

MENTOR GRAPHICS TUTORIAL

ECE 221

Fall 2003

I. OBJECTIVE

The objective of this laboratory is to work through the process of creating and simulating a digital circuit using the Mentor Graphics computer-aided design software.

II. INTRODUCTION

In this design project you will construct and test various logic gates using tools from the Mentor Graphics tool suite. Mentor Graphics is a set of integrated programs for computer aided design of electronic circuits. It includes many tools and applications, most of which are outside the scope of this introduction. We will be looking at the following three applications:

1. *Design Architect*: A multi-level design environment that includes: a Schematic Editor to create and modify schematics, a Symbol Editor to create and modify symbols for schematics, and a VHDL Editor to edit and compile System-1076 VHDL models.
2. *QuickSim II*: An interactive logic simulator that allows you to simulate the function and timing of digital logic designs (such as schematics created in Design Architect).
3. *Design Manager*: Design Manager helps you to manage your designs by providing an easy interface to copy, delete and move design data, access previous versions of designs, and perform other tasks. This tool is very important because you need to **ALWAYS** use Design Manager to move or copy Mentor Graphics design data (**NEVER** use Unix commands).

You will also use the Adobe Acrobat Exchange tool to access the Mentor Graphics documentation and on-line help system.

After you have finished this tutorial you should be able to:

- Create a digital design schematic using Design Architect.
- Verify that a digital design created in Design Architect operates correctly using QuickSim II.
- Copy or move designs created using the Mentor tools using Design Manager.
- Use the Mentor documentation to answer questions you have about the software.

III. PROCEDURE

CREATING A DIGITAL DESIGN (SCHEMATIC CAPTURE)

The objective of this section is to electronically create a simple circuit provided at the end of this handout (use the circuit as a reference when creating the design). The procedure for creating the design is outlined below. The tool we will use to create the design is called Design Architect.

1. In this laboratory you will be using the Sun Workstations. Log onto the workstation and open up a terminal window. To do this, type the following command in a terminal window.

```
textedit .cshrc
```

This file contains a number of environment variables and paths that must be set to run the Mentor Graphics software. The .cshrc file is a start-up file that is executed whenever you open a terminal window. Add the following line to your .cshrc file:

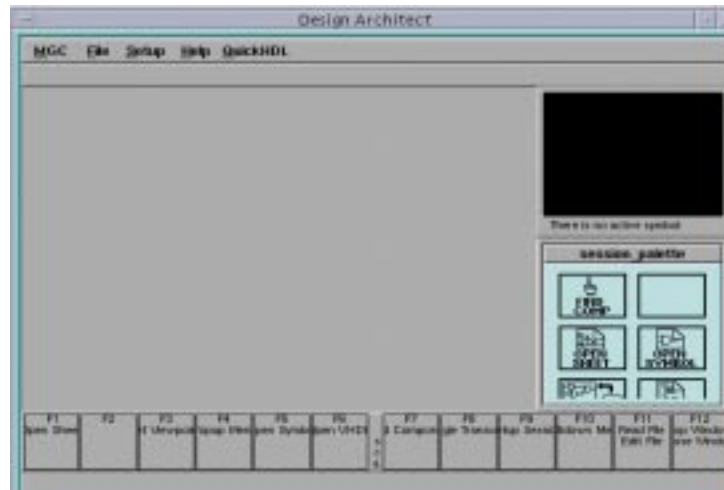
```
source /usr/local/mentor/user/admin/cad_cshrc
```

NOTE: If these lines are at the end of your .cshrc file you will have to add at least one carriage return at the end. After you have saved the file, execute the command:

```
source .cshrc
```

in a terminal window. This will reread your start-up file.

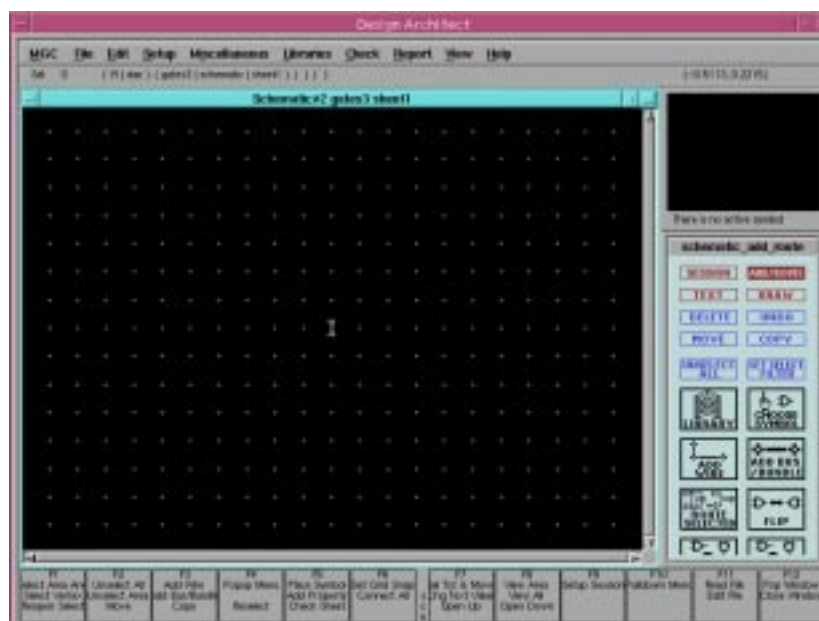
2. In the same command window, type in **da** and then hit **<enter>**. This will open the schematic capture tool, Design Architect. The Design Architect main window will appear as shown below:



3. Open a new design and enlarge your work area.

3a. Select the **OPEN SHEET** option from the palette on the right. A form will appear asking you for a component name. Delete the current name, then type in **gates** and hit return. This will create a new design named **gates** in your current directory.

3b. When the design opens, you should see a black square. For the remainder of this tutorial we will call this area the **design workspace**. To make it easier to create the design, enlarge the design architect window. You can then maximize your design workspace by single clicking the square found in the upper right corner of the design workspace window.



3c. Add the borders and title block to your design by selecting the **Add Valpo Borders (Small)** under the **Edit** menu. After the borders have been added, a form bar on the bottom of the window will appear. This will allow you to change the attributes in the title block. Type in **Logic Gates** then hit <tab> followed by **your name** <tab>, the **date** <tab> and **ECE 221**. Then hit <enter>. The title block should be updated with the new information you just entered.

The Mentor software graphical user interface has a heavy emphasis on palettes and popup menus. *Palettes* are *context-sensitive* groups of icons which provide easy access to tools and functions that are relevant to what you are doing. For example, when you invoked Design Architect, the `session_palette` was active so that you could open a file. When you opened a schematic, the `schematic_add_route` palette appeared, which contains many of the standard schematic function icons.

All popup menus are accessed via the **right mouse button (RMB)** and are context-sensitive. This means that a different popup menu will appear depending on where the mouse is. For example, **RMB** in the main schematic window will bring up one menu, while **RMB** in the palette window will bring up a different menu.

4. Add the components (gates) to the design.

4a. To add a gate, select the **MGC DIGITAL LIBRARIES** option under the **LIBRARIES** pull down menu using the **LMB**. The palette on the right will change to show a list of libraries. Select the library, **gen_lib** with the **LMB**. Now the palette contains a list of components in the general library. Select the **and2** component with the **LMB**. This makes that component active. Move the mouse where you want to place the part in your design and then single click on the **LMB** to add the part to the schematic. Next, place the cursor in the `gen_lib` window and hit the <page-down> until you see the **or2** component (you can also add the scroll bars by placing the cursor in the `gen_lib` window, using the **RMB** to bring up a pull down menu and then selecting the **Show Scroll Bars** option). Add the **or2** to the schematic using the same procedure you used for the **and2**. Add the **inv**, **nand2** and **nor2** in a similar fashion (use the schematic at the end of this handout as a guide for placing components). When you add a part, it shows up bright white in your design workspace. This means that it is selected. Any selected components (or wires) can be moved, copied, etc. To unselect all of the components hit the function key **F2**.

4b. Add the port components: These components signify the inputs and outputs to your design. They are also located in the **gen_lib** under the names **portin** and **portout**. Add two **portin** components to the schematic and five **portout** components. Follow the same procedure for getting and placing a new part as described in 4a. Once you have placed all of your parts, change back to the main palette by placing the cursor over the **palette** window and holding down the **RMB**, and then selecting the option **Display Schematic Palette**.

4c. If you need to move any component, first unselect all components, single click with the **LMB** on the component you want to move (this re-selects it), and then choose the **MOVE** option on the palette. The component will then follow your cursor until you single click with the **LMB**. Use the move command to place the components at their correct location on the schematic (if they are not already there).

5. Connect the components: Now that you've placed all of the components, you need to connect them together by adding wires. To draw a wire in the design follow the procedure outlined below.

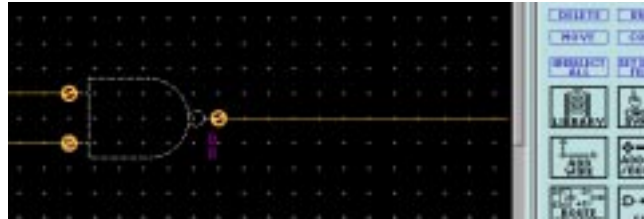
5a. Unselect all of the components by selecting with your **LMB** the **UNSELECT ALL** option on the palette or by hitting **F2** (A component is selected if it appears bright white on the screen).

5b. Select the **Zoom In -> 2.0** option under the **View** pull-down menu to zoom in on your

components. You can now use the scroll bars to move in the design workspace window.

5c. On the palette you will see the **ADD WIRE** option. Once this is selected, a plus sign will appear under your cursor. This means you are in draw wire mode. In this mode, a single click on the **LMB** will start a wire, another single click will create a vertex at that location in your design, and a double click will end a wire. Once you've completed a wire, you can start another right away because you remain in draw wire mode. To get out of the draw wire mode, hit <esc>. Connect the wires as shown in the given schematic. After you've added all of the wires, unselect them.

5d. If you have any crossed-out symbols (a.k.a. the ghostbuster symbol) in your design as shown below:



This symbol means that the component is not connected to the nets but is sitting on top of them. To connect the nets to the component, select **CONNECT ALL** from the palette. This will make the connections between the parts and the wires and remove all of the crossed out symbols in your design.

6. Change the labels for the inputs and outputs.

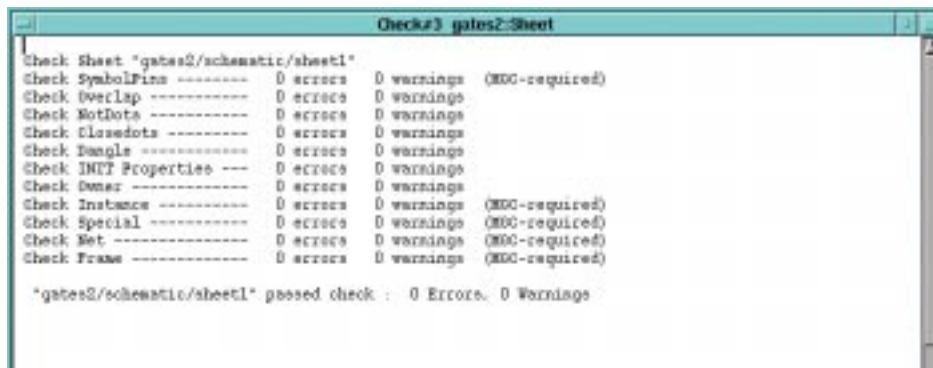
6a. Select the **View All** option under the **View** pull-down menu to show the whole design.

6b. Select the **TEXT** option on the palette.

6c. Select the **CHANGE VALUE** option on the palette. Then holding the **LMB** down, draw a box around the word **NET** associated with the two portin components. The words **NET** associated with the portin symbols should change to bright white one at a time. A dialog box should appear at the bottom of the window. Type in **IN1** and hit <enter>, then type in **IN2** and hit <enter>. The two NET labels for the portin components should be changed to IN1 and IN2.

6d. Change the NET labels for the portout symbols following the procedure outlined in the previous step. Use the labels for the outputs as shown on the schematic: **AND_OUT**, **OR_OUT**, **INV_OUT**, **NAND_OUT**, and **NOR_OUT**.

7. Check your design to make sure it does not have any errors by selecting the **Sheet** option under the **Check** pull-down menu. The window given below should appear showing no errors or warnings.



Close the window by clicking on the bar in the upper left corner of the window with the **RMB**, holding it down, and selecting the **Close** option.

8. Save your design. Use the **Save Sheet** option under the **File** pull-down menu.

There are a variety of ways built into the user interface to execute various commands. For example, at the bottom of the user interface, there are the **Function Key** shortcuts for many of the commands found in the menus. Each function key has four options, <function key>, <shift>-<function-key>, <ctrl>-<function-key> and <alt>-<function-key>. For example, if you press F8 you will be able to view a specified area while <shift>-F8 will allow you to view your whole design (View-All). The other shortcut technique is by creating **Keystrokes**. Keystrokes are created when the **middle mouse button (MMB)** is pressed in the design workspace window. The cursor is moved in a specific pattern while the **MMB** is held down and finally released. The software recognizes the pattern that is drawn and executes a specific command. For example, place your cursor in the design workspace window, hold down the **MMB** and draw a question mark (you should see the question mark appear in red) and then release the **MMB**. This will bring up a window showing you different keystroke shortcuts for various commands (if you don't get the window the first time try again). If you want, try some of these commands on your design.

9. To print out your design, select **Print Sheet** under the **File** pull-down menu. For the printer name, type in **psmgc**. This printer name will always send the file to printer in room 130.

10. Close the design architect tool by again using the bar at the top of the window.

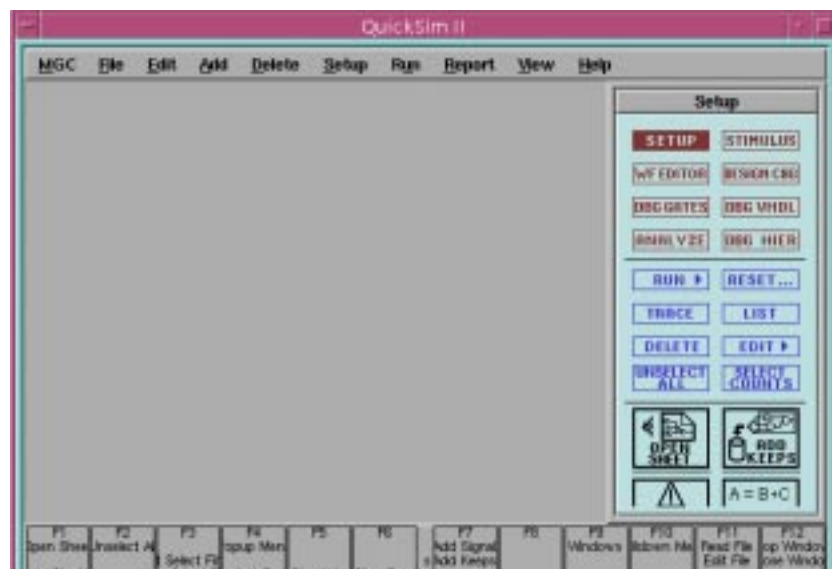
SIMULATING A DIGITAL DESIGN

Recall that simulating or verifying a circuit involves placing values on the inputs and then viewing the output values. To verify the design you need three items: 1) the design itself, 2) a set of values placed on the inputs at various times (stimulus), and 3) a simulator. The procedure for simulating the circuit is outlined below:

11. The simulator within the Mentor Graphics tool suite is called **QuickSimII**. To access QuickSimII, type

```
quicksim gates
```

in a command window. This will initialize the simulator using the design that you created and bring up the following window:



Enlarge the window so you can see more of the palette.

12. Select the **OPEN SHEET** option on the palette. This will allow you to view the design as you are simulating it.

13. Next add stimulus to the inputs.

13a. Select the **STIMULUS** option at the top of the palette. This brings up a separate palette that handles all stimulus operations.

13b. Select the **ADD CLOCK** option on the palette. In the form that pops up, place the cursor over the phrase "Signal name" and type in **IN1**, hit <tab> and then type in **400** for the period. Then select **OK** with the **LMB**.

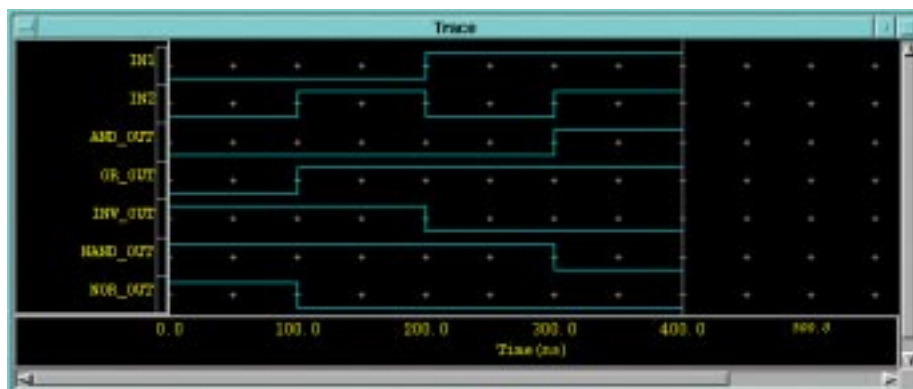
13c. Again select the **ADD CLOCK** option on the palette. In the form type in **IN2** for the signal name, and **200** for the period. Then select **OK** with the **LMB**.

14. The next step is to set-up a trace window to view the results. Select the **SETUP** option at the top of the palette. This brings you back to the main palette. Select the **TRACE** option on the palette. In the window that pops up, move the cursor over the phrase "Signal name" and type in the following signals (these should match the input and output labels in your design):

```
IN1 <tab>
IN2 <tab>
AND_OUT <tab>
OR_OUT <tab>
INV_OUT <tab>
NAND_OUT <tab>
NOR_OUT <tab>
```

Once you've typed them in, hit the **OK** button or <enter>. A trace window will then appear within the quicksim window showing all of your signals.

15. Run the Simulation: To run the simulation, select the **RUN** option on the palette with the **LMB**. When the pop-up menu appears, select **FOR TIME:**. Type in **400** and then hit <enter>. The simulation will run and the trace window will be updated showing the waveforms for each of the signals as the simulation was run. Select **View All** under the **View** menu. Compare your waveforms with those shown below.



16. To print out the waveforms, select **Print->Active Window** under the **File** menu. A form will appear asking you for the printer name and domain start and finish. Put **psmgc** as the printer name and hit **OK** (for this example you can leave the domain start and finish blank).

17. If your waveforms are not correct, check your design and make sure it matches the one found in this handout. If you want to rerun your simulation, select the **RESET** option on the palette, click on the **State** box and then hit **OK**. The waveforms in the trace window should disappear. You can then add new stimulus to your inputs or rerun your simulation. If your design is not correct, close QuickSim, reopen Design Architect, make your changes, check the design, and then save it. Then start up QuickSim again.

NOTE: Resetting the simulation does not remove the forces (i.e., the stimuli, or inputs) from the components. To do that, select the **STIMULUS** button on the palette. The palette will change to the Stimulus palette. Select **Delete Forces** on the palette, and then the **All signals** button, followed by **OK**.

18. Exit the simulation window using the procedure outlined in Step 7. A form will appear asking if you want to save your simulation environment. Select the option **Without saving** and press **OK**.

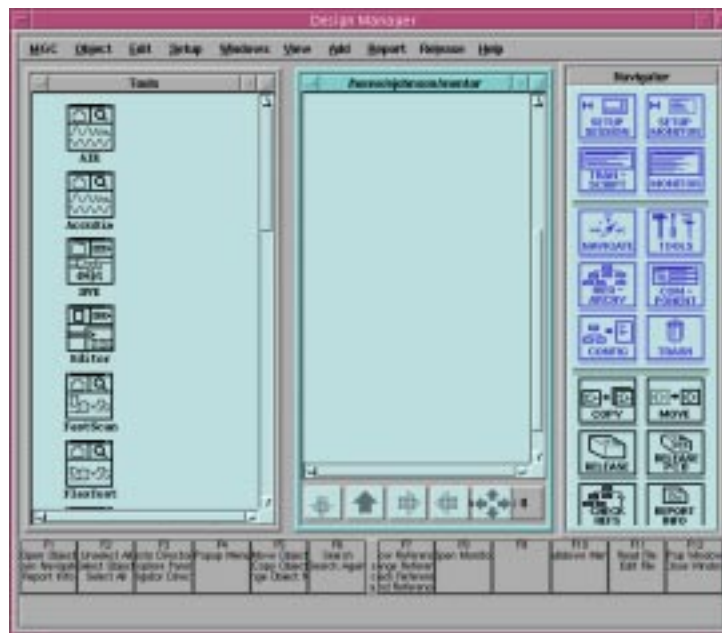
FILE MANAGEMENT

Now we will briefly look at the Design Manager. Because the Mentor Graphics software creates a number of directories and linked files you **MUST** use the Design Manager to move or copy ALL Mentor Graphics files. If you don't, your design data will be corrupt and you will have to start again.

19. To invoke Design Manager. Type:

```
dmgr
```

Design Manager will take several seconds to load. When it finishes loading, you will see the following window:



There are three sections in the window. The left-hand section allows you to invoke specific tools or applications. This is accomplished by selecting the tool and then choosing the **Open** option under the **Object** menu (you can invoke both Design Architect and QuickSim from the Design Manager instead of the command line if you want). The middle section is similar to a file manager and shows you all the files in your current directory. To give you some practice with moving Mentor objects, create a new directory (using **mkdir** in a command window or selecting **Directory** under the **Add** menu). Call

the directory *mentor* (or some other name). This will allow you to have a directory for all of your Mentor designs. Move the design you created (gates) to this directory by executing the following commands:

19a. Go to the location of your design using the up and down arrows at the bottom of the window. The up arrow will move you up one directory. The down arrow is used to move down a directory after it has been selected.

19b. Select the design. All Mentor design icons should look like this:



19c. Click on the design icon and select the **Move** command under the **Edit** menu. A small text bar will appear asking you for the location to move it. Type in:

```
/home/<your_username>/<your_mentor_directory>
```

and then hit <enter>. The design will be successfully moved to your Mentor directory.

The right hand section of the window is the design manager palette. For now we will not discuss the operations on the palette. Close the Design Manager when you have completed the move.

DOCUMENTATION

All on-line documentation can be accessed through either a Mentor Graphics application called Bold Browser or by using Adobe's Acrobat Exchange. In this class we will be using the Adobe Acrobat Exchange option. To invoke Acrobat Exchange, type in.

```
mgc_acro
```

If a message window appears (just hit **OK**) and look for the following basic acrobat window to appear:



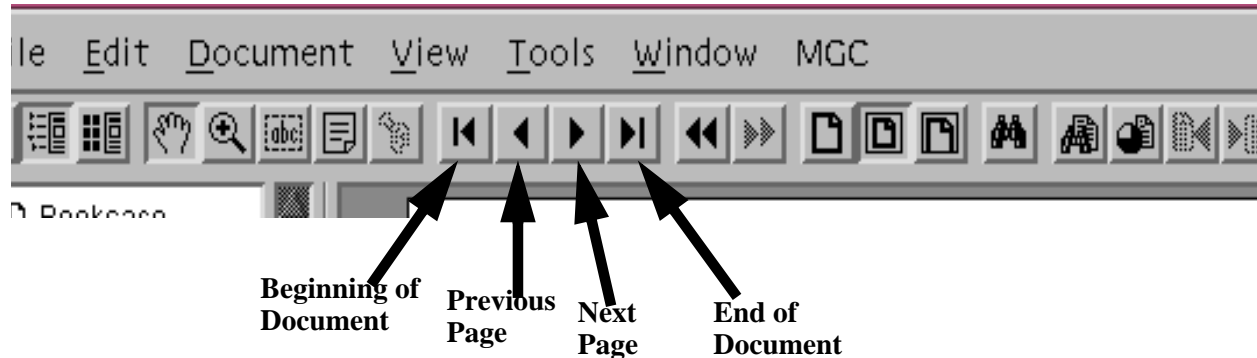
Select the **Open** command under the **File** menu. In the window that appears you will see a "files" section. That subwindow lists a number of different bookcases that contain documents you may use. The two bookcases that contain the documents you will need are:

```
_bk_da.pdf    Bookcase for Design Architect
```

_bk_qsim.pdf Bookcase for Quicksim

Select the **_bk_da.pdf** and then hit **Open**. A window will appear that contains all the documents you can reference that are associated with Design Architect. In that list select **Design Architect User's Manual**.

A window will appear containing that document. It is helpful to maximize this window. Click and hold down on the **100%** button at the bottom of the page and when the pop-up menu appears select **Fit Page**. To move forward or backward pages in the document use the sideways triangles located along the top bar of the window (see below).

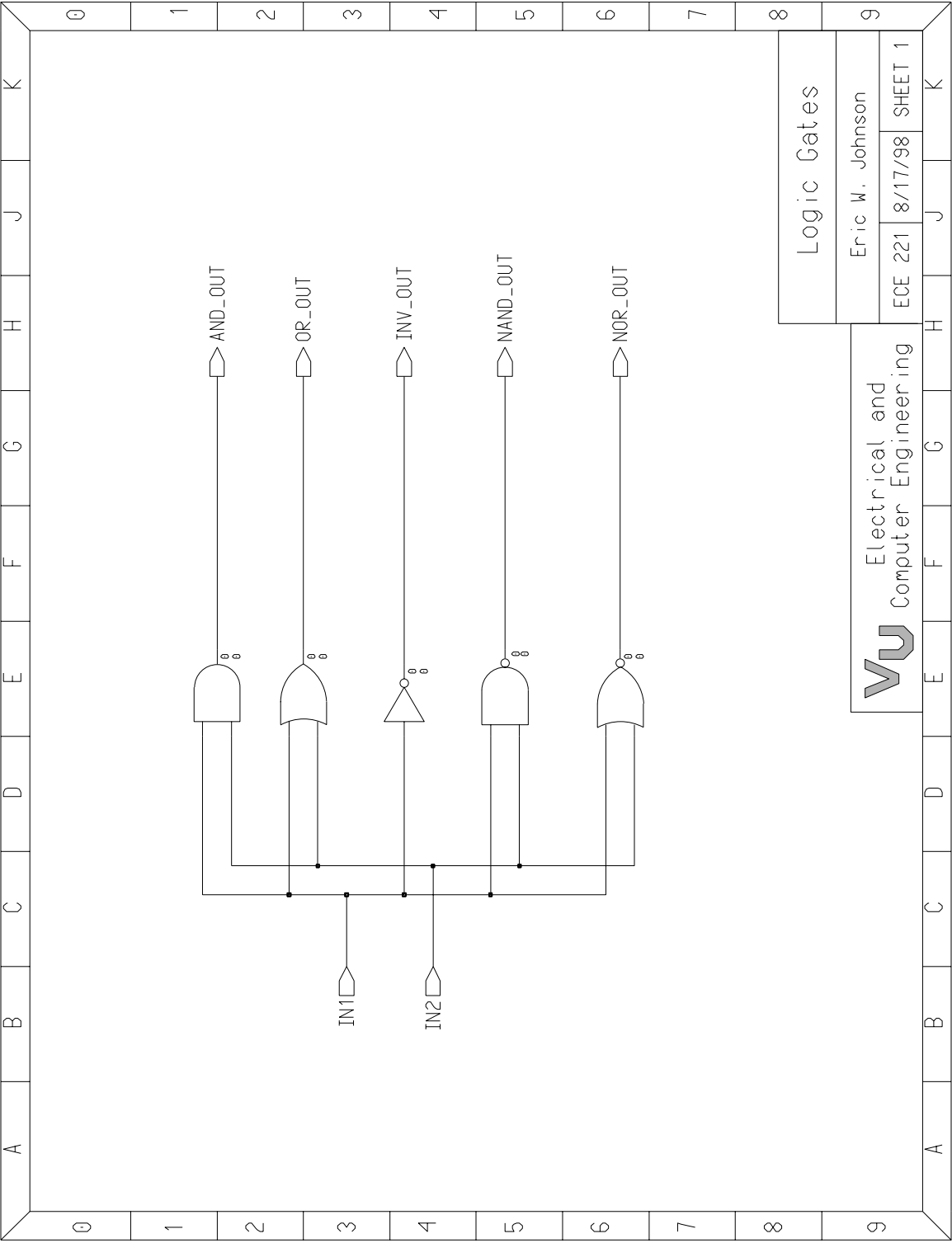


Blue text is hypertext; that is, if you single **LMB** click on this text, you will jump to that location. Spend some time using Exchange to work through the document. Search capability is available using the **Find** command under the **Tools** menu. To exit Acrobat Exchange select **Exit** under the **File** menu.

IV. SUMMARY AND ANALYSIS

Place a copy of your finished schematic and the correct simulation results in your lab notebook. Write a paragraph summarizing what you did in the lab today. Finally, using the Mentor documentation answer the following questions:

1. In Design Architect, what is the **match** command? How do you use it?
2. What are strokes? How do you make them? In Design Architect, what is the window stroke of **View All**?
3. In *Module 3: Lab Exercises* of the *Getting Started with QuickSim II* document, how are the signals in the Trace window added?



Logic Gates

Eric W. Johnson

ECE 221 8/17/98 SHEET 1