I. Introduction
This document contains a step-by-step tutorial for simulating an inverter in the Mentor Graphics tool Design Architect. It covers simulation of the inverter and analyzing and viewing the results.

II. Starting Design Architect IC Simulation
1) Invoke 'Design Manager' and invoke da_ic as you did in the previous tutorial.
2) Open the simulation schematic sheet you had created with the symbol of your inverter.
   In the Open Schematic window you can use the Navigator to go to the directory and the schematic you want to simulate.
   Modify the input to be a time varying source so as to run transient analysis.
   Click on Sources Lib and place a pulse source at the input.
3) Involve simulation by clicking Simulation from the palette menu.
4) You will be using the `eldo` simulation kernel in DA for your simulations.

III. Setting up the models
1) Adding the transistor models
   To include TSMC0.35 µm BSIM3 model, click on Setup Other -> Library
   Type `$ADK/technology/accusim/tsmc035.mod` in Library path and then click OK

IV. DC Analysis
In this section, you will run a DC sweep of the input and view the results.
1) Setting up the simulation
   Click Setup Sim -> Analysis.
   Select DC. Click Setup.
   In the setup box that appears, type `Vin` for `Voltage Source`, '0' for 'Start' box and '3.3' for 'Stop' field. Then click on OK
2) Setting up the signals to be probed
You need to setup the signals to be plotted and probed in order to view the results after simulation.

To view the input and output of your inverter, highlight Vin and Vout and then,

Click Setup Results>Plots. Check DC, select Voltage, and then click OK
Click Setup Results>Probes. Check DC, select Voltage, and then click OK

3) **Running the simulation**

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

4) **Viewing the output netlist**

Select **Simulation View Log** from the palette menu to see netlist and analysis setup.

5) **Viewing the results**

To view the results of your dc sweep,

Click **Results>View.Invoke Viewer**.

Click **Result> Xprobe>Voltages-Default**.

A chart will appear. A signal list window will also appear on the left.

Highlight V(Vin) and V(Vout). Press the right mouse bottom and select **Chart> Selected**.

6) **Using the viewing tool and making measurements.**

The next step is to add and move cursors and get familiar with the trace window.

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) **Viewing the listed output**

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) **Printing your plots**

Print your waveform by selecting the trace window, and selecting

**File > Print Chart with ICPrint** from the menu.

Type `laser` for the printer name.
V. Transient Analysis

In this section, you will provide a pulse at the input and view the results.

1) Providing the input
In your schematic define vin to be a pulse

2) Setting up the simulation
Click Setup Sim -> Analysis.
Select Transient. Click Setup.
In the setup box that appears,
type '10n' for 'Time Step' box and '10U' for 'Stop Time' field. Then click on OK.

3) Setting up the signals to be probed
You need to setup the signals to be plotted and probed in order to view the results after simulation.
To view the input and output of your inverter, highlight Vin and Vout and then,
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) Running the simulation
Click Netlist Write and Simulation->Run.

5) Viewing the output netlist
Select Simulation View Log from the palette menu to see netlist and analysis setup.

6) Viewing the results
To view the results of your dc sweep,
Click Results>View>Invoke Viewer.
Click Result> Xprobe>Voltages-Default.
A chart will appear as in Figure 1. A signal list windows will appear on the left.
Highlight V(Vin) and V(Vout). Press the right mouse bottom and select Chart> Selected.

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

VI. 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**