Table of Content

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 1** Student's Name:

Objective: To get familiar with Analog & Digital Training System (M21-7000).

Goal: In this experiment students will develop practical understanding with different functions of Analog & Digital Training System (M21-7000).

Required Tools/Equipment:-

1) Analog & Digital Training System (M21-7000).
2) Connecting Wires (Jumpers).
3) NOT Gates IC (74HC04).

Introduction:-

S.M.I.U Electronic Laboratory equipped with Analog & Digital Training System (M21-7000). This training system has almost all essential functions which are required for both Analog and Digital experiments. Because this training system will be used throughout **Digital Logic and Design** subject's practical work, it need to be examined thoroughly.

This training system has a power switch 

to enable/disable all functions. It contains following main sections:

Solderless Breadboard:

This training system has a breadboard which can be removed from its place if required. This breadboard has 2820 connecting points (holes). It has horizontal, vertical and separation region.

Function Generator:

This trainer has a section of Function Generator. Function Generator is a device which is used to generate different ac voltage wave foam with selected amplitude and frequency. It has 3 connecting holes to connect circuit input. Output and GND are used to connect selected frequency and amplitude output waveform to any circuit input. TTL and GND is used to connect fixed 4V (p-p) TTL mode voltage waveform to any circuit input.

How to select frequency**:** To select desired frequency, rotate “Frequency Range” knob to select desired frequency multiplier “x1, x10, x100, x1k and x10k” and select suitable “Frequency” by rotating knob from 1 to 10.

Example: To select 2.5 KHz frequency signal from function generator,

Rotate “Frequency Range” multiplier knob to select **x1k** and “Frequency” knob to **2.5**. So frequency will be $2.5 \times 1kHz = 2.5\ kHz$

How to select amplitude**:** To select desired amplitude of voltage waveform, rotate “Amplitude” knob to suitable position from 0 to 10.

Example: To select amplitude of output waveform to 5 Volts (p-p), rotate “Amplitude” knob to 5.

How to select function of waveform: To select desired function of output waveform, rotate “Function” knob to one of the three given functions (Triangular, Sinusoidal or Square).

<u>Dc Power Supply</u>:

Analog & Digital Training System (M21-7000) has separate DC power supply section which provides both variable and fixed DC voltage levels. It has total 6 connecting terminals. Three terminals +5V, GND and -5V to select +5V/-5V fixed voltage and three terminals 0~+15V, GND and 0~-15V to select variable voltage range from 0 to +15V and 0 to -15V.

***How to select +5 fixed voltage*:** To provide +5V fixed dc voltage, connect one jumper to +5V terminal and second to GND terminal to connect input of any circuit which requires +5V dc voltage**.**

***How to select -5 fixed voltage*:** To provide -5V fixed dc voltage, connect one jumper to -5V terminal and second to GND terminal to connect input of any circuit which requires -5V dc voltage**.**

***How to select any voltage from 0 to +15 volts*:** To select any voltage level from 0 to +15V, connect one jumper to terminal labeled as 0~+15V which will serve as positive terminal of DC supply and connect second jumper to terminal labeled as GND which will serve as ground terminal. Rotate knob which is labeled as +V to select desired voltage level.

***How to select any voltage from 0 to -15 volts*:** To select any voltage level from 0 to -15V, connect one jumper to terminal labeled as 0~-15V which will serve as negative terminal of DC supply and connect second jumper to terminal labeled as GND which will serve as ground terminal. Rotate knob which is labeled as -V to select desired voltage level.

<u>Potentiometer (Variable Resistor)</u>:

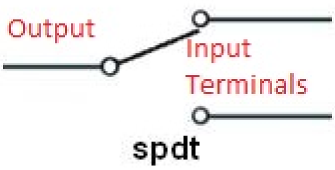
There are two potentiometers available in this section. One range from 0~100KΩ and second range from 0~1KΩ. Each potentiometer has three terminals and one rotating knob.

How to select any resistance value from 0 to 100KΩ: Connect three jumpers to terminal 1, 2 and 3 respectively. Conventional current enters in Terminal 1 and leaves from adjustable terminal 2 and 3. When rotating dial at 0, adjustable terminal 2 coincides with the terminal 1, all current passes through terminal 2 hence potentiometer offer 0Ω resistance. When rotating dial at 10, adjustable terminal 2 coincides with the terminal 3, all current passes through terminal 3 hence potentiometer offer 100KΩ resistance. When rotating dial position between 0 and 10, current passes through terminal 2 and 3 hence potentiometer offer resistance R> 0 <100KΩ.

How to select any resistance value from 0 to 1KΩ: Connect three jumpers to terminal 4, 5 and 6 respectively. Conventional current enters in Terminal 4 and leaves from adjustable terminal 5 and fixed terminal 6. When rotating dial at 0, adjustable terminal 5 coincides with the terminal 4, all current passes through terminal 5 hence potentiometer offer 0Ω resistance. When rotating dial at 10, adjustable terminal 5 coincides with the terminal 6, all current passes through terminal 6 hence potentiometer offer 1KΩ resistance. When rotating dial position between 0 and 10, current passes through terminal 5 and 6 hence potentiometer offer resistance R> 0 <1KΩ.

<u>Toggle Switches:</u>

There are 16 toggle switches from SW0 to SW15 and their relevant connecting holes for output terminals. These switches are used to provide Boolean input (0 or 1) to the digital logic components/circuit. There are three terminals of toggle switch. Two terminals are used for inputs and one terminal is used for output. One input terminal is internally connected with positive terminal of DC supply and second terminal is internally connected with ground/very low voltage. When switch position is upward, output terminal is connected with first input terminal and logical 1 appears on the connecting hole/terminal. When switch position is downward, output is connected with second input terminal and logical 0 appears on the connecting hole/terminal.



<u>LED Data Display:</u>

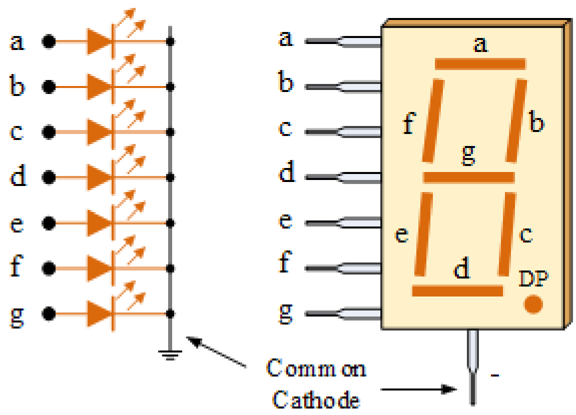
This section contain 16 LED's and their relevant connecting holes from 0 to 15. Anode side of all LED's are connected to their relevant connecting holes and Cathode is internally connected to ground. When no input or logical 0 is provided to the LED input, it becomes reverse biased and remain turned off. When logical 1 is provided to the LED input, it becomes forward biased and turned on.

Pulse Switches:

There are two pulse switches A and B with four output terminals A, $\overline{A}$, B and $\overline{B}$. Two terminals for each pulse switch. When Switch A is pressed, terminal A connecting hole provides logic HIGH output. When Switch A is released, terminal A connecting hole provides logic LOW output. Same applies for Switch B and its output terminals.

BCD Displays:

This section contains two Common Cathode 7 Segment display units and four connecting holes as 4 input terminals for each 7 Segment Display unit. Two 7 Segment Display Decoder driver IC's are internally connected to each display unit.



Following is the output pattern of 7 Segment Display unit's individual LEDs with the relevant BCD inputs.

Decimal Or Function	BCD Inputs D	C	B	A	Outputs a	b	c	d	e	f	g
0	L	L	L	L	H	H	H	H	H	H	L
1	L	L	L	H	L	H	H	L	L	L	L
2	L	L	H	L	H	H	L	H	H	L	H
3	L	L	H	H	H	H	H	H	L	L	H
4	L	H	L	L	L	H	H	L	L	H	H
5	L	H	L	H	H	L	H	H	L	H	H
6	L	H	H	L	L	L	H	H	H	H	H
7	L	H	H	H	H	H	H	L	L	L	L
8	H	L	L	L	H	H	H	H	H	H	H
9	H	L	L	H	H	H	H	L	L	H	H
10	H	L	H	L	L	L	L	H	H	L	H
11	H	L	H	H	L	L	H	H	L	L	H
12	H	H	L	L	L	H	L	L	L	H	H
13	H	H	L	H	H	L	L	H	L	H	H
14	H	H	H	L	L	L	L	H	H	H	H
15	H	H	H	H	L	L	L	L	L	L	L

There are two other sections in M21-7000 which are not covered here.

Conclusion:-

In this lab we have learned about the analogue and digital training system how we use training system with IC's I online classes we get only concept of this training system.

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 2** Student's Name:

Objective: Introduction of NI Multisim Circuit Design Software.

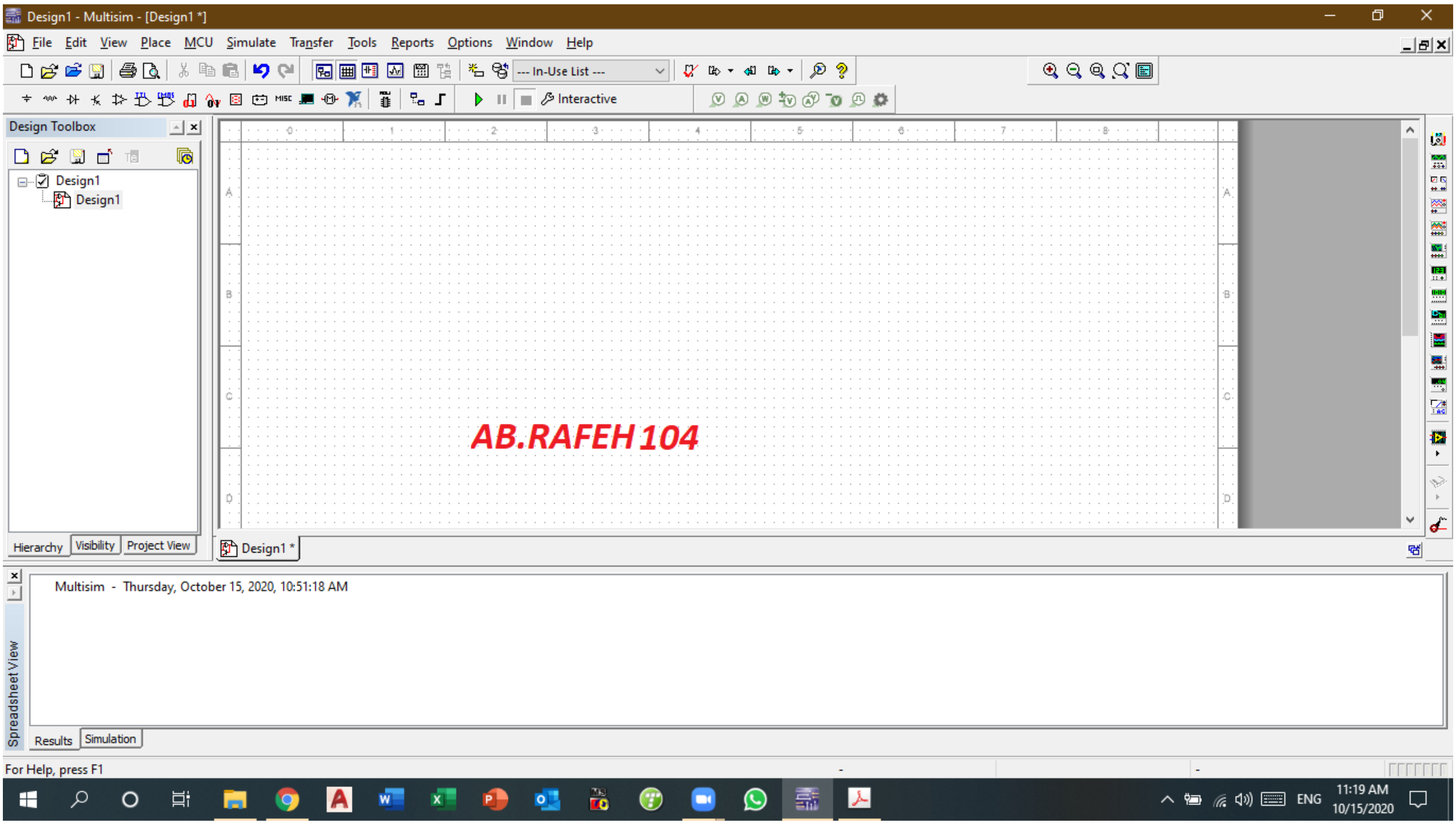
Goal: In this experiment will familiar with NI Multisim circuit Design software and will go through of its menus/function.

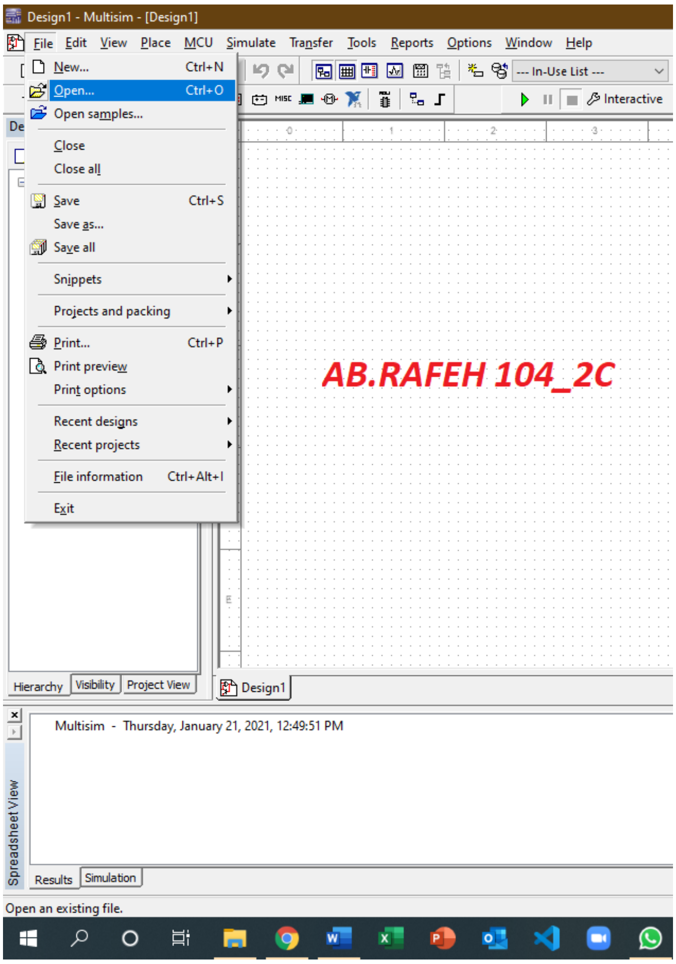
Required Tools/Equipment:-

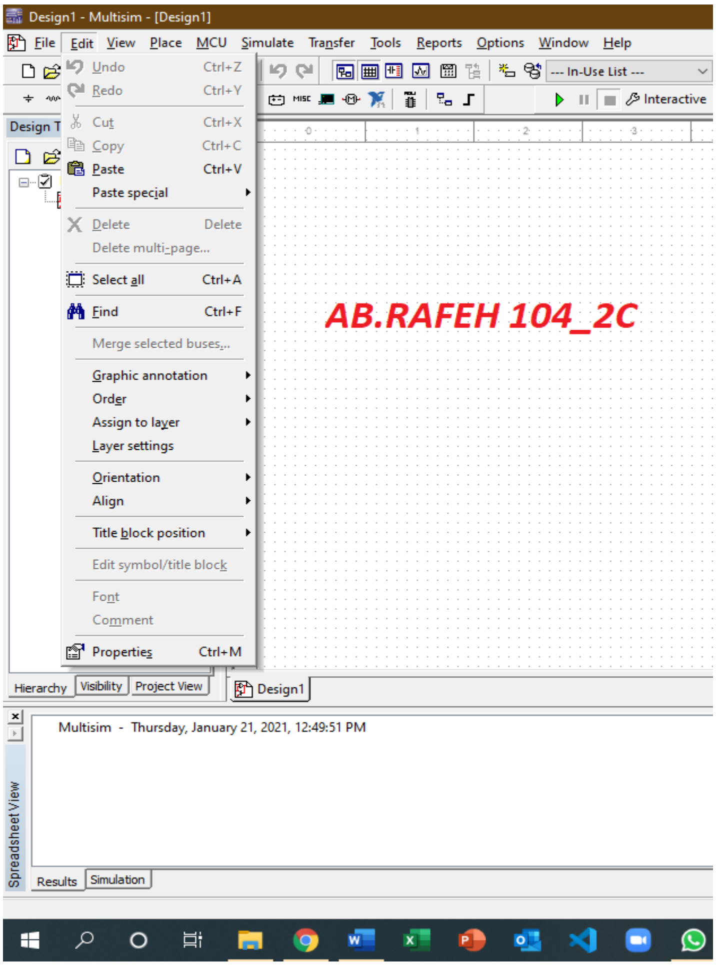
4) Laptop / Computer
5) NI Multisim Circuit Software setup

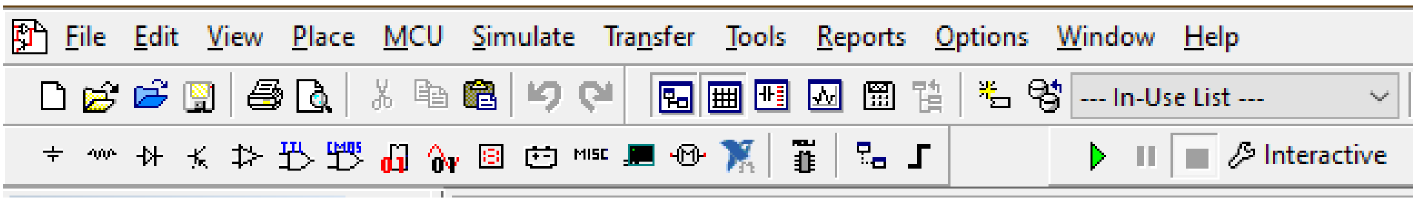
Introduction:-

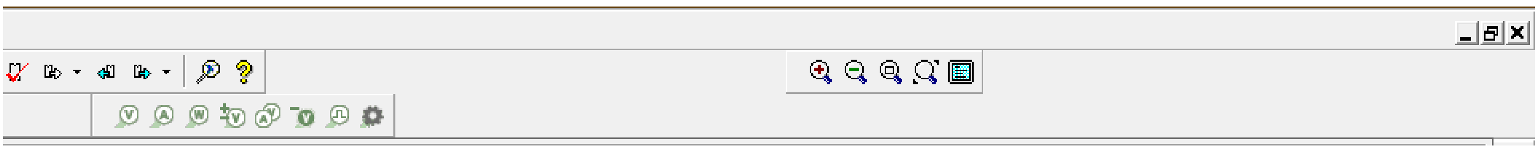
NI Multisim is an electronic schematic capture and simulation program which is a part of a suite of circuit design programs, along with NI Ultiboard. Multisim is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation. Multisim was originally created by a company named Electronics workbench, which is now a division of National Instruments (NI).

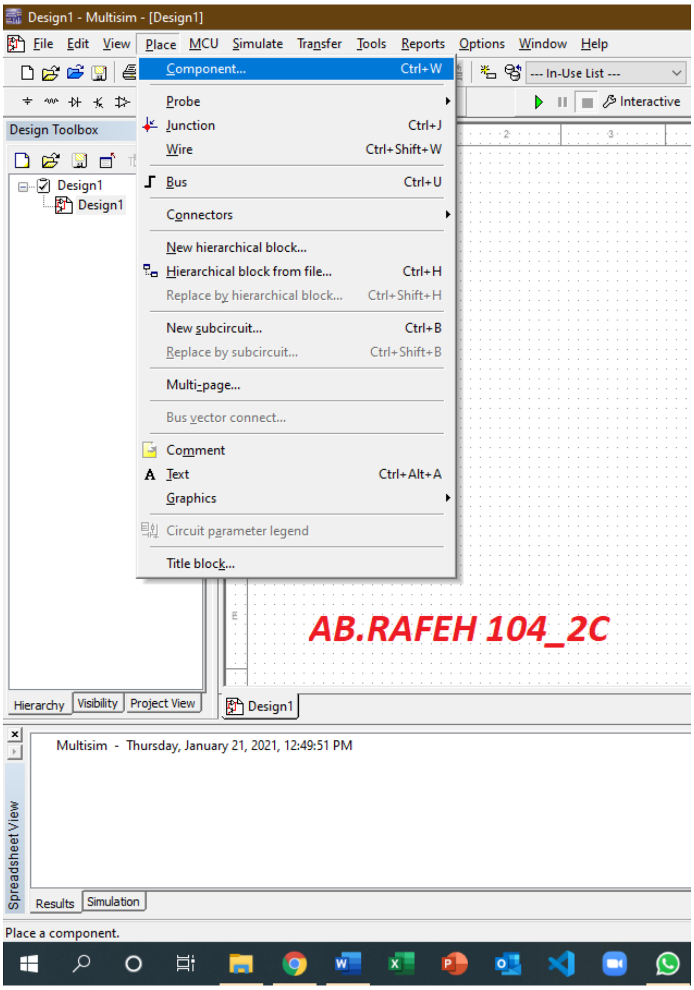


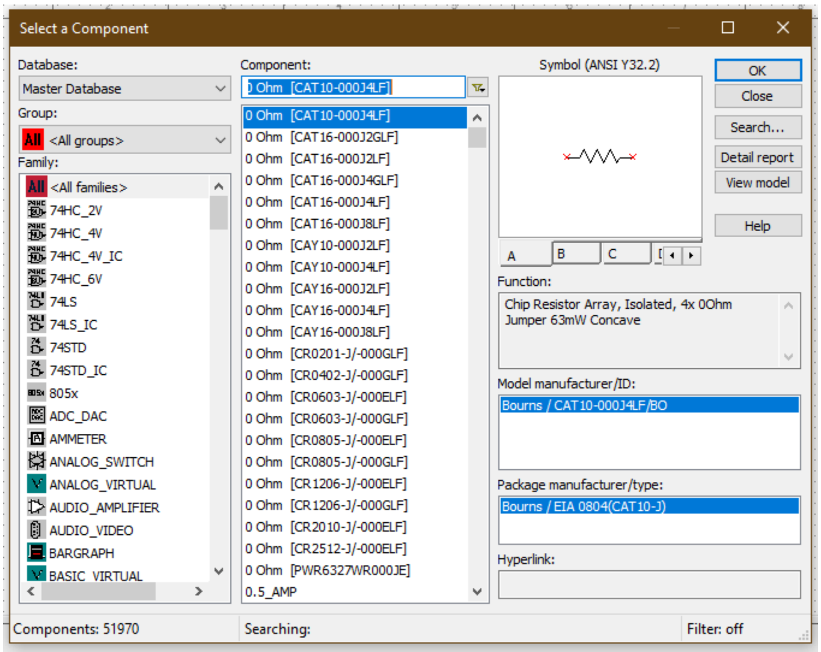

Design1 - Multisim - [Design1]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
New... Ctrl+N
Open... Ctrl+O
Open samples...
Close
Close all
Save Ctrl+S
Save as...
Save all
Snippets
Projects and packing
Print... Ctrl+P
Print preview
Print options
Recent designs
Recent projects
File information Ctrl+Alt+I
Exit
AB.RAFEH 104_2C
Hierarchy Visibility Project View
Design1
Multisim - Thursday, January 21, 2021, 12:49:51 PM
Spreadsheet View
Results Simulation
Open an existing file.

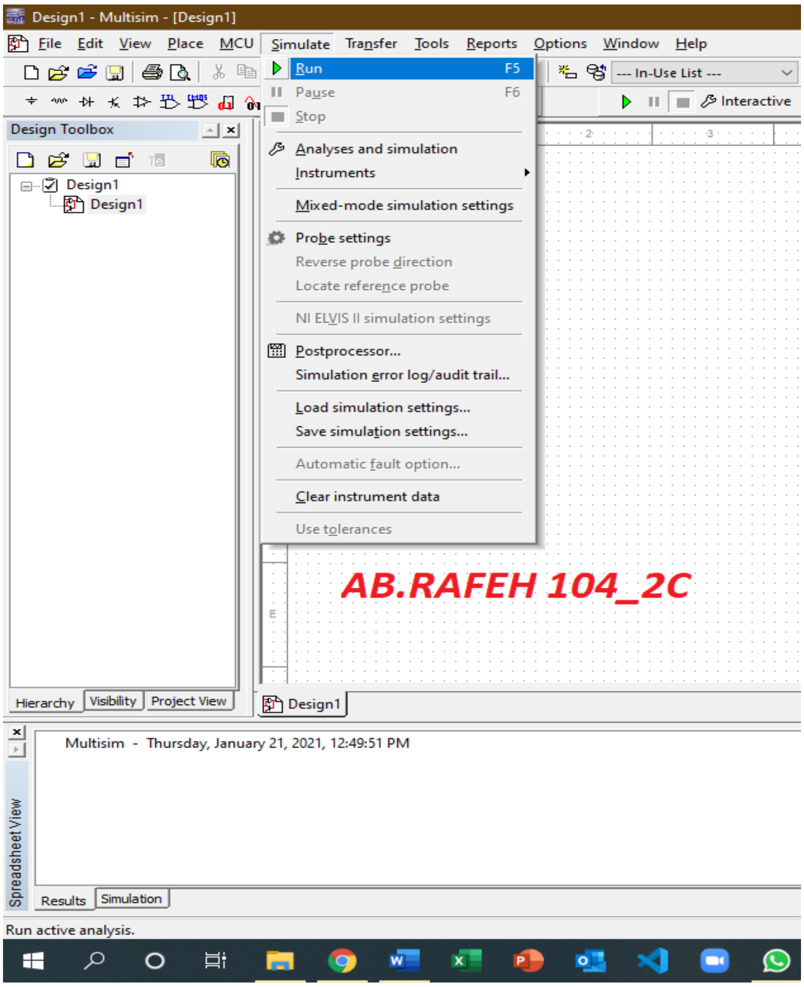

Design1 - Multisim - [Design1]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Undo Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Paste special
Delete Delete
Delete multi-page...
Select all Ctrl+A
Find Ctrl+F
Merge selected buses...
Graphic annotation
Order
Assign to layer
Layer settings
Orientation
Align
Title block position
Edit symbol/title block
Font
Comment
Properties Ctrl+M
AB.RAFEH 104_2C
Hierarchy Visibility Project View
Design1
Multisim - Thursday, January 21, 2021, 12:49:51 PM
Spreadsheet View
Results Simulation


File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
--- In-Use List ---
MISC
Interactive









Conclusion:-

In this lab we have learned About multisim software how to use and how to get any circuit components in short we familiar with multisim circuit design software.

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 3** Student's Name:

Objective: Cracking of NI Multisim Circuit Design Software

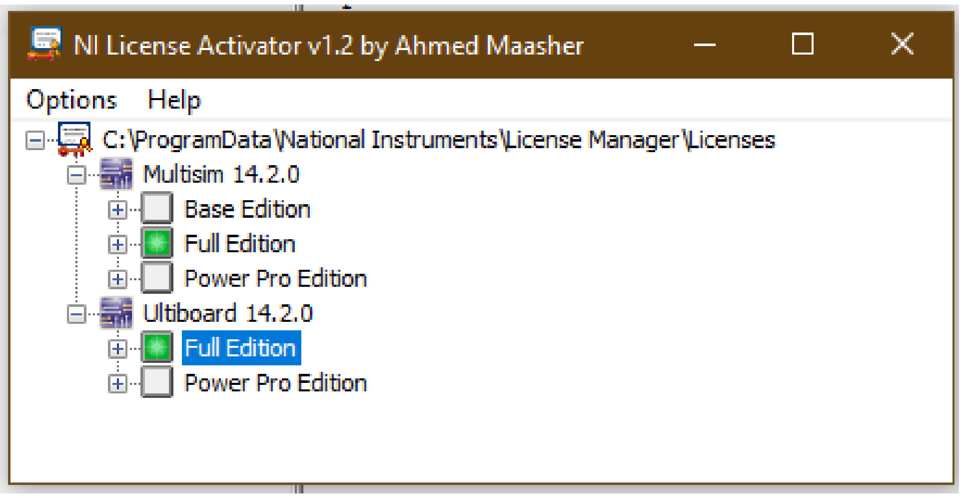
Goal: In this experiment will crack the NI Multisim circuit Design software and will go through of its main function.

Required Tools/Equipment:-

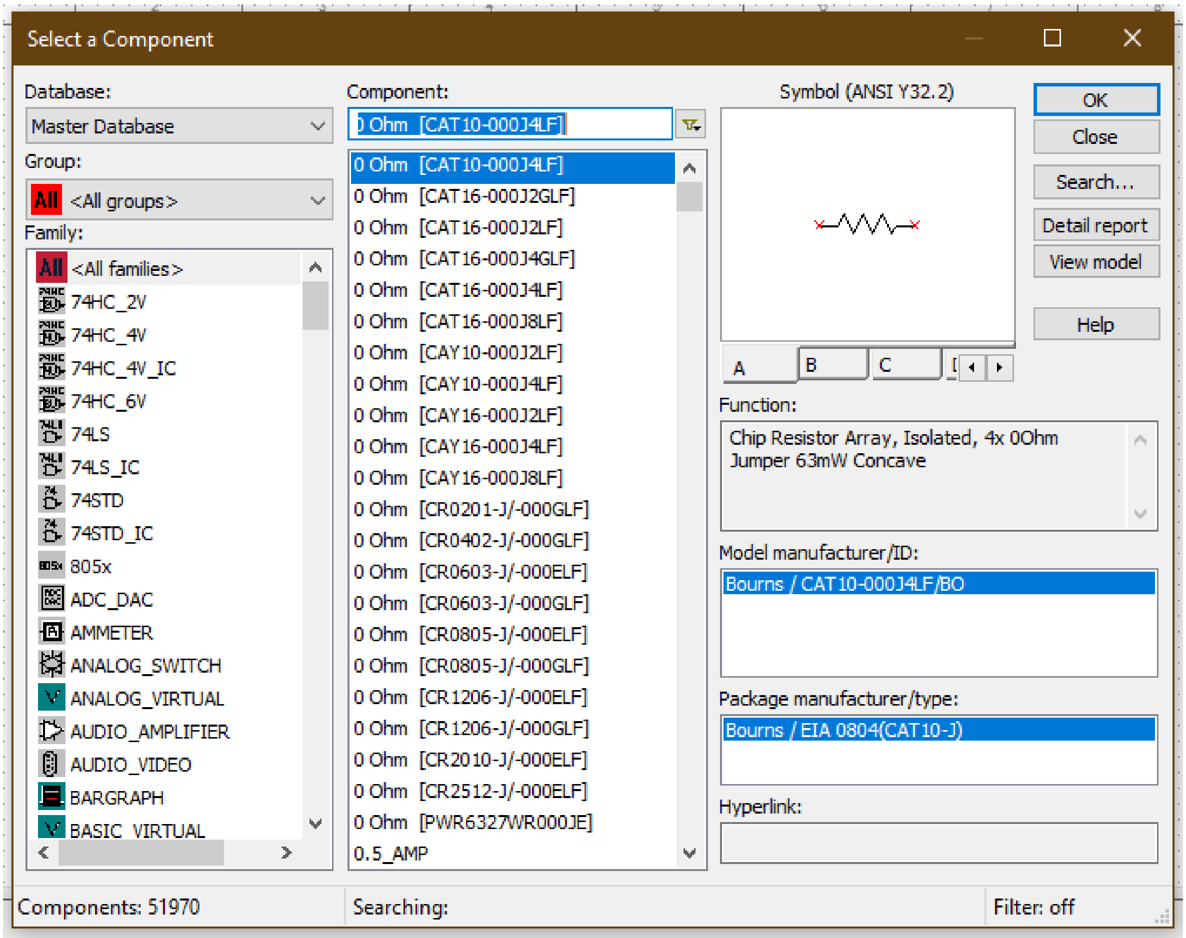
6) Laptop / Computer
7) NI Multisim Circuit Software Setup

Procedure:-

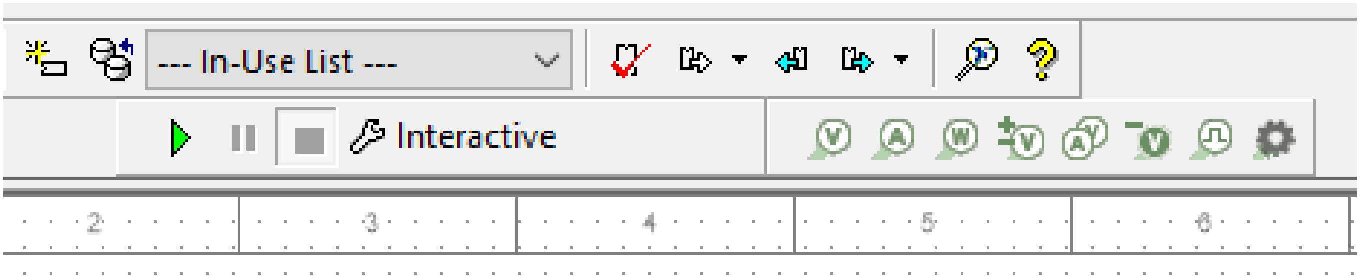
1. Go to the download file location where NI Multisim setup downloaded.
2. Run the setup and Install the software of Multisim.
3. After Complete of Installation now this time to crack the software.
4. Go to the same folder where file setup is downloaded in same folder there is Crack folder open and run the file then setup is running SS mentioned below.



5. Click right button & activate the Multisim full edition.
6. Same with the below Ultiboard click right button & activate the full edition.
7. After all of this step NI Multisim fully Installed & Activated.
8. Now this time to run the software & overview of it functions.
9. For circuit component go to the menu bar their option is place click on it then many options will see but click on component there is all component related to the circuit.



10. For seeing a result of circuit in menu bar have play, pause & stop options.



CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 4** Student's Name:

Objective: To familiarize with basic NOT Gate.

Goal: In this experiment students will implement basic Logical NOT gate practically and will observe its function.

Required Tools/Equipment:

1) Laptop / Computer
2) NI Multisim Circuit Software
3) Current Controlled SPST Switch
4) DC Power Supply
5) Resistor 1kΩ
6) Probe bulb
7) NOT gate
8) Ground

Theory:

A logic gate is an electronic circuit which makes different binary decision with respect to binary inputs. A logic gate can have one or multiple inputs but only has single output. Binary 1 is taken as Logic HIGH and binary 0 as Logic LOW. Logic gate activates only for specific input combinations. If same input combination is applied to the inputs of different logic gates, output might be 1 or 0 depends on Logic gate function. Two different logic gates might give same output for same input combinations because there are only two logic levels possible in digital electronics (i-e Logic LOW and Logic HIGH).

Logic gates are implemented through Diodes or Transistors as discrete circuit but they are also implemented directly in the form of integrated circuit (I.C). Logic gates function can also be implemented with switches, electromagnetic relays, solenoid vales, pneumatic relays and mechanical components.

Basic Logic Gates:

1) NOT.
2) AND.
3) OR.

NOT Gate:

NOT gate is the simplest logic gate among all basic logic gates. It is also referred as "Inverting Buffer" or "Digital Inverter" or even sometimes just as "Inverter" when discussing digital circuit/logic. NOT gate is the only Logic gate which requires one logical input. Its function is like a unary operator. When a single binary 0 or Logic LOW is applied to its input, it provides binary 1 or logic HIGH on its output. When a single binary 1 or logic HIGH is applied to its input, it provides binary 0 or logic LOW on its output. Clearly it inverts or complement whatever logic level is applied to its input. If A is input and Z is output then $Z = A'$. (‘) apostrophe character indicate inverting operation.

Symbol:-

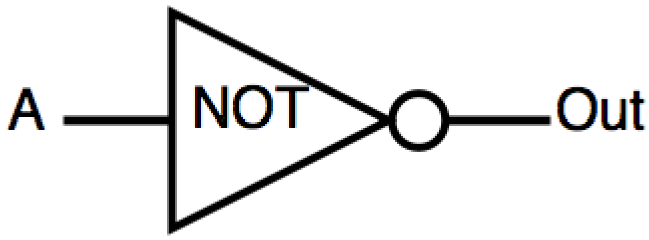
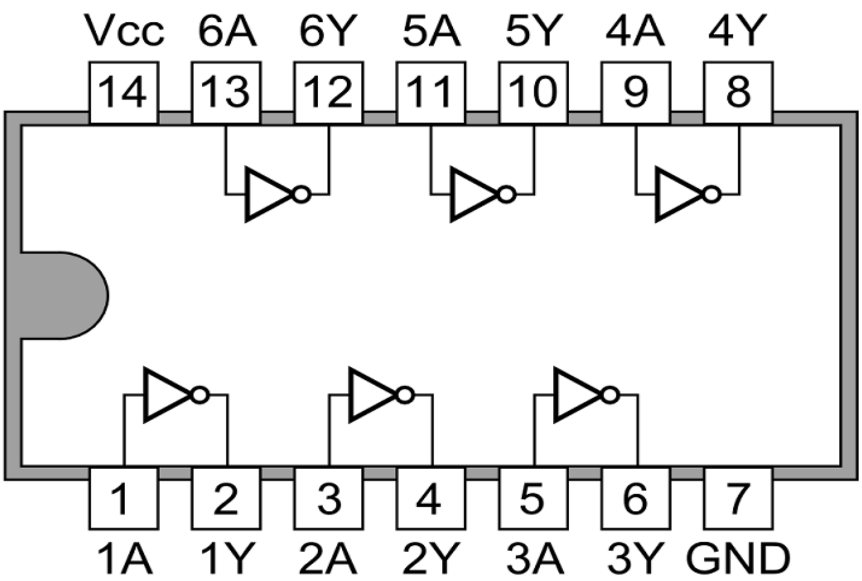


Fig: 2.1

Pin Configuration:-



74HC04 Pin Configuration Fig: 2.2

Procedure:

1) Run Multisim software.
2) Go to the menu bar.
3) In menu bar > Place > Components.
4) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
5) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
6) We put DC Power Supply 12V so equipment location is Place > Components > Master

Database > Group Sources > Power_Sources > DC_Power then click on ok and place it.

7) We put Current_Controlled SPST Switch so equipment location is Place > Components > Master Database > Group Basic > Switch > Current_Controlled_SPST then click on ok and place it.
8) We put resistor in this circuit resistor location is Place > Components > Master Database > Group Basic > Resistor > 1k then click on ok and place it.
9) NOT gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > NOT then click on ok and place it.
10) Last equipment we use in this circuit is Probe_blue bulb for getting result of this circuit the bulb location is Place > Components > Master Database > Group Indicators > Probe > Probe_blue then click on ok and place it.
11) In upward mention 10Points we discuss how we select equipment for using in this circuit through multisim software.

Now in this stage we discuss how we connected the circuit. Power supply terminal connected to the switch and the other end of switch is connected to the resistor then from the switch we take loop for the NOT gate from switch then after NOT gate we put one prob bulb for the output and ground the circuit. Example mentions in below truth table.

Input (A)	**Output (Z)**
0 (LOW)	1 (HIGH)
1 (HIGH)	0 (LOW)

Table: 2.1

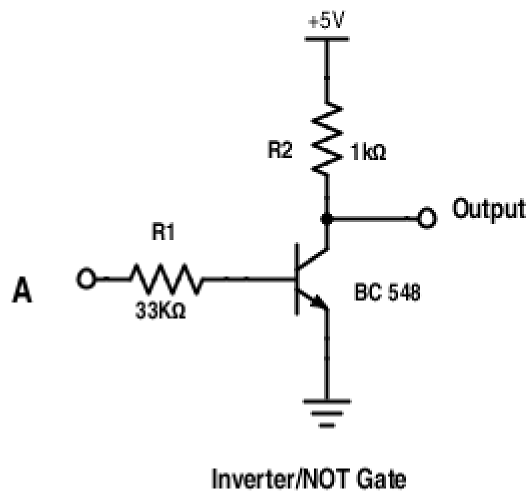
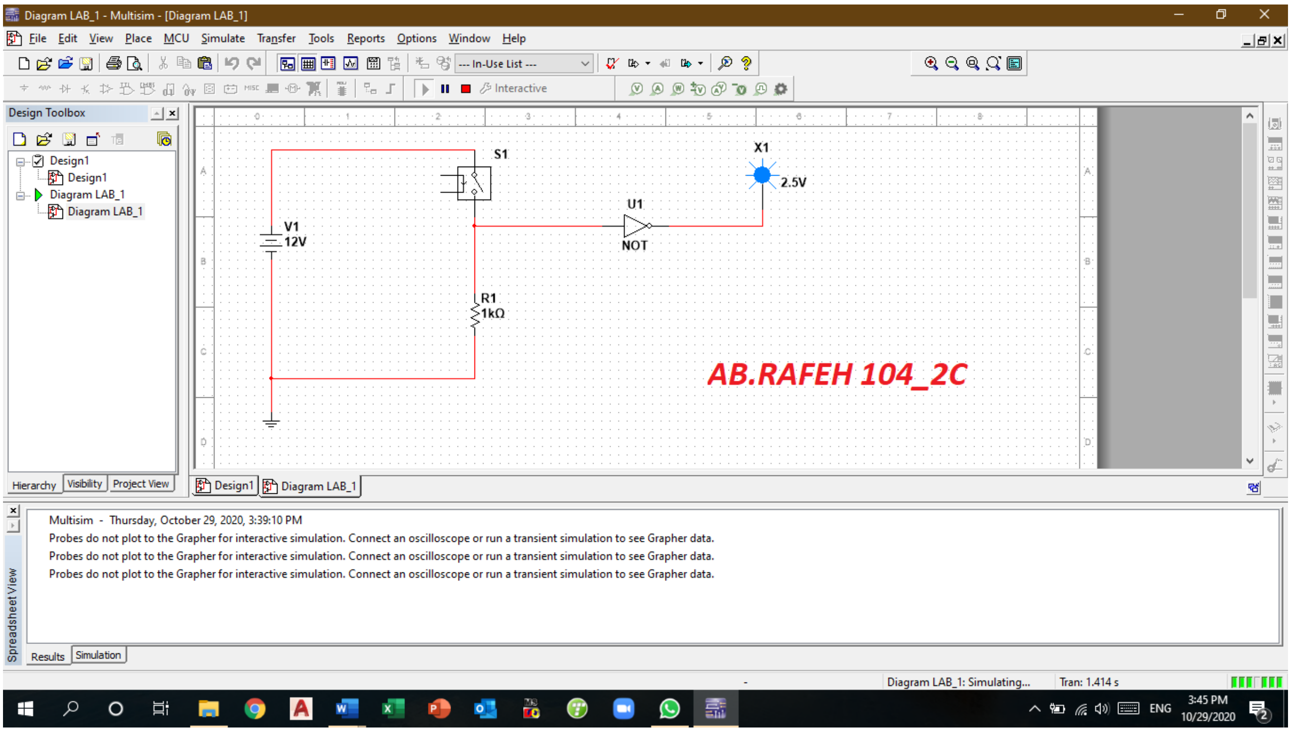


Fig 2.3 NOT Gate with Discrete Components

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of NOT gate.

Diagram:-



Answer the following questions:-

Q # 1. Is this possible to apply more than one inputs to NOT gate? Justify your answer.

Answer:

A simple 2-input logic NOT gate can be constructed using a RTL Resistor-transistor switches as shown below with the input connected directly to the transistor base. The transistor must be saturated “ON” for an inverted output “OFF” at Q.

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 5** Student's Name:

Objective: To familiarize with basic AND Gate.

Goal: In this experiment students will implement basic Logical AND gate practically and will observe its function.

Required Tools/Equipment:

1) Laptop / Computer
2) NI Multisim Circuit Software
3) SPDT Switches
4) VCC 5.0V
5) Led blue
6) AND gate
7) Ground

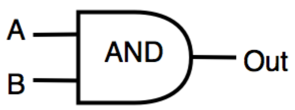
Theory:

AND Gate:

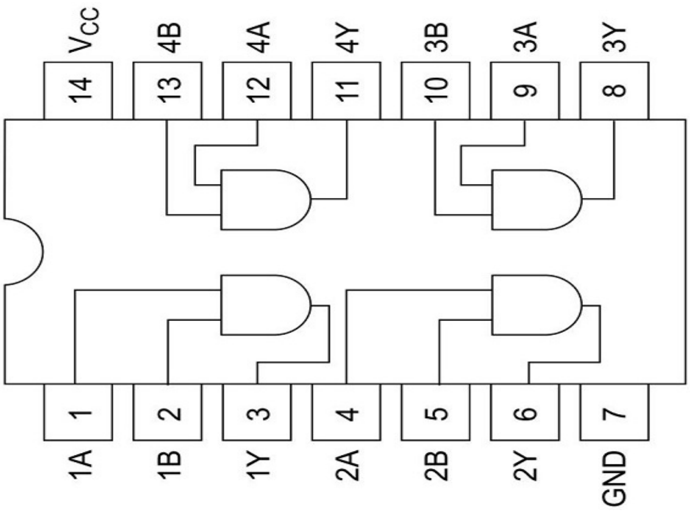
AND Gate provides binary 1 (HIGH) only when binary 1 (HIGH) is applied to all its inputs. In other words AND Gate provides binary 0 (LOW) if binary 0 (LOW) is applied to any of its inputs. If A and B are two inputs to AND Gate and Z is its output then $Z = A.B$

Dot operator is used to indicate AND operation between two or more than two logic inputs. Sometimes dot operator is omitted.

Symbol:-



Pin Configuration:-



Procedure:-

1) Run Multisim software.
2) Go to the menu bar.
3) In menu bar > Place > Components.
4) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
5) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
6) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
7) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
8) We are using two SPDT switches.
9) AND gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > AND2 then click on ok and place it.
10) Last equipment we use in this circuit is LED for getting result of this circuit the led location is Place > Components > Master Database > Group Diodes > LED > LED_blue then click on ok and place it.
11) In upward mention 10Points we discuss how we select equipment for using in this circuit through multisim software.

Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 1 connect to the VCC and switch 2 also connect to the wire which connected to VCC. Then 2 input of AND gate we connect to the Switch 1 & Switch 2 then output of AND gate connected to the LED and last ground the circuit. After complete this circuit connection we run the circuit and we get results. Example mention in below table.

Truth Table:-

Input (A)	**Input (B)**	**Output (Z)**
0 (LOW)	0 (LOW)	0 (LOW)
0 (LOW)	1 (HIGH)	0 (LOW)
1 (HIGH)	0 (LOW)	0 (LOW)
1 (HIGH)	1 (HIGH)	1 (HIGH)

Table: 3.1

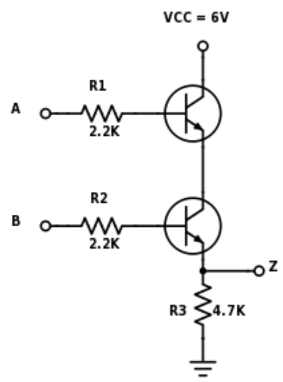
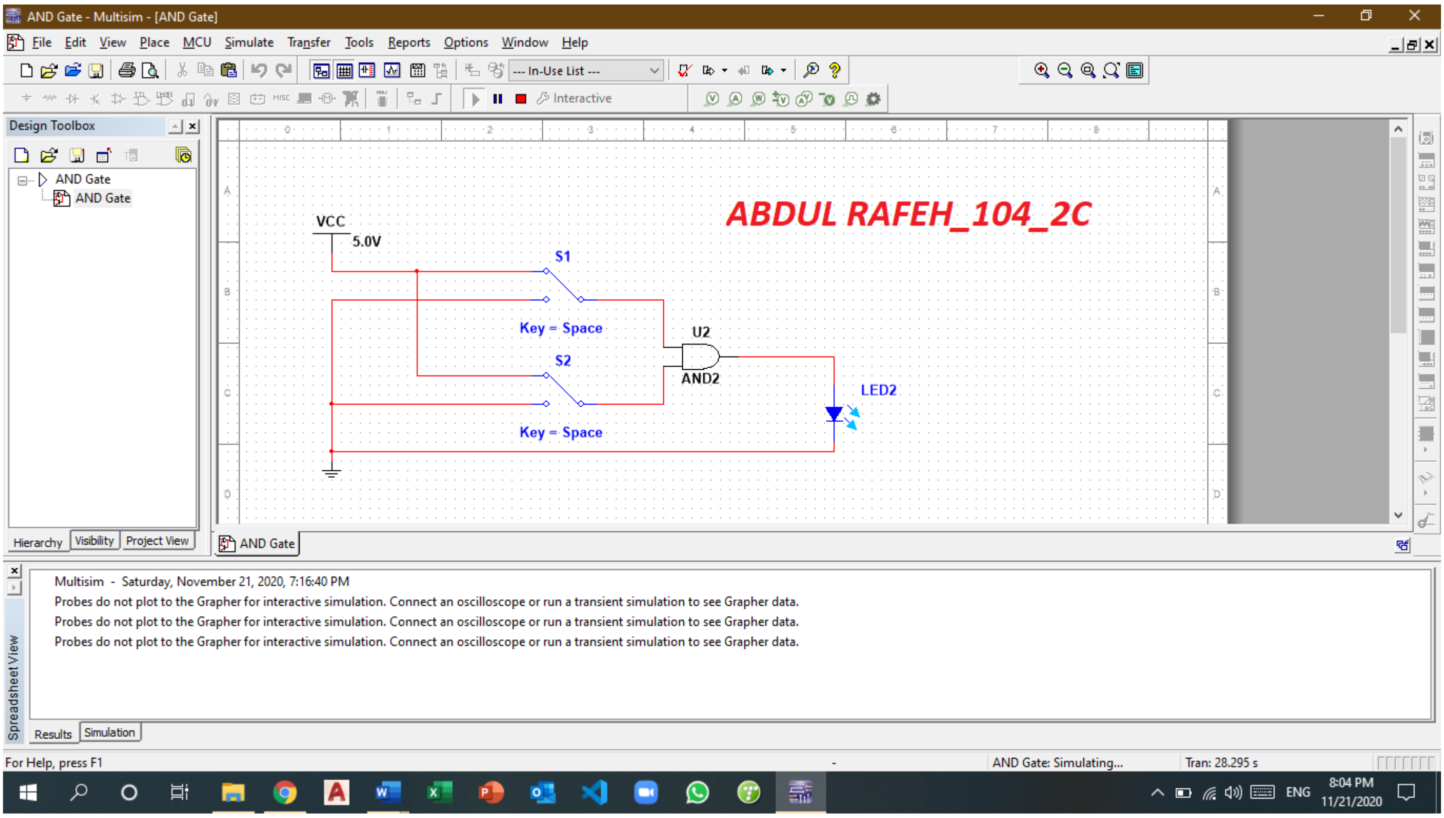


Fig: 3.3 AND Gate with discrete Components

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of AND gate.

Diagram:-



Answer the following questions:-

Q # 1. What do you understand by “&” character mentioned with AND gate symbol?

Answer:

There are three symbols for AND gates: the American (ANSI or 'military') symbol and the IEC ('European' or 'rectangular') symbol, as well as the deprecated DIN symbol. Additional inputs can be added as needed. It can also be denoted as symbol "^" or "&".

The AND gate with inputs A and B and output C implements the logical expression C=A·B This expression also may be denoted as **C=A^B** or **C=A&B**.

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 6** Student's Name:

Objective: To familiarize with basic logic OR gate.

Goal: In this experiment students will implement basic Logic OR Gate practically and will observe its function.

Required Tools/Equipment:-

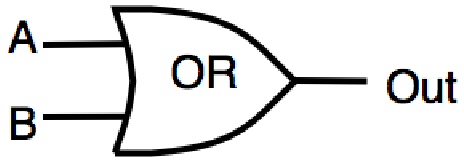
8) Laptop / Computer
9) NI Multisim Circuit Software
10) SPDT Switches
11) VCC 5.0V
12) Led red
13) OR gate
14) Ground

Theory:-

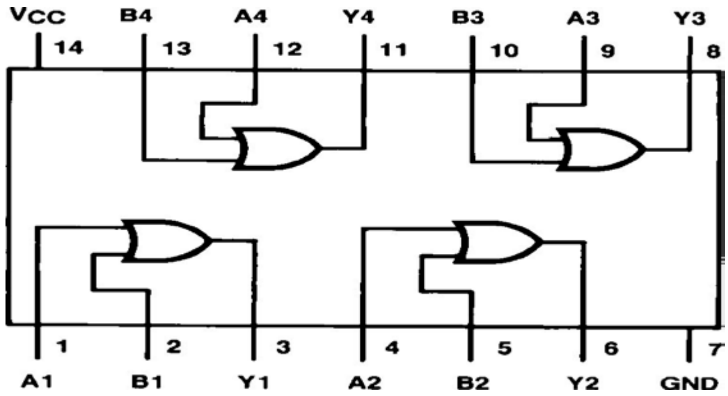
OR Gate:-

OR Gate provides binary 0 (LOW) only when binary 0 (LOW) is applied to all its input. In other words OR Gate provides binary 1 (HIGH) whenever binary 1 (HIGH) is applied to any of its input. If A and B are two inputs to OR Gate and Z is its output then $Z = A + B$. Plus sign '+' is used to indicate OR operation between two or more than two logic inputs. It is **never** omitted.

Symbol:-



Pin Configuration:-



Procedure:-

1) Run Multisim software.
2) Go to the menu bar.
3) In menu bar > Place > Components.
4) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
5) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
6) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
7) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
8) We are using two SPDT switches.
9) OR gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > OR2 then click on ok and place it.
10) Last equipment we use in this circuit is LED for getting result of this circuit the led location is Place > Components > Master Database > Group Diodes > LED > LED_red then click on ok and place it.
11) In upward mention 10Points we discuss how we select equipment for using in this circuit through multisim software.

Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 2 connect to the VCC and switch 1 also connect to the wire which connected to VCC. Then 2 input of OR gate we connect to the Switch 1 & Switch 2 then output of OR gate connected to the LED and last ground the circuit. After complete this circuit connection we run the circuit and we get results. Example mention in below table.

Input (A)	**Input (B)**	**Output (Z)**
0 (LOW)	0 (LOW)	0 (LOW)
0 (LOW)	1 (HIGH)	1 (HIGH)

1 (HIGH)	0 (LOW)	1 (HIGH)
1 (HIGH)	1 (HIGH)	1 (HIGH)

Table: 4.1

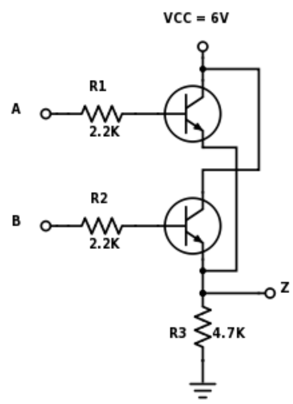
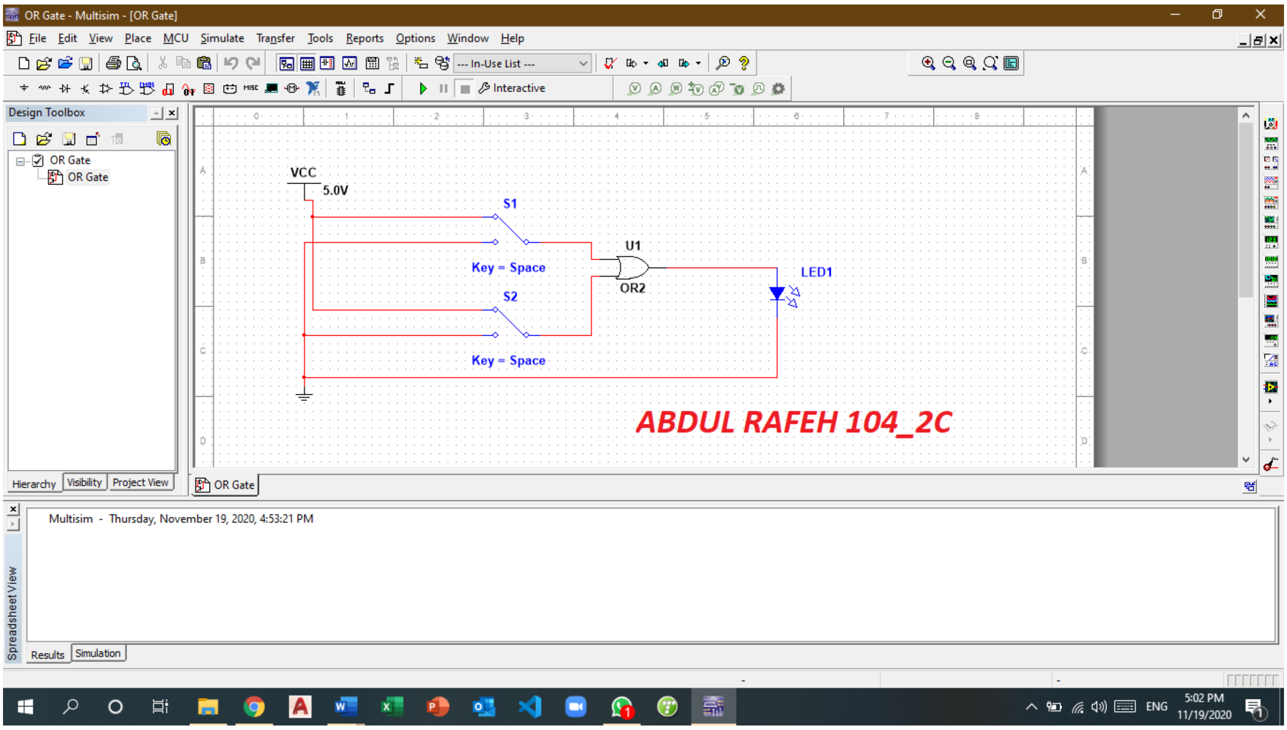


Fig: 4.3 OR Gate with discrete Components

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of OR gate.

Diagram:-



Answer the following questions:-

Q # 1. What do you understand by " ≥ 1 " character mentioned with OR gate symbol?

Answer:

OR gate or function. The symbol was chosen to indicate that at least one active input is needed to activate the output.

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 7** Student's Name:

Objective: To familiarize with basic logic NOR gate.

Goal: In this experiment students will implement basic Logic NOR Gate practically and will observe its function.

Required Tools/Equipment:-

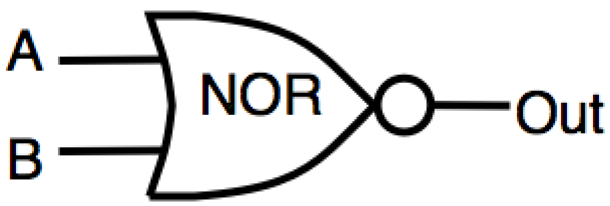
1) Laptop / Computer
2) NI Multisim Circuit Software
3) SPDT Switches
4) VCC 5.0V
5) Resistor 1kΩ
6) Transistor 2N2712 as per NI Multisim software component.
7) Probe blue
8) Ground

Theory:-

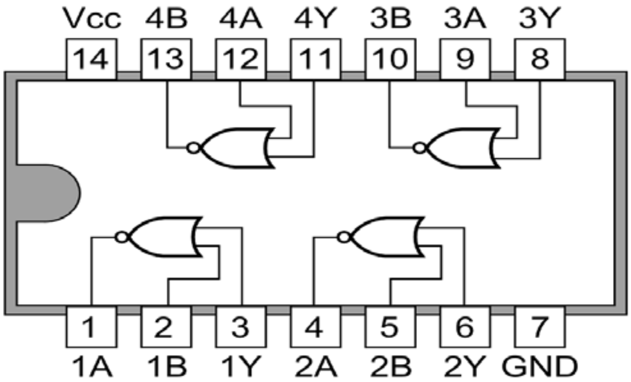
NOR:-

NOR Gate provides binary 1 (HIGH) only when binary 0 (LOW) is applied to all its input. In other words, NOR Gate provides binary 0 (LOW) whenever binary 1 (HIGH) is applied to any of its input. This Logic Gate can also be implemented with OR gate and NOT gate by connecting OR gate output to NOT gate input. In this condition, Inputs will be applied to AND gate input pins and output will be taken at NOT gate output pin. If A and B are two inputs to NOR Gate and Z is its output then $Z = (A + B)'$.

Symbol:-



Pin Configuration:-



Procedure:-

1) Run Multisim software.
2) Go to the menu bar.
3) In menu bar > Place > Components.
4) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
5) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
6) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
7) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
8) We are using two SPDT switches.
9) NOR gate using discrete components location Multisim software is Place > Components > Master Database > Group Basic > Resistor > 1kΩ then click on ok and place it.
10) We are using two transistor 2N2712 as per Multisim software coding location is Place > Components > Master Database > Group Transistor > BJT_NPN > 2N2712 then click on ok and place it.
11) Last equipment we use in this circuit is Probe bulb for getting result of this circuit the probe location is Place > Components > Master Database > Group Indicators > PROBE > PROBE_BLUE then click on ok and place it.
12) In upward mention 11Points we discuss how we select equipment for using in this NOR gate discrete circuit components through multisim software.

Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 1 & 2 connect to the VCC and Resistor 1kΩ one side connect to the VCC and other side of resistor we connected to the transistor. Then 2 transistors first we connected both together then switch 1 connect to the transistor 1 and same we did for switch 2 after that we put one probe and connect to the other side of resistor wire which connected to the transistors and last ground the circuit. After complete this circuit connection we run the circuit and we get results. Example mentions in below table.

Input (A)	Input (B)	Output (Z)
0 (LOW)	0 (LOW)	1 (HIGH)
0 (LOW)	1 (HIGH)	0 (LOW)
1 (HIGH)	0 (LOW)	0 (LOW)
1 (HIGH)	1 (HIGH)	0 (LOW)

Table: 5.1

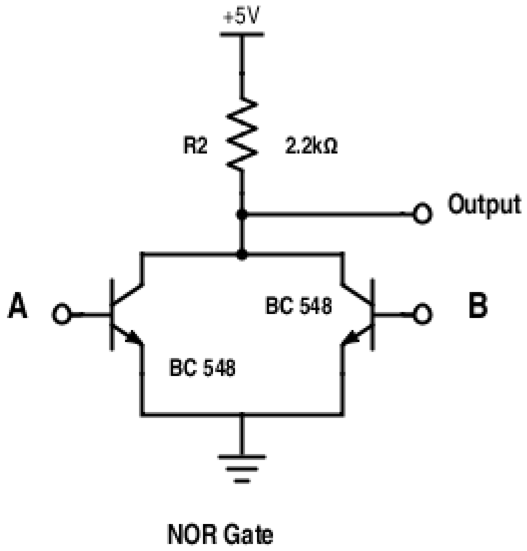
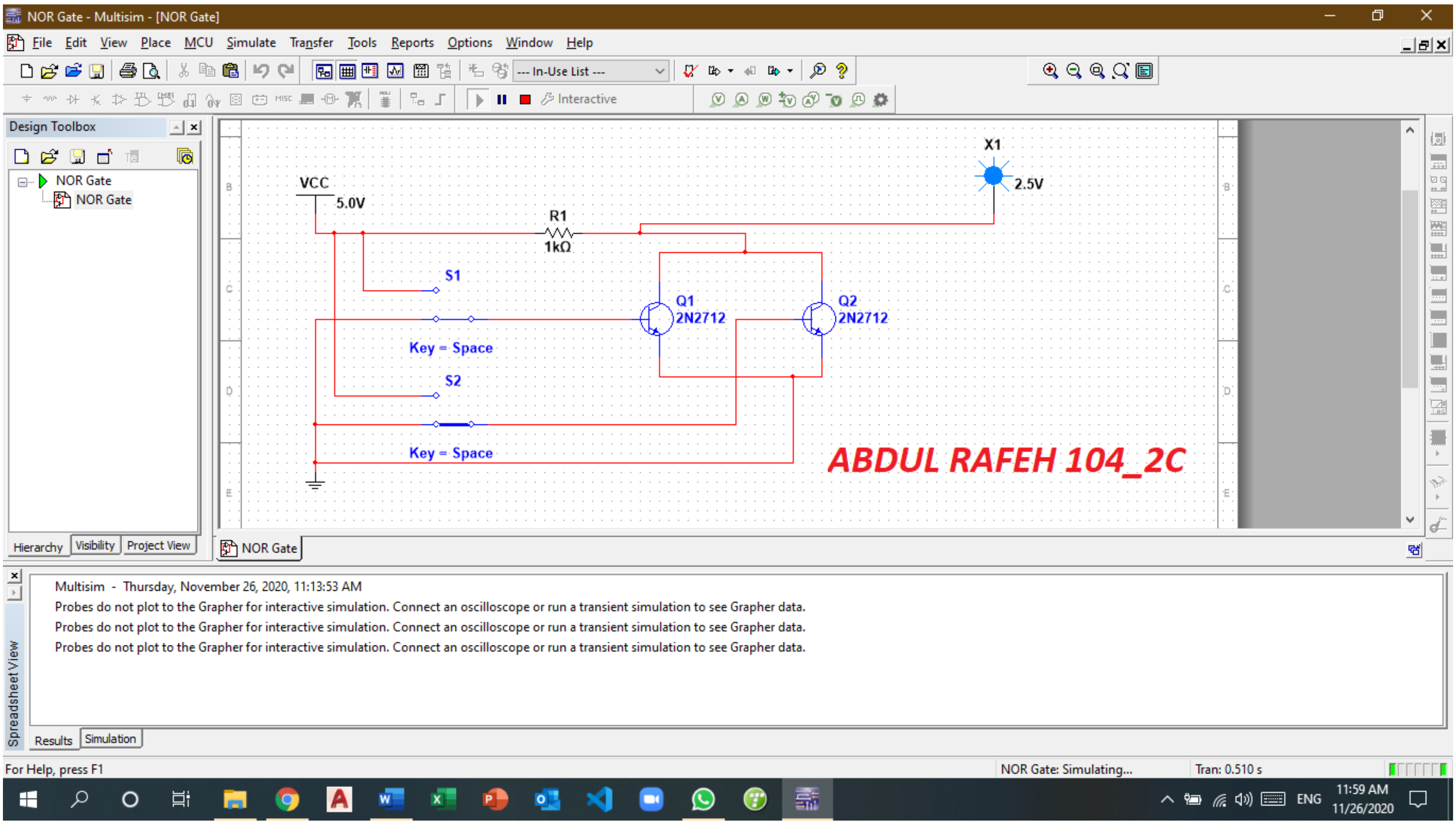


Fig: 5.3 NOR Gate with discrete Components

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of NOR gate.

Diagram:-



Answer the following questions:-

Q # 1. Describe working principle of NOR logic gate in words?

Answer:

A NOR gate (“not OR gate”) is a logic gate that produces a high output (1) only if all its inputs are false, and low output (0) otherwise. Hence the NOR gate is the inverse of an OR gate, and its circuit is produced by connecting an OR gate to a NOT gate. Just like an OR gate, a NOR gate may have any number of input probes but only one output probe. A NOR gate is also referred to as a universal gate. Because all binary operations can be realized by only using NOR gates. As we know that there are only three basic operations AND, OR, and NOT. Also, we know that all complex binary operations can be realized by using these three basic operations. If we can prove that AND, OR, and NOT operations can be realized by using only the NOR gate, then we can easily say that all binary operations can be realized by using only NOR gates.

CSC20S104 **AB.RAFEH**

Student's ID: # Laboratory Exercise No: 8 Student's Name:

Objective: To familiarize with basic logic NAND gate.

Goal: In this experiment students will implement basic Logic NAND Gate practically and will observe its function.

Required Tools/Equipment:-

1) Laptop / Computer
2) NI Multisim Circuit Software
3) SPDT Switches
4) VCC 5.0V
5) Resistor 1kΩ
6) Transistor 2N2712 as per NI Multisim software component.
7) Probe blue
8) Ground

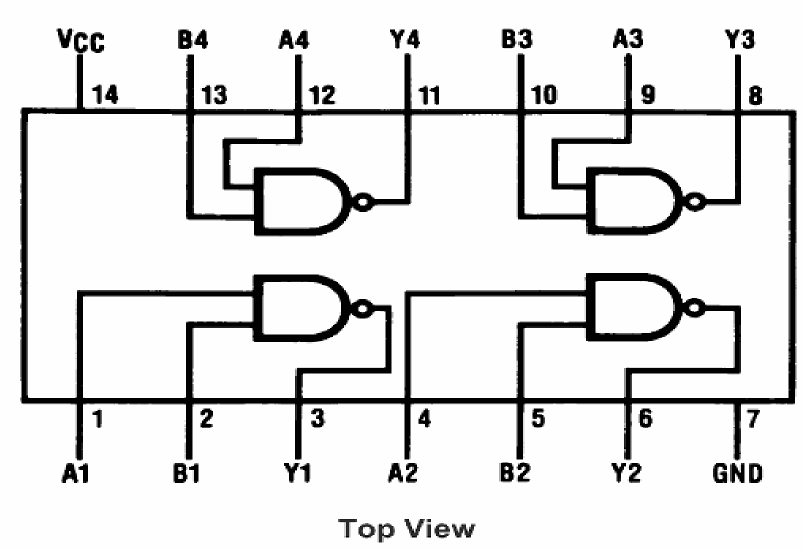
Theory:-

NAND:-

NAND Gate provides binary 0 (LOW) only when binary 1 (HIGH) is applied to all its inputs. In other words NAND Gate provides binary 1 (HIGH) if binary 0 (LOW) is applied to any of its inputs. This Logic Gate can be implemented with AND gate and NOT gate by connecting AND gate output to NOT gate input. In this condition, Inputs will be applied to AND gate input pins and output will be taken at NOT gate output pin. If A and B are two inputs to NAND Gate and Z is its output then $Z = (A.B)'$.

Symbol:-

Pin Configuration:-



Procedure:-

1) Run Multisim software.
2) Go to the menu bar.
3) In menu bar > Place > Components.
4) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
5) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
6) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
7) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
8) We are using two SPDT switches.
9) NAND gate using discrete components, we are using total 3 Resistors 1kΩ location Multisim software is Place > Components > Master Database > Group Basic > Resistor > 1kΩ then click on ok and place it.
10) We are using two transistor 2N2712 as per Multisim software coding location is Place > Components > Master Database > Group Transistor > BJT_NPN > 2N2712 then click on ok and place it.
11) Last equipment we use in this circuit is Probe bulb for getting result of this circuit the probe location is Place > Components > Master Database > Group Indicators > PROBE > PROBE_BLUE then click on ok and place it.
12) In upward mention 11Points we discuss how we select equipment for using in this NAND gate discrete circuit components through multisim software.

Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 1 & 2 connect to the VCC and Resistor 1kΩ one side connect to the VCC and other side of resistor we connected to the transistor. Then 2 transistors first we connected both together then we put 2 resistors between two switches & transistors then connected all of them after that we put one probe and connect to the other side of resistor wire which connected to the transistors and last ground the circuit. After complete this circuit connection we run the circuit and we get results. Example mentions in below table.

Input (A)	**Input (B)**	**Output (Z)**
0 (LOW)	0 (LOW)	1 (HIGH)
0 (LOW)	1 (HIGH)	1 (HIGH)
1 (HIGH)	0 (LOW)	1 (HIGH)
1 (HIGH)	1 (HIGH)	0 (LOW)

Table: 6.1

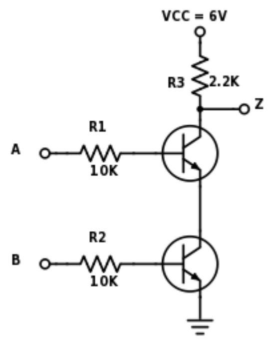
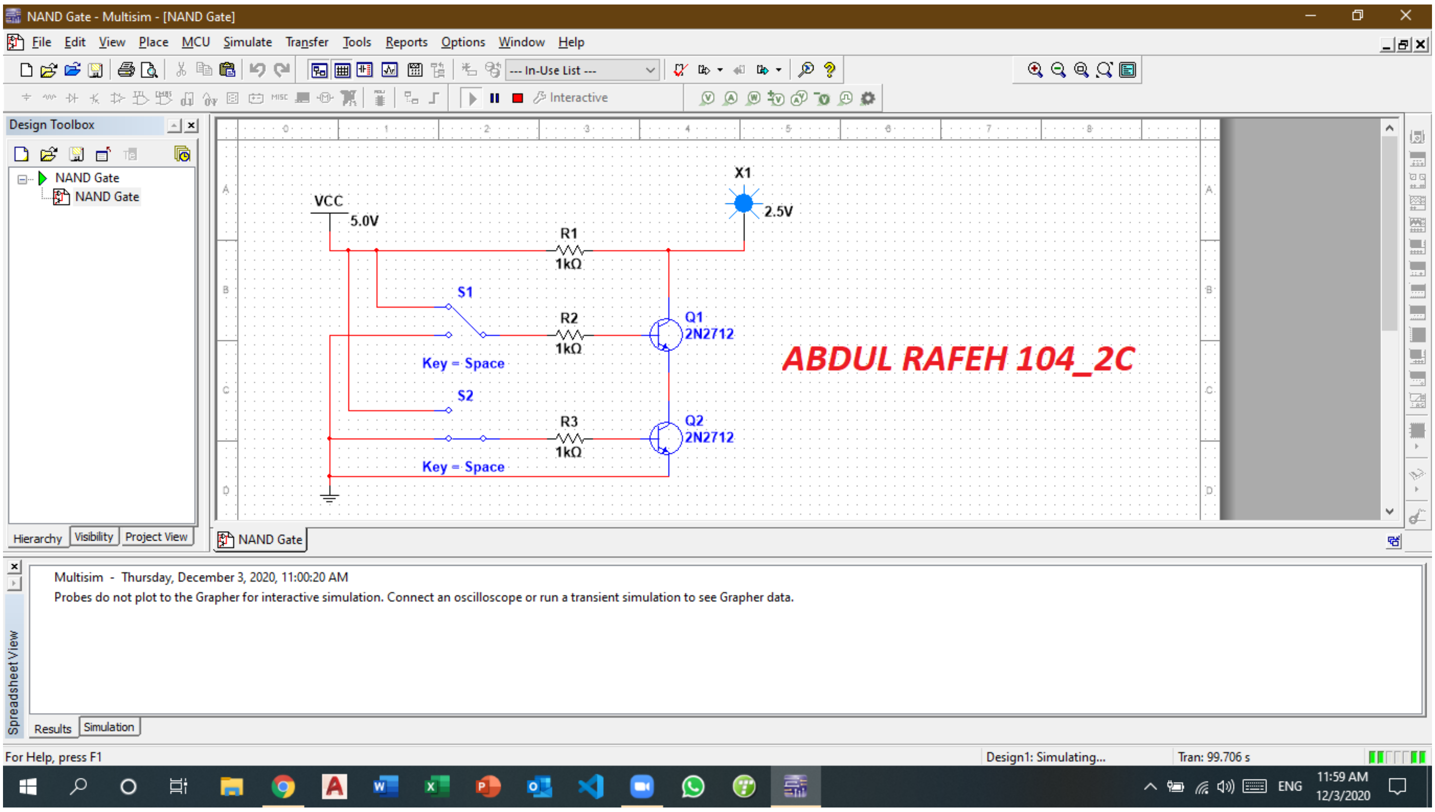


Fig: 6.3 NAND Gate with discrete Components

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of NAND gate.

Diagram:-



Answer the following questions:-

Q # 1. How many logic gates are integrated in 74HC00?

Answer:

The IC has four NAND gates in it, each gate can be used separately.

CSC20S104 **AB.RAFEH**

Student's ID: # Laboratory Exercise No: 9 Student's Name:

Objective: To familiarize with basic logic XOR gate.

Goal: In this experiment students will implement basic Logic XOR Gate practically and will observe its function.

Required Tools/Equipment:-

1) Laptop / Computer
2) NI Multisim Circuit Software
3) SPDT Switches
4) VCC 5.0V
5) Led blue
6) XOR gate
7) Ground

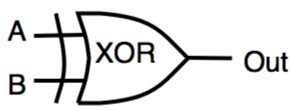
Theory:-

XOR:-

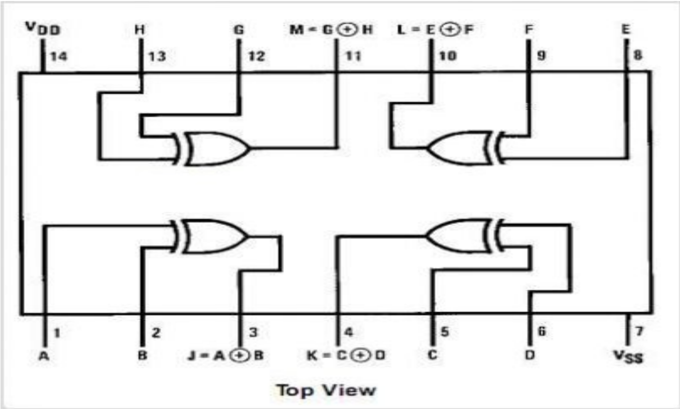
XOR Gate provides binary 1 (HIGH) only when not same logic levels are applied to its input (i-e 0 (LOW) is applied to one input and 1 (HIGH) is applied to second input). In other words, XOR Gate provides binary 0 (LOW) whenever same logic levels are applied to its input (i-e 0 (LOW) or 1 (HIGH) is applied to both inputs). This Logic Gate can also be implemented with one AND, NAND and OR gate by connecting one input pin of OR gate and NAND gate to one common junction, this junction is one input of XOR gate. By connecting second input pin of OR gate and NAND gate to second common junction, this is second input of XOR gate. Connecting these OR gate and NAND outputs to AND gate inputs, Output of this AND gate will provide XOR output. If A and B are two inputs to XOR Gate and Z is its output then $Z = A \oplus B = A'.B + A.B'$.

$\oplus$ Sign express Exclusive OR function between A and B.

Symbol:-



Pin Configuration:-



Procedure:-

1) Run Multisim software.
2) Go to the menu bar.
3) In menu bar > Place > Components.
4) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
5) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
6) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
7) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
8) We are using two SPDT switches.
9) XOR gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > XOR2 then click on ok and place it.
10) Last equipment we use in this circuit is LED for getting result of this circuit the led location is Place > Components > Master Database > Group Diodes > LED > LED_blue then click on ok and place it.
11) In upward mention 10Points we discuss how we select equipment for using in this circuit through multisim software.

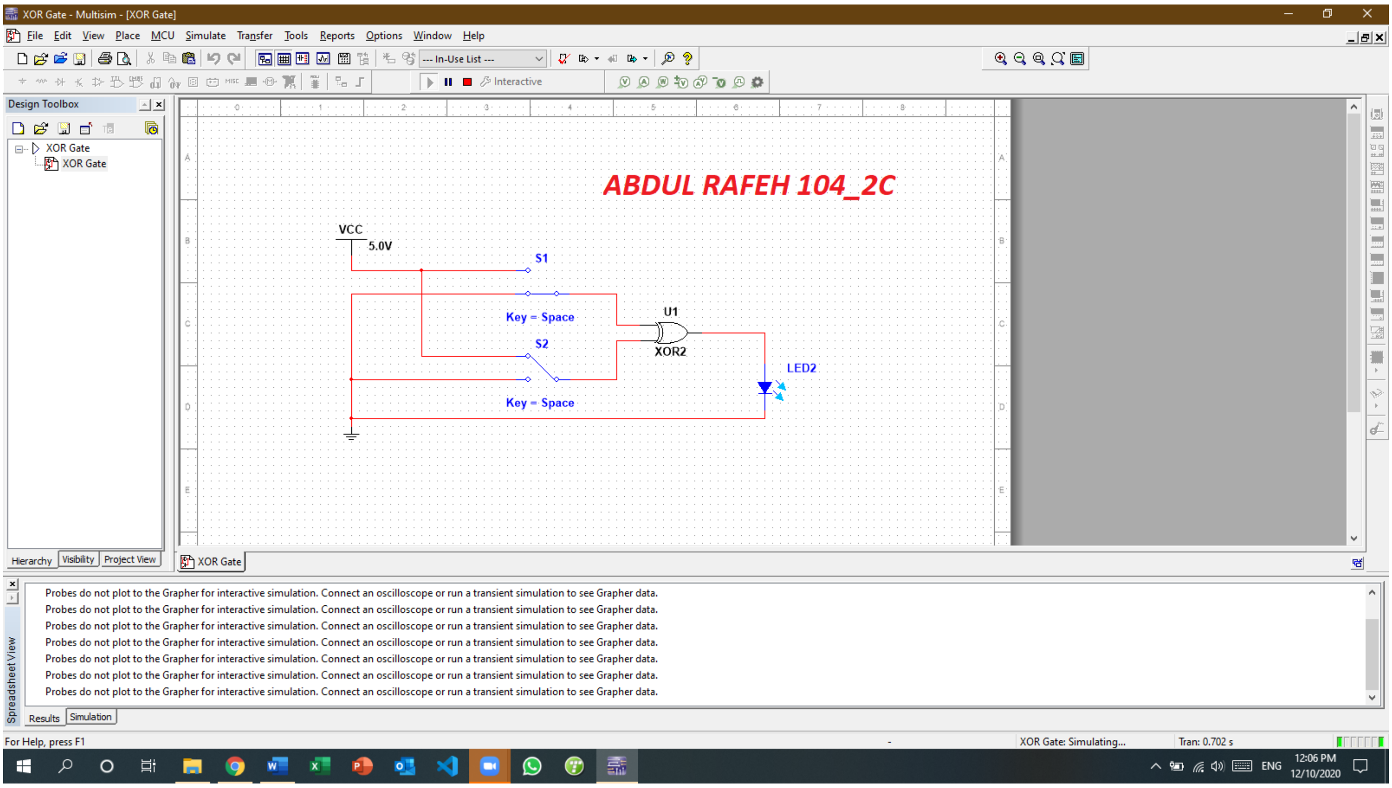
Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 1 connect to the VCC and switch 2 also connect to the wire which connected to VCC. Then 2 input of XOR gate we connect to the Switch 1 & Switch 2 then output of XOR gate connected to the LED and last ground the circuit. After complete this circuit connection we run the circuit and we get result mention in below truth table.

Input (A)	**Input (B)**	**Output (Z)**
0 (LOW)	0 (LOW)	0 (LOW)
0 (LOW)	1 (HIGH)	1 (HIGH)
1 (HIGH)	0 (LOW)	1 (HIGH)
1 (HIGH)	1 (HIGH)	0 (LOW)

Table: 7.1

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of XOR gate.

Diagram:-



Answer the following questions:-

Q # 1. How many inputs are available to each gate of XOR I.C?

Answer:

An XOR gate is a digital logic gate with two or more inputs and one output that performs exclusive disjunction. The output of an XOR gate is true only when exactly one of its inputs is true. Because gates are manufactured in IC form, typically containing two to six gates of the same type, it is often uneconomical to use a complete IC of six gates to perform a particular logic function. A better solution may be to use just a single type of gate to perform any of the logic operations required.

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 10** Student's Name:

Objective: To familiarize with basic logic XNOR gate.

Goal: In this experiment students will implement basic Logic XNOR Gate practically and will observe its function.

Required Tools/Equipment:-

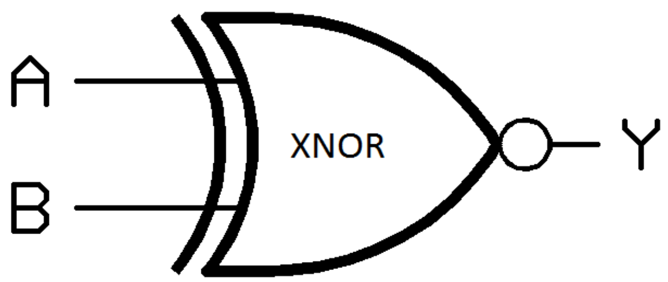
9) Laptop / Computer
10) NI Multisim Circuit Software
11) SPDT Switches
12) VCC 5.0V
13) Led blue
14) XNOR gate
15) Ground

Theory:-

XNOR:

XNOR Gate provides binary 0 (LOW) only when not same logic levels are applied to its input (i-e 0 (LOW) is applied to one input and 1 (HIGH) is applied to second input). In other words, XNOR Gate provides binary 1 (HIGH) whenever same logic levels are applied to its input (i-e 0 (LOW) or 1 (HIGH) is applied to both inputs). This Logic Gate can be implemented with XOR gate and NOT gate by connecting XOR gate output to NOT gate input. In this condition, Inputs will be applied to XOR gate input pins and output will be taken at NOT gate output pin. If A and B are two inputs to XNOR Gate and Z is its output then $Z = (A \oplus B)' = A.B + A'.B'$.

Symbol:-



Procedure:-

1) Run Multisim software.
2) Go to the menu bar.
3) In menu bar > Place > Components.
4) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
5) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
6) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
7) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
8) We are using two SPDT switches.
9) XNOR gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > XNOR2 then click on ok and place it.
10) Last equipment we use in this circuit is LED for getting result of this circuit the led location is Place > Components > Master Database > Group Diodes > LED > LED_blue then click on ok and place it.
11) In upward mention 10Points we discuss how we select equipment for using in this circuit through multisim software.

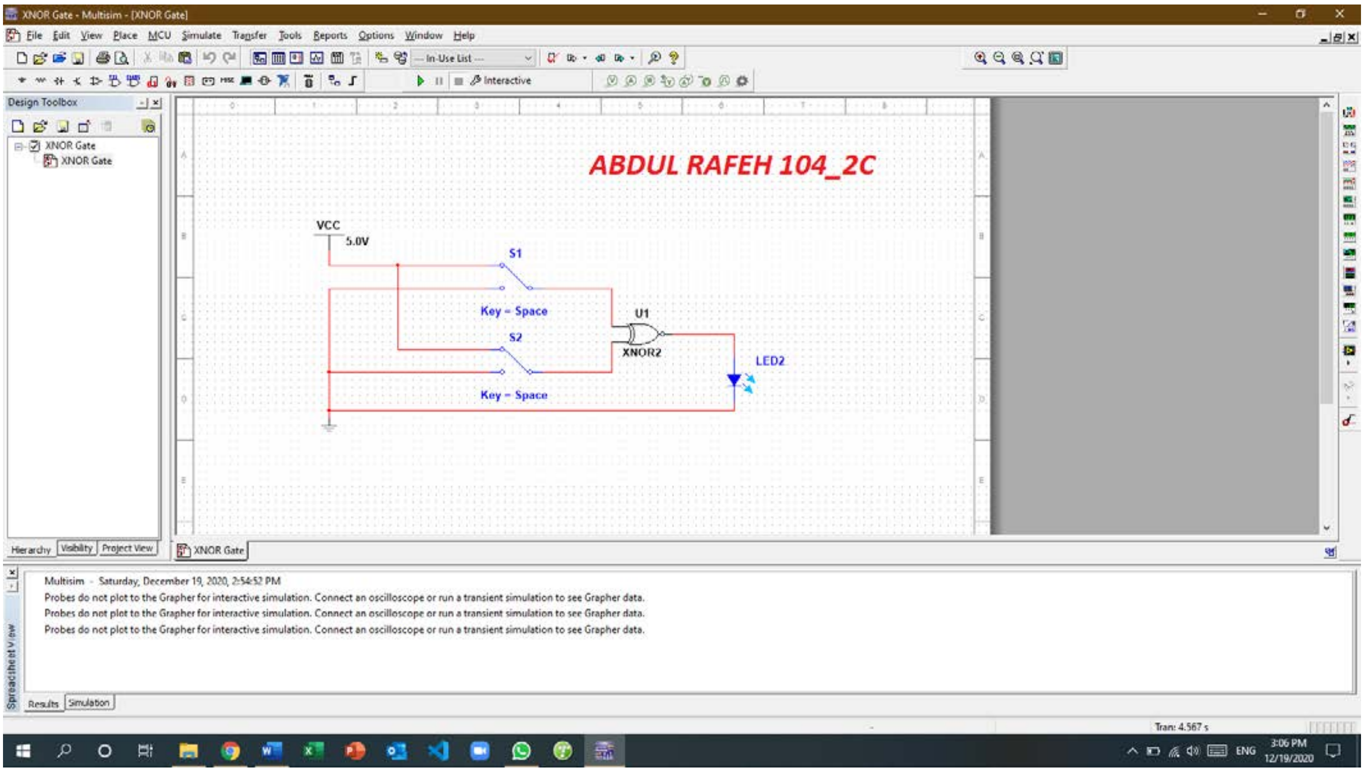
Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 1 connect to the VCC and switch 2 also connect to the wire which connected to VCC. Then 2 input of XNOR gate we connect to the Switch 1 & Switch 2 then output of XNOR gate connected to the LED and last ground the circuit. After complete this circuit connection we run the circuit and we get result mention in below truth table.

Input (A)	**Input (B)**	**Output (Z)**
0 (LOW)	0 (LOW)	1 (HIGH)
0 (LOW)	1 (HIGH)	0 (LOW)
1 (HIGH)	0 (LOW)	0 (LOW)
1 (HIGH)	1 (HIGH)	1 (HIGH)

Table: 8.1

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of XNOR gate.

Diagram:-



CSC20S104 **AB.RAFEH**

Student's ID: # Laboratory Exercise No: 11 Student's Name:

Objective: To construct XOR gate using discrete components with transistor.

Goal: In this experiment students will implement XOR Logic Gate practically using discrete components and will observe its function.

Required Tools/Equipment:-

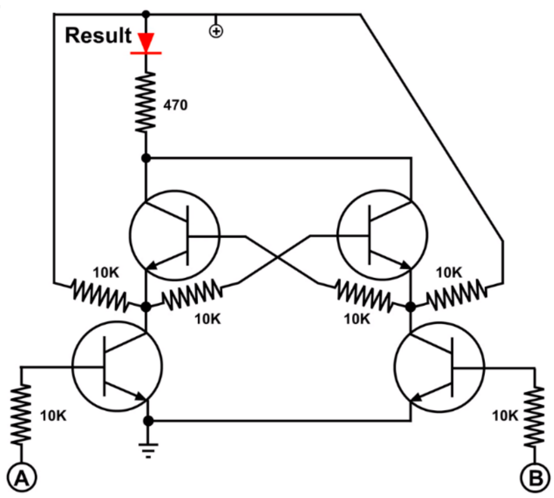
1) Laptop / Computer
2) NI Multisim Circuit Software
3) SPDT Switches
4) VCC 5.0V
5) Resistors 1kΩ
6) Transistors
7) Led_red
8) Ground

Theory:-

When the voltages at terminals A and B are at opposite logic states forward biases the Emitter-Base junction and turns ON the transistor. Thus an approximate Logic HIGH voltage VH−0.6V−VCE is available at output terminal Z. The Logic LOW voltage is approximately 0V but the sink current limited by the collector resistance 10K ohm.

As the Logic HIGH input current for TTL is approximately 0.4mA, which is the transistors emitter current and would generate a voltage drop of approximately 4V across 10K ohm resistor. But the problem is the 10K ohm resistor cannot provide the required sink current 0.4mA when the output Z is at Logic ZERO. Thus this XOR configuration seems to be suitable only for CMOS or TTL inputs at A and B and capable of driving only CMOS at output Y.

Schematic:-



Procedure:-

13) Run Multisim software.
14) Go to the menu bar.
15) In menu bar > Place > Components.
16) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
17) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
18) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
19) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
20) We are using two SPDT switches.
21) XOR gate using discrete components, we are using total 7 Resistors 1kΩ location Multisim software is Place > Components > Master Database > Group Basic > Resistor > 1kΩ then click on ok and place it.
22) We are using four transistor location is Place > Components > Master Database > Transistor > BJT_NPN > 2N2712 then click on ok and place it.
23) Last equipment we use in this circuit is Probe bulb for getting result of this circuit the probe location is Place > Components > Master Database > Diodes > LED > Led_red then click on ok and place it.
24) In upward mention 10Points we discuss how we select equipment for using in this XOR gate discrete circuit components through multisim software.

Now in this stage we discuss how we connected the circuit. For complete connection detail see in the SS and also schematic diagram

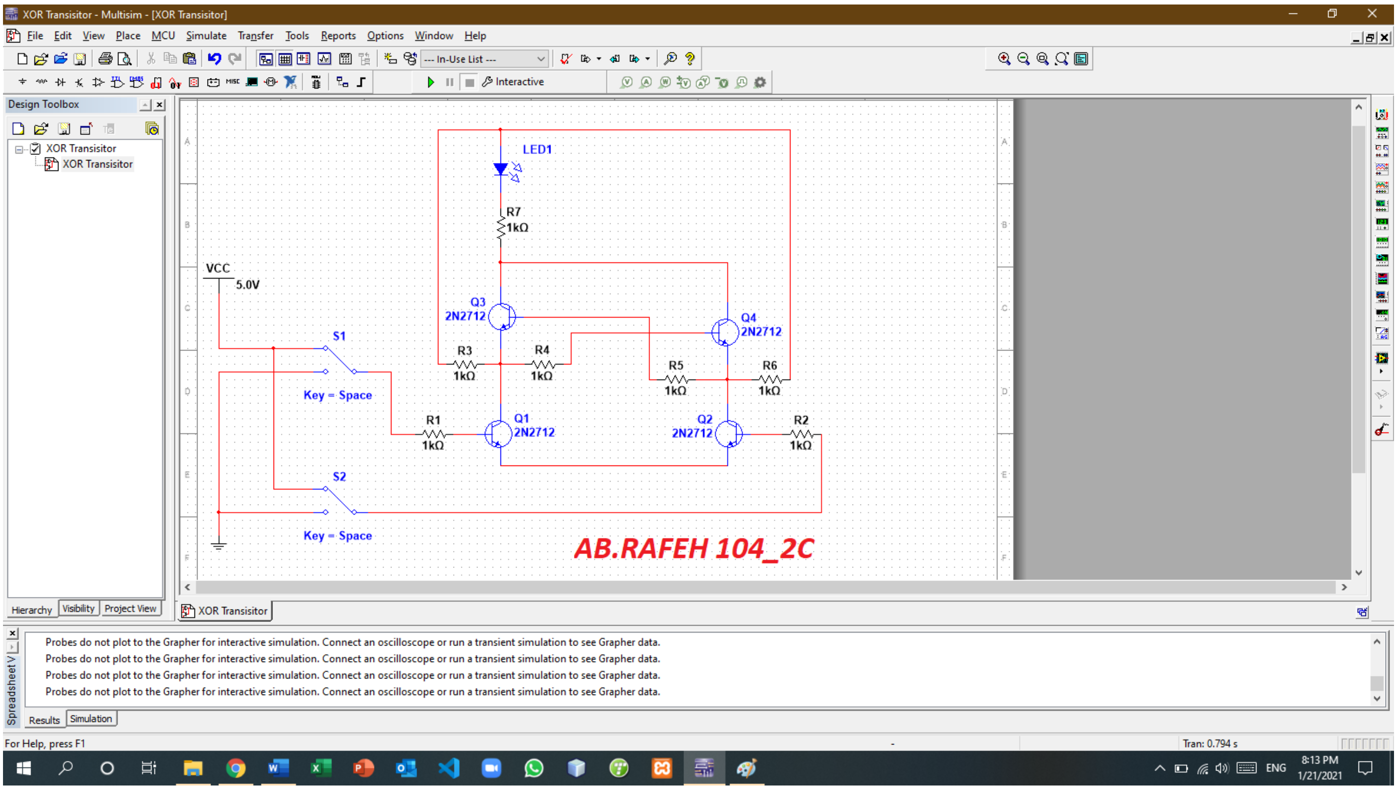
. After complete this circuit connection we run the circuit and we get results. Example mentions in below table.

Input (A)	**Input (B)**	**Output (Z)**
0 (LOW)	0 (LOW)	0 (LOW)
0 (LOW)	1 (HIGH)	1 (HIGH)
1 (HIGH)	0 (LOW)	1 (HIGH)
1 (HIGH)	1 (HIGH)	0 (LOW)

Table: 9.1

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of XOR gate using transistor discrete components.

Diagram:-



Answer the following questions:-

Q # 1. What are TTL and CMOS voltage level?

Answer:

Unlike TTL, which is restricted to a power supply voltage of 5 volts, CMOS may be powered by voltages as high as 15 volts (some CMOS circuits as high as 18 volts).

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 12** Student's Name:

Objective: To construct XOR gate using discrete components with diodes.

Goal: In this experiment students will implement XOR Logic Gate practically using discrete components and will observe its function.

Required Tools/Equipment:-

9) Laptop / Computer
10) NI Multisim Circuit Software
11) SPDT Switches
12) VCC 5.0V
13) Resistors 1kΩ
14) Transistors
15) Diodes
16) Probe Dig blue
17) Ground

Theory:-

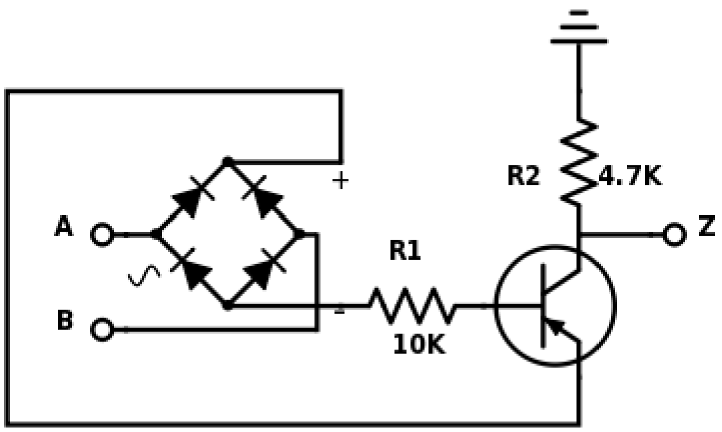


Fig: 9.1 XOR Gate with discrete Components

When the voltages at terminals A and B are at opposite logic states forward biases the Emitter-Base junction and turns ON the transistor. Thus an approximate Logic HIGH voltage VH−0.6V−VCE is available at output terminal Z. The Logic LOW voltage is approximately 0V but the sink current limited by the collector resistance 10K ohm.

As the Logic HIGH input current for TTL is approximately 0.4mA, which is the transistors emitter current and would generate a voltage drop of approximately 4V across 10K ohm resistor. But the problem is the 10K ohm resistor cannot provide the required sink current 0.4mA when the output Z is at Logic ZERO. Thus this XOR configuration seems to be suitable only for CMOS or TTL inputs at A and B and capable of driving only CMOS at output Y.

Procedure:-

25) Run Multisim software.
26) Go to the menu bar.
27) In menu bar > Place > Components.
28) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
29) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
30) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
31) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
32) We are using two SPDT switches.
33) XOR gate using discrete components, we are using total 4 Resistors 1kΩ location Multisim software is Place > Components > Master Database > Group Basic > Resistor > 1kΩ then click on ok and place it.
34) We are using two transistor location is Place > Components > Master Database > Group Power > SWITCHES > transistor then click on ok and place it.
35) We are using 4 Diodes location is Place > Components > Master Database > Group Power > SWITCHES > Diode then click on ok and place it.
36) Last equipment we use in this circuit is Probe bulb for getting result of this circuit the probe location is Place > Components > Master Database > Group Indicators > PROBE > PROBE_DIG_BLUE then click on ok and place it.
37) In upward mention 11Points we discuss how we select equipment for using in this XOR gate discrete circuit components through multisim software.

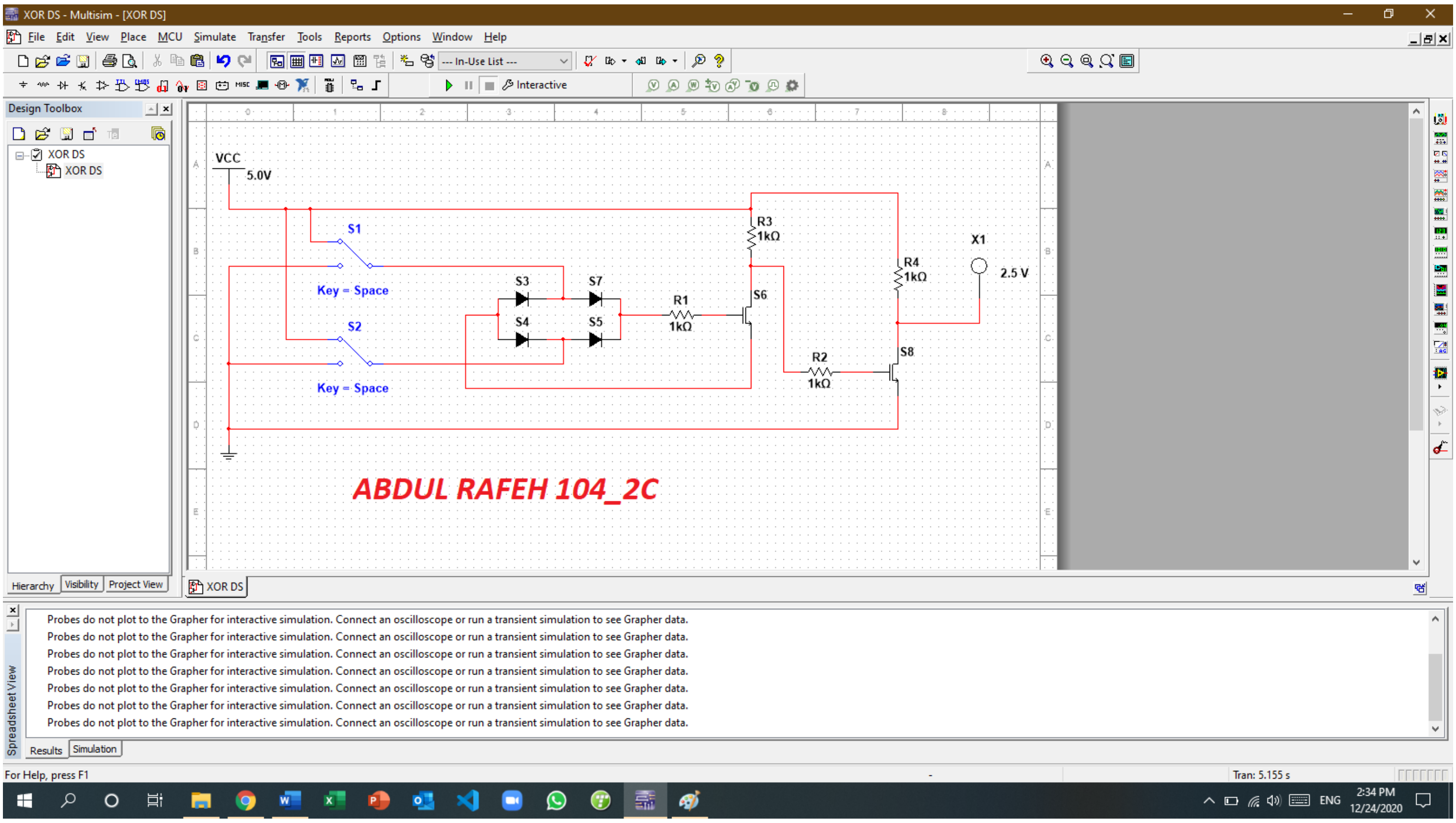
Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 1 & 2 connect to the VCC and Resistor3 1kΩ one side connect to the VCC and other side of resistor we connected to the transistor. Then first transistor we connect R1 & R3 and other side of transistor connect to the diodes and second transistor connect to the R2 & R4 and we use we use 4nos. diodes first we connect 4 together then connect to the switch 1 & 2 and R1 after that for getting result we take wire from transistor and R4 we get output for probe and last ground the circuit. After complete this circuit connection we run the circuit and we get results. Example mentions in below table.

Input (A)	**Input (B)**	**Output (Z)**
0 (LOW)	0 (LOW)	0 (LOW)
0 (LOW)	1 (HIGH)	1 (HIGH)
1 (HIGH)	0 (LOW)	1 (HIGH)
1 (HIGH)	1 (HIGH)	0 (LOW)

Table: 9.1

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of XOR gate using diodes & more discrete components.

Diagram:-



CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 13** Student's Name:

Objective: To construct XNOR gate using discrete components with transistor.

Goal: In this experiment students will implement XNOR Logic Gate practically using discrete components and will observe its function.

Required Tools/Equipment:-

1) Laptop / Computer
2) NI Multisim Circuit Software
3) SPDT Switches
4) VCC 5.0V
5) Resistors 1kΩ
6) Transistors
7) Probe Dig Orange
8) Ground

Theory:-

XNOR:

When the voltage at A and B terminals are at opposite logic state, a voltage of higher voltage minus lower voltage minus 1.2V (voltage drop between two diodes) forward bias the Emitter-Base junction of the Transistor. This turns ON the transistor and the Logic LOW voltage available at the collector of the transistor is approximately equal to 0.6+VL+VCE, where VL is the Logic LOW input and VCE is the Collector to Emitter voltage of the transistor. When the both inputs A and B are at the same Logic Levels, the Emitter to Base junction of the transistor cannot be forward biased, thus the transistor is in OFF state and the output Y is at supply voltage.

Procedure:-

38) Run Multisim software.
39) Go to the menu bar.
40) In menu bar > Place > Components.
41) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
42) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
43) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
44) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
45) We are using two SPDT switches.

46) XNOR gate using discrete components, we are using total 3 Resistors 1kΩ location Multisim software is Place > Components > Master Database > Group Basic > Resistor > 1kΩ then click on ok and place it.
47) We are using two transistor location is Place > Components > Master Database > Group Power > SWITCHES > transistor then click on ok and place it.
48) Last equipment we use in this circuit is Probe bulb for getting result of this circuit the probe location is Place > Components > Master Database > Group Indicators > PROBE > PROBE_DIG_ORANGE then click on ok and place it.
49) In upward mention 10Points we discuss how we select equipment for using in this XNOR gate discrete circuit components through multisim software.

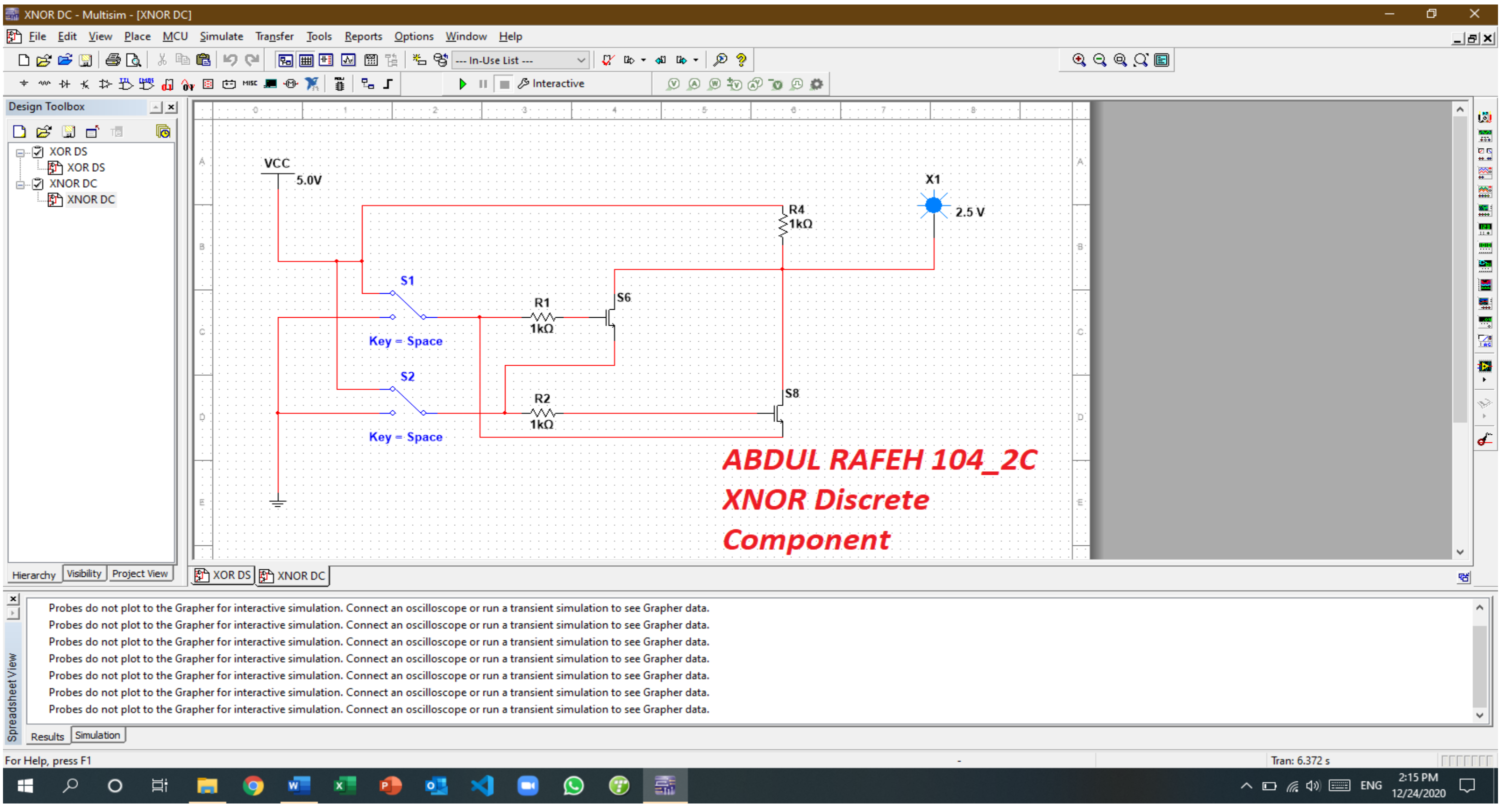
Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 1 & 2 connect to the VCC and Resistor3 1kΩ one side connect to the VCC and other side of resistor we connected to the transistor. For more connection see in the below SS after that for getting result we take wire from transistor and R3 we get output for probe and last ground the circuit. After complete this circuit connection we run the circuit and we get results. Example mentions in below table.

Input (A)	**Input (B)**	**Output (Z)**
0 (LOW)	0 (LOW)	1 (HIGH)
0 (LOW)	1 (HIGH)	0 (LOW)
1 (HIGH)	0 (LOW)	0 (LOW)
1 (HIGH)	1 (HIGH)	1 (HIGH)

Table: 10.1

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of XNOR gate using transistor discrete components.

Diagram:-


XNOR DC - Multisim - [XNOR DC]
VCC
5.0V
X1
2.5 V
R4
1kΩ
S1
R1
1kΩ
S6
Key = Space
S2
R2
1kΩ
S8
Key = Space
ABDUL RAFEH 104_2C
XNOR Discrete
Component
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
For Help, press F1
Tran: 6.372 s
2:15 PM
12/24/2020

CSC20S104 **AB.RAFEH**

Student's ID: **Laboratory Exercise No: 14** Student's Name:

Objective: To construct XNOR gate using discrete components with diodes.

Goal: In this experiment students will implement XNOR Logic Gate practically using discrete components and will observe its function.

Required Tools/Equipment:-

9) Laptop / Computer
10) NI Multisim Circuit Software
11) SPDT Switches
12) VCC 5.0V
13) Resistors 1kΩ
14) Transistors
15) Diodes
16) Probe Dig Orange
17) Ground

Theory:-
XNOR:

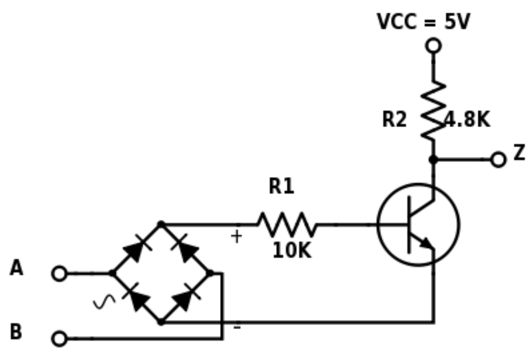


Fig: 10.1 XNOR Gate with discrete

When the voltage at A and B terminals are at opposite logic state, a voltage of higher voltage minus lower voltage minus 1.2V (voltage drop between two diodes) forward bias the Emitter-Base junction of the Transistor. This turns ON the transistor and the Logic LOW voltage available at the collector of the transistor is approximately equal to 0.6+VL+VCE, where VL is the Logic LOW input and VCE is the Collector to Emitter voltage of the transistor. When the both inputs A and B are at the same Logic Levels, the Emitter to Base junction of the transistor cannot be forward biased, thus the transistor is in OFF state and the output Y is at supply voltage.

Procedure:-

50) Run Multisim software.
51) Go to the menu bar.
52) In menu bar > Place > Components.
53) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
54) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
55) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
56) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
57) We are using two SPDT switches.
58) XNOR gate using discrete components, we are using total 2 Resistors 1kΩ location Multisim software is Place > Components > Master Database > Group Basic > Resistor > 1kΩ then click on ok and place it.
59) We are using one transistor location is Place > Components > Master Database > Group Power > SWITCHES > Transistor then click on ok and place it.
60) We are using 4 Diodes location is Place > Components > Master Database > Group Power > SWITCHES > Diode then click on ok and place it.
61) Last equipment we use in this circuit is Probe bulb for getting result of this circuit the probe location is Place > Components > Master Database > Group Indicators > PROBE > PROBE_DIG_ORANGE then click on ok and place it.
62) In upward mention 11Points we discuss how we select equipment for using in this XNOR gate discrete circuit components through multisim software.

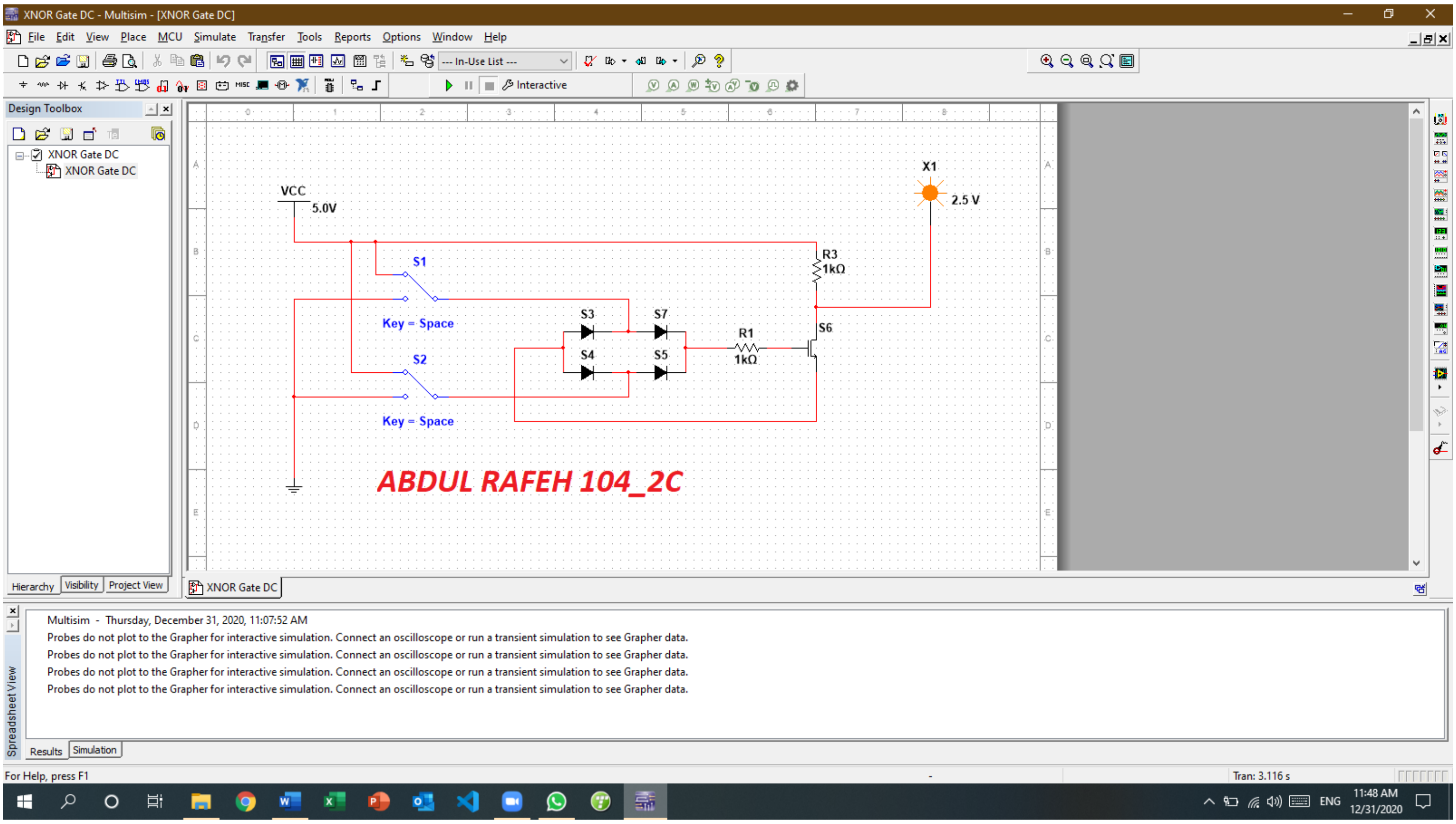
Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that switch 1 & 2 connect to the VCC and Resistor3 1kΩ one side connect to the VCC and other side of resistor we connected to the transistor. Then transistor we connect R1 & R3 and other side of transistor connect to the diodes and we use we use 4nos. diodes first we connect 4 together then connect to the switch 1 & 2 and R1 after that for getting result we take wire from transistor and R3 we get output for probe and last ground the circuit. After complete this circuit connection we run the circuit and we get results. Example mentions in below table.

Input (A)	Input (B)	Output (Z)
0 (LOW)	0 (LOW)	1 (HIGH)
0 (LOW)	1 (HIGH)	0 (LOW)
1 (HIGH)	0 (LOW)	0 (LOW)
1 (HIGH)	1 (HIGH)	1 (HIGH)

Table: 10.1

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of XNOR gate using diodes & more discrete components.

Diagram:-



CSC20S104 **AB.RAFEH**

Student's ID: # Laboratory Exercise No: 15 Student's Name:

Objective: To familiarize with half adder digital circuit.

Goal: In this experiment students will implement half adder digital circuit practically and will observe its function.

Required Tools/Equipment:-

8) Laptop / Computer
9) NI Multisim Circuit Software
10) SPDT Switches
11) VCC 5.0V
12) Led orange
13) Led red
14) XOR gate
15) AND gate
16) Ground

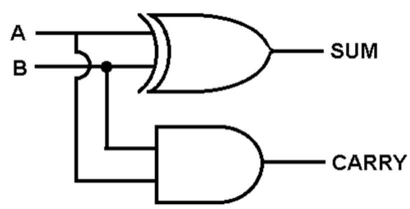
Theory:-

Half Adder Digital Circuit:-

Half Adder is a combinational logic circuit which is designed by connecting one EX-OR gate and one AND gate. The half adder circuit has two inputs: A and B, which add two input digits and generates a carry and a sum. The output obtained from the EX-OR gate is the sum of the two numbers while that obtained by AND gate is the carry. There will be no forwarding of carry addition because there is no logic gate to process that. Thus, this is called Half Adder circuit.

Logical Expression for half adder is:
$S=a\oplus b$; $C=a*b$.

Symbol:-



Procedure:-

12) Run Multisim software.
13) Go to the menu bar.
14) In menu bar > Place > Components.
15) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
16) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
17) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
18) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
19) We are using two SPDT switches.
20) XOR gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > XOR2 then click on ok and place it.
21) AND gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > AND2 then click on ok and place it.
22) Last equipment we use in this circuit is LED for getting result of this circuit the led location is Place > Components > Master Database > Group Diodes > LED > LED_orange & LED_red then click on ok and place it.
23) In upward mention 10Points we discuss how we select equipment for using in this circuit through multisim software.

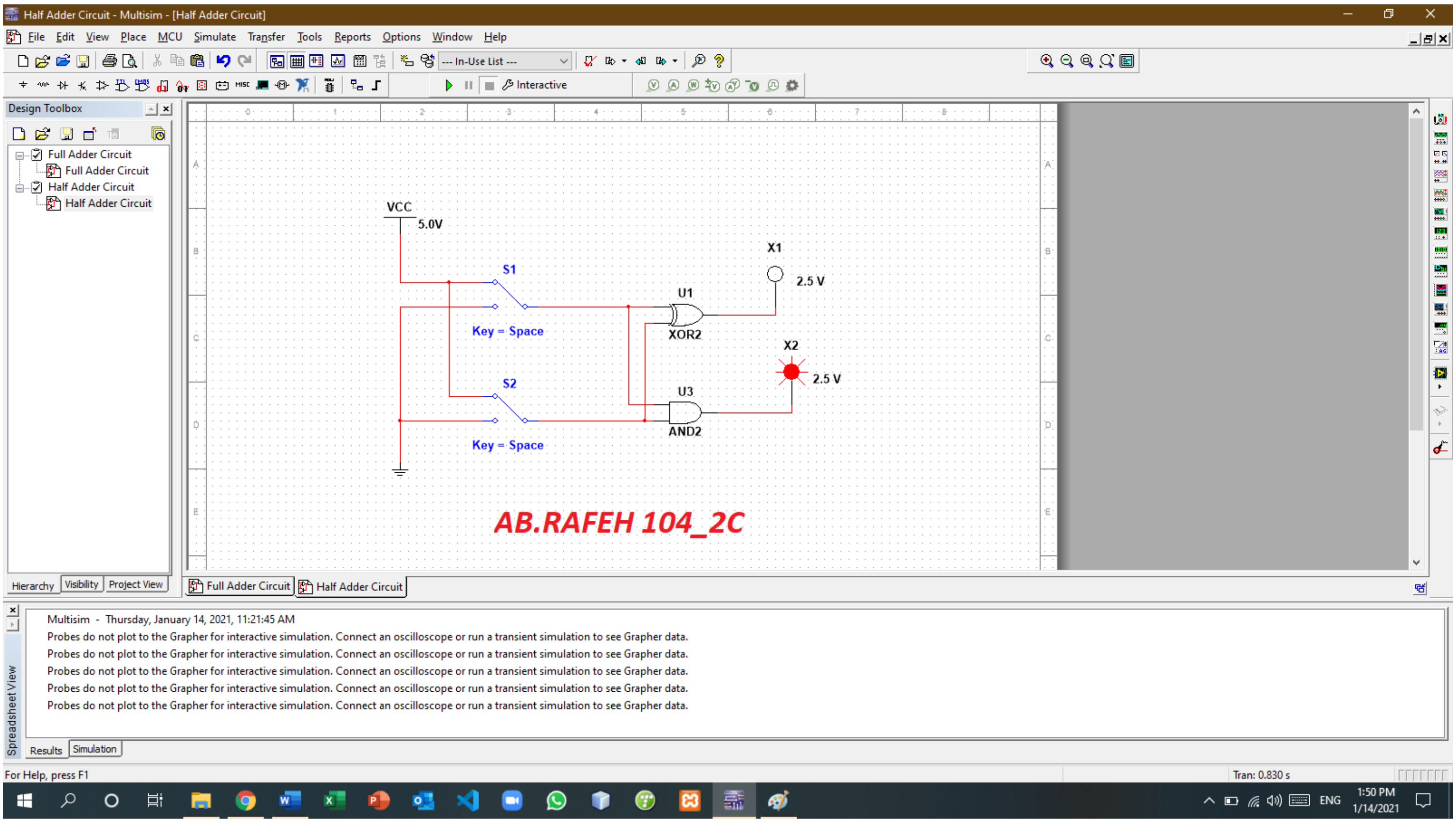
Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that we connected all the logic gates which using in this circuit you can see in the below attached diagram. After complete this circuit connection we run the circuit and we get result mention in below truth table.

Input (A)	**Input (B)**	**Output (Sum)**	**Output (Carry)**
0 (LOW)	0 (LOW)	0 (LOW)	0 (LOW)
0 (LOW)	1 (HIGH)	1 (HIGH)	0 (LOW)
1 (HIGH)	0 (LOW)	1 (HIGH)	0 (LOW)
1 (HIGH)	1 (HIGH)	0 (LOW)	1 (HIGH)

Table: 11.1

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of half adder digital circuit.

Diagram:-


Half Adder Circuit - Multisim - [Half Adder Circuit]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Design Toolbox
Full Adder Circuit
Full Adder Circuit
Half Adder Circuit
Half Adder Circuit
VCC
5.0V
S1
Key = Space
S2
Key = Space
U1
XOR2
U3
AND2
X1
2.5 V
X2
2.5 V
AB.RAFEH 104_2C
Hierarchy Visibility Project View
Full Adder Circuit Half Adder Circuit
Multisim - Thursday, January 14, 2021, 11:21:45 AM
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Probes do not plot to the Grapher for interactive simulation. Connect an oscilloscope or run a transient simulation to see Grapher data.
Results Simulation
For Help, press F1
Tran: 0.830 s
1:50 PM
1/14/2021

CSC20S104 **AB.RAFEH**

Student's ID: # Laboratory Exercise No: 16 Student's Name:

Objective: To familiarize with full adder digital circuit.

Goal: In this experiment students will implement full adder digital circuit practically and will observe its function.

Required Tools/Equipment:-

17) Laptop / Computer
18) NI Multisim Circuit Software
19) SPDT Switches
20) VCC 5.0V
21) Led orange
22) Led red
23) XOR gate
24) AND gate
25) OR gate
26) Ground

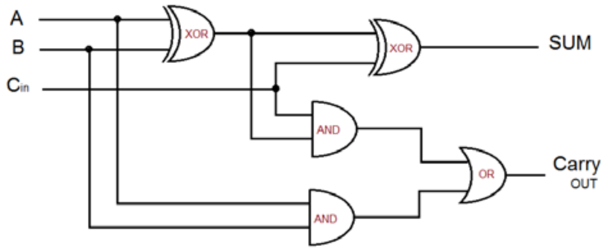
Theory:-

Full Adder Digital Circuit:-

Full Adder is the circuit which consists of two EX-OR gates, two AND gates and one OR gate. Full Adder is the adder which adds three inputs and produces two outputs which consists of two EX-OR gates, two AND gates and one OR gate. The first two inputs are A and B and the third input is an input carry as C-IN. The output carry is designated as C-OUT and the normal output is designated as S which is SUM. Equation obtained by EX-OR gate is the sum of the binary digits. While the output obtained by AND gate is the carry obtained by addition.

Logical Expression for Full adder is:
$S=a\oplus b\oplus Cin$; $Cout=(a*b)+(Cin*(a\oplus b))$.

Symbol:-



Procedure:-

24) Run Multisim software.
25) Go to the menu bar.
26) In menu bar > Place > Components.
27) In Component > Database > Master database after database there is option for Group choose group which related to the equipment you needed.
28) In this experiment first we put ground so ground equipment location is Place > Components > Master Database > Group Sources > Power_Sources > Ground then click on ok and place it.
29) We put VCC 5.0V so equipment location is Place > Components > Master Database > Group Sources > Power_Sources > VCC then click on ok and place it.
30) We put SPDT Switches so equipment location is Place > Components > Master Database > Group Basic > Switch > SPDT then click on ok and place it.
31) We are using three SPDT switches.
32) We are using two XOR gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > XOR2 then click on ok and place it.
33) We are using two AND gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > AND2 then click on ok and place it.
34) OR gate location Multisim software is Place > Components > Master Database > Group Misc Digital > TIL > OR2 then click on ok and place it.
35) Last equipment we use in this circuit is LED for getting result of this circuit the led location is Place > Components > Master Database > Group Diodes > LED > LED_orange & LED_red then click on ok and place it.
36) In upward mention 10Points we discuss how we select equipment for using in this circuit through multisim software.

Now in this stage we discuss how we connected the circuit. First switch 1 connect to the ground then switch 2 also connected to the same wire which switch 1 connected to ground after that we connected all the logic gates which using in this circuit you can see in the below attached diagram. After complete this circuit connection we run the circuit and we get result mention in below truth table.

Input (A)	**Input (B)**	**Input (Cin)**	**Output (Sum)**	**Output (Carry)**
0 (LOW)	0 (LOW)	0 (LOW)	0 (LOW)	0 (LOW)
0 (LOW)	0 (LOW)	1 (HIGH)	1 (HIGH)	0 (LOW)
0 (LOW)	1 (HIGH)	0 (LOW)	1 (HIGH)	0 (LOW)
0 (LOW)	1 (HIGH)	1 (HIGH)	0 (LOW)	1 (HIGH)
1 (HIGH)	0 (LOW)	0 (LOW)	1 (HIGH)	0 (LOW)
1 (HIGH)	0 (LOW)	1 (HIGH)	0 (LOW)	1 (HIGH)
1 (HIGH)	1 (HIGH)	0 (LOW)	0 (LOW)	1 (HIGH)
1 (HIGH)	1 (HIGH)	1 (HIGH)	1 (HIGH)	1 (HIGH)

Table: 12.1

Conclusion:- In this circuit by using **Multisim** Software we have observed the functionality of full adder digital circuit.

Diagram:-