School of Science, Engineering and Technology

Engineering Design 2

EEET-2602: Session 01 – Team 02

Assessment 4

Submitted by

Student #1: [Duong Bao Khang – s3926606]

Student #2: [Dam Ta Minh Nhat – s3926946]

Student #3: [Le Huu Duc - s3926088]

Student #4: [Ngo Nhat Anh Khoa - s3924481]

Authors Statement: This lab is the original work of the above authors. We as authors understand resubmissions of others work without proper citation is an act of plagiarism and is subject to the penalties as put forth by RMIT University Vietnam and the School of Science, Engineering and Technology.

Date Submitted:	22/09/2023.
Lab Session:	01
Course Instructor :	Mr. Khuong Nguyen

Table of Contents

List of Figures

List of Tables

Nomenclature

V_{DD} = DC Voltage Source

V_{dd} = AC Volatge Source

I_{ref} = Reference Current

NI = National Instruments

IC = Integrated Circuit

DC = Direct Current

AC = Alternating Current

PCB = Printed Circuit Board

Introduction

This report is part of the assessing process of Assessment 4, and is to:

- Help student understand the circuit design process.
- Get familiar with the printed circuit board (PCB) design process.
- Understand the standards of a good PCB solder in the industry.
- Understand the application of the circuit design in the industry.
- Understand the circuit test process after getting the circuit board ready.

Circuit Design and Simulation

In this section, students will need to do the simulation of the circuit in Figure 1.

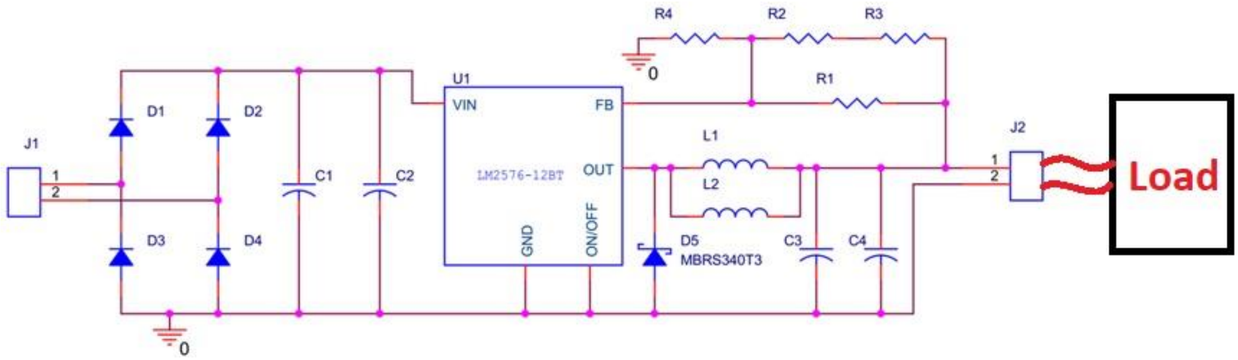

Figure 1. Real circuit to be simulated and tested

Detailed component values and models are shown in Table 1.

Table 1. Components' values and models

Device	Model	Name on Schematic
Diode (normal)	1N4001	D1, D2, D3, D4
Diode (Schottky)	MBRS340T3	D5
Linear Regulator IC	LM2576	U1
Inductor	220uH	L1, L2
Capacitor	330uF	C1, C2, C3, C4
Resistor	8.2kΩ	R3
Resistor	1.3kΩ	R4
Resistor	2.4kΩ	R1, R2

In this circuit, a design of a Voltage Regulator using an IC Linear Regulator was taken into account. Prior to having the actual circuit board designed, simulation process should be taken by the team members. NI Multisim 14.2 is the tool that will be used in this simulation process. The IC chip in the real circuit is LM2576 model while this model is not available in NI Multisim. The team proactively used LM2592HVS-ADJ/NOPB, which its characteristic can be sufficiently compared to the LM2576 one. The input of the experiment is 5Vrms which will be taken directly from the NI Elvis II board and to be tested at 100kHz. The reason testing the circuit of a circuit with an IC Linear Regulator at 100kHz is because this model can output a stable DC voltage with low noise [1]. Additionally, the model is cost saving to be designed since it requires a low external number of components with a wide range of capacitance selection [2]. The load is simulated at 328 Ohms.

The simulations process was run with the simulation circuit in NI Multisim as shown in Figure 2.

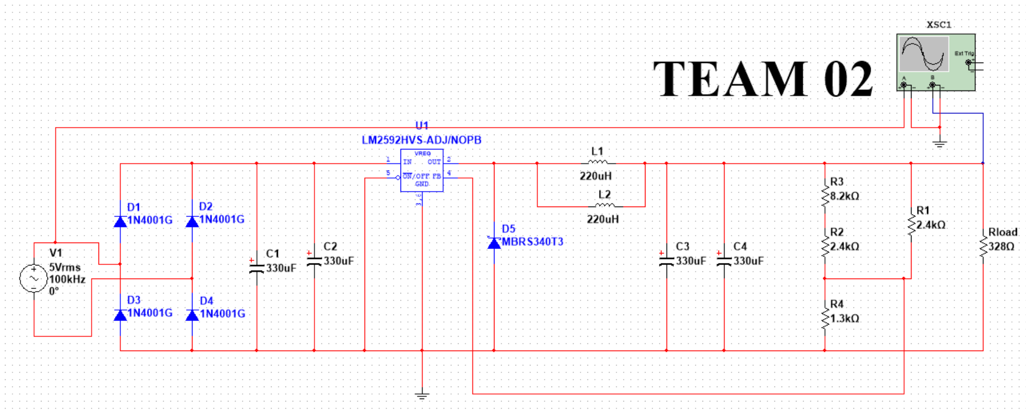

Figure 2. Simulation circuit on Multisim

The simulated results are shown in Figure 3.

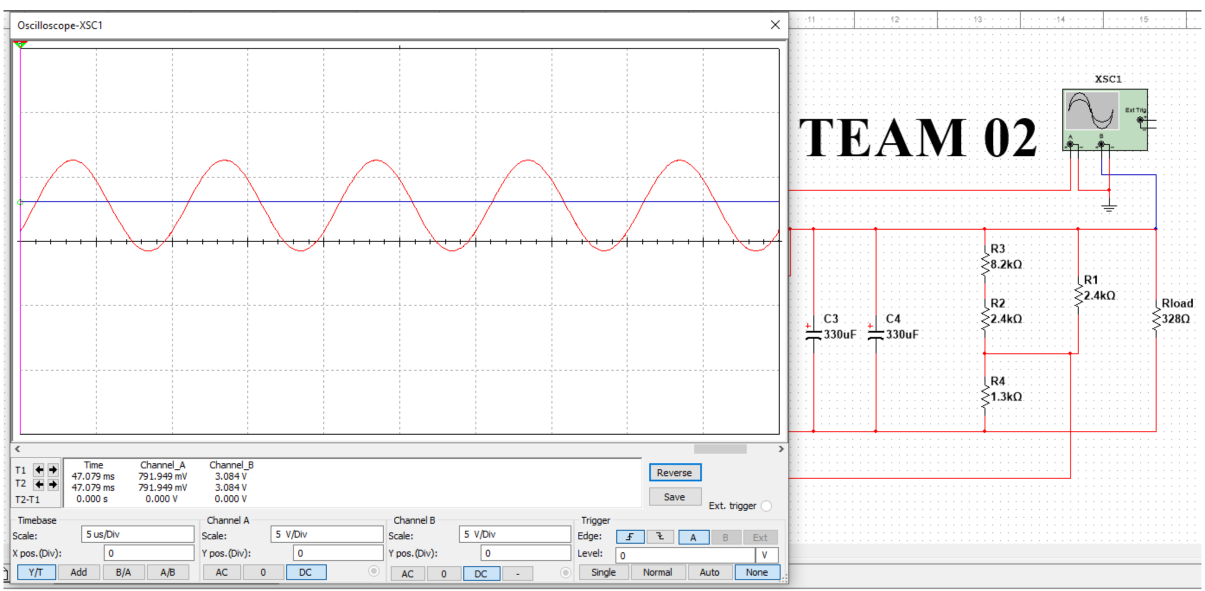

Figure 3. Simulation results on Multisim

The simulated results are analyzed by the using the cursors as shown in Figure 4.

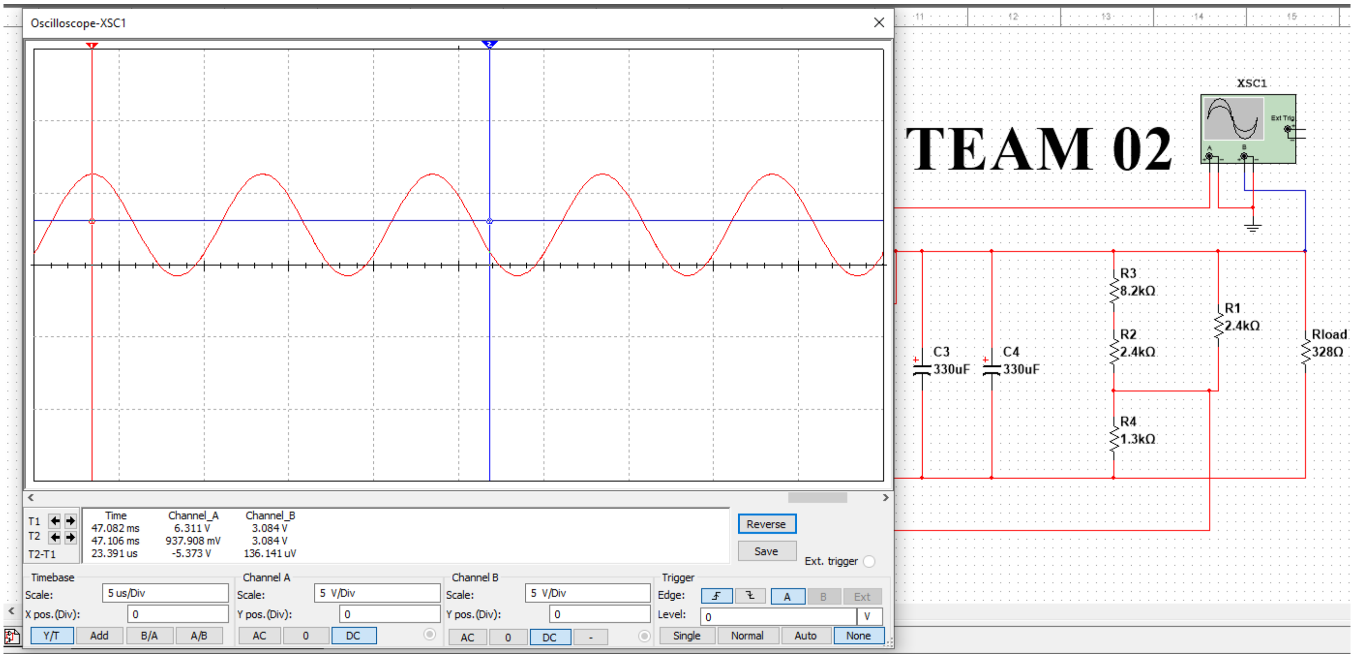

Figure 4. Simulation results analysis with cursors on Multisim

In the design, two inductors were connected parallel in order to prevent circuit-failure when a sudden change in the current occurs [3]. Furthermore, these inductors combine with the 2 capacitors C3 and C4 to make an LC circuit that stores energy and helps maintain the load voltage [4]. Capacitors C1 and C2 were used for filtering the ripple of rectified voltage [5] while capacitors C3 and C4 were used for smoothing the output voltage to provide a stable DC current [5]. Schottky Diode was placed at the output of the IC chip to protect the IC from reverse current and voltage spike [6]. Additionally, the diode helps save power consumption with its fast-switching time characteristic [7].

From the information that was found through the cursors in Figure 4, the team came up with the data of simulation in Table 2.

Table 2. Simulation Data Summary

Data name	Value	Unit
Output voltage	3.084	V
Ripple voltage	136.141	μV
Output current	9.402	mA

PCB Design

After simulating the circuit, the team started to design the PCB and have its schematic designed as the same to Figure 1. Detailed components' footprints are in Figure 5.

		Part Reference	PCB Footprint	Power Pins Visible
1	SCHEMATIC1 : PAGE1 : C1	C1	cyld400ls200034	☐
2	SCHEMATIC1 : PAGE1 : C4	C4	cyld400ls200034	☐
3	SCHEMATIC1 : PAGE1 : C2	C2	cyld400ls200034	☐
4	SCHEMATIC1 : PAGE1 : C3	C3	cyld400ls200034	☐
5	SCHEMATIC1 : PAGE1 : D5	D5	DIOM7959X261N	☐
6	SCHEMATIC1 : PAGE1 : D3	D3	DO41	☐
7	SCHEMATIC1 : PAGE1 : D1	D1	DO41	☐
8	SCHEMATIC1 : PAGE1 : D4	D4	DO41	☐
9	SCHEMATIC1 : PAGE1 : D2	D2	DO41	☐
10	SCHEMATIC1 : PAGE1 : L2	L2	INDP125125X600N	☐
11	SCHEMATIC1 : PAGE1 : L1	L1	INDP125125X600N	☐
12	SCHEMATIC1 : PAGE1 : J1	J1	Jumper2	☐
13	SCHEMATIC1 : PAGE1 : J2	J2	Jumper2	☐
14	SCHEMATIC1 : PAGE1 : R2	R2	res400	☐
15	SCHEMATIC1 : PAGE1 : R4	R4	res400	☐
16	SCHEMATIC1 : PAGE1 : R1	R1	res400	☐
17	SCHEMATIC1 : PAGE1 : R3	R3	res400	☐
18	SCHEMATIC1 : PAGE1 : U1	U1	VREG_TPS79625KTTR	☐

Figure 5. Component's Footprints in Cadence Capture design

Throughout the PCB design process that was learned in class, the team came up with the PCB layout design as shown in Figure 6.

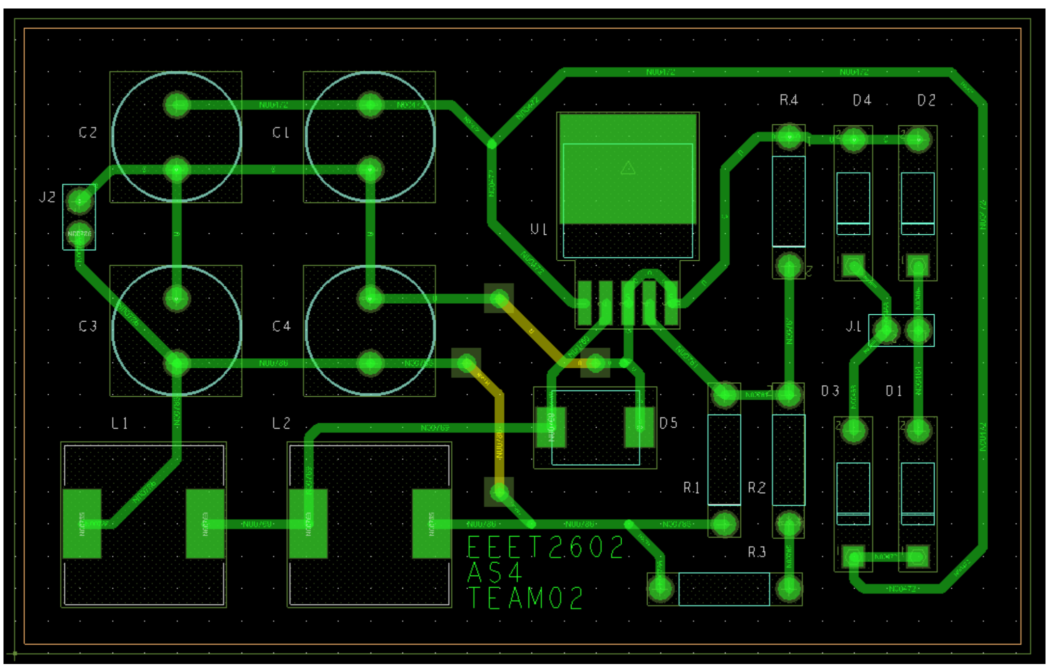

Figure 6. PCB Layout in Cadence PCB Editor design

PCB Manufacturing and Soldering Process

Once the PCB layout was completed, through the Gerber files and Drill file which were generated from the design, the team started to put the parts onto the PCB. The process of PCB soldering was strictly followed the 4 key regulations: Risking mitigation, sequential assembly, quality control, and iron handling [8].

1. First, about risking mitigation, before step in the soldering process, the team need to strictly follow this rule: Avoid, Deny, and Defense which is represented by the acronym ADD. The members need to review the schematic carefully with the pins, verify solder joint quality, ensure proper heat management, and employ proper ESD (Electrostatic Discharge) protection. Then in the soldering section, the teams need to stop right away when something wrong happens by applying the sucker and start the de-soldering again. This is to avoid errors, deny any potential risk and defense against potential issues.
2. The soldering order is from the SMD (Surface-Mounted Device) components to through-hole components, from low components to high components.
3. In the quality control process, the team needs to pay attention to these factors to avoid mistakes which are component orientation, solder joint quality, temperature control, flux residue and cleaning, ESD protection, and soldering Voltera-made PCBs. For this project, some common mistakes include solder bridges, insufficient solder, excessive solder, cold joint or even component damage. By focusing on these aspects and avoiding these mistakes, the team can guarantee high-quality soldering and reliable PCB assemblies.
4. The soldering iron must be handled with some cautions such as:
    - Do not point the soldering iron’s tip towards a person.
    - The joint region should be exposed to the iron's tip while making sure there is adequate contact and heat transfer.
    - Do not touch the metal part of the iron when the tool is heated up. Always hold the plastic handle.
    - Do not make any physical contact with the tip of the soldering iron.
    - Clean the soldering iron with the wet sponge and put it into the holder after finishing soldering to avoid accidental burn or damage.

The team finished soldering the PCB as the figures, from Figure 7 to Figure 14, are showing the different sides of the completed PCB.

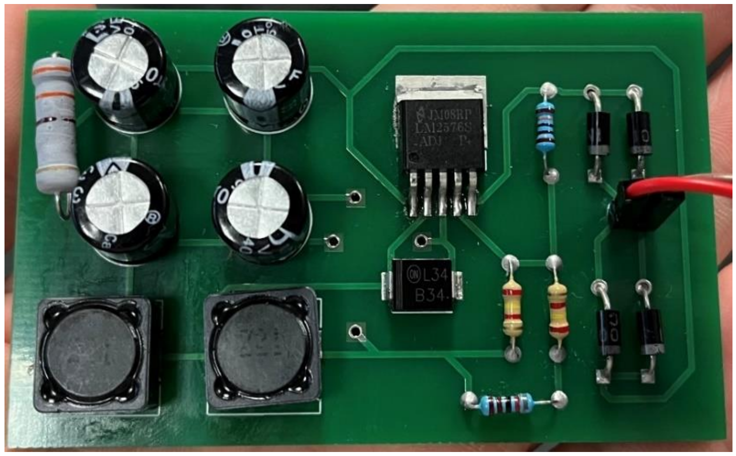

Figure 7. Completed PCB looking from TOP view

Figure 8. Completed PCB looking from the BOTTOM view

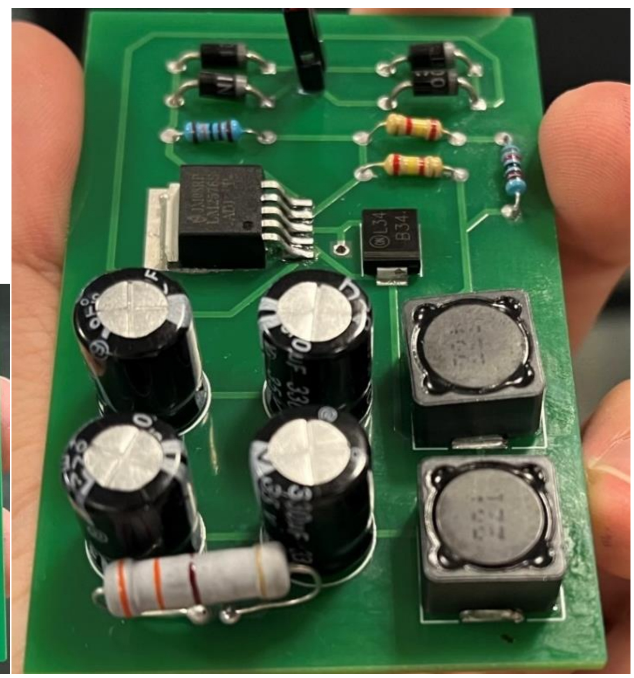

Figure 9. Completed PCB looking from left to right of TOP view

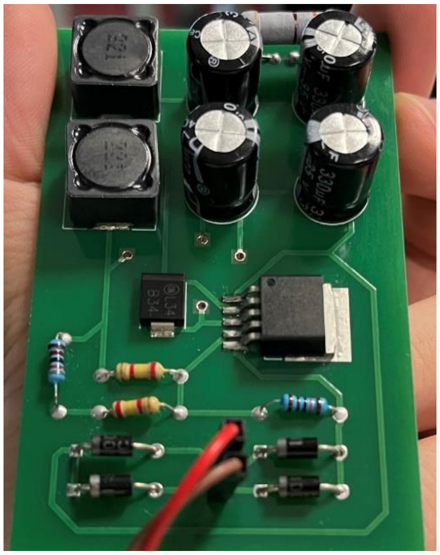

Figure 10. Completed PCB looking from right to left of Top view

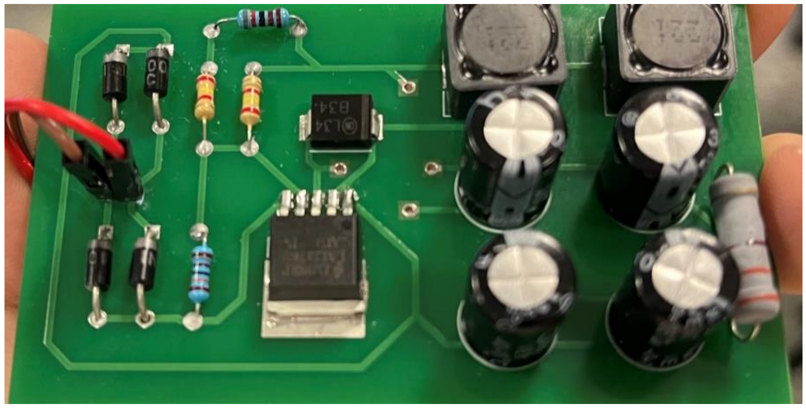

Figure 11. Completed PCB looking from upside to downside of TOP view

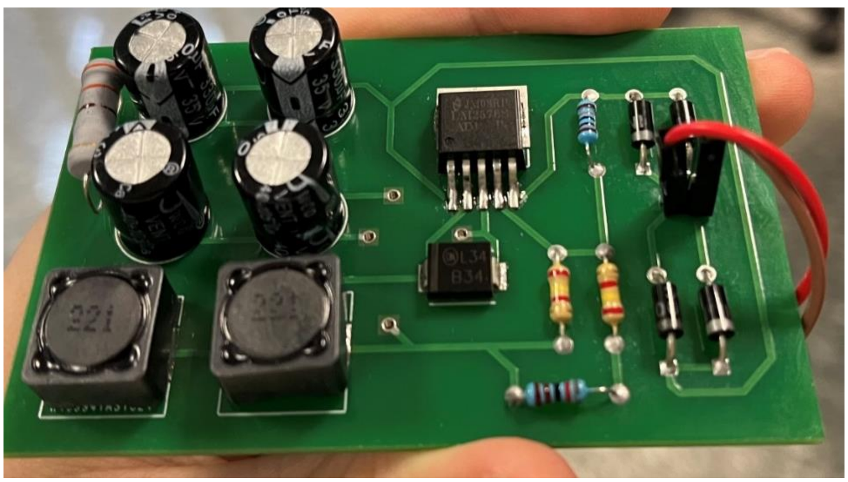

Figure 12. Completed PCB looking from downside to upside of TOP view

Figure 13. Completed PCB looking from right to left of BOTTOM view

Figure 14. Completed PCB looking from left to right of BOTTOM view

Circuit Test

General Information

After the PCB was completed, the circuit was tested by NI Elvis Board by using FGEN. The load for testing is 328-Ohm resistor. The circuit was setup based on the schematic shown in Figure 1. The setup of circuit test is shown in Figure 15.

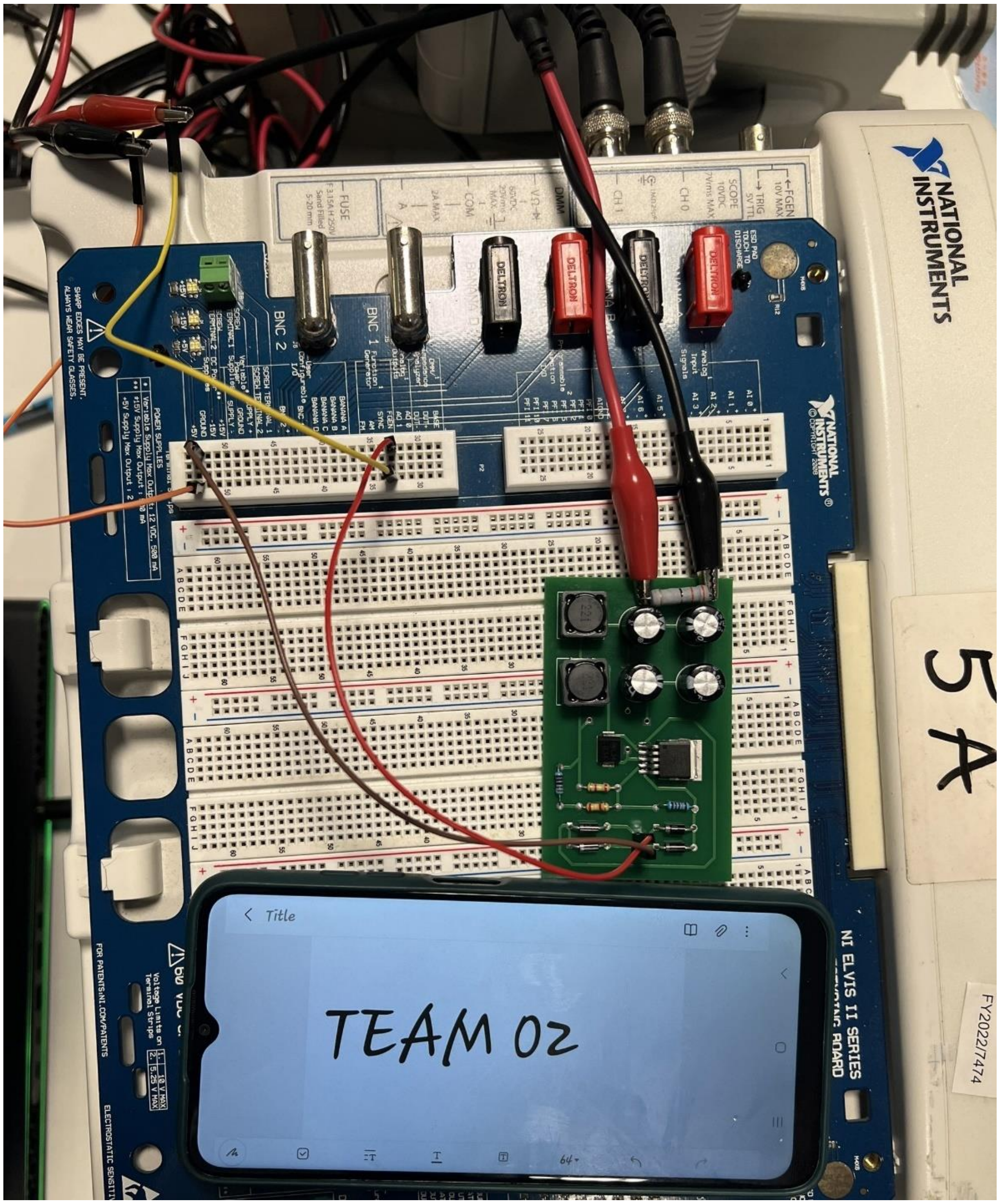

Figure 15. Actual circuit test setup

Circuit Test Data Collection

The actual tested results are shown in Figure 16.

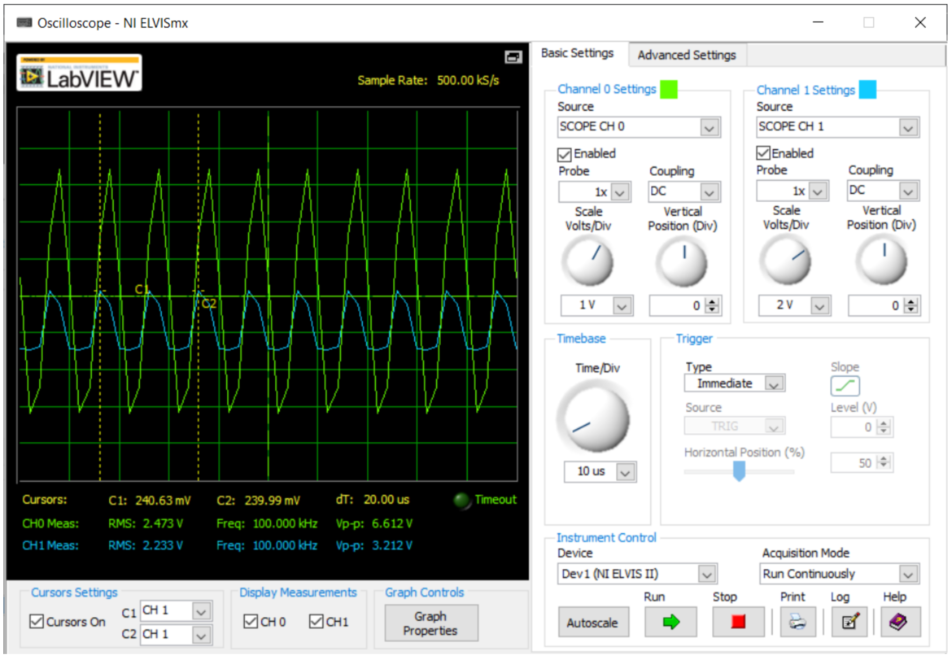

Figure 16. Actual measured waveform

Data from actual experiment was recorded into Table 3.

Table 3. Actual Measured Data Summary

Data name	Value	Unit
Output voltage	3.212	V
Ripple voltage	240.630 – 239.990 = 0.640	mV
Output current	9.793	mA

Discussion of Result

After the circuit was tested. The data from Table 3 were used to compare with the data in Table 2. The difference was recognized and can be explained as the components inserted in Multisim are assumed to be in ideal conditions. In contrast, the real-world ones are non-ideal as they have manufacturing tolerances. The capacitors and inductors provided have various charging stages and internal resistance, whose factors are not considered in Multisim simulation. Furthermore, another reason for the results difference is the temperature influence in the testing environment, which can affect the diode characteristics [9] and the IC Linear Regulator performance in regulating the output voltage [10]. The leverage of long jumper wires can also lead to the instability of the input signal, resulting in a difference in peak input voltage and sharp-edged waveform.

Moreover, compare to what the team designed in Assessment 2, the similarity between the circuits can be listed out as: both designs use an IC Linear Voltage Regulator to control the stability of the output voltage, a full-wave rectifier at the beginning of the circuit, a load resistor at the circuit's output end and capacitor(s) to smoothen the signal.

While there are several differences between them, such as: additional capacitor for rectified voltage, LC circuit, Schottky Diode, resistors at the output of IC and different IC chip's model in Assignment 3 circuit.

Those differences can be explained as: The second capacitor increases the total capacitance of the filtering circuit and therefore decreases the ripple voltage of rectifier. Next, the combination of capacitors and inductors helps in energy storage and further voltage and current stabilization. With the addition of a Schottky Diode, the circuit is protected from reverse current and has a faster switching time. Finally, resistors are added to adjust a desired value for the output current.

References

[1] R. Keim. "What Is a Linear Voltage Regulator?" ALL ABOUT CIRCUITS. https://www.allaboutcircuits.com/technical-articles/what-is-a-linear-voltage-regulator/#:~:text=Linear%20Voltage%20Regulator%20Applications&text=Major%20advantages%20are%20ease%20of,the%20design%20task%20very%20straightforward (accessed Sep. 21, 2023).

[2] CADENCE PCB SOLUTIONS. "The Role of Linear Electronics in Voltage Regulation." cadence. https://resources.pcb.cadence.com/blog/2022-the-role-of-linear-electronics-in-voltage-regulation (accessed Sep. 21, 2023).

[3] ElectronicsTutorials. "Inductors in Parallel." ElectronicsTutorials. https://www.electronics-tutorials.ws/inductor/parallel-inductors.html (accessed Sep. 21, 2023).

[4] L. James. "Understanding the role of inductors in power Electronics. "ELECTRONIC COMPONENTS. https://www.power-and-beyond.com/understanding-the-role-of-inductors-in-power-electronics-a-dc947f2aa88d7ac3009791f798478d1c/ (accessed Sep. 21, 2023).

[5] Learning about Electronics. “What is a Smoothing Capacitor?” Learning about Electronics. http://www.learningaboutelectronics.com/Articles/What-is-a-smoothing-capacitor (accessed Sep. 21, 2023).

[6] Wolfspeed. “Schottky Diode Characteristics and Applications.” Wolfspeed. https://www.wolfspeed.com/knowledge-center/article/schottky-diode-characteristics-and-applications/ (accessed Sep. 21, 2023).

[7] O. M. Urias. “Schottky Diode: A Beginners’ Guide” buildelectroniccircuits. https://www.build-electronic-circuits.com/schottky-diode/ (accessed Sep. 21, 2023).

[8] N. V. Khuong, *11 – EEET2602 – PCB Soldering.* [Online] https://rmit.instructure.com/courses/121705/pages/eeet2602-pcb-soldering?module_item_id=5229442 (accessed Sep. 21, 2023).

[9] Semiconductor For You. “Effect of temperature on diode characteristics.” SemiconductorForu.com. https://www.semiconductorforu.com/effect-temperature-v-characteristics/ (accessed Sep. 21, 2023).

[10] Electrical Volt. “Effect of Temperature on Resistance.” Electrical Volt. https://www.electricalvolt.com/2020/11/effect-of-temperature-on-resistance/(accessed Sep. 21, 2023).