

Lab Experiment 2

Using Multisim to Build Digital Circuit

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 2.1

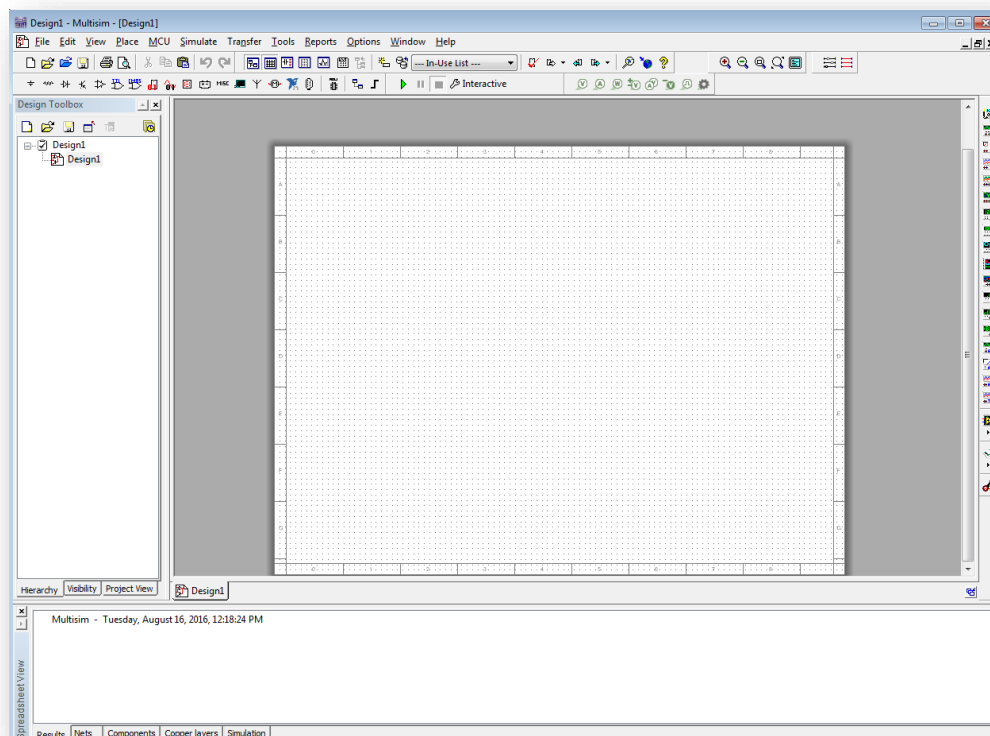
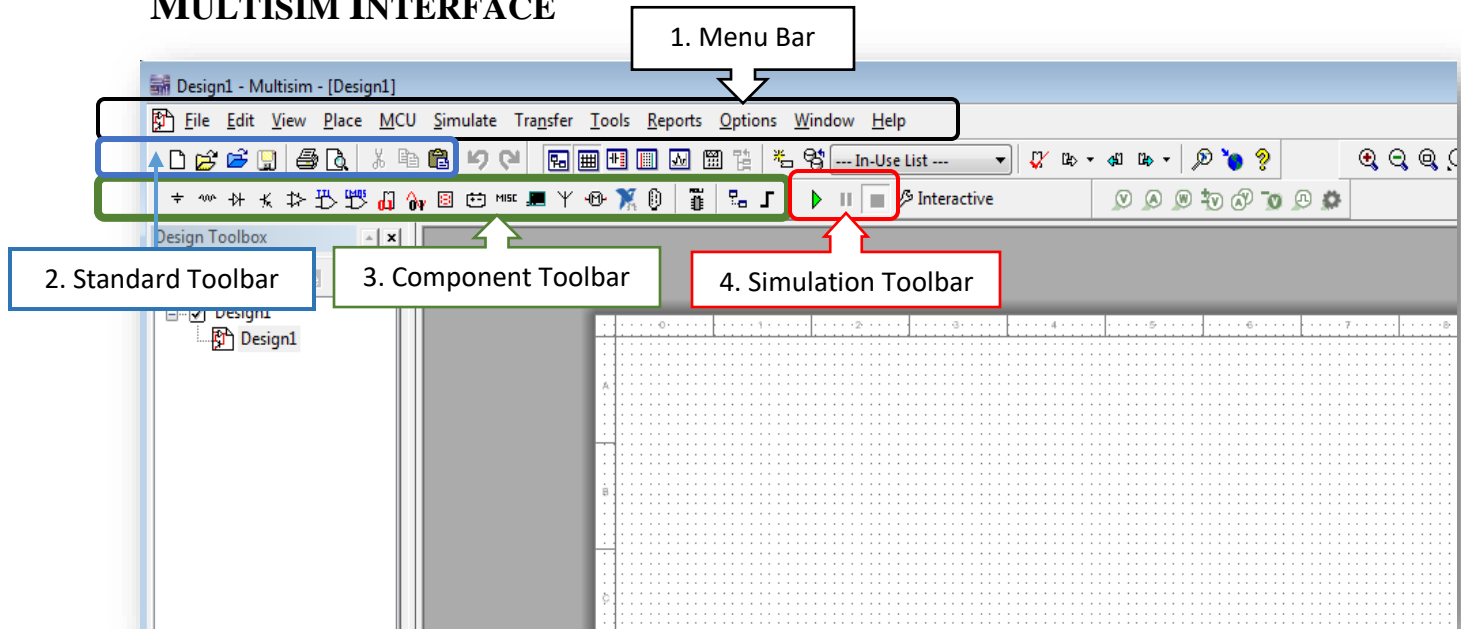


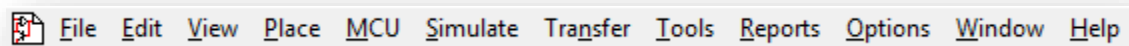
Figure 2.1 – Multisim Interface

MULTISIM INTERFACE



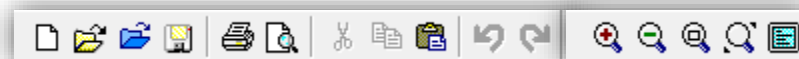
1. Menu Bar

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

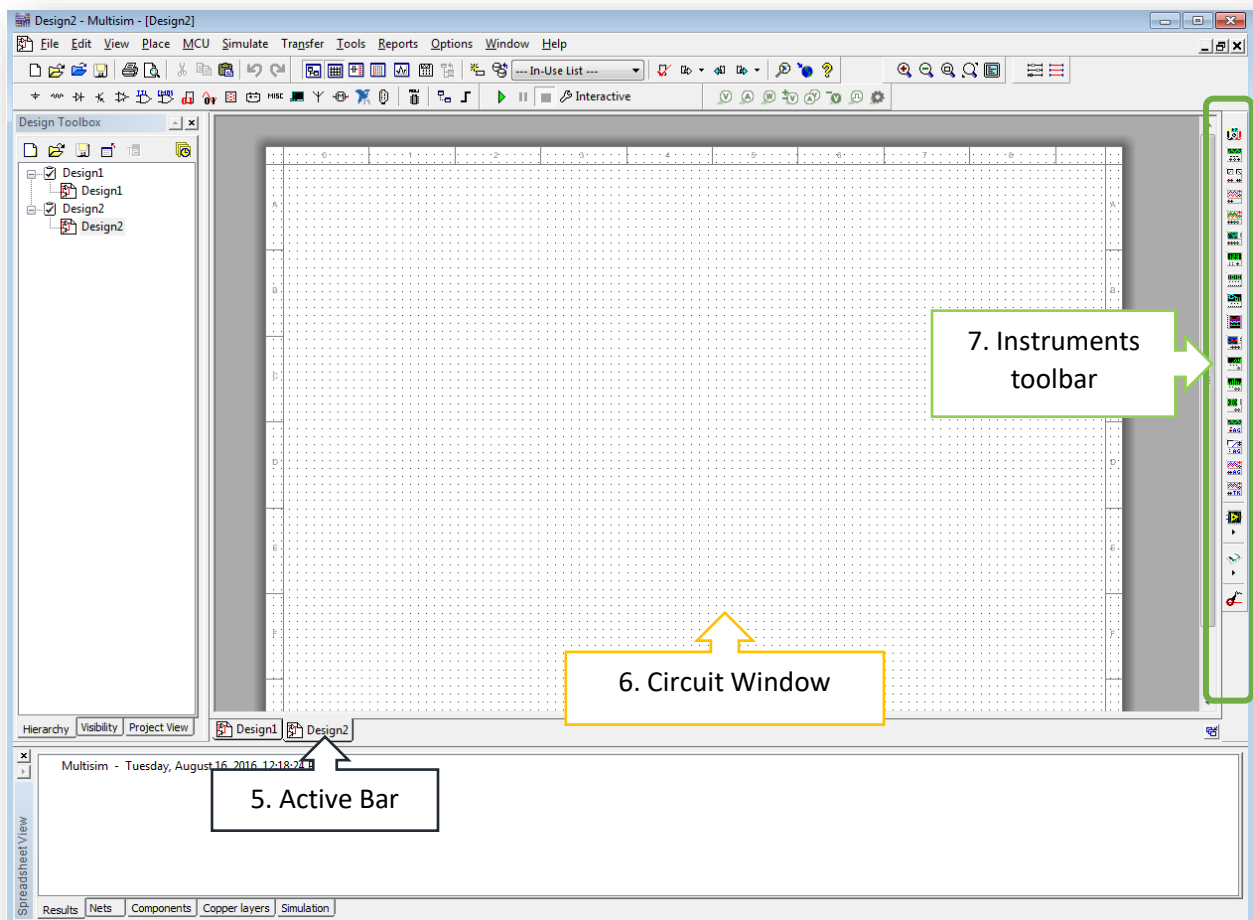
Component toolbar contains button that launches to the component browser of a selected Group: Source, Basic, Diode, Transistor, Analog, TTL (Transistor-Transistor-Logic), CMOS (Complementary metal-oxide-semiconductor), Mixed, Indicator, Power Component, Miscellaneous, Advance peripherals, RF, Electromechanical,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

LAB EXPERIMENT PROCEDURE

For this lab exercise, you will introduce to create a digital circuit counter in Multisim. During the lab, you will expose to find, place, and connect components in the workspace. Indeed, you will also experiment how to run, troubleshoot, and analyze a complete circuit using the simulation toolbar.

The complete circuit will be as shown below, Figure 2.2

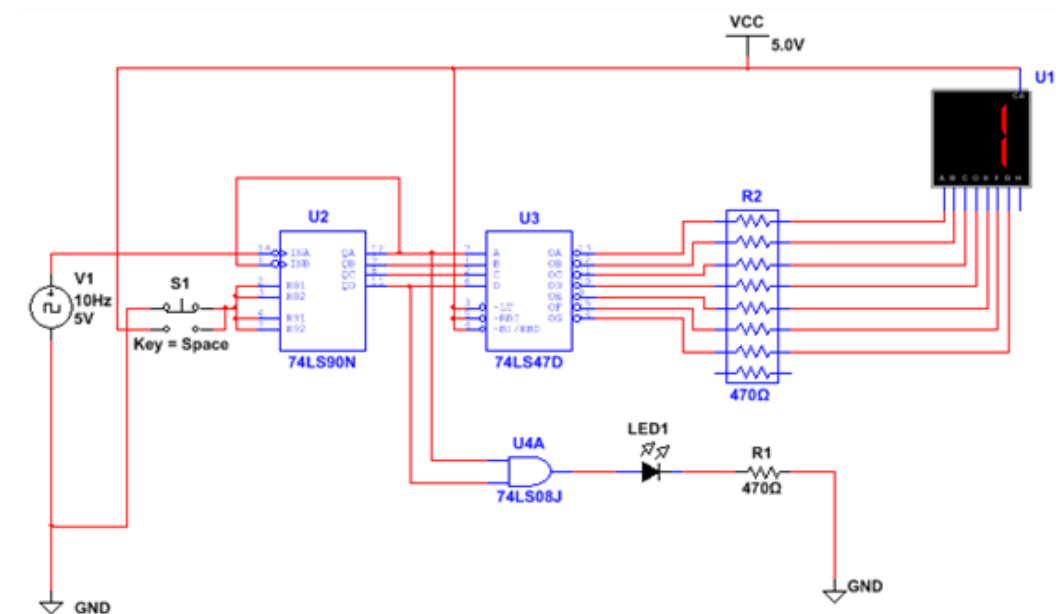
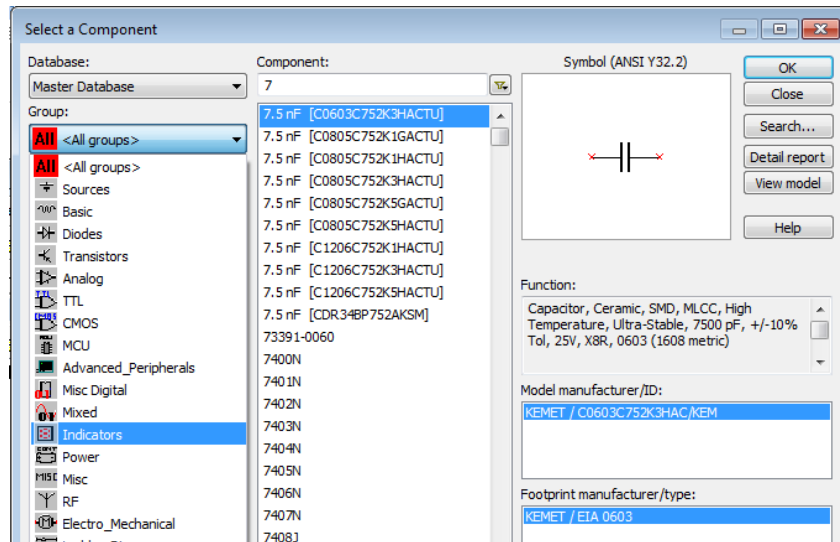


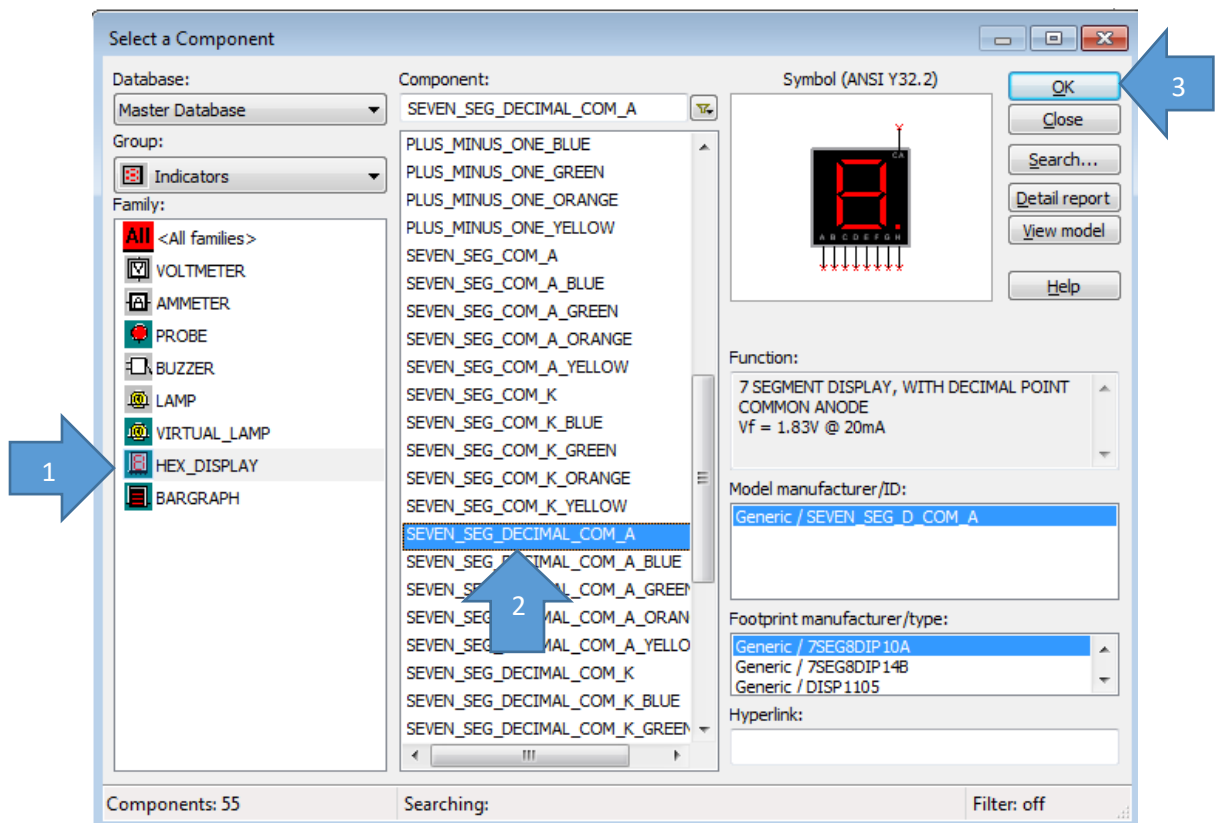
Figure 2.2 – Digital Counter

Steps:

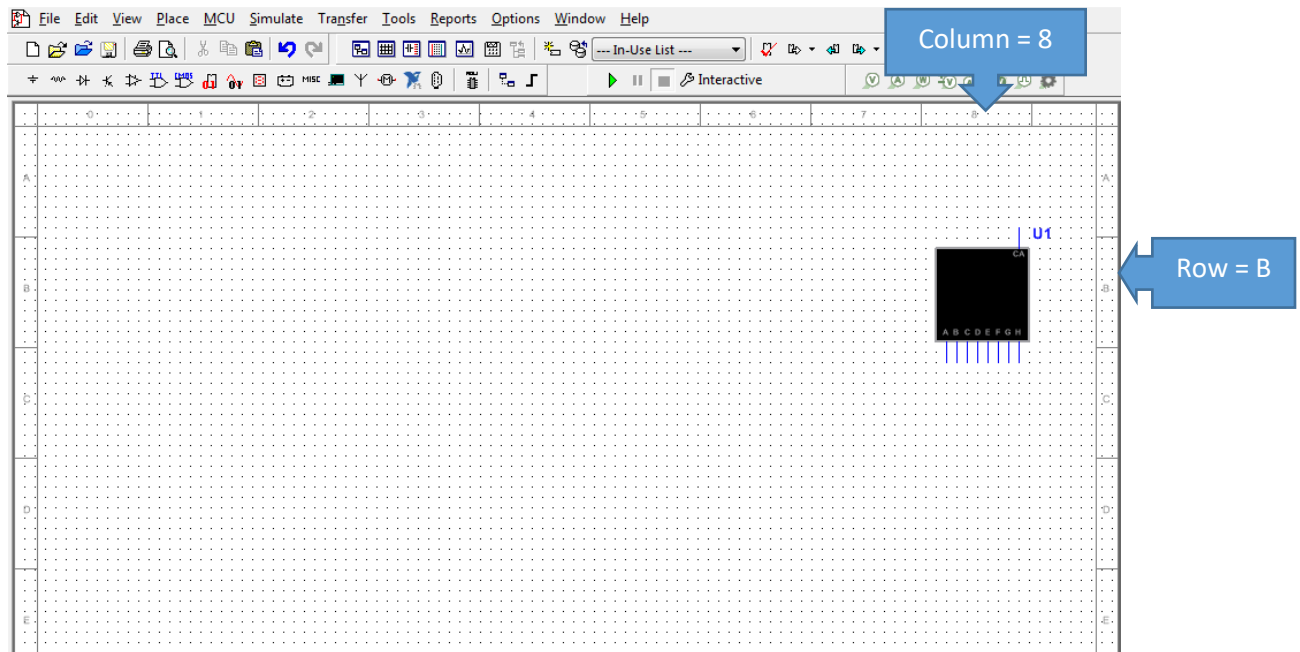
1. **Opening a circuit design workspace.** Go to Start → All Programs → NI Multisim 14.0
2. **Saving the workspace file.** Select File → Save As ... Save the file with the student's last name, for example, if your name is John Lee, save the multisim file as "Lee_Lab1". Make sure that the **Save as Type** 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



Place the remaining components in the workspace.

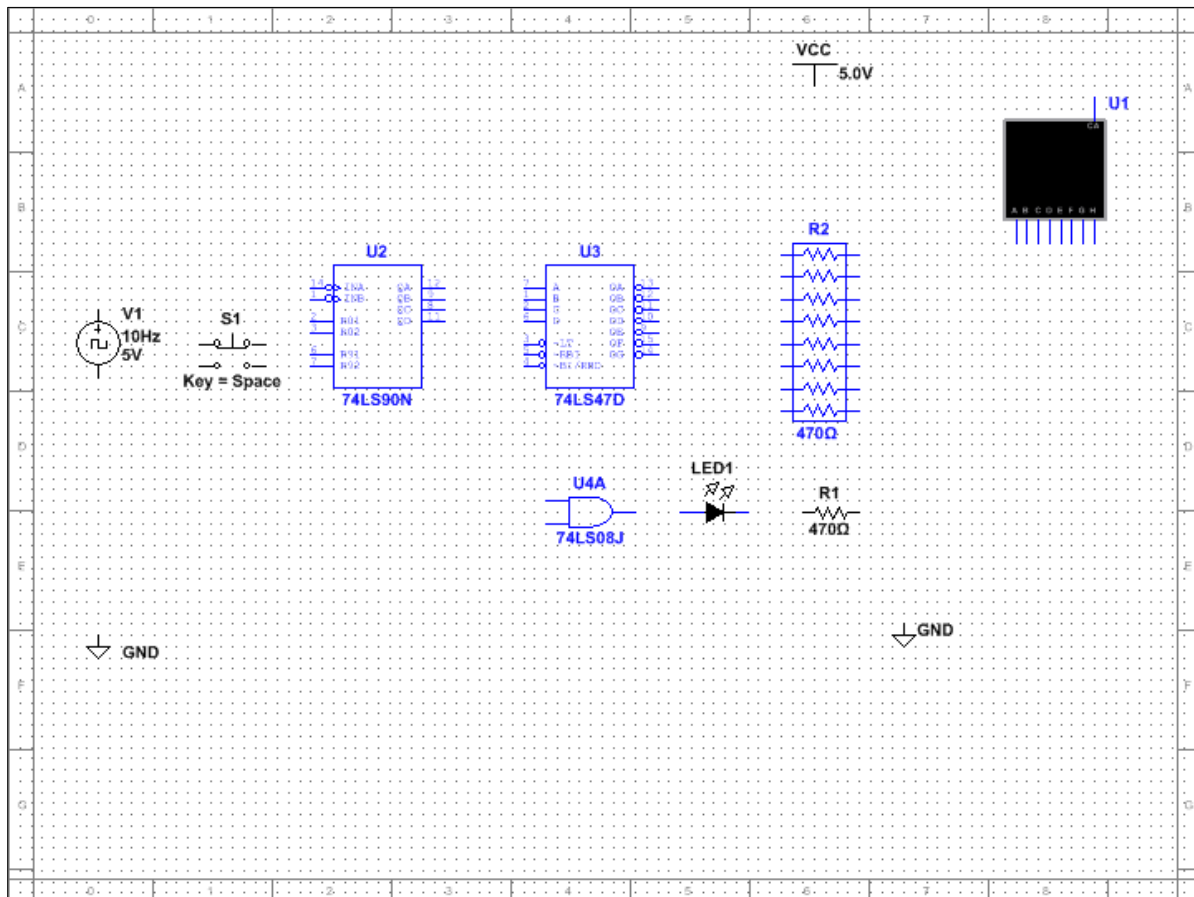
List	Group	Family	Component	Location
LED	Diodes	LED	LED_red Note: Rotate the LED using the key combination “Ctrl + r”	5E
Signal Voltage Source	Sources	Signal Voltage Source	Clock Voltage	0C
Transistor Voltage Sources	Sources	Power Sources	VCC	6A
Ground	Sources	Power Source	DGND Note: need 2 of them	0F, 7F
8 line pack of Resistors	Basic	RPACK	8Line_Isolated Note: double click the value “1k Ω ” and change it to 470 Ω	6C-6D
Resistor	Basic	Resistor	470 Ω	6E
Integrated Circuit	TTL	74LS	74LS90N ¹	2C-3C
Integrated Circuit	TTL	74LS	74LS47D ²	4C
Integrated Circuit	TTL	74LS	74LS08J ³	4E
Push-button	Electromechanical	Supplementary Switches	PB_DPST	1C

¹ 74LS90N – Decade and 4-bit binary counters. Appendix 2

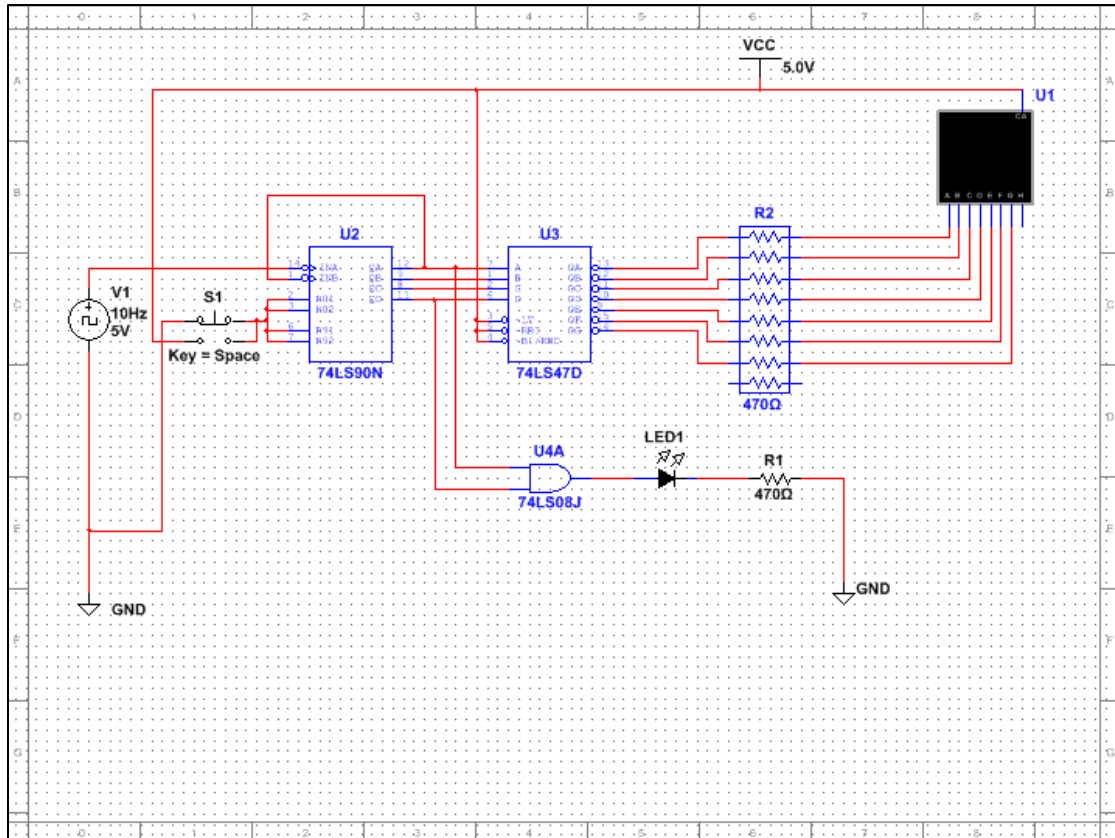
² 74LS47D - BCD to 7- Segment Decode. Appendix 2



³ 74LS08J – AND gate. Appendix 2

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 Circuit Counter
 Description: Lab 1: Introduction to Multisim: Digital Circuit Design

 Designed by: Student Name (type your name)
 Date: Insert the data
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 2.3:

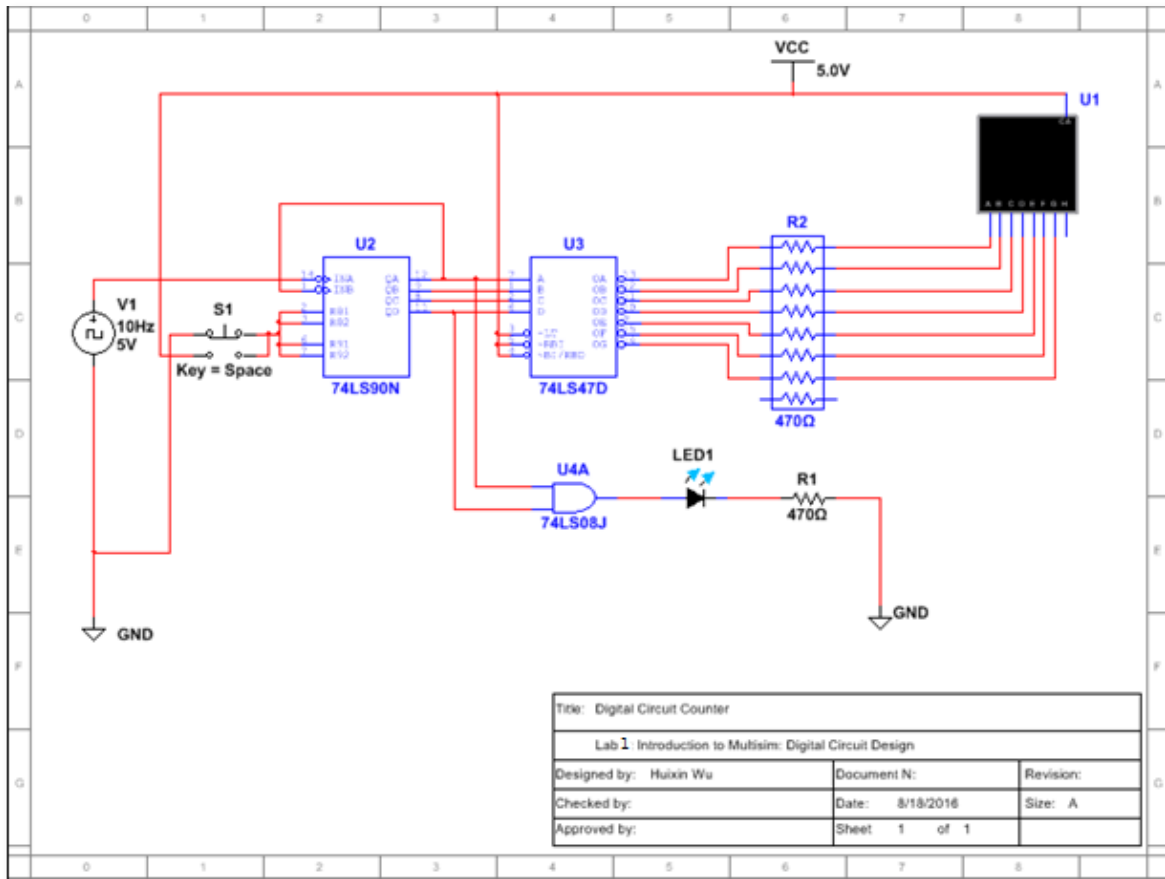


Figure 2.3 – Complete Digital Counter in Multisim

QUESTIONS

Multisim Envelopment

1. In which tab from the main menu the command **Junction** is located?
2. Mention three different ways to simulate a circuit in multisim
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 k Ω 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?
5. What is the function of the push button switch in the digital counter circuit?

Student's name: _____ Lab Instructor's signature: _____

----- **LAB EXPERIMENT 2 ENDS HERE** -----