10-MINUTE TUTORIAL

DIGITAL LOGIC CIRCUIT MODELING AND SIMULATION WITH MULTISIM

Multisim is a schematic capture and simulation program for analog, digital and mixed analog/digital circuits, and is one application program of the National Instruments "Circuit Design Suite", which also includes printed circuit board design tools and an interface to the ELVIS breadboarding platform.

It is suggested that you begin by taking a few minutes to read the short *Multisim* tutorial, which is presented as Chapter 2 of "Getting Started with NI Circuit Design Suite", which can be accessed via:

Start Menu > All Programs > National Instruments > Circuit Design Suite 11.0 > Documentation > Getting Started

The basic steps in modeling and analysis of a digital logic circuit are:

- 1. Open *Multisim* and create a "design".
- 2. Draw a schematic diagram of the circuit (components and interconnections).
- 3. Define digital test patterns to be applied to the circuit inputs to stimulate the circuit and connect signal sources to the inputs to produce these patterns.
- 4. Connect the circuit outputs to one or more indicators to display the response of the circuit to the test patterns.
- 5. Run the simulation and examine the results, copying and pasting *Multisim* windows into lab reports and other documents as needed.
- 6. Save the design.

Step 1. Open Multisim and create a design

Multisim is opened from the Start Menu:

Start Menu > All Programs > National Instruments > Circuit Design Suite 11.0 > Multisim 11.0 (Version numbers may differ)

This creates a blank design called "Design1", as illustrated in Figure 1. Save the file with the desired design name via menu bar **File>Save As** to use the standard Windows Save dialog, shown in Figure 2. Navigate to the directory in which you want to save your design, enter the desired file name, and click the **Save** button. The default file extension for Multisim 11.0 design files is *.ms11*.

Note: Create a separate directory on a flash drive or your network drive (H:) to be used exclusively for storing your Multisim designs and related files. This will ensure that (1) your files will be accessible from any lab computer and (2) your Multisim files will not be mixed in with your other files. It is also a good idea to periodically make backup copies of your files as protection against loss of data.

A previously created design can be opened via **File>Open**. In the dialog window, navigate to the directory in which the design is stored, select the file, and click the **Open** button.



Figure 1. Blank design with default name "Design 1"



Figure 2. "File > Save As" dialog window. Design to be saved as file *ELEC2210_Lab2.ms11*.

Step 2. Draw a schematic diagram of the circuit

Placing Components

A schematic diagram comprises one or more circuit components, interconnected by wires. Optionally, signal "sources" may be connected to the circuit inputs, and "indicators" to the circuit outputs. Each component is selected from the *Multisim* library and placed on the drawing sheet in the Circuit Window (also called the Workspace). The *Multisim* library is organized into "groups" of related components (Transistors, Diodes, Misc Digital, TTL, etc.). Each group comprises one or more "families", in which the components are implemented with a common technology. For designing and simulating digital logic circuits in ELEC 2210, two groups are to be used: "Misc Digital" (TIL family only) and "TTL".

The "Misc Digital" group has three families of components, of which family "TIL" contains models of generic logic gates, flip-flops, and modular functions. These components are technology-independent, which means that they have only nominal circuit delays and power dissipation, unrelated to any particular technology. Generic components can be used to test the basic functionality of a design, whereas realistic timing information requires the use of technology-specific part models, such as those in the TTL group.

To place a component on the drawing sheet, select it via the Component Browser, which is opened via the component toolbar or the menu bar. From the menu bar, select **Place>Component** to open the Component Browser window, illustrated in Figure 3. You can also open this window by clicking on the **Misc Digital** icon in the component toolbar. On the left side of the window, select "Master Database", group "Misc Digital", and family "TIL". The component panel in the center lists all components in the selected family. Scroll down to and click on the desired gate (NAND2 in Figure 3); its symbol and description are displayed on the right side of the window. Then click the OK button. The selected gate will be shown on the drawing sheet next to the cursor; move the cursor to position the gate at the desired location, and then click to fix the position of the component. The component can later be moved to a different location, deleted, rotated, etc. by right clicking on the component and select the desired action. You may also select these operations via the menu bar **Edit** menu.

After a component has been positioned, the component browser is redisplayed and additional components can be placed by repeating the above actions. When the last component has been placed, click the **Close** button to close the component selection window. You may return to the component browser at any time to add additional components.

🗃 Select a Component				
Database:	Component:	Symbol (ANSI)		
Master Database 🛛 🗸	NAND2			
Group:	LA_CARRY_GEN			
🚮 Misc Digital 🔽 🗸	MAG_CMP4	Search		
Family:	MICRO_4BIT_ALU	Detail report		
All Select all families	MUX_16TO1	View model		
	MUX_2TO1	Help		
	MUX_4TO1			
	MUX_8TO1	Function:		
LINE_TRANSCEIVER	MUX_LATCH			
	NAND2			
	NAND3			
	NANDS			
	NAND6	i Model manufacturer (ID)		
	NAND7	IIT / NAND2		
	NAND8			
	NOR2			
	NOR3	Footprint manufacturer (type)		
	NOR4			
	NOR5			
	NOR6			
	NOR7	Hyperlink:		
	NOR8			
Components: 105 Searching:				

Figure 3. Component Browser: Misc Digital TIL family NAND2 gate component selected.



Figure 4. A third NAND2 gate is about to be placed on the drawing sheet.

The TIL family part naming convention is as follows. Look at the symbol and function panels on the right side of the component selection window to determine the specific function of a selected component.

- 1. Basic logic gates: ANDx, NANDx, NOT, NORx, ORx, XNORx, XORx, where x is the number of gate inputs (2-8).
- Flip flops: D_FF, JK_FF, SR_FF, T_FF, and latches D_LATCH, SR_LATCH, plus versions that have active-high or active-low asynchronous Set and Reset pins (ex. D_FF_POSSR and D_FF_NEGSR, respectively).
- 3. Standard digital modules: multiplexers, decoders, encoders, counters, ALU, BCD-to-7 segment decoder, registers, shift registers, etc.
- 4. DIGITAL_PULLUP, DIGITAL_PULLDOWN to pull a pin to a constant logic 1 and 0, respectively.

Figure 5 shows the schematic diagram with four placed components. Note that each placed gate has a "designator" (U2, U3, U4, U5), which can be used when referring to that gate. You can change a designator by right clicking on the component, selecting Properties, and entering the desired name on the Label tab.



Figure 5. Schematic diagram with all placed components.

Drawing Wires

Wires are drawn between component pins to interconnect them. Moving the cursor over a component pin changes the pointer to a crosshair, at which time you may click to initiate a wire from that pin. This causes a wire to appear, connected to the pin and the cursor. Move the cursor to the corresponding pin of the second component (the wire follows the cursor) and click to terminate the wire on that pin. If you do not like the path selected for the wire, you may click at a point on the drawing sheet to fix the wire to that point and then you can move the cursor to continue the wire from that point. You may also initiate or terminate a wire by clicking in the middle of a wire segment, creating a "junction" at that point. This is necessary when a wire is to be fanned out to more than one component input. A partially-wired circuit, including one junction point, is illustrated in Figure 6.



Figure 6. Partially wired circuit, with one junction point.

Step 3. Generating test input patterns.

To drive circuit simulations, *Multisim* provides several types of "sources" to generate and apply patterns of logic values to digital circuit inputs. Sources are placed on the schematic sheet and connected to circuit inputs in the same way as circuit components, selecting them from the "Digital_Sources" family of the "Sources" group in the component browser. Note that there is a **Place Source** shortcut icon in the tool bar.

There are three basic digital sources:

- 1. DIGITAL_CONSTANT this is a box with a constant logic 1 or 0 output, and would be used where the logic values is not to be changed during simulation. To change the output value, right click on the box, select Properties, select the desired value on the Value tab, and click the OK button.
- 2. INTERACTIVE_DIGITAL_CONSTANT this is a clickable box that can be connected to a circuit input. Clicking on the box toggles its output between 0 and 1. This can be used to interactively change a circuit input during simulation.
- DIGITAL_CLOCK this is a box that produces a repeating pulse train (square waveform), oscillating between 0 and 1 at a specified frequency. To set the frequency and duty cycle, right click on the box, select Properties, select the desired frequency and duty cycle value on the Value tab, and click the OK button.

Figure 7 shows the circuit of Figure 6 with an INTERACTIVE_DIGITAL_CONSTANT connected to each input. Note that the initial state of each is logic 0. Since this circuit has only three inputs, all 8 input patterns can be produced (to generate a truth table for the circuit) by manually toggling the inputs.



Figure 7. INTERACTIVE_DIGITAL_CONSTANT sources connected to circuit inputs.

Another mechanism to apply all possible input combinations to a circuit is to connect a DIGITAL_CLOCK to each input, with the frequency of the first clock set to some value N, the second to 2N, the third to 4N, etc. This produces the pattern shown in Figure 7. Looking at these waveforms, it can be seen that the initial pattern is 000, changing to 001, 010, etc.



Figure 7. Three clocks used to produce a sequence of 8 digital patterns

Of course, DIGITAL_CLOCK sources would also be used to drive the clock inputs of sequential circuits.

One additional mechanism for generating digital patterns is the "Word Generator" instrument, which can be added to the circuit via the shortcut icon on the right side of the main window, or via the menu bar **Simulate>Instruments>Word Generator**. Figure 8 shows a Word Generator (XWG1) connected to the three circuit inputs. The Word Generator produces a sequence of patterns, each containing from 1 to 32 bits.

To specify the patterns and the rate at which they should be produced, double click on the Word Generator symbol, producing the window shown in Figure 9. The Control buttons on the left of this window allow patterns (1) to be continuously applied and repeated (Cycle button), (2) a single set of patterns to be applied (Burst button), or (3) a single pattern to be applied (Step button). The patterns displayed in this window will be applied in the order listed, and can be entered manually or generated automatically. For automatically-generated patterns, press the Set button, producing the Settings window of Figure 10. In this example, "Up Counter" is selected to produce a sequence of binary numbers, "Buffer Size" is set to 8 to limit the sequence to 8 numbers, and "Initial pattern" is set to 0. Note that the 8 binary numbers are displayed in the Word Generator window of Figure 9. The rate at which the patterns are to be applied to the circuit is specified via the Frequency box of the Word Generator window. In Figure 9, the frequency has been set to 1KHz, which means that 1000 patterns will be generated each second, or one pattern every 1msec.

XW61	••••••••••••••••••••••••••••••••••••••
	XIA1
	······································
	NOT NAND2
	NAND2
R T	
: : 	



Word Generator-XWG1			
Controls Display Opec Step Display Dec Dec Display Dec Dec Display Dec Dec Display Dec Display Dec Dec Display Dec Display Dec Display Display Dec Display Dec Display Display Dec Display Dec Display Dec Display Dec Display Dec Display Dec Display Dec Display Dec Display Display Dec Display Display Display Dec Display D	▼ 0000000 00000001 00000002 00000003 00000004 ● 00000005 00000006 1 00000007		
Ready Trigger			
31 ©©©©©©©©©©©©©©©©©©©©©©©©©©©©©©©			

Figure 9. Word Generator window showing count up pattern



Figure 10. Word Generator Settings window

Step 4. Connect circuit outputs to indicators

To facilitate studying the digital circuit output(s), *Multisim* provides a variety of "indicators". For digital simulation, the most useful are digital "probes", hex displays, and the Logic Analyzer instrument. A probe, illustrated in Figure 11, displays a single digital value as ON or OFF (the probe is "illuminated" in Figure 11, indicating an ON condition). The PROBE family of the Indicators group includes a generic PROBE_DIG and several PROBE_DIG_color indicators (color = BLUE, GREEN, ORANGE, RED, YELLOW). The probe in Figure 11 is PROBE_DIG_BLUE. This circuit can be verified by manually changing the three INTERACTIVE_DIGITAL_CONSTANT inputs to each of the 8 possible combinations, and recording the probe value for each combination to create a truth table.



Figure 11. DIGITAL_PROBE_BLUE connected to circuit output.

The HEX_DISPLAY family contains a variety of displays that show multi-bit values as hexadecimal numbers. This would be useful for such circuits as binary counters, as illustrated in Figure 12. In this example, the DCD_HEX_GREEN indicator is a "decoded" hex display digit, which means that the indicator displays the hex number corresponding to the states of its four inputs.



Figure 12. DCD_HEX_GREEN decoded hex indicator displays a counter output.

Another useful indicator is the Logic Analyzer instrument, shown in Figures 8 and 12. A logic analyzer is an instrument that captures and displays sequences of digital values over time, with the sequences displayed as waveforms rather than as tables of numbers. The *Multisim* Logic Analyzer instrument can capture and display up to 16 signals. Samples are triggered either by an internal clock (N samples per second) or by an external clock. Figure 13 shows the Logic Analyzer display of the three inputs and the output of the circuit in Figure 8, showing the response of the circuit to the burst of Word Generator values shown in Figure 9, plus the internal clock that was used to capture samples.

The logic analyzer must be configured to capture values at the correct sampling times. This is done by clicking on the Set button in the Clock area under the logic analyzer display, opening the Clock Setup window shown in Figure 14. Since the word generator was configured to generate patterns at a rate of 1KHz, a clock rate of 2KHz was set for the logic analyzer to make it capture two values for each input pattern.



Figure 13. Logic Analyzer display of the 3 circuit inputs and circuit output of Figure 8, for the 8 patters produced by the Word Generator.

Clock Setup		×		
Clock source O External O Inte Clock rate		<u>Accept</u> <u>Cancel</u>		
2 kHz X				
Pre-trigger samples:	1			
Post-trigger samples:	100000			
Threshold volt. (V):	2.5	V		

Figure 14. Logic analyzer sampling clock setup.

Step 5. Run the simulation

A simulation is initiated by pressing the Run (green arrow) button in the toolbar or via the menu bar via **Simulate>Run**. Alternatively, simulation can be initiated from a Word Generator by pressing the Cycle, Burst, or Step buttons.

You may capture any window and paste it into a Word or other document for generating reports. An individual window is captured by pressing the ALT and Print Screen keys concurrently. You may then

"paste" the captured window into a document via the editing features of that document. To capture a circuit diagram in the main window, the simplest method is via the menu bar Tools>Capture Screen Area. This produces a rectangle whose corners can be stretched to include the screen area to be captured; the "copy" icon on the top left corner is pressed to copy the area, which may then be pasted into a document. The circuit images in Figures 8 and 12 were copied in this manner.

Step 6. Save the design and close Multisim

The simplest way to save a design is to click the Save icon in the Design Toolbar on the left side of the window, directly above the design name. Alternatively, you may use the standard menu bar **File>Save**. As mentioned earlier, you should save all designs in a special course directory on either your network (H:) drive or on a flash memory device.

Multisim is exited as any other Windows program.