CONSTRUCTING CIRCUIT DIAGRAMS

1. Log in to your workstation. Click on Start, then select "Designworks Professional 5" to launch DesignWorks.
2. Click on "cancel" in the "Welcome" window.
3. The screen display includes a menu bar, a toolbar providing simulation functions (the "simulation toolbar"), a large design window (shaded grey), a timing window (at the bottom) and a parts palette (on the right).
4. From the "File" pulldown menu select "New". A small list window opens. Highlight "Design" and then click "OK".
5. A small "New Design" list window will appear. Select "Generic Simulation" and click "Create". The large Design Window should now display a grid. It is within this window that you will create the schematic of your digital system. As well, a second toolbar should appear beneath the simulation toolbar. This toolbar is the "drawing toolbar" and provides functions to create and edit your schematic.
6. At the top of the parts palette window on the right-hand side, the phrase "ALL LIBRARIES" should be displayed. Click on this and then select "Simulation Gates.clf". A list of gate level components will appear in the display underneath.
7. In this lab you will construct a circuit that implements an important digital system called a "1-bit full adder":

$$s(x, y, c0) = (x \oplus y) \oplus c0$$

$$c1(x, y, c0) = x \cdot y + c0 \cdot (x \oplus y)$$

   NOTE: The symbol "⊕" is the traditional symbol for the XOR function. The symbol used in C for this operator is "ˆ."

   To implement the operators in the expressions, you will require XOR-2 gates to implement the "⊕" operator, AND-2 gates to implement the "·" operator and OR-2 gates to implement the "+" operator. Do not use any gates that have additional descriptions such as "[1-Inv]" or "O.C."

8. Scroll down the parts palette window until the XOR-2 component is located. Note that components are listed in alphabetical order. The number following the gate name indicates the number of inputs.
9. Double-Click on the XOR-2 component name and move your cursor onto the design screen (the one with the grid displayed). Your cursor will change to the entity representation of a 2-input XOR gate. Position this image somewhere near centre and top of the design screen, and click your left mouse button.
10. A copy of the XOR gate will be placed on the main window, but you will continue to have an XOR gate attached to your cursor. Place this second XOR gate about two small grid squares to the right of the first, making sure that no signal port leads are touching, and left click your mouse again. A label may also appear above each gate, but you can ignore this for now.
11. A second XOR gate will now be on your design sheet, and a third XOR gate will be attached to your cursor. Since you will only need two XOR gates, push the space bar, and the third XOR gate will be replaced by the cursor arrow.

NOTE: If you place a component on the design sheet and you want to remove it, then move the cursor over the gate, click the left mouse button, which will display the component area in “reverse video”, and then press the delete key.

12. Before adding more gates to the design, the two XOR gates are to be named. To do this, Click on the “pencil icon” in the Drawing Toolbar at the top of the screen and move your cursor onto the design palette. The cursor arrow will be replaced by the “pencil” icon.

13. Position the point of the pencil in the centre of the left XOR gate and click the left mouse button. A highlighted rectangular text field will appear above the gate, containing the default label. Type the name “XOR1” into the text field, and press “Enter”. Similarly, label the rightmost XOR gate “XOR2”.

14. When you have finished entering text, click on the“arrow icon” in the Drawing toolbar.

15. From the parts palette find and select a 2-input AND gate. Place one such gate approximately two grid-squares below the XOR1 gate. Place a second AND gate about 1 grid square below the XOR2 gate.

    Your design sheet should now have four gates on it. Label the leftmost AND gate “AND1” and the rightmost gate “AND2”.

16. Find a 2-input OR gate in the parts palette and place one such gate about one grid square to the right of the rightmost AND gate.

    NOTE: You can place a component anywhere and then move it to the desired location. Place the cursor arrow on the component and, while holding the left mouse button down, drag it to the desired location.

17. Although you have placed the necessary gates on your design, they have yet to be connected. Before doing so however, you must introduce some components that will generate input values for some of the gates, and some components that will display the signals generated at the some of the outputs. For this example, you will use switches and probes.

    A binary switch allows you to place a signal value (0 or 1) onto the signal line to which it is connected and a binary probe displays the current signal value (0 or 1) on the signal line to which it is connected.

    Return to the top of the parts palette window and select “Simulation IO.clf” rather than “Simulation Gates.clf”.

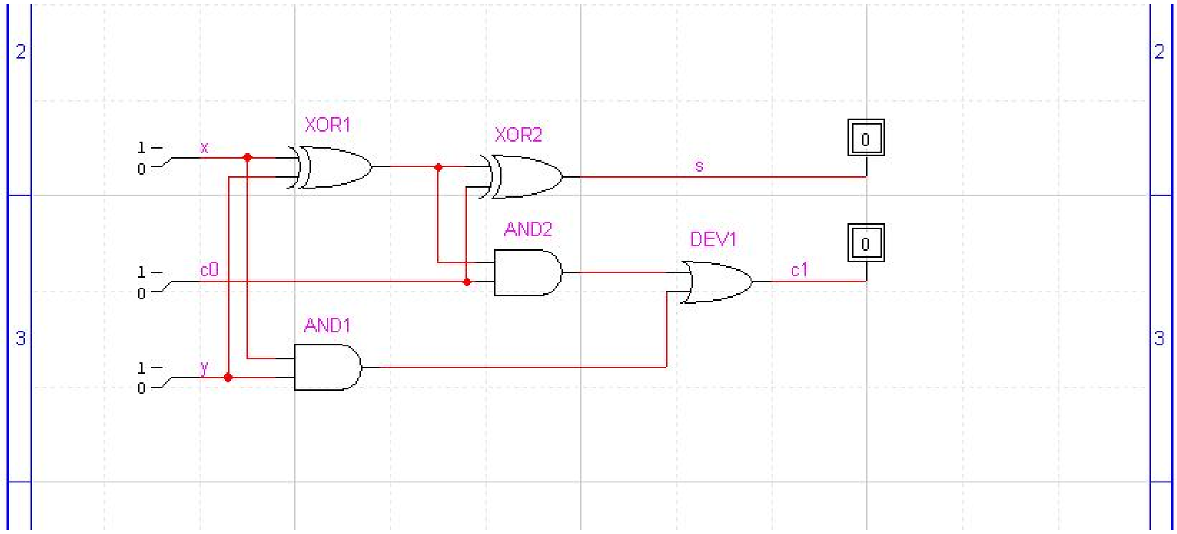
18. From the list of parts displayed, double click “Binary Probe” and place the part about three grid squares to the right of the output from the rightmost XOR gate. Place a second binary probe one grid square to the right of the output from the OR gate, underneath the first binary probe.

19. From the list of parts, select “Binary Switch” and place two switches on the design sheet. Place one switch about 2 grid squares to the left of the inputs of the “XOR1” gate and the second about one grid square to the left of the“AND1” gate.

20. Place a third binary switch between the first two switches as shown in the diagram below.

    Your resulting diagram so far should look similar to this diagram without the signal lines which have yet to be added.

21. If you wish, you can reposition your gates as described previously. Probes can be relocated in this way as well,.



To reposition input devices such as switches, you must first hold the shift key down before clicking but not releasing your mouse. The switch (or other input device) can then be dragged to the desired position. The reason for this approach with input devices is that normally you click on an input device to change the value of the input it is placing on the signal line to which it is connected.

22. Signal lines can now be drawn between components. To draw a signal line, place the cursor arrow point on the end of a component input or output, press the left mouse button, and **while holding the button down** draw a line to the input or output signal of another gate.

    Notice that it is possible to make one right-angled turn. Any signal line that ends up not connected will be terminated by a short line perpendicular to the signal line. Any signal line that terminates at another signal line will be connected to it and this connection will be indicated by a solid red dot. For additional turns, left-click your mouse before making the turn.

23. To delete a signal line, select the "lightning bolt" symbol from the drawing tools menu (between the arrow and the "A" symbols). Place the lower tip of the lightning bolt on the signal line to be deleted and click the left mouse button.

    Note that this only deletes a straight-line segment of a signal line. For signal lines with bends, repeat this process for each straight segment. The lightning bolt will remain until you return to the tools menu and select the arrow or "A" symbols.

24. Signal lines can be labelled the same way that components were labelled. Label the signal lines as indicated in the diagram above.

25. The labels on signal lines can be moved to more convenient locations. To do so, click on the label, causing it to be displayed in "reverse video". Then place the cursor arrow tip on the highlighted label, and drag it to the desired location while holding the left mouse button down.

26. Comments can be added to a schematic by selecting the "A" drawing tool, then pointing the cursor at any "open" space in the diagram and pressing the left mouse button. A text field will appear and any desired text can be entered. Pressing "enter" permits multi-line comments, so to exit, simply click the left mouse button again and the text field "box" will disappear.

27. Comments can be moved to other parts of the design sheet by placing the cursor on the text, clicking the left mouse button, and then dragging the text to the desired location.

28. Complete your digital design to obtain a schematic similar to that in the sample above. Save this schematic in a file named “FullAdder.cct”. The extension “.cct” identifies this as a “circuit file”.

29. To test your circuit, you need to launch the simulator. This is done by clicking on the “running man” symbol shown near the middle of the simulation toolbar at the top of the window.

    If you click on a binary switch, two things will happen. The switch will change state from “0” to “1” or vice-versa. At the same time, the value displayed on the binary probes may change to either “0” or “1”.

30. By trying various input values and observing the values on the outpus, confirm that the following function table defines the behavior of the 1-bit full adder digital system:

x y c0	c1	s
0 0 0	0	0
0 0 1	0	1
0 1 0	0	1
0 1 1	1	0
1 0 0	0	1
1 0 1	1	0
1 1 0	1	0
1 1 1	1	1