I. **Introduction**

This document contains a step-by-step tutorial for creating an inverter in the Mentor Graphics tool Design Architect. It covers entering the schematic, creating a symbol, and simulation of the inverter.

II. **Setup and Preparation**

The set of directives listed below is applicable to users of the Engineering Design Center at Santa Clara University. If you are working in a different environment please check with your system administrator.

The steps below are necessary the first time you change the settings in your .profile or .cshrc file.

1. Add the following lines in your .profile:
   ```bash
   . /usr/local/scripts/setup.mentor-2001.2.sh
   ``
   Remember to execute
   ```bash
   $ . .profile
   ```

2. If using C-shell add the following lines in your .cshrc:
   ```bash
   source /usr/local/scripts/setup.mentor-2001.2.csh
   ``
   Remember to execute
   ```bash
   $ source .cshrc
   ```

3. Create a working directory by typing:
   ```bash
   $ cd
   $ mkdir projects
   $ cd projects
   $ mkdir tutorial
   ```
III. Starting Design Architect IC

1) Invoke 'Design Manager' by typing:
   `$ dmgr&`

The window shown below will appear.

2) Double click on `da_ic`. The window shown below will appear.
3) Open a schematic sheet following the below selections.

In the palette menu select **Schematic** and a window as shown below will appear.

In the Open Schematic window select **Options**

In the Open Schematic Options window that appears select **New Sheet**. Then click **OK**.

In the Open Schematic window use the **Navigator** to go to the directory you created and type the schematic name **inv**. Then click **OK**.

Note: You will see an empty sheet in the **Design Architect** window. This sheet is where you will draw your schematic by placing the parts for your circuit and wiring them together.
IV. Schematic Entry

We will be drawing the schematic for the CMOS inverter shown in the Figure below.

1) Adding components and ports

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

In the palette menu select **Library → IC Libraries**

Click on **Device Lib** to place the transistors - an NMOS(4-pin) and a PMOS(4-pin). Resistors(ideal) and capacitors(ideal) can also be obtained from this library.

Click on **Generic Lib** to place Portin, Portout, VDD, and Ground.

The **Sources Lib** will provide AC and DC sources which be later used for simulation.

2) Wiring the components

Press **F3** to wire up the components according to the circuit diagram in Figure 2. Pressing F3 activates the wiring command. Clicking on **Cancel** in the prompt bar or pressing **Esc** on your keyboard disables the wire command.

You can also add wires by using the palette menu and selecting ADD – WIRE
3) **Changing attributes and values of the components**

To change the value(INSTPAR) of the components, select the value of the component or move cursor beside value of component and press `SHIFT F7`. Change the values for the sizes of the transistors of your circuit.

To change the labels of the ports, select the net and then use the same method to change the label of the net.

4) **Changing the reference number of the components**

To change the reference number(INST), select the component, click right mouse button, **Properties > Modify Multiple**. In the box that appears change the INST value.

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

5) **Checking and saving the schematic sheet**

Press **Check& Save** from the palette menu to check your sheet. If your sheet does not pass check, you cannot simulate the design. Check the log if errors are listed.

6) **Creating a viewpoint**

To prepare for simulation in DA_IC, a Design Viewpoint needs to be created. The tool automatically creates the viewpoint. You will use the viewpoint during layout of the circuit.
V. Creating a Symbol

Hierarchical design allows you to instantiate lower level cells (circuits) into upper level cells to create a tree structure. Since, at higher levels, we really don't need to see the detailed transistor-level description of the base cells, we create symbols for them.

1) Automatically generating the symbol

To automatically generate a symbol in DA_IC, from the main menu select Miscellaneous > Generate Symbol
Do not change any of the options in the dialog box and click the 'OK' button to generate a symbol for the cell. The generated symbol will be a simple rectangle with two pins, IN and OUT.

2) Drawing the symbol

You may want to make your inverter symbol look like an inverter.

Open a symbol sheet following the below selections.
In the palette menu select Symbol and a window as shown below will appear.
In the Open Symbol window select Options
In the Open Symbol Options window that appears select New Sheet. Then click OK.
In the Open Symbol window use the Navigator to go to the directory you created and type the schematic name inv. Then click OK.

In the sheet that is opened, you will draw your inverter symbol.

Click DRAW \rightarrow ADD POLYLINE and draw a triangle.
Click DRAW \rightarrow ADD CIRCLE to draw a circle at the front of the triangle.
Press F2 to unselect these shapes.

3) Adding the pins

To add pins to your inverter, click on PIN in the ADD menu from the palette.
Select an optimal height for the pin name. Enter the pin name, appropriate pintype and pin placement. You can move the pins and their texts to the appropriate locations.
To move text, click TEXT \rightarrow CHANGE VALUE, select pin names and the click MOVE to move pin names to appropriate position.

4) Checking and saving the symbol

To check the symbol for errors, from the main menu select Check > With Defaults
You can ignore the warnings about the properties not being on the interface. If you have any other errors fix them before moving on.

Save your symbol by selecting File > Save Symbol > Default
Close the symbol window.
VI. Making a Simulation Schematic

1) Creating a test circuit

To simulate your circuit in DA_IC, you should create a test circuit of your inverter. Open another sheet and name it test_inv.

Add one instantiation of your inverter cell inv by clicking left mouse bottom and selecting ADD->Instance- Choose Symbol and select inv from the list of symbols.

Click on Sources Lib to place DC source at the input.

Click on Sources Lib to place DC source and connect it to VDD. Setup VDD=3.3.

Check the schematic and close it.