Queensborough Community College

The City University Of New York

Department of Engineering Technology

ET 540 - Introduction to Digital Computer Theory Lab

Lab 2: Introduction to Multisim: Digital Circuit Design

COMPONENTS

Multisim

INTRODUCTION

Multisim is a schematic capture and simulation application that assists you in carrying out the major steps in the circuit design flow. Multisim can be used for both analog and digital circuits and also includes mixed analog/digital simulation capability, and microcontroller co-simulation. Simulating the circuits before building them, catches errors early in the design flow, saving time and money. The Multisim's user interface and its main elements can be seen in Figure 1.1

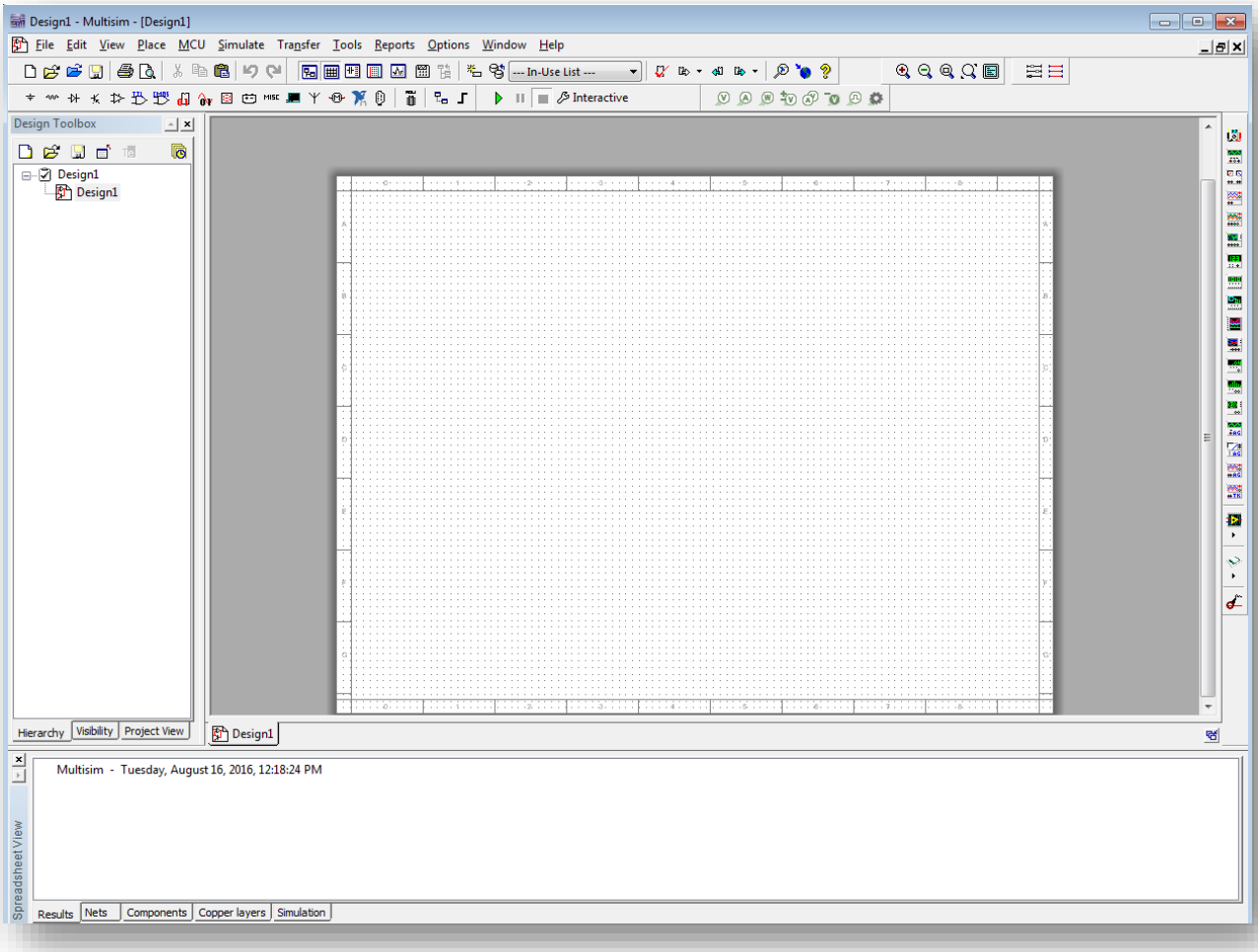
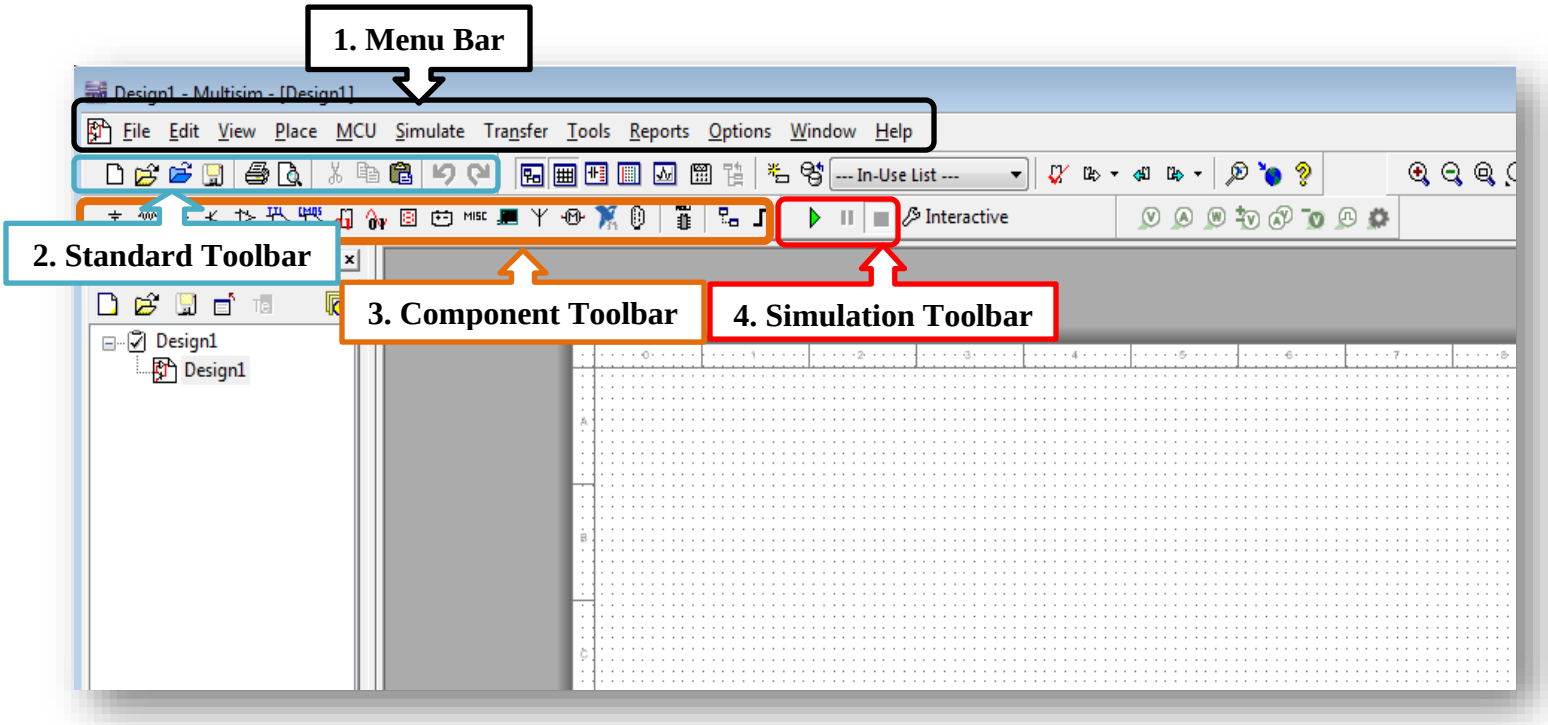


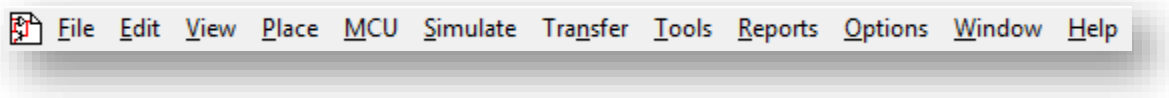
Figure 1.1 – Multisim Interface

MULTISIM INTERFACE



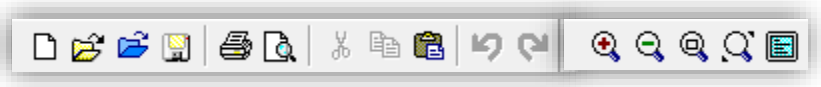
1. Menu Bar

Menu bar contains the tabs or commands for all main functions: File, Edit, View, Place, MCU, Simulate, Transfer, Tools, Reports, Options, Window, and Help



2. Standard Toolbar

The standard toolbar contains buttons for commonly-performed functions: New, Open, Open Sample, Save, Print Circuit , Print Preview, Cut, Copy, Paste, Undo, Redo, Zoom In, Zoom Out, Zoom to Specific Area, Zoom Sheet, and Full Screen button

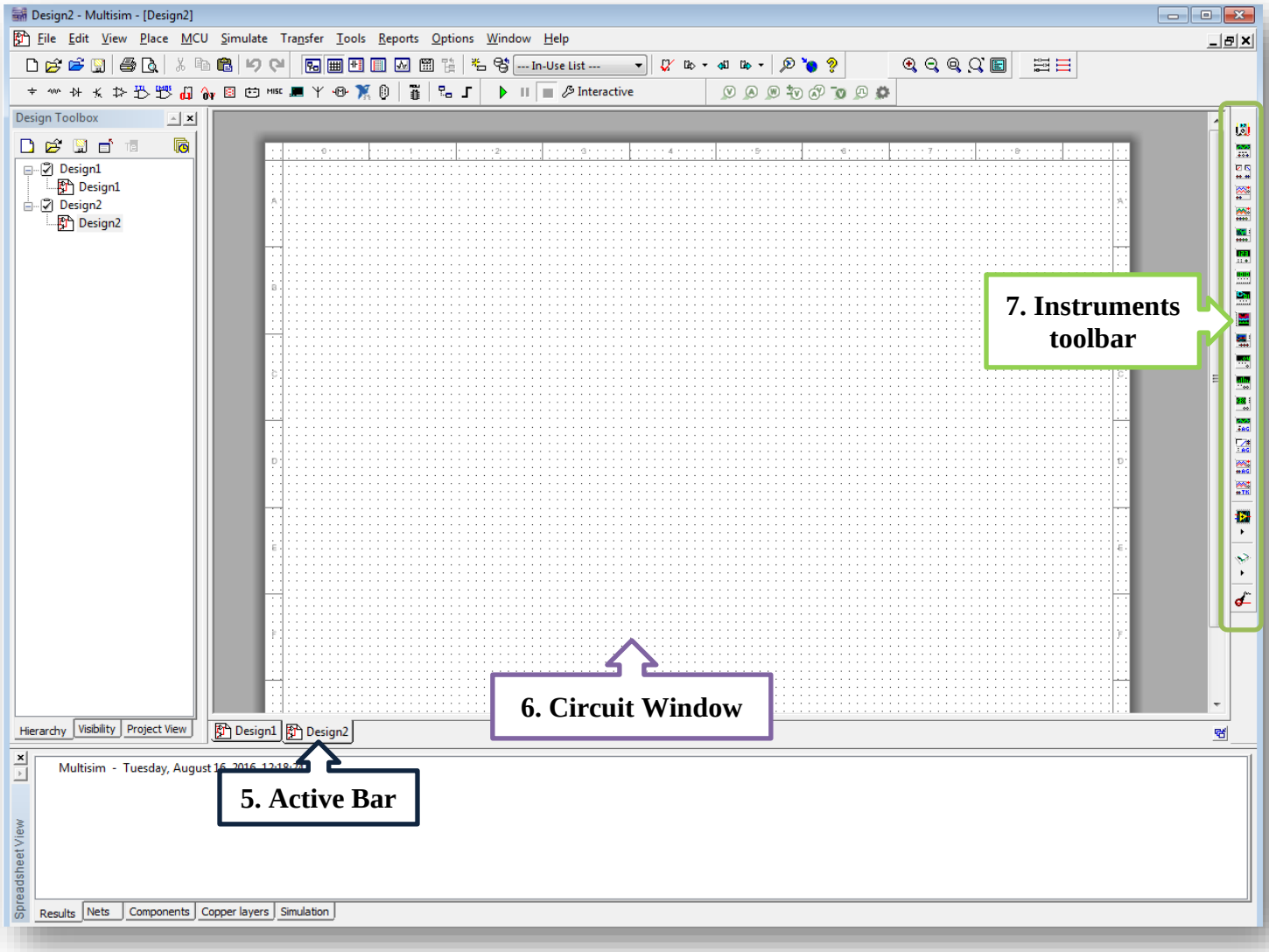


3. Component Toolbar

Component toolbar contains button that launches to the component browser of a selected Group: Source, Basic, Diode, Transistor, Analog, TTL (Transistor-Transistor-Logic), CMOS (Complementary metal–oxide–semiconductor), Mixed, Indicator, Power Component, Miscellaneous, Advance peripherals, RF, Electromechanical, Educational resources, and Connectors button

4. Simulation Toolbar

Simulation toolbar contains the buttons to run, pause, or stop the simulation of the circuit.



5. Active Bar

Active bar shows the current workspace.

6. Circuit Window

Circuit window is the active workspace where the circuit is built.

7. Instruments Toolbar

Instruments toolbar contains buttons that place a specific instrument on the workspace: Multimeter, Function generator, wattmeter, oscilloscope, four channel oscilloscope, bode platter, frequency counter, word generator, logic converter, logic analyzer, IV analyzer, distortion analyzer, spectrum analyzer, network analyzer, agilent function generator, Agilent multimeter, Agilent oscilloscope, Textronics oscilloscope, and LABView instructions.

EXPERIMENTAL PROCEDURE

For this lab exercise, you will be introduced to Miltisym by creating a digital counter circuit. During the lab, you will find, place, and connect components on the workspace. Furthermore, you will also simulate or run and operate the circuit.

The complete circuit is shown below, Figure 1.2

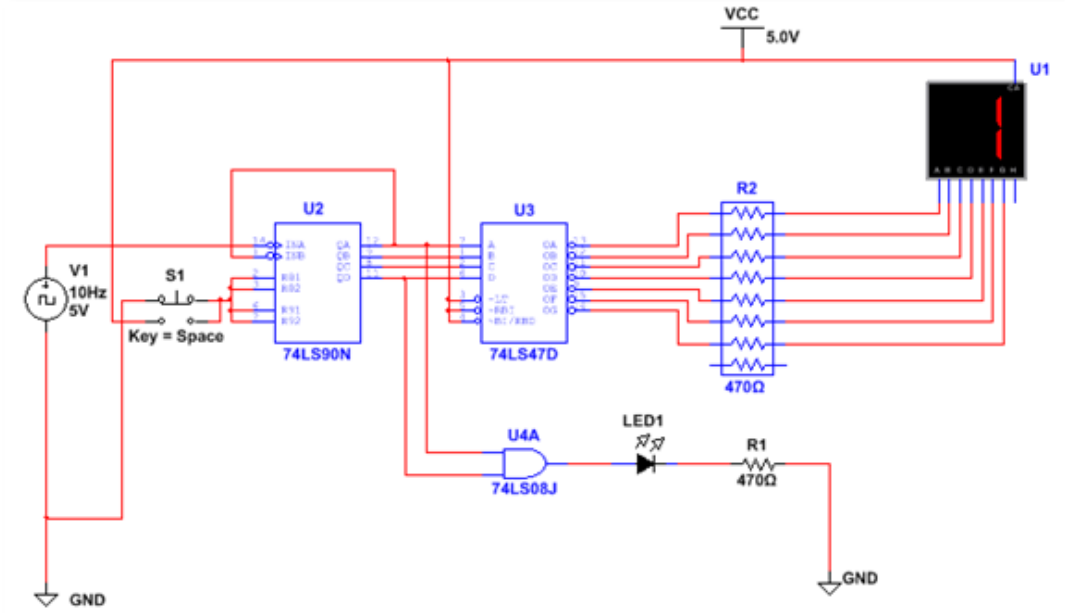
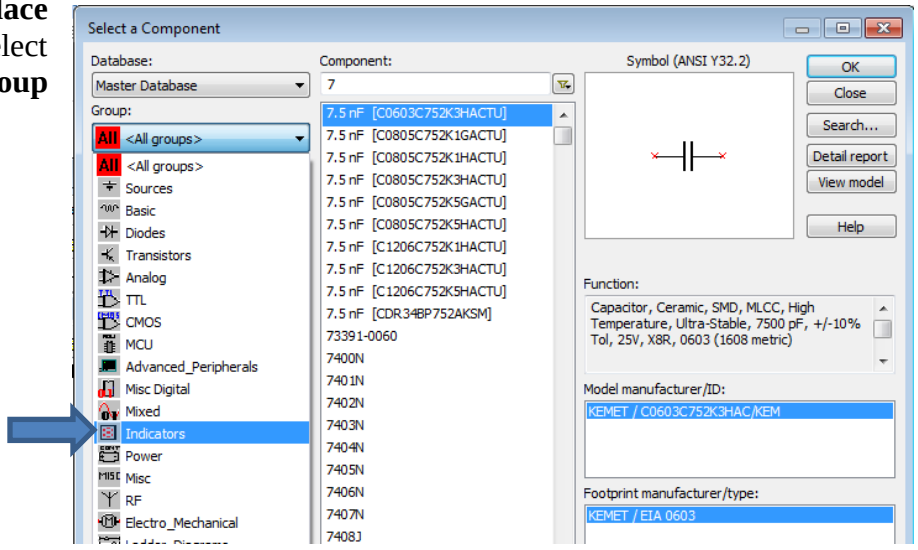


Figure 1.2 – Digital Counter

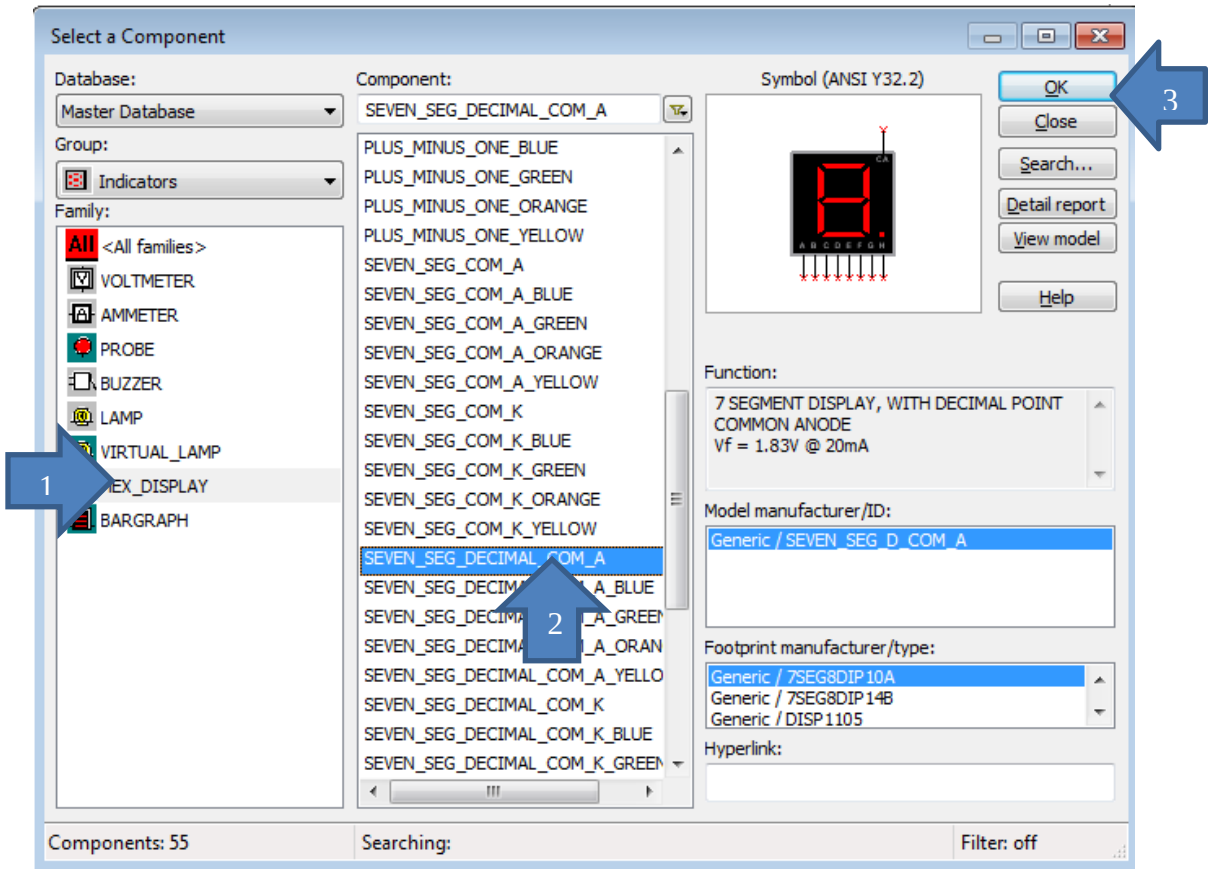
Steps:

1. **Opening a circuit design workspace**. Go to Start ➔ NI Multisim 14.1.
2. **Saving the workspace file**. Select File ➔ Save As … Save the file with the student's last name, for example, if your name is John Lee, save the multisim file as "Lee_Lab2". Make sure that the **Save as Type** is selected as **Multisim 14 files**, and the file location should be in your personal storage drive. If you don't have a personal storage drive, save the file at the computer desktop.

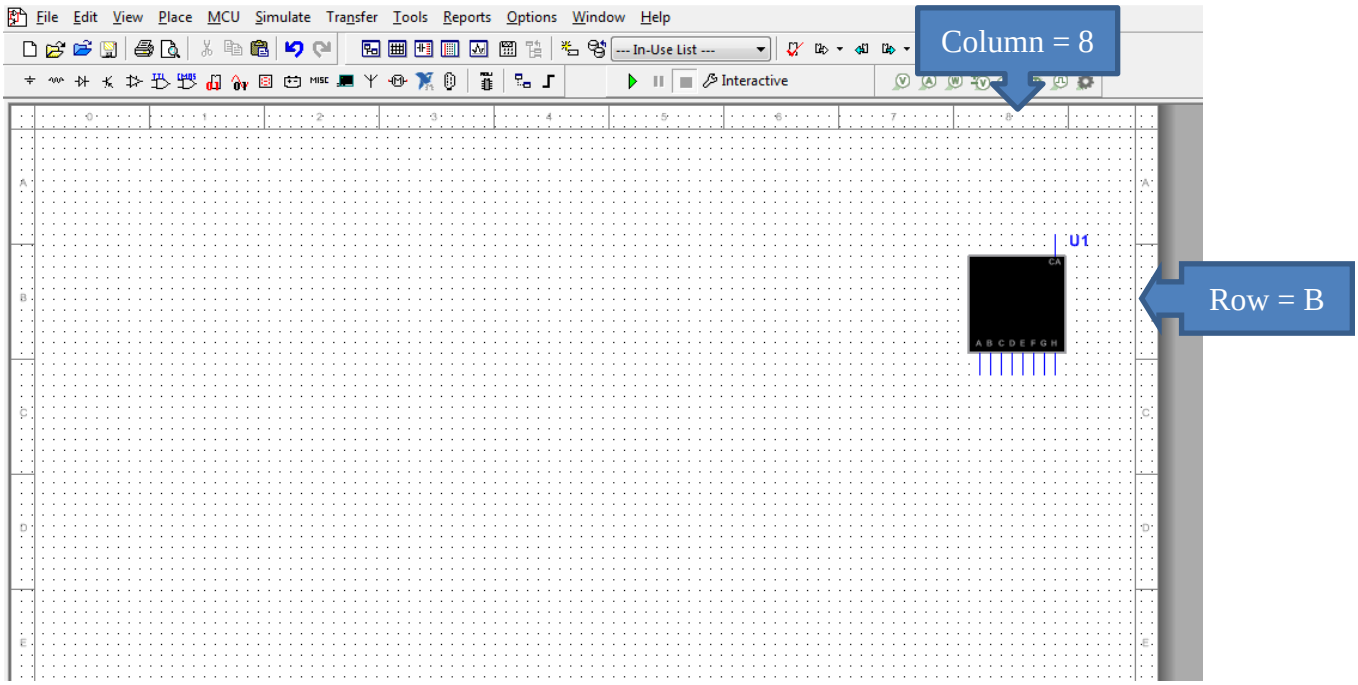
3. **Placing components.** Select **Place ➔ Component** to display the Select a Component browser. In the **Group** list, select **Indicators**.



Once the Indicator group is selected, select **HEX_DISPLAY** from the Family list. Now with the HEX_DISPLAY selected, navigate the component list and select **SEVEN_SEG_DECIMAL_COM_A**, and click Ok



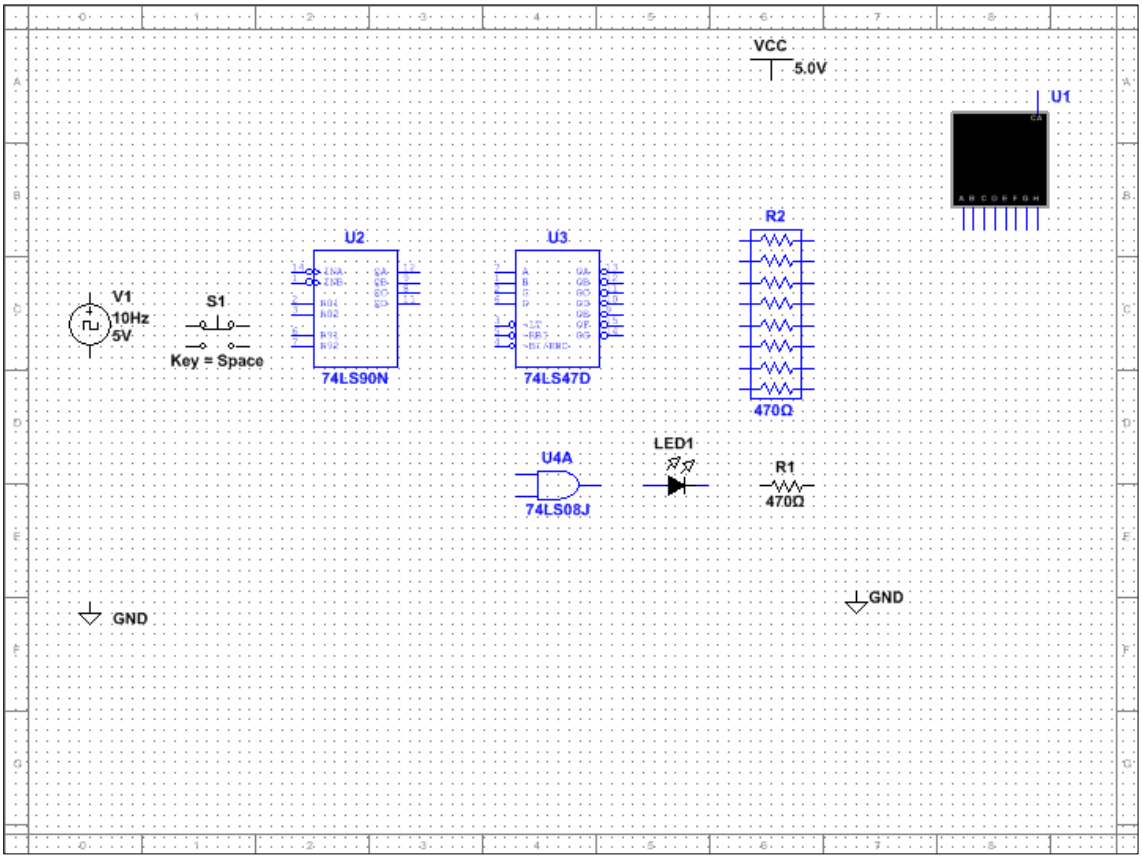
The component appears as a ghost on the cursor. Move the component to position B8 in the workspace and click to release the component



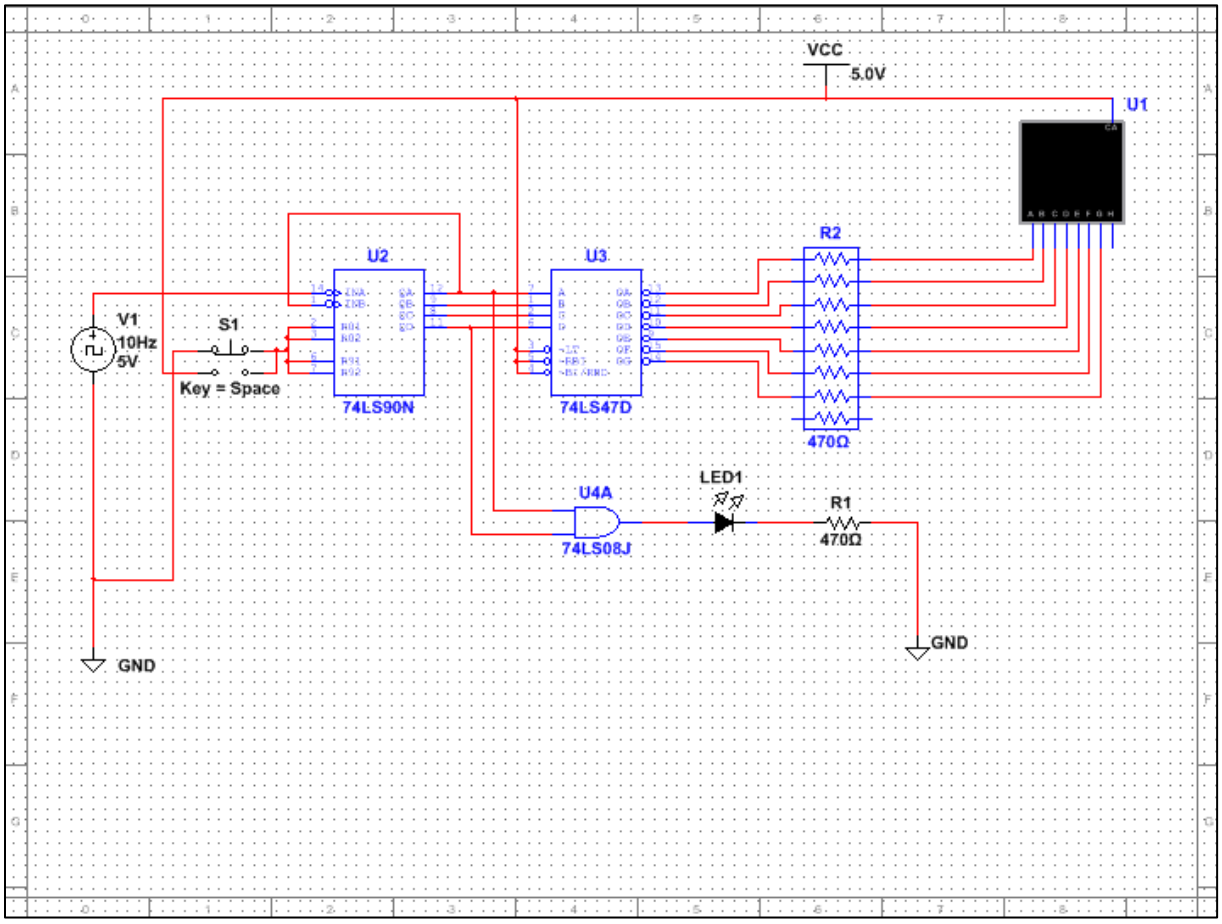
Following the previous step, place the remaining components in the workspace.

List	Group	Family	Component	Location
LED	Diodes	LED	**LED_Red** *Note: Rotate the LED using the key combination "Ctrl + r"*	5E
Signal Voltage Source	Sources	Signal Voltage Source	**Clock Voltage**	0C
Transistor Voltage Sources	Sources	Power Sources	**VCC**	6A
Ground	Sources	Power Source	**DGND** *Note: need 2 of them. One is placed in cell 0F and the other one in cell 7F*	0F, 7F
8 line pack of Resistors	Basic	RPACK	**8Line_Isolated** *Note: double click the value "1k Ω" and change it to 470 Ω*	Between 6C and 6D
Resistor	Basic	Resistor	**470 Ω**	6E
Integrated Circuit	TTL	74LS	**74LS90N**	Between 2C and 3C
Integrated Circuit	TTL	74LS	**74LS47D**	4C
Integrated Circuit	TTL	74LS	**74LS08J** *Note: when a window appears, click on A from the list.*	4E
Push-button	Electromechanical	Supplementary Switches	**PB_DPST**	1C

Your workspace should look like the following:



4. **Wiring the circuit.** To wire the components, click at the end of a terminal of a component to start the connection (the pointer turns into a crosshair) and move the mouse. A wire appears attached to your cursor. Click on the terminal on the second component to finish the connection. Multisim automatically places the wire, which conveniently snaps to an appropriate configuration. You can also control the flow of the wire by clicking on points as you move the mouse. Each click "fixes" the wire to that point. Wire your circuit as the following:



5. **Running the simulation.** Select **Simulate / Run** or press the Run button in the simulation toolbar . As the circuit simulated the 7-segment display counts up and the LED flashes at the end of each count cycle. The pushbutton is an interactive component to reset the counter to zero. To slow the display of the counter, double click on the signal voltage source and change the frequency to a lower frequency: try it with 1 Hz, 10 Hz, and 1 kHZ. Remember, the simulation must stop before changing the frequency of the signal voltage source.

6. **Inserting title block.** Click on **Place / Title Block / DefaultV6.tb7.** Position the title block at the lower right corner of the workspace and click to settle the position. Double click on the title block and fill up the block with the following information:

   Title: Digital Counter Circuit
   Description: Lab 2: Introduction to Multisim: Digital Circuit Design
   Designed by: Student Name (type your name)
   Date: Insert the date

7. **Finalizing the workspace.** Take the grid off the workspace by clicking on **View** and uncheck the **Grid** box. The final workspace should look as Figure 1.3:

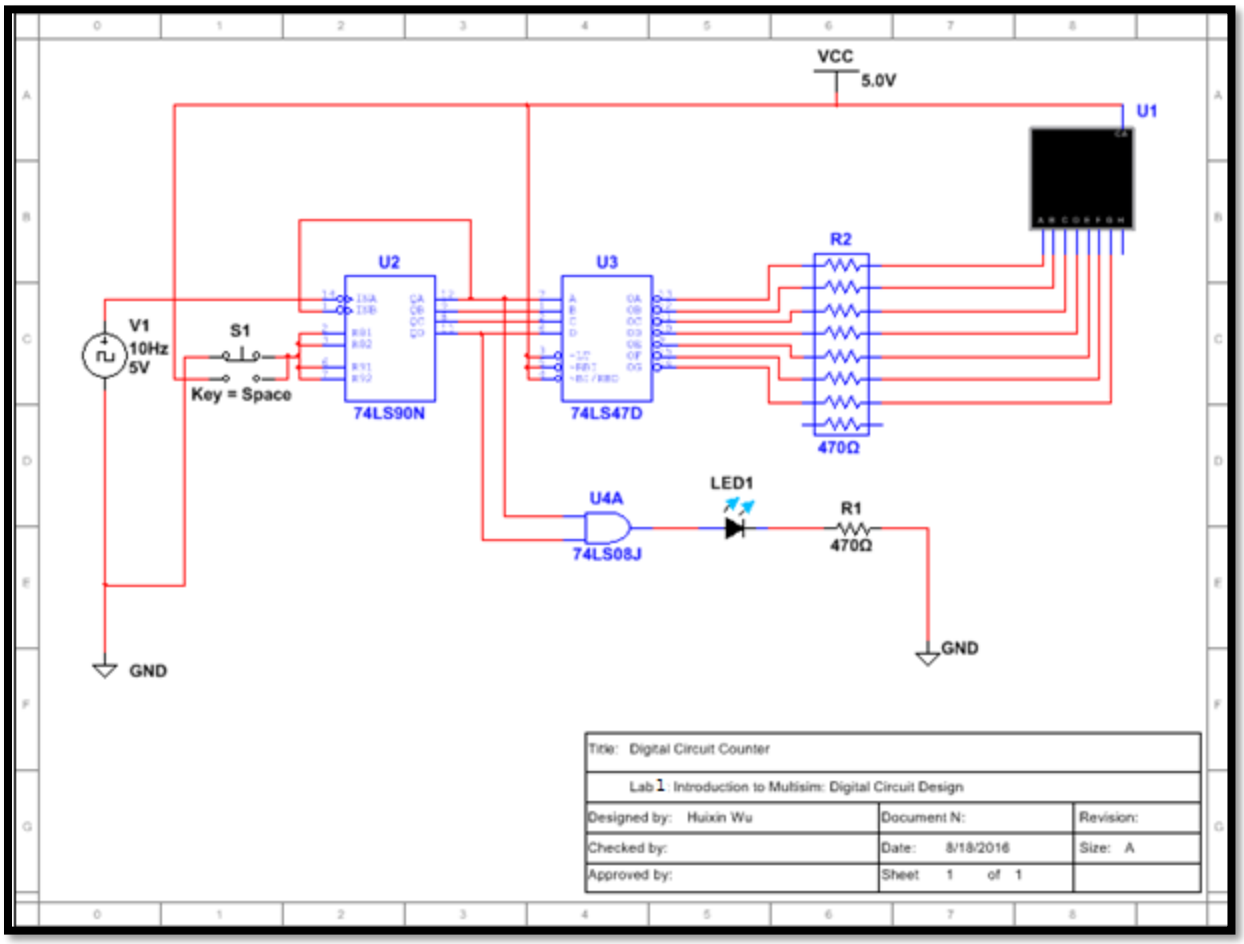


Figure 1.3 – Complete Digital Counter in Multisim

QUESTIONS

Multisim Envelopment

1. In which tab from the main menu the command **Junction** is located?

2. Once the circuit is built in Multisim, mention three different ways that you can run the circuit simulation?

3. Fill up the following table with the corresponding Group and Family of the given components:

Component	Group	Family
Capacitor: 1 µ Capacitor		
Power Source: AC Current		
TTL logic Gate: Hex Inverter – 74LS04N		
Ground: digital ground		
Switch: Single-Pole, Double-Throw (SPDT) switch		
Potentiometer: 2 kΩ potentiometer		
Resistor Pack: 7 resistor isolated resistor pack		

Circuit data analysis and results

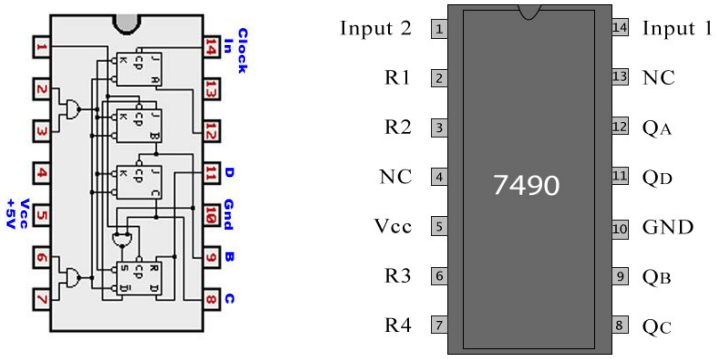
4. What is the behavior of the digital counter if the input frequency is decreased to 1 Hz? and when is increased to 1 kHz? *Hint: run the circuit built in Multisim and test it with 1kHz and then with 1Hz. Don't forget to stop the simulation to change the frequency.*

5. What is the function of the push button switch in the digital counter circuit? *Hint: run the circuit built in Multisim and press the push -button, or press the space key from the keyboard, and observe the behavior of the digital counter.*

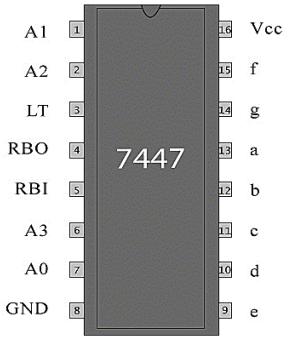
Student's name: _____ Lab instructor signature: _____

APPENDIX

74LS90N - Decade and 4-bit Binary Counters: Type of digital counter which can count from 0 to 9.



74LS47D - BCD to 7- Segment Decode: It is an encoding in which each digit of a number is represented by its own binary sequence of 4-bit



---------- LAB EXPERIMENT ENDS HERE, PROCEED WITH LAB REPORT ----------