
CISE 204 – DIGITAL SYSTEM DESIGN

LABORATORY MANUAL

This manual helps to understand basic elements of CISE 204 and its implementation.

Dr. Muhammad Faizan Mysorewala
Mr. Syed Ameenuddin Hussain

Revised in February 2011 by M. Mysorewala



TABLE OF CONTENTS

LAB 1: Basic operation of logic gates, instruments and circuit board.....	3
LAB 2: TTL Logic & Voltage switching characteristics of basic logic gates	11
LAB 3: Decimal to Binary Encoder.....	15
Lab 4: Introduction to Multisim + Example of Molasses Process	19
LAB 5: BCD to Seven-Segment Decoder	27
Lab 6: 4-bit Binary Adder/Subtractor and Magnitude Comparator	31
LAB 7: JK Flop-flop	33
LAB #8: One-shot and Astable Multivibrators for Traffic-signal control system	36
LAB 9: Truncated Sequence using Asynchronous Counter.....	40
LAB 10: Storing to, and Reading from Memory.....	43

LAB 1: INSTRUMENTS, CIRCUIT BOARD AND A SIMPLE CIRCUIT

Objectives:

The objective of this lab is to learn how to:

- Build a simple circuit on the circuit board
- Use the instruments such as digital multi-meter (DMM), oscilloscope, power supply, function generator, etc

Materials Needed:

Power Supply, Function Generator, Oscilloscope, Multimeter, Light-Emitting Diodes (LEDs), Resistors, Capacitors, and an IC.



(a) Power Supply



(b) Function Generator



(c) Oscilloscope



(d) Multimeter

Figure L1.1: Instruments

Power Supply (Figure L1.1a): All active electronic devices such as the integrated circuits used in digital electronics require a stable source of DC voltage to function properly. The power supply provides the proper level of dc voltage. It is very important that the correct voltage be set before connecting it to the ICs on your board or permanent damage can result. The power supply at your bench may have more than one output and normally will have a built in meter to help you set the voltage. Usually for all circuits power supply should be set to +5.0 V.

Digital Multimeter (Figure L1.1b): the DMM is a multipurpose measuring instrument that combines in one instrument the characteristics of a dc and ac voltmeter, a ac and dc ammeter and ohmmeter. The DMM indicates the measured quantity as a digital number, avoiding the necessity to interpret the scales as was necessary on older instruments. You need to select the desired function by the switch provided. For current measurements a separate set of lead connections are made to the meter. The ohmmeter function of a DMM is used only in circuits that are not powered.

Function Generator (Figure L1.1c): A function generator is used to produce signals required for testing various kinds of circuits. For digital circuits, a periodic rectangular pulse is the basic signal used for testing logic circuits. It is important that proper voltage be set up before connecting the function generator to the circuit or else damage may occur. Function generators normally have controls for adjusting the peak amplitude of a signal and may also have a means of adjusting the 0 volt level.

Oscilloscope (Figure L1.1d): The oscilloscope is the most important test instrument for testing circuits and you should become completely familiar with its operations. It's a versatile test instrument letting you see a graph of the voltage as a function of time in a circuit and compare waves. Because an oscilloscope allows you to measure various parameters, it is considered to be an instrument capable of parametric measurements important in both digital and analog work. Nearly all complex digital circuits have specific timing requirement that can be readily measured with a two channel oscilloscope. There are two basic types of oscilloscope; analog and digital. Both types of scopes have four main control groups: display controls, vertical and horizontal control and trigger controls.

Circuit board (protoboards): Circuit boards are convenient way to construct circuit for testing and experimenting. Most of them look like the one shown in the figure L1.2 . You will notice the top and bottom horizontal rows are connected as a continuous row. Vertical groups of five holes are connected together; the vertical group above the center strip is not connected to vertical group below the center strip. The holes are 0.1 inch apart, which is the same spacing as the pins on an integrated circuit DIP (Dual In-line Pins). Integrated circuits (ICs) are inserted to straddle the center; in this manner, wires can be connected to the pins of the IC by connecting them to the same vertical group as the desired pin.

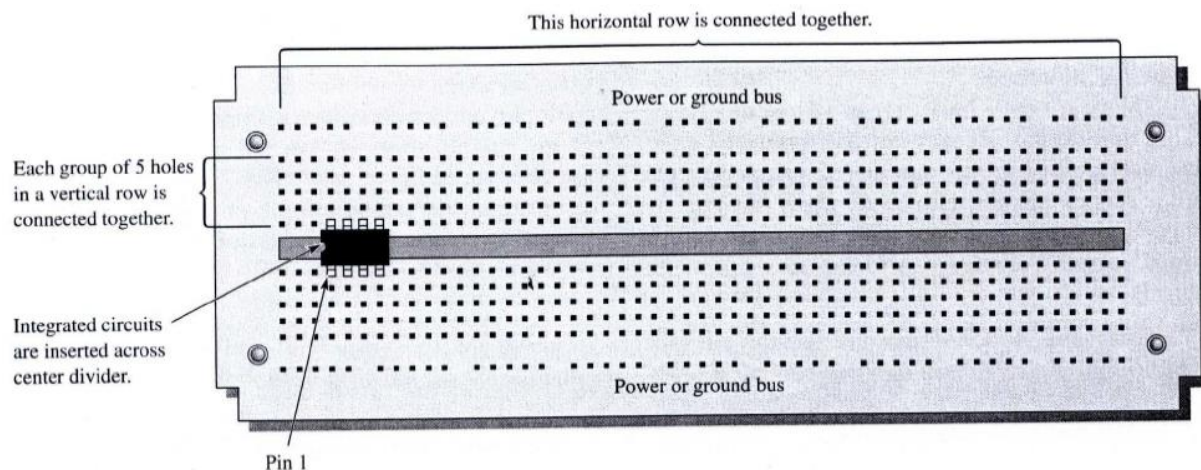


Figure L1.2: Circuit Board

Exercise 1: Measurement of DC Voltage with the DMM

1. Review the operator's manual or information supplied by your instructor for the power supply at your lab station. Generally, power supplies have a meter or meters that enable you to set the output voltage and monitor the current. Set the voltage based on the power supply meter to +5.0 V and record the reading in Table 1-1 (in the Data and Observation section).
2. The +5.0 V is the voltage you will use for nearly all of the experiments in this manual. For most TTL circuits, the power supply should be from 4.75 V to 5.25 V. To check that you have correctly set up the supply, measure the voltage with the DMM. Record the reading of the DMM in Table 1-1.

Exercise 2: Measurement of DC Voltage with the Oscilloscope

3. In this step, you will confirm the dc voltage from the power supply using the oscilloscope. Set the SEC/DIV control of your oscilloscope to a convenient value (a value near 0.2 ms/div is suggested to give a steady line on the display). Set the trigger controls to AUTO and INT (internal trigger) to assure a sweep is on the display. Select channel 1 as the input channel, and connect a scope probe to the vertical input. Put the input coupling control on GND to disconnect the input signal and find the ground position on the oscilloscope (digital scopes may have a marker for the GND level). Adjust the beam for a sharp, horizontal line across the scope face.
4. Since you will be measuring a positive voltage, position the ground on a convenient graticule line near the bottom of the display using the vertical POSITION control. If you are using an analog scope, check that the vertical VOLTS/DIV variable knobs are in their calibrated positions. A digital scope is always calibrated, and there is no vernier control.
5. Move the channel I input coupling control from the GND position to the dc position. For almost all digital work, the input coupling control should be in the DC position. Clip the ground lead of the scope probe to the ground of the power supply and touch the probe itself to the power supply output. The line

on the face of the oscilloscope should jump up 5 divisions. You can determine the dc voltage by multiplying the vertical sensitivity (1.0 V/div) by the number of divisions observed between ground and this line (5 divisions). Record the measured voltage (to the nearest 0.1 V) in Table 1-1.

Exercise 3: Measurement of Pulses with the Oscilloscope

6. Now you will set up the function generator or pulse generator for a logic pulse and measure some characteristics of the pulse using the oscilloscope. Review the operator's manual or information supplied by your instructor for the function generator at your lab station. Select the pulse function and set the frequency for 1.0 kHz. (If you do not have a pulse function, a square wave may be substituted.)
7. Set up and measure the pulse amplitude of the function generator. The vertical sensitivity (VOLTS/DIV) control of the oscilloscope should be set for 1.0 V/div and the SEC/DIV should be left at 0.2 ms/div. Check that both controls are in their calibrated positions. Check the ground level on the oscilloscope as you did in Step 3 and set it for a convenient graticule near the bottom of the scope face. Switch the scope back to dc coupling and clip the ground lead of the scope probe to a ground on the generator. Touch the probe to the function generator's pulse output. If the generator has a variable amplitude control, adjust it for a 4.0 V pulse (4 divisions of deflection). Some generators have a separate control to adjust the dc level of the pulse; others do not. If your generator has a dc offset control, adjust the ground level of the pulse for zero volts.
8. You should obtain a stable display that allows you to measure both the time information and the voltage parameters of the waveform. (If the waveform is not stable, check triggering controls.) In Plot I of your report, sketch the observed waveform on the scope display. It is a good idea, whenever you sketch a waveform from a scope, to record the VOLTS/DIV and SEC/DIV settings of controls next to the sketch and to show the ground level. Measure the pulse width (tti), period (T), and amplitude of the waveform and record these values in Table 1-2. The amplitude is measured in volts.
9. Connect the LED and series-limiting resistor, R1, to the pulse generator as shown in Figure L1.3. Note that the LED is a polarized component and must be connected in the correct direction to work. The schematic and an example of protoboard wiring are shown. Measure the signal across the LED with the oscilloscope and show it in Plot 2 of your report. Label the scope settings as in step 8 and show the ground level.

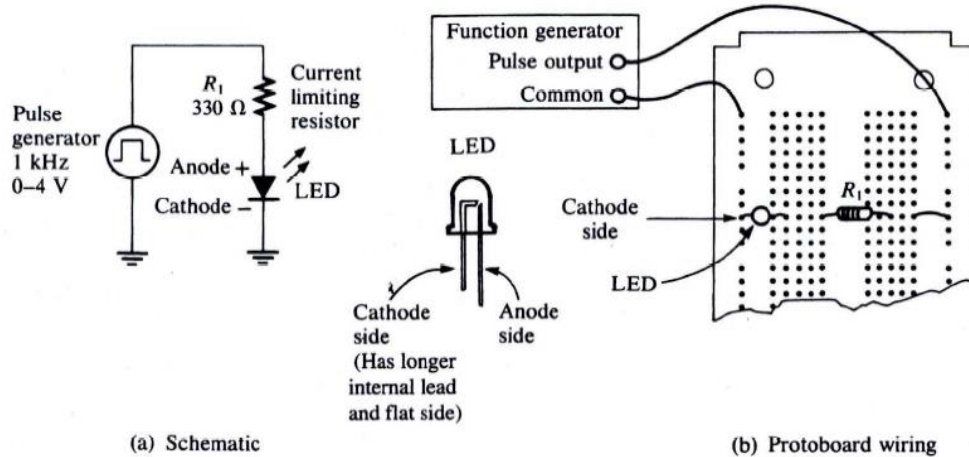


Figure L1.3: Example

10. Sometimes it is useful to use an oscilloscope to measure the voltage across an ungrounded component. The current-limiting resistor, R_1 , in Figure L1.3 is an ungrounded component. To measure the voltage across it, connect both channels of your oscilloscope as shown in Figure L1.4. Make sure that both channels are calibrated and that the vertical sensitivity (VOLTS/DIV) is 1 V/div for each channel. If you are using a newer scope, the difference operation (Channel 1-Channel 2) is likely to be shown as a menu item. On older scopes, the difference measurement is done by inverting channel 2 and selecting the ADD function. Consult the operator's manual if you are not sure. Measure the signal across R_1 and show the result on Plot 3. As a check, the sum of the voltages across the LED and resistor should be equal to the voltage of the generator.

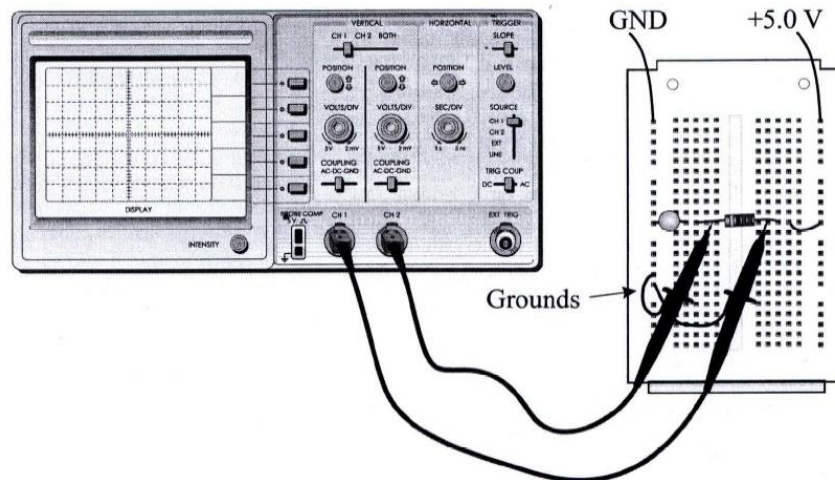


Figure L1.4: Oscilloscope for measuring voltage across the resistor

Exercise 4: Constructing and Measuring Parameters in a Digital Circuit

11. In this step, you will construct a small digital oscillator. This oscillator generates pulses that could be used to drive other digital circuits. The basic integrated circuit for the oscillator is the 555 timer, which will be covered in detail later. The schematic and sample protoboard wiring is shown in Figure L1.5. Construct the circuit as shown.

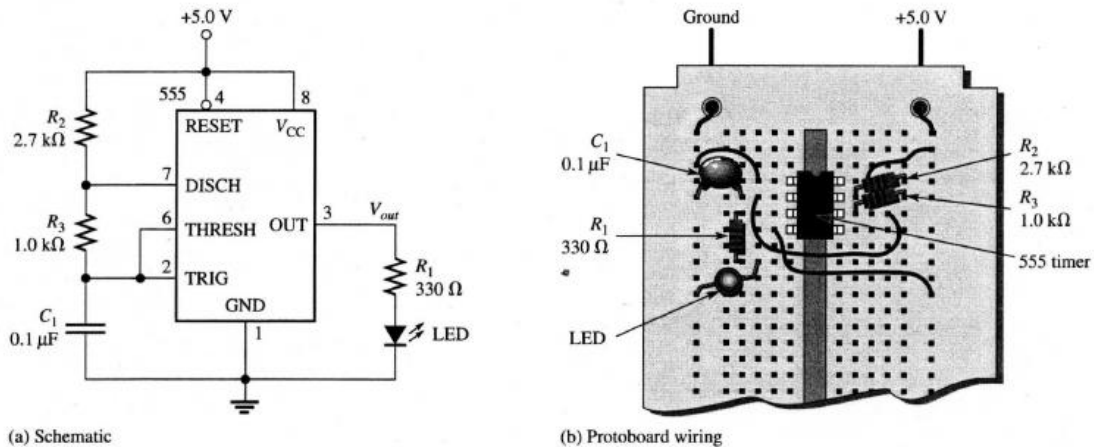


Figure L1.5: Example circuit for rectangular pulse generation

12. Using your oscilloscope, observe the signal on pin 3. Sketch the observed signal in Plot 4. Be sure to label the plot with the scope settings (VOLTS/DIV and SEC/DIV). Measure the parameters listed in the first four rows of Table 1-3. The frequency is computed from the period measurement ($f = 1/T$).
13. Replace C_1 with a 100 μF capacitor. The light should blink at a relatively slow rate. A slow frequency like this is useful for visual tests of a circuit or for simulating the opening and closing of a manual switch. Measure the period and frequency of the oscillator with the 100 μF capacitor. This signal, with a low frequency like this, may give you difficulty if you are using an analog scope. You will need to use NORMAL triggering instead of auto triggering and you may need to adjust the trigger LEVEL control to obtain a stable display. Record your measured values in Table 1-3.

Data and Observations:

Exercise 1 & 2: Measurement of DC Voltage with the DMM and Oscilloscope

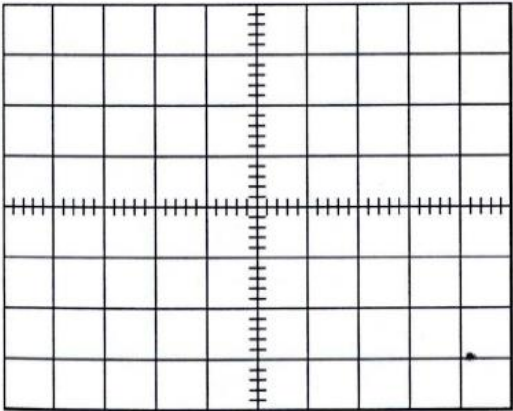
Voltage Setting = 5.0 V	Voltage Reading
Power supply Meter	
DMM (Ex. 1)	
Oscilloscope (Ex. 2)	

Table 1-1

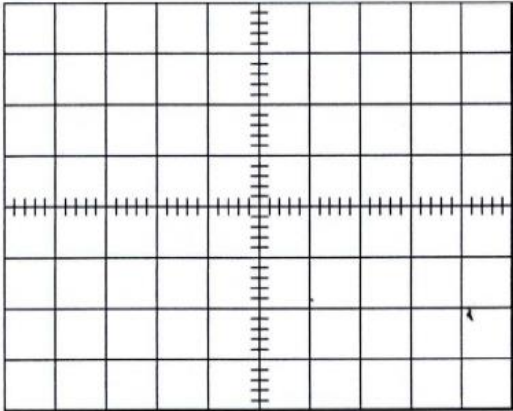
Exercise 3: Measurement of digital parameters

Function Generator Parameters (at 1 kHz)	Measured Values
Pulse width	
Period	
Amplitude	

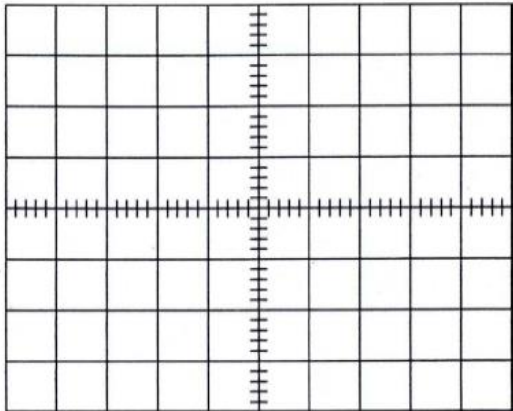
Table 1-2



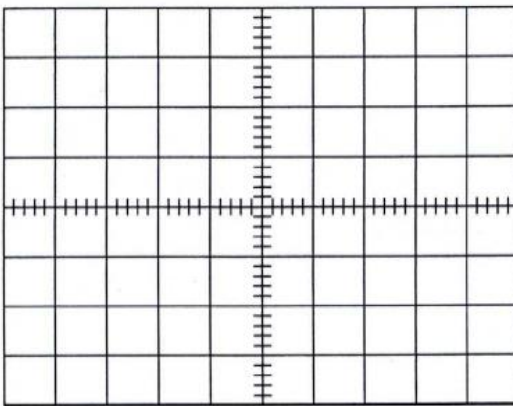
PLOT 1 Generator waveform



PLOT 2 Voltage across LED



PLOT 3 Voltage across R_1



PLOT 4 Digital oscillator output (pin 3)

Exercise 4: Measurement of digital parameters

Step	Digital Oscillator Parameters	Measured Values
12	Period	
	Duty cycle	
	Amplitude	
	Frequency	
13	Period	
	Frequency	

Table 1-3

Group Members:

- (1) _____
- (2) _____
- (3) _____
- (4) _____

LAB 2: TTL LOGIC & VOLTAGE SWITCHING CHARACTERISTICS OF BASIC LOGIC GATES

Objectives:

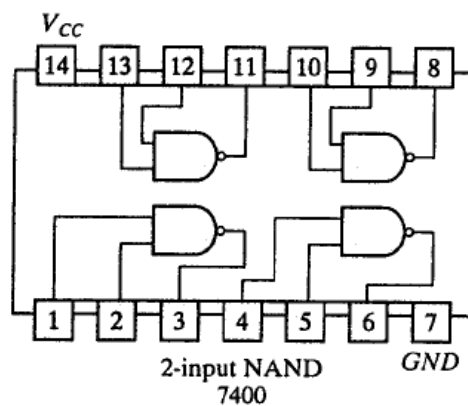
To learn and implement the TTL logic and switching characteristics of basic logic gates like NAND and Hex Inverter.

Materials needed:

Quad 2-input NAND gate IC (7400), Hex Inverter IC (7404)

Exercise 1: Testing truth table for NAND

In this experiment, you will test the truth tables for NAND gate. Keep in mind that if any two truth tables are identical, then the logic circuits that they represent are equivalent.



Procedure:

1. Find the connection diagram for the 7400 quad 2-input NAND gate in the manufacturer's specification sheet. Note that there are four gates on each of these ICs.
2. Apply VCC and ground to the appropriate pins.
3. Then test one of the NAND gates by connecting all possible combinations of inputs, as listed in Table 2-1.
4. Apply logic 1 through a series 1.0 W resistor and a logic 0 by connecting directly to ground.
5. Show the logic output (1 or 0) as well as the measured output voltage in Table 2-1.
6. Use the DMM to measure the output voltage.

NAND gate.

Inputs		Output	Measured Output Voltage
A	B	X	
0	0		
0	1		
1	0		
1	1		

Table 2-1

Exercise 2:

In this experiment, you will test the switching characteristics of the Hex Inverter.

Procedure:

1. Find the connection diagram for the 7404 Hex Inverter in the manufacturer's specification sheet. Note that there are five inverters on each of these ICs.
2. Apply V_{CC} and ground to the appropriate pins. Then test Hex Inverter gates by applying the different voltage inputs V_{in} as highlighted in the Table 2-2.
3. Measure and note the V_{out} for different values of V_{in} in Table 2-2. Use the DMM to measure the output voltage.
4. Start with small step and increase the step size if output voltage is not changing significantly.
5. Repeat the experiment with different values of $V_{CC} = 4.8\text{ V}$ and $V_{CC} = 5.2\text{ V}$.

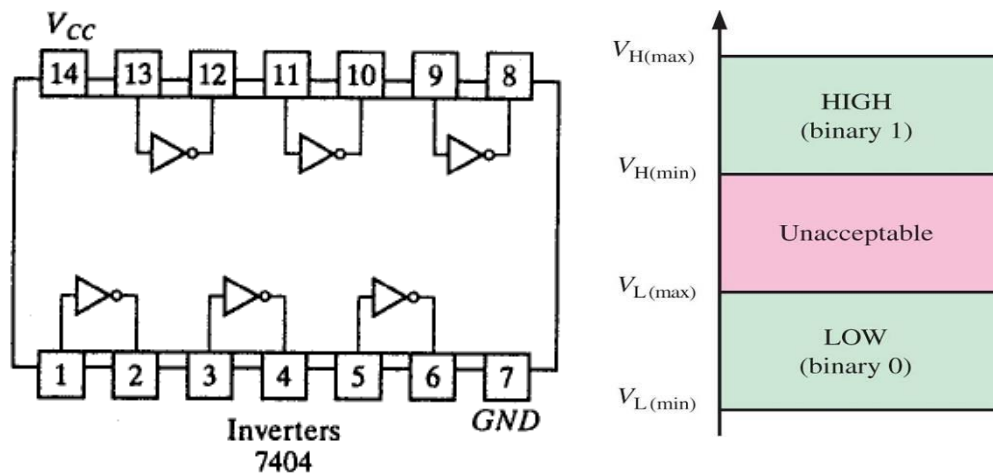


Figure L2.1: Hex Inverter 7404 and TTL Levels

Complete the following table for inverter IC 7404.

[illegible]

Table 2-1

Find from the data collected the following for the TTL hex Inverter:

1. Minimum input High voltage (V_{IH} min) : _____
2. Maximum input Low voltage (V_{IL} max) : _____
3. Minimum output High voltage (V_{OH} min) : _____
4. Maximum output Low voltage (V_{OL} max) : _____

Plot V_{in} versus V_{out} for different V_{cc} values

Group Members:

(1) _____

(2) _____

(3) _____

(4) _____

LAB 3: DECIMAL TO BINARY ENCODER

Objective:

Design, build and evaluate a decimal to binary encoder

Materials Needed:

7432 quadruple 2-input positive-OR gate (Qty=3)

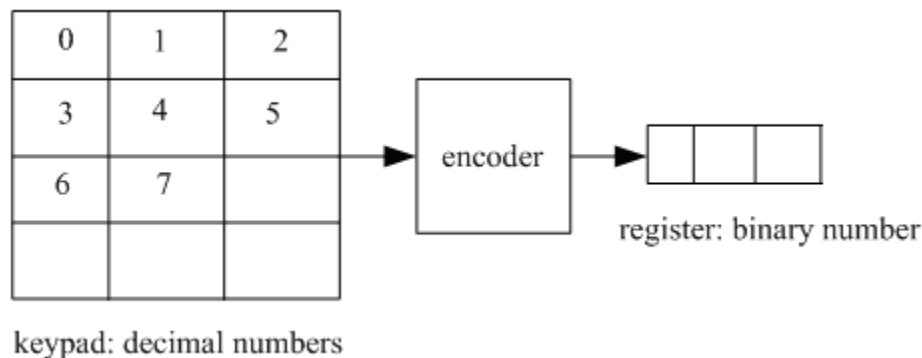
Dipswitch (Qty=1)

330Ω resistors (Qty=3)

LEDs (Qty: 3)

Design of decimal to binary converter

Our objective is to obtain a Boolean function relating the decimal inputs to binary outputs. We will restrict for simplicity to 7 decimal numbers and 3-bit binary number.



The decimal numbers range from 0,1,2,3,4,5,6 and 7. Let the decimal number inputs be denoted as follows

$$A = [A_0 \ A_1 \ A_2 \ A_3 \ A_4 \ A_5 \ A_6 \ A_7]$$

where $A_0 = 0$, $A_1 = 1$, $A_2 = 2$, $A_3 = 3$, $A_4 = 4$, $A_5 = 5$, $A_6 = 6$ and $A_7 = 7$

The 3-bit binary outputs be denoted by $Y = [a_1 \ a_2 \ a_3]$,

Boolean expression relating the output, Y, and the input, A is obtained from the truth table. There are 8 inputs and 3 outputs. At a given instant, only one input, A_i , may be activated: two or more inputs may not be activated at the same time. To appreciate this restriction, consider a computer keyboard. One can press only one number key at any time instant and pressing simultaneously two or more number keys are not allowed. Truth table for decimal to binary encoder is given in the Table below. For values of decimal numbers from 0 to 7, corresponding binary numbers are listed in Table 3-1.

Table 3-1: The truth table for decimal to binary

Input								Output		
A ₀	A ₁	A ₂	A ₃	A ₄	A ₅	A ₆	A ₇	a ₂	a ₁	a ₀
1	0	0	0	0	0	0	0	0	0	0
0	1	0	0	0	0	0	0	0	0	1
0	0	1	0	0	0	0	0	0	1	0
0	0	0	1	0	0	0	0	0	1	1
0	0	0	0	1	0	0	0	1	0	0
0	0	0	0	0	1	0	0	1	0	1
0	0	0	0	0	0	1	0	1	1	0
0	0	0	0	0	0	0	1	1	1	1

Boolean expression for each of the outputs; a_0 , a_1 and a_2 is obtained by analyzing the row corresponding to $a_0 = 1$, $a_1 = 1$, and $a_2 = 1$ respectively. From the truth table, it can be deduced that each binary output is a sum of 4 of the 7 decimal inputs as given below:

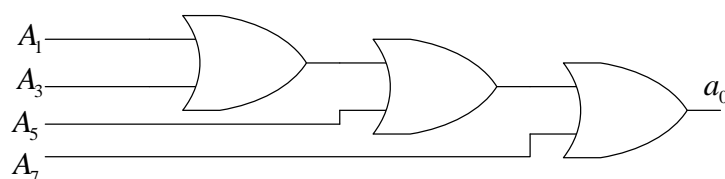
$$a_0 = A_1 + A_3 + A_5 + A_7$$

$$a_1 = A_2 + A_3 + A_6 + A_7$$

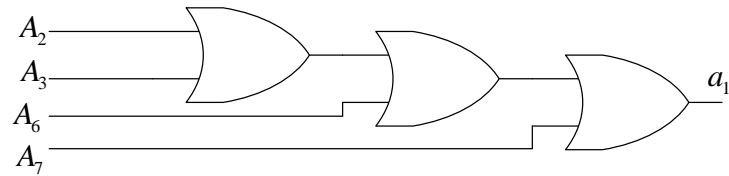
$$a_2 = A_4 + A_5 + A_6 + A_7$$

Implementation of the decimal-binary encoder

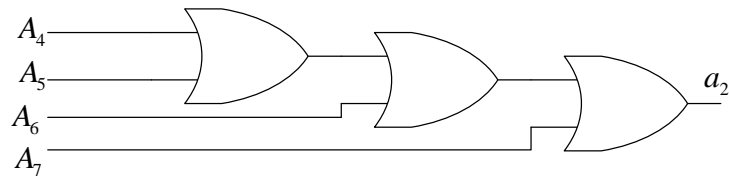
Each binary output is obtained by using 4 input and one output OR gate. Since OR gates are available as only two-input and one output gate, three OR gates are cascade connected.



$$a_0 = A_1 + A_3 + A_5 + A_7$$



$$a_1 = A_2 + A_3 + A_6 + A_7$$



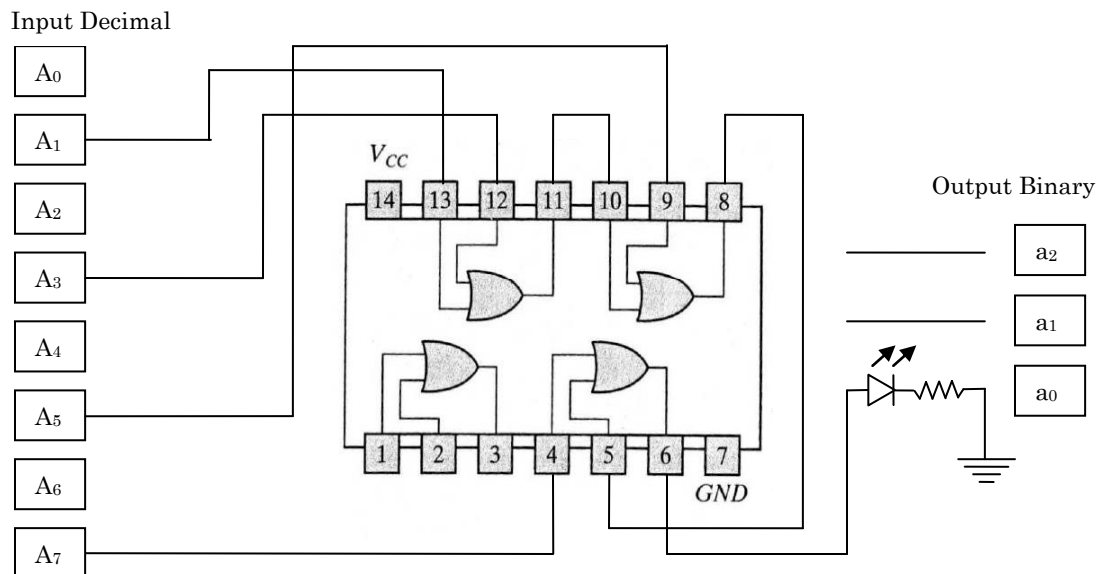
$$a_2 = A_4 + A_5 + A_6 + A_7$$

Evaluation and testing of the encoder:

An LED may be connected to each of the outputs. Each of the decimal inputs, A_i , where $i = 1, 2, \dots, 7$ representing decimal numbers $0, 1, \dots, 7$ is applied by setting the input to $A_i = 5V$.

For example when $A_0 = 5V$ none of LEDs connected to the binary outputs will light, when $A_1 = 5V$ the LED connected to the binary output a_0 will light, when $A_2 = 5V$ the LED connected to the binary output a_1 will light, when $A_3 = 5V$ the LEDs connected to the binary outputs a_0 and a_1 will light, and so on.

Circuit for output a_0 :



Input Output Connections for $a_0 = A_1 + A_3 + A_5 + A_7$

Group Members:

(1) _____

(2) _____

(3) _____

(4) _____

LAB 4: INTRODUCTION TO MULTISIM + EXAMPLE OF MOLASSES PROCESS

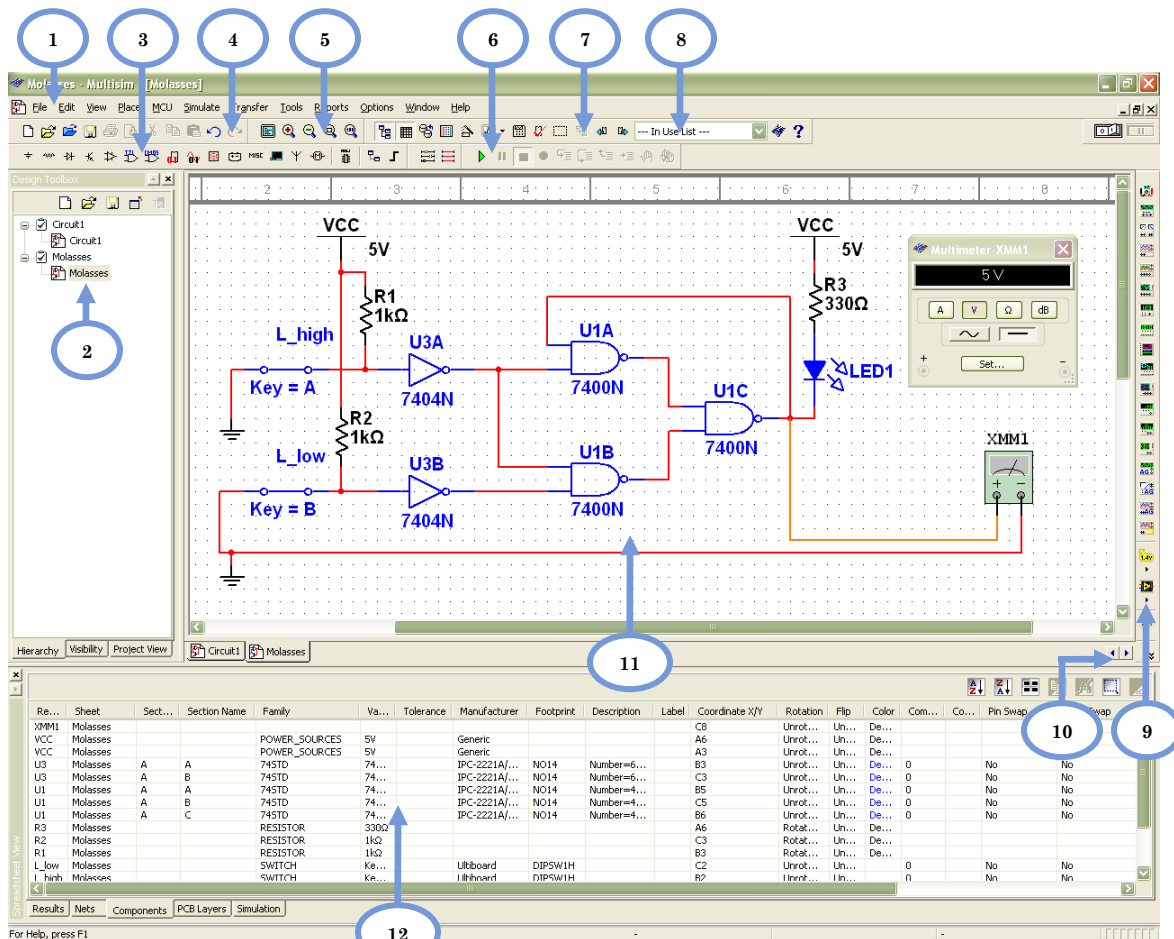
Objective:

The objective of this experiment is introducing you to Multisim programming software with the implementation of simple Molasses Process.

Introduction:

Multisim is the schematic capture and simulation application of National Instruments Circuit Design Suite, a suite of EDA (Electronic Design Automation) tools that assists you in carrying out the major steps in the circuit design flow. Multisim is designed for schematic entry, simulation, and feeding to downstage steps, such as PCB layout.

Multisim's user interface consists of the following basic elements:



- | | | |
|----------------------|-----------------------|------------------------|
| 1. Menu Bar | 5. View Toolbar | 9. Instruments Toolbar |
| 2. Design Toolbox | 6. Simulation Toolbar | 10. Scroll Left/Right |
| 3. Component Toolbar | 7. Main Toolbar | 11. Circuit Window |
| 4. Standard Toolbar | 8. In Use List | 12. Spreadsheet View |

- The **Menu Bar** is where you find commands for all functions.
- The **Design Toolbox** lets you navigate through the different types of files in a project (schematics, PCBs, reports), view a schematic's hierarchy and show or hide different layers.
- The **Component toolbar** contains buttons that let you select components from the Multisim databases for placement in your schematic.
- The **Standard toolbar** contains buttons for commonly-performed functions such as Save, Print, Cut, and Paste.
- The **View toolbar** contains buttons for modifying the way the screen is displayed.
- The **Simulation toolbar** contains buttons for starting, stopping, and other simulation functions.
- The **Main toolbar** contains buttons for common Multisim functions.
- The **In Use List** contains a list of all components used in the design.
- The **Instruments toolbar** contains buttons for each instrument.
- The **Circuit Window (or workspace)** is where you build your circuit designs.
- The **Spreadsheet View** allows fast advanced viewing and editing of parameters including component details such as footprints, RefDes, attributes and design constraints. Users can change parameters for some or all components in one step and perform a number of other functions.

Problem statement for Molasses Cookie Storage Tank Process Setup:

Control systems typically use sophisticated programmable controllers to provide flexibility in implementing logic and allowing for changes in the design as requirements change. For simple systems, the control logic can be designed from fixed function logic, as will be the case in this experiment. The problem posed in this experiment is to design the control logic for two of the four outputs in a tank control system. The specific requirements are described in the Problem Statement. The problem stated in the experiment is similar, but not identical, to the system application for the pancake syrup tank explained in Chapter 5. The key difference in the experiment is that there is feedback present. Strictly speaking this means it is sequential logic, but the methods employed in the design use combinational logic design methods (Karnaugh maps) like those discussed in Chapter 5. The feedback prevents the tank from refilling until it has emptied below a certain level as you will see. A flow sensor is not needed in this design. The Crumbly Cookie Company has a problem with the storage tank for its new line of molasses cookies.

The problem is that the molasses in the winter months runs too slow for the batch process. As a new employee, you have been assigned to design logic for the model-2

tank controller that will assure that the molasses is warm enough before the outlet valve, V_{OUT} , is opened. After it is opened, it must remain open until the lower sensor is uncovered, indicating the tank is empty. The best way to understand the problem is to review the model - 1 tank controller design. In the model- 1 design, the molasses tank had two level sensors, one high, L_H , and one low, L_L . The tank was emptied (outlet valve opened) only when molasses reached the upper sensor. After opening, the valve was closed only when the lower sensor was uncovered.

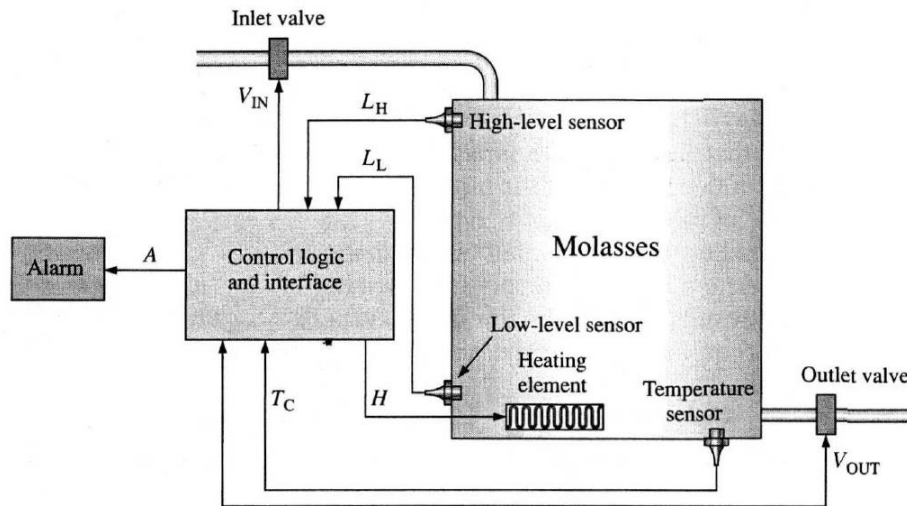


Figure 1: Molasses Process Setup

As mentioned, the model-1 system opens the outlet valve only when both sensors are covered but, once opened; it remains open until both sensors are uncovered. This concept requires knowing the current state of the output valve; hence in the design, it is considered as both *an output* and an *input* to the logic. This idea is summarized with the truth table shown as Table 1. Because the system is designed for TTL logic, the outlet valve is *opened* with a LOW signal.

Inputs			Output	Action
L_H	L_L	V_{OUT}	V_{OUT}	
0	0	0	1	Close valve.
0	0	1	1	Leave valve closed.
0	1	0	0	Valve is open; leave open.
0	1	1	1	Valve is closed; leave closed.
1	0	0	0	Sensor error; open valve.
1	0	1	0	Sensor error; open valve.

1	1	0	0	Sensors covered; leave valve open.
1	1	1	0	Sensors covered; open valve.

Table 1 – Truth table Summarized

V_{OUT}	0	1
$L_H L_L$		
00	1	1
01	0	1
10	0	0
11	0	0

$$V_{OUT} = \bar{L}_H \bar{L}_L + \bar{L}_H V_{OUT}$$

Figure 2: K-Map

A Karnaugh map that represents the truth table is shown in Figure 2. From the map, the minimum logic is determined. The circuit for the outlet valve of the model-1 tank controller is shown in Figure 3.

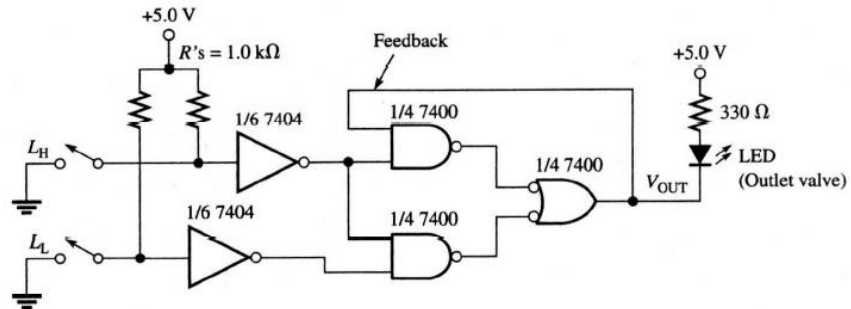


Figure 3: Circuit for output valve of the model-1 tank

Notice that V_{OUT} is returned to one of the inputs, which is the feedback referred to previously. After constructing the circuit for the outlet valve of the model-1 tank controller, it will be tested using switches for the inputs and an LED to represent the outlet valve.

Multisim Tutorial:

This tutorial leads you through the circuit design flow, from schematic capture to simulation. After following the steps outlined on the following pages, you will have designed a circuit that you can use to implement the testing procedure without building a circuit on Multisim.

Schematic Capture:

Opening and saving files

Complete the following step to launch Multisim:

1. Select **Start»All Programs»National Instruments»Circuit Design Suite 10.1»Multisim 10.1**. A blank file opens on the workspace called Circuit1.

Complete the following steps to save the file with a new name:

1. Select **File»Save As** to display a standard Windows Save dialog.
2. Navigate to the location where you wish the file to reside, enter MyFirstCircuit as the filename, and click the Save button.

Tip: To guard against accidental loss of data, set up a timed auto-backup of the file in the Save tab of the Preferences dialog box.

Complete the following step to open an existing file:

1. Select **File»Open**, navigate to the location where the file resides, highlight the file, and click on the Open button.

Tip: To view files from earlier versions of Multisim, select the desired version in the Files of Type drop-down in the Open dialog.

Components Needed:

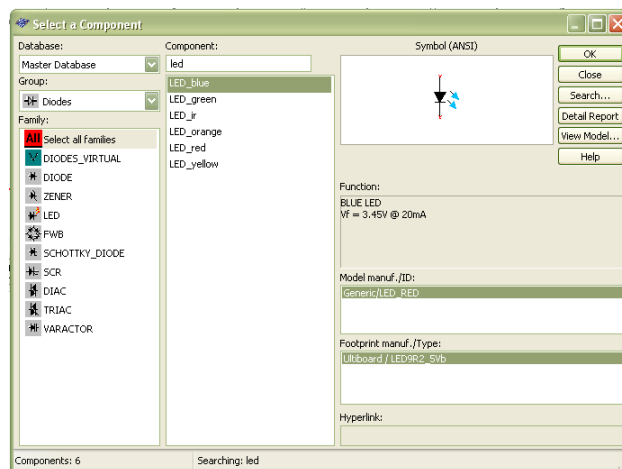
- One LED
- Two single-position DIP switches
- Resistors: two 1.0 k4, one 330 S2
- IC – 7400,7404 as determined by the student

Placing components:

Complete the following steps to start placing components:

1. Open MyFirstCircuit.ms10 as described above.
2. Select **Place»Component** to display the **Select a Component browser**, search for LED as shown below and click OK. The component appears as a “ghost” on the cursor.

Tip: Once you have selected the desired Group and Family, start typing the component’s name in the browser’s Component field. As you type, the string appears in the Searching field at the bottom of the browser. In the example below, type led. Matches are displayed as you type.



3. Move the cursor to the bottom-right of the workspace and left-click to place the component. Note that the Reference Designator for this component is “LED1.”
4. Place the remaining components in the Digital Counter area as shown below.

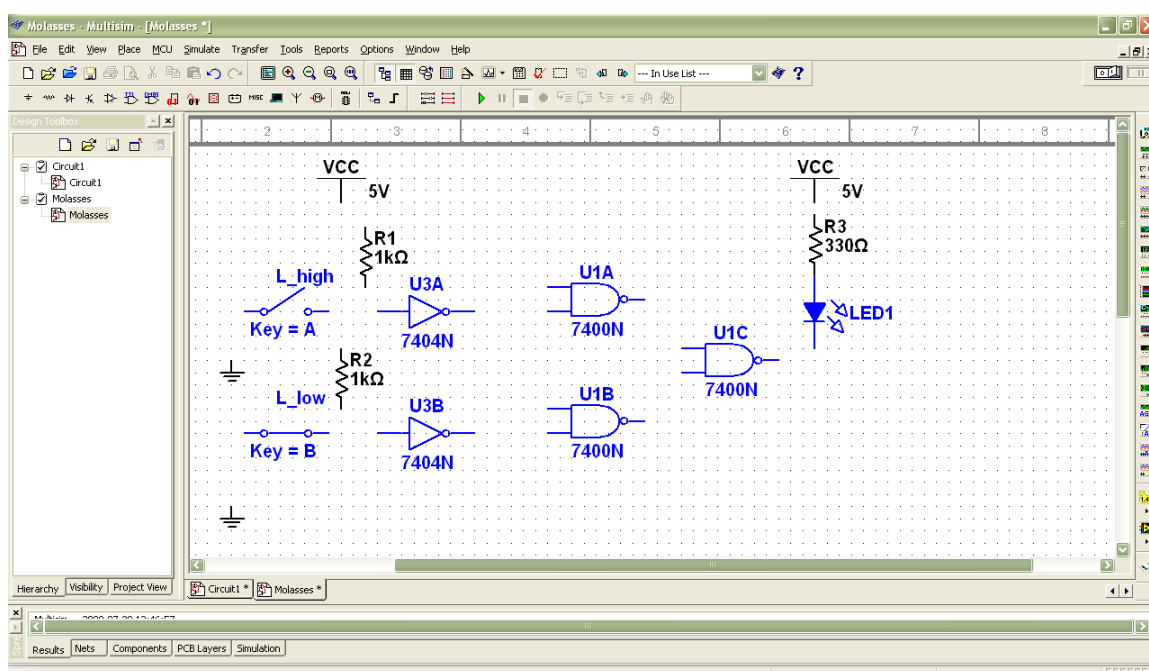
Note: When placing resistors, inductors, or capacitors (RLC components), the Select a Component browser has slightly different fields than for other components. When placing any of these components, you can choose any combination of: the component’s value (for example, the resistance value); type (for example, carbon film); tolerance; footprint and manufacturer. If you are placing a component that will be ultimately exported to PCB layout, and become part of a Bill of Materials,

you must be careful that the combination of values that you select in the Select a Component dialog box are available in a real-world, purchasable component.

Tip: When placing RLC components, type the value of the device that you want to place in the field at the top of the Component list. The value does not need to appear in the list to be placed on the schematic.

Tip: While placing the $200\ \Omega$ resistor, rotate it to a vertical orientation by pressing Ctrl-R on your keyboard.

Tip: Reference Designators (for example, U1, U2) are assigned in the order the components are placed. If you place components in a different order than in the original circuit, the numbering will differ. This will not affect the operation of the circuit in any way.



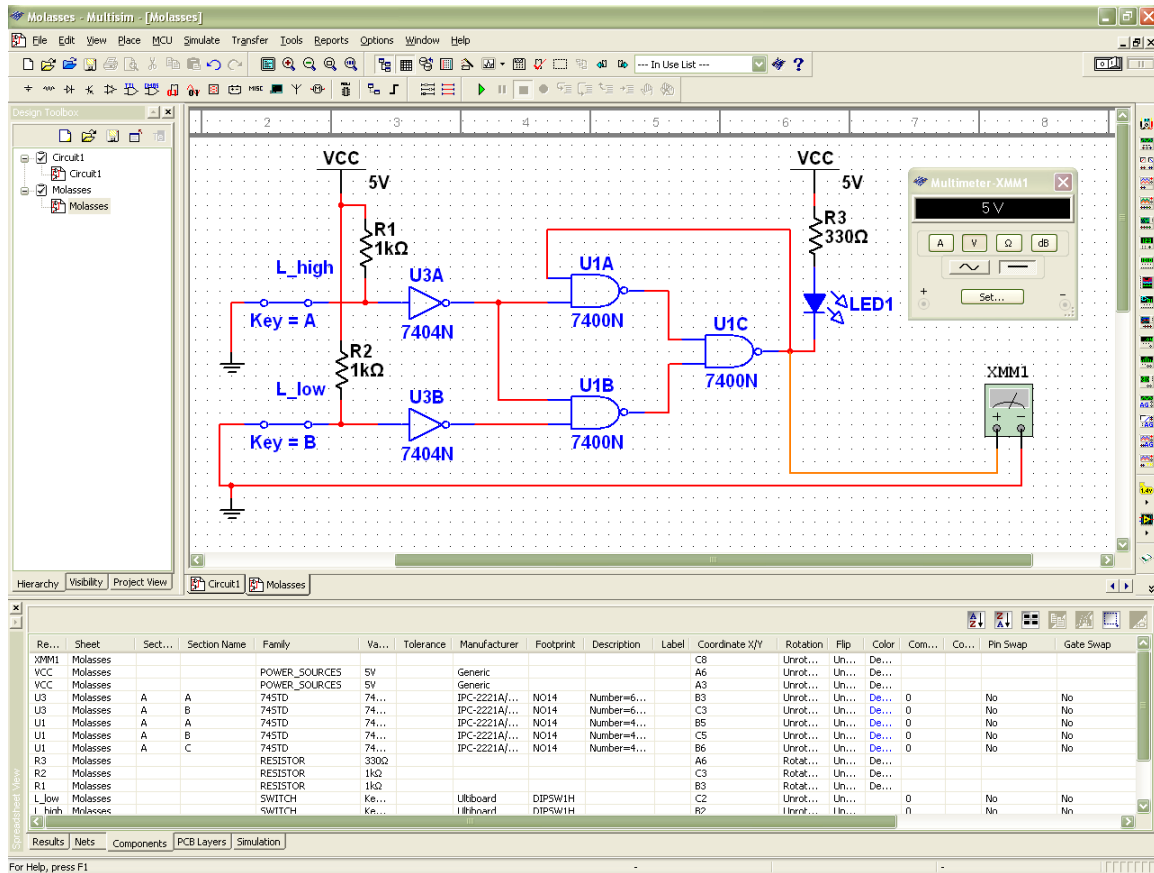
Wiring the Circuit:

All components have pins that you use to wire them to other components or instruments. As soon as your cursor is over a pin, Multisim knows you want to wire and the pointer changes to a crosshair.

Complete the following steps to wire the circuit:

1. Click on a pin on a component to start the connection (your pointer turns into a crosshair) and move the mouse. A wire appears, attached to your cursor.
2. Click on a pin on the second component to finish the connection. Multisim automatically places the wire, which conveniently snaps to an appropriate configuration, as shown below. This feature saves a great deal of time when wiring large circuits.

- 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. Finish wiring the circuit as shown below.



Simulation:

Simulating your circuits with Multisim catches errors early in the design flow, saving time and money. You can simulate the circuit by pressing either “F5” function button or by navigating to file menu >>**Simulate**>>**Run** or by clicking the green play or run button in the simulation toolbar.

Testing Procedure:

The test begins with both switches closed, meaning the level inputs are both LOW. As the tank fills, L_L is covered, so it will open and changes to a HIGH level. The LED, representing the valve, remains OFF. Later, L_H is covered, so it is opened and changes to a HIGH level. This causes the LED to turn ON. At this point, closing the HIGH level switch will have no effect on the LED as it remains ON until L_L is again covered, represented by a LOW.

Group Members:

(1) _____

(2) _____

(3) _____

(4) _____

LAB 5: BCD TO SEVEN-SEGMENT DECODER

Objective:

Simulate, build and test a simple decoder circuit using a BCD to seven-segment decoder and a seven-segment display.

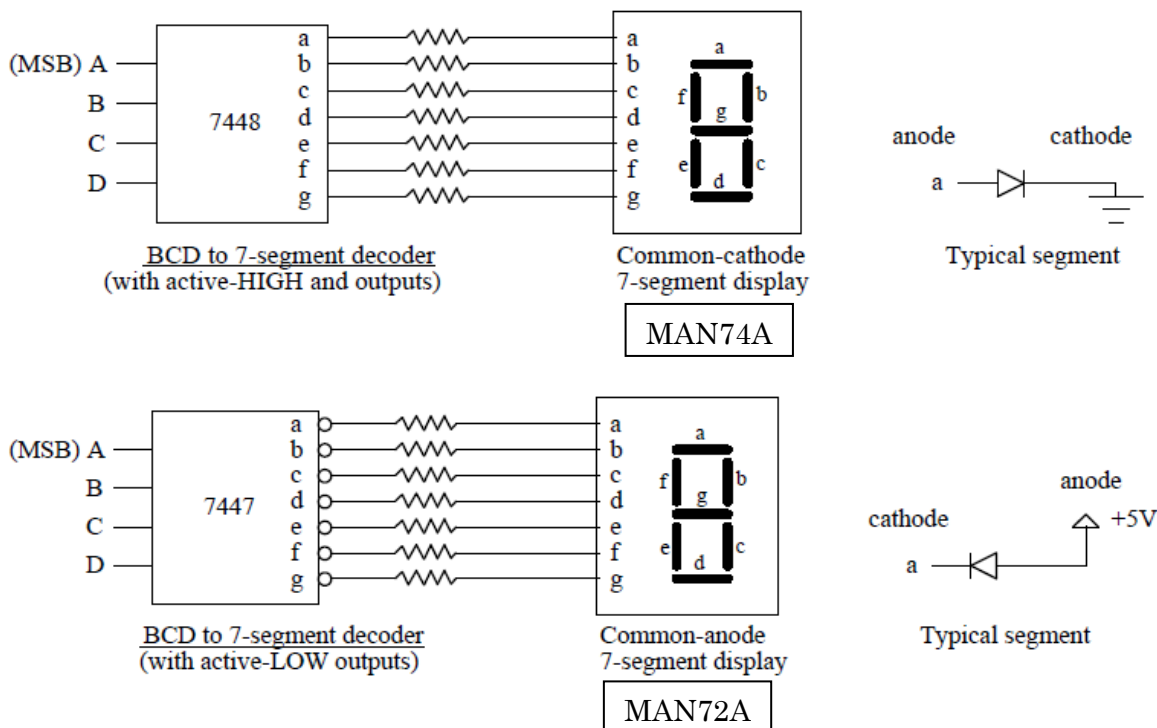
Material Needed:

BCD to seven-segment decoder (7447) (Qty=1)
 Seven Segment display MAN72A (Qty=1), LEDs (Qty=4)
 1k Ω resistor (Qty=1), 330 Ω resistors (Qty=11)
 Dipswitch (Qty=1)

Theory:

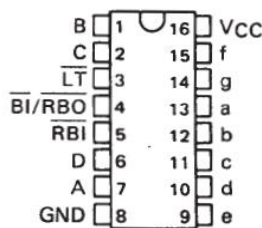
The purpose of BCD-to-seven segment decoder is to decode the input in BCD into 7 output segments for displaying. These decoders are used in conjunction with the segment displays. There are two main types of decoders as shown in figure below:

- IC 7448, whose inputs and outputs are active HIGH. It is used with MAN74A (common-cathode type) seven-segment display.
- IC 7447, whose inputs and outputs are active LOW. It is used with MAN72A (common-anode type) seven-segment display.



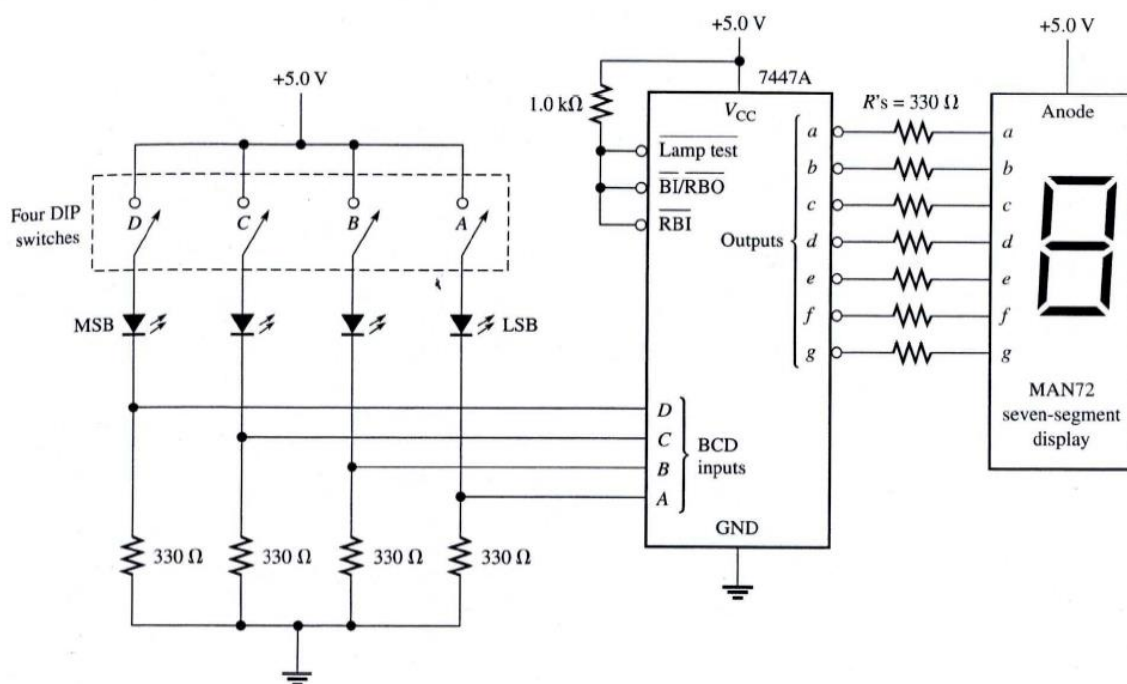
We are using 7447, MAN72A & MAN74. The pin-connections for both are given below:

Pin No.	MAN72	MAN74
1	Cathode A	Anode F
2	Cathode F	Anode G
3	Common Anode	No Pin
4	No Pin	Common Cathode
5	No Pin	No Pin
6	Cathode D.P.	Anode E
7	Cathode E	Anode D
8	Cathode D	Anode C
9	No Connection	Anode D.P.
10	Cathode C	No Pin
11	Cathode G	No Pin
12	No Pin	Common Cathode
13	Cathode B	Anode B
14	Common Anode	Anode A



Procedure:

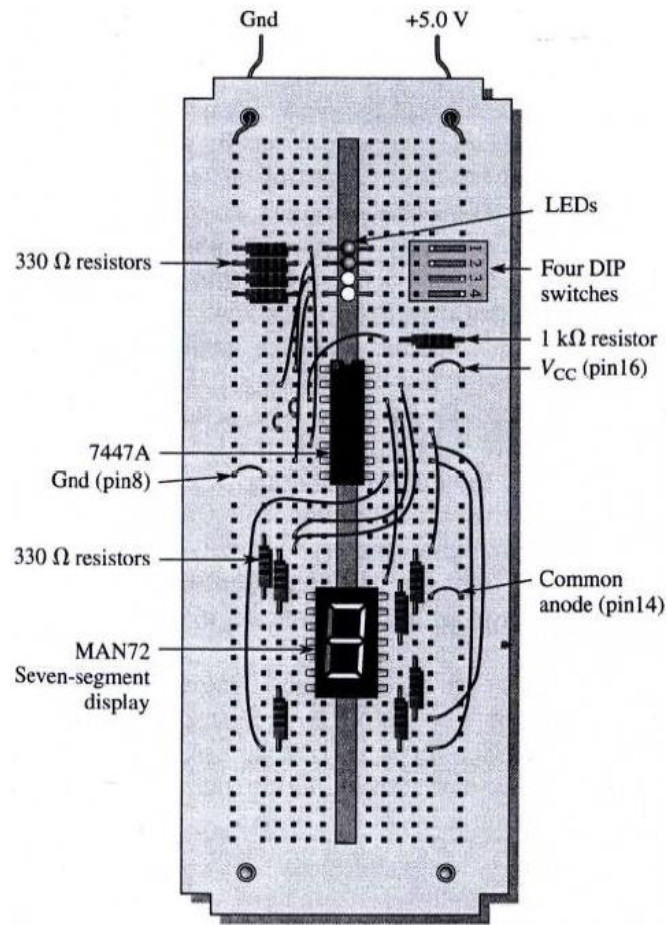
- First simulate the circuit in Multisim software.



Configure the BCD-to-seven-segment decoder (7447) such that it takes inputs from four switches and drives the seven-segment display. The 7447 is designed to properly control a common-anode type of seven-segment display (MAN72).

- Provide +5 volt on pin 16 and ground on pin 8 to the 7447 chip.
- Apply +5 volt on pins 3, 4, and 5 of the 7447 chip.
- Connect 330-ohm resistors to pins 13, 12, 11, 10, 9, 15 and 14 of the 7447, and connect those resistors to the a, b, c, d, e, f, and g segments of the 7-segment display.
- Connect the common-anode pins 9, 14 of the 7-segment display to Vcc.

Apply the 4-bit BCD digits through switch and observe the decimal display from 0 to 9. Inputs 1010 through 1111 have no meaning in BCD.



Results:

(1) Show your simulation and the circuit to me, or the lab assistant.

Evaluator' signatures:_____ (simulation) _____ (experiment)

(2) Email you simulation file to : xyz@kfupm.edu.sa

(3) Fill the following table using your experimental results.

Inputs		Output
Binary Number	BCD Number	Seven-Segment Display
0000		8
0001		8
0010		8
0011		8'
0100		8
0101		8
0110		8
0111		8
1000		8
1001		8
1010	INVALID	8
1011	INVALID	8
1100	INVALID	8
1101	INVALID	8
1110	INVALID	8
1111	INVALID	8

Submit this report to me, or the lab assistant.

Group Members:

(5) _____

(6) _____

(7) _____

(8) _____

LAB 6: 4-BIT BINARY ADDER/SUBTRACTOR

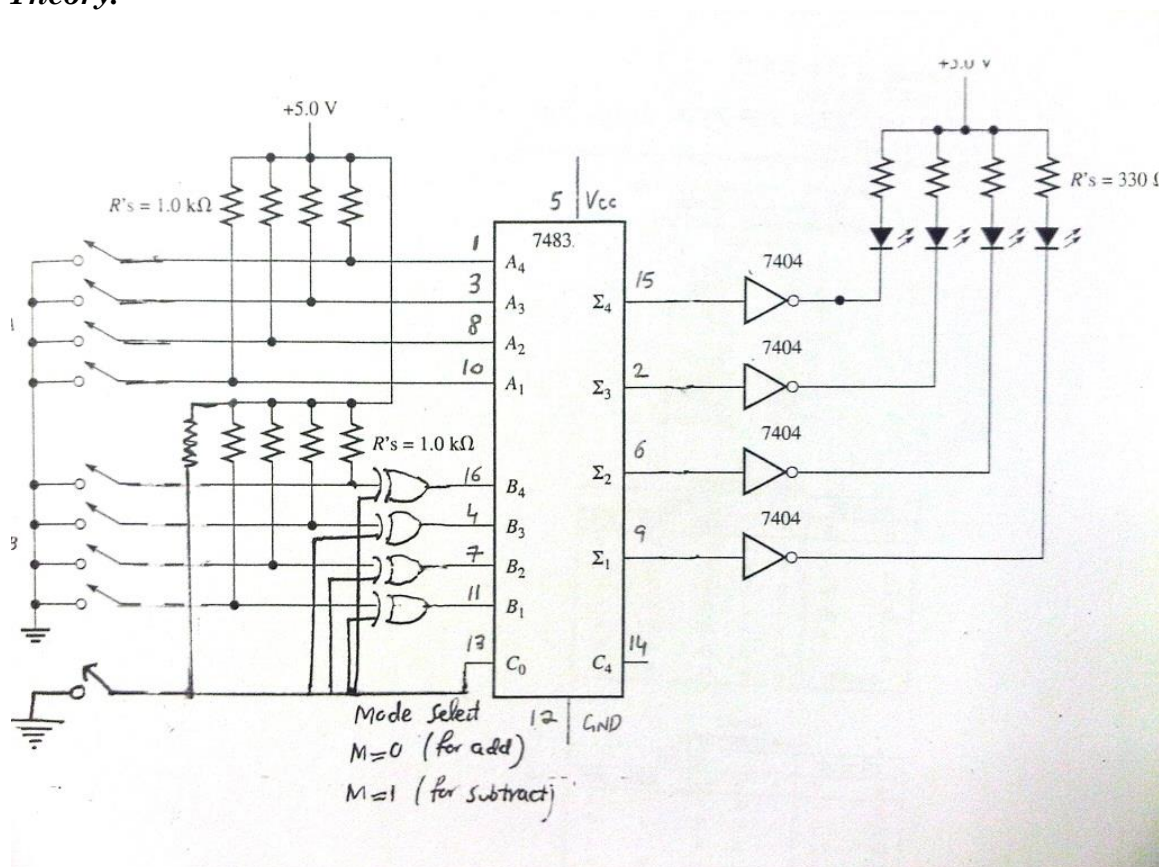
Objectives:

- Build and test the 4-bit binary adder/subtractor circuit.

Material Needed:

7483 (Qty=1), 7404 (Qty=1), 330 ohms resistors (Qty=5), 1 KOhms (Qty=10), LEDs (Qty=5), Switch (Qty=2)

Theory:



IC type 7483 is a four-bit binary parallel adder. The four bit binary input numbers are A₁ through A₄, and B₁ through B₄. The four-bit sum is obtained from S₁ through S₄. C₀ is the input carry and C₄ the output carry.

Two binary numbers can be subtracted by taking the 2's complement of the subtrahend and adding it to the minuend. The 2's complement can be obtained by taking the 1's complement and adding 1. To perform A-B, the complement of B is taken and added to the four bits of A, and 1 is added from the input carry. As shown in figure below, four XOR gates complement the bits of B when the mode select M=1, and leaves the bits of B unchanged when M=0. Thus, when the mode select M is equal to 1, the input carry C₀ is equal to 1, and the sum output is A plus the 2's

complement of B. When M is equal to 0, the input carry is equal to 0 and the sum generates A+B.

Procedure:

- (1) Connect the adder-subtractor circuit by giving input through switches and outputs connected to LEDs. Complete the table shown below:

Operations	Inputs/Outputs			Comment on your result
	C0	S	C4	
9+5				
9+15				
9-5				
5-9				
9+9				
9-9				

Group Members: _____, _____
_____, _____

LAB 7: JK FLOP-FLOP

Objective:

To learn to use flip-flops in digital circuits

Materials needed:

7476A dual JK flip-flop (Qty=1)

390 Ω resistors (Qty=3), 1k Ω resistor (Qty=4)

LEDs Red, Green, Yellow (Qty=1 each)

4-position DIP switch (Qty=1)

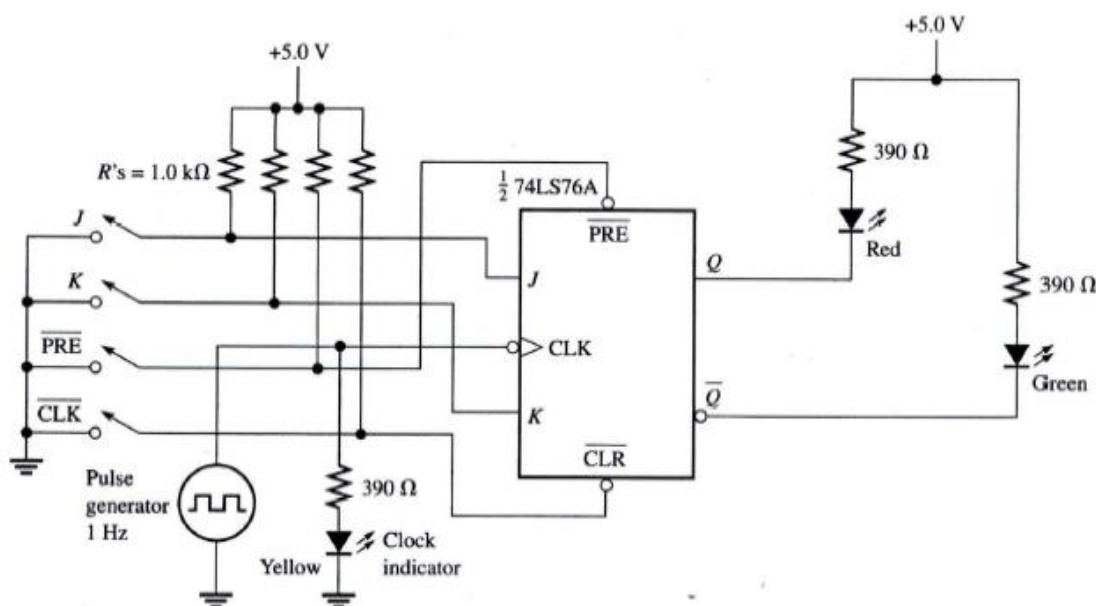
Theory:

The JK flip-flop is the most versatile of the three basic flip-flops (D, SR). All applications for flip-flops can be accomplished with either the D or JK flip-flop. The clocked SR is seldom used; it is used mostly as an internal component of the integrated circuit.

JK flip-flops are available as either edge or pulse triggered devices. The older 7476 is a level triggered flip-flop. The 74LS76A is edge-triggered on the HIGH to LOW transition of the clock.

Procedure:

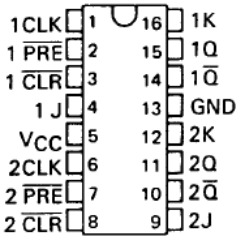
Build the circuit given below on the circuit board, and take different observations.



Data and Observations:

J	K	\overline{PRE}	\overline{CLR}	Q	\overline{Q}
0	0	1	1		
0	1	1	1		
1	0	1	1		
1	1	1	1		
0	1	0	1		
1	0	1	0		
1	1	1	0		
1	1	0	1		

SN5476, SN54LS76A . . . J PACKAGE
 SN7476 . . . N PACKAGE
 SN74LS76A . . . D OR N PACKAGE
 (TOP VIEW)



'76
FUNCTION TABLE

INPUTS					OUTPUTS	
PRE	CLR	CLK	J	K	Q	Q̄
L	H	X	X	X	H	L
H	L	X	X	X	L	H
L	L	X	X	X	H↑	H↑
H	H	⌋	L	L	Q ₀	Q̄ ₀
H	H	⌋	H	L	H	L
H	H	⌋	L	H	L	H
H	H	⌋	H	H	TOGGLE	

'LS76A
FUNCTION TABLE

INPUTS					OUTPUTS	
PRE	CLR	CLK	J	K	Q	Q̄
L	H	X	X	X	H	L
H	L	X	X	X	L	H
L	L	X	X	X	H↑	H↑
H	H	↓	L	L	Q ₀	Q̄ ₀
H	H	↓	H	L	H	L
H	H	↓	L	H	L	H
H	H	↓	H	H	TOGGLE	
H	H	H	X	X	Q ₀	Q̄ ₀

Submit this report to me, or the lab assistant.

Group Members:

- (1) _____
- (2) _____
- (3) _____
- (4) _____

LAB #8: ONE-SHOT AND ASTABLE MULTIVIBRATORS FOR TRAFFIC-SIGNAL CONTROL SYSTEM

Objective:

Learn how to design, simulate, built, and test, one-shot and astable multivibrators, using the 555 timer.

Materials needed:

555 Timer (Qty=1)

Theory & Procedure:

There are three types of multivibrators: the bistable, the monostable (or one-shot), and the astable. The name of each type refers to the number of stable states. The bistable is simply a latch or flip-flop that can be either set or reset and will remain in either state indefinitely. The one-shot has one stable (or in-active) state and one active state, which requires an input trigger to assert. When triggered, the one-shot enters the active state for a precise length of time and returns to the stable state to await another trigger. Finally, the astable multivibrator has no stable state and alternates between HIGH and LOW by itself. It frequently functions as a clock generator, since its output is a constant stream of pulses. Many systems require one-shot or astable multivibrators.

The traffic signal control system requires two one-shots and an astable multivibrator as a clock. In this experiment the components for the one-shot and astable multivibrator is specified, and it is required to test the on-time, frequency and duty-cycle for the output.

555 is a general purpose IC that can operate as an astable or as a one-shot. As a one-shot, the timer is limited to pulse widths of not less than about 10 μ s or frequencies not greater than 100 kHz. This experiment includes an introduction to the 555 timer, the first and still the most popular timer. It is not a TTL device but can operate on +5V (and up to +18V) so it can be TTL or CMOS compatible. The timer is extremely versatile but has limited triggering logic. Some applications include accurate time-delay generation, pulse generation, missing pulse detectors, and voltage-controlled oscillators (VCOs).

(a) Monostable multivibrator

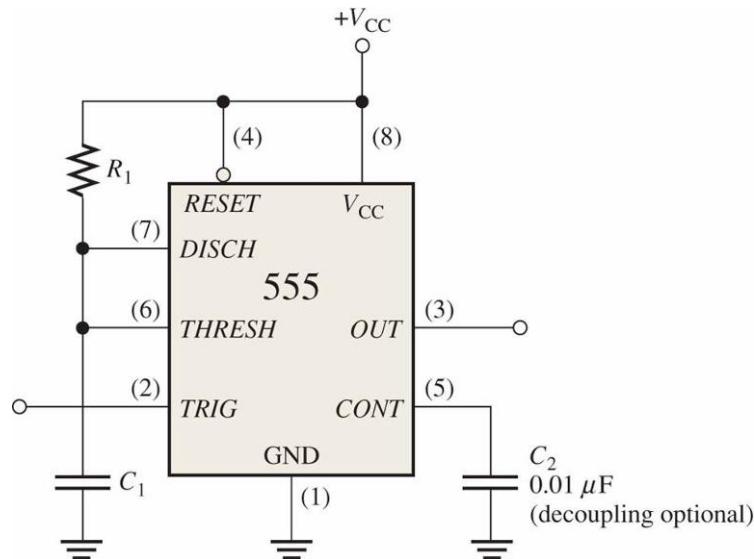
The following figure below shows the circuit diagram in order to use 555 as a monostable multivibrator. Assuming LOW state is the stable state of the multivibrator. Upon giving a HIGH pulse at the TRIG pin (pin 2) for a short interval of time switches the output pin (pin 3) from LOW to HIGH state. The on-time for the output pulse is given by:

$$t_w = 1.1R_1C_1$$

TASK1: (EXPERIMENT)

Design, built and test the circuit to use 555 as monostable multivibrator to generate a pulse of width = 4 seconds (for the orange light) for the traffic signal when triggered. Select appropriate values for R1 and C1.

A complete circuit is given below for your reference.



Check the output at pin3 on the Oscilloscope. **On-time = _____ seconds**

(b) Astable multivibrator

A basic astable circuit is shown in the figure below. There are two timing resistors. The capacitor is charged though both but is discharged only through R2. The duty-cycle and frequency of the output at pin3 is given by the following relationships.

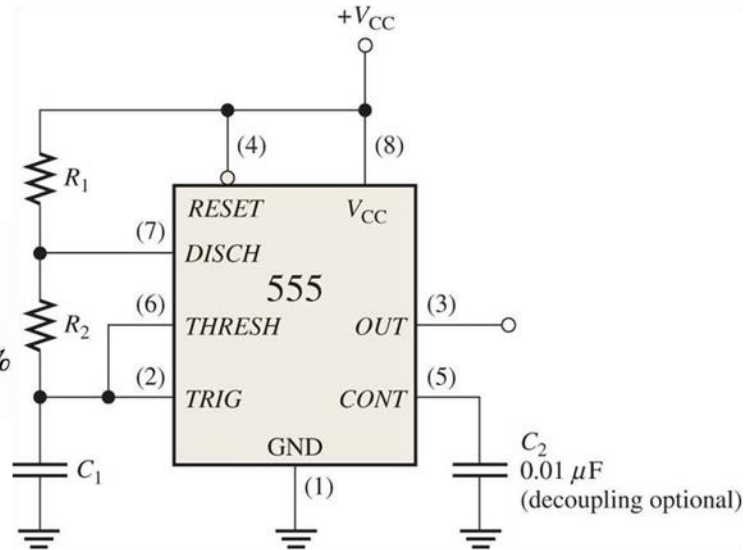
TASK2: (EXPERIMENT)

Design, built and test the circuit to use 555 as astable multivibrator to generate a clock pulse of 10kHz to operate the traffic signal. Select appropriate values for R1, R2 and C1.

$$f = \frac{1.44}{(R_1 + 2R_2)C_1}$$

$$\text{Duty cycle} = \frac{t_H}{T} = \frac{t_H}{t_H + t_L}$$

$$\text{Duty cycle} = \left(\frac{R_1 + R_2}{R_1 + 2R_2} \right) 100\%$$

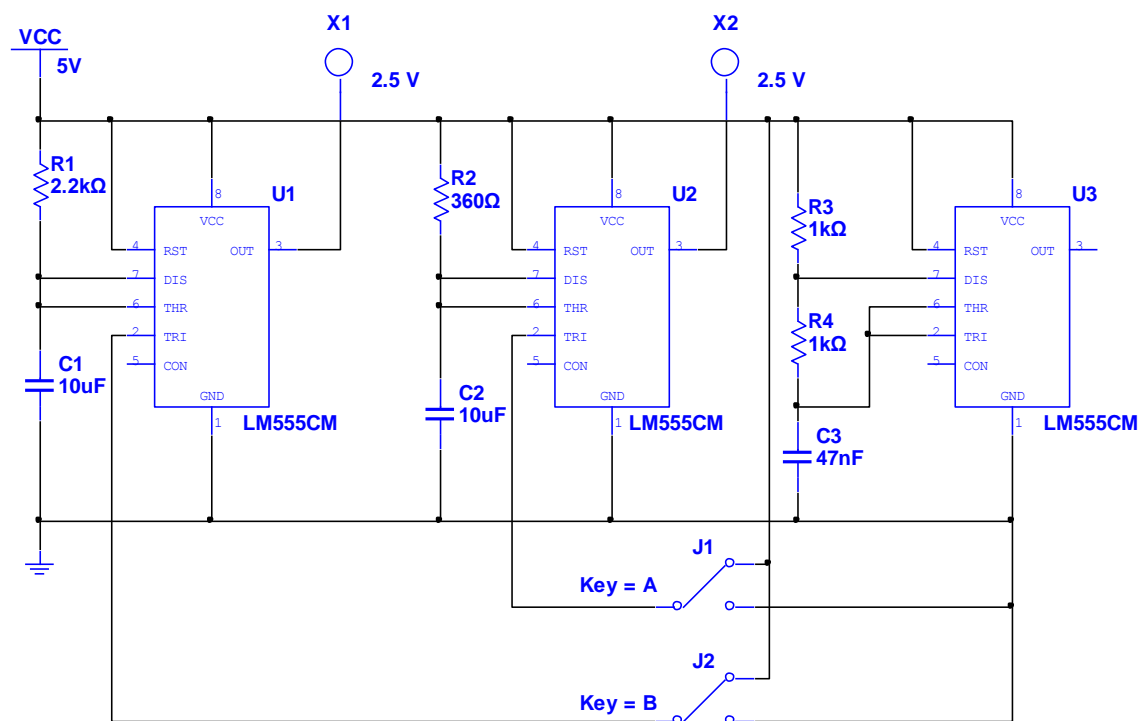


Check the output at pin3 on the Oscilloscope. **On-time** = _____ seconds, **Duty-cycle** = _____ %

TASK 3: (SIMULATION)

Below is the circuit for traffic signal control. Simulate the circuit in Multi-Sim.

- Output 1 generates pulse of width = 25 seconds (Red and Green light) when triggered. (monostable)
- Output 2 generates pulse of width = 4 seconds (Orange light) when triggered. (monostable)
- Output 3 generates a clock of 10kHz. (astable)



Group Members:

- (1) _____
- (2) _____
- (3) _____
- (4) _____

LAB 9: TRUNCATED SEQUENCE USING ASYNCHRONOUS COUNTER

Objective:

To learn to use counters in digital circuits using the Asynchronous Counter

Materials Needed:

7493A binary Counter (Qty=1)

BCD to seven-segment decoder (7447 or 7448) (Qty=1)

Seven Segment display MAN72A or MAN74A (Qty=1)

1k Ω resistor (Qty=2), 330 Ω resistors (Qty=7)

Theory & Procedure:

Digital counters are classified as either synchronous or asynchronous, depending on how they are clocked. Synchronous counters are made up from a series of flip-flops that are clocked together. By contrast, asynchronous counters are a series of flip-flops, each clocked by the previous stage, one after the other. Since all stages of the counter are not clocked together, a ripple effect propagates as various flip-flops are clocked. For this reason, asynchronous counters are called *ripple counters*. Ripple counter can be made several stages of JK or D flip-flops.

For 7393A, the count advances as the clock input becomes low (on the falling-edge), this is indicated by the bar over the clock label. This is the usual clock behavior of ripple counters and it means a counter output can directly drive the clock input of the next counter in a chain.

The counter is in two sections: clockA-QA and clockB-QB-QC-QD. For normal use connect QA to clockB to link the two sections, and connect the external clock signal to clockA. For normal operation at least one reset0 input should be low, making both high resets the counter to zero (0000, QA-QD low).

Counting to less than the maximum (9 or 15) can be achieved by connecting the appropriate output(s) to the two reset0 inputs. If only one reset input is required the two inputs can be connected together. For example: to count 0 to 8 connect QA (1) and QD (8) to the reset inputs.

Display the outputs QA-QD seven-segment display for counting from 0-9 for 3 different input frequencies ($f=0.1, 1, 10\text{Hz}$). Use the figure L9.2 given above as reference.

Connect 1k Ω resistors before Vcc and CLK.

NOTE: Retain your circuit for the next Lab.

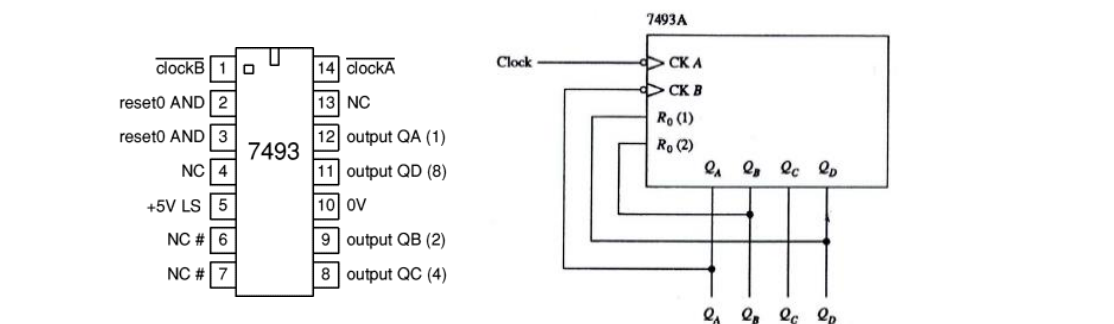


Figure L9.1 (a) Pin connections for 74LS93A, (b) Connections for observations

Data and Observations:

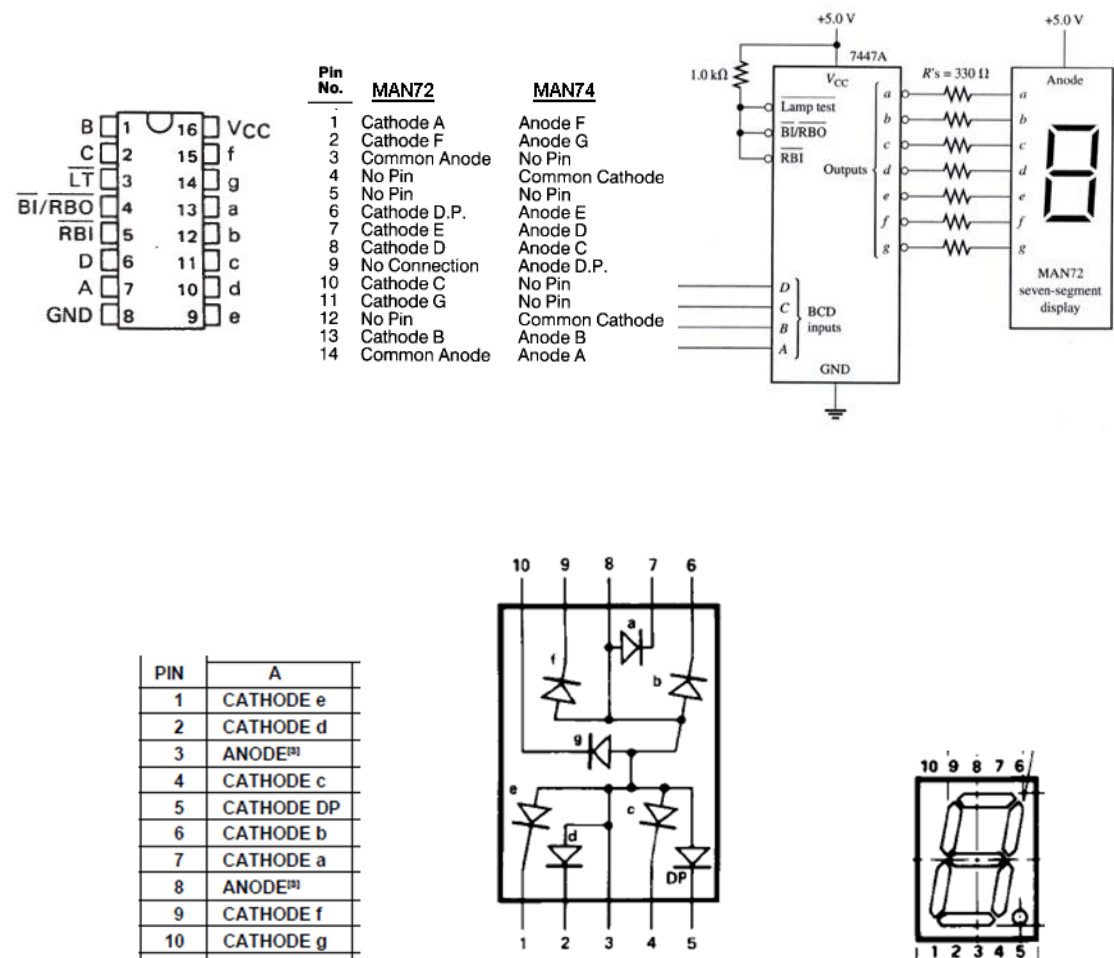


Figure L9.2 Pin-outs for (Left) BCD-to-Seven segment decoder (7447/7448), (Right) display (MAN72A, MAN74A), (Bottom) HDSP-5501 Common-Anode Seven-segment Display

Group Members:

(1) _____

(2) _____

(3) _____

(4) _____

LAB 10: STORING TO, AND READING FROM MEMORY

Objective:

Learn to use memory by building digital circuits using a 16x4 RAM

Materials needed:

74LS189 (16x4) RAM (Qty=1)

4-position DIP switches (Qty=2)

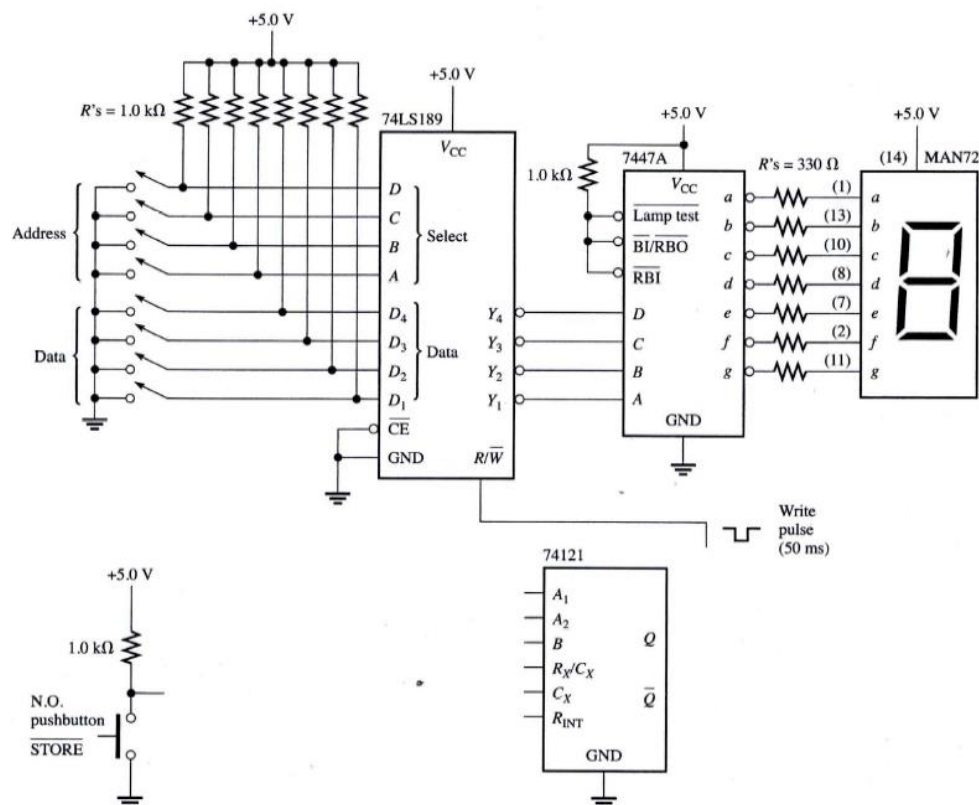
BCD to seven-segment decoder (7447) (Qty=1)

Seven Segment display MAN72A (Qty=1)

1k Ω resistor (Qty=9), 330 Ω resistors (Qty=7)

Push button (Qty=1)

Theory and Procedure:



The basic cell in a static RAM is a flip-flop; it can be set or reset for writing operations or tested without changing its state for read operations. In addition, the RAM contains logic gates to control the read and write functions and decoding circuitry. The RAM is organized as an array containing the memory cells in row and

columns. In memories, the number of bits treated as one entity is considered the word size. Each word in a read/write memory is accessed by a set of address bits, representing the location of the word. There are also control lines to enable the chip, and to perform read or write operation.

The outputs of memory and other devices are frequently connected to a common set of wires known as a bus. For this experiment, data and address information is input using switches. The memory used is 74LS189 tri-state RAM, a 64 bit memory organized as sixteen 4-bit words. There are 4 address lines (A-D), four data inputs (D1-D4), and four output lines (Y1-Y4) plus chip enable \overline{CE} and a read/write control line R/\overline{W} . The output are shown with inverting bubbles, hence the complement of the input data is present at the output when reading the data. The tri-state outputs are in high impedance state when R/\overline{W} line is LOW. To write the data, the address is place on the address bus, data is place on the data bus, and \overline{CE} and R/\overline{W} are low to WRITE. To read data, the address is placed on the address bus; \overline{CE} is LOW and R/\overline{W} is HIGH.

Data and Observations

Write the five-digit postal code for KFUPM into the first five locations of memory, then read those and display them on the seven-segment.

Table 1: Write operation

\overline{CE}	R/\overline{W}	$A-D$				$D1-D4$			

Table 2: Read operation

\overline{CE}	R/\overline{W}	$A-D$	Seven-Segment Output