EE281 Introduction to Design Architect

Getting Started

The application used to create a Mentor Graphics design is Design Architect, the “schematic capture” tool. To use it, first make sure Design Manager is running (it can be started by selecting Design tools ⇒ Mentor Graphics from the HP’s pop-up menu). Scroll down in Design Manager’s Tools window until you see the design_arch icon. Double-click on it, then wait for a while. When Design Architect is done loading, it will maximize its window to fill the screen. The following procedure will help you enter and save a simple circuit.

First, click on Open Sheet in the palette window (the rightmost window in every Mentor tool we use). “Sheet” is equivalent to “schematic” for our purposes; Mentor calls them “sheets” because the schematic for one circuit could span more than one virtual “sheet” of paper. A dialog box will pop up, asking you to name the component (all circuits are called “components” by Mentor, as any circuit could be used as a component of a higher-level design). The Component Name box will contain the path to your mgc directory. Add “/small” after the end and OK the dialog box. This will create a circuit named “small”.

FIGURE 1. Circuit to draw.

To draw a schematic that looks something like the figure, follow these steps:
Select Libraries⇒MGC Digital Libraries from the main menu of the window. **Mentor Oddity Alert:** There will be an arrow next to MGC Digital Libraries, indicating a sub-menu. If you click on (or drag the mouse over) the arrow, the sub-menu will appear. However, if you just click on (or release the mouse over) MGC Digital Libraries, the sub-menu will not appear. Instead, the first item on the sub-menu will be selected by default. In this case, that’s OK. In this (and other) Mentor labs, if the instructions say to pick (for example) Check⇒Sheet, there’s no need to follow the menu structure below “Sheet.”

Click (once) on “ls_lib” in the palette. This will bring up a list of parts to choose from. First, click on “74ls08” (this is an AND gate). Its symbol will appear above the palette. Click on the symbol, then move the mouse over the schematic window. A ghost image of the gate will follow the mouse around. Click in the schematic window to place the part. Place three of these gates in the schematic window, roughly in the locations indicated by Figure 1. You’ll probably have to scroll around and/or use the View⇒Zoom Out command to leave enough room between the gates. The View⇒View All command will zoom in or out so that the circuit fills the entire window.

Now, click on “portin” in the palette. This will produce a pointy symbol that stands for an input. Place four of these in the circuit. Then, click on “portout” in the palette. Place one of these in the circuit. Now, all of the parts should be in place. Move the mouse into the palette area, hold down the right mouse button to display the pop-up menu, and choose Display Schematic Palette. This will get rid of the parts library.

Select “Add Wire” from the palette. (If you don’t see Add Wire, click on the red button labeled “Add/Route”). This tool will let you connect the parts together. To start a wire, click once. To add a bend in a wire, click once. To end the wire, double-click. To undo a bend you place in the wire, hit the Backspace key. Add some wires (starting and ending right on the purple diamonds next to the parts) so that your circuit looks something like Figure 1. When you’re done adding wires, click the Cancel button on the gray prompt bar at the bottom of the screen. (It says ADD WI on its left side).

At this point, use View⇒All to see the entire circuit. All of the parts and wires will probably be selected (highlighted in white). To de-select them, click on the blue “Unselect All” box in the palette. (You could also pick Unselect⇒All from the pop-up menu, or Edit⇒Unselect⇒All from the window’s menu, or hit the F2 key).

The next step is to rename the inputs and outputs to something other than NET. (This isn’t just for aesthetic reasons; if these ports have the same name, Mentor assumes they’re connected together). Click on the red “Text” button on the palette. This brings up a new set of tools in the bottom part of the palette. Hold down the Shift key and click on the top left NET. It’ll become highlighted. Now, click on “Change Value” in the palette. Type A into the leftmost text box inside the gray prompt bar. Then hit ENTER or click OK. The input should change name to A. Unselect
everything (using any of the methods above). Select another NET and repeat this process. Continue until all of the inputs and outputs have unique names.

Now, the circuit is complete. However, before it can be saved, it must be checked for electrical correctness. Select Check⇒Sheet from the window’s menu. A window will pop up, showing any errors in, or warnings about, your circuit that Mentor can find. If there are any errors, Design Architect will not allow you to save the sheet; the errors must be corrected first. If there are warnings, they probably should be looked into, but the circuit can still be saved.

Finally, save the circuit. This is accomplished by using File⇒fiSave Sheet from the menu. If the file is saved successfully, the status area (the bottom left of the screen) will display a message similar to “Version 2 of sheet /rcc4/...../mgc/small/schematic/sheet1 has been written.”

**Experimentation**

Now that the circuit is finished, you can modify it to your heart’s content. Try moving parts and wires around (there’s a blue “Move” button on the palette). When a part is moved, the wires connected to it will “stretch” to stay connected to the part. You can also copy and delete parts or wires. Note that these Move, Copy and Delete commands operate on everything that’s selected. **Mentor Oddity Alert:** Items are selected in Mentor by clicking on them (no big surprise). However, once selected, they stay selected, so clicking on several parts will leave them all highlighted. You must click on an item again to deselect it, or use one of the many forms of Unselect All.

**Cleaning Up**

When you’re finished editing the circuit, close the Design Architect window. This can be done by double-clicking on the box at the top left of the window (next to the title bar - it contains a minus sign). In Design Manager, click in the Navigator (middle) window so it becomes active. Bring up the pop-up menu and select Update Window. Your circuit “small” should appear. Click on the circuit’s icon to highlight it. Then, hold down the right mouse button to bring up the pop-up menu. One of the choices is Open. Selecting the arrow to the right of Open, you’ll notice a list of tools that can be used on the circuit. design_arch can be opened, to edit the circuit further. QuickSim II can be opened, to simulate the circuit’s behavior. (If it were an analog circuit, AccuSim would be used instead). Go ahead and close the Design Manager window.

Note that any tools you are using must be closed before closing Design Manger (otherwise, Design Manager complains). Likewise, Design Manager must be closed before you log out. Otherwise, the HP environment will try to restart Design Manager when you next log in. This will take a long time, and Design Manager will get confused, causing it to function improperly.