

EE261 Computer Project 1: Using Mentor Graphics for Digital Simulation

Introduction

In this project, you will begin to explore the digital simulation tools of the **Mentor Graphics** package available on the HP workstations in the Department of Electrical Engineering.

Getting Started

If you have not yet worked the “**Using Mentor Graphics: Design Manager and Bold Browser**” tutorial, stop and work that now before going any further.

Once you have worked the tutorial, log into one of the HP workstations in DL517, DL557 or CL137 using the instructions found in the sheet “Logging into the HPs,” available in CL137. Invoke **Design Manager** by choosing `Design tools⇒Mentor Graphics` from the pop-up menu. To get to the pop-up menu in the X Windows environment (and in **Mentor Graphics**), press the right mouse button (RMB). Selecting this menu item will start **Design Manager** and setup your environment. If you have trouble while in **Mentor Graphics**, consult the **Bold Browser** online help tool. The browser can be started by double clicking on the “bold_browser” icon in the Tools window of **Design Manager**.

Beginning the Simulation

In this part, you will examine the characteristics of a simple logic circuit. The circuit has already been entered for you.

If you have not already done so, be sure to maximize the **Design Manager** window now. Look at the top of the Navigator window. (Remember: it’s the one with the arrows at the bottom.) You will see a path that looks like `/rcc4/user6/student/drawn/mgc`. This is the path to your `mgc` directory under your home directory. (Remember? You made that `mgc` directory in the first tutorial.) Write down this path now, you will need it in a minute. Now, using the Navigator, let’s go to the EE261 directory. To do this, click on the “goto” button (the one with four arrows). When the dialog box pops up, click in the blank text box and enter:

```
/usr/local/classwork/ee261/mentor/project1/
```

then press return. You will see a file come up, `261Circuit1`. You will need to select and then copy that file. Select the file and choose `Edit⇒Copy`: from the menu bar or the pop-up menu (RMB). At the destination prompt, enter the path to *your* `mgc` directory (that you wrote down earlier) and press return.

Now, using the `goto` button (four arrows) again, return to your Mentor directory. You should now see an icon called “`261Circuit1`” in the Navigator window. To start the simulator you must “invoke it on a design”. To do that, follow these steps:

1. Select the `261Circuit1` icon.
2. From the pop-up menu, choose `Open⇒QuickSimII`. Remember, it may take some time before you see anything happen.
3. Minimize the **Design Manager** window. When **QuickSimII** comes up it will eventually maximize itself to fill the screen. Once that happens, it will be done loading and you can go on.

At this point, Mentor only knows about the circuit and nothing about how you want to simulate it. Therefore, you will need to set up the simulation before you actually simulate the circuit. First, let’s take a look at the circuit. (See Figure 1) If you look at the palette (far right), you will see eight brown buttons at the top. These are the palette selector buttons and are there in every palette. With them you can switch between the available palettes, each containing a different set of tools. Get to the `SETUP` palette now by clicking on its brown button. To bring up a view of the schematic, click on the `OPEN SHEET` icon in the `SETUP` palette. When the schematic appears, notice the input and output ports (or pads) in the schematic. (They are at the edges of the circuit and look like a box with a point at one end. They have text on one side and a wire on the other). In a simulation you want to watch what electrical activity is happening on various nets (wires). To watch nets you need to tell the simulator to display the activity on them.

First, you need to tell it what nets to watch by selecting them. To select the nets (wires) attached to the ports, click on each port. (The attached net should turn white if it is selected). To unselect a net, click on it again. To unselect *everything* use the `F2` key. You could also use `Unselect⇒All` from the pop-up menu. This method of selecting and unselecting works for all objects in **Mentor Graphics**. Make sure everything is unselected, then select all three inputs (X, Y and Z) and the output (C) by clicking on each of their ports. Now click the `TRACE` and `LIST` buttons on the palette. When the Trace and List windows appear, notice that the signal names are highlighted in all three windows. This is called *cross-selection* and is a key feature of Mentor. If a signal is selected or unselected in one window, it will be selected or unselected in all windows.

Next, you will need to set up the timing mode and define some input signals. The timing mode defines the timing parameters for each gate (you will learn more about this later). Unselect everything, then select just the four gates (three AND gates and one OR gate) by clicking on them. From the `SETUP` palette click the `TIMING MODE` button. In the dialog box that appears, just

make sure the Timing Mode is set to unit. OK the form. To finish the setup, create a clock signal for Z and input forces for the other two inputs. Get to the STIMULUS palette and, with *only* the Z signal selected, click on the ADD CLOCK button. Enter a period of 40 (the default time period is ns) in the Period box and OK the form. This sets up an infinite clock signal on that input that is active low (the default) with a 50% duty cycle (default again) and has a period of 40 ns. Also by default, it replaces any other signals (clocks or otherwise) that were on that input before. Select input X (only) and from the STIMULUS palette click ADD FORCE. Enter 0 for the Value, 0 for the Time and OK the form. This places a signal of value logical zero on the selected input at time 0.0 ns. Repeat this step for the input Y.

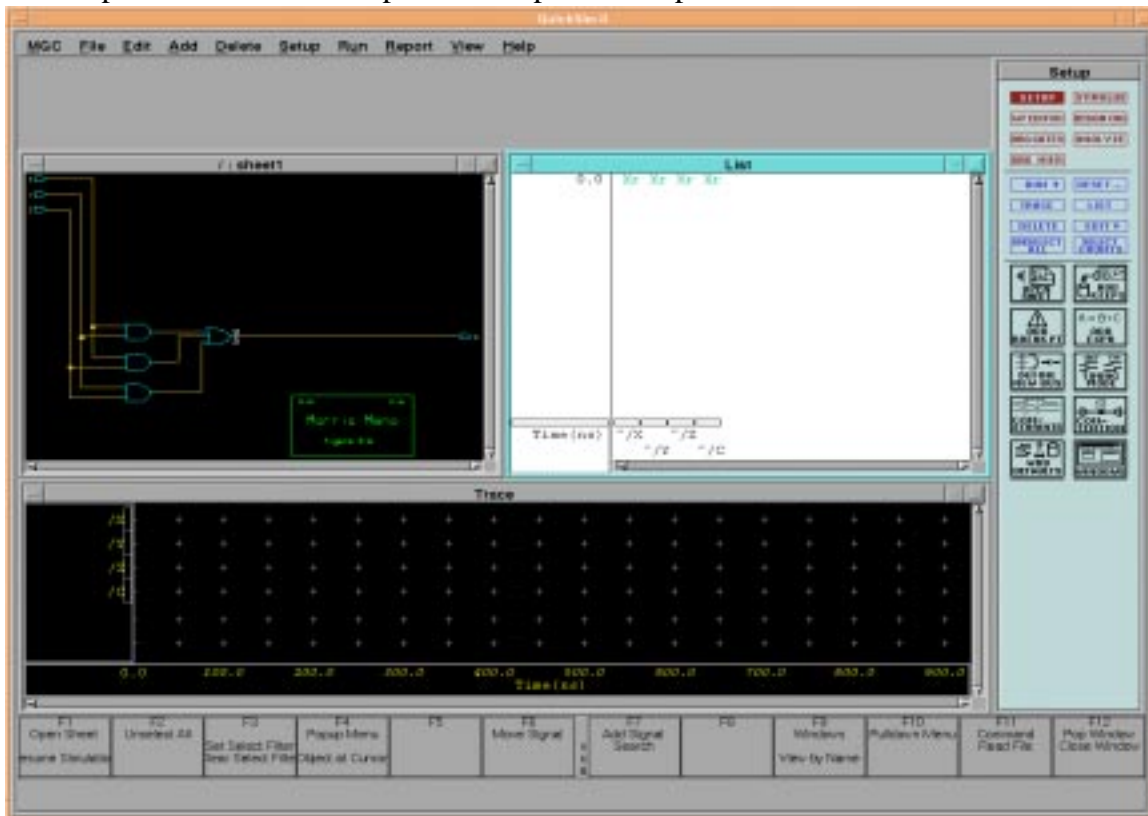


FIGURE 1. QuickSimII Screen for Project 1

The circuit is now all set up and you are ready to simulate it. Be careful to read *all* the steps below before you start the simulation. To run the simulation, follow these steps:

1. You will run, stop, change the input, run and stop the simulation again. Read this entire section before doing anything. To start a simulation, type `run 40` and press return with the cursor in any window *except* the palette. This will simulate the circuit for 40 ns and stop after that. This way you can run the simulator for any amount of time you like. The Trace and List windows display what events occurred during simulation. The List only adds a new line when a signal you are watching changes (changes appear in green). The Trace shows what happened for the entire time the simulation has run. To see all of the Trace output you may have to use the

`View⇒All` command from the pop-up menu in the Trace window. The input Z which is driven by a clock changes between 0 and 1 automatically every 20ns. This is because the clock has a 50% duty cycle and the period of the clock was set to 40ns.

2. The last step showed you how to run the simulation with inputs X and Y at logical zero and Z driven by a clock (as you set them up). Now you need to run the simulation with one input at logical zero and the other at logical one. First, unselect everything. Next, change input Y to a logical one. To do this, select the input you want to change. Then, from the `STIMULUS` palette, click the `FORCE TO STATE` button. Choose `1 (TRUE)`. This will cause the selected input to have a value of logical one from the current simulation time on, or until you change that input again. It should be noted that this will *not* affect the signal value prior to the current simulation time (i.e. you can't change the past).
3. Now, run the simulation for another 40ns with this new set of inputs. This can be done by typing `run 40` again. Note that the time you give it is relative to the current simulation's time. (i.e. `run 500` adds 500ns of simulation to the total simulation time). Notice the green text in the List window. This indicates a change in value. You may have to scroll back through the list to see changes on signals. To see the changes in the Trace window you will need to zoom out. To do this, use the pop-up menu command `VIEW⇒ZOOM OUT` or `VIEW⇒ALL` while in the Trace window. You may also need to use the scroll bar to move around in the trace window.
4. Use the process described above to try all possible logical combinations of inputs X and Y.

To interrupt simulation at any time type `ctrl-c`. Only type `ctrl-c` once. If you type it more than once, you may be dumped out of the simulator completely and have to start over. Be careful. To resume the simulation type `run` followed by the duration and press return again.

Questions:

1. What does the output do for each case?
2. When you change one of the inputs, does the output change at the same simulation time? (Note: you may have to zoom in around a change to examine this. You could also use the List window to see possible small delays in time.)
3. If there is a delay, is it the same for inputs X and Y?
4. What is the delay (in ns) from the input X to the output C?

Drawing Waveforms

In this section, you'll use **Mentor Graphics** to draw/simulate the circuit in Figure 3-6 (page 109) of the course text. Begin by comparing the circuit on the screen to the circuit in Figure 3-6 in the text.

Now you'll draw waveforms to match Figure 3-8 in the text. When you are done studying the traces from the previous section, you will need to reset the simulation time to 0.0 ns and remove the input forces you applied to the input signals. To reset the time, click `RESET` from the palette area. It is a blue button just above the `List` button. On the form that appears check the

State box, *uncheck* the Save 'results' box and OK the form. This resets the simulation time to 0.0 ns. Now, use the `unselect all` button to unselect all the signals. Then select the inputs X and Y. From the STIMULUS palette click the DELETE FORCES button. In that form select Selected signals and OK the form. This removes the forces applied on the selected signals. Now the clock that was setup on the input Z is retained. **Note:** the timing mode you set for the gates earlier is unaffected by any of these actions.

Now, select input X and go to the WF EDITOR palette (brown button). Take a second to look at the tools you can use to create and edit waveforms (inputs). Next, click on EDIT WAVEFORM. This will product an editable copy of the selected input(s) in the trace window. You only need to click EDIT WAVEFORM once to create the editable waveform. If you leave the same signal selected and click that button again, a duplicate (unwanted) copy will appear. To remove that copy (or any other unwanted trace), select the small hollow rectangle next to the trace name or above the label in the List window. This is called a *gadget*. It will become highlighted when it is selected. To delete it, press the Delete key or select delete from the edit menu. The editable waveform will have a name like `forces@@/signalname`. Take some time and experiment with the different ways of editing the waveform. With the trace window active, click on a tool in the palette then click on the waveform to edit it. When you are done with a tool, choose CANCEL in the lower left corner of the screen. For example, if you use the PULSE tool, you need to hold and drag over the area you wish to add the pulse. Also note, a pulse will not be added if you cross two traces (vertically) while editing. If you tell a signal to go high at a certain time, and later use the same or a different tool tell it to go high at the same time, a red dot will appear. This dot means that a duplicate force is being applied at that time. To remove these, select the CLEANUP tool and click on the dots.

When you feel comfortable drawing waveforms, make sure the signal(s) you were editing are the only things selected, then DELETE FORCES on them. Scroll to time 0.0 ns if you need to, and, using the methods above and the TOGGLE tool, draw waveforms on X and Y to match Figure 3-7 of the text. You'll probably have trouble getting the TOGGLE tool to change the input at exactly time 0.0 ns. However, you can use the ADD FORCE tool like you did in the last section to set the desired value at time 0. To get your inputs to line up with the clock signal (Z), you can either add a new clock and draw your inputs to match it or you can draw your clock signal by hand, just like you did for inputs X and Y.

Now you need to simulate the circuit with this set of inputs. The simulator stops either when you tell it to (like in the last section) or when it doesn't see any more inputs changing (i.e. it runs out of input). In the first section you specified the duration of simulation, because the clock is an infinite signal that never stops changing. If you drew your clock by hand in this section, the simulator will stop when it reaches the end of your drawn signals (the last time where one of them changes).

When you have run the simulation (and are happy with how it looks) you need to get a print out of the trace. (Note: in general, try to avoid printing out the List window. It can be *very* long and doesn't provide much information. Use cut and paste to get the few lines you need if you think the List data will better illustrate a point you are making.) First, set up which printer you want to use. From the menu bar choose `MGC⇒Setup⇒Printer...` In the form, click on the printer you want to use, click the `Select Printer` button, and then `OK` the form. Make the window you want to print active (click in the Trace window in this case) and select `File⇒Print⇒Active Window...` from the menu bar. Make any modifications you want, `OK` the form and go collect your print out. **WARNING:** Be aware that there is a print quota in the department. You can only print a total of 500 pages per quarter. Be careful to do your editing online and only print what you need. You can check your print quota anytime by typing `pquota` at a system prompt.

Drawing Circuits

In this section, you'll begin to learn about the **Mentor Graphics** schematic capture abilities. You will use another tool to draw circuits called **Design Architect**. Minimize the **QuickSimII** window and double click on the **Design Manager** icon to bring the window back up. Again select the `261Circuit1` icon, then, from the pop-up menu, choose `Open⇒design_arch`. Remember it will take some time before anything happens. Use any of the zoom functions (or `View⇒All`) to adjust the view so you can see all of the circuit clearly, once it has been completely drawn.

You will be modifying your circuit to look like the carry part (C) of Figure 3-6 in the text book. There are two ways you can change the circuit. You could move the gates and output port with the wires attached. You would then have to move the bends in the wires to clean up your circuit. The other method is to delete the wires, move the gates and then re-draw the wires.

To make the change without removing the wires, move the mouse cursor onto the 3-input OR gate and click the left mouse button to select it. The gate will become highlighted. From the pop-up menu or the palette, choose the `MOVE` command. Notice as you move the ghost image around, the wires connected to the gate stretch to move with it. Move the OR gate so that the middle input is straight across from the output of the middle AND gate, then click the LMB to place the gate there. Be careful of what is selected when using the `MOVE` (or any other) command as the command will affect *everything* that is selected. To see how many items you have selected, look in the upper left of the session window, below the menu bar. You will see a `Sel:` followed by a number. That number is how many items you have selected. Now move the output port C so that it is level with the output from the OR gate. In **Design Architect**, a port and its attached wire are separate objects, unlike in **QuickSimII**. You should select the output port and move it.

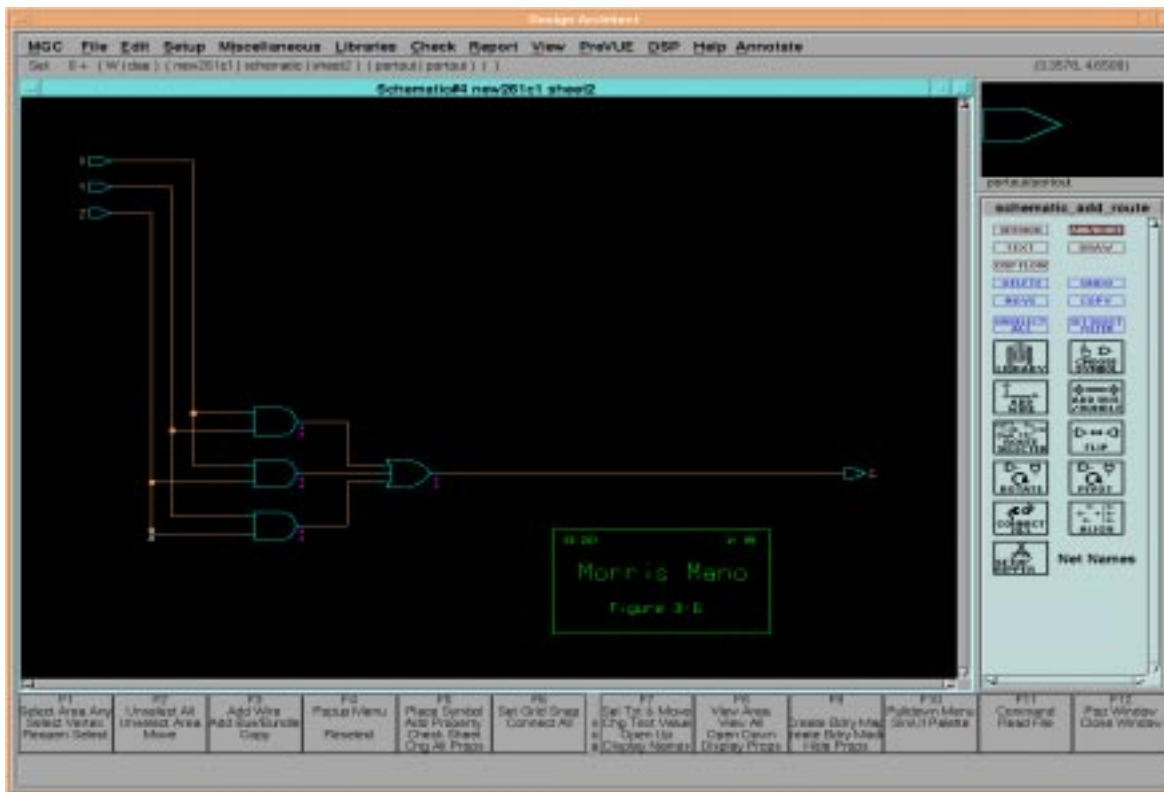


FIGURE 2. Design Architect Screen for Project 1

At this point, your circuit is a mess, with the wires crossing everywhere. To fix this, you should move the points (bends) in the wires. They are selected and moved in the same way any wire or gate is. It may take some practice to select just the point and not the wire. (Hint: When a point is selected, it is highlighted by a white cross centered on the point.) You can also select and move a point that is attached to the pin of a gate. Move the points around so your circuit looks similar to Figure 2. There is another way to do this automatically. When you click and hold down the LMB while moving the mouse, the mouse pointer forms a rectangle and selects everything inside the rectangle when you release the button. Use this to select the output wires of the AND gates along with the OR gate. Stop the right edge of the rectangle at the middle of the OR gate so that the output wire of the OR gate is not selected. Now, from the ADD/ROUTE palette, click the ROUTE SELECTED tool icon. This will automatically re-route the wires cleaning up all the mess.

If you are unable to fix the circuit by moving the points you can use the other method, deleting and redrawing the wires. You can select the things (wires in this case) you want to delete and press the Delete key. You could do this before or after you move the gates. After you delete the wires and you have placed your gates you need to replace the wires. From the ADD/ROUTE palette, click the ADD WIRE tool icon or use the F3 key to start the wiring tool. To be electrically correct, you should start and end your wires on the purple diamonds of gates. To start a wire, click

once. To add a bend or anchor point anywhere, click once again. To end a wire, double click. To undo your last click, use the `backspace` key. When you are done, your circuit should look like Figure 2. `Cancel` the wiring tool on the form at the lower left when you are done with it.

Now all you need to do is put your name on the circuit. Place the mouse over “Morris Mano” and click while holding down this shift key. This action allows you to select comment text, which is not included in the selection filter by default. From the `TEXT` palette select `CHANGE VALUE`. A text prompt will appear in the lower left. Type your name and press `Enter`. If you need to, move your name around to fit better in the box.

Before you save the changes you have made, you need to make sure your circuit is electrically correct. Do this by selecting `Check⇒Sheet` from the menu bar. A report will appear. If you have any errors or warnings you, will need to fix them, and then check the sheet again. Remember cross-selection from `QuickSimII`? It works here also and is a good way to find the source of your errors and warnings. If you have an error or warning it will tell you the location: something like `N$486` or `I$271`. This is the handle tag Mentor uses to keep track of things. To figure out what it refers to, highlight the handle in the report, then look at your schematic. The corresponding part will be highlighted in your circuit as well. Fix any problems you have and check the circuit again. When your circuit checks with no errors or warnings, save it by selecting `File⇒Save Sheet` from the menu bar. When you have saved it, get a printout. You will have to setup the printer just like you did in `QuickSimII` (`MGC⇒Setup⇒Printer...`), then choose `File⇒Print Sheet:` from the menu bar.

More Drawing

Now try adding `AND`, `OR` and `NOT` gates to the circuit so that it is logically equivalent to Figure 3-6 from the text. You will also need to add and label the output `S`. To get the new parts choose `Libraries⇒MGC Digital Libraries` from the menu bar. This will display a list of all the libraries available in the palette area. To get back to the schematic palette or to move around in the libraries, use the pop-up menu on the palette. `Back` takes you up one level in the library and `root` will take you to the first list of libraries you saw. Scroll down to `gen_lib` and click on it. (If you don’t have scroll bars, turn them on from the pop-up palette menu.) You should now see a list of all the parts available in that library (listed in a short form). To place a part on the sheet, click on the part name in the palette. A ghost image will follow the mouse pointer when you move around the sheet area. Click where you want to place the part. The parts you will need are an output port, named `portout`, a 3-input `AND` gate named `and3`, a 2-input `AND` gate named `and2`, a 2-input `OR` gate named `or2` and an inverter (`NOT` gate) called `inv`. For the 3-input `OR` gate you can copy the one that is already in the circuit.

When you place the ports on the sheet, they will have a name property of “`NET`”. You need to change this name, using the same method you used to change “Morris Mano” to your name.

See the figure in the text to determine which port gets which name. To add the wires, use the pop-up menu in the palette to display the schematic palette, then click the `ADD WIRE` tool. Use the tool as described previously to add the necessary wires. If you are drawing a wire and it crosses a pin or wire junction of another wire, and you do not explicitly connect it, a “not dot” will appear. This simply means that there is no electrical connection at that point. While you can leave these in place, it is suggested that you choose an alternate route for your wire to prevent the not dot. When you are done wiring `Cancel` the wire tool in the lower left corner. Check the sheet as before to make sure you have not made any errors. If you have, identify them and correct the problem. Save the sheet with `File⇒Save Sheet`. When you save the sheet, it will save over any previous version of the circuit you had. If you want to keep the original version, save the circuit with a different name by using `File⇒Save Sheet As . . .`. Enter a component name for the new circuit and `OK` the form. **Note:** if you do this the schematic you have open is still the original. Any changes you make to this will be saved to the original unless you use the `Save Sheet As . . .` command again. Get a print out of your new circuit for your report.

Optional Extra Step

If you want to simulate your new design and don't have a **QuickSimII** session running, select the circuit from **Design Manager** and start **QuickSimII** on it. Set up the circuit as you did previously and simulate it. If you still have a **QuickSimII** session running, and you saved the changes to the original circuit, you do not need to exit **QuickSimII** and restart it. To load in the changes you have made, close the Trace and List windows and get to the `DESIGN CHANGE` palette. From there select `RELOAD MODEL⇒ALL`. This will cause **QuickSimII** to look at the circuit file and load any changes you have made to the circuit. You will have to set up the timing mode for the gates again. It is also a good idea to delete all applied forces and create new ones before simulating the circuit.

Report

The report (one report per group) should be typed and follow the report format handed out in class. Be sure to include:

- Answers to the questions (1-4)
- A trace output from the simulation of the first circuit
- The schematic of the modified circuit (like Figure 2)
- The schematic of the final circuit (equivalent to Figure 3-6 of the textbook)
- Trace output from the simulation of the final circuit (optional)