

Introduction to **Mentor Graphics** Design Architect and Accusim

1. Introduction

This tutorial has been devised to walk you through all the steps involved in the design and simulation of a resistive load NMOS inverter using Mentor Graphics CAD tools. Before invoking the Mentor Graphics tools, it is essential to first set up the environment under which these tools will be used throughout the semester.

2. Getting started

The following is an overview of the steps involved in creating and simulating a resistive load NMOS inverter in Mentor Graphics Design Architect and Accusim SPICE model simulators. We first need to draw the schematic and then simulate it.

(a) Schematic capture

The circuit representation is captured by the Mentor Graphics Design Architect (DA) program.

(b) Circuit Simulation

The design that was created in DA is now translated into a SPICE model and simulated using the Mentor Graphics Accusim program.

2.1. Setting up environment variables

Before running any Mentor tool, you have to setup certain environmental variables in your .cshrc file.

2.2. Schematic Capture Using DA

After making sure that you are in your class directory, invoke DA by typing the following at the Unix shell prompt:

```
%da &
```

A window with the title Design Architect will open. This window can be maximized by clicking the **LMB** on the square button at the extreme top right corner of the DA window. In the Menu palette on the right part of the window, click the **LMB** on the OPEN SHEET icon. A pop up dialog box will appear, prompting you for the name of the component. Use the mouse to highlight the component box (red border should appear around the box) and enter the name of the component, say introlab. Then click the **LMB** on the OK button. A schematic window will appear and you are ready to edit a schematic.

Note:

Alternately, you can use the **LMB** to click on **File** at the top navigation bar. You will see a pull down menu. Click on **Open Sheet>Sheet**. Thereafter, the component naming procedure is similar to the previous case.

The circuit we need to create is shown in figure 1. This is a ratioed logic NMOS inverter with a resistive load. You need to assemble the components, connect them together in the circuit and modify certain properties.

First, we need to get the NMOS transistor. Go to the **Libraries** menu at the top of the menu bar and select **MGC Analog Libraries>Display Libraries Palette**. From the palette menu on the right hand side of the window, choose **Generic Parts**. If the scroll bars are hidden, then click on the **RMB** and select **Show scroll bars**. In the palette, a menu will appear showing various generic circuit elements. For our lab, we also require a resistor, an input port, an output port, power supply and ground.

To instantiate a resistor, use the **LMB** to click on the **RESIST** symbol. A dialog box will appear at the bottom of the schematic window and if you move the mouse in the schematic window, a ghost image of the resistor will be seen. Position the resistor appropriately in the schematic window with the mouse and click the **LMB**. Instantiate capacitance, power supply and ground in a similar way by clicking on **VCC** and **GROUND** respectively. For this lab, we shall use an NMOS transistor with 4 terminals - source, gate, drain and substrate. The substrate lead is useful to simulate body effects. Instantiate the NMOS transistor by clicking on **NENH4** symbol and position in the schematic window.

The next step is to connect all these components. Click the **RMB** on the library palette menu on the right hand side of the schematic window and select **Display Schematic Palette**.

Click the LMB on the **ADD WIRE** icon in the Menu Palette. A dialog box appears at the bottom of the window with the corresponding command name. To draw the nets, click the LMB at the desired starting point in the schematic window and drag the mouse till the destination and double click the LMB at the end point. You can draw as many different nets as you can since the DRAW NET command stays in effect until it is cancelled by clicking the LMB on the CANCEL button in the dialog box at the bottom of the screen. After completion, the schematic looks like Figure 1.

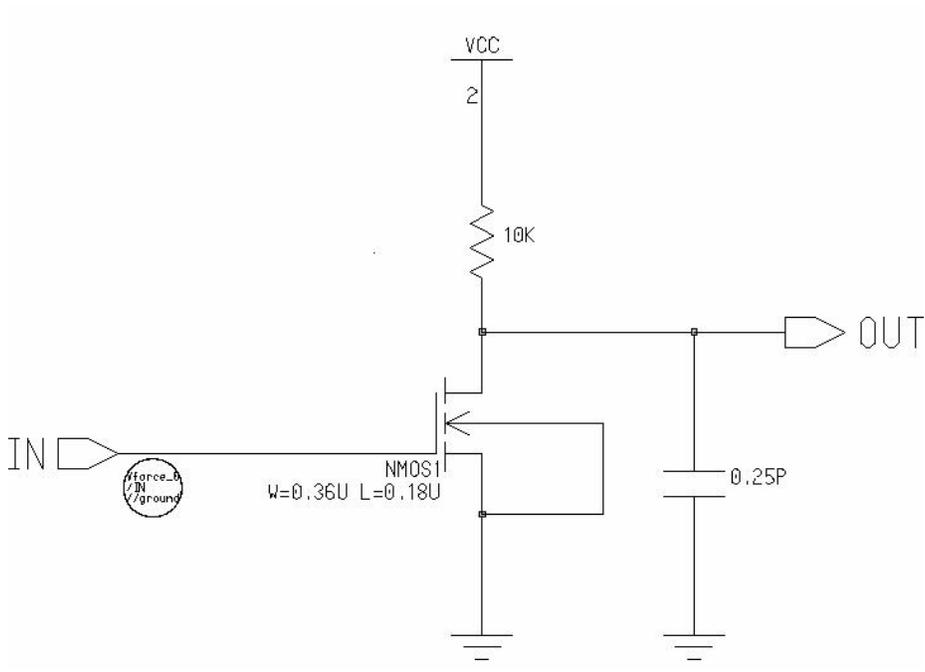


Figure 1. Schematic of NMOS inverter with resistance load

The next step is to define input and output ports for the circuit. This can be done by selecting the **PORTIN** and **PRTOUT** symbols in the Generic parts library. Place the ports in the appropriate nodes and connect them.

Now we need to alter the properties associated with the nets and instances. For instance, you can see that both the input and output ports have the word **NET** written near them. This is the value of the property net associated with these wires. These default values need to be altered according our requirements. To change the name of a port, say the input port to **IN**, place the cursor over the word **NET**. Then press **Shift+F7**. A dialog box will appear at the bottom of the window prompting you for a new value for the property. Type **IN** and click the LMB on the OK button. The input port should now be named **IN**. If now you hit **F2** function key, the port is unselected and the name **IN** appears in yellow. Similarly, the output port name and resistor values can be changed appropriately.

Using the above technique, property values of only one instance can be changed at a time. To change all the property values at a time, you will need to do the following. First, unselect everything in the schematic window by hitting **F2** key on the keyboard. Place cursor in the schematic window and click on RMB. A pop-up menu will appear. Choose **Other menus > Instance menu**. Now a new pop-up menu appears. From this window, select **Properties>Change text values**. A dialog box will appear in the window prompting you to enter the text values that need to be changed. Enter all the text values that you want to be changed(e.g. **IN,OUT** etc.). After you have entered all the values, click on **OK**.A small dialog box will appear at the bottom of the schematic window and message will appear telling you what the next changed text value is. Click the LMB on the default text that has to be changed to this value. For example, if the first text value that you wish to change is the resistor value to **10K**, click on the default resistor value. The resistor value should have now changed to **10K**. Continue this until all the text values have been changed. You should pay attention that here **Vcc=2** instead of the default value of **5**. Please remember to change it.

The next step is to change the properties of the NMOS transistor. Click the LMB on the transistor. This selects the transistor, and now click the RMB. A pop-up menu will appear. From the menu, select **Properties>Modify multiple**. A new window will open enlisting the properties of the component. Change the text value of **ASIM_MODEL** and **MODEL** to **NMOS1**. This refers to the NMOS model file located elsewhere. Next, you need to change the **INSTPAR** property. This property refers to the dimensions of the MOS transistor. Change the **INSTPAR** property as follows:

$$W=0.36U \quad L=0.18U$$

This implies that the width of the MOS transistor is 0.36 microns and its gate length is 0.18U.

Note on SPICE models and levels:

SPICE models are a set of parameter values which the circuit simulator uses to model the physical behavior of transistors. To simulate a MOS transistor, several models of varying degrees of accuracy and computing efficiency are available. Depending upon the model chosen for simulation, the MOS transistor can be simulated for many of its parasitic and second order effects. The model file includes process parameters and other parameters to compute the parasitics. In this course, we shall be using the TSMC,0.18U level 53 model to simulate in Accusim. Here **NMOS1** is the name of the NMOS transistor model. Similarly, the PMOS model is named **PMOS1**. So whenever you encounter a PMOS transistor in the design, make sure that you change its **ASIM_MODEL** and **MODEL** property to **PMOS1**.

The final step is to do a Design check. In the menu bar on the top of the DA window, you will see an entry **Check**. Click the LMB on this and a pull down menu will appear. Select **Sheet>With defaults**. If all the design steps mentioned above have been correctly

implemented, you should not get any warnings or errors in the report generated by the check. If you do get errors, talk to your GSI. Finally, from the File menu at the top of the window, you can save the schematic by doing the following:

File > Save Sheet > Default Registration

This completes schematic capture in DA.

2.3. Design Viewpoint

A design viewpoint is needed to translate the schematic design into a form recognizable by the SPICE analog simulator. This can be created within DA as follows. From the Libraries menu, choose

Libraries>MGC Analog Libraries > Display Libraries Palette

This brings up another menu on the right hand side of the schematic window. Click on the **Analog M/S utilites**. A new menu opens up. Click on **Create viewpoint**. The Target simulator should be Accusim II. The design viewpoint name should be **default** and the Back-Annotation name is **sim_ba**. The netlistfile should be pointing to the location of the SPICE model file. Click on the navigator and select the correct path where the model file is located.

Press Ok and Accusim viewpoint is created. This will take few seconds to a couple of minutes depending upon your design. A message at the bottom of the DA window saying "Viewpoint created successfully" indicates that an Accusim viewpoint has been successfully created.

If you list the files in your working directory, you should find the files: **introlab.mgc_component.attr** and the directory **introlab**. Do not move or alter these files using Unix shell commands. To copy or move a design, use the **MGC>Design Management** menu from within the DA program.

2.4. Analog simulation

Analog simulations are performed in Accusim. You invoke it for this design with the command:

```
% accusim introlab &
```

The schematic design sheet for the design will appear in a window. You can use **Shift+F8** key to VIEW ALL the schematic. There is also another window that reads "Simulation: NOT RUNNING". It also mentions the type of simulation performed, the current simulation temperature and nominal temperature (TNOM). You can also see that the simulator usage MODE reads "Schematic design".

First let us obtain the Input-Output waveform for the circuit. To do this, we need to do a TIME MODE analysis. The order in which we shall perform the scheme of things is:

Setup analysis-->Add Force-->Add Keeps-->Add signals to be traced ---> Run simulation.

Follow the steps to run a timing simulation.

2.5. Setting up Analysis

(a) Click on the **SETUP ANALYSIS** button in the palette. You will be prompted for the type of analysis to be performed. Select **Transient**.

(b) Select time step as **0.2N** and Stop time as **11N**. Click OK.

2.6. Adding forces

(a) Now you need to force an input. Click on **ADD FORCE** button in the palette.

- (b) You will be prompted to specify the kind of forcing signal. Select **voltage**.
- (c) Then select the forcing signal as **IN**.
- (d) The Force type is to be selected as Pulse with following values: (see figure 2)
Initial Value:0 Pulsed Value:2 Delay Time:2N Rise Time:1N Fall Time:0N
- (e) Select Width & Period in the Specify column with Pulse Width as **8N** and Period as **11N**.

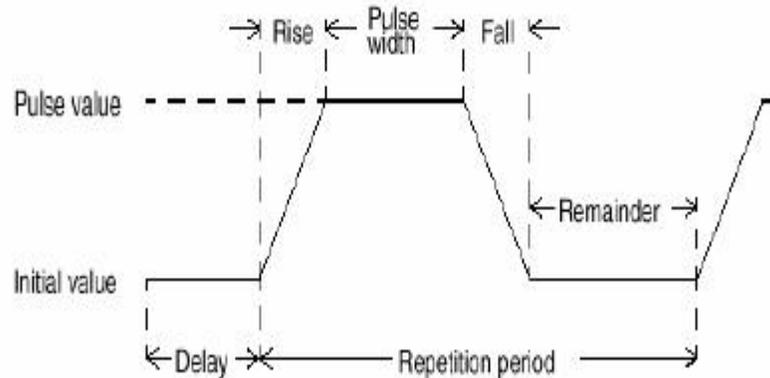


Figure 2. Waveform

2.7. Adding Keeps

- (a) Click on **ADD KEEPS** and Select **All** and click OK. (If the design is complex, then we need to selectively keep the nodes that we are interested in)

2.8. Adding signals to be traced

- (a) **Select** the instance connecting the OUT and IN ports and click on **TRACE**. You will see a waveform window with the empty **OUT** and **IN** waveforms.

Now click on **Run**. You will see that the Simulation status changes to "Running" for a few seconds and goes back to its original NOT RUNNING status in the other window. You will have the output waveforms displayed like the one in figure 3.

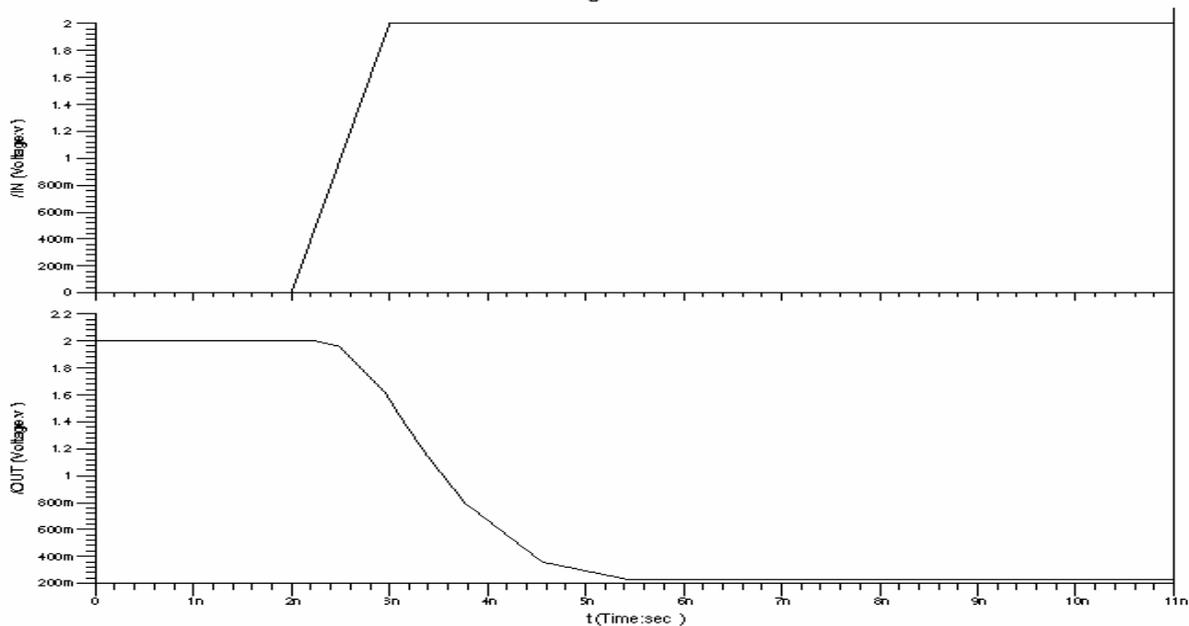


Figure 3. Input-Output waveform of the resistive load NMOS inverter

Exercise #1:

Please print out this diagram either by **Snapshot** or by the way taught at the end of this tutorial. Label the 7 parameters from 2.6(d) and 2.6(e) in your final output diagram.

2.9. Editing in the waveform window

You can edit the text, add cursors, change titles etc in the waveform window. Click the RMB and select **Chart>Hide legend**. This hides the waveform legend. Hold RMB again and select **Chart>Change Chart Title**. It should read Analog Trace by default. Now a small dialog box will appear at the bottom of the window. Type the title as Input-Output waveform of the resistive load NMOS inverter for R=10K and click OK.

Next we need to do a sweep analysis. We need to find out the role of the resistance in the circuit. To do this, do not change the ADD FORCE or SETUP ANALYSIS options from previous simulation.

2.10. Sweep analysis

Follow the instructions given below:

- (a) Click the LMB on the resistance. The resistance will be **selected**.
- (b) From the top navigation bar, Click on **Run**. A pull down menu appears. Select **Sweep Runs>Property on Instance**.
- (c) Since you have already selected the resistance, it displays the instance name of the item selected as /\$X where X is any numeric value.
- (d) Now click on **Select Property** and select **instpar** property (It should be equal to **10K**, the current resistor value). Click **OK**.

(e) Select **Sweep** and **Linear**. In the From column enter **5K** and enter **20K** in the To column. Step size is **5K**.

(f) Select **Mag** in the "For AC analysis, plot" option.

(g) To observe the output waveforms, type **/OUT** in the "Chart these signals when sweep completed".

The simulator will do the analysis and plot the output waveforms as resistance value is swept. What do you infer from the results? Does your waveform look like figure 4?

2.11. Adding cursors

To add cursors, click the RMB in the waveform window. Select **Chart>Add** cursor. A dialog box appears at the bottom of the screen. Click on **click +** and place the cursor at the appropriate location and click the LMB. If you click on the cursor, it becomes fluorescent green in color. You can now move the cursor to any desired location.

Transient Sweep Analysis for R=5K, 10K, 15K and 20K (W/L=2)

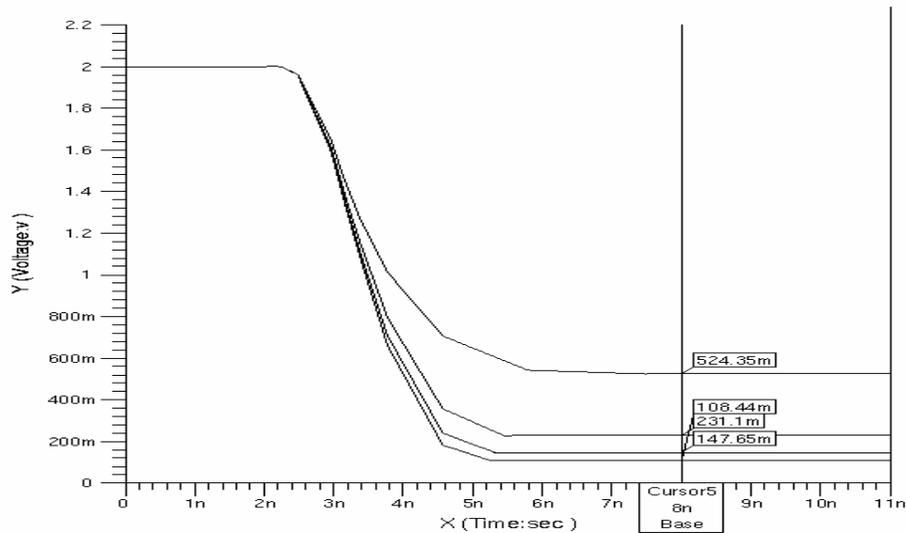


Figure 4 . Sweep analysis for R=5K,10K,15K and 20K (W/L=2)

Exercises #2:

Please hand in the Fig. 4 output from your own simulation exactly labeled as above.

How does the resistance influence the output signal (delay time, output voltage, fall time etc.)?

2.12. Effect of W/L on the transients

We are also interested in the effect of W/L ratio of the NMOS transistor on the transients.

- (a) Go back to DA and open up the schematic.
- (b) Select the NMOS transistor and click the RMB. **Select Properties>Modify multiple.**
- (c) Change the W from 0.36U to **0.72U**. Check the schematic and save as default registration.
- (d) Now come back to accusim window. On the right hand side palette, click on Design **Change** and click LMB on **load DA change**. This loads the changes incorporated in DA to accusim window.
- (e) Click on **Run**. Select **Sweep Runs>Property on instance**. Make sure that all the fields are properly filled and hit OK.
- (f) Observe the waveform.

2.13. More on editing in the waveform window

To add or change text values, click the RMB. **Select Chart > Add Text**. Type the text and hit OK. You can place the text, increase its size in the waveform window.

3. Printing in Mentor Graphics

You can print the various charts by following the steps listed below:

- (a) Make sure that the window of the plot you want to print is selected. If not, click on that window and you can observe that the border of the window will become blue in color indicating that the window is selected.
- (b) The 1st way is to save your plot as a separate postscript file from the main menu **File> Print >Active window>Export Graphics**. You can define the **Path Name** and file name by yourself. Later if you can review and print this plot from Ghostview as **%gv &**
- (c) The 2nd way is to **Snapshot** the window and print from it. Snapshot command can be obtained by clicking RMB at the desktop. Choose **Program...** in the pop menu.

A note on Design Viewpoints

For any schematic you enter in DA, you need to create a design viewpoint only once. If, after the viewpoint is created, you change the schematic, you do not need to create another viewpoint. All you need to do is check and save the schematic. Then if you click the **Design Change** icon on the right hand side of the **accusim** window and click on Load DA change, you can see that the schematic has been updated. A design viewpoint is a configuration file, which along with the schematic sheet tells the analog simulator that this is an electronic design. It seems trivial now and you may be wondering what the use of this file is since you are simulating all these schematics you are creating. But, this may not be the case when you are building a huge circuit with various smaller schematics as sub circuits.

Exercise #3:

Using a pulse waveform with a delay time of 2ns, rise and fall time of 1ns each, pulse width of 10ns and period of 30ns, calculate the t_r and t_f for the 2 cases of W/L from the waveforms. Before simulation, click on **Setup Analysis**, the **Stop Time** changes to 30N.

(1). You can then verify the results from the accusim window by clicking on **Result** on the right hand side palette. Then select **W/F measure** and you can find out rise time and fall time (10% to 90% of the swing) for your simulations. What is your inference from the observed results?

(2) Output a plot for W/L=2 similar to Fig. 3. And using cursors to figure out what are the two delay times corresponding to this pulse. Cursors should be contained in your final plot to hand in. Are these two delays the same? If not, what is the possible reason?