Tutorial for IC Design Using Mentor Graphics™

Design and simulation an inverter

Wei Wang and Dongsoo S. Kim

1. Introduction

In this tutorial, we are going to create transistor level schematic of an inverter by using the Design Architect of Mentor Graphic. The final schematic of the inverter will look like the figure as shown in Figure 1-1. It also covers the simulation of the inverter.

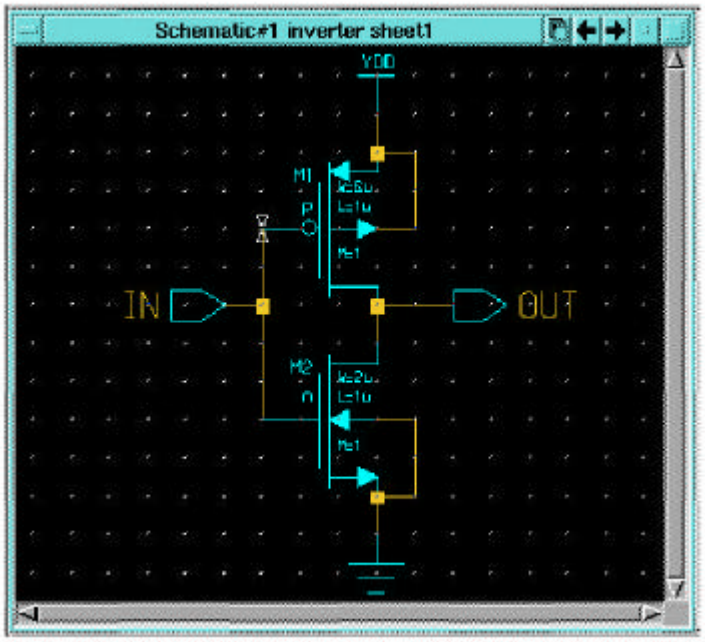

Figure 1-1. Transistor level schematic of an inverter

2. Preparation

Check you have a correct configuration for using Mentor Graphics tools by typing as :

```
% module list
Currently Loaded Modulefiles:
  1) base     2) tex      3) mentor
```

If you have don't have the mentor in the list, you need to type the following to load the Mentor Graphics tools:

```
% module load mentor
```

Anyhow, you must have `MyMentor` directory under your home directory, which will contain an html file for redirecting to ECAD online community of ECE department.

3. Starting Design Architect

1. Invoke **Design Manager** by typing:

```
$ adk_daic &
```

The **Design Manager** window will appear as shown in Figure 3-1.

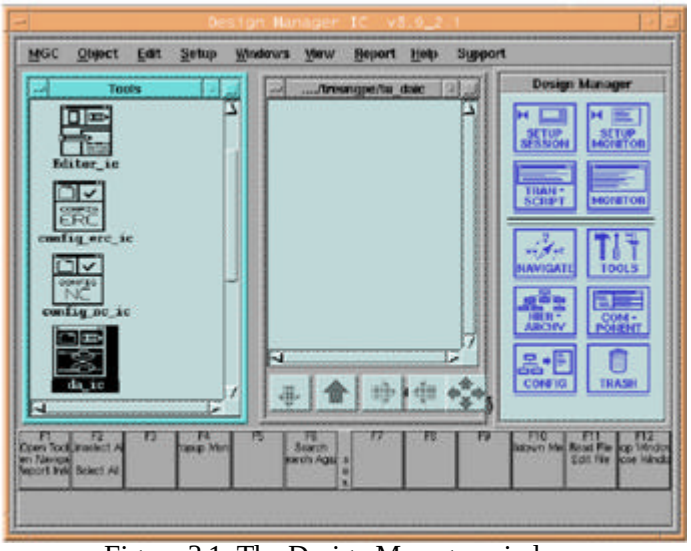

Figure 3-1. The Design Manager window

3. Double click **da_ic** icon in the 'Tools' window and wait for a couple of seconds, the **Design Architect** window will appear. Select **Schematic** in the **Session Palette** window. A dialogbox pops up as shown in Figure 3-2.

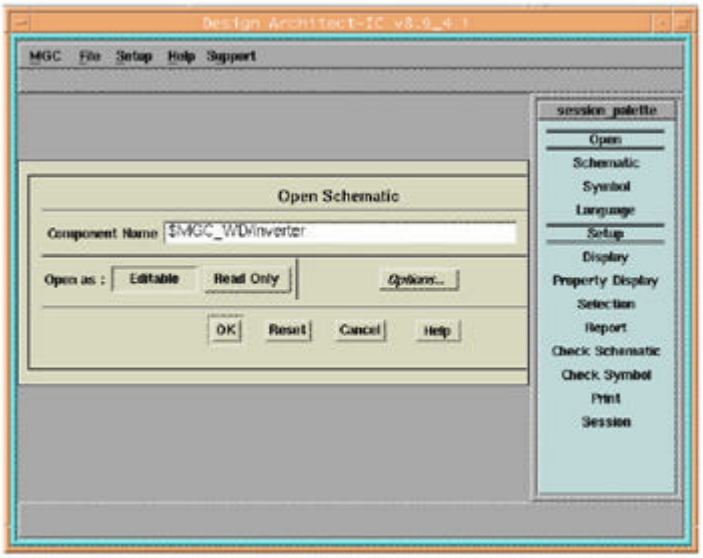

Figure 3-2 Dialog box for open new sheet

4. In the box labeled **Component Name** , type the schematic name **inverter** at the end of default path. If you want to indicate the another directory, use the **Navigator**, select the directory name, and type **inverter** at the end.
5. Leave other boxes untouched and click **OK**. For an existing design, you

may use the navigator to locate the component in your directories.

4. Schematic Entry

4.1 Adding components and ports to the schematic

Select the parts for your schematic of the inverter by choosing from a library of components.

1. In the **schematic_edit** palette, select **Library.** Click on **Device Lib** to place the transistors - an NMOS(4-pin) and a PMOS(4-pin).
2. Click on **Generic Lib** to place **Portin**, **Portout**, **VDD**, and **Ground** as shown in Figure 4-1.

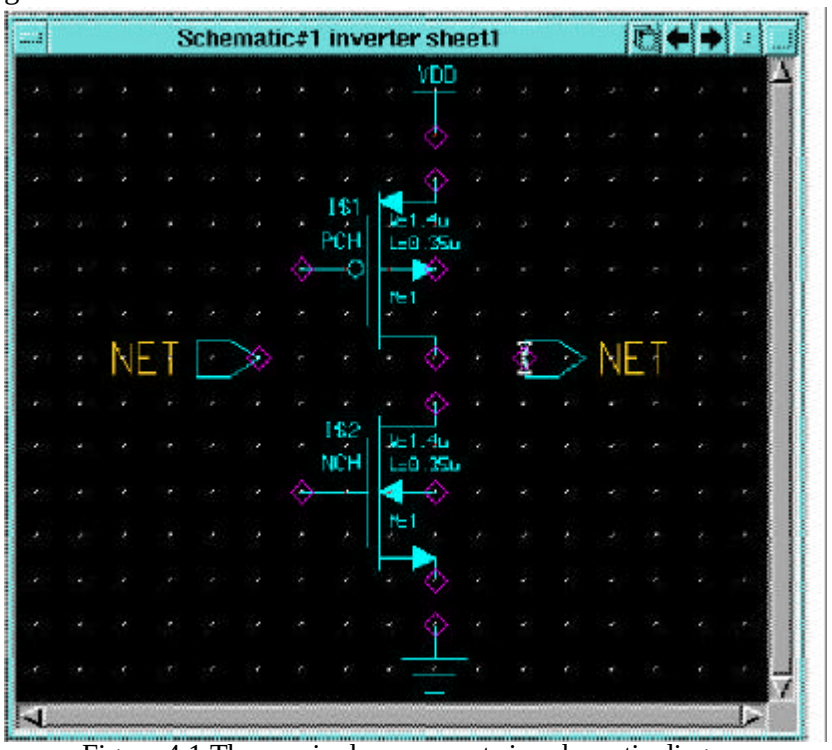

Figure 4-1 The required components in schematic diagram

4.2 Add Text/Change Labels

- To add text on the sheet:
  1. Click **Text** on the right side palette, then select **Comment Text.**
  2. Enter the text in the prompt bar, then click on **OK**.
  3. Move cursor and click to place the text on the sheet.
- To change the text or name of the ports:
  1. Place the cursor on the port name. Press **Shift key and F7** simultaneously.
  2. A **Display Prompt Bar** appears in the bottom of the the schematic window. Input the new name in the **New Value** box, then click **OK** as shown in Figure 4-2.

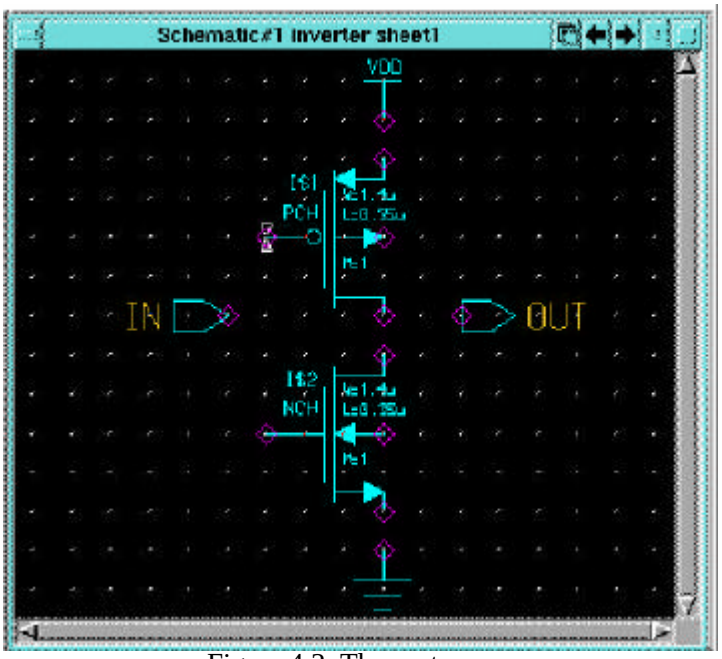

Figure 4-2. The port name.

4.3 Wiring

In the schematic_edit palette, select **WIRE** (or press F3). For adding a wire between two points click once at the starting point and at all intermediate points to define the net route and double click at the end point to complete the routing.

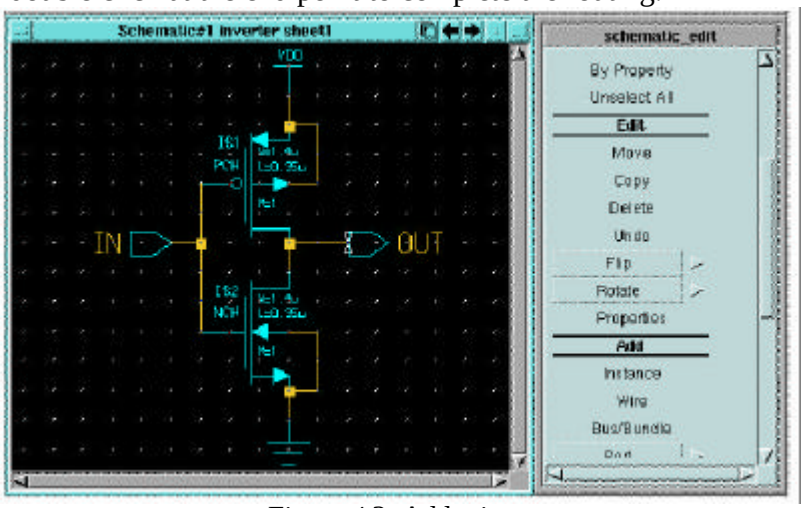

Figure 4-3. Add wires.

4.4 Change the instance name of the components

To change the reference number (INST), select the component, click right mouse button, and then select **Properties** > **Modify Multipl**e. In the box of **Instance**, change the ASIM_MODEL, INST, L, and W values as shown in Figure 4-4.

To change the reference number automatically highlight all components, click the right mouse button, **Instance** > **Alte**r. The reference numbers will be automatically generated by the tool.

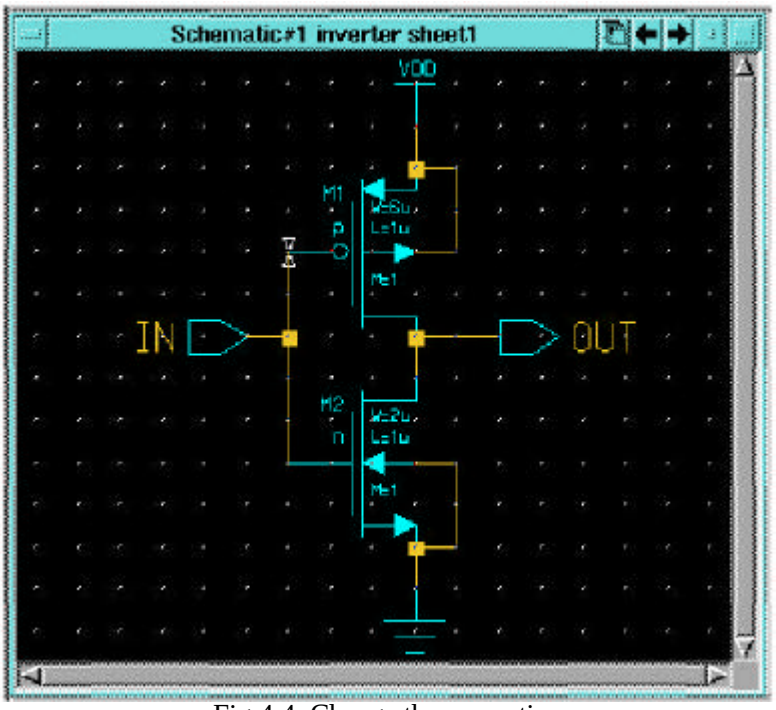

Fig 4-4. Change the properties

5. Check/Save the sheet

1. The schematic you have created needs to be checked for connection errors before it is saved into a file. Press **Check& Save** from the **schematic_edit** palette to check and save your sheet. If your sheet does not pass the check, you cannot get the result of simulation of the design. Check the log if errors are listed.

2. To prepare a **Viewpoin**t for simulation in DA_IC, click **Simulation** in the palette, click **OK,** and then click **EndSim** (To make sure the Netlist is SPICE_Netlister and Viewpoint name is eldonet by click option as shown in Figure 5-1.)

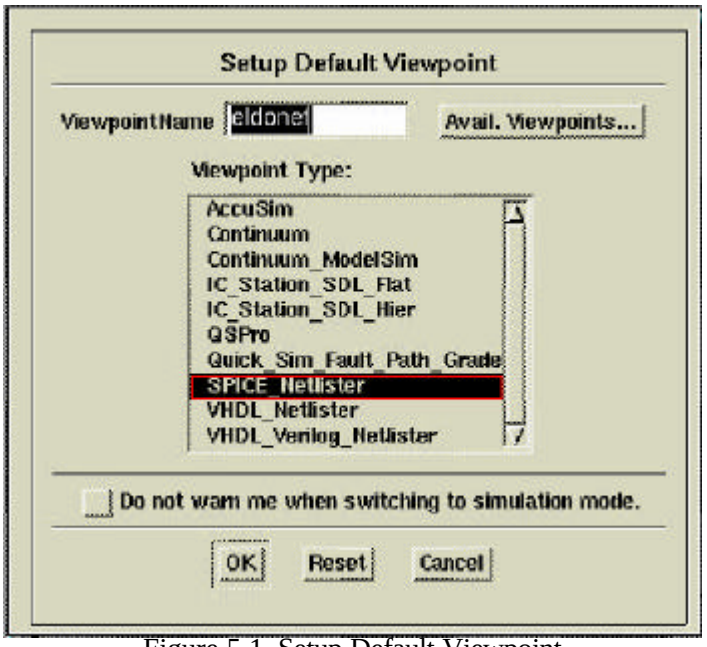

Figure 5-1. Setup Default Viewpoint

6. Generate Symbol

1. After your schematic is completed, you may want to create a block symbol to represent this circuit. This symbol can be used in other schematics to perform the same function as this circuit, and you do not need to draw the same circuit again.
2. Make sure that the schematic is checked and saved before making the symbol. To generate a symbol, select:
Miscellaneous -> Generate Symbol...
from the pull-down menu bar.
3. In **General Symbol** dialog box, select **Choose Shape**, and select **Buffer**, click **OK**, and click **OK**. A **Symbol** is created automatically and is displayed in a new window as shown in Figure 6-1.

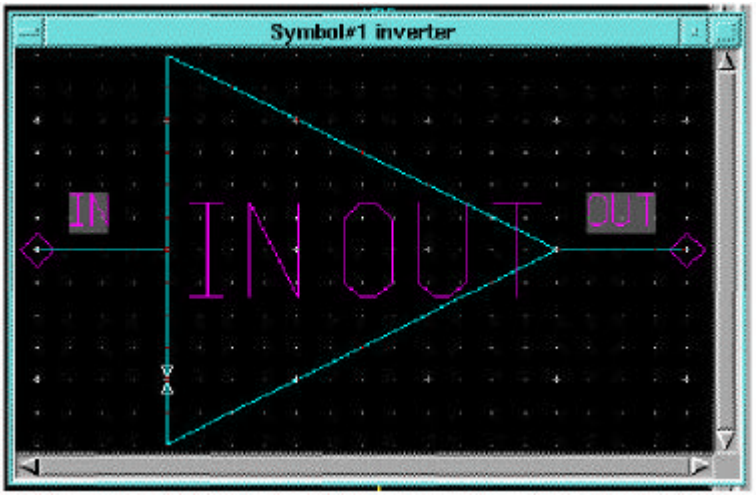

Fig 6-1. The symbol of Inverter.

4. Edit symbol if necessary. Use the same skill as in editing the schematic.
5. To save the symbol, press **Check& Save** from the **schematic_edit** palette to check and save your sheet.

7. Simulation

7.1 Create a test circuit

1. Close your inverter and symbol, open a new sheet, and name it **test_inverter**.
2. Add one instantiation of inverter cell by selecting
Add -> Instrance -> Choose Symbol
Choose **Inverter** and click **OK**, we will get the symbol of the inverter as shown in Figure 7-1.

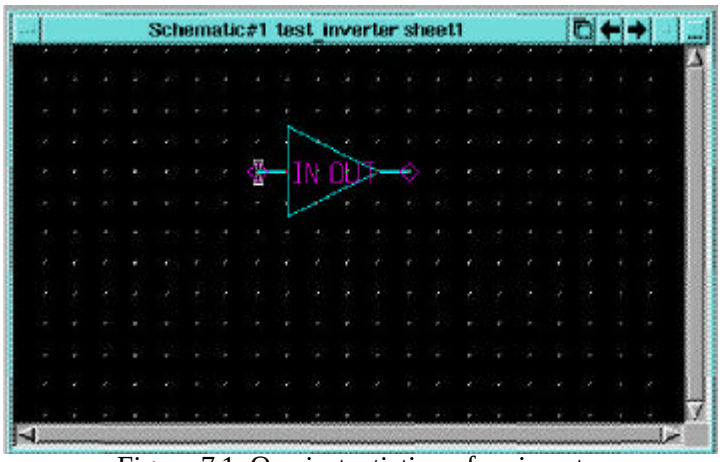

Figure 7-1. One instantiation of an inverter

3. Add **DC source** at the input of inverter, add **Portout** at the output, create **power supply** for the circuit, and change the value of DC source and Pulse source from **1** V to **3.3** V.
4. Check & Save

7.2 Set up the model

1. Click **Simulation** in the **schematic_edit** palette to invoke the simulation, and click **OK**.
2. Include TSMC0.35 m BSIM3 model, click **Library** in **schematic_sim_palette**, and type **'$ADK/technology/accusim/tsmc035.mod'** in **Library path** and then click **OK.**

7.3 DC Analysis

You will run a DC sweep of the input and view the results.
1. Cliick **Analysis** in **Setup Sim** header inside **schematic_sim_palette**, select **DC** and click **Setup**.
In the setup box that appears, type '**IN**' for '**Voltage Source**', '**0**' for '**Start**' box and '**3.3**' for '**Stop**' field, click **OK**, and then click **OK** as shown in Figure 7-2.

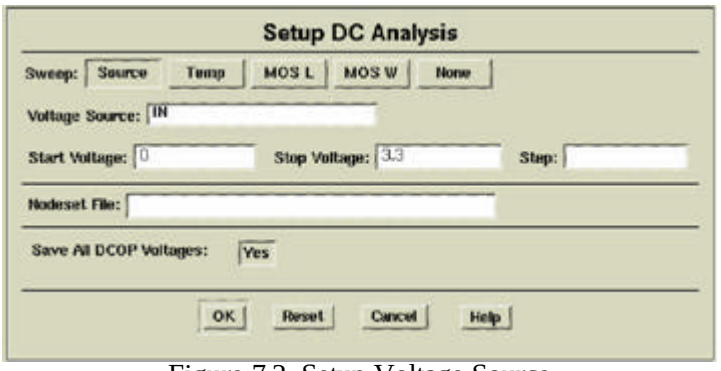

Figure 7-2. Setup Voltage Source

2. To view the input and output of your inverter, highlight **IN** and **OUT**, click **Setup Result**s>**Plots**. Select **D**C, select **Voltag**e, and then click **OK**
Click **Setup Result**s>**Probe**s. Check **D**C, select **Voltag**e, and then click **OK**
3. To run the simulation, click **Netlist Write** and **Simulation->Run.**
4. To view the results of your dc sweep, click **Results>View>Invoke Viewe**r. Click **Result> Xprobe>Voltages-Default**.
5. Drag cover input port and output port.
6. Press the right mouse bottom and from the popup menu select **Chart >Add cursor.**
Then type **Cursor name: T0 > OK**
Press the left mouse button on the cursor and drag the cursor to the voltage point you want
to measure.
To obtain another cursor to measure between points, press the right mouse bottom and from
the popup menu select **Chart >Add cursor.** Then type **Cursor name: T1 > OK**
Press the left mouse button on the cursor and drag the cursor to the second voltage point you
want to measure.
7. Another method to view the results of the analysis is by listing the signal values. Select **Vin Vout** and the right mouse bottom, and select **LIST** . A LIST window will appear
with a list of values for Vin and Vout.
8. Print your waveform by selecting the trace window, and selecting **File > Print Chart with ICPrint** from the menu. Type `laser` for the printer name.

7.4 Transient Analysis

In this section, you will provide a pulse at the input and view the results.
1. Instead of using DC Source by using Pulse Source, connect to the input of inverter as shown in Figure 7-3.

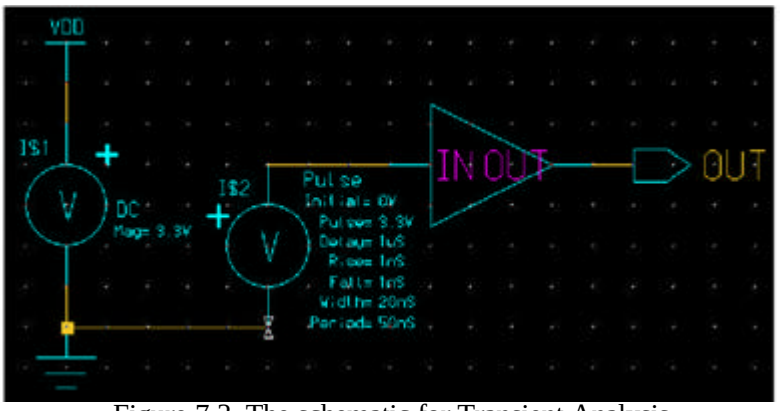

Figure 7-3. The schematic for Transient Analysis

2. Click **Setup Sim** ->**Analysis**, select **Transient**, click **Setup**. In the setup box that appears, type **10u** for **Stop Time** field and **50n** for **Time Step** box. Then click on **OK**.

3. To view the input and output of your inverter, highlight **IN** and **OUT**, and click **Setup Results**>**Plots**. Check **TRAN**, select **Voltage**, and then click **OK.**

Click **Setup Results**>**Probes**. Check **TRAN**, select **Voltage**, and then click **OK**

4. To run the simulation, click **Netlist Write** and **Simulation->Run.**

5. View the results by clicking **Results>View>Invoke Viewe**r. Click **Result> Xprobe>Voltages-Default**as shown in Figure 7-4.

6. You can probe, measure and view your results as in the previous section.

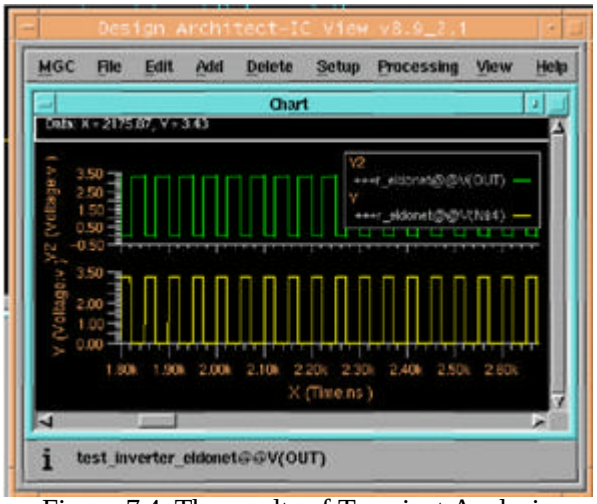

Figure 7-4. The results of Transient Analysis

7.5 DC Operating point Analysis

In this section, you will find the DC operating point of several nodes in the circuit and look at the transfer function of the input.

1. **Setting up the simulation**

Click **Setup Sim** ->**Analysis**.

Select **DCOP** in the dialog box that appears,then click **OK**

2. **Running the simulation**

Click **Netlist Write** and **Simulation->Run.**

3. **Viewing the results**

The next step is to view the results of your simulation run.
To see all the DC voltages and currents in your schematic,
Click **DCOP_Show >Both**