Escuela Argentina de Microelectrónica 2006

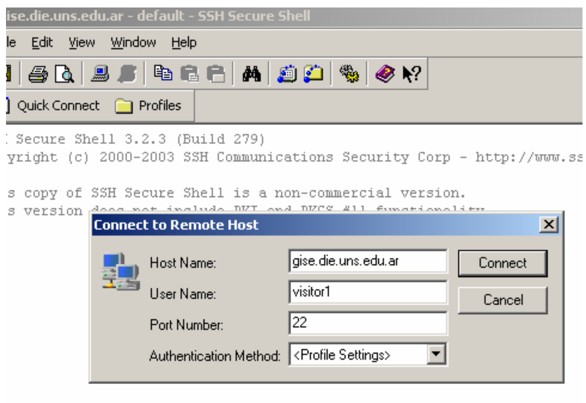
Tutor sobre

Mentor Graphics IC Design

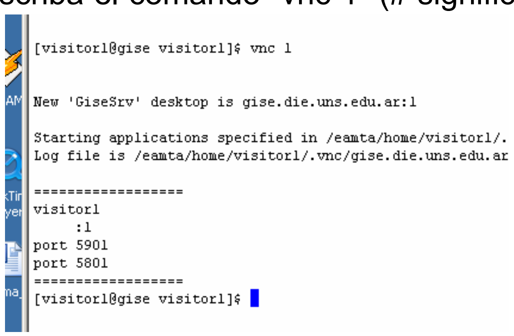
Introducción

Este tutor está basado en uno provisto por Mentor Graphics para una biblioteca de 0.5 micrones. Ha sido brevemente modificado para adaptarse a las necesidades de esta Escuela.

Step 1

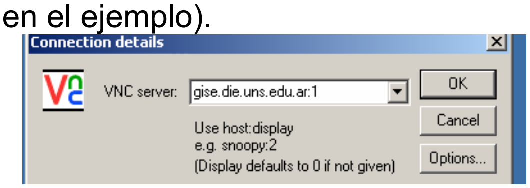


Ejecute el programa SSH Secure Shell (en el escritorio). Haga clic en Quick Connect. Conéctese al servidor gise.die.edu.ar utilizando el login *visitorX*, contraseña *eamta06*. La X debe sustituirse por el número asignado a su comisión.
En la línea de comando que aparece, escriba el comando "vnc 1" (# significa el *prompt*), como se muestra en la figura 2 (el 1 define un tipo de resolución gráfica). Ingrese una contraseña, puede ser la misma de su *login* (eamta06).



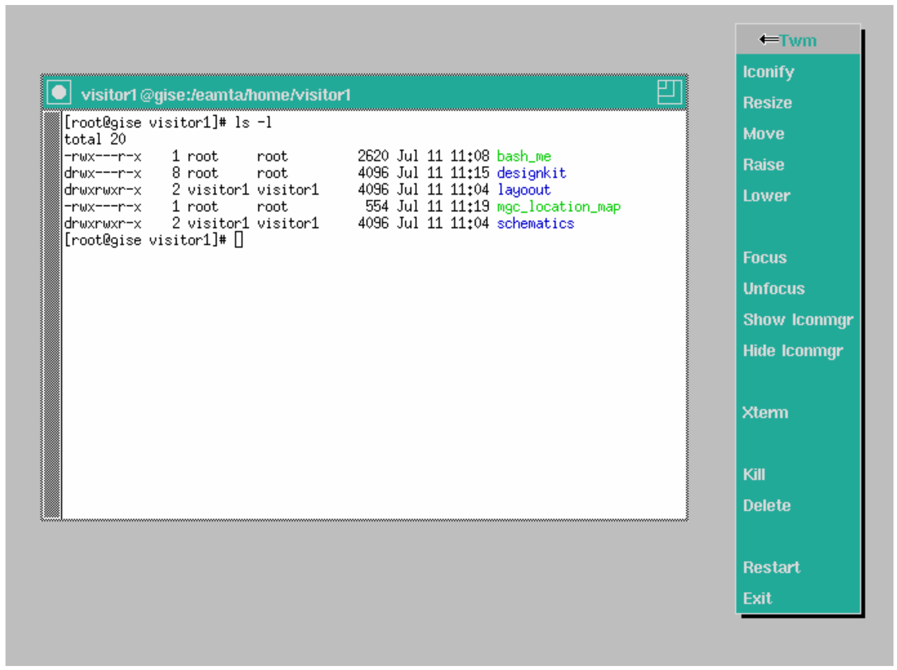
Recuerde el último número (N) que entrega la salida del comando (":1" en la figura de ejemplo). Una vez hecho este paso, puede cerrar si lo desea la sesión SSH (teclee # *exit*).

Step 2. Invoque el vncviewer en el escritorio y conéctese al servidor gise.die.uns.edu.ar:N (gise.die.uns.edu.ar:1 en el ejemplo).



Obtendrá una terminal gráfica básica Twm de Linux. Esta terminal no es la mejor que ofrece Linux, pero es muy liviana para el servidor.

Step 3. Verifique que posee los siguientes directorios (#ls –l) : *designkit, schematic, layout*, y el archivo *mgc_location_map* (siga el ejemplo de la figura adjunta). Nota: si se llega a cerrar por error la ventana terminal, sostener el botón izquierdo provee un menú para obtener nuevas terminales (XTerm). Cuando invoque una aplicación, añada un & al comando, para liberar la terminal. El cuadro que aparece indica dónde desea colocar su ventana. Haga clic donde le plazca.



Otros comandos útiles:
cd Cambia directorio. Ejemplo: #cd ./designkit
cp Copia archivos o directorios (opción –R)
rm Remueve archivos (cuidado, en Linux remover no puede revertirse)
Ctrl-c Detiene una ejecución
Ctrl-z Envía una aplicación al fondo (para recuperar #fg N, N número de aplicación. Si se desea dejar funcionando en el fondo #bg N. #jobs da los números de las aplicaciones hijas de la terminal actual)
nedit Invoca a un editor muy versátil, con atajos similares a los de Word, con revisión de sintaxis para varios lenguajes de programación, scripting o HDL (bash, pearl, verilog, C, etc.)

A continuación se detalla el contenido de alguno de los directorios del Design Kit usado en este tutor.

process - This directory contains the design kit's process definition file as well as where the foundry Calibre DRC and LVS command files are stored.

rule_deck - This directory contains foundry Calibre DRC and LVS rule files.

schematics - This directory can store all the schematic files.

sim - This directory can be used to store all the simulation data.

symbols - This directory contains all the symbols used in creating schematics and setting up simulations.

userware - This directory contains the da_ic and ic station userware including device name mapping and palette menus.

.

mgc_location_map - Provides the location of the design

Step 4 Check your variables. Type "echo $MGC_LOCATION_MAP". It should point

to your design directory. Repeat for variables MGC_HOME, MGC_DESIGN_KIT,

and AMPLE_PATH. Si no le parecen correctas, comuníquelo de inmediato

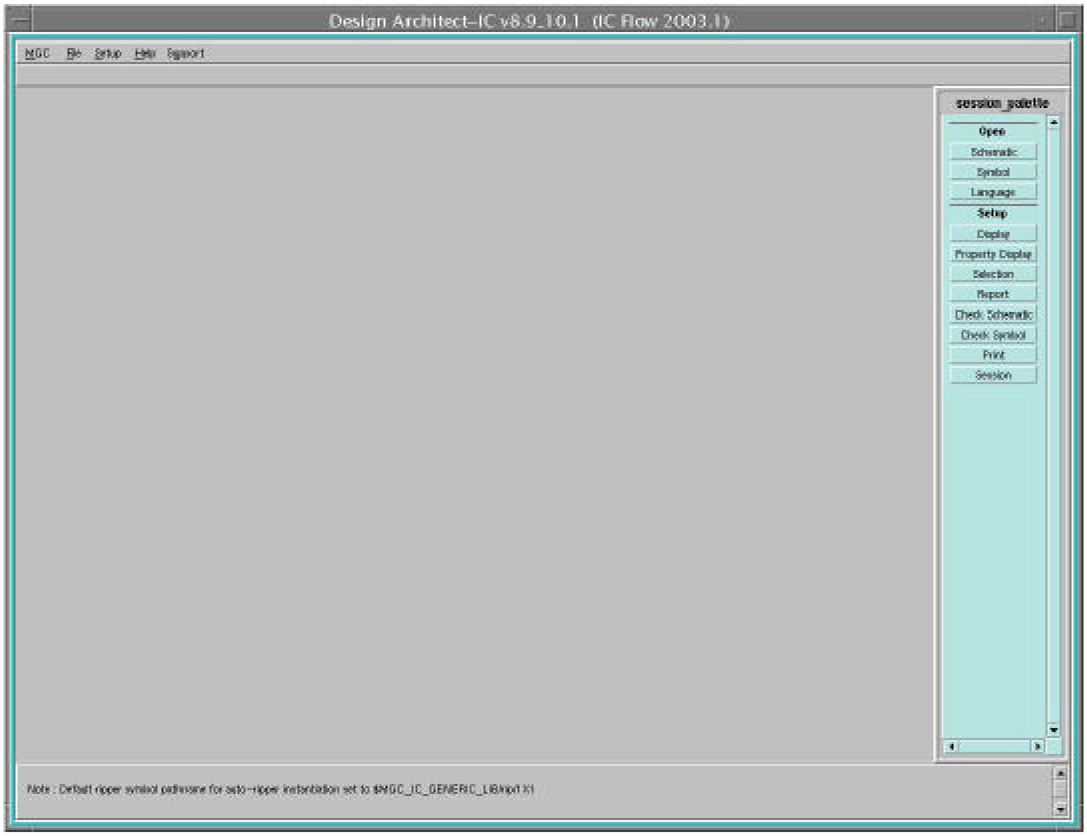
Step 5 To invoke DA-IC do the following:

Open Design Architect-IC by typing the command "**da_ic&".**

The da_ic graphic interface will be invoked and appear on the screen.

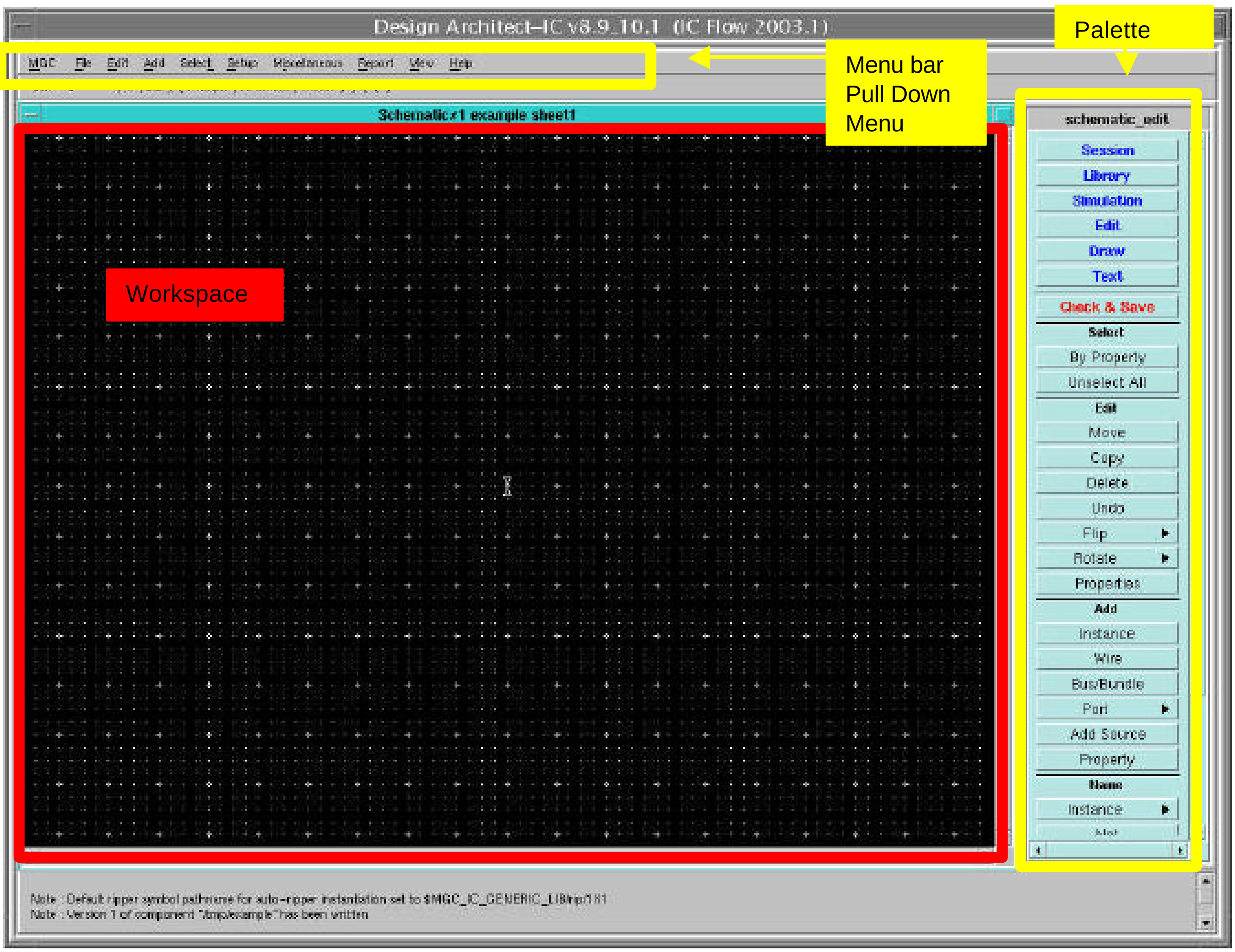
Step 6 To open or create a schematic, click on the “open > schematic” in the session_pallete.

A blank schematic sheet will appear with the name you gave it at the top.



Nota: Si no aparece el menú de la izquierda, vaya a MGC>Setup y marque Show Palette

Introduction to the Design Architect-IC Graphic Interface



The graphic interface is divided into three main sections – a menu bar, a workspace, and a palette.

The menu bar and palette change to reflect the design task you select – laying out a schematic, setting up a testbench, or simulating a design. [As you are working with your design you will notice that the gray bar below the menu (the Info bar) contains information that changes dynamically as you go about your tasks.] The workspace provides a snap grid for manipulating primitive-level components or higher level symbols.

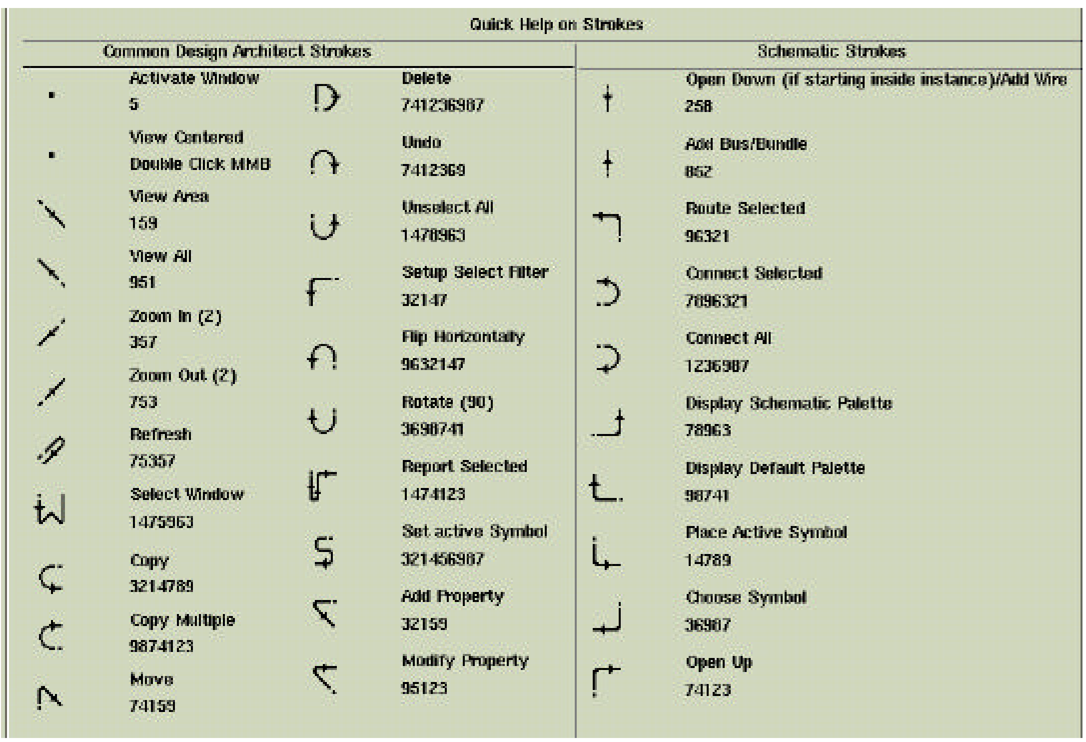
Function key shortcuts

Function key shortcuts provide quick one- or two-button access to common tasks like Copy, Paste, Select All, Unselect All. The function keys work in conjunction with the Shift, Control and Alt keys to provide up to four separate actions per key.

To display function key shortcuts select *MGC > Setup > Session* and select *Show Softkey Area*.

"Strokes with middle mouse button"

Design Architect–IC provides a number of shortcuts to common actions by utilizing the middle mouse button. While holding down the middle mouse button you can make simple, alphabet-like strokes in the workspace to quickly copy, move, zoom in or out, view an area or view all, delete or undo, unselect all and flip horizontally... to name just a few. The **Quick Help on Strokes** dialog box is available from the Help menu.



In the lessons that follow, actions that can be simplified with strokes will be presented with the appropriate stroke in the left margin, as shown here, or within the paragraph.

Creating a Schematic

In this lesson you will become familiar with placing primitive analog devices for a inverter. You'll learn how to:

- open a schematic workspace
- place primitives on the schematic
- select and manipulate devices
- customizing hotkeys for placing devices
- route devices
- edit device parameter values
- name instances
- check and save the schematic

Primitive devices are selected from the ***design kit*** symbol library shipped with Design Architect-IC. If problems are encountered placing the parts, make sure the entries $MGC_HOME and $MGC_DESIGN_KIT have been properly set to valid paths in the location map and the location map is loaded. To see the current location in the session, go to ***MGC > Location Map > Show Location Map*** (in the menu bar).

In later lessons, we will simulate this inverter using Eldo and view the results.

A completed inverter schematic and testbench can be found in $MGC_DESIGN_KIT/schematics/inv_complete and testbench_complete

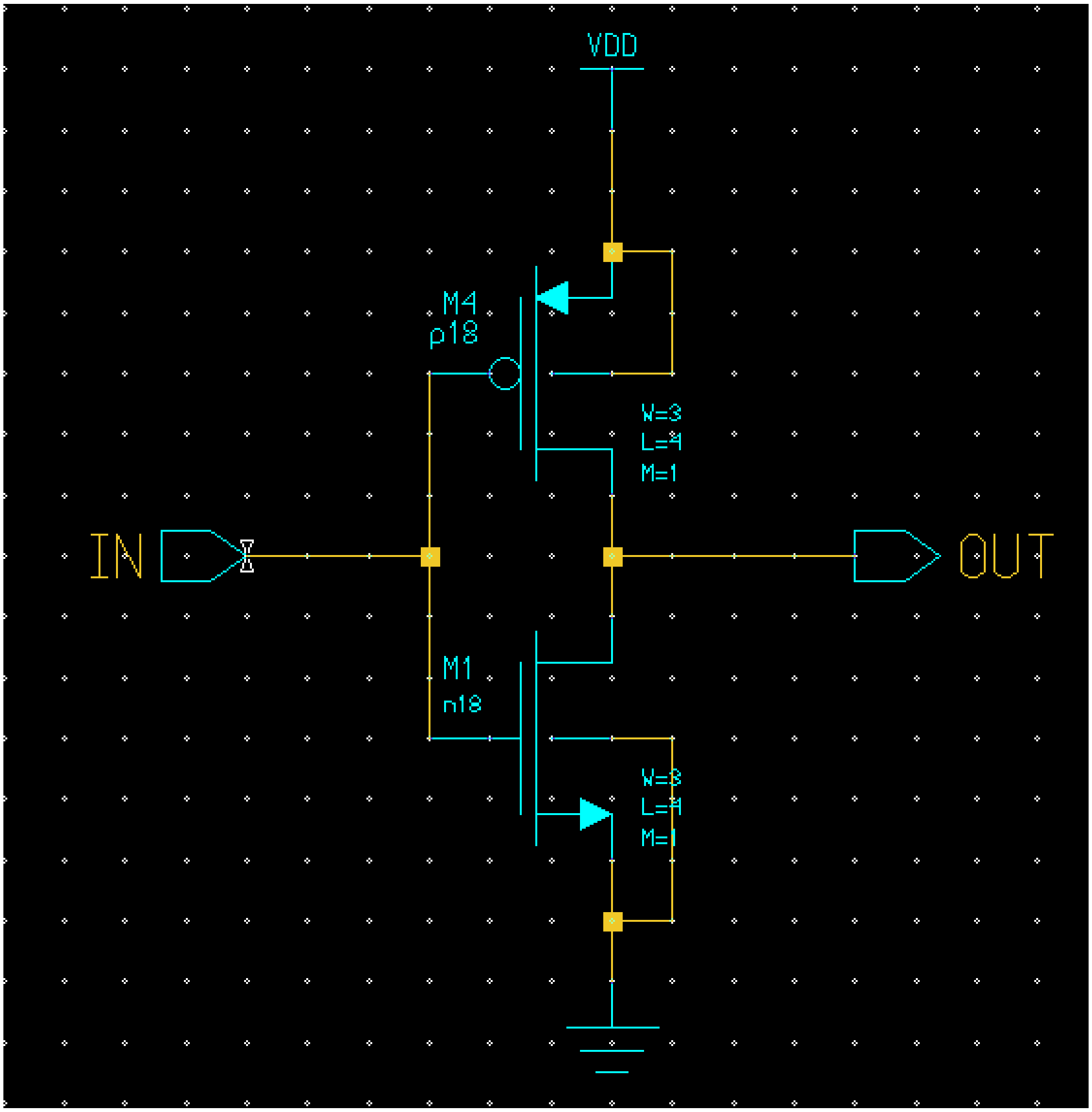
Setup

Step 1 Open Design Architect-IC : **da_ic&**

Step 2 Expand the Design Architect-IC session window to the maximum fit

Editing the Inverter Schematic

The first step in preparing the design for simulation is to complete the schematic shown in the figure below.

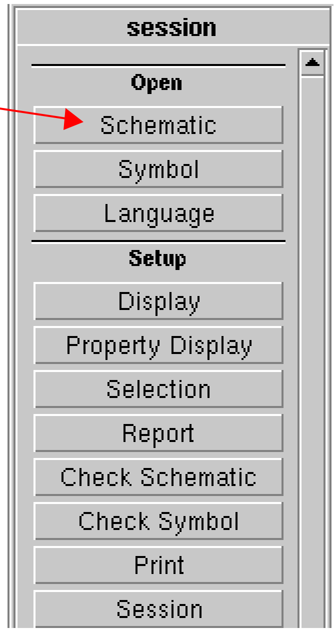


This schematic is a simple inverter circuit created from the components of the design kit. In later lessons, we will simulate this inverter using Eldo and view the results in the waveform viewer EZWave. In this lesson you will become familiar with placing primitive components from the 0.5 mixed-mode design kit symbol library. We will place some primitive components that make up this circuit using the hot keys feature, wire them up, change some of the default parameters for the devices to the values required, then check and save the schematic.

Open empty schematic to build the inverter

The first step is to open a window that will contain the inverter schematic.

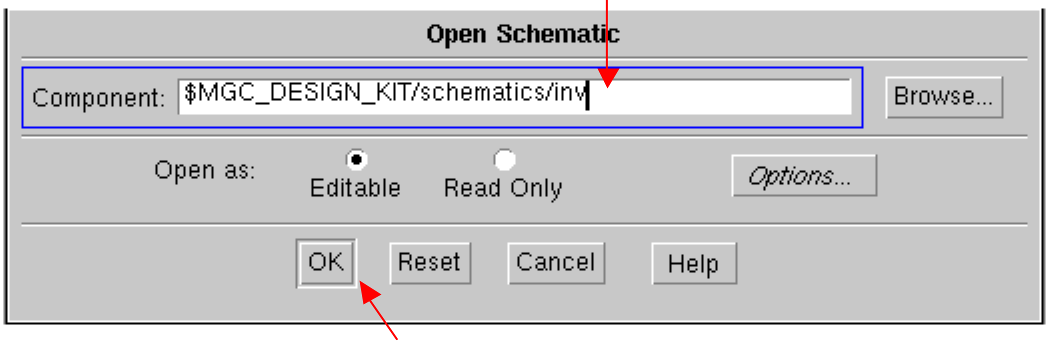
Step 1 Select *Open->Schematic* with left mouse button



This will pop up a Open Schematic form

Step 2 Type in the following for the Component Name:

$MGC_DESIGN_KIT/schematics/inv

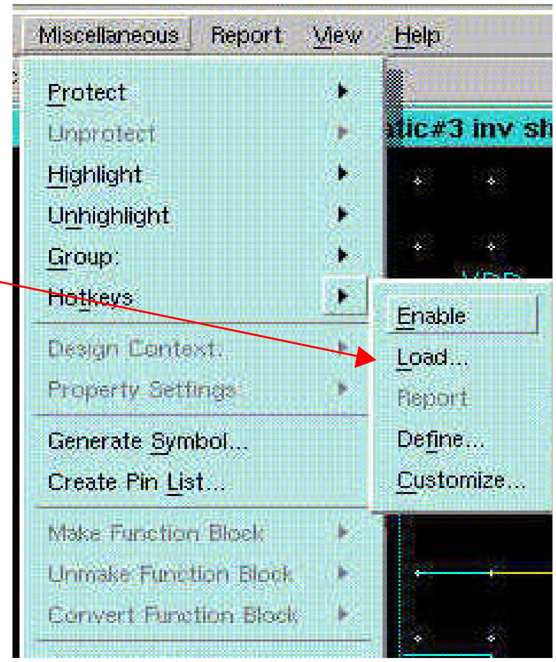


Step 3 OK the form with the left mouse button

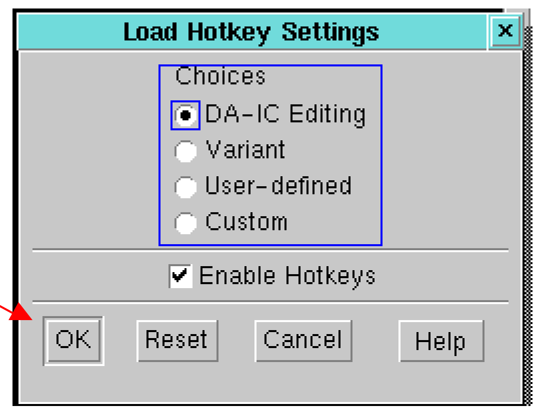
The inverter schematic window is opened. Design Architect-IC is a multiple window environment, meaning multiple schematics and symbols can be edited simultaneously in the same Design Architect-IC session. We will now begin editing this schematic using commands found on the palette menu (the static menu on the right side of the application) and commands that have been mapped to keyboard keys (hot keys).

Load Hot Keys

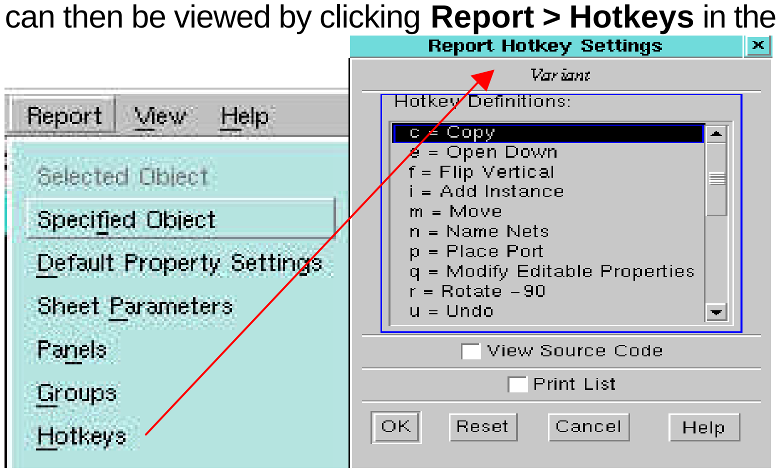
Step 1 Enable the hot keys. Select **Miscellaneous > Hotkeys >Load** from the pulldown menu



Step 2 Select **DA-IC Editing and Enable Hotkeys** and OK the form



Step 3 The hotkeys can then be viewed by clicking **Report > Hotkeys** in the pulldown menu



Two keys that are useful when manipulating components are the ***f*** and ***r*** keys. The ***f*** key will flip the selected component horizontally. The ***r*** key will rotate it.

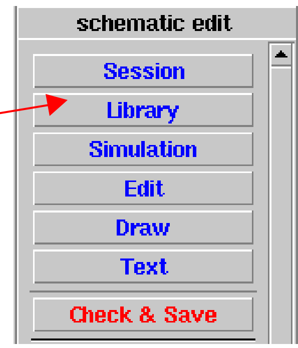
In the following lessons, actions that can be simplified with hotkeys will be presented with the appropriate hotkey in the left margin, as shown here, or within the paragraph.

Creating the Inverter with the 0.5 mixed-mode Library

Now you will build up the inverter schematic by completing the following steps.

Place Primitives on the schematic

Step 1 Click on the **Library** button on the schematic palette menu (the static menu on the right hand side) using the left mouse button



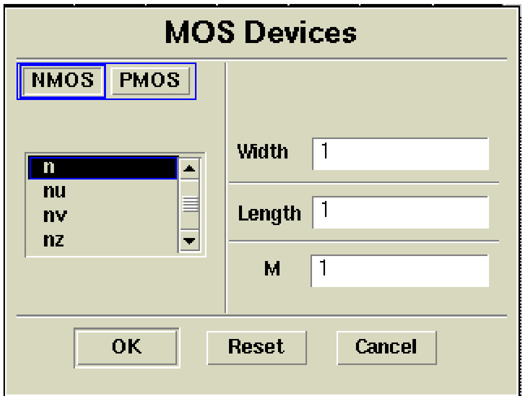
The palette menu is split into parts that have been qualified and have a corresponding layout for the 0.5 mixed-mode process

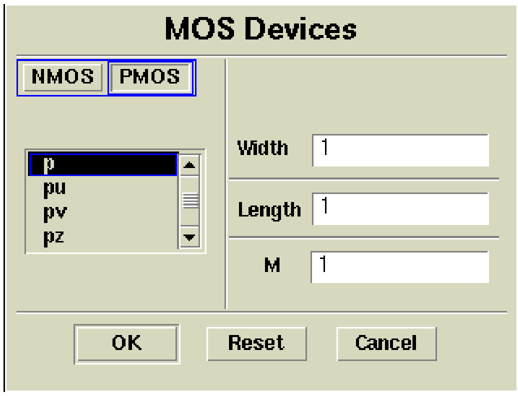
You can see the mixed-mode components and basic components

Basic consist of ideal parts, generic parts such as port and off-page symbols, and voltage/current sources to create test benches

Step 2 Select Mosfets with left mouse button

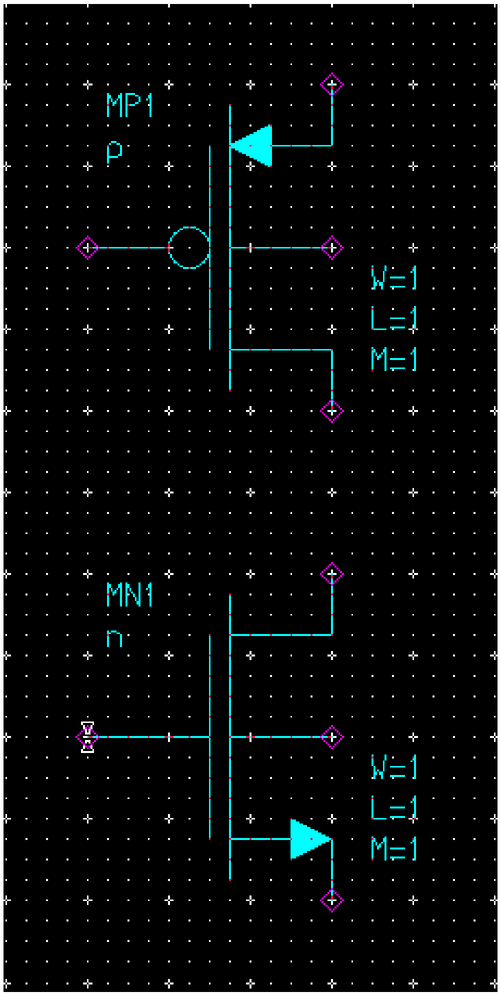
Step 3 You can choose PMOS, with width equal to 1, length equal to 1 and M eqaul to 1, and click OK, and do the same for NMOS.





A ghost image of the transistor should be tied to the cursor. Place the mos transistors on the schematic

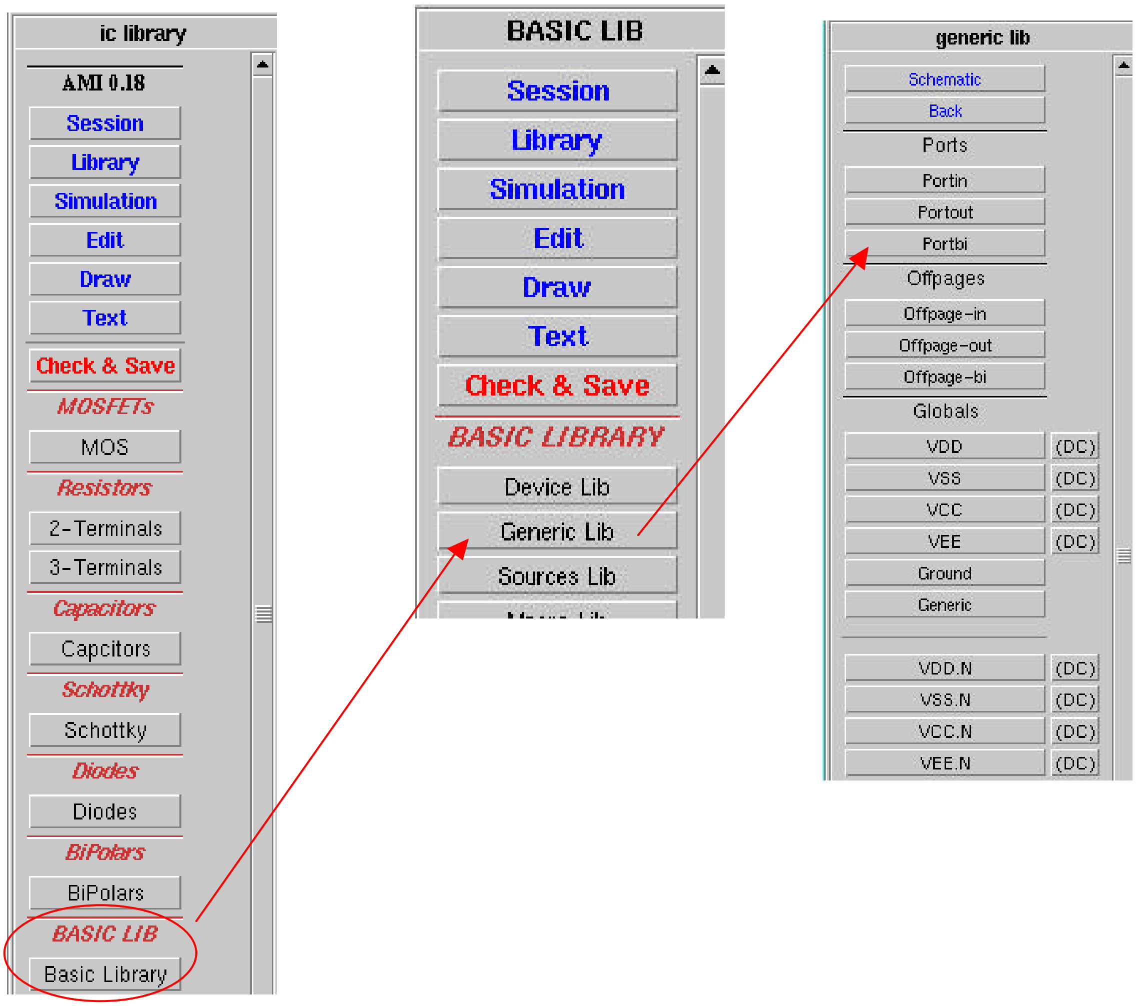
Step 4 Place the pmos transistor above the nmos so the source of pmos can easily connect to the drain of nmos as shown to the right



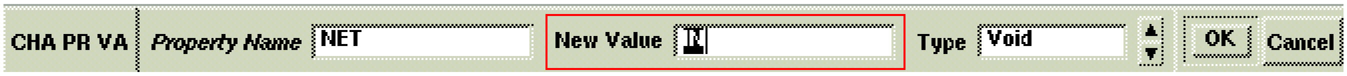
Adding Nets

Step 1 Add input and output nets by selecting the Basic Library Entry on the ic_library palette, then Generic lib, then choose Portbi .

Nota: En nuestro caso, escoja Portin para entrada, Portout para Salida, en lugar de Portbi.

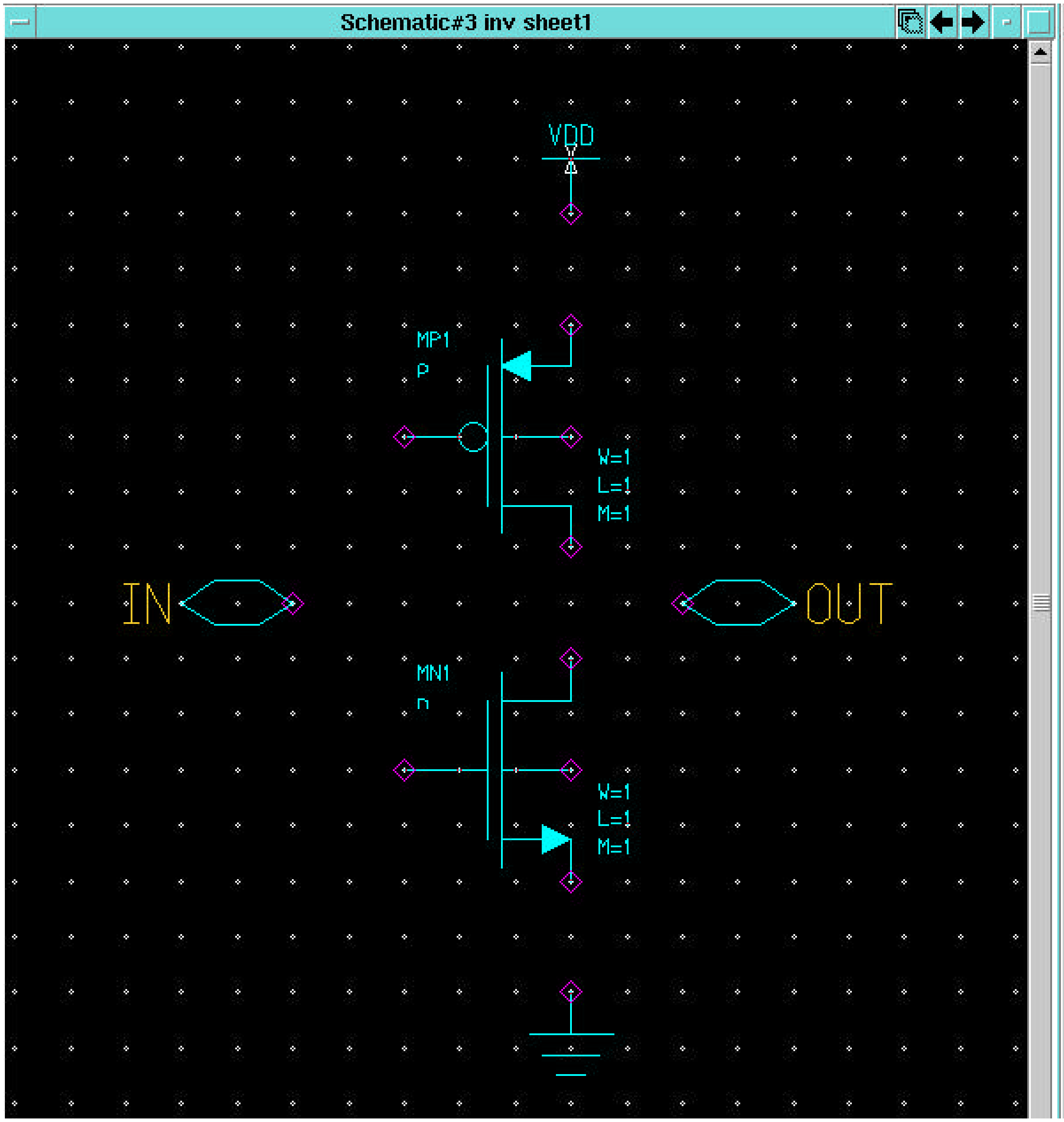


Step 2 Add VDD and Ground ports in a similar fashion



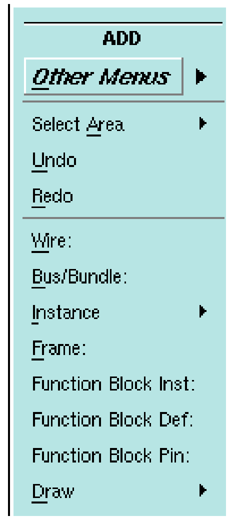
Step 3 Place the cursor over the NET on the Left and select shift-F7 key. This will bring up a form as shown above and enter IN for the net name. Do the same for the OUT net. Your schematic should be similar to what is shown below

Nota: Los puertos serán levemente diferentes, segun su dirección IN u Out.



Wiring Devices

You can wire the devices together by using the Add-Wire selection in the schematic_edit palette or by clicking the right mouse button in the workspace and selecting Wire: from the ADD popup menu.

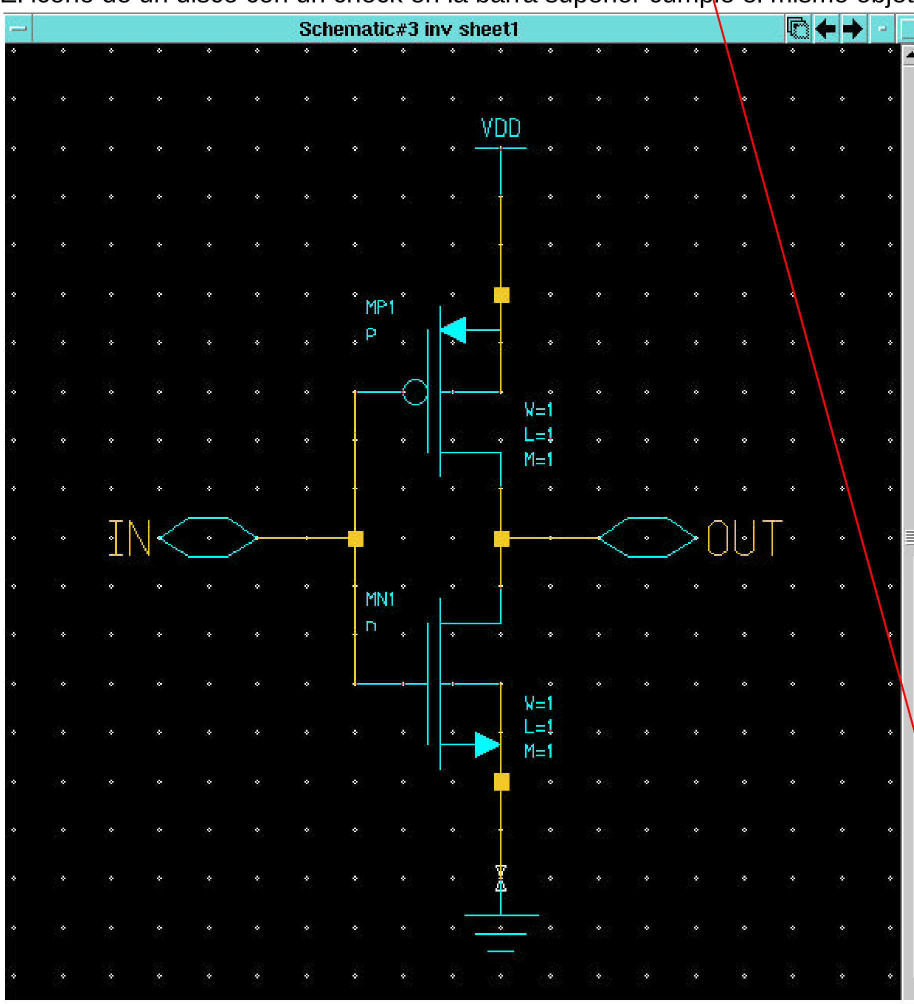


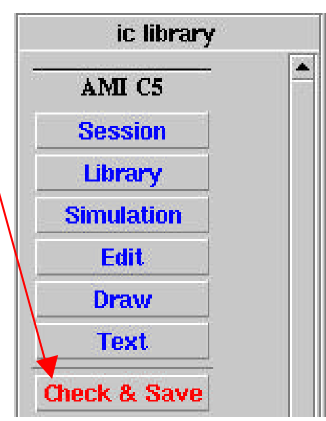
You could also use the hotkey “w” to starting wiring the devices

Step 1 Select “w” and wire the schematic as shown below.

Step 2 When completed select the Check and Save on the ic_library palette

El icono de un disco con un check en la barra superior cumple el mismo objetivo



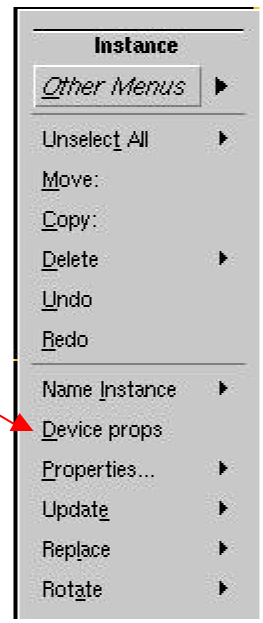


Changing parameter values

Your schematic is now complete and you will now need to create a testbench for simulation. You can manipulate devices and change device parameters by the following method that is descibed below. If you are satisifed with the devices you may skip to the section entitled Generating a Symbol to continue.

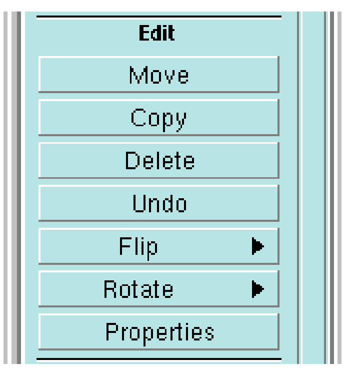
Change Device Parameters

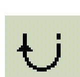
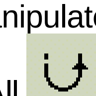
By selecting the device you want to change its parameters, then selecting the mouse right button this menu will appear. Select *Device props* and a menu like the one that appeared when instantiating the device will appear and you can change the narameters you want.



Selecting and manipulating instances

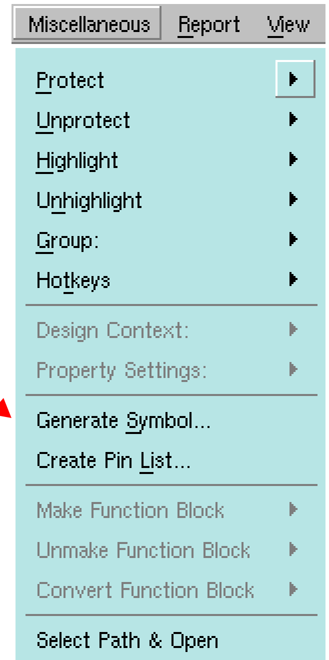
Devices can be selected with the left mouse button or by pressing the F1 function key while the cursor is over the device. With the Edit section of the schematic_edit palette you can Move, Copy, Delete, Undo, Flip or Rotate the instance. You can also use the left palette in da_ic.



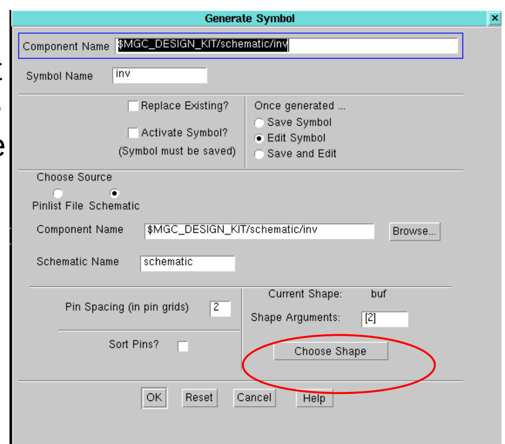
The Flip Horizontal  and Rotate  strokes allow you to quickly manipulate selected devices as do the "f" and "r" hotkeys. Unselect a device with the Unselect All  stroke or by pressing the F2 key

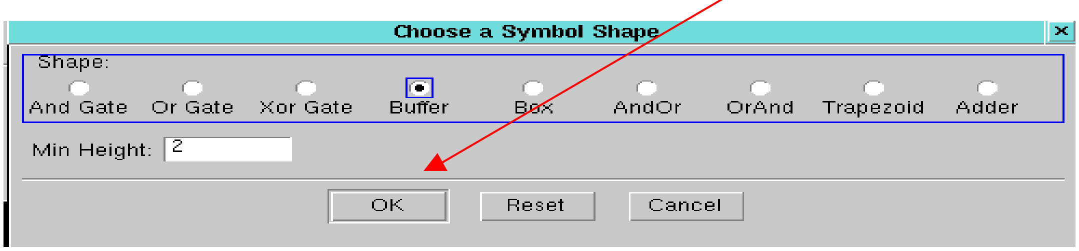
Generating a Symbol

Step 1 Select Miscellaneous and then the Generate Symbol from the pull down area



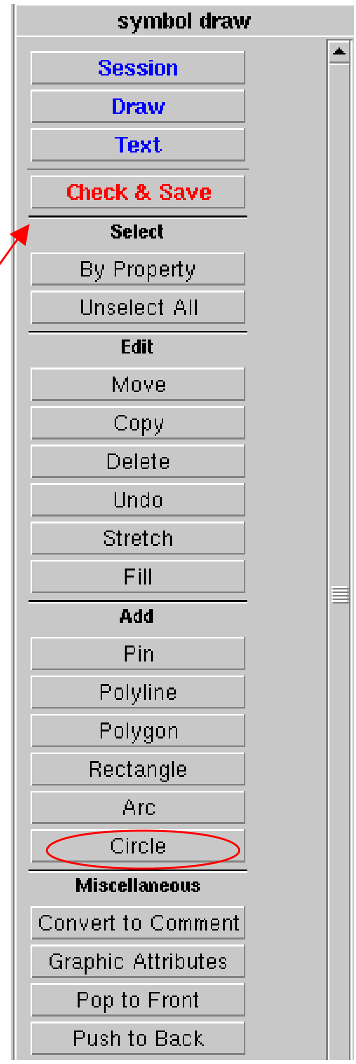
Step 2 When you select generate symbol the form to the right will popup. You can OK the form or you can change the shape to be a buffer and add a circle to create a classic inverter shape.



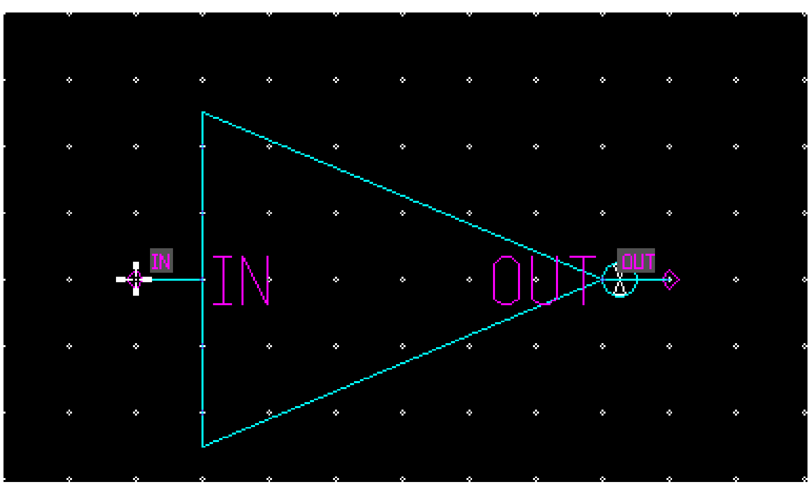


Step 3 You can select the circle from the symbol draw menu to add a cicle to the inverter symbol, or from the left palette.

Step 4 When you are happy with the symbol perform a check and save.



It should look as shown below (or just a box shape depending on your choice)

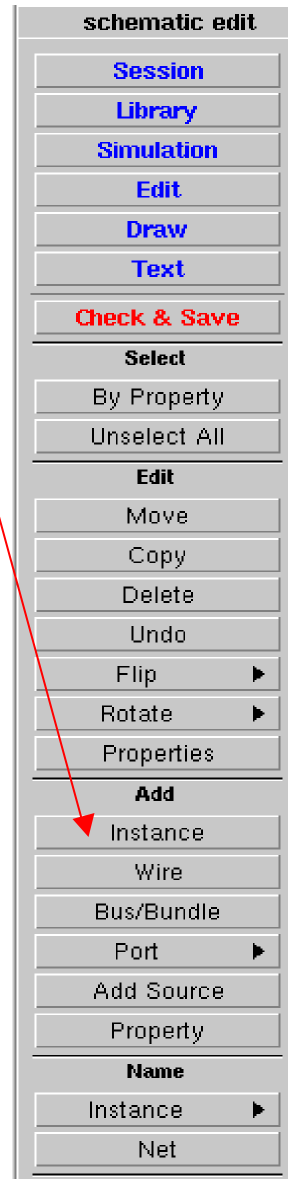


Now you can use the symbol to create a testbench for simulation

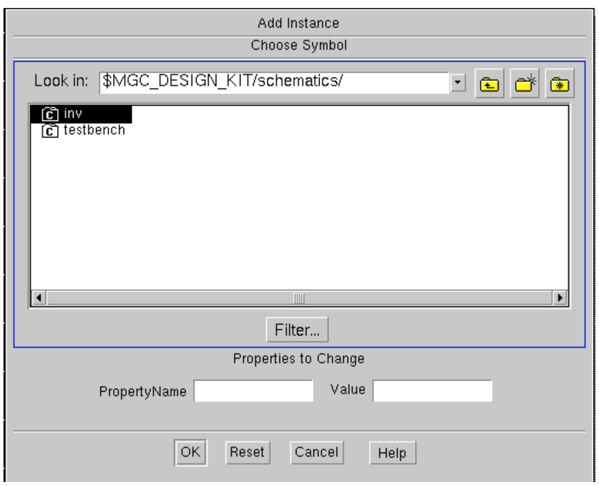
Creating a Testbench

Step 1 Exit both the schematic and symbol window. Open a new schematic with the name testbench.

Step 2 Now instantiate the new inverter symbol by selecting Add Instance from the Schematic Edit Palette menu



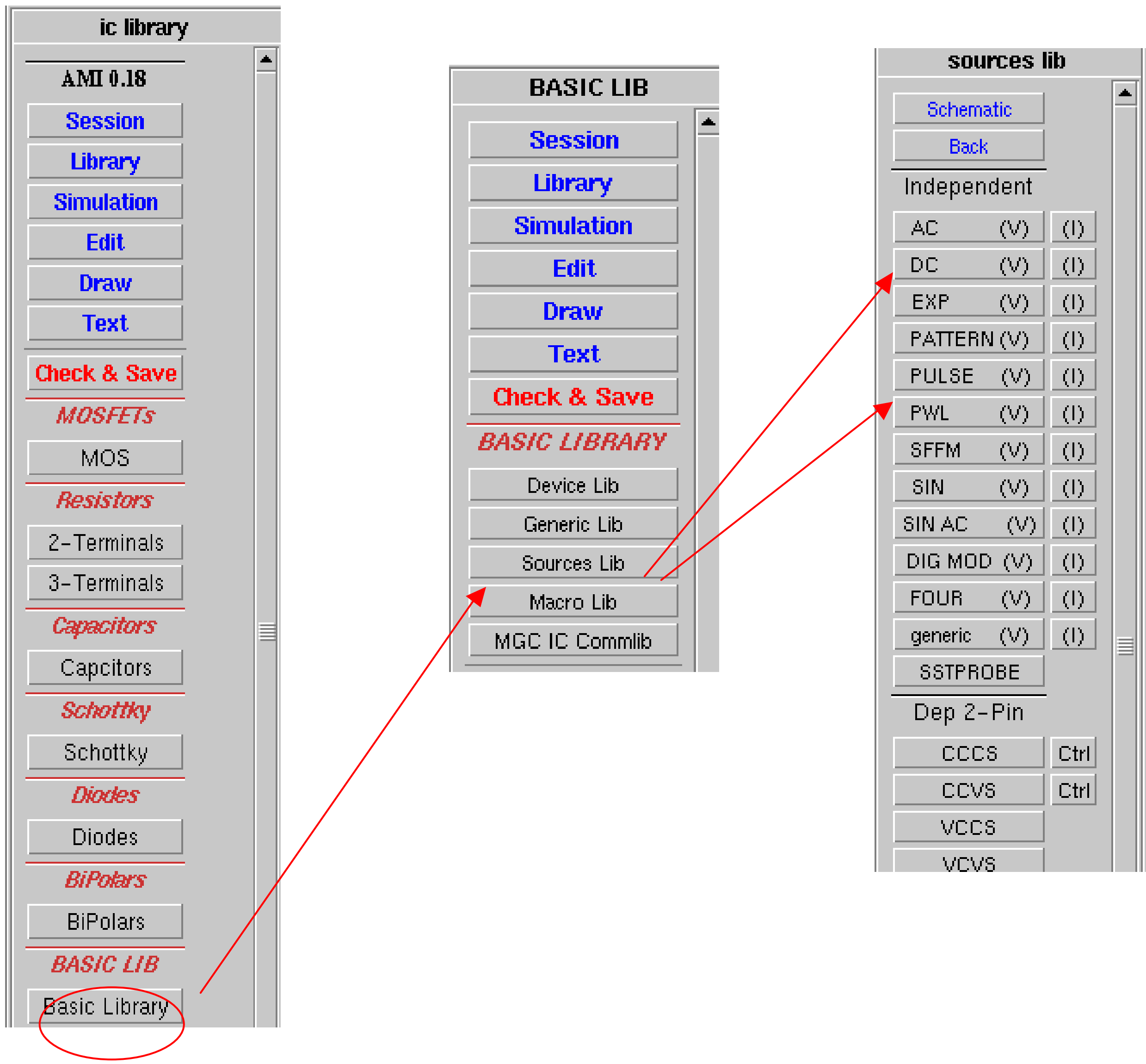
Step 3 Select inv and OK the form



Step 4 Add IN and OUT nets as before by selecting the Basic Library Entry on the ic_library palette. Name the nets with shift-F7 function key

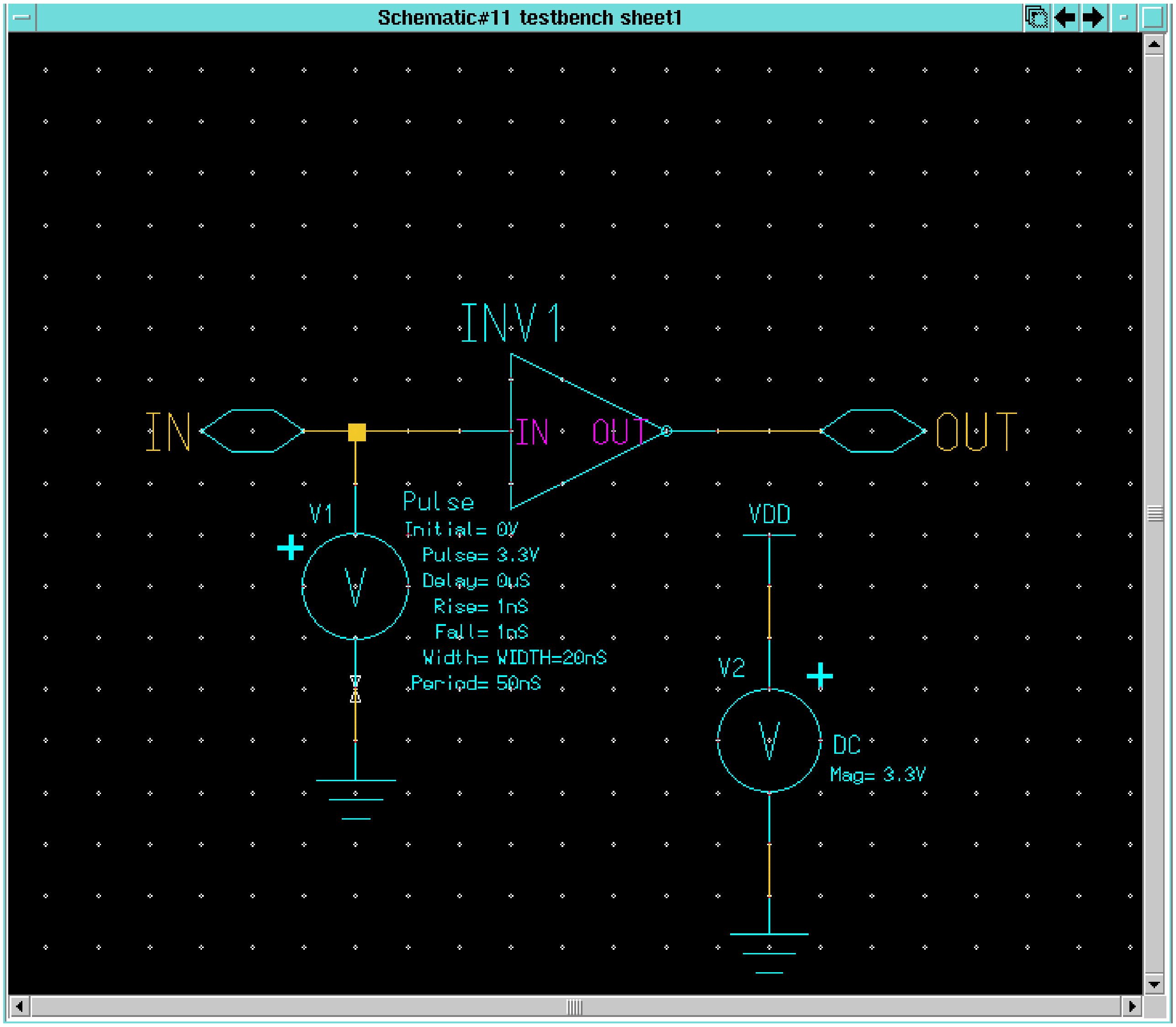
Step 5 Add VDD and Ground ports in a similar fashion

Step 6 Add PULSE(V) and DC(V) from the ic_library Sources sub menu as shown below



Change the DC magnitude value of the DC source and the amplitude of the Pulse generator to 3.3v, by selecting the device, use the mouse right button, click on it, choose properties… from the pop-up menu, then modify multiple.

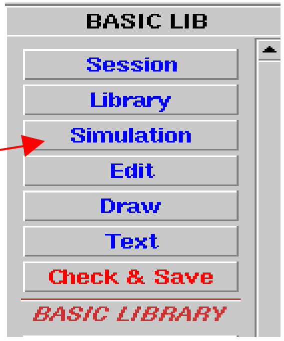
The completed schematic should look similar to the one below:

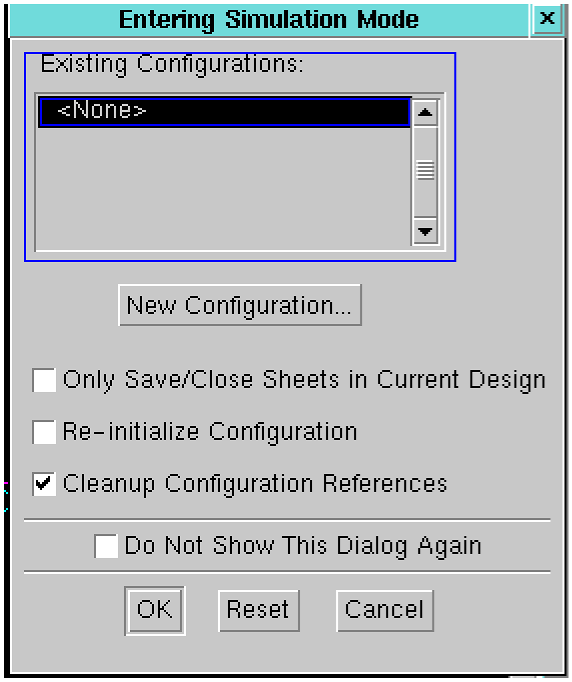


Simulating the Testbench

Step 1 Select Check & Save to Save schematic changes.

Step 2 When you have no errors select the Simulation Button to go into design context and simulate our design

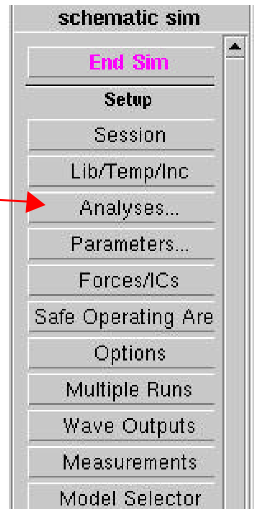




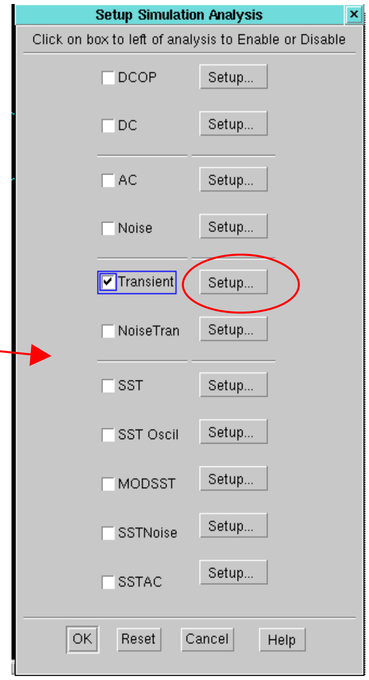
Step 3 Click ok when this form appears.

Now you are in the Design context and need to setup the analysis type, plots and load in the eldo models.

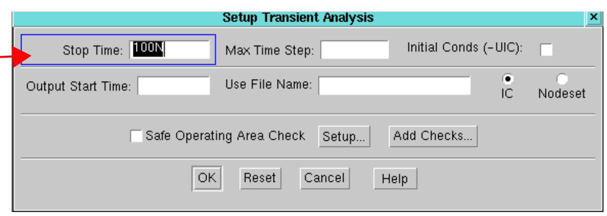
Step 4 Select Analyses



Step 5 Select Transient followed by selecting the Setup button

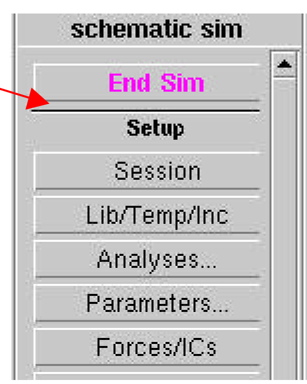


Step 6 Enter Stop Time as 100ns and select OK
Nota: la ventana puede ser un poco diferente.

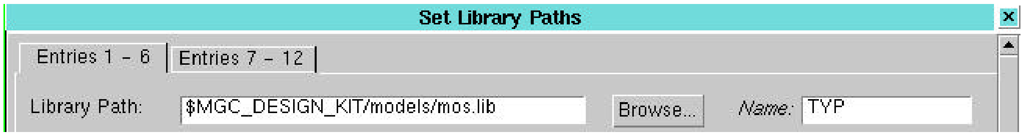


Step 7 Include models from "Lib/Temp/Inc" to tell Eldo where the Eldo models are located

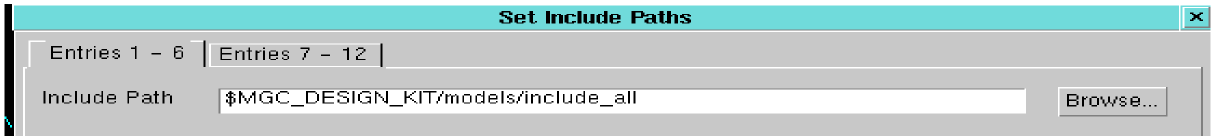
Choose libraries then include files



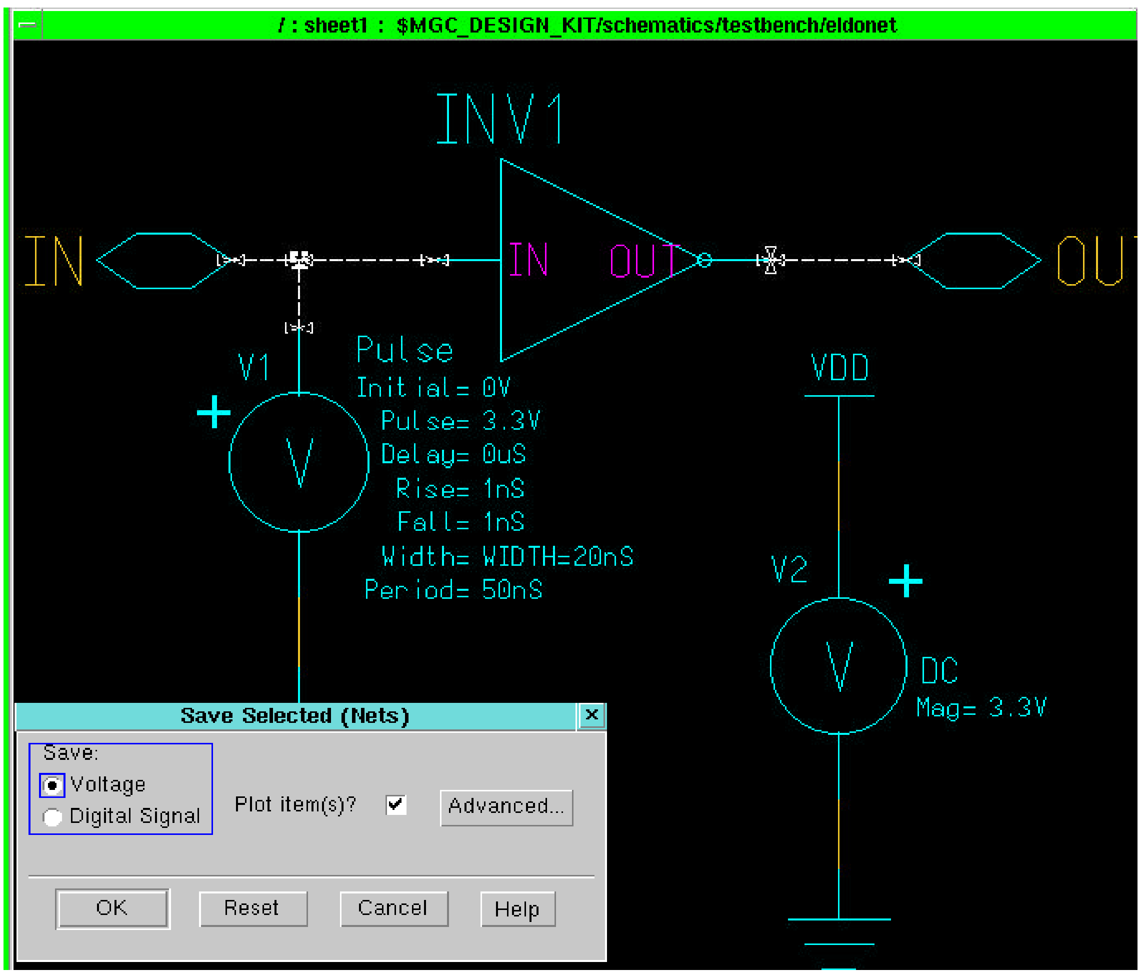
Step 8 Libraries :Navigate to the $MGC_DESIGN_KIT/models directory and select the
mos.lib and specify typical for Name and OK the form.



Step 9 You can just use Include Files :Navigate to the $MGC_DESIGN_KIT/models directory and include the include_all file, that will include all the needed models

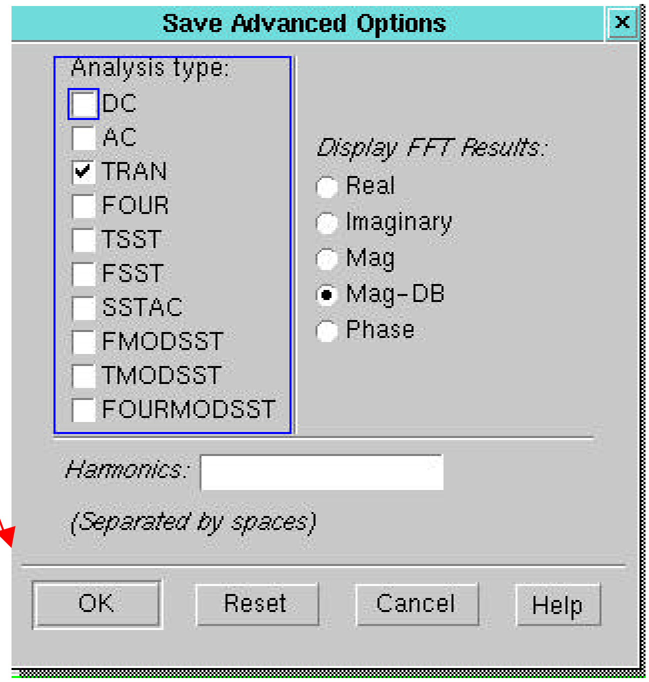


Step 10 Now select the the IN and OUT terminals by holding down the left mouse button and drawing a box around the terminals. This will cause the wires to be highlighted

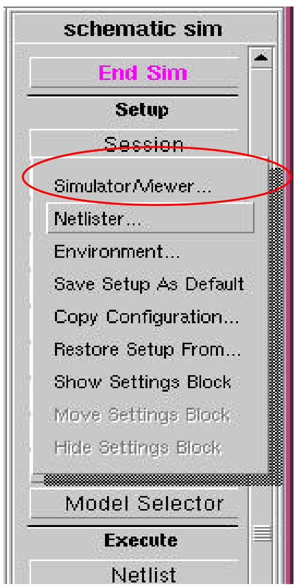


Step 11 While the IN and OUT nets are Selected go the Simulation Palette and select Wave Outputs, save selected and choose voltage in Save, and mark plot items

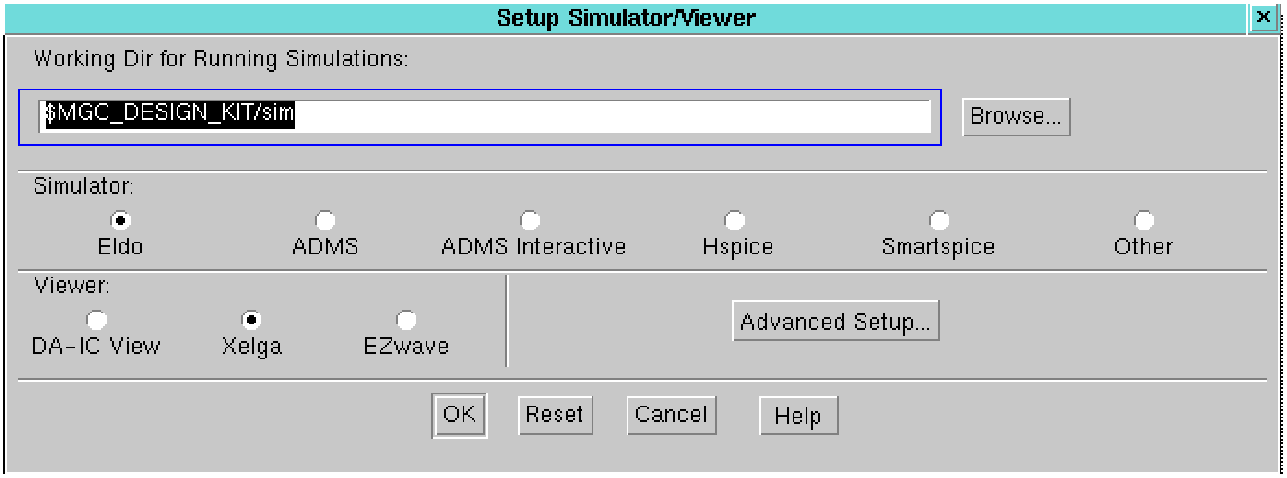
Step 12 Click advanced, Select Analysis type **TRAN**, and click OK.



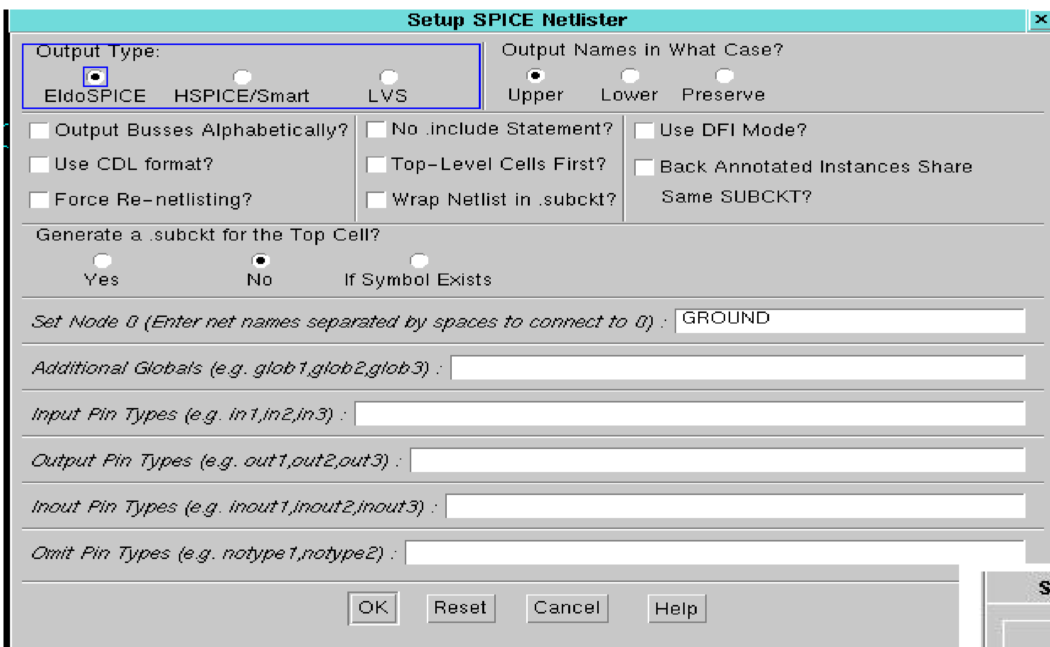
Step 13 Set the viewer to be EZWave by selecting Session then Simulator/Viewer from the Session Pull down menu as seen below



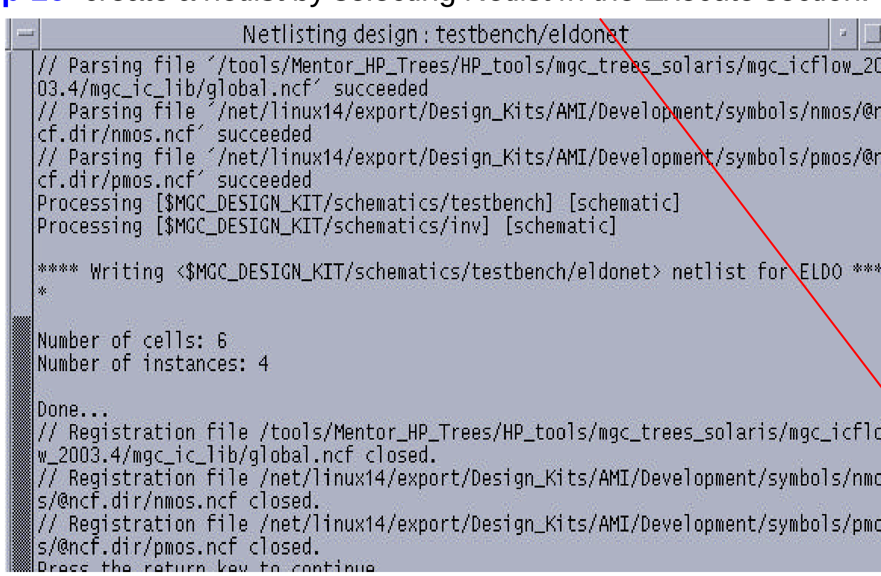
Step 14 Select Ezwave and OK the form. (Figura es diferente,cuidado). Escoja Ezwave.



Step 15 Set the Netlister to be EldoSpice by selecting Session then Netlister from the Session Pull down menu as seen below

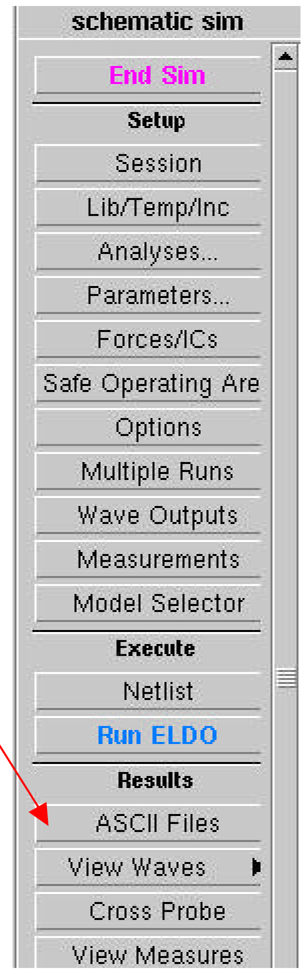


Step 16 create a netlist by selecting Netlist in the Execute section.

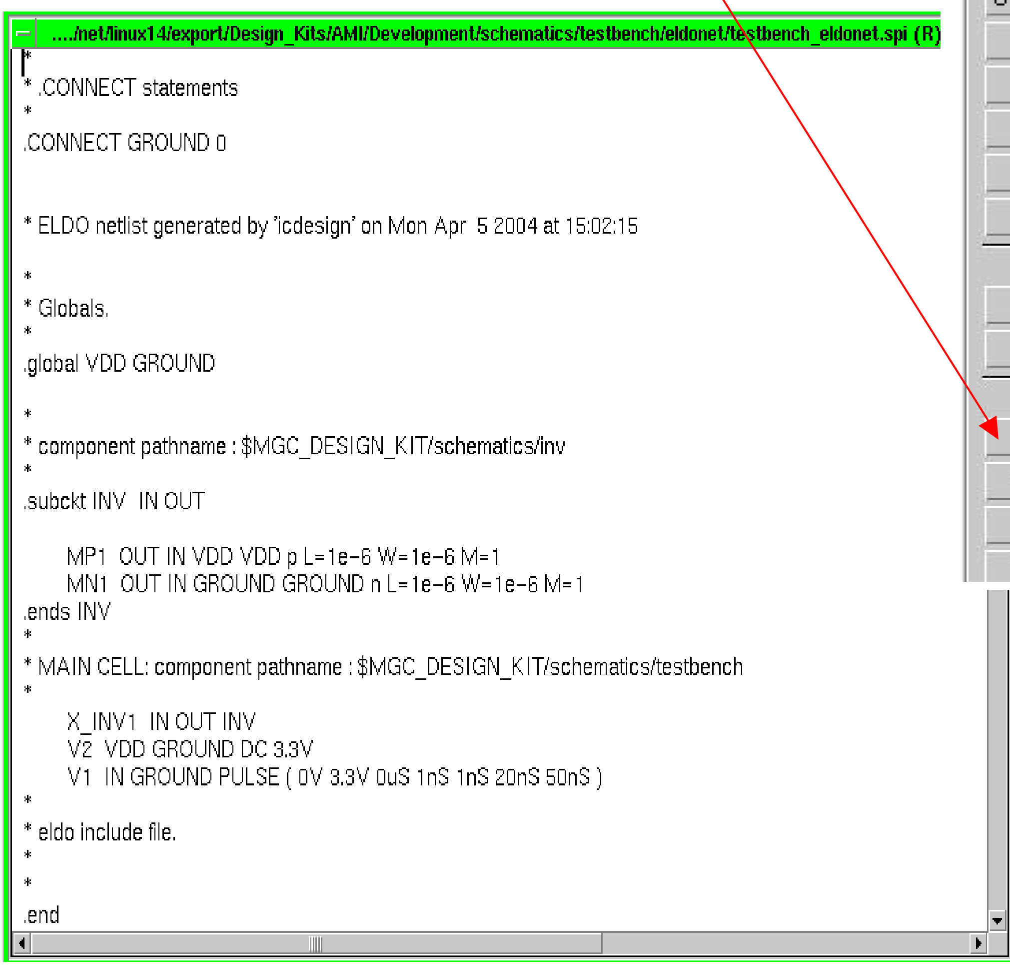


Step 17 Xterm will show the steps of netlisting, Press Enter to close the netlist xterm window

Step 18 View the netlist by selecting the View Netlist in the ASCII files of the Results session



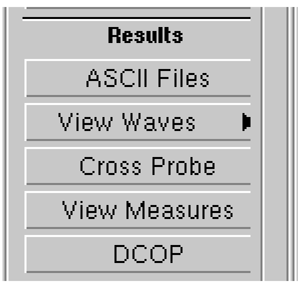
The netlist should look similar to this:

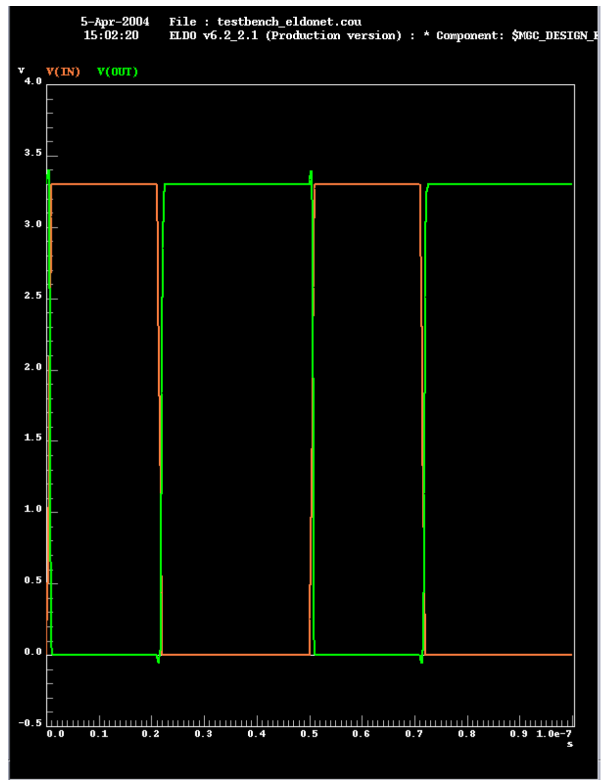


```
..../net/linux14/export/Design_Kits/AMI/Development/schematics/testbench/eldonet/testbench_eldonet.spi (R)
*
* .CONNECT statements
*
.CONNECT GROUND 0


* ELDO netlist generated by 'icdesign' on Mon Apr  5 2004 at 15:02:15

*
* Globals.
*
.global VDD GROUND

*
* component pathname : $MGC_DESIGN_KIT/schematics/inv
*
.subckt INV  IN OUT

      MP1  OUT IN VDD VDD p L=1e-6 W=1e-6 M=1
      MN1  OUT IN GROUND GROUND n L=1e-6 W=1e-6 M=1
.ends INV
*
* MAIN CELL: component pathname : $MGC_DESIGN_KIT/schematics/testbench
*
      X_INV1  IN OUT INV
      V2  VDD GROUND DC 3.3V
      V1  IN GROUND PULSE ( 0V 3.3V 0uS 1nS 1nS 20nS 50nS )
*
* eldo include file.
*
*
.end
```

Step 19 Close the netlist window (Shift – F12) or Right-to-Left stroke, then select the Run ELDO button under the Simulation Section. Press Enter to close the Simulation xterm.

Step 20 View the simulation results by selecting the View Waves button under the Results section of the Simulation Palette



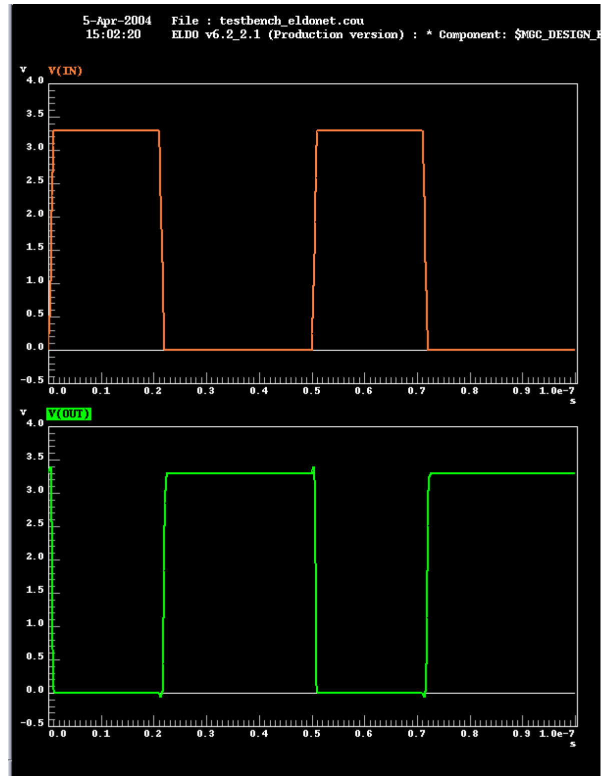


You will see the IN and OUT waveforms
Las verá en Ezwave

Step 21 Sobre EzWave, trate de crear dos plots separados. Solicite ayuda si la requiere.

Pista: use Waveform List, en Tran.

You should see something similar to these graphs oh the right (pero en Ezwave) .



Creating Layout

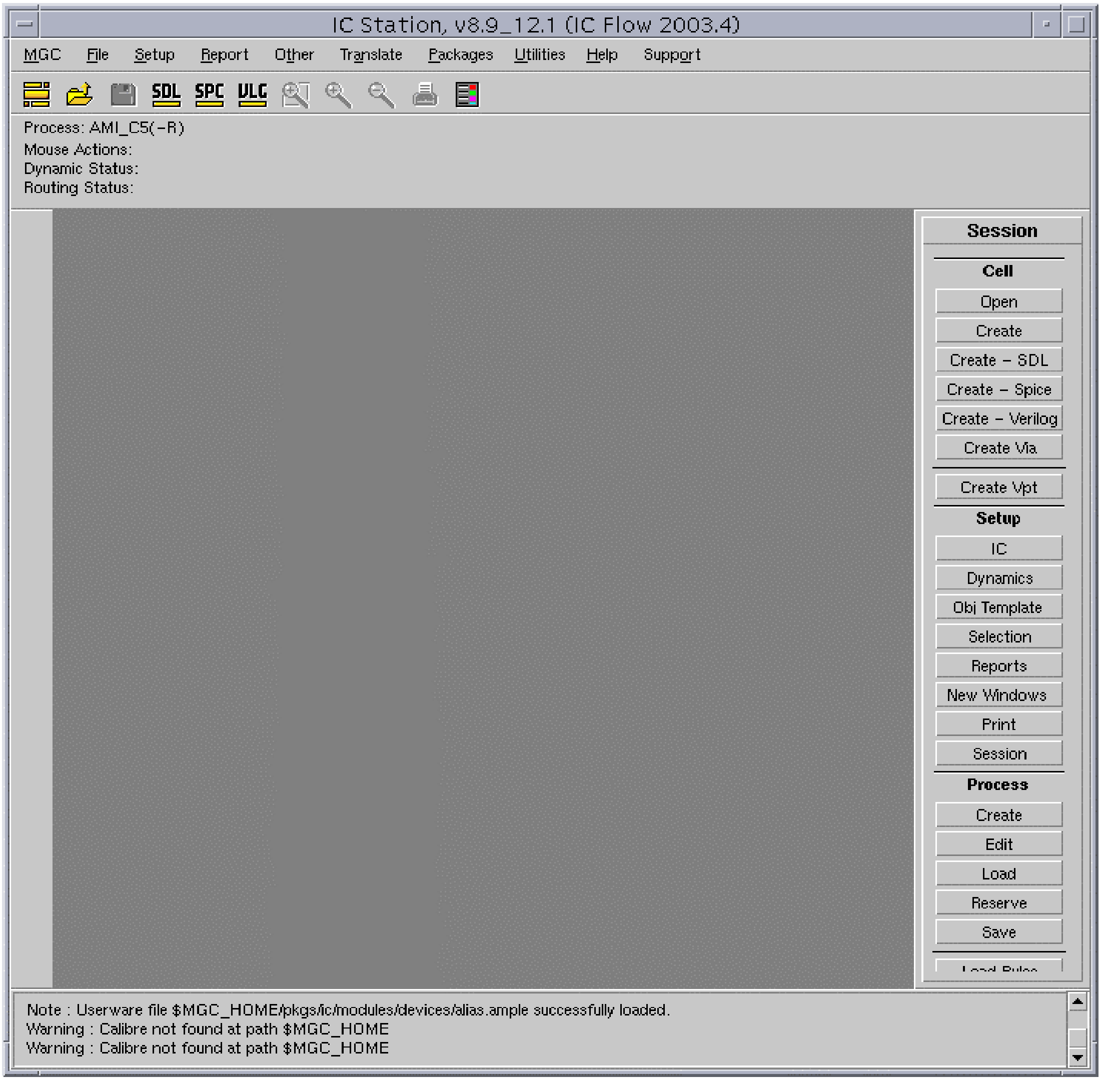
Once you are happy with the simulation results you can now setup the new design kit components to function on the back end. The symbols already have the properties required to drive the device generators using SDL .

Invoking and Setting Options in Layout

You can now invoke IC Station that will use the same inverter database that was used in simulation.

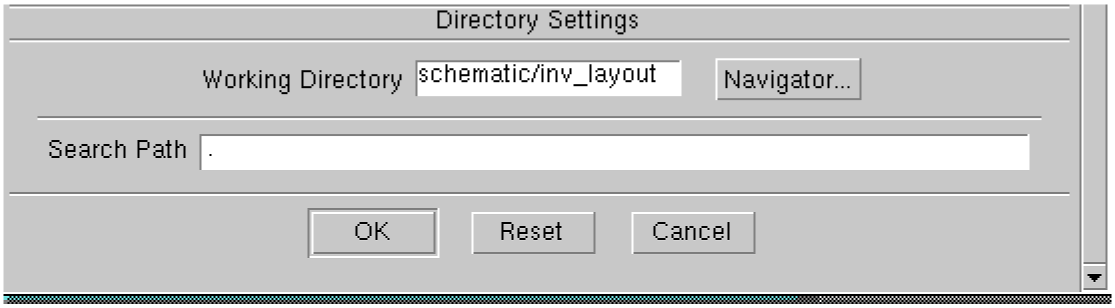
Step 1 Open IC Station by entering ***ic&***

You will see the following:



Setup Layout Directory

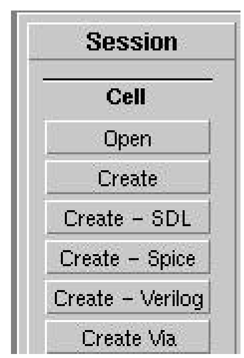
Step 1 Select IC from the Setup menu and navigate to the $MGC_DESIGN_KIT/lschematics/inv_layout directory



This will tell IC Station to store the layout cells in the $MGC_DESIGN_KIT/schematics/inv_layout directory.

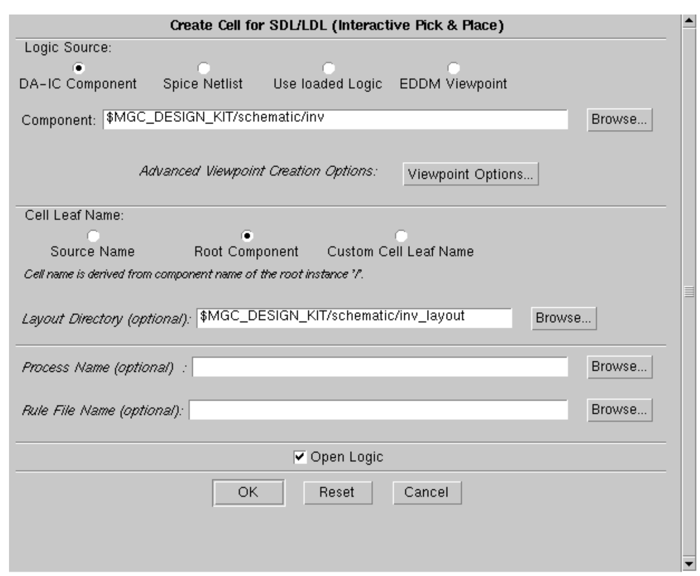
Creating SDL

Step 1 Select Create-SDL from the right hand palette menu which should popup the load rules form.

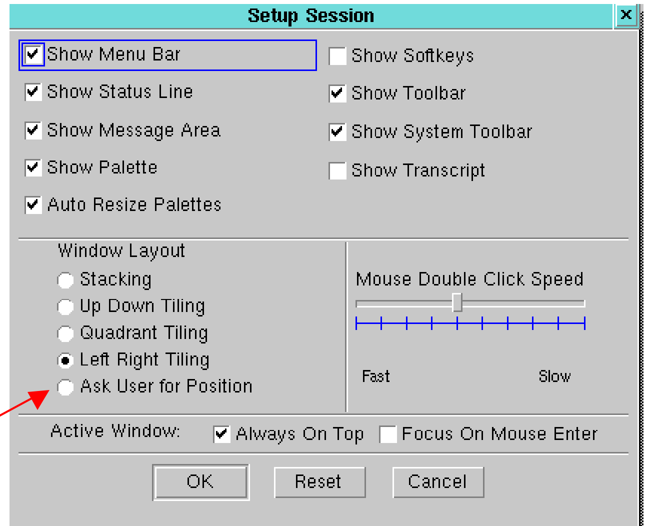


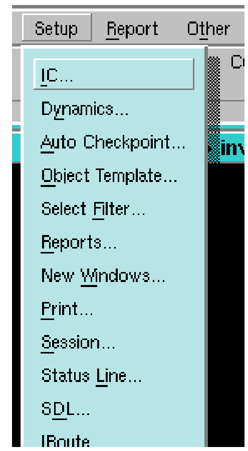
Step 2 Navigate to Cell Name $MGC_DESIGN_KIT/schematics/inv_layout

Step 3 select DA-IC component $MGC_DESIGN_KIT/schematics/inv and OK the form



IC Station will now bring up two overlapping windows. One window will be the logic source (schematic that you just created) and a blank window where the layout will be created.

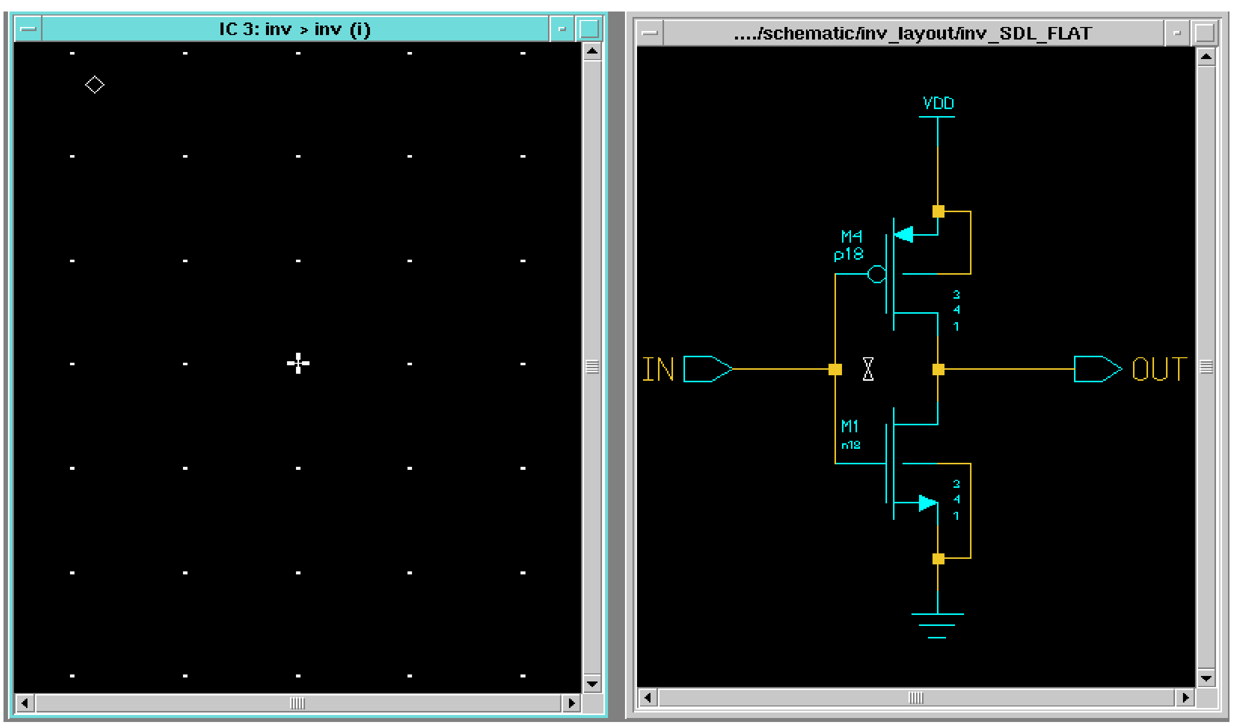




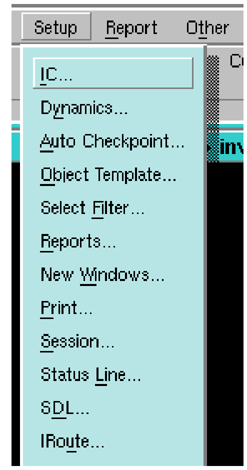
OK

Step 1 Select from the pulldown menu Setup->Session and select Left Right Tiling and form

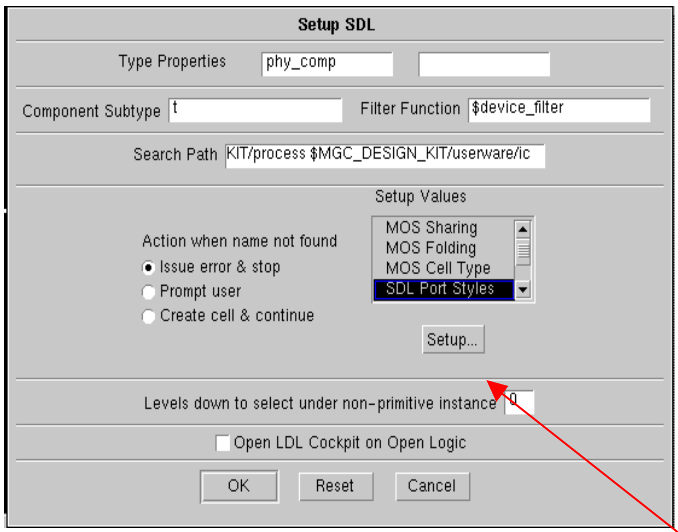
You will now see the windows side by side as shown below. Let's go ahead and setup a few other options that will be used as we generate the layout for your inverter.



Step 2 Select Setup->SDL from the Pulldown menu

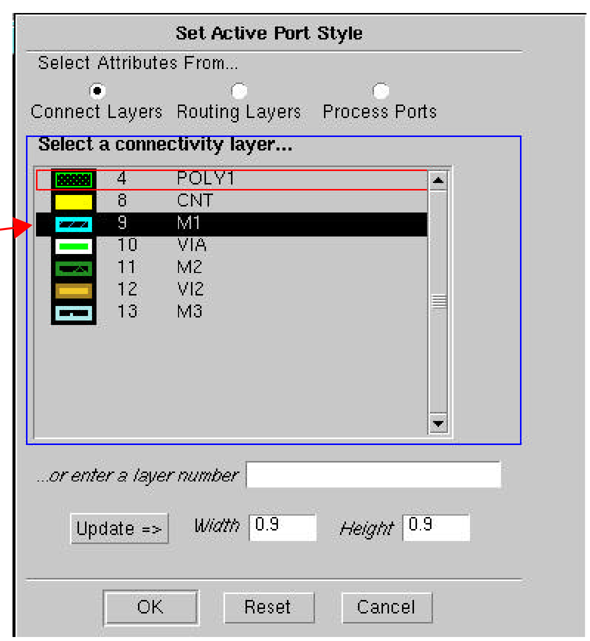


You will now be setting up the default port styles to be used with routing VDD, GROUND, IN and OUT

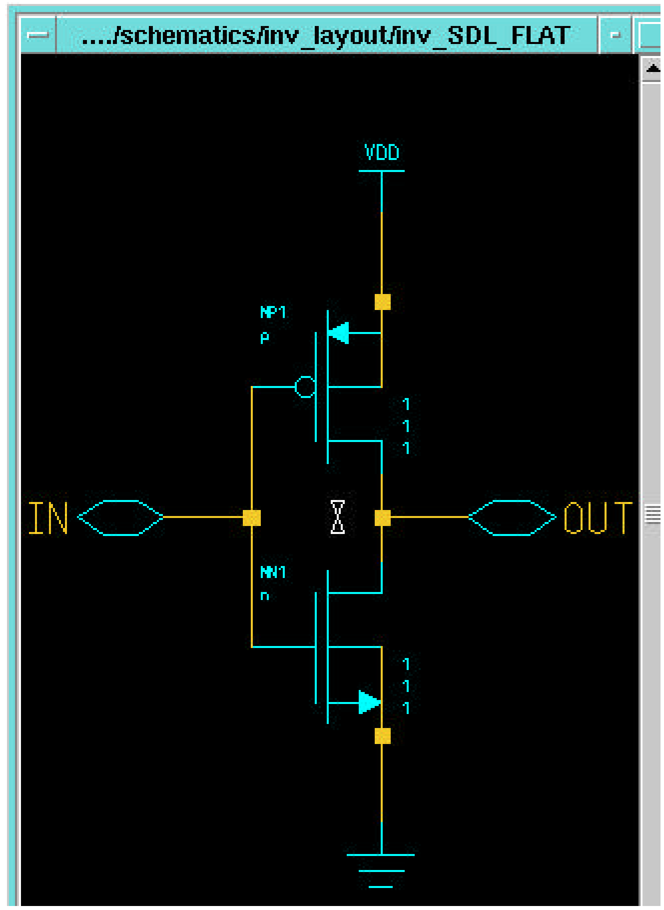


Step 3 Select SDL Port Styles from the Setup Values popup menu and then Setup

Step 4 Select M1 from the Select a connectivety layer and OK

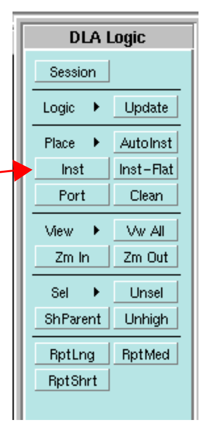


Step 5 Make the schematic window active by selecting it with the LMB



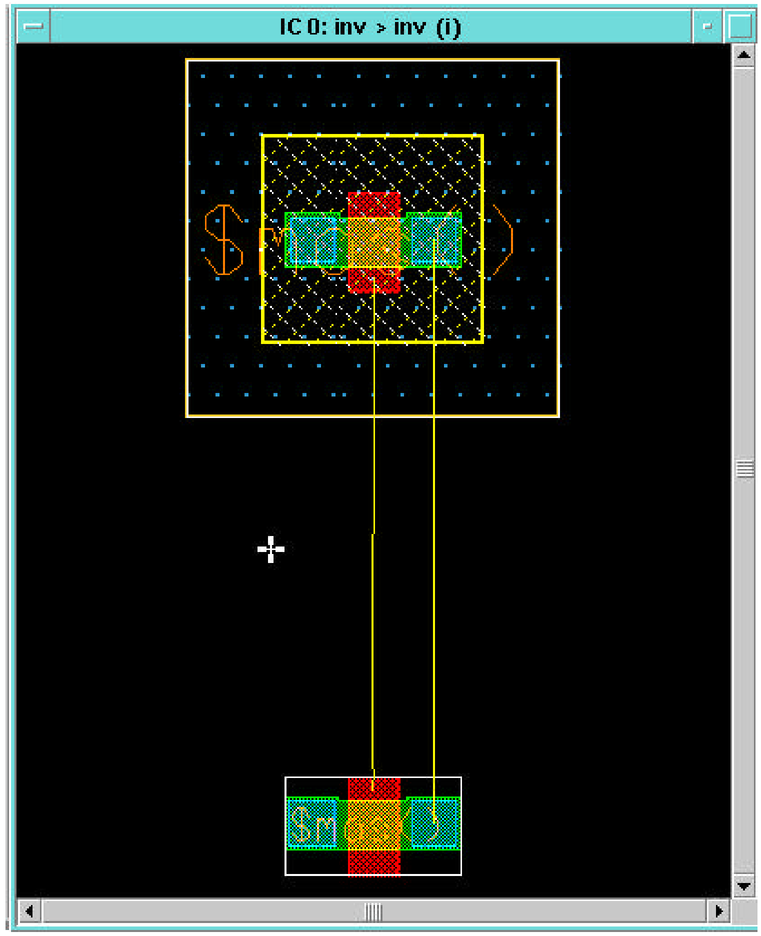
Step 6 Select the pmos component with the LMB

Step 7 While the pmos is selected, select the Inst-Flat from the DLA Logic Palette.



Step 8 Place the ghost image of the pmos device in the layout window

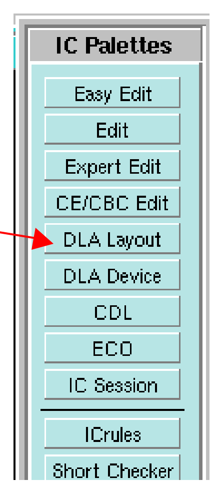
Step 9 Now select nmos in the Logic Window followed by Inst from the DLA logic and place the nmos device as shown on top.

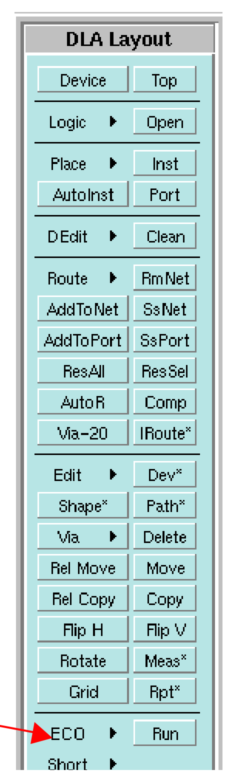


You have now instantiated pmos and nmos and the connectivity is maintained as illustrated by the fly lines. Next you will add ports and complete the routing.

Before routing you will need to adjust the grid as follows:

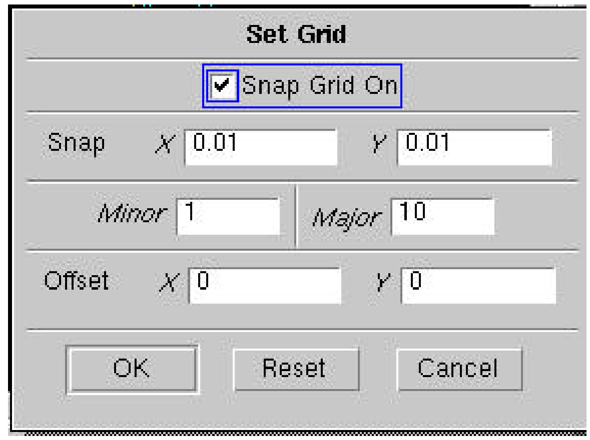
Step 10 Select DLA Layout form the IC Palettes menu





Step 11 Select Grid from the DLA Layout Palette

Step 12 Set the Snap X and Y to 0.01 and OK the form



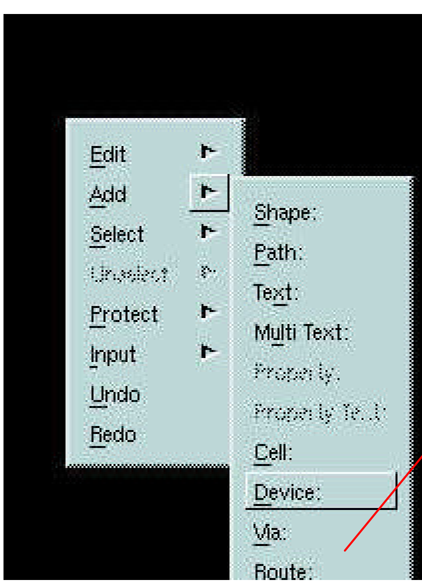
Routing Layout

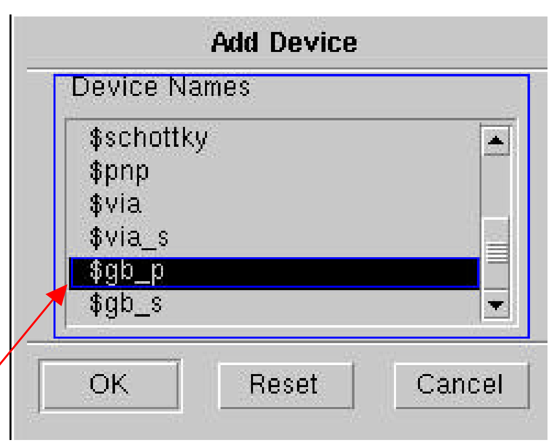
Now let's zoom in to take a closer look at the layout and get ready to route.

Step 1 Select the Layout window and select the shift-F8 button. This will create a full window image of the layout.

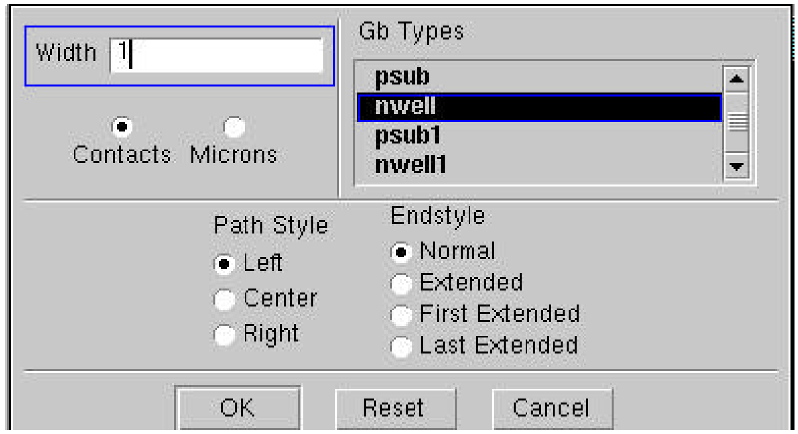
Now you can add the NWELL tie down for the pmos devices and the PSUB tiedown for the nmos devices, such that the well of the nwell/psub is intersecting the well of pmos/nmos.This is used to connect the bulk terminals.

Step13 For pmos device, add nwell guard band from add device popup menu "$gb_p",as shown using the right mouse button.

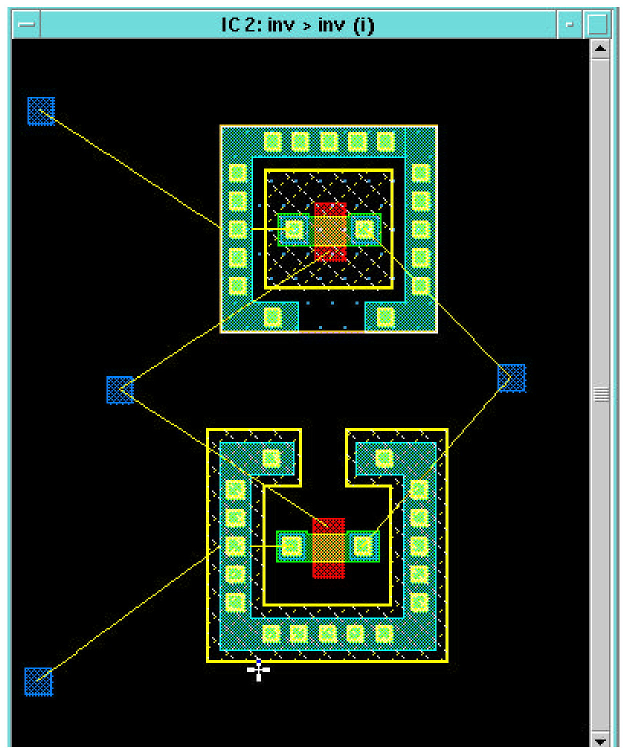




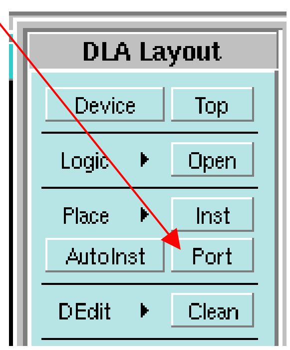
step14 For nmos device, do the same, add psub guard band from add device “$gb_p” popup menu, as mentioned above.



Now let’s add the ports.



Step 1 With the layout window active, select the Port entry from the DLA Layout palette.



Step 2 You will be asked to place the GROUND, VDD, IN and OUT port in order. Place the ports as illustrated.

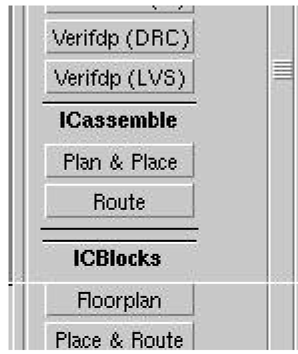
Si desea ver los contactos a bulk, dibuje una L invertida con el boton central sobre las guardas

Step 3 Select the GOUND port, and use the right mouse button, choose add then text from the pop-up menu, the add text menu will pop up as below, write Ground in the Value field then click on the port.

Step 4 Do the same with the rest of the ports.

Step 5 Select Route from the IC Palettes. (Top siempre lleva al IC Palette).

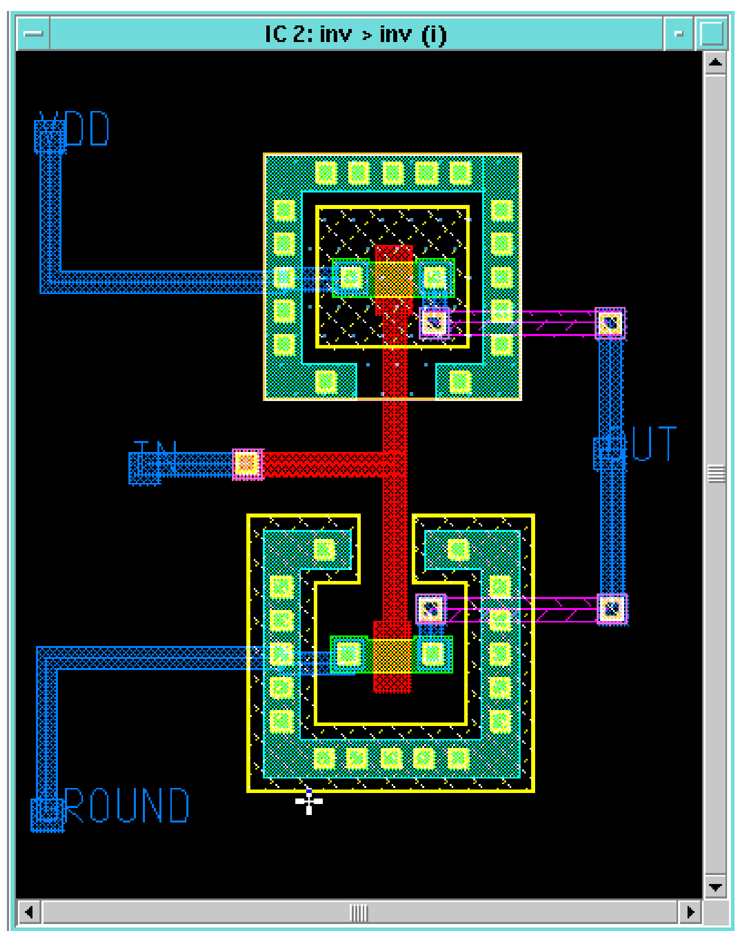
Step 6 Press on the green button, “RUN” on the AROUTE menu for auto routing.



Es probable que el AutoRoute no haga bein su trabajo. Ya verificaremos con Calibre (DRC)

Nota: File>Cell>Reserve>Current Context permite volver a editar la celda en caso de que se bloquee.

After completion,You should see something similar to the one on the right.



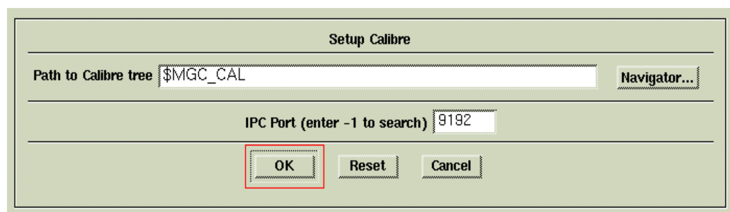
Verification of Layout

Now you can verify the layout by running DRC and LVS checks. we will run Calibre Interactive.

Running Calibre Interactive DRC:

Step 1: Select Calibre from the Pulldown menu followed by selecting the Run DRC entry

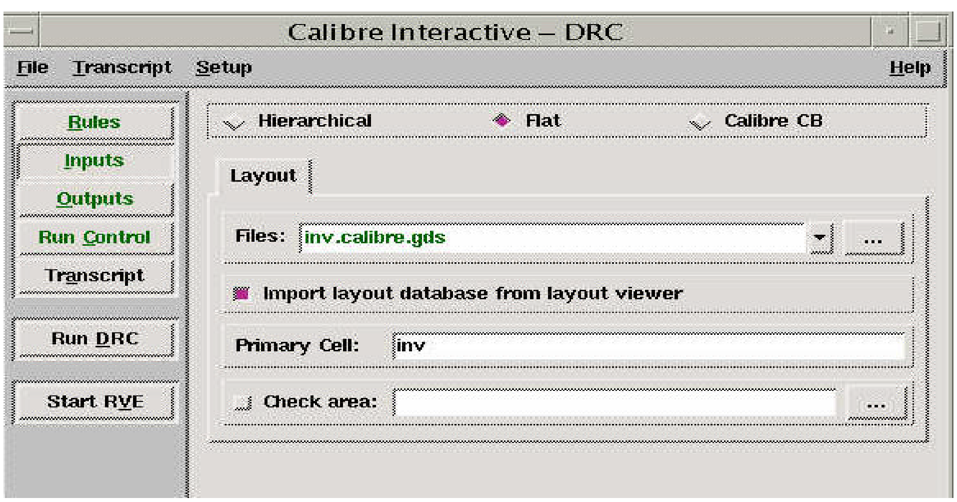
Step 2: Since the path to Calibre has not been setup enter $MGC_CAL for the path to the Calibre tree as seen to the right and OK the form



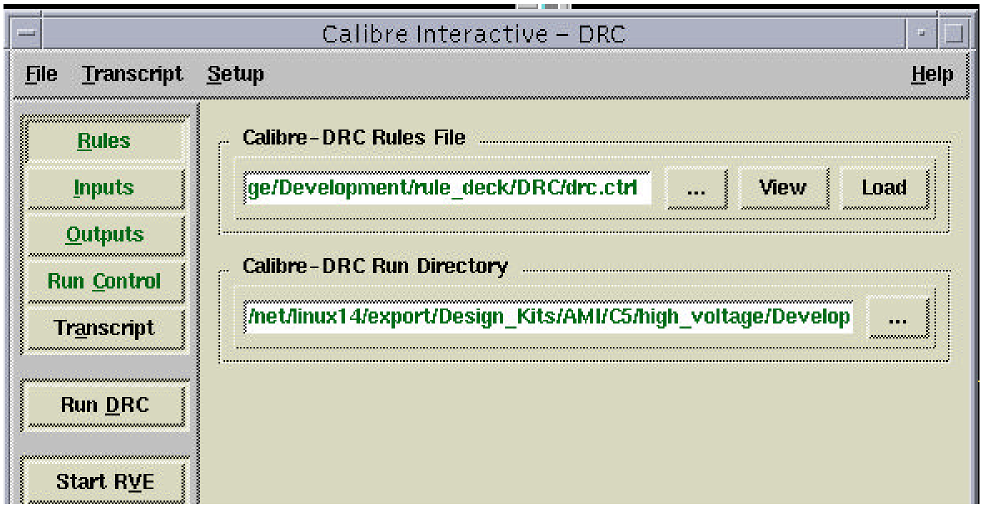
$MGC_CAL: Ask for correct path. $CALIBRE tiene el path correcto. Introduza esta variable.

Step 3: Go back to the Calibre-Run DRC from the pull down again. This will bring up the Calibre Interactive –DRC menu.

Select Flat and make sure Import layout database from the layout viewer is selected when the Inputs tab is selected as shown to the right

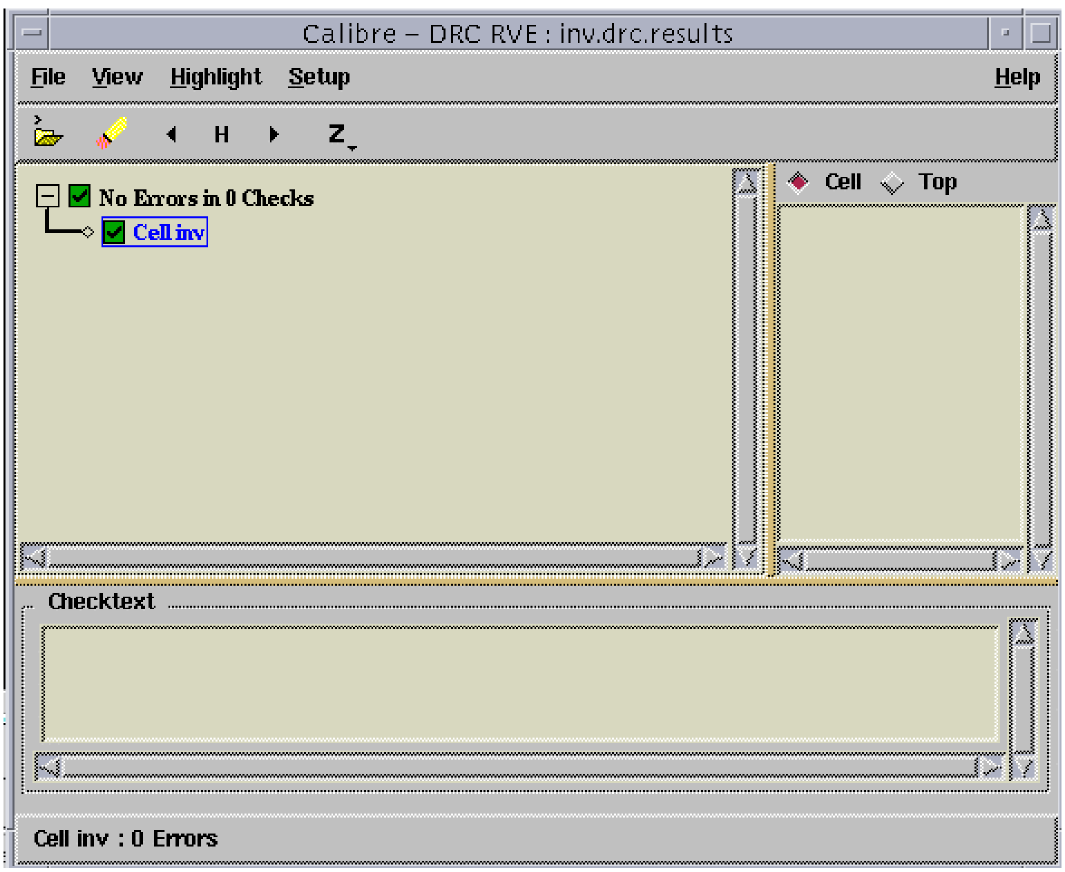


Step 4: Select Rules button and navigate to $MGC_DESIGN_KIT/rule_deck/DRC/drc.ctrl load Calibre file.



El camino correcto en el Run DIrectory es: $HOME/designkit/

Step 5: Select Run DRC
Step 6: The Calibre RVE window will popup and you should see the following results.

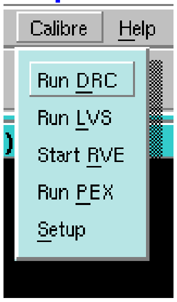


Es probable que aparezcan gran cantidad de errores. Ello debido a que el ARoute no hizo bien su trabajo. En particular, si el ruteo se hace con polisilicio, al pasar por los contactos a bulk, se crean transistores. Haremos un ruteo con el IRoute, que es el enrutador manual interactivo.
Busque el botón de IRoute en la paleta. Al seleccionarla y mover el cursor sobre la pantalla, verá alrededor de este un cuadro que indica el material que se usará para rutear. La barra espaciadora permite ir intercambiando el material (el ancho es determinado por las reglas mínimas, por ahora no nos preocupamos del asunto). Borre las líneas que le parezca que están mal. Colóquese luego en los puntos que quedan desconectados (indicados por las líneas amarillas, overflows) e inicie el ruteo. Salga con el material indicado. Cuando desee cambiar de material (Metal 1 vertical, Metal 2 Horizontal, por ejemplo), use la espaciadora. La herramienta añade las vías o contactos necesarios para pasar entre materiales. Fíjese que el IRoute le sugiere la dirección más corta siempre, y le impide pasar por dónde no debe, debido a las reglas del proceso y a la conectividad del circuito.
Una vez que tiene todo enrutado (no hay líneas amarillas visibles), ejecute el DRC. Debería pasar.

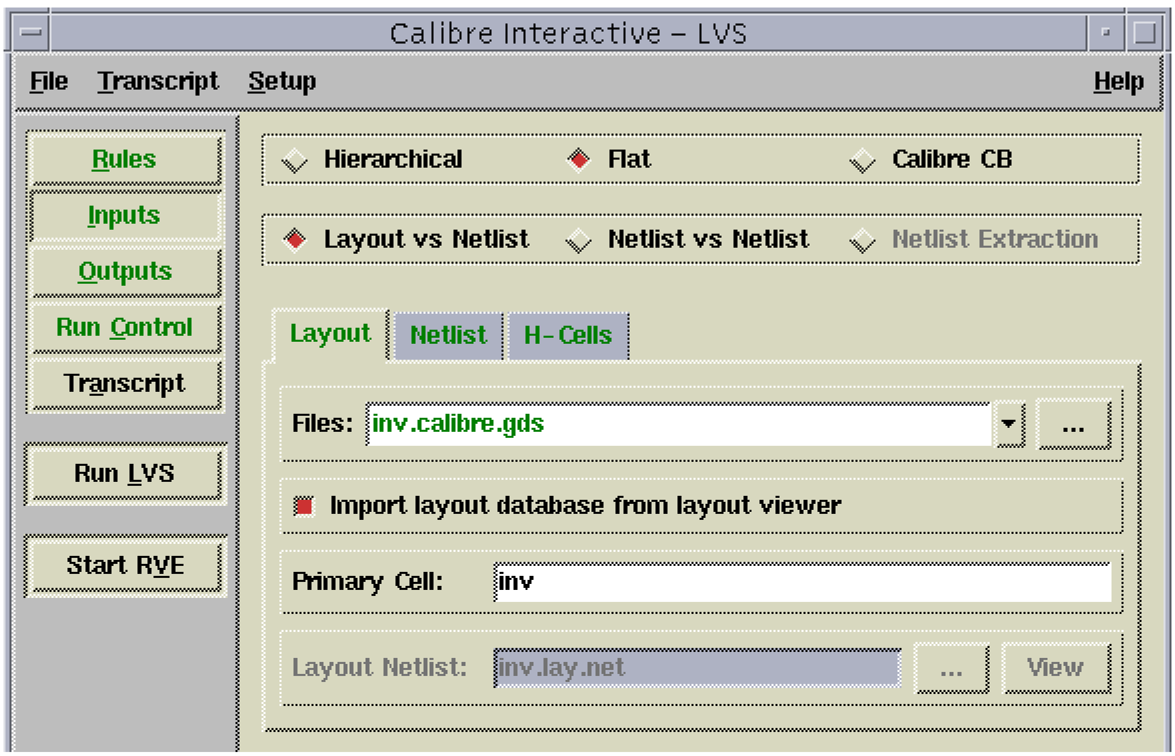
En la próxima sección, usted hará un LVS. Al finalizar, sería interesante ver qué pasa si usted borra una conexión. Hágalo y corra el LVS de nuevo. Calibre le ayuda a averiguar los nodos sueltos. El instructor puede ayudarle con esto.

Running Calibre Interactive LVS:

Step1: Select Calibre from the Pulldown menu followed by selecting the Run LVS entry.

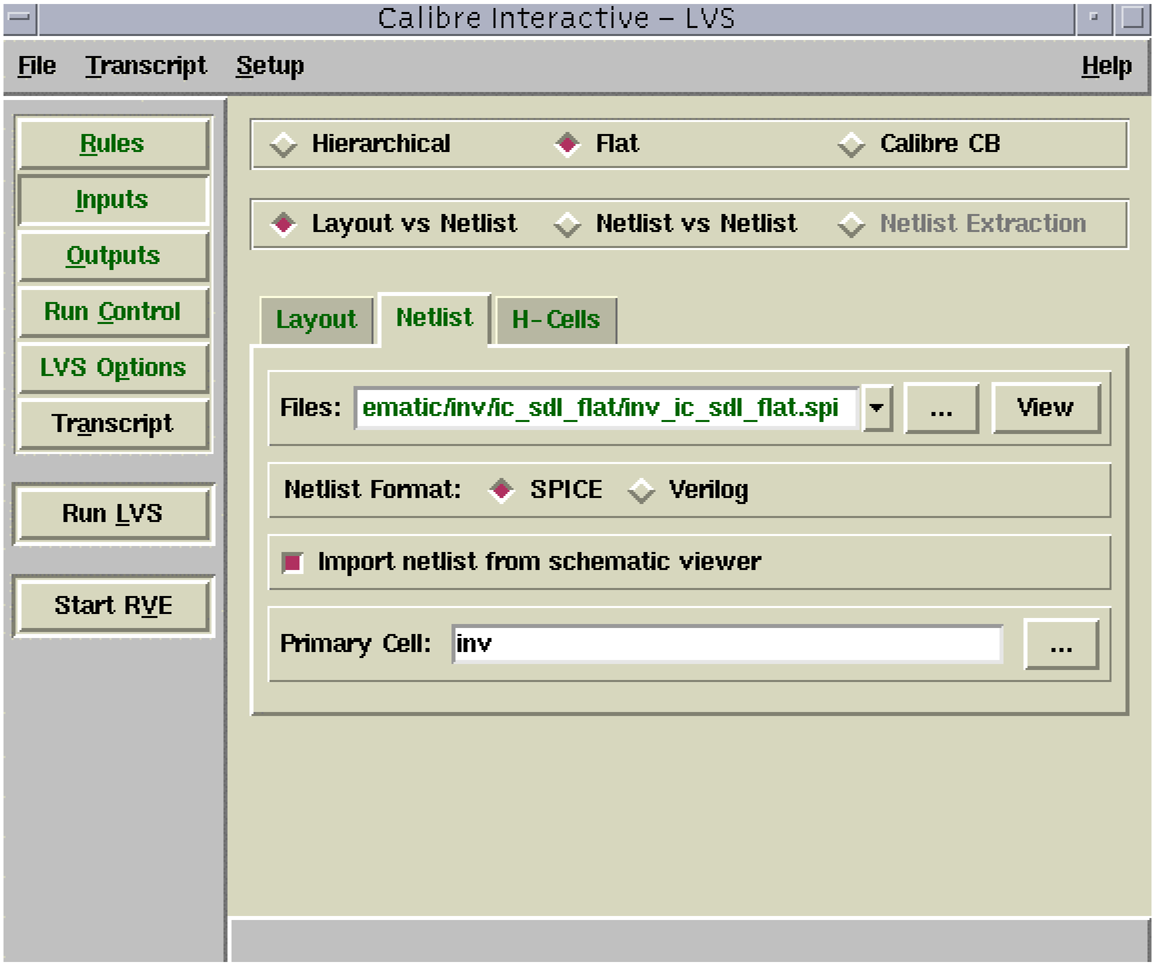


Step2: The Calibre Interactive – LVS window should pop up. Make sure Flat and Import layout database from layout viewer is selected while the Inputs and Layout tab are active as shown .



Step3: Select Rules button and navigate to $MGC_DESIGN_KIT/rule_deck/LVS/lvs.ctrl load Calibre file.

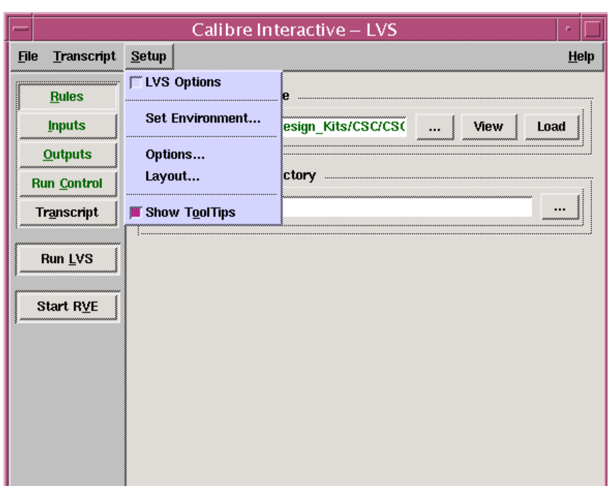
Step4: Select Inputs button and Netlist Tab and make sure that Flat and navigate to Import netlist as shown.

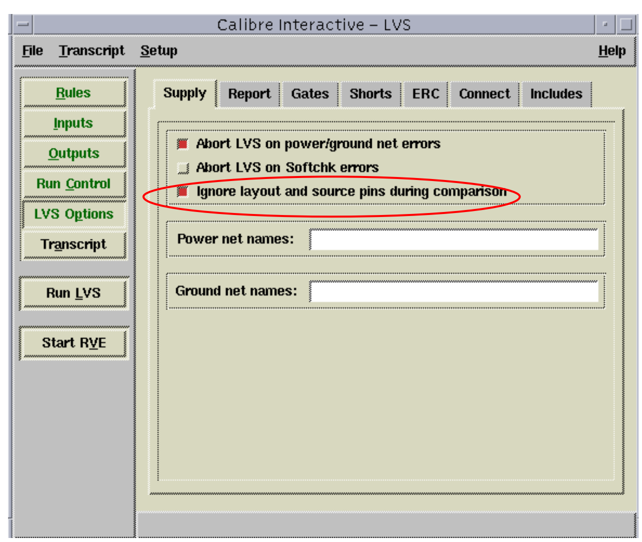


El camino para el Netlist es $HOME/schematics/... Donde usted guardo su esquematico.

Step5: Select Setup the LVS Options. Select Ignore layout and source pins during comparison.

Si tiene problemas (no inv in database), escriba '.' (sin comillas) en Primary Cell y ejecute de nuevo





Step6: Calibre LVS RVE window will popup and you should see results similar to this.