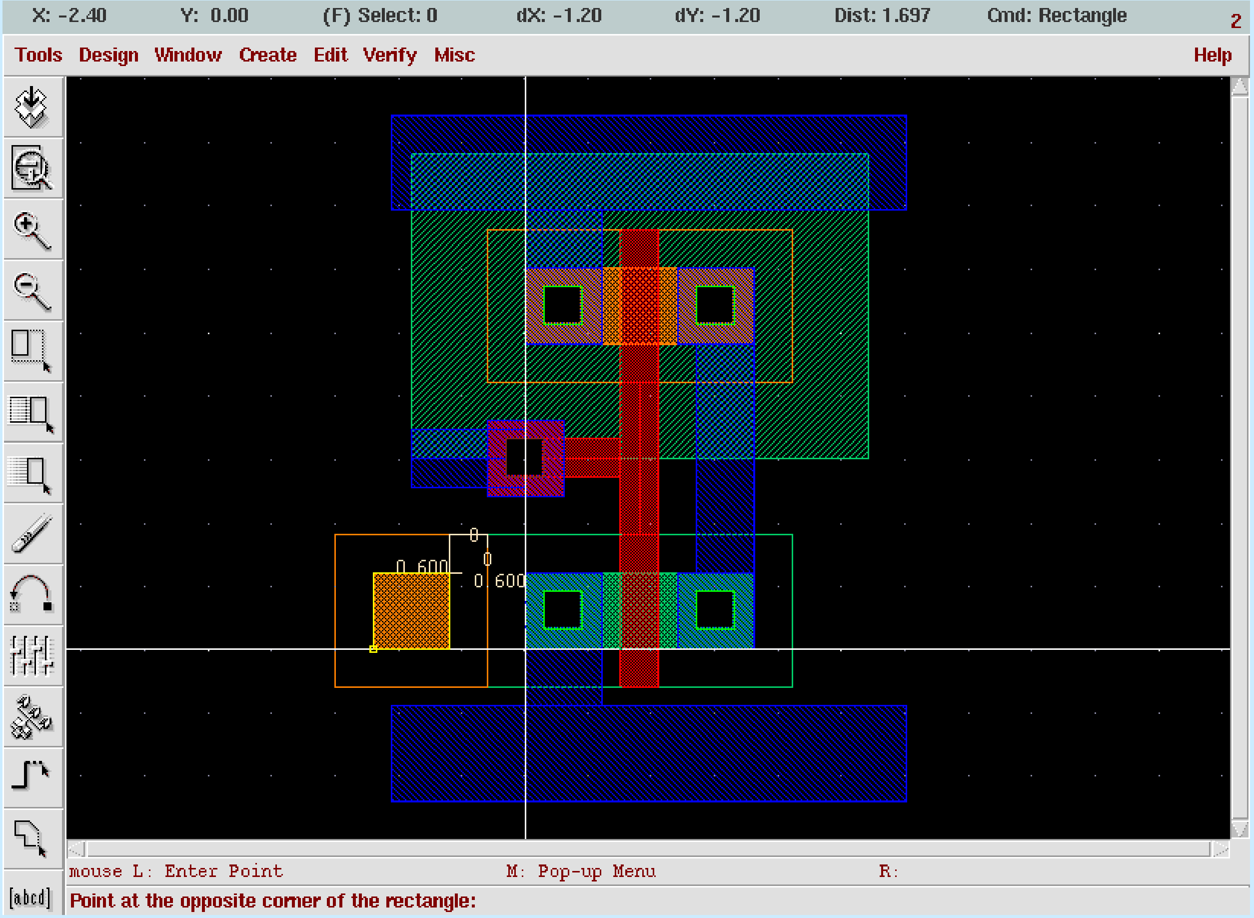


# P-Substrate Contact

The substrate on which the transistors are built must be properly biased.

1. Draw a P-select square next to the NMOS transistor.

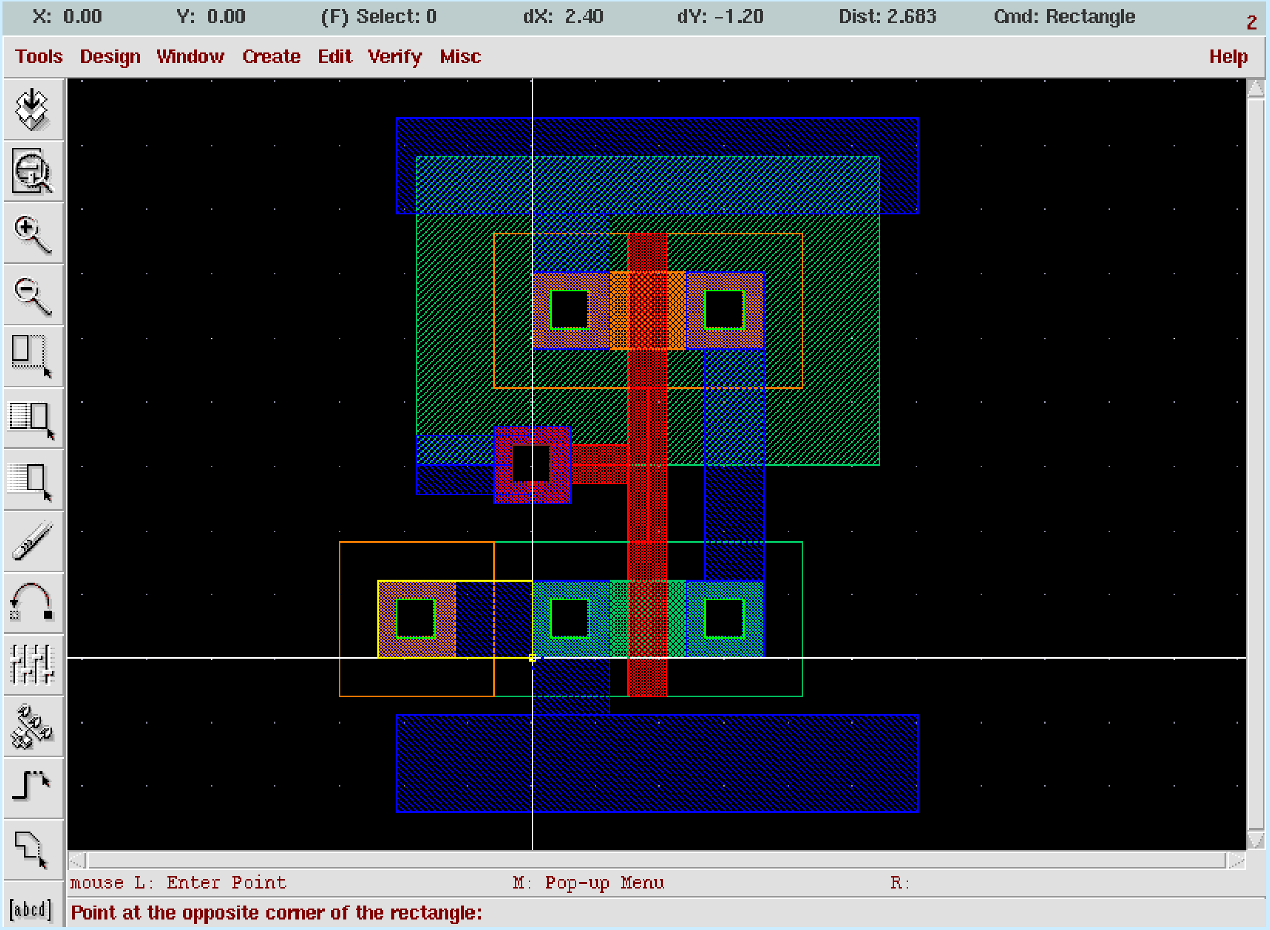
2. Draw a P-active square inside the P-select area.



# P-Substrate Contact

3. Draw the active contact square inside the p-type active area.

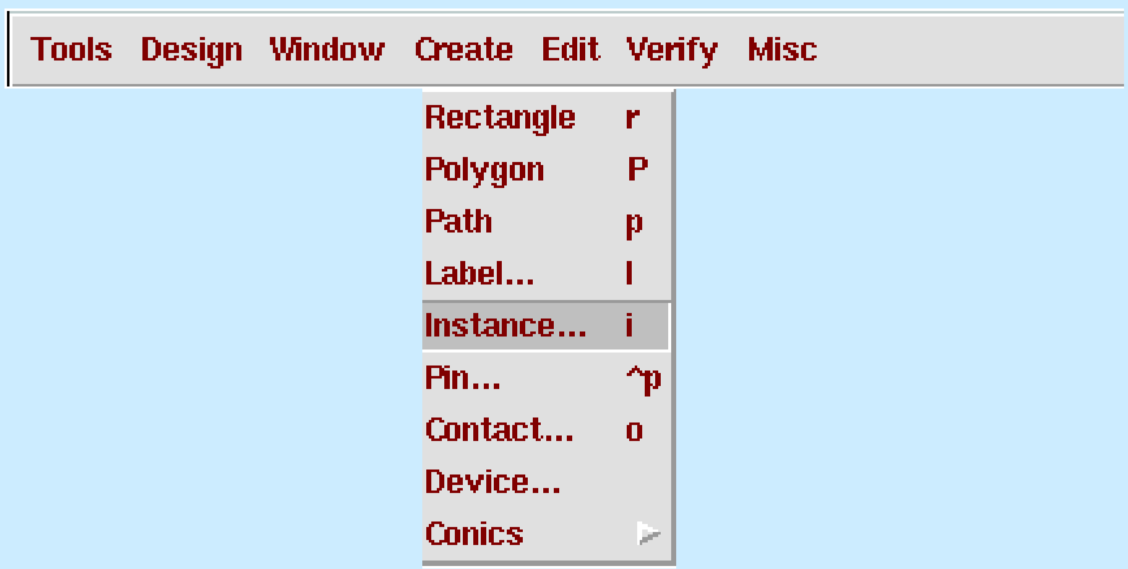
4. Make a metal connection to ground, covering the entire substrate contact.



# N-Substrate Contact

The PMOS transistor was placed within the n-well, which has to be biased with the VDD potential.

1. From the menu *Create* select option *Instance*

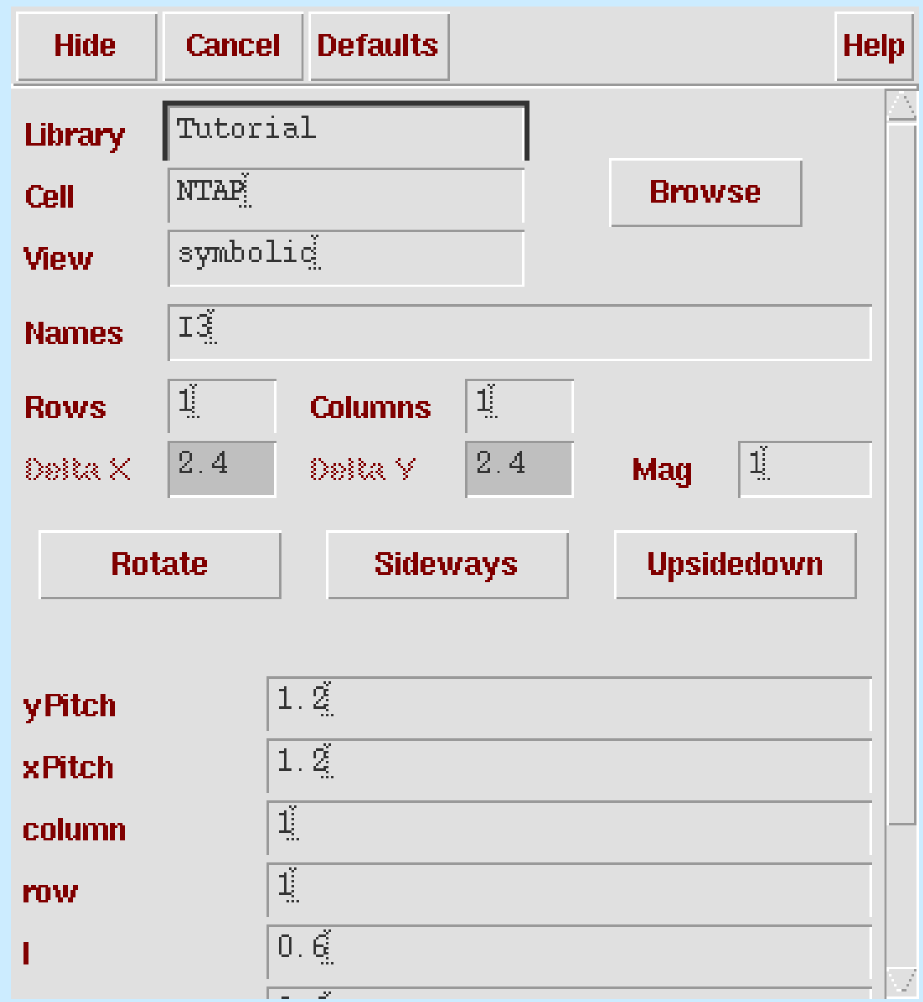


# N-Substrate Contact

The instance options menu will pop-up:

**Provide:**

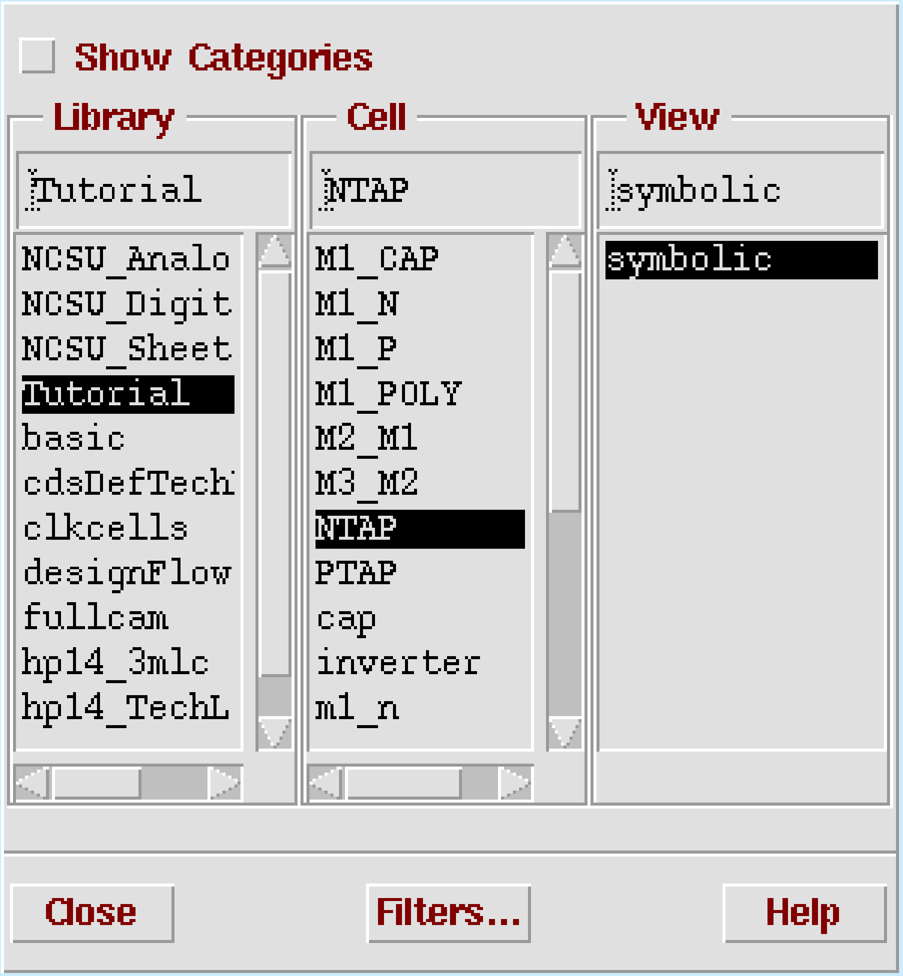
- a cell name
- library



# N-Substrate Contact

Choose the library, cell and cell view.

Your selection will be transferred to the Instance options menu.

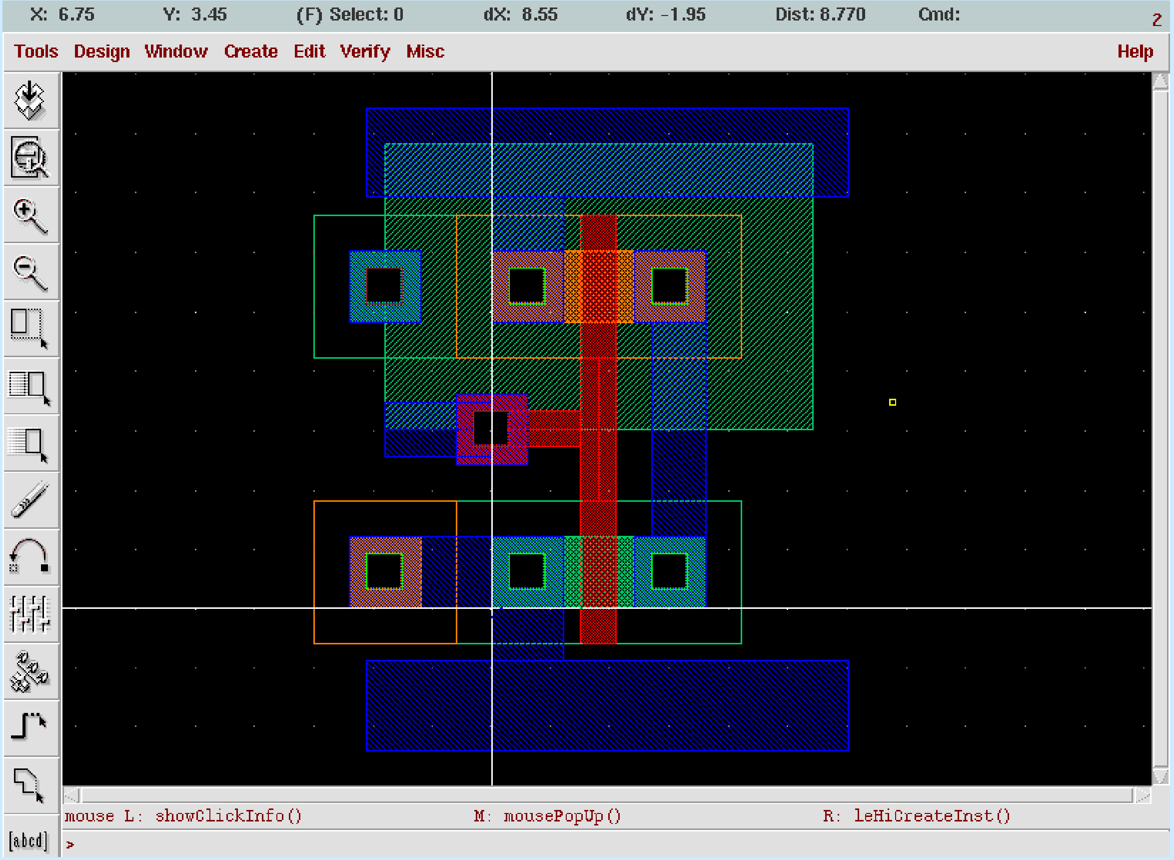


# N-Substrate Contact

2. Move the instance to the desired location.

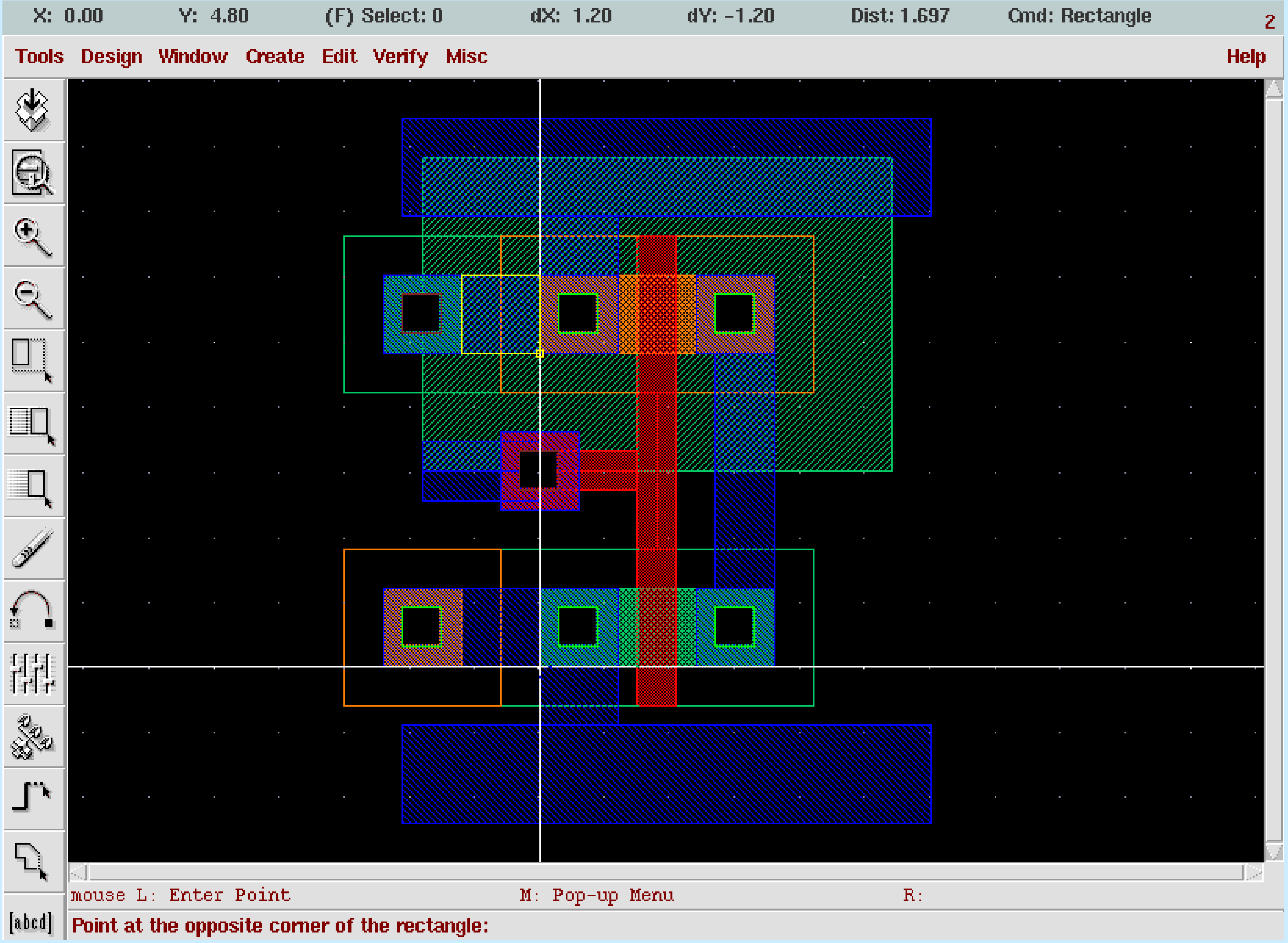
3. Place the instance.

The n-well in this exapmle is not wide enough to accomodate both the PMOS transistor and rge contact, which will obviously generate a rule violation. This will have to be dealt with in the next step.



# N-Substrate Contact

4. Make the power connection.



# Enclosing the substrate contact

Enlarge the n-well, so that it also covers the substrate contact.

• draw an adjoining rectangle using the n-well layer
or
• modify the existing rectangle

1. Press *F4* on the keyboard to toggle selection mode.

The information bar will start displaying "(P) Select" (P for partial) instead of "(F) Select" (F for Full).

# Enclosing the substrate contact

2. Move cursor over the left edge of the n-well.

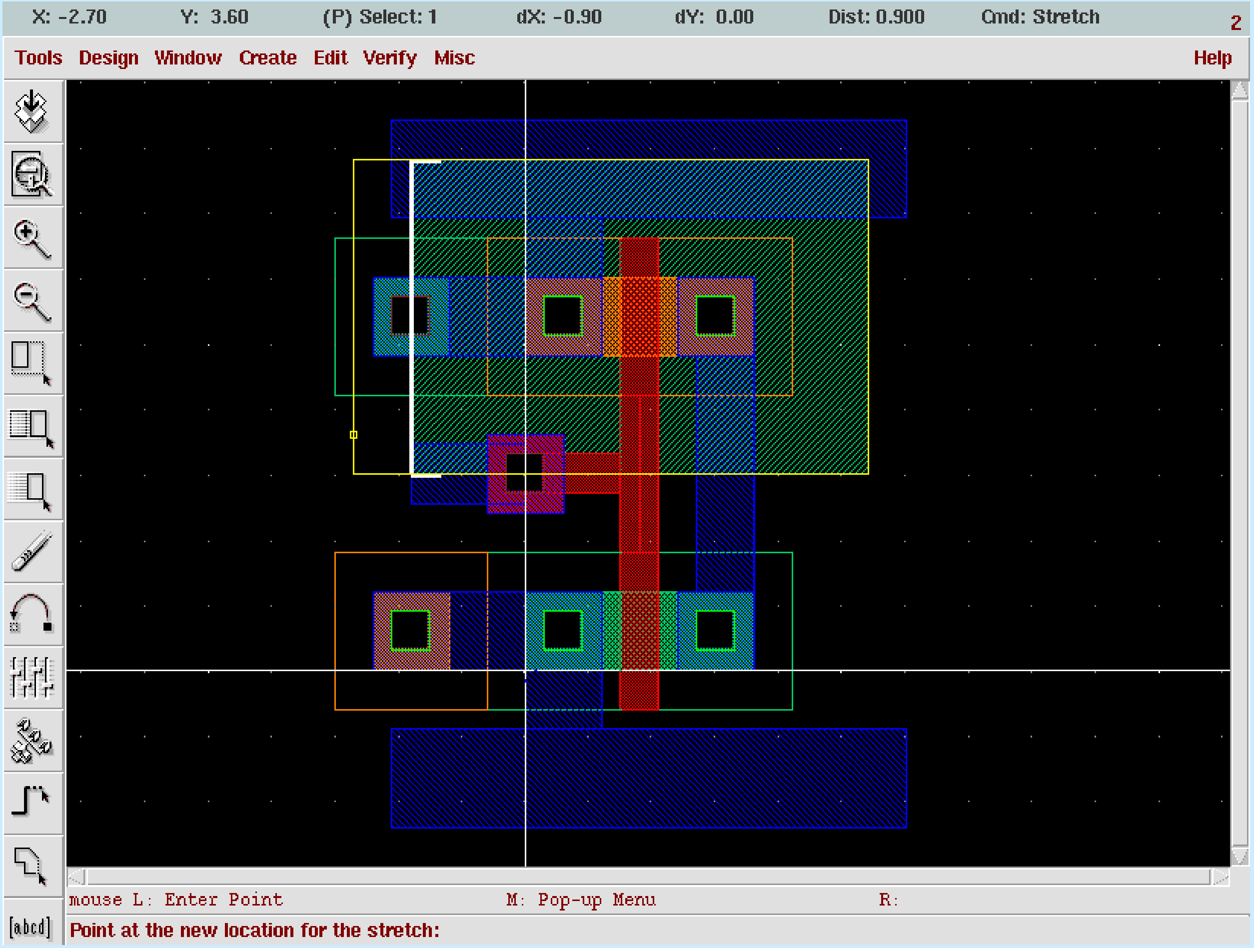
3. Click once to select the edge.

4. Move mouse over the selected edge (without pressing any mouse buttons).

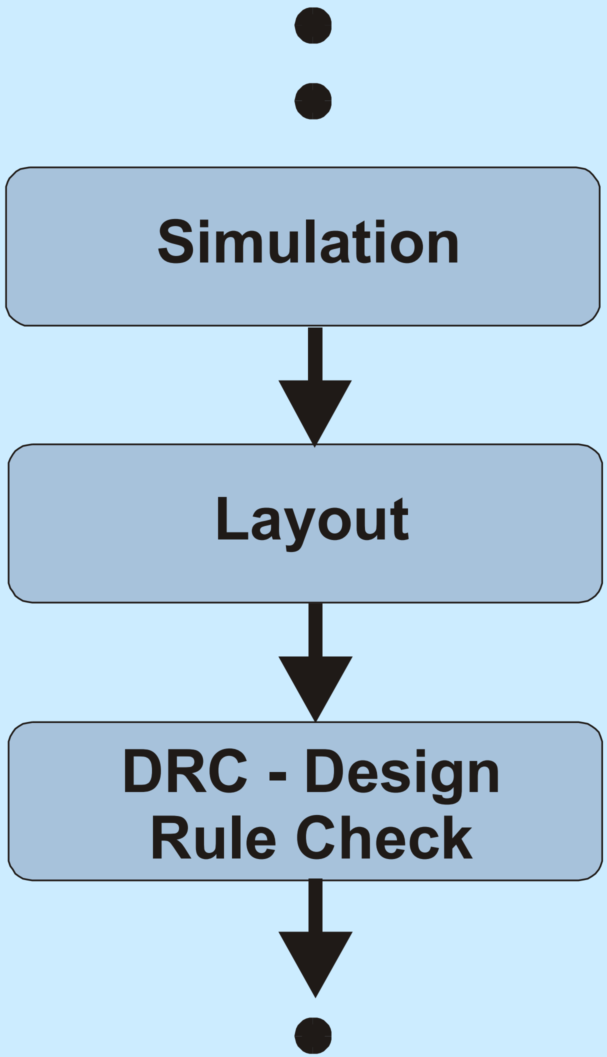
Cursor changes shape when you are close to the edge.

# Enclosing the substrate contact

5. Press and hold left mouse button when cursor changes above the selected edge.



# Design Rule Check (DRC)



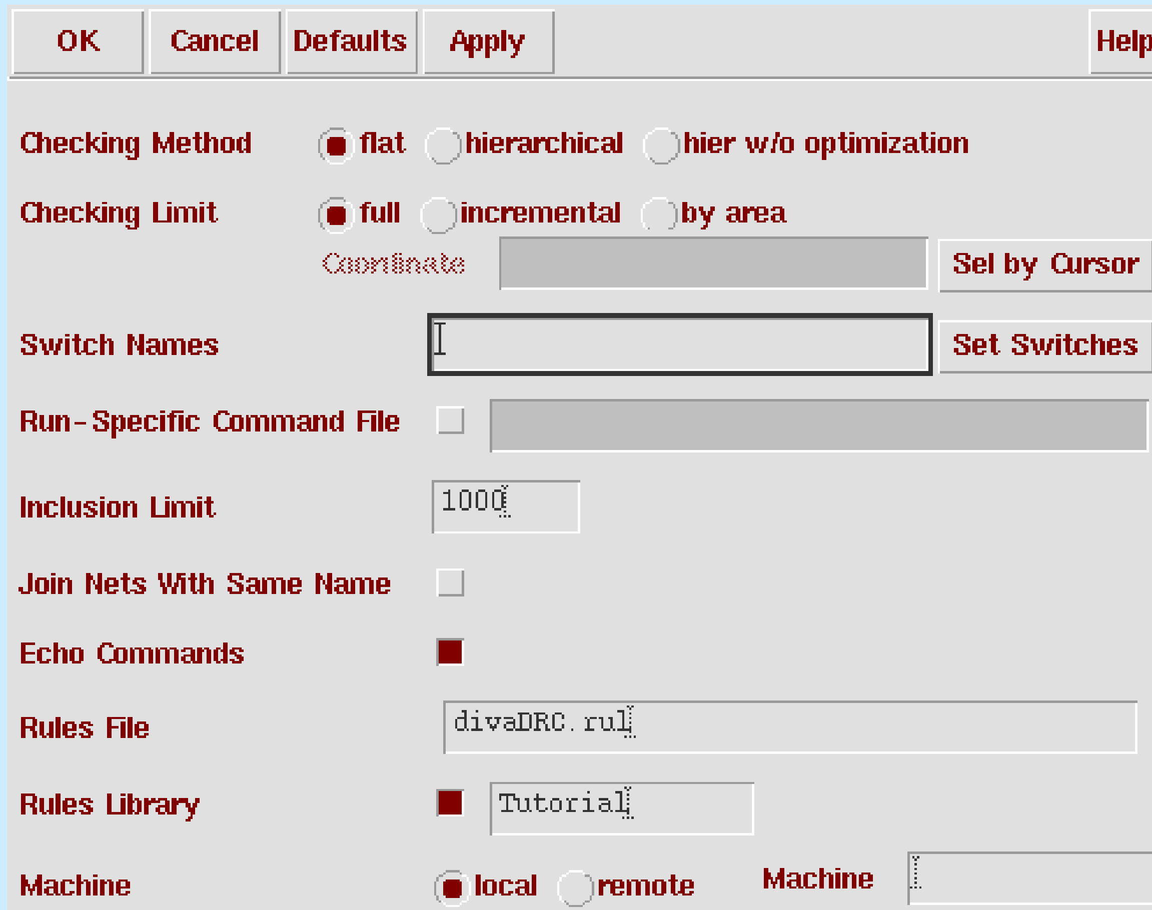

- The created mask layout must conform to a complex set of design rules, in order to ensure a lower probability of fabrication defects.

- *A* tool built into the Layout Editor, called Design Rule Checker, is used to detect any design rule violations during and after the mask layout design.

# Design Rule Checking

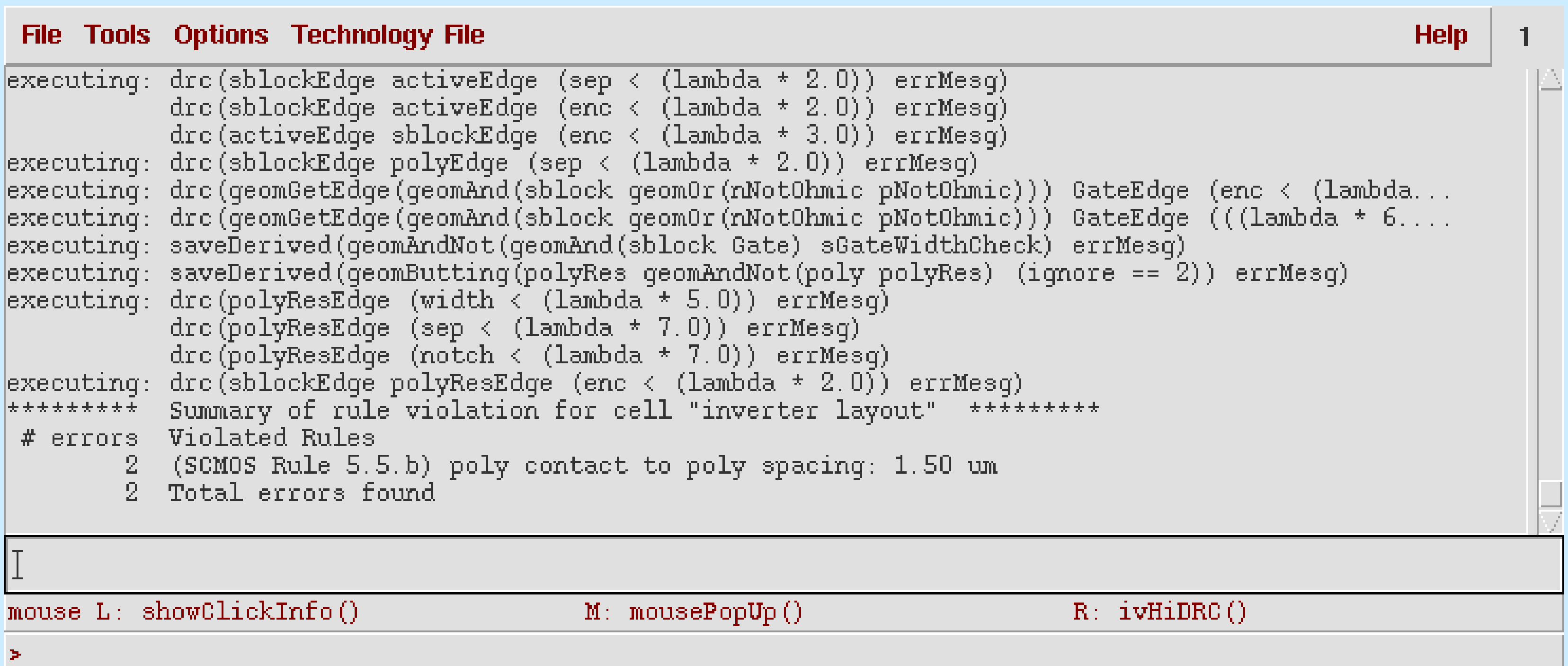
1. From the menu *Verify* select option *DRC*

This will pop-up the DRC options dialog box: ►



# Design Rule Checking

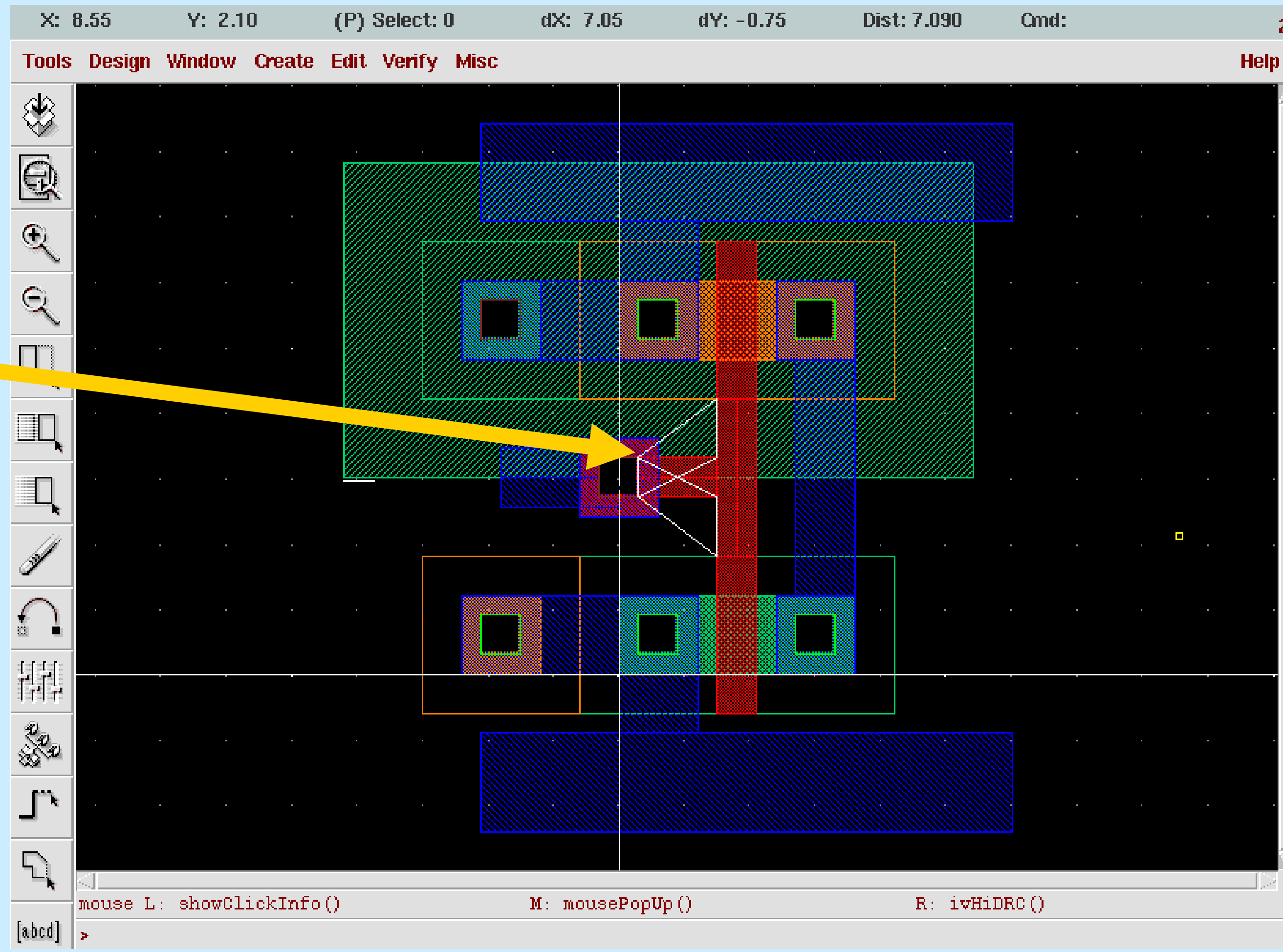
## 2. Start DRC




DRC results and progress will be displayed in the CIW.

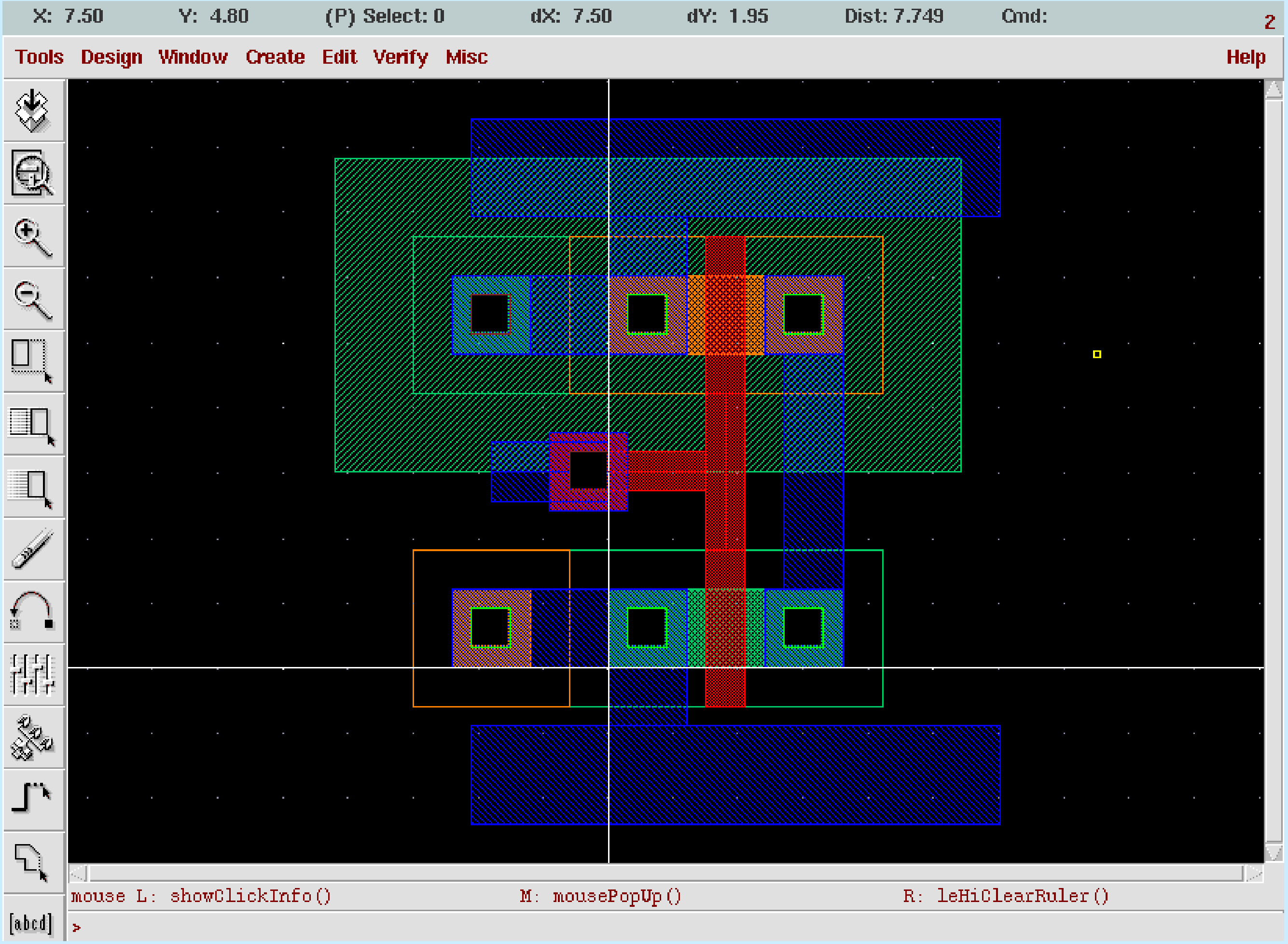
# Design Rule Checking

The errors are highlighted on the layout.

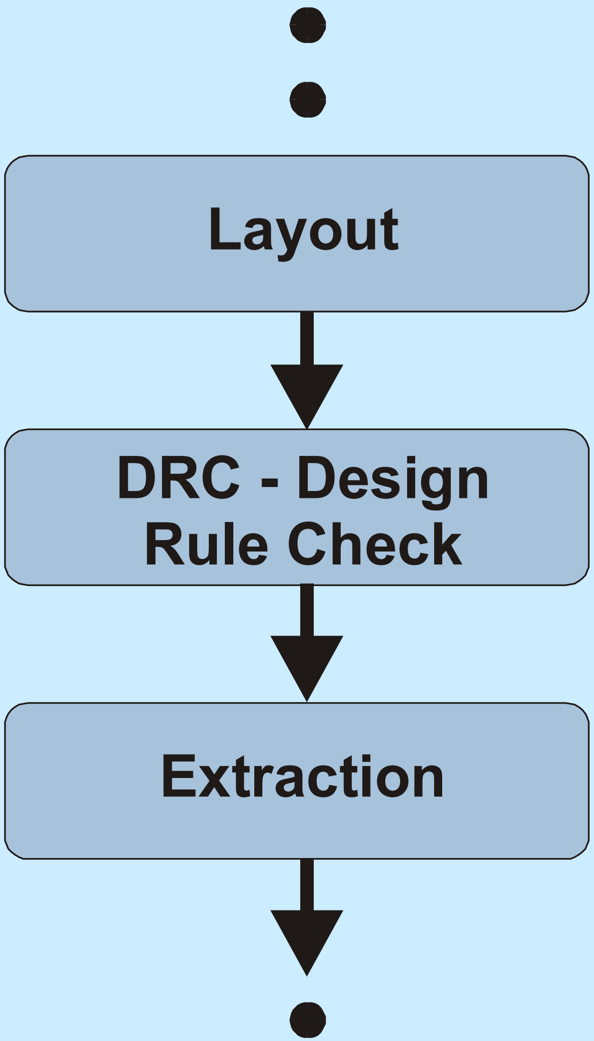


# Final Layout

This is the completed layout of the CMOS inverter.



# Circuit Extraction




- The mask layout only contains physical data. (In fact it just contains coordinates of rectangles drawn in different layers).

- The extraction process identifies the devices and generates a netlist associated with the layout.

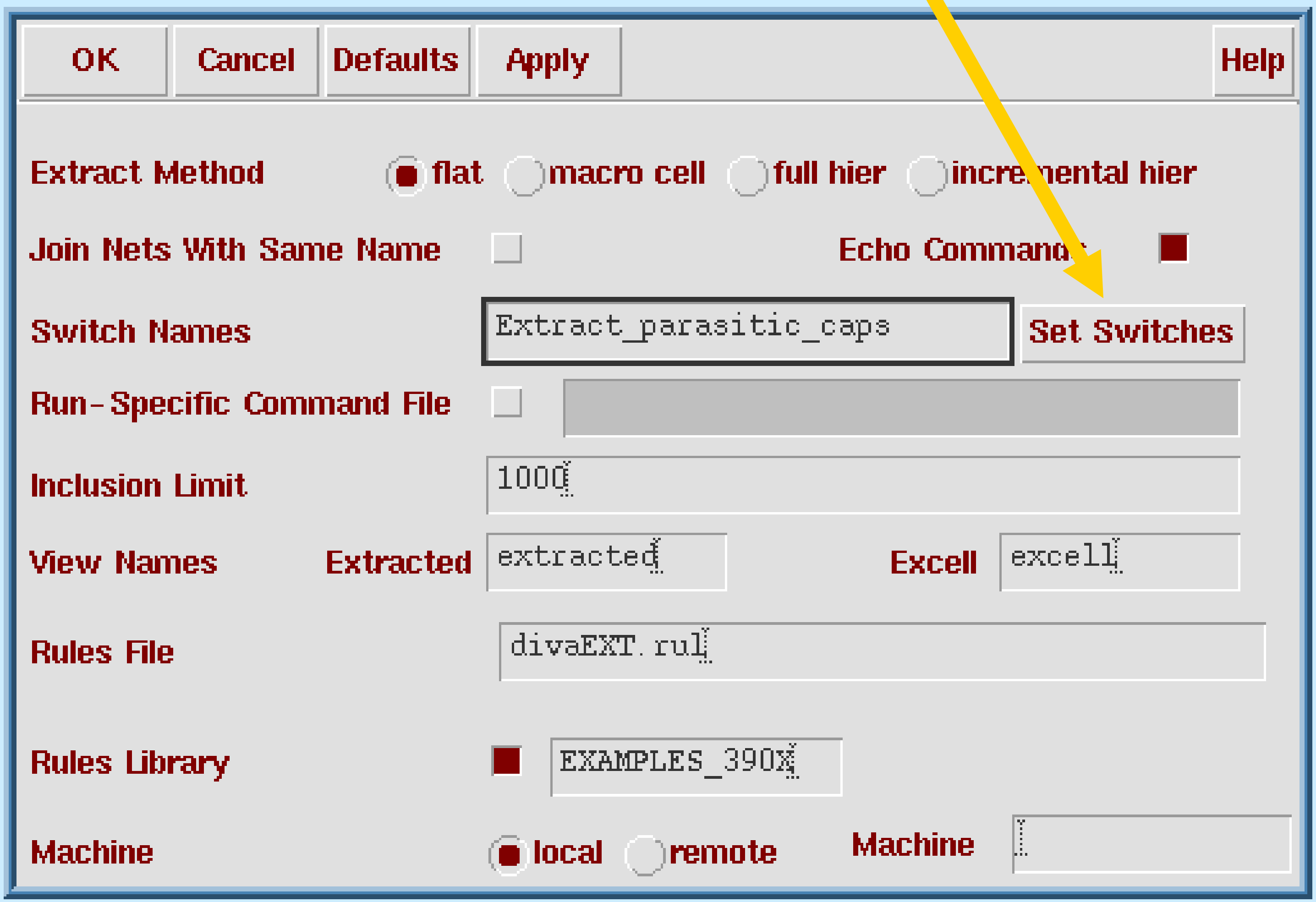
# Extracting from the Layout

Before extraction make sure that the design does not contain any DRC errors.

1. From the *Verify* menu select the option *Extract*

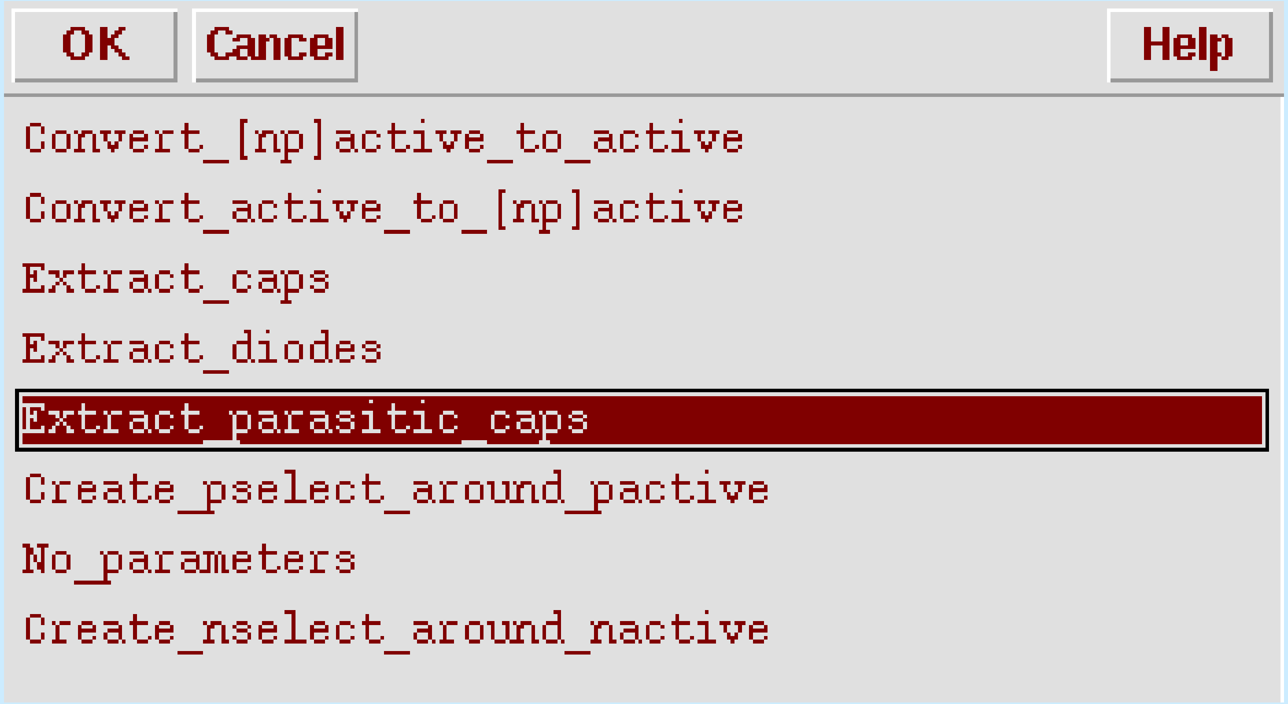
# Extracting from the Layout

To enable the extraction of parasitic devices, a selection parameter called a switch has to be specified.



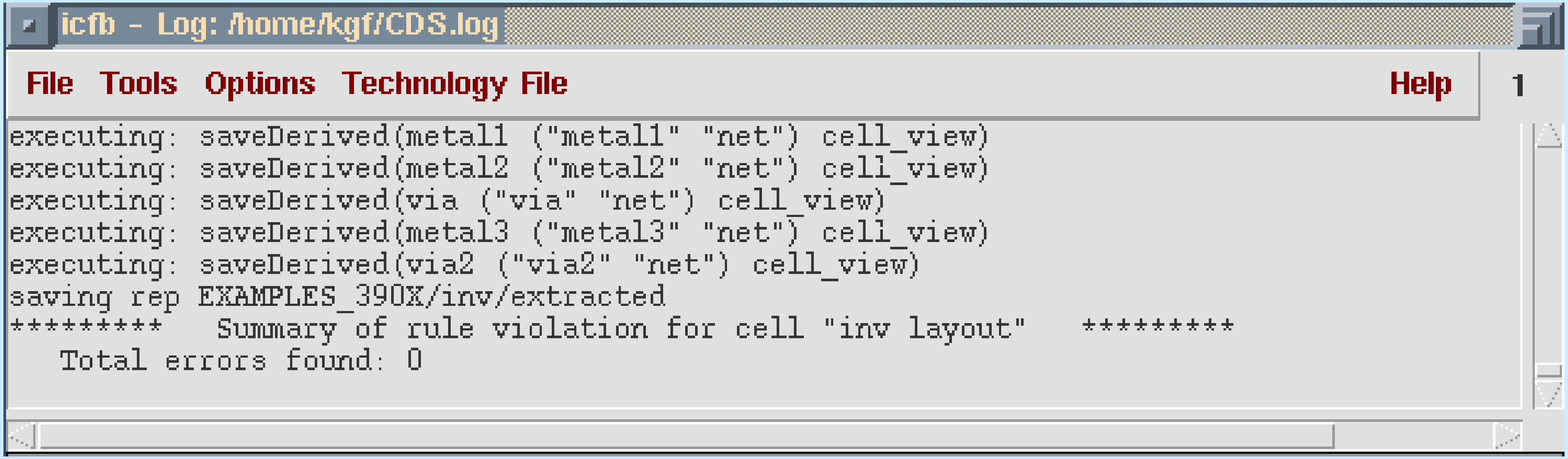
# Extracting from the Layout

The list of switches



# Extracting from the Layout

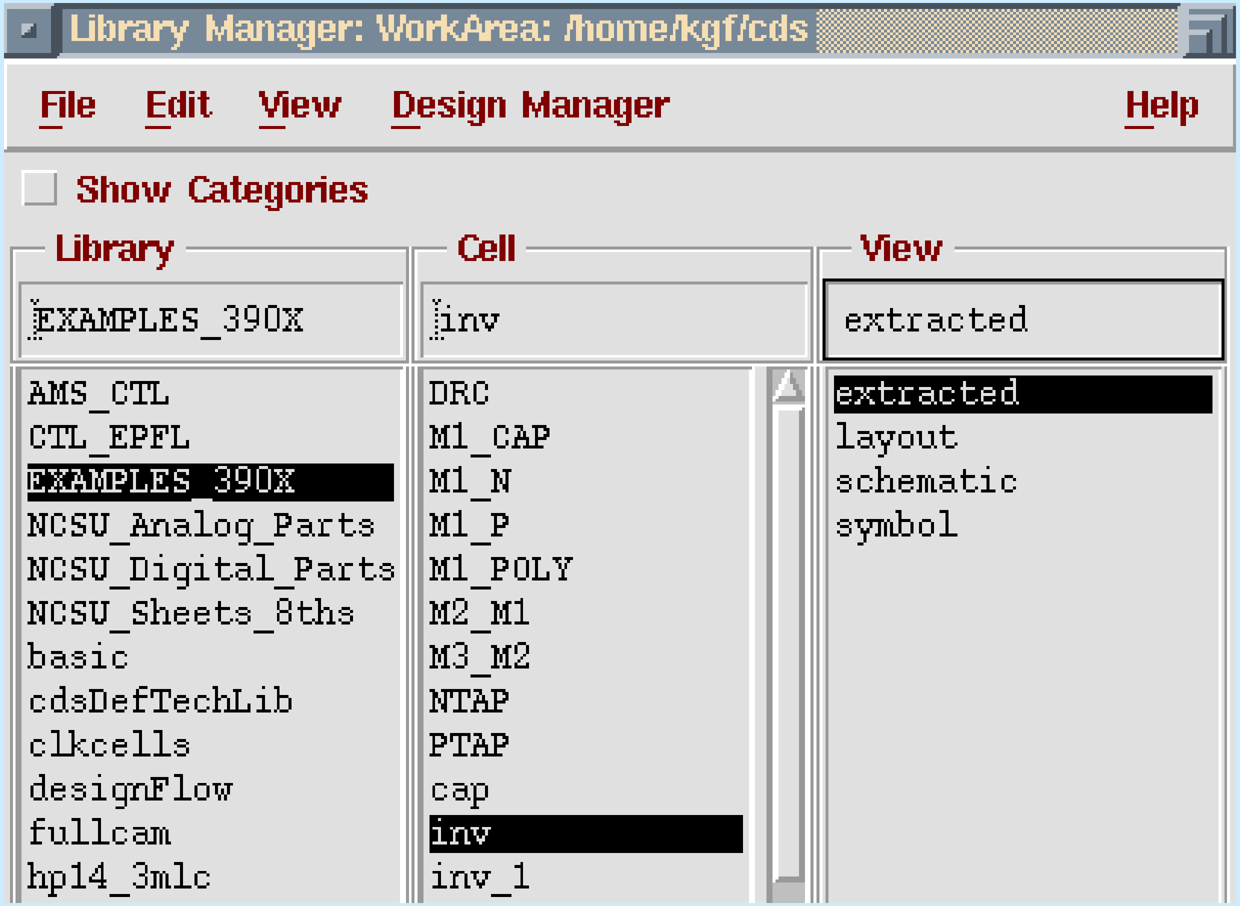
Check the Command Interpreter Window (the main window when you start Cadence) for errors after extraction.




**Following a successfull extraction you will see a new cell view called *extracted* for your cell in the library manager.**

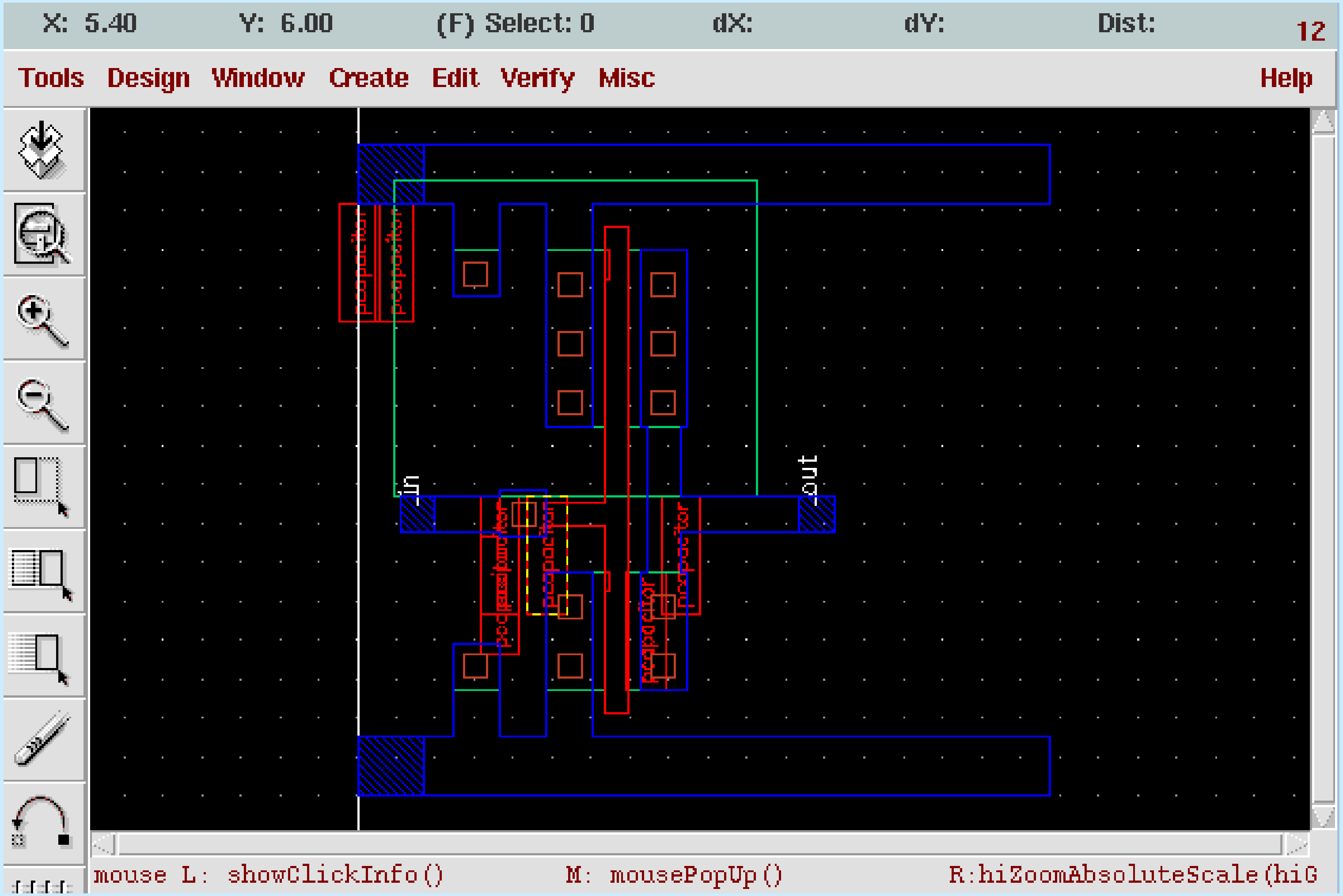
# The Extracted Cell View

A new cellview (called *extracted*) is generated in your library.



# The Extracted Cell View

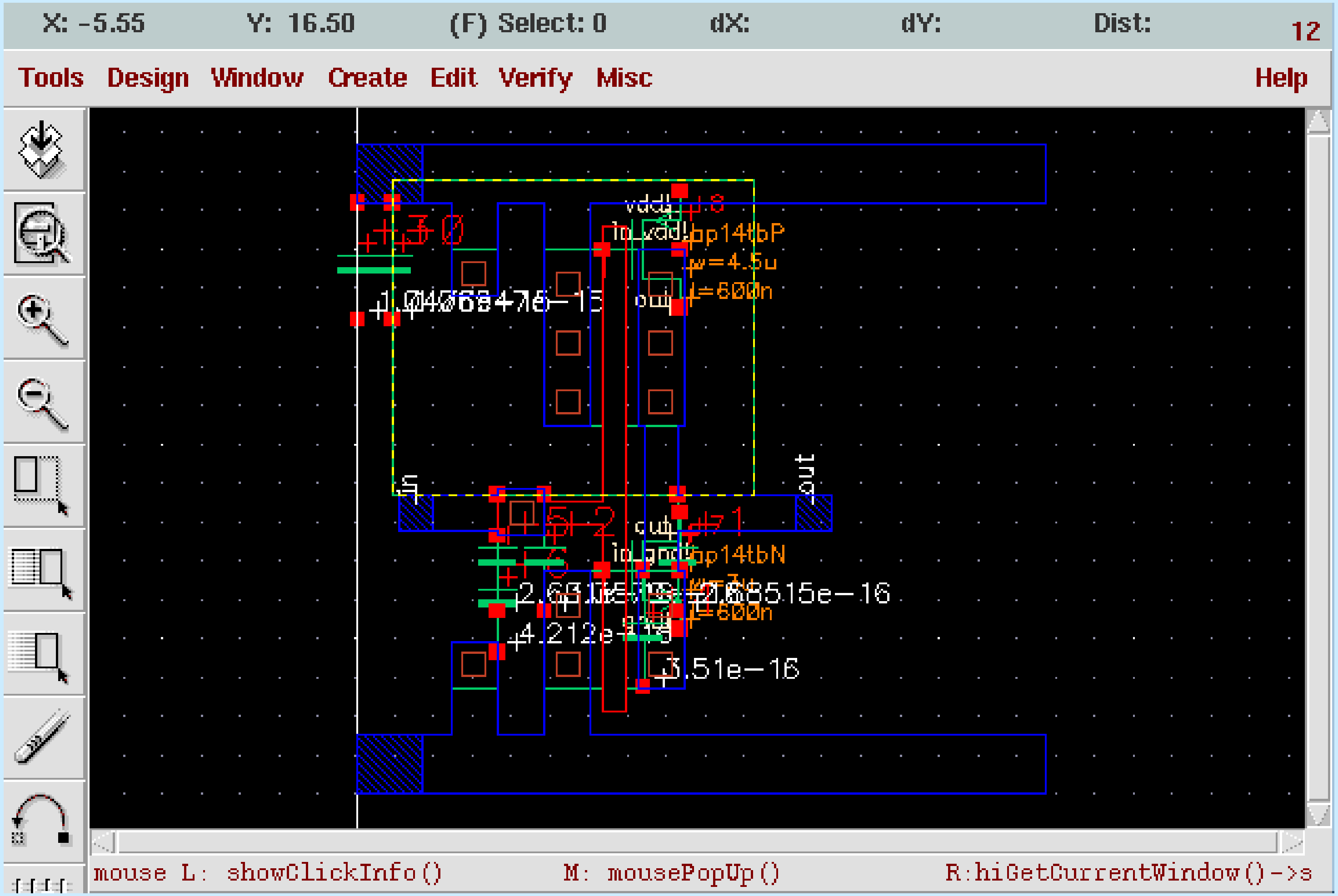
Load the cellview.




Notice that only the I/O pins appear as solid blocks and all other shapes appear as outlines.

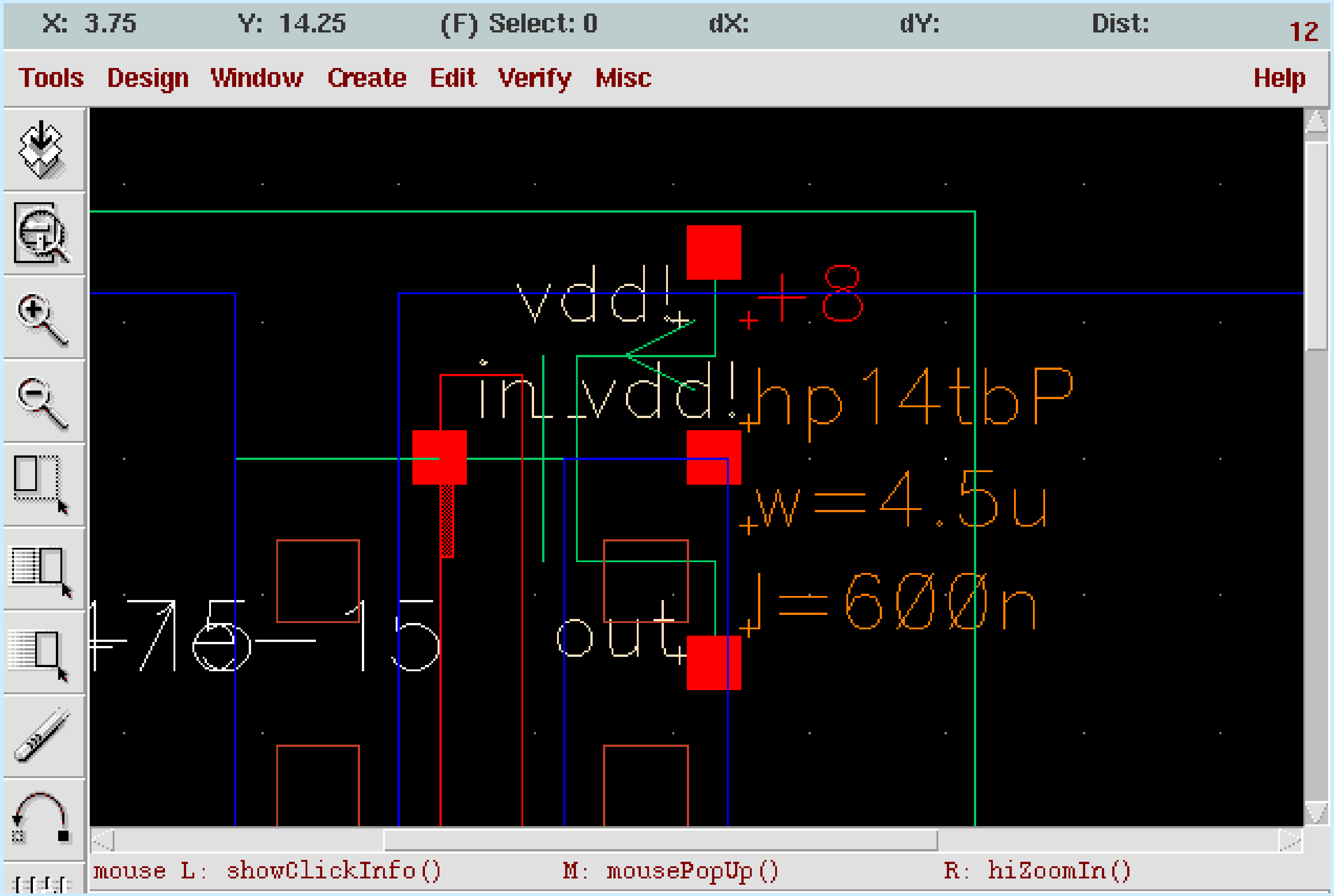
# The Extracted Cell View

The red rectangles indicate that there are a number of instances within this hierarchy.



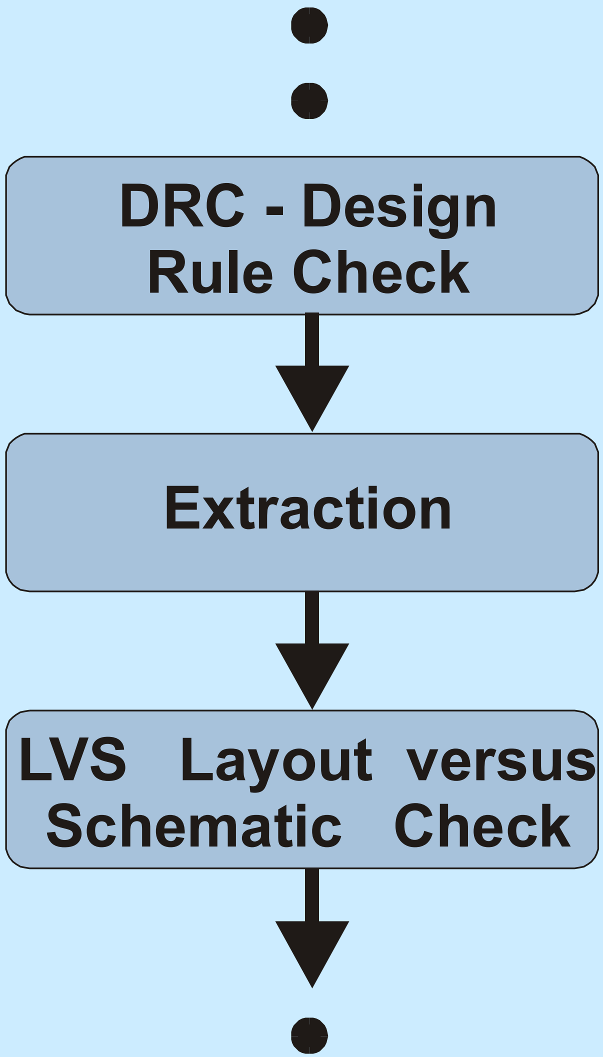

**Press *Shift-F* to see all of the hierarchy.**

# The Extracted Cell View




Notice a number of elements, mainly capacitors.
They are parasitic capacitances.

# Layout versus Schematic Check

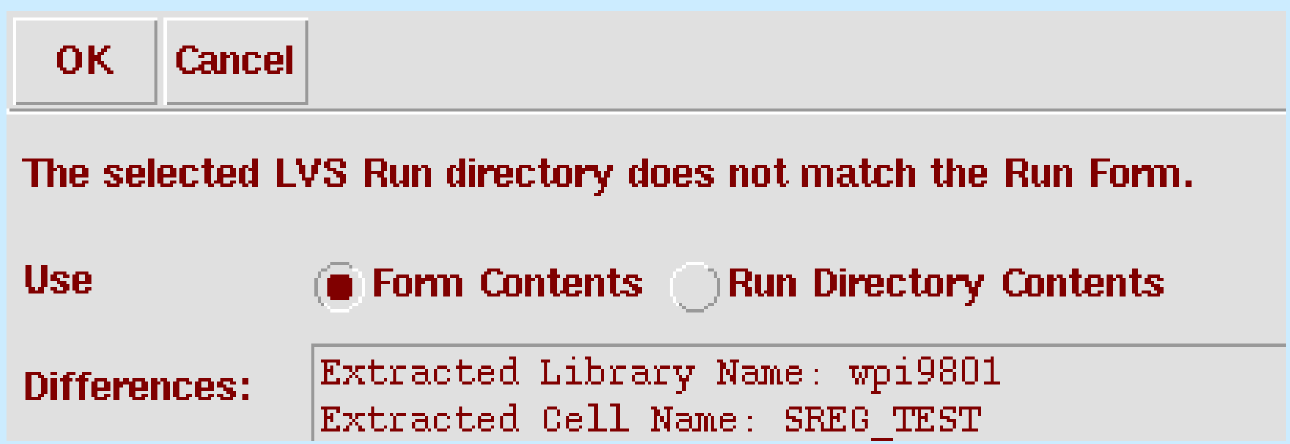



- Compares the original network with the one extracted from the mask layout
- Proves that the two networks are indeed equivalent
- Provides an additional level of confidence for the integrity of the design
- Ensures that the mask layout is a correct realization of the intended circuit topology

# Layout versus Schematic Check

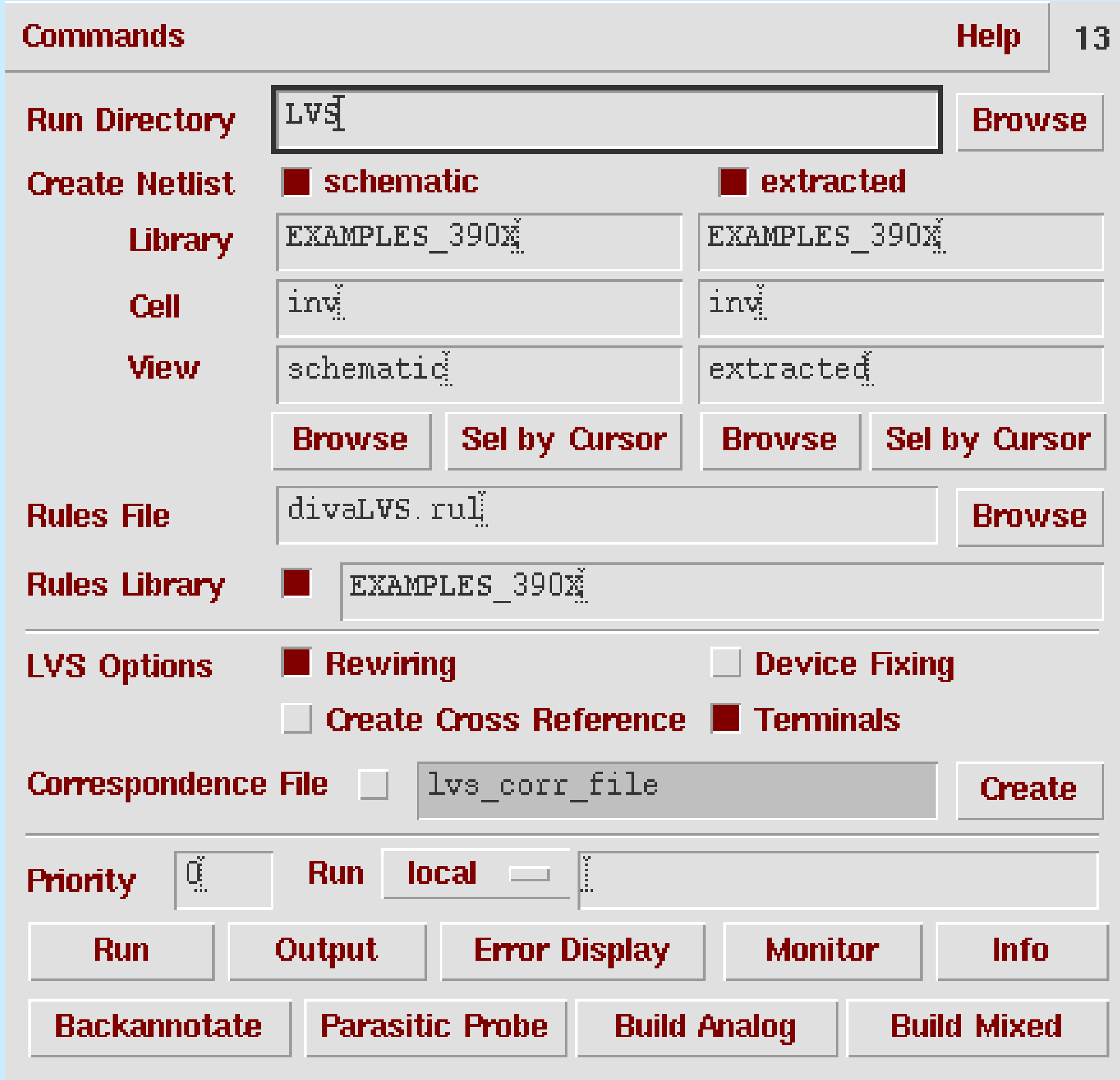
1. From the *Verify* menu select the option *LVS*.


**If you had previously run a LVS check, this would pop-up a small warning box. Make sure that the option *Form Contents* is selected in this box.**



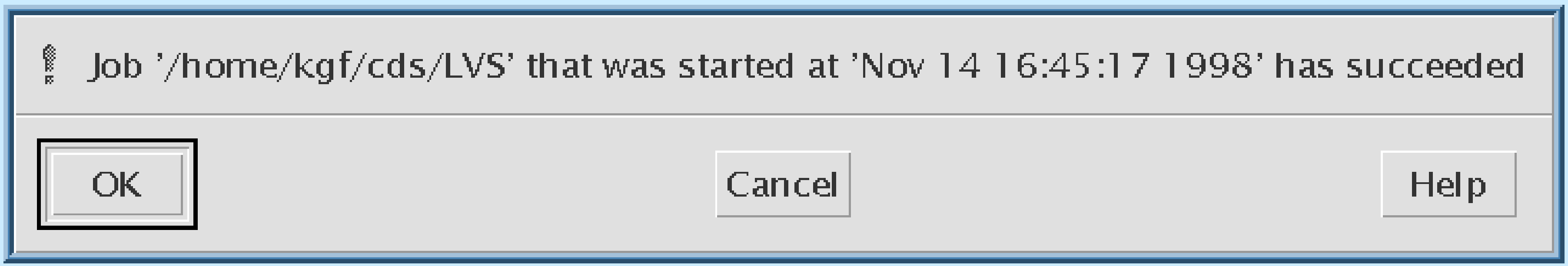
# Layout versus Schematic Check

Although there are a number of options for LVS, the default options will be enough for basic operations, select *Run* to start the comparison.



# Layout versus Schematic Check

Even for a very small design the LVS run can take some time (minutes).




The *succeeded* message box

The above message box, indicates that the LVS program has finished comparing the netlists, **NOT THAT THE CIRCUITS MATCH**. It might be the case that the LVS was successful in comparing the netlists and came up with the result that both circuits were different.

# Layout versus Schematic Check

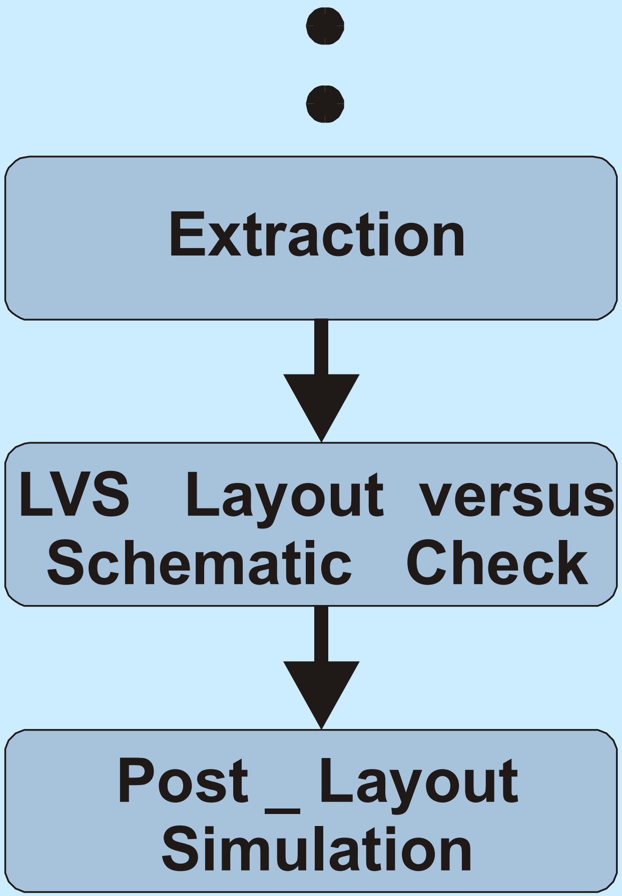
To see the actual result of an LVS run you have to examine the output of the LVS run. The *Output* option is right next to the *Run* command

```
    Terminal correspondence points
          1     gnd!
          2     in
          3     out
          4     vdd!

The net-lists match.

                             layout  schematic
                                instances
        un-matched              0        0
        rewired                 0        0
        size errors             0        0
        pruned                  0        0
        active                  2        2
        total                   2        2
```

# Post-layout Simulation




**Steps of Postlayout Simulation**

- Extracting from the Layout
- The Extracted Cell View
- Layout Versus Schematic
- Summary of the Cell Views
- Simulating the Extracted Cell View

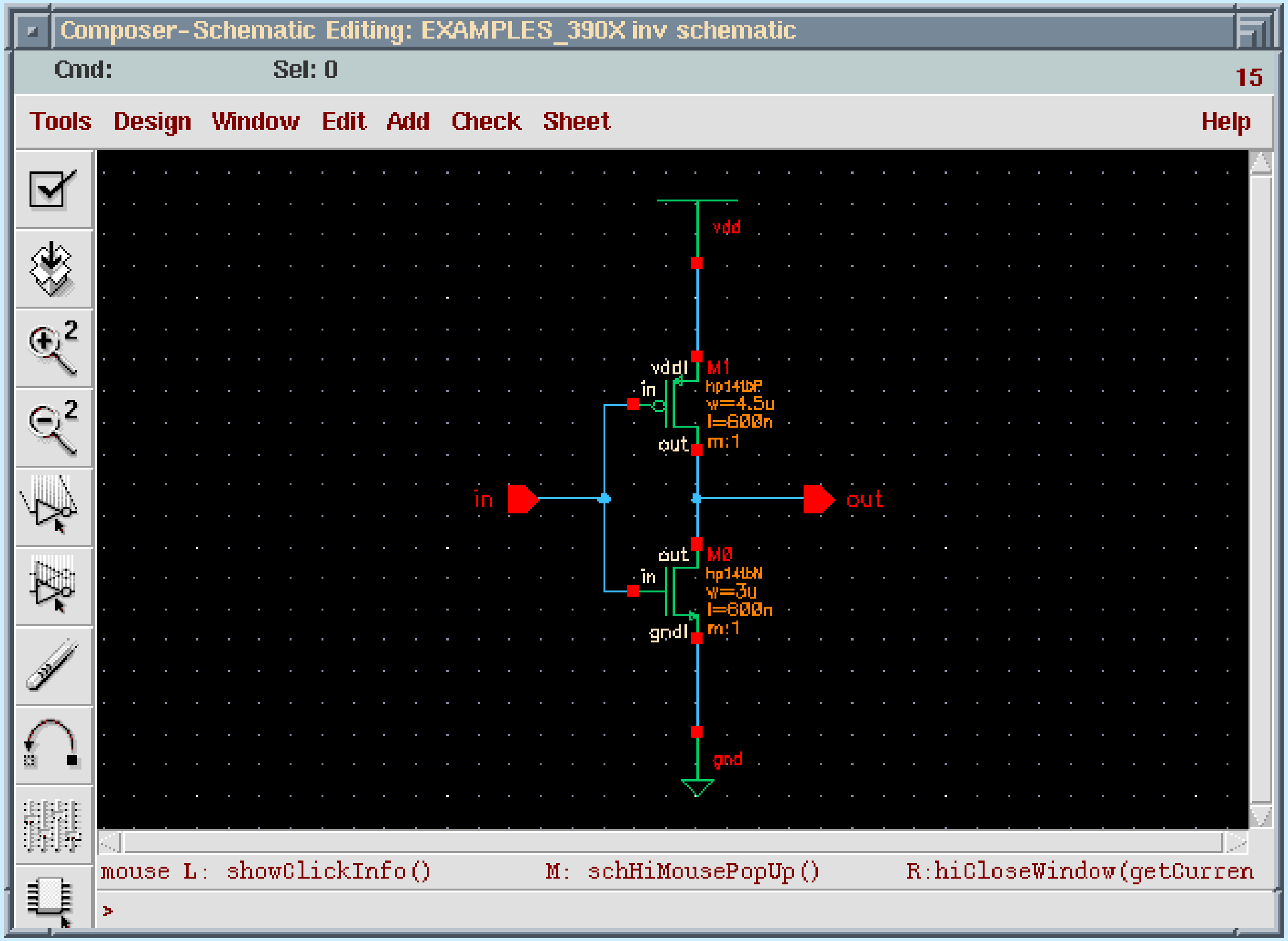
(First three described earlier)

FINISH

# Summary of the Cell Views

1. Schematic view

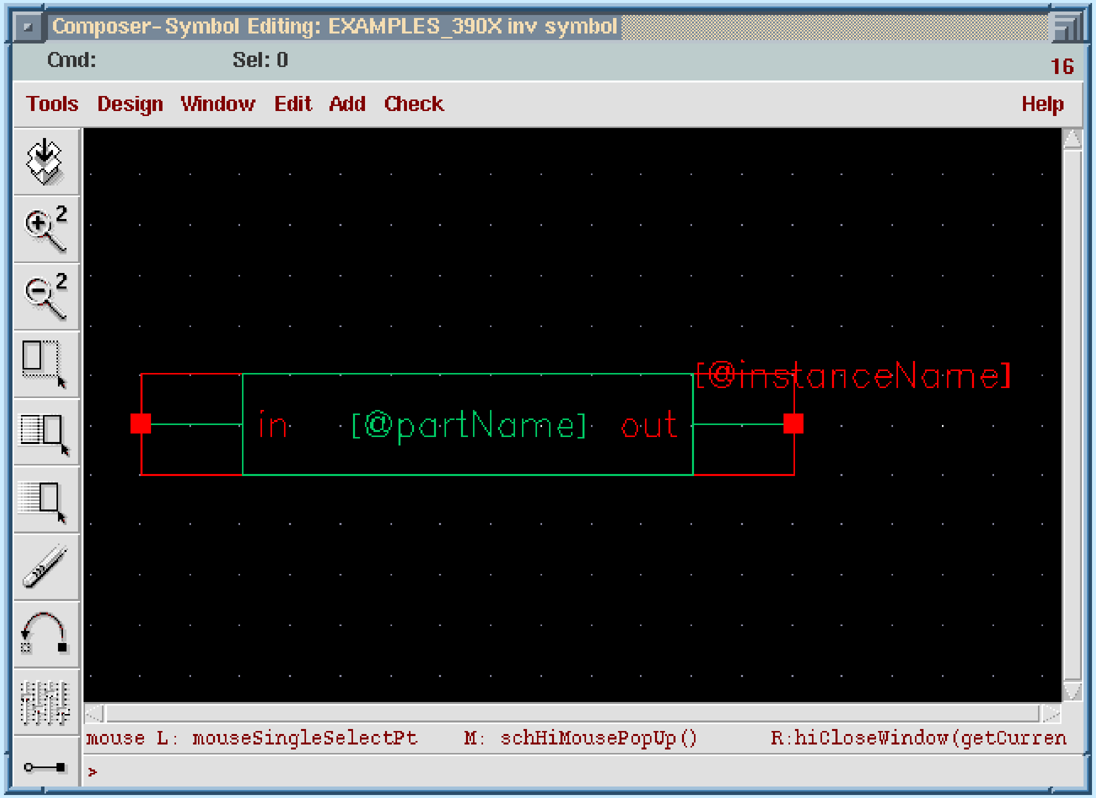
For any design, the schematic should be the first cell view to be created. The schematic will be the basic reference of your circuit.



# Summary of the Cell Views

## 2. Symbol view

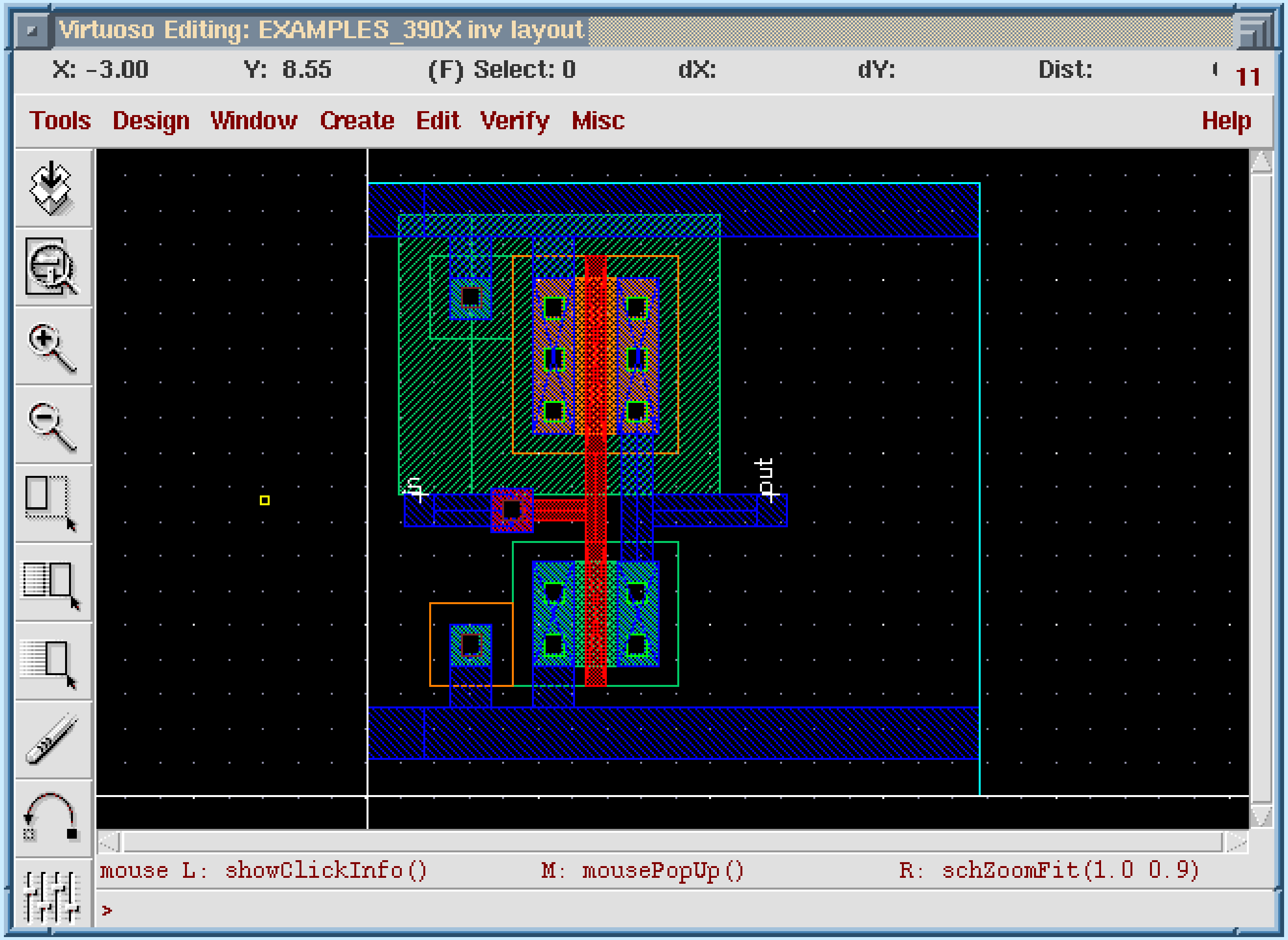
After you are done with the schematic, you will need to simulate your design. The proper way of doing this is to create a seperate test schematic and include your circuit as a block. Therefore you will need to create a symbol.



# Summary of the Cell Views

3. Layout view

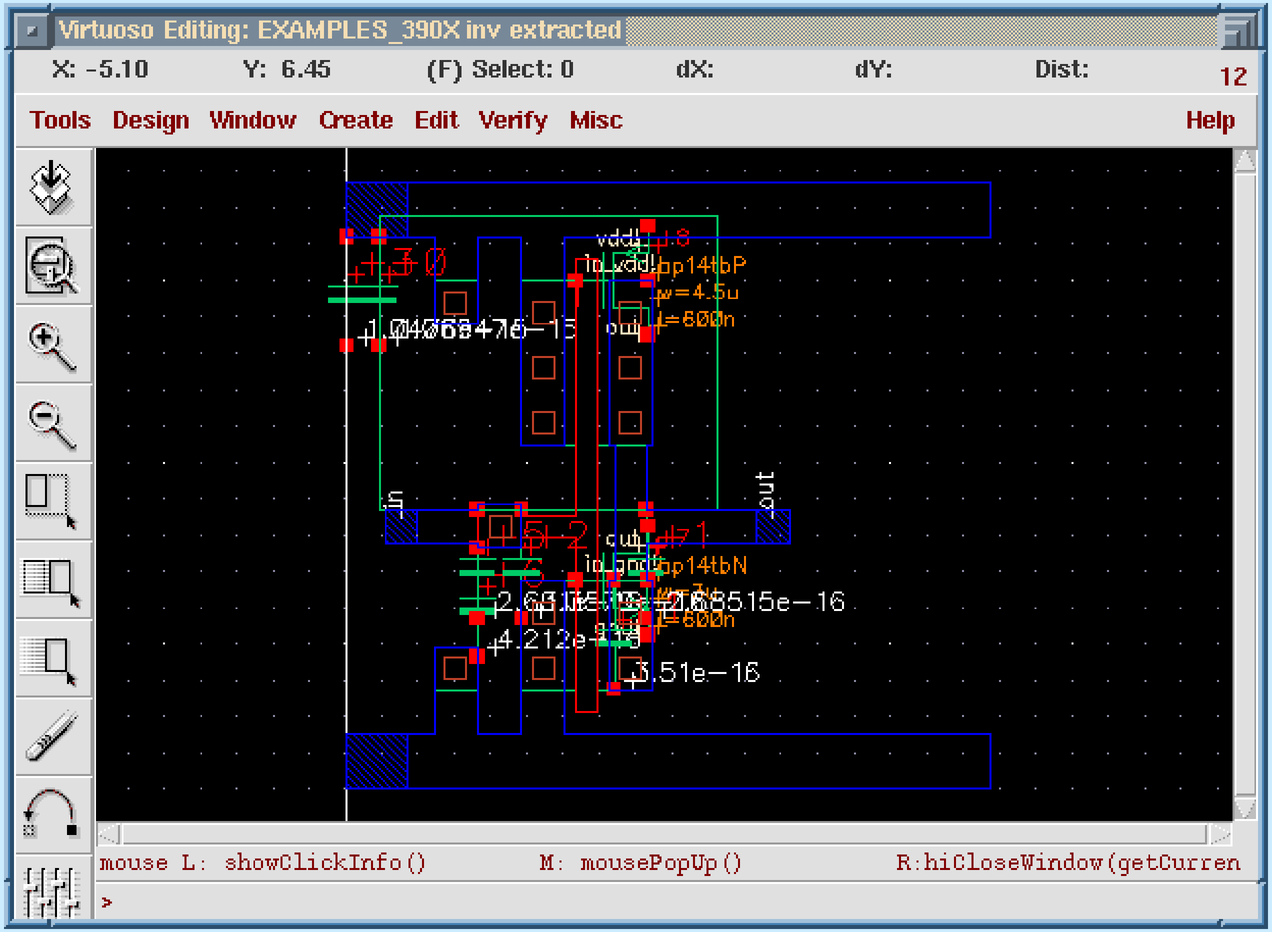
This is the actual layout mask data that will be fabricated.



# Summary of the Cell Views

4. Extracted view

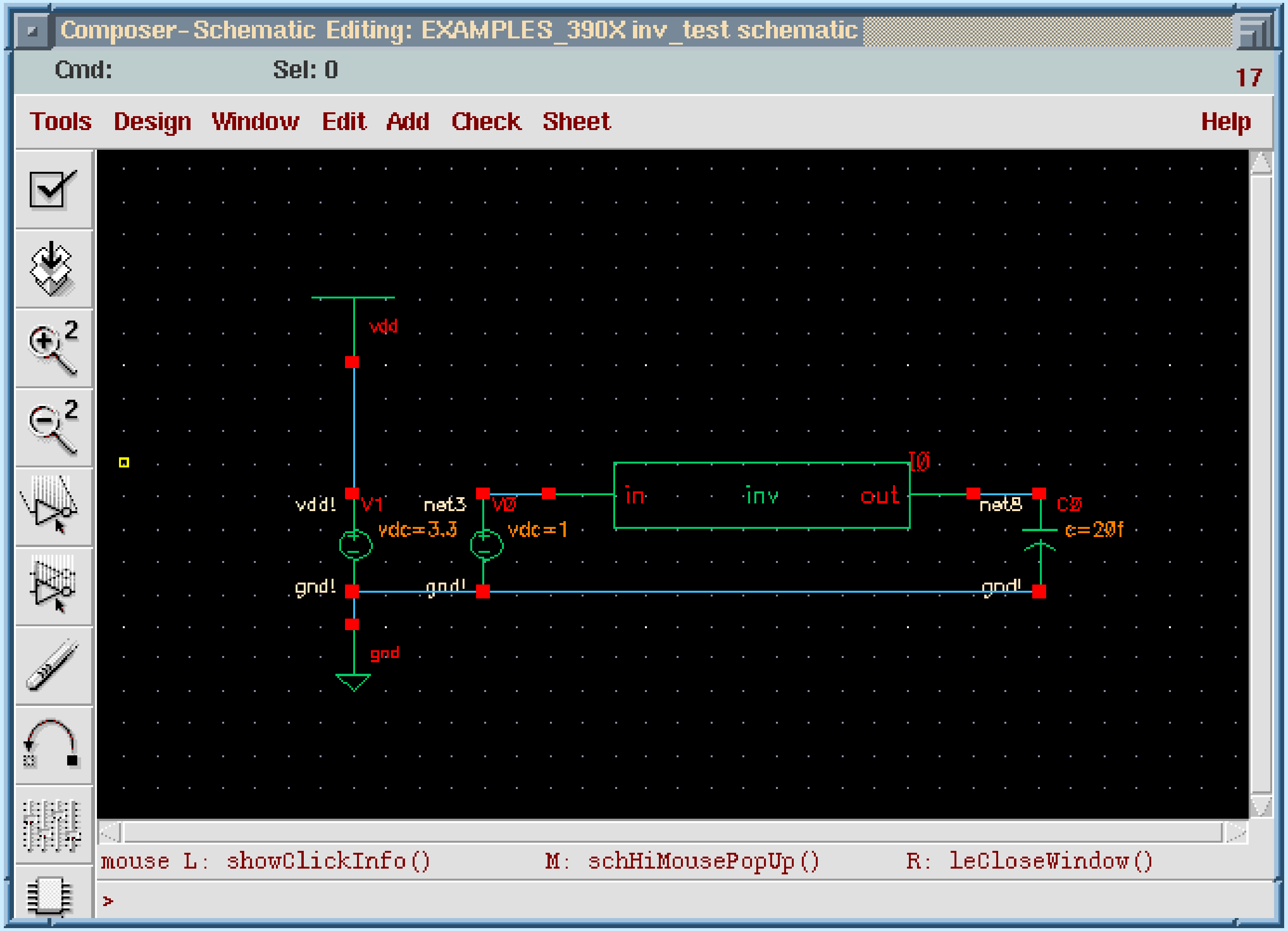
After the layout has been finalized, it is extracted, devices and parasitic elements are identified and a netlist is formed.



# Summary of the Cell Views

## 5. Test Schematic

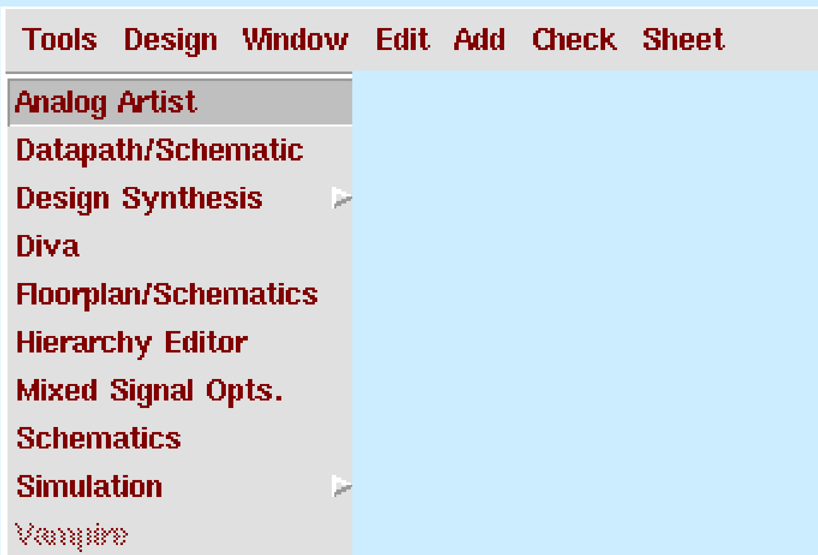
A separate cell is used to as a test bench. This test bench includes sources, loads and the circuit to be tested. The test cell usually consists of a single schematic only.

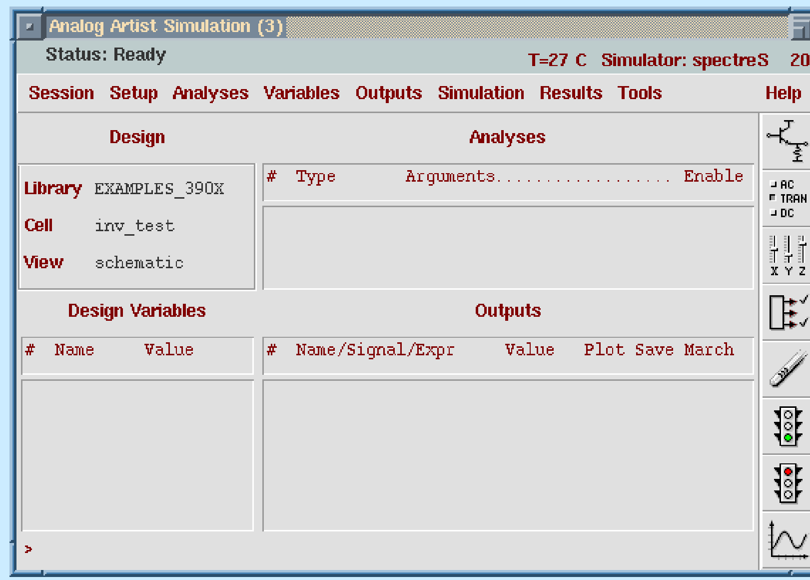


# Simulating the Extracted Cell View

Make sure that you are in the test schematic, that you used to simulate your design earlier.

1. *Start Analog Artist using Tools --> Analog Artist*

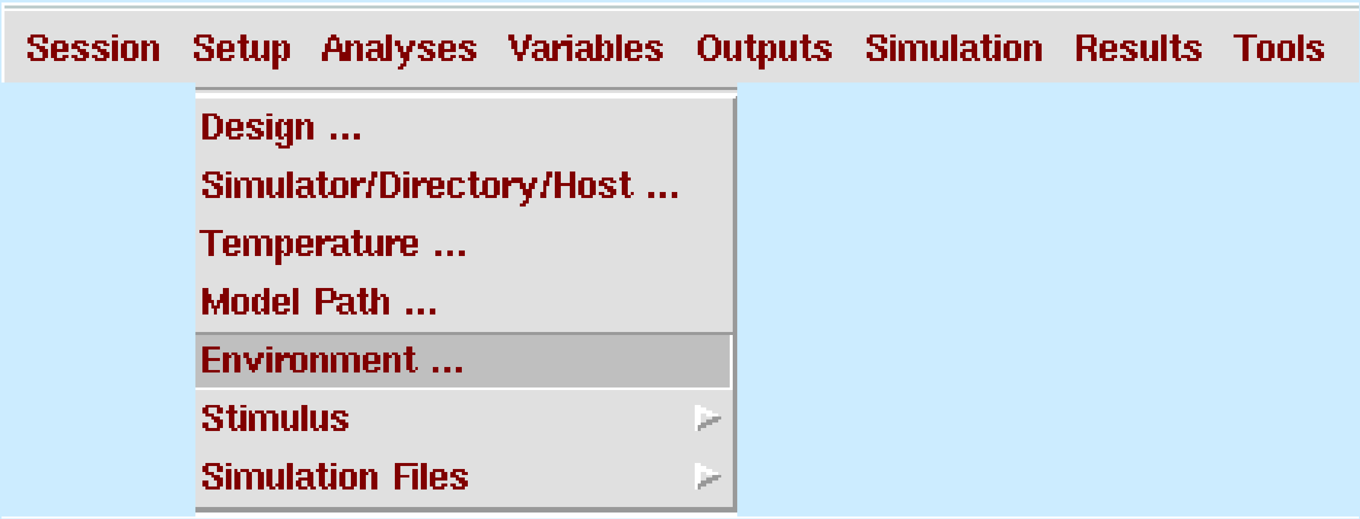



The Analog Artist window will pop-up.

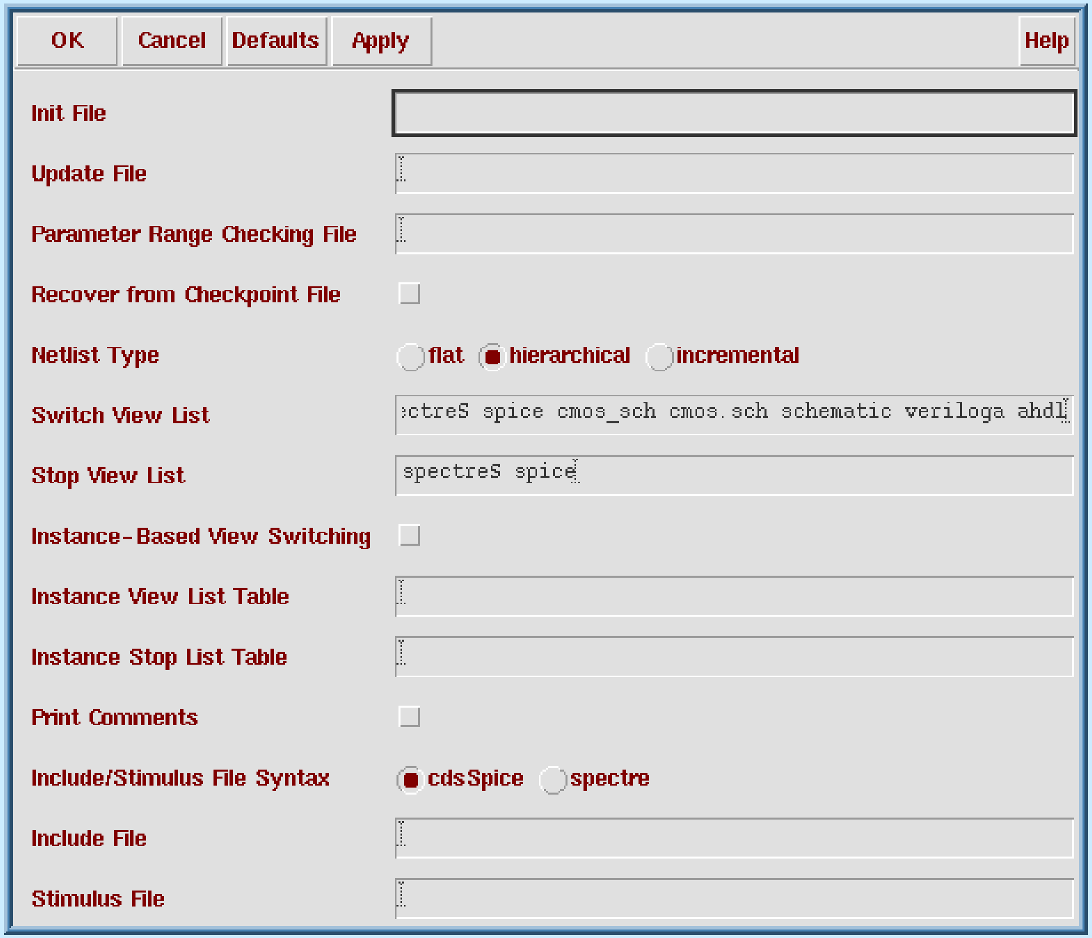
# Simulating the Extracted Cell View

2. From the *Setup* menu choose the *Environment* option.




**A new dialog box controlling various parameters of Analog Artist will pop-up...**

# Simulating the Extracted Cell View

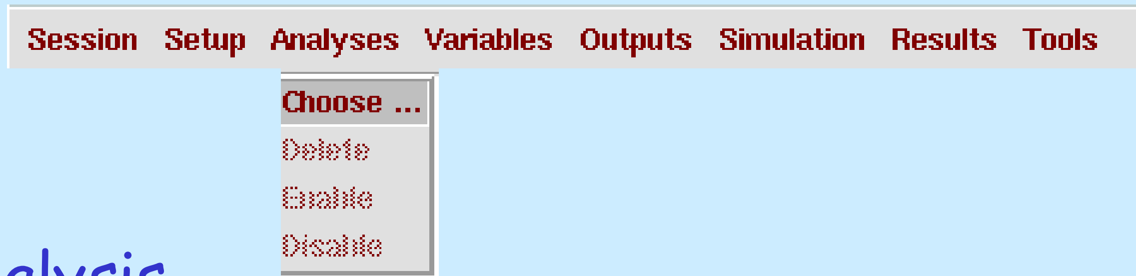



Alter the line called the *Switch View List*.

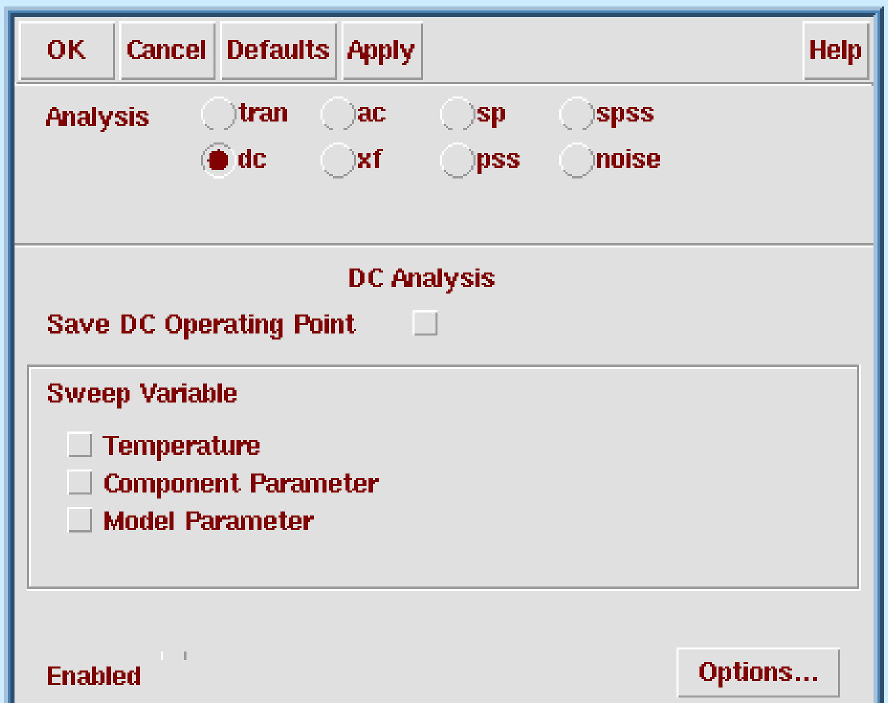
This entry is an ordered list of cell views that contain information that can be simulated.

# Simulating the Extracted Cell View

3. Choose analyses



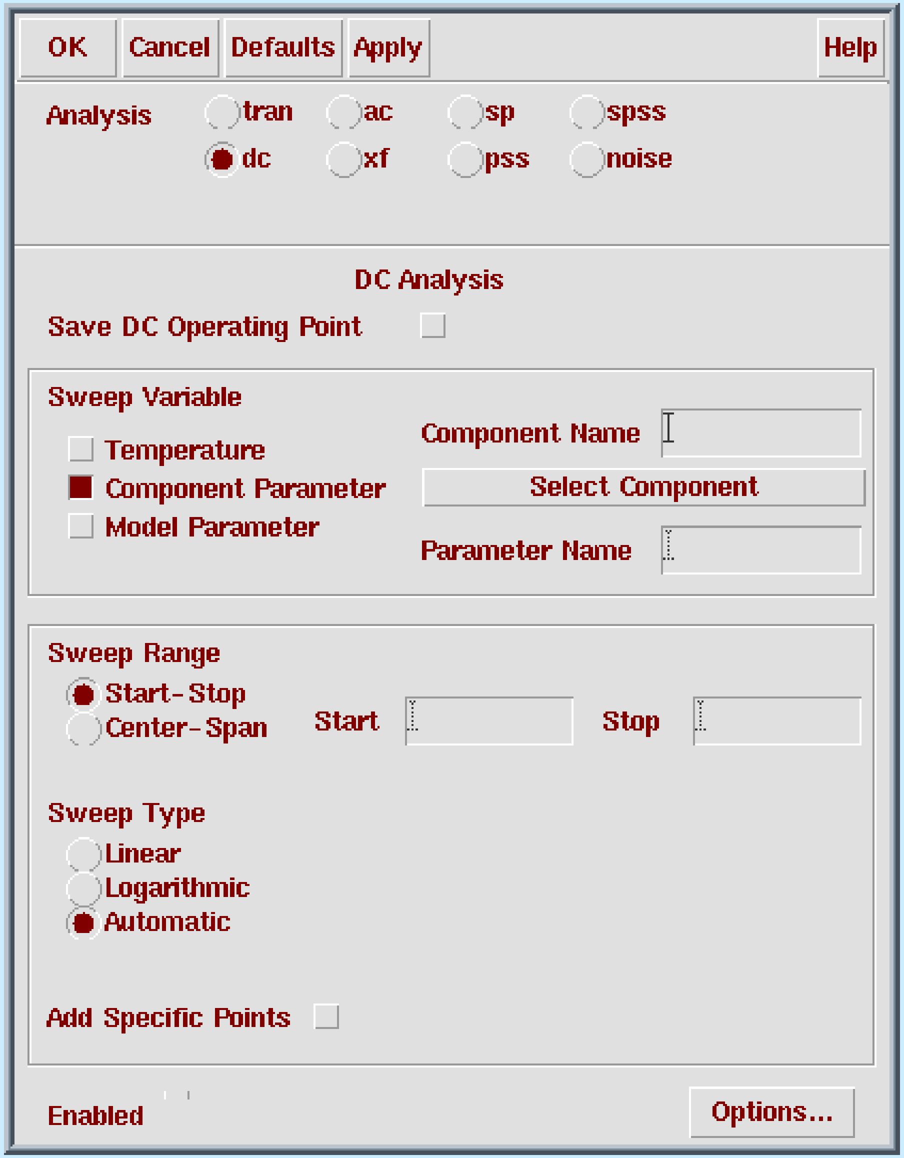

For example DC analysis




The first step is to determine what parameter will be swept.

# Simulating the Extracted Cell View

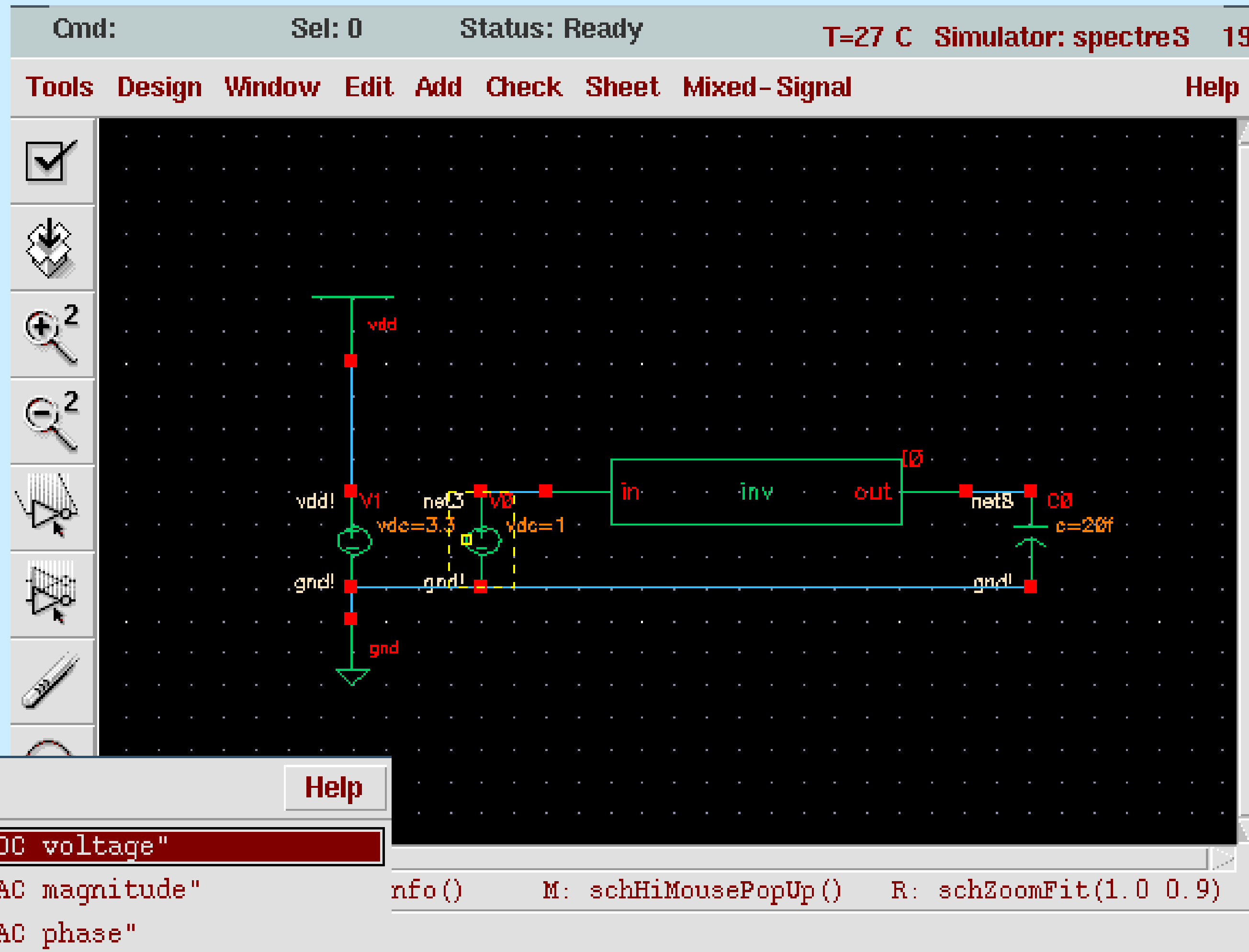
Choose *Component Parameter* as the Sweep Variable.

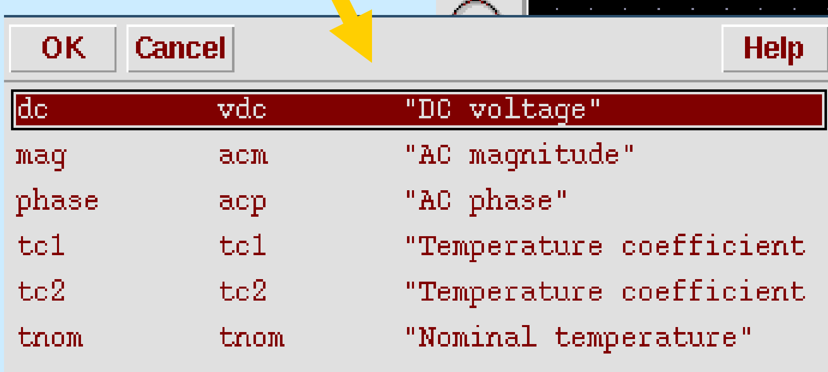



**You can select the parameter from the schematic window after you click on *Select Component*...**

# Simulating the Extracted Cell View

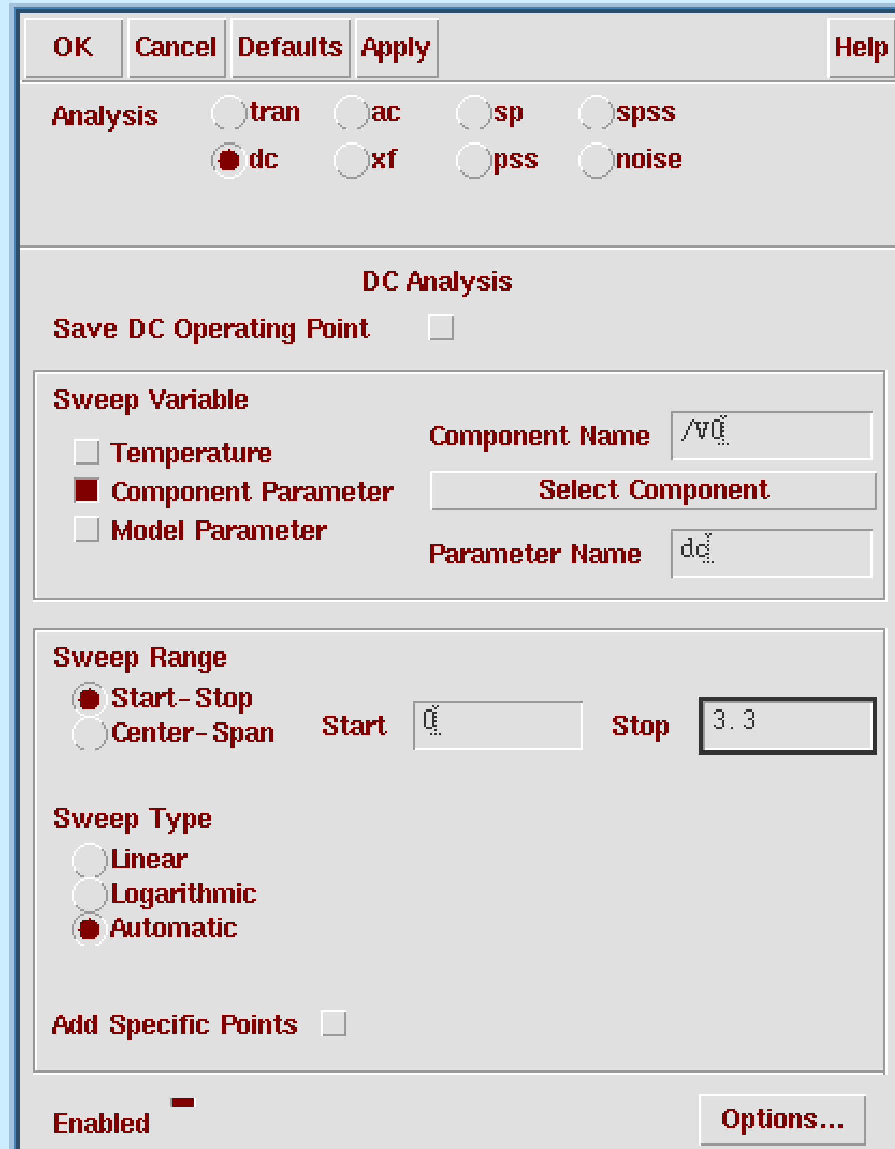
As each component has a number of parameters, you will be given a list of parameters associated with the component you select.



# Simulating the Extracted Cell View

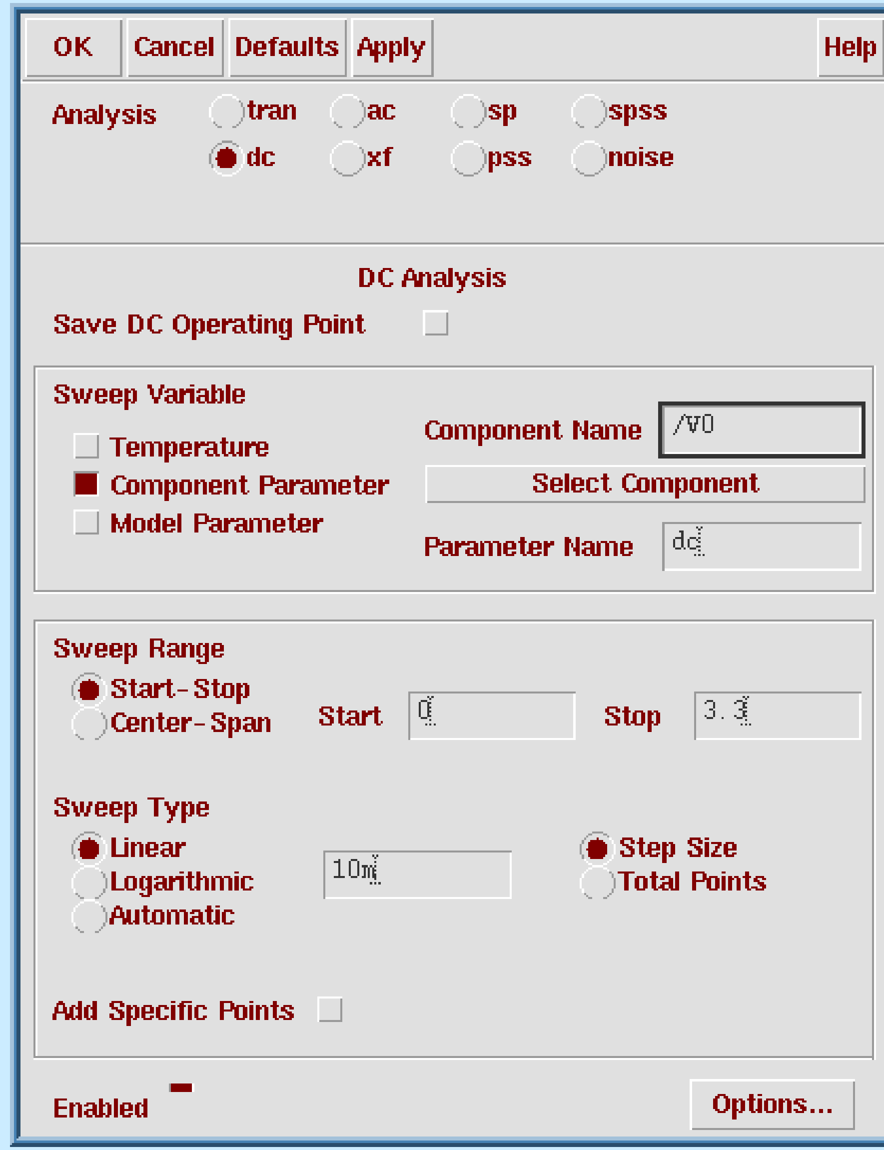
After we have selected the variable we can decide, the range where the variable will change.




This example changes the DC voltage source connected to the input from 0 Volts to 3.3 Volts.

# Simulating the Extracted Cell View

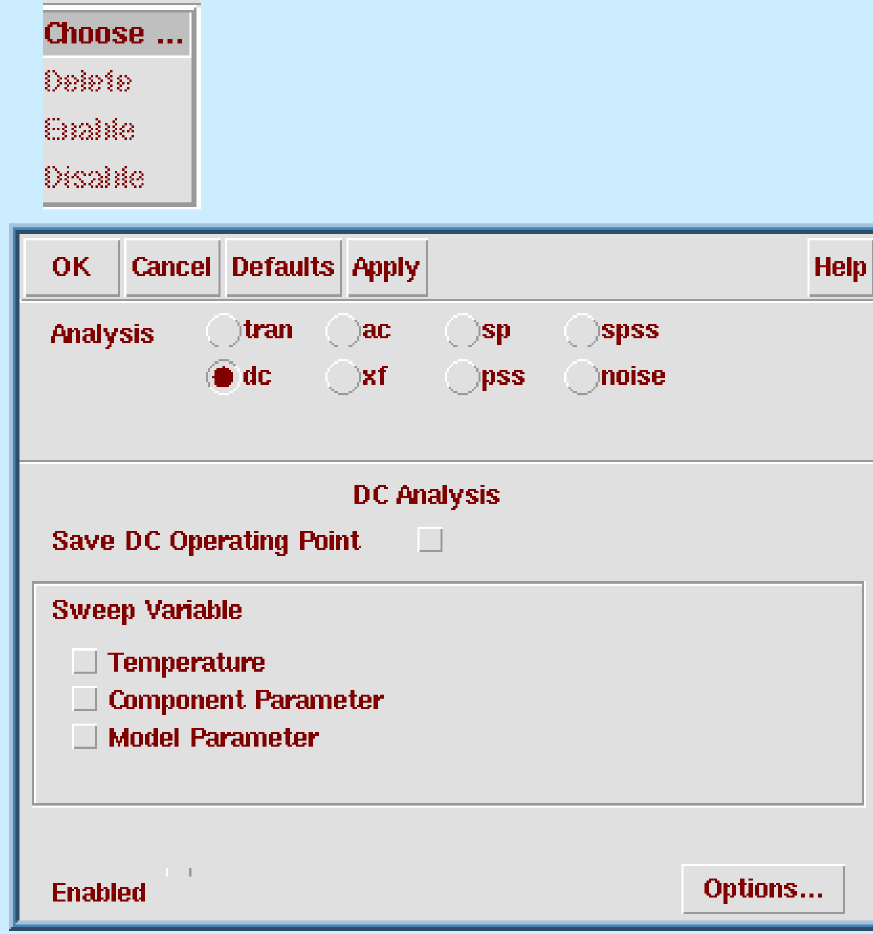
The last parameter determines how the sweep will be performed. A linear sweep will increment the value of the sweep variable by a fixed amount.




**The example uses a step size of 10 millivolts.**

# Simulating the Extracted Cell View

Choose another analyses


For example **tran**, **noise ect.** till you will be satisfied of your circuit.

**End of Design**
**SEND to FOUNDRY**