AMERICAN INTERNATIONAL UNIVERSITY-BANGLADESH (AIUB)

Faculty of Engineering

Department of Electrical and Electronic Engineering

EXPERIMENT NO: 01 **MID TERM**

SEC: EE GROUP :07

Electronic Device Lab

<u>EXPERIMENT TITLE:</u> *Determination of Characteristic Curve of a Diode.*

<u>Submitted to:</u>

Raja Rashidul Hasan

Assistant Professor

Faculty of Engineering

Department of EEE

Semester: Spring 2021-22

Date of Submission: 09/02/2022

SUBMITTED BY: Anam Zawad **GROUP NO: 07**

Serial no	Student's Name	ID	Dept.
1.	Anam Zawad	20-43967-2	CSE
2.	Shamiur Rahman Shakir	20-43543-1	CSE
3.	Iqbal, Asif	20-43187-1	CSE
4.	Shah, Faysal Mohammed	19-39367-1	CSE
5.	Das, Sudipta Kumar	20-43658-2	CSE

·

Table of Contents

Title: **Determination of Characteristic Curve of a Diode**

Abstract:

A diode is one of the simplest electronic devices. It is a unidirectional device. A diode does not behave linearly with respect to applied voltage and has an exponential I-V relationship. There are two operating regions for the diode, reverse biased region, and forward biased region. The diode is simply a semiconductor pn junction. In addition to being applied as a diode, the pn junction is the basic element of bipolar-junction transistors (BJTs) and field-effect transistors (FETs).

Introduction:

The objectives of the Experiment 1 of the Electronic Devices Lab are,

i) To become familiar with semiconductor diode.
ii) To determine the characteristic curve of a semiconductor diode.

Theory & Methodology:

Diode Structure

The semiconductor diode is created by simply joining an n-type and a p-type material together [1]. It is a pn junction as shown in Figure 1. As indicated, the pn junction consists of p-type semiconductor material in contact with n-type semiconductor material. A variety of semiconductor materials can be used to form pn junctions like silicon, germanium, gallium arsenide etc. However, we will concentrate on silicon, as this is the most widely used material in microelectronics. In actual practice, both the p and n regions are part of the same silicon crystal. The pn junction is formed by creating regions of different doping (p and n regions) within a single piece of silicon. The material is doped by bringing in additional atoms (impurities). The impurities can be either donors or acceptors atoms. The words acceptor and donor can be associated with donating and accepting electrons.

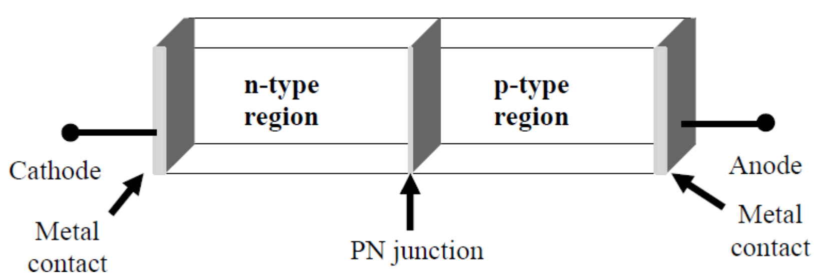


Figure 1: pn junction diode structure

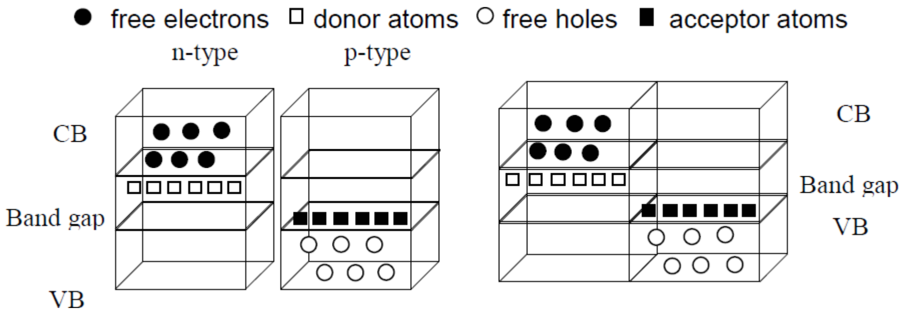


Figure 2: a) separate pieces b) pn junction

<u>PN Junction</u>

To understand how a pn junction is formed we will start by imagining two separate pieces of semiconductor, one n-type and the other p-type as shown in Figure 2(a). Now we bring the two pieces together to make one piece of semiconductor. This results in the formation of a pn junction (Figure 2(b)).

<u>Forward/Reverse Bias Characteristics</u>

If a negative voltage is applied to the pn junction, the diode is in reverse biased. In response, free holes and electrons are pulled towards the end of the crystal and away from the junction. The result is that all available carriers are attracted away from the junction, and the depletion region is extended. There is no current flow through under such conditions. If the applied voltage is positive, the diode operates in forward bias. This has the effect of shrinking the depletion region. Now, electrons in the p-type end are attracted to the positive applied voltage, while holes in the n-type end are attracted to the negative applied voltage.

<u>Diode Characteristics</u>

In forward bias condition, a cut-in voltage has to be overcome for the diode to start conduction. In silicon, this voltage is about 0.7 volts. In reverse-bias condition, the current is limited to IS (reverse saturation current). For higher value of reverse voltages, the junction breaks down. Figure 3 shows the diode I-V characteristics.

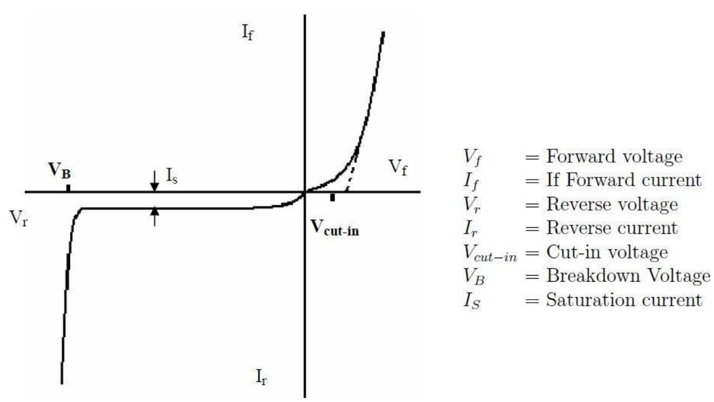


Figure 3: Diode IV Characteristics

Circuit Diagram:

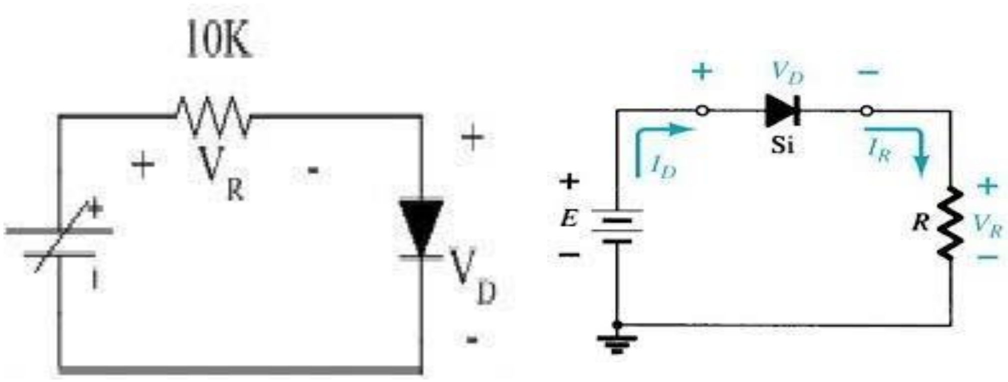


Figure 4: Circuit diagram for determining diode characteristic

Apparatus:

No.	Apparatus	Quantity
1	Diode	1
2	10 k Resistance	1
3	Project Board	1
4	DC Power Supply	1
5	Multimeter	1

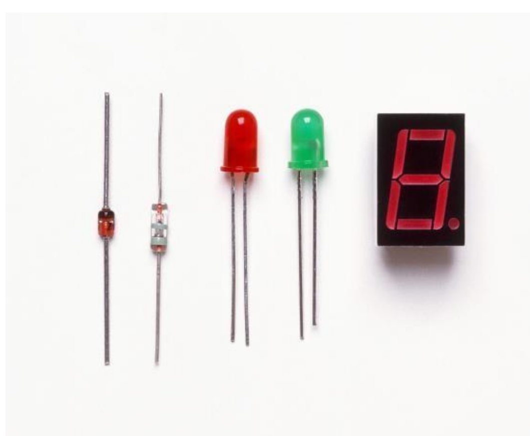


Fig: Diode

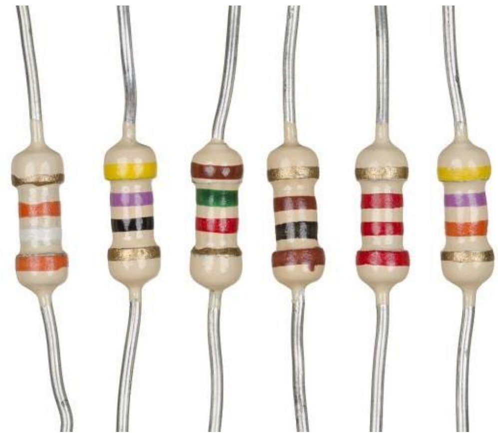


Fig: Resistor

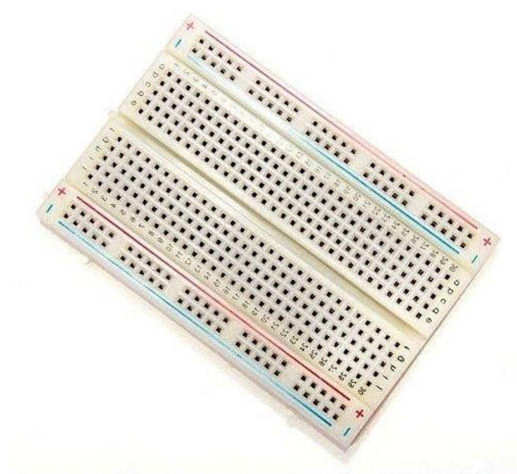


Fig: Project Board

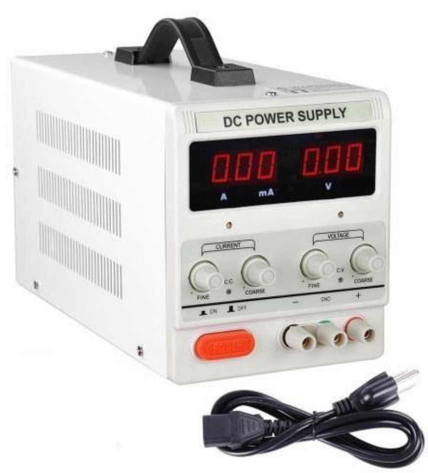


Fig: DC Power Supply

Fig: Multimeter

Data Table:

Vin(V)	Vd(V)	Vr(V)	Id= VR/(10K)(mA)
0	0	0	0
0.1	0.056	0.043	0.004
0.3	0.137	0.163	0.016
0.5	0.190	0.309	0.031
0.6	0.210	0.389	0.039
0.7	0.228	0.471	0.047
0.9	0.243	0.556	0.056
2	0.350	1.650	0.165
3	0.396	2.604	0.260
4	0.429	3.571	0.357
5	0.453	4.547	0.455

So, from the theory analysis, we basically know that the voltage remains constant after threshold voltage and the current keeps increasing after that. The same thing we got from our practical analysis. From the data table we seen that, as we kept increasing the input voltage, the forward voltage kept increasing too but the forward current almost stayed in constant state until it reached the threshold voltage. Here it can be seen that the forward voltage kept increasing until it reached the threshold voltage which is 0.7 here for Silicon. And after that current gets to flow while the forward voltage stays in constant state. So, we can easily said that, what we learned in theory, is justified through this practical analysis.

Calculations:

Here, $I_d = V_R/R$ For

V_R=0
I_d= 0/10K Ohm=0mA

For V_R=0.043
I_d= 0.043/10K Ohm=0.0043mA

For V_R=0.163
I_d= 0.163/10K Ohm= 0.0163mA

For V_R=0.309
I_d= 0.309/10K Ohm=0.0309mA

For V_R=0.389
I_d= 0.389/10K Ohm=0.0389mA

For V_R=0.471
I_d= 0.471/10K Ohm= 0.0471mA

For V_R=0.556
I_d= 0.556/10K Ohm=0.0556mA

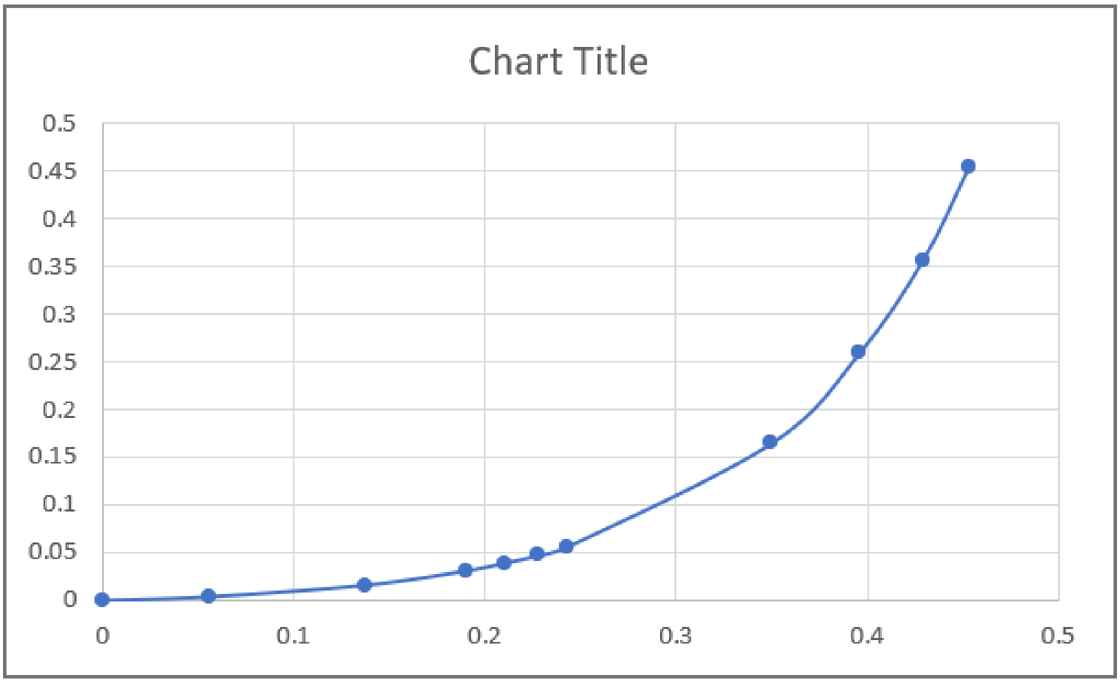
For V_R=1.650
I_d= 1.650/10K Ohm= 0.165mA

For V_R=2.604
I_d= 2.604/10K Ohm=0.2604Ma

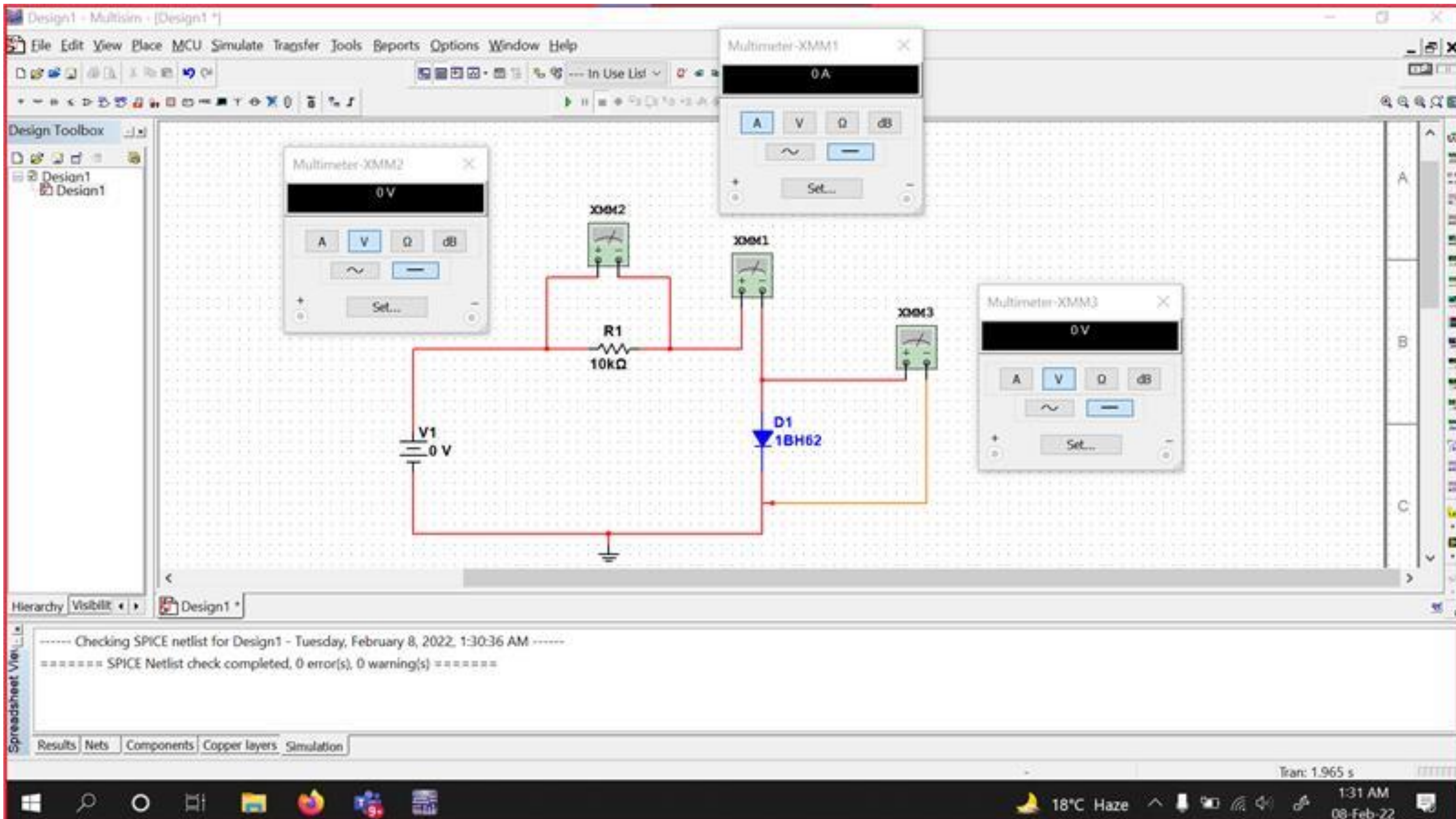
For VR=3.571
Id= 3.571/10K Ohm= 0.3571Ma

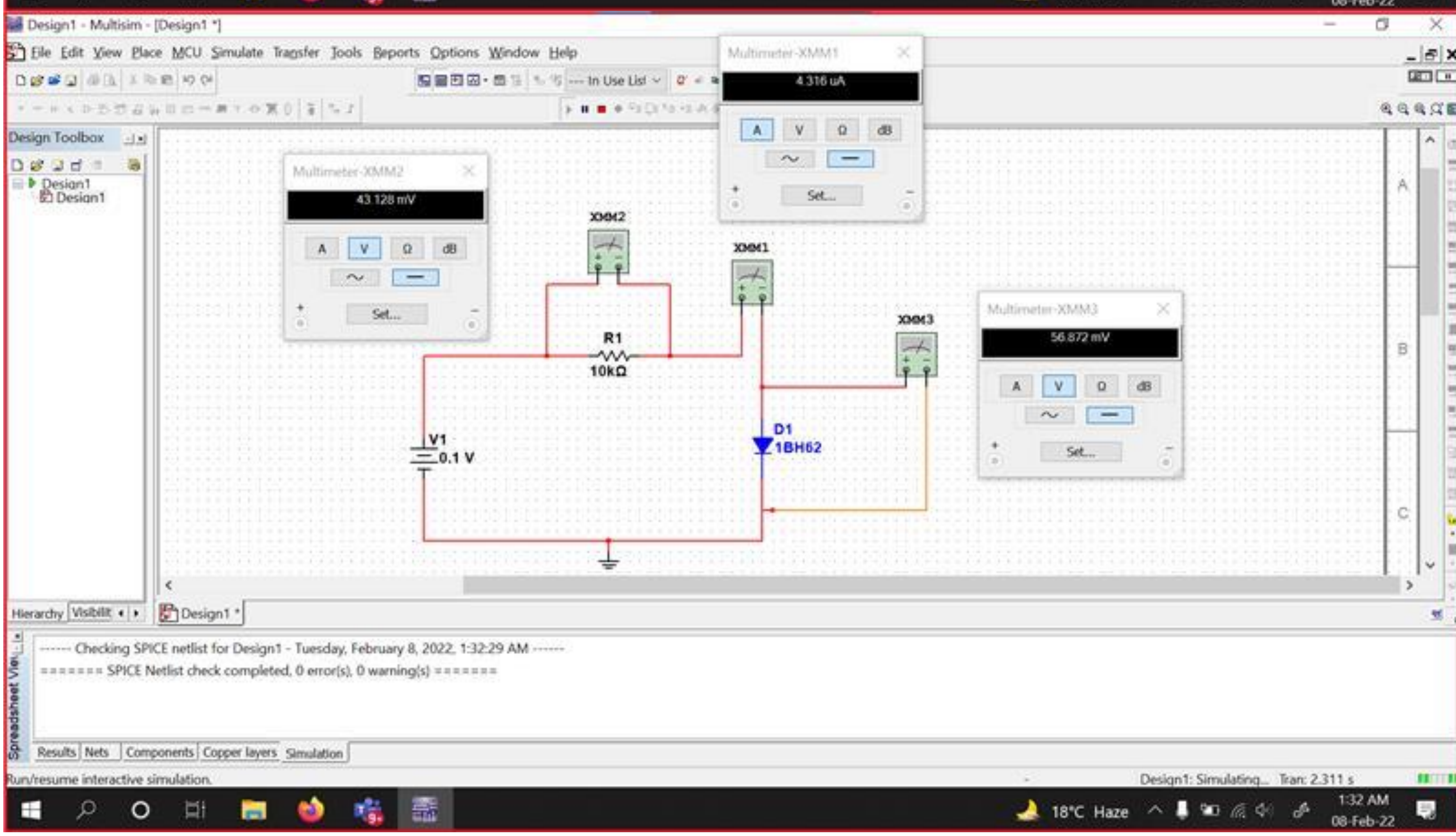
For VR=4.547
Id= 4.547/10K Ohm=0.4547mA

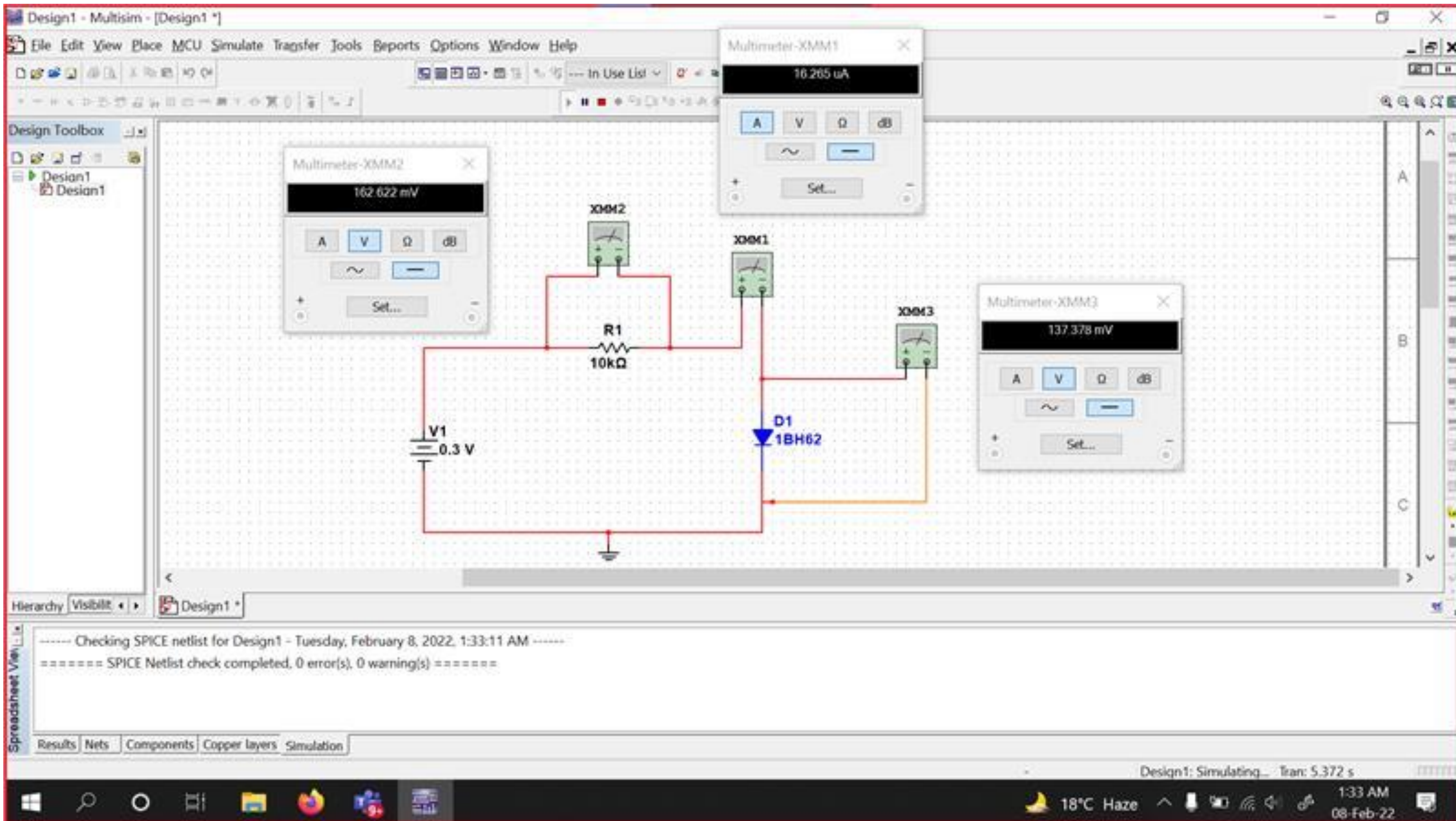
Graph

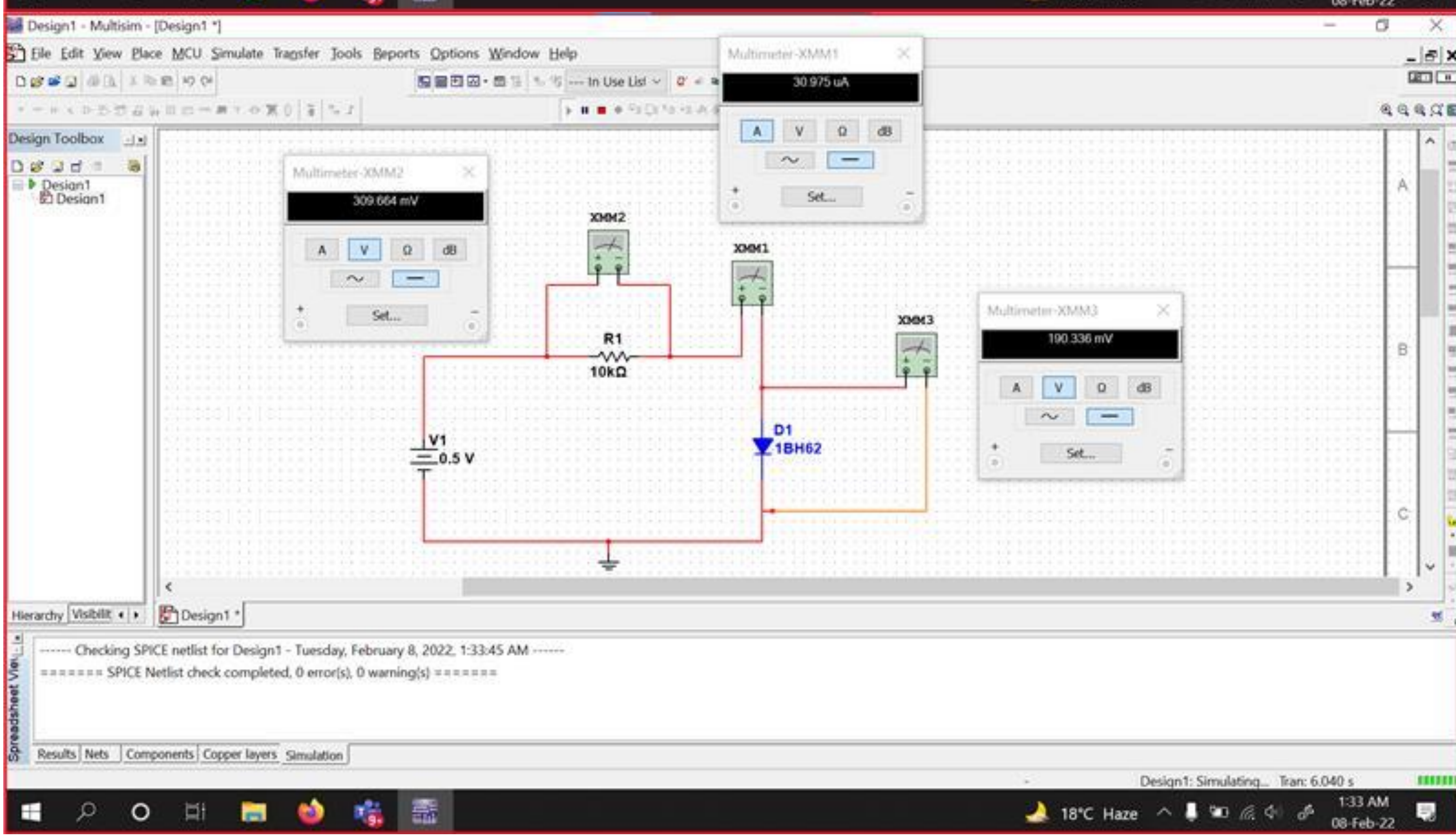


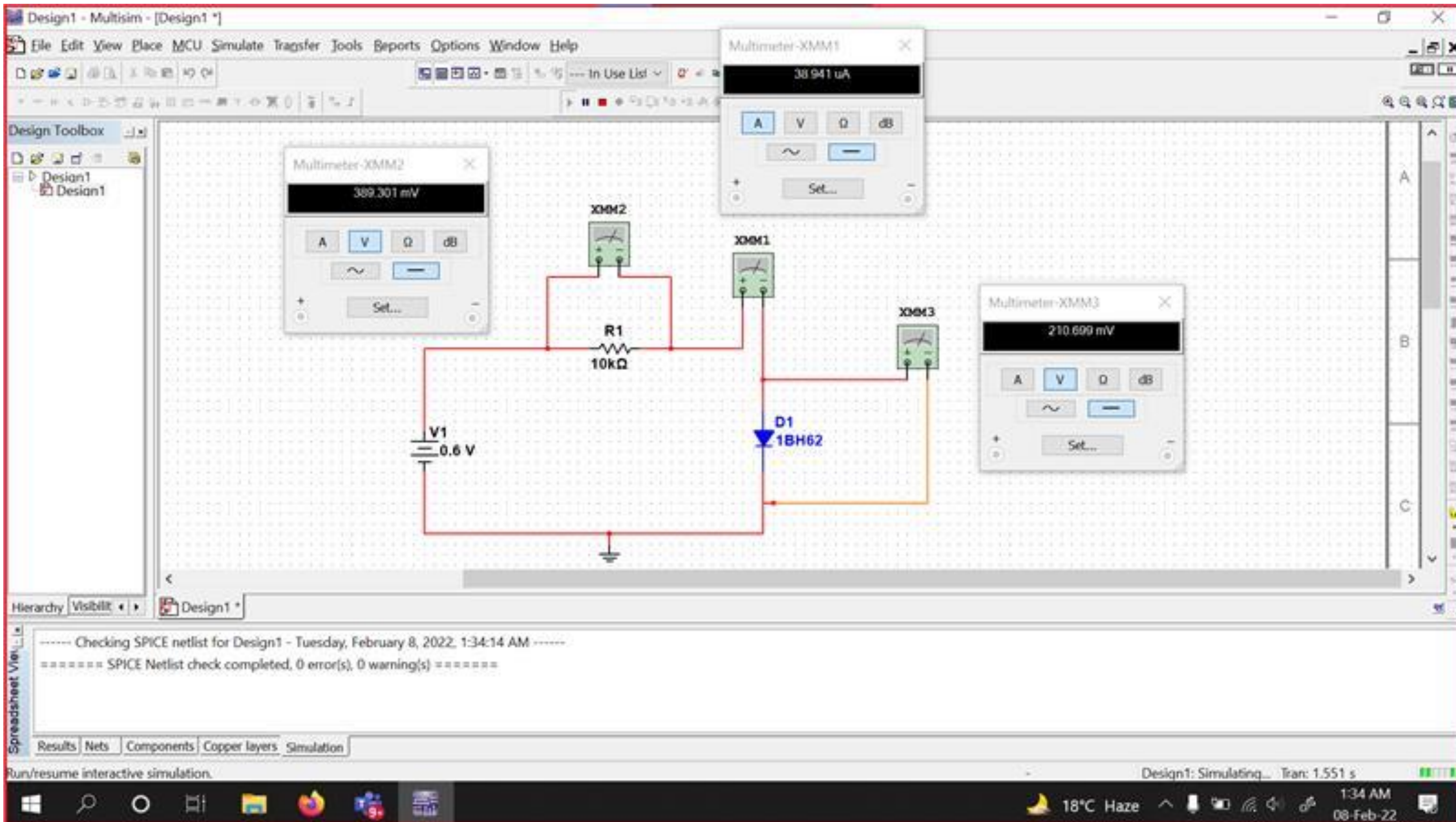
Simulation:

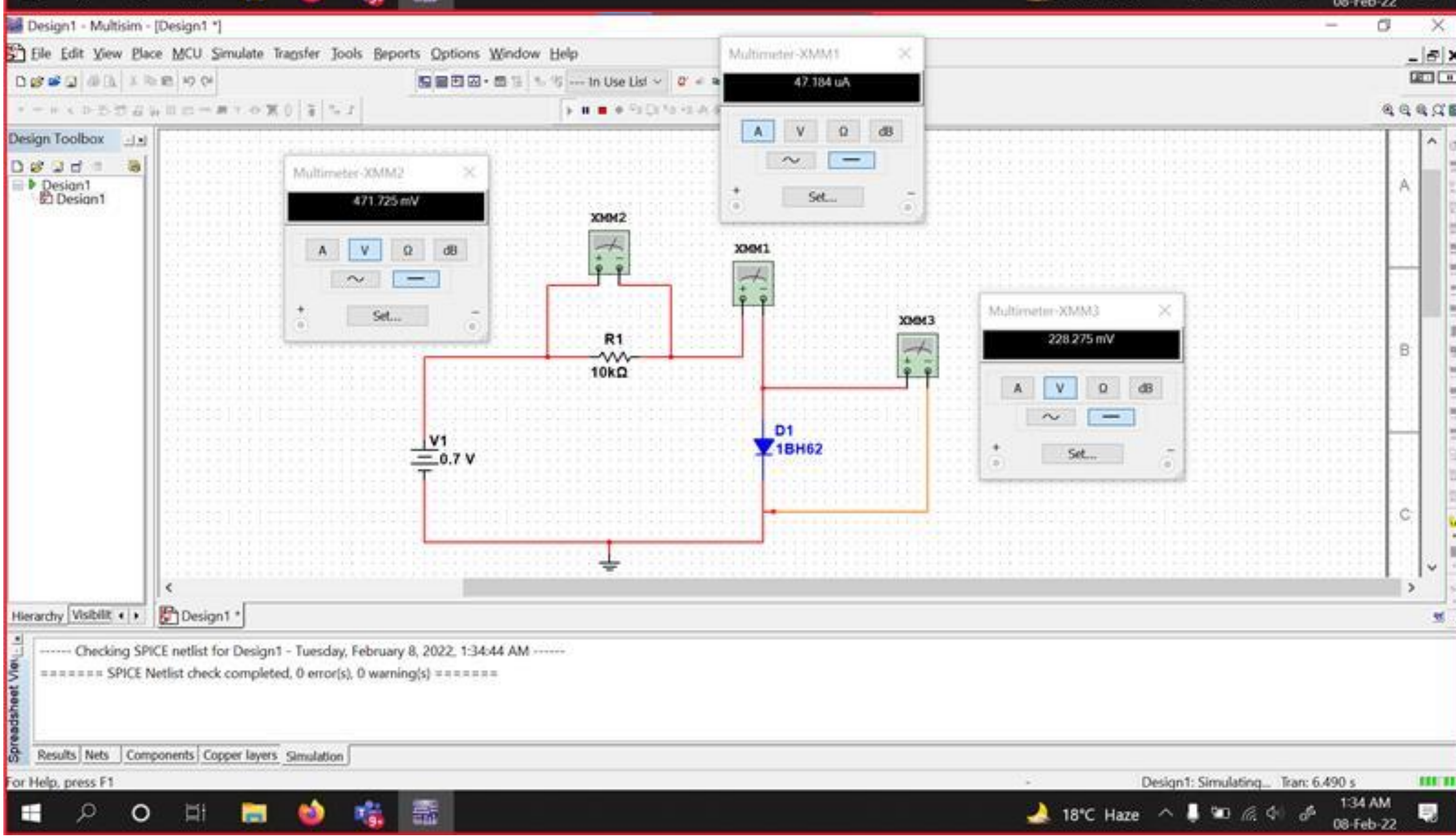

Design1 - Multisim - [Design1 *]
Multimeter-XMM1
0 A
Multimeter-XMM2
0 V
Multimeter-XMM3
0 V
R1
10kΩ
V1
0 V
D1
1BH62
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:30:36 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Tran: 1.965 s

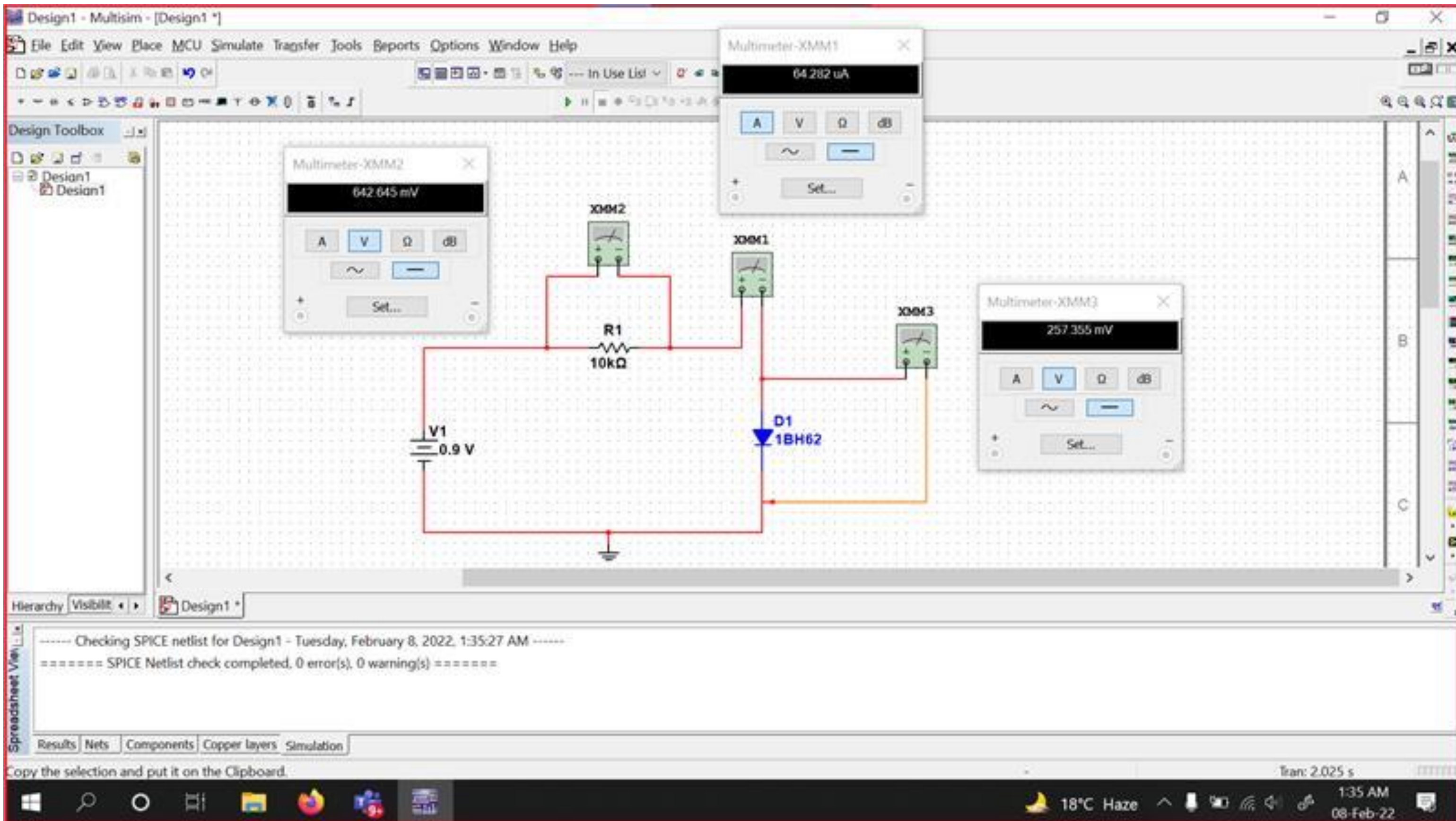

Design1 - Multisim - [Design1 *]
Multimeter-XMM1
4.316 uA
Multimeter-XMM2
43.128 mV
Multimeter-XMM3
56.872 mV
R1
10kΩ
V1
0.1 V
D1
1BH62
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:32:29 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Run/resume interactive simulation.
Design1: Simulating... Tran: 2.311 s

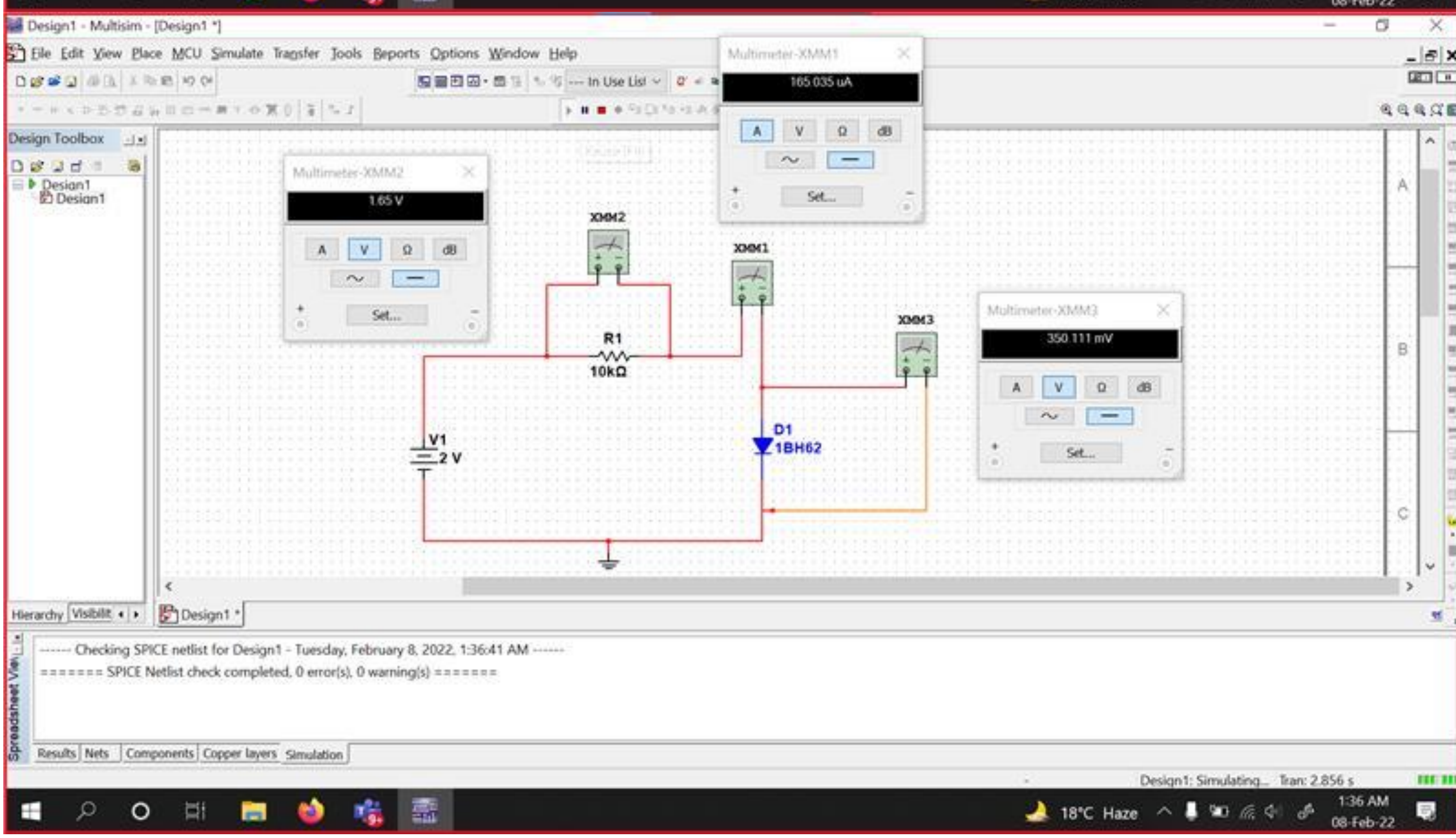

Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Design Toolbox
Multimeter-XMM2
162.622 mV
Multimeter-XMM1
16.265 uA
Multimeter-XMM3
137.378 mV
XMM2
XMM1
XMM3
R1
10kΩ
V1
0.3 V
D1
1BH62
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:33:11 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Results Nets Components Copper layers Simulation
Design1: Simulating... Tran: 5.372 s
18°C Haze
1:33 AM
08-Feb-22

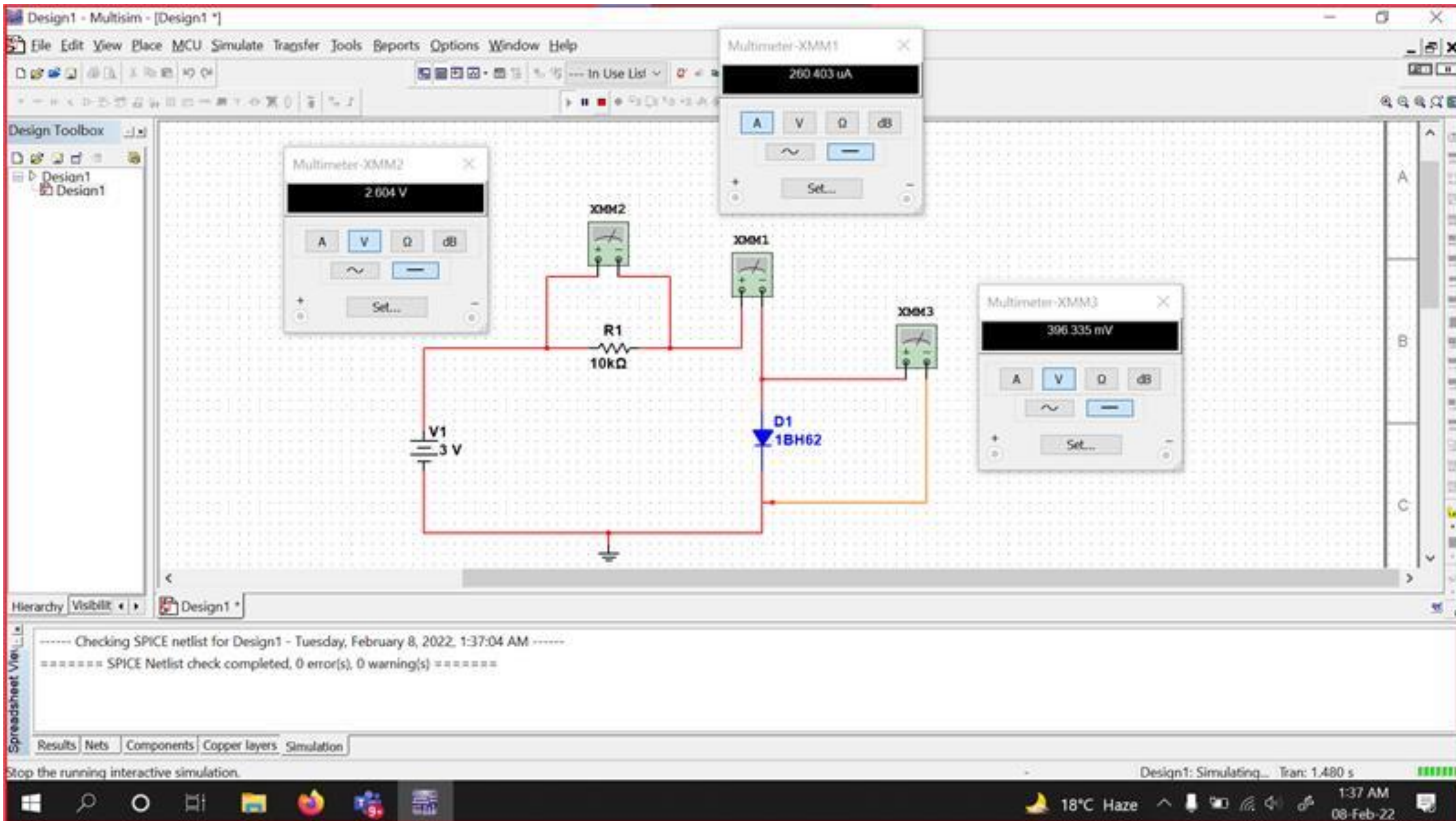

Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Design Toolbox
Multimeter-XMM2
309.664 mV
Multimeter-XMM1
30.975 uA
Multimeter-XMM3
190.336 mV
XMM2
XMM1
XMM3
R1
10kΩ
V1
0.5 V
D1
1BH62
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:33:45 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Results Nets Components Copper layers Simulation
Design1: Simulating... Tran: 6.040 s
18°C Haze
1:33 AM
08-Feb-22

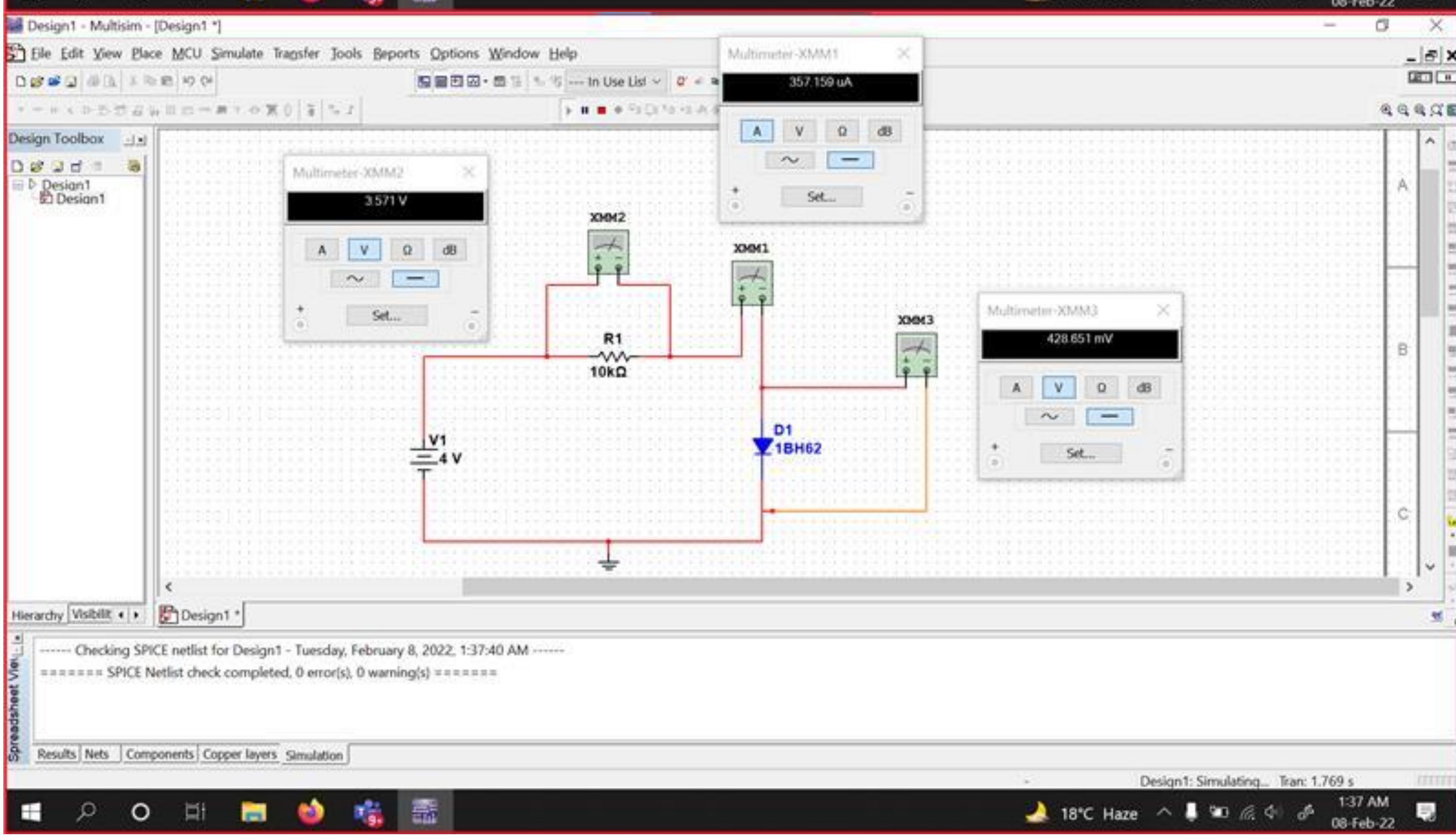

Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Multimeter-XMM1
38.941 uA
Multimeter-XMM2
389.301 mV
Multimeter-XMM3
210.699 mV
XMM1
XMM2
XMM3
R1
10kΩ
V1
0.6 V
D1
1BH62
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:34:14 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Run/resume interactive simulation.
Design1: Simulating... Tran: 1.551 s
18°C Haze
1:34 AM
08-Feb-22

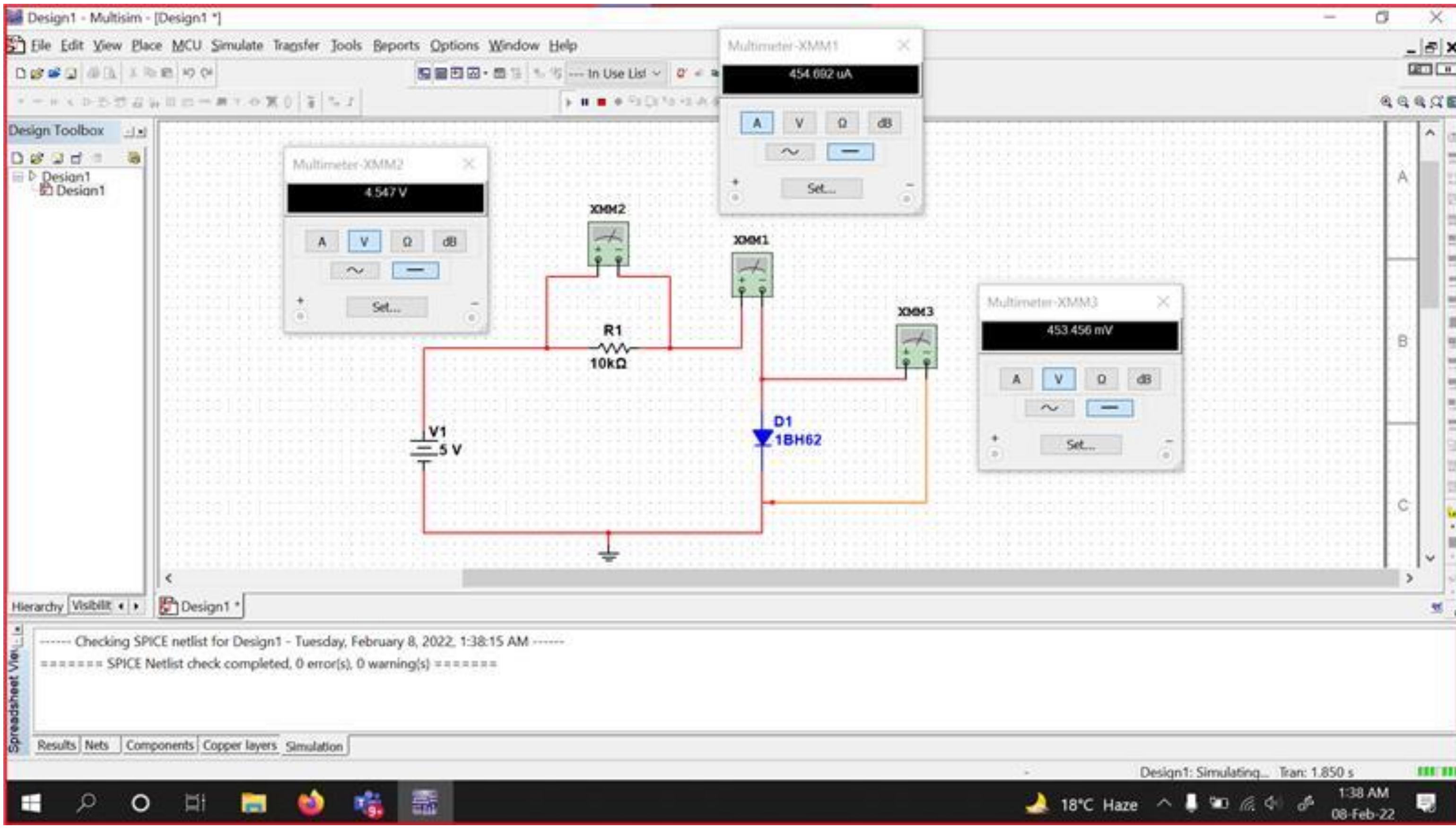

Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Multimeter-XMM1
47.184 uA
Multimeter-XMM2
471.725 mV
Multimeter-XMM3
228.275 mV
XMM1
XMM2
XMM3
R1
10kΩ
V1
0.7 V
D1
1BH62
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:34:44 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
For Help, press F1
Design1: Simulating... Tran: 6.490 s
18°C Haze
1:34 AM
08-Feb-22


Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Multimeter-XMM1
64.282 uA
Multimeter-XMM2
642.645 mV
Multimeter-XMM3
257.355 mV
XMM1
XMM2
XMM3
R1
10kΩ
V1
0.9 V
D1
1BH62
Design Toolbox
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:35:27 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Results Nets Components Copper layers Simulation
Copy the selection and put it on the Clipboard.
Tran: 2.025 s
18°C Haze
1:35 AM
08-Feb-22


Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Multimeter-XMM1
165.035 uA
Multimeter-XMM2
1.65 V
Multimeter-XMM3
350.111 mV
XMM1
XMM2
XMM3
R1
10kΩ
V1
2 V
D1
1BH62
Design Toolbox
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:36:41 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Results Nets Components Copper layers Simulation
Design1: Simulating... Tran: 2.856 s
18°C Haze
1:36 AM
08-Feb-22


Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Multimeter-XMM1
260.403 uA
Multimeter-XMM2
2.604 V
Multimeter-XMM3
396.335 mV
XMM2
XMM1
XMM3
R1
10kΩ
V1
3 V
D1
1BH62
Design Toolbox
Design1
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:37:04 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Results Nets Components Copper layers Simulation
Stop the running interactive simulation.
Design1: Simulating... Tran: 1.480 s
18°C Haze
1:37 AM
08-Feb-22


Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Multimeter-XMM1
357.159 uA
Multimeter-XMM2
3.571 V
Multimeter-XMM3
428.651 mV
XMM2
XMM1
XMM3
R1
10kΩ
V1
4 V
D1
1BH62
Design Toolbox
Design1
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:37:40 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Results Nets Components Copper layers Simulation
Design1: Simulating... Tran: 1.769 s
18°C Haze
1:37 AM
08-Feb-22


Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Multimeter-XMM1
454.692 uA
Multimeter-XMM2
4.547 V
Multimeter-XMM3
453.456 mV
XMM1
XMM2
XMM3
R1
10kΩ
V1
5 V
D1
1BH62
------ Checking SPICE netlist for Design1 - Tuesday, February 8, 2022, 1:38:15 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Design1: Simulating... Tran: 1.850 s
18°C Haze
1:38 AM
08-Feb-22

Here is the drive link of our simulation part,

https://drive.google.com/drive/folders/13zr6DkAPRiPGiuaWGRUY42enjekaIYLL?usp=sharing

Discussion:

From this overall experiment we mainly got to learn about the construction of a semiconductor diode and how to operate a semiconductor diode. Si diode Improved temperature sensitivity and easily available. Electronics have become more sensitive to "Speed" concerns in recent years. The outcome corresponds to the theoretical value. The curves show that the diode's knee of curve is around 0.7 V. It has been proven that our experimental result matches the theoretical outcome using the data table for diode characteristic. As a result, the theory was proven correct, and the experiment went off without a hitch. The entire experiment went off without a hitch.

References:

1. Robert L. Boylestad, Louis Nashelsky, Electronic Devices and Circuit Theory, Ninth Edition, 2007-2008
2. Adel S. Sedra, Kennth C. Smith, Microelectronic Circuits, Saunders College Publishing, 3rd ed., ISBN: 0-03-051648-X, 1991.
3. American International University–Bangladesh (AIUB) Electronic Devices Lab Manual.
4. 4. David J. Comer, Donald T. Comer, Fundamentals of Electronic Circuit Design, john Wiley & Sons Canada, Ltd.; ISBN: 0471410160, 2002.

Video Link: https://youtu.be/UqJ258EPTkI