School of Science, Engineering and Technology

Engineering Design 2

EEET2602: Session 4 – Team 16

Assessment 4

Submitted by

Student #1: [Hoang Thai Kiet – s3855250]

Student #2: [Dang Huu Nghia – s3878116]

Student #3: [Pham Khanh Nguyen – s3845568]

Student #4: [Nguyen Huynh Hai Long – s3877624]

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:	20/09/2022
Lab Session:	Session 4 - Brown
Course Instructor:	Mr. Khuong Nguyen

Table of Contents

List of Figures

List of Tables

Nomenclature

V_{DD} = DC Voltage Source

V_{dd} = AC Voltage Source

I_{ref} = Reference Current

$V_{rec,\ min}$ = Minimum Rectifier Voltage

$V_{rec,\ max}$ = Maximum Rectifier Voltage

V_r = Ripple Voltage

$V_{IC\ drop,\ LM2592HV}$ = Ideal Dropout voltage of IC LM2592HVS – ADJ/NOPB (assumption of 0.900V at $25^{o}C$ due to closest current load value in the datasheet) [1]

$V_{IC\ drop,\ LM2576}$ = Ideal Dropout voltage of IC LM2576 (assumption of 0.600V at $25^{o}C$ due to closest current load value in the datasheet) [2]

$V_{IC\ out}$ = Output voltage of IC

V_P = peak voltage of the input source, V_D = Forward voltage drop of the diode

Introduction

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

- Help students 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.

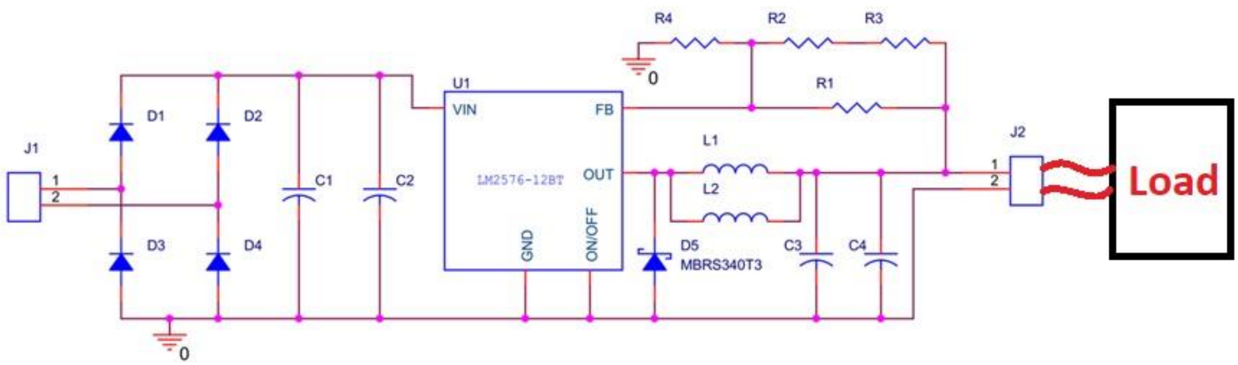

Figure 1. Real circuit to be simulated and tested

Table 1 below shows detailed component values and models.

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 consideration. Prior to having the actual circuit board designed, the simulation process should be carried out by the team members. NI Multisim 14.2 was used as a tool in this simulation process. The IC chip in the real circuit is the LM2576 model while this model is not available in NI

Multisim. The team proactively used LM2592HVS-ADJ/NOPB for its characteristics, of which 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 be evaluated at 100kHz. The reason to test the circuit of a circuit with an IC Linear Regulator at 100kHz is because this model's IC linear regulator has op-amps that compare the feedback signal and reference voltage signals to control the output current of op-amps to be smooth. Hence, produce a stable output voltage despite the changes in temperature. A higher frequency also allows the switching regulator to have smaller sizes for the inductors and capacitors, whereas a smaller inductor value reduces the ripple current, and input and output capacitance, is required [3]. The load is simulated at 298 Ohms. The simulation process was run with the simulation circuit in NI Multisim as shown in Figure 2.

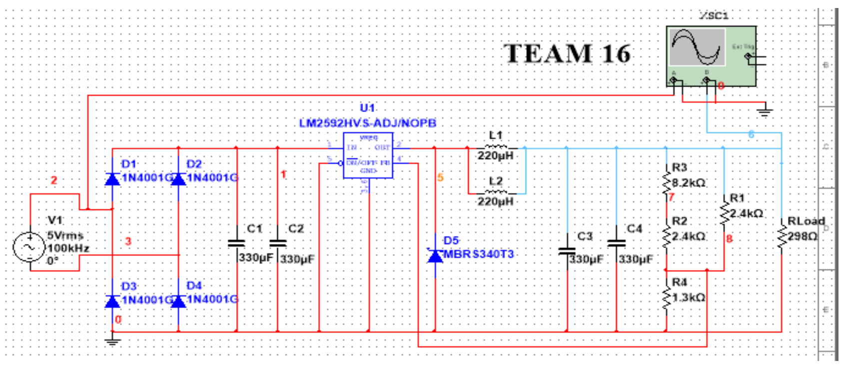

Figure 2. Simulation circuit on Multisim

The simulated results are shown in Figure 3 after clicking on the RUN button in Multisim, at the Oscilloscope display.

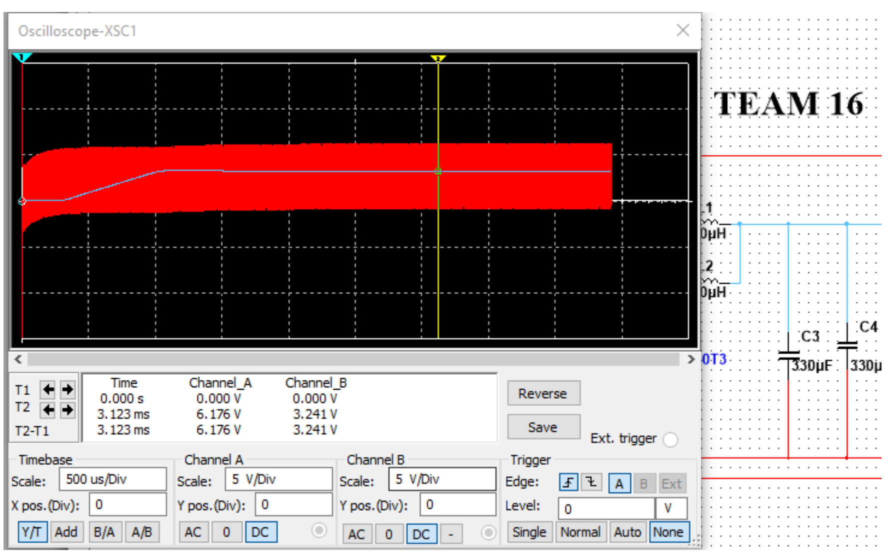

Figure 3. Simulation results on Multisim

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

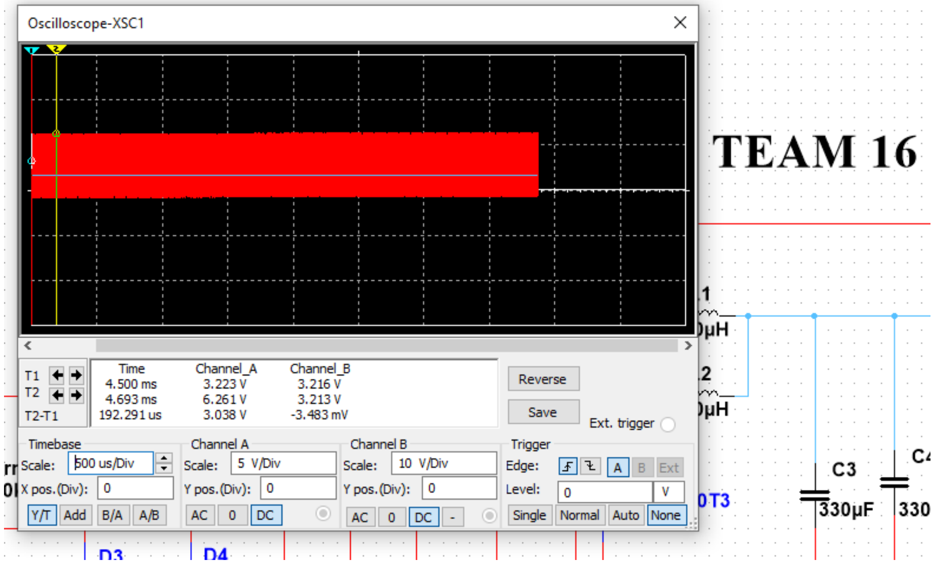

Figure 4. Simulation results analysis with cursors: cursor 1 and cursor2; on Multisim

In the design, two inductors are connected parallel in order to decrease the value of the total inductance, therefore reducing the ripple current output in the inductor at the desired instance [4]. Capacitors C1 and C2 were used for supplying a current source whenever the output suddenly draws current from the input when the switch changes, which reduces the voltage decline at that time [5], while capacitors C3 and C4 were used for achieving fast response to load transients by forming a second-order low pass filter with the inductors, in which their impedance enhances the stability of the feedback loop by minimizing the ripple noises and their most effectiveness in the frequency range that cover the ideal fixed IC's oscillator frequency [6]. Schottky Diode was placed at the output of the IC chip to reduce the forward voltage drop compared to a normal diode, which is from 0.2V to 0.3V, and to react quickly to high frequency due to its switching speed [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	3.279	V
Ripple voltage	0.138	V
Output current	11.003	mA

The Ripple voltage is evaluated as follows [1]:
$V_{rec,\ min} \geq V_{IC\ drop\ LM2592HV} + V_{IC\ out} = 0.9\ V + 3.279\ V = 4.179\ V$

$V_{rec,\ max} = V_P - V_D = 6.177\ V - 2 * 0.93\ V = 4.317\ V$

$V_r \leq V_{rec,\ max} - V_{rec,\ min} = 4.317\ V - 4.179\ V = 0.138\ V$

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
1	SCHEMATIC1 : PA	C1	CAP300
2	SCHEMATIC1 : PA	C2	CAP300
3	SCHEMATIC1 : PA	C3	CAP300
4	SCHEMATIC1 : PA	C4	CAP300
5	SCHEMATIC1 : PA	D1	DO41
6	SCHEMATIC1 : PA	D2	DO41
7	SCHEMATIC1 : PA	D3	DO41
8	SCHEMATIC1 : PA	D4	DO41
9	SCHEMATIC1 : PA	D5	DIOM7959X261N
10	SCHEMATIC1 : PA	J1	JUMPER2
11	SCHEMATIC1 : PA	J2	JUMPER2
12	SCHEMATIC1 : PA	L1	INDP125125X600N
13	SCHEMATIC1 : PA	L2	INDP125125X600N
14	SCHEMATIC1 : PA	R1	RES400
15	SCHEMATIC1 : PA	R2	RES400
16	SCHEMATIC1 : PA	R3	RES400
17	SCHEMATIC1 : PA	R4	RES400
18	SCHEMATIC1 : PA	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.

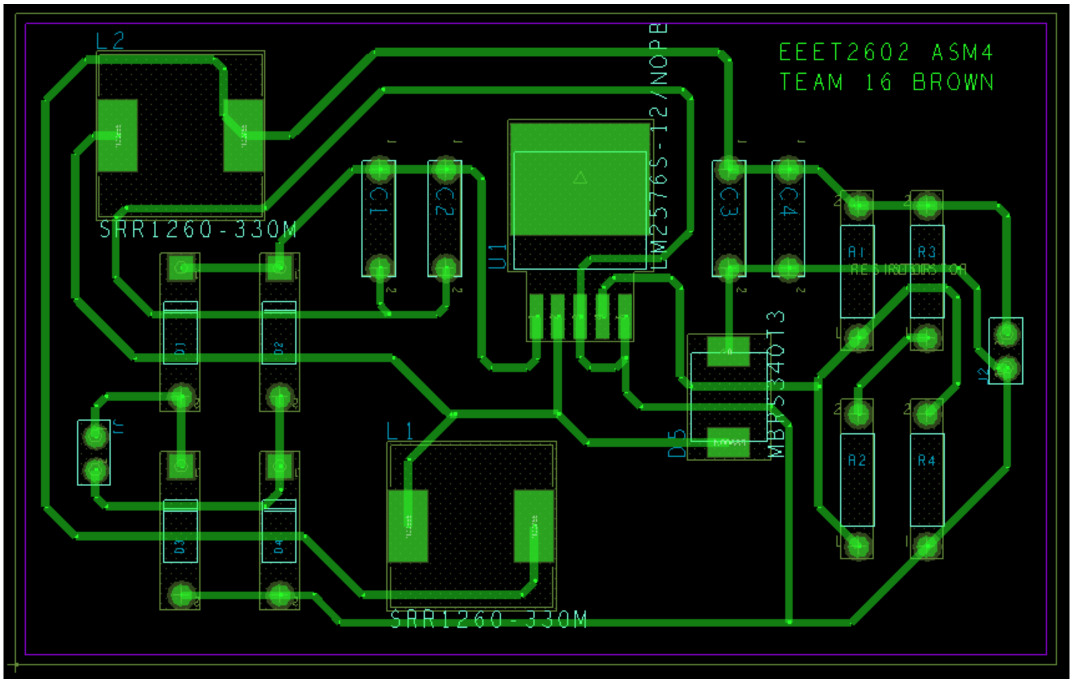

Figure 6. PCB Layout in Cadence PCB Editor design

PCB Manufacturing and Soldering Process (10pts)

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:

1. **Basic Safety Rule During Soldering Operation**

Avoid – Deny – Defense:

- **AVOID:** Go through the schematic, check the pins and PCB board carefully beforehand.
- **DENY:** At any step, if something is done wrong, all action must be stopped to prevent further risks or damage.
- **DEFENSE:** If there are risks of components explosion or fire, flee the scene. If the situation is in control, start the de-soldering process or restart work on the new PCB.

2. **Soldering Process Order**

The priority is toward soldering smaller components to larger ones. There will be two types that require a different technique of solder.

a. SMD parts: The soldering process starts with the SMD parts as it requires the Voltera machine to bake the metal parts. First, a solder paste is applied on the metal parts to prevent the components from detaching, and the order starts with the Schottky diode and is followed by LM2576, and 2 inductors according to size priority.

b. Thru-holes parts: the pins of the components are inserted through the holes and the PCB board is flipped over as the soldering process begins. This process is repeated 3 times in regard to the order of priority of size, as it starts with the Resistor, and is followed by Didoes, and Capacitor. Tic tac glues are used to stabilize the board during soldering.

3. **Problems to avoid**

- Bad Solder joints -> adjust the temperature of the soldering iron and the soldering technique (avoid cold joints, overheating, exceeded or insufficient solder paste…)
- Give enough paste thick enough (to avoid insufficient wetting, Solder-starved)
- Pad diameter (75 mil -> 90 mil) (avoid little pad) can be solved by using thin lead wire to solder, but it is highly recommended to enlarge the pad diameter.
- Thru-hole (45 mil -> 30 mil or specify holes for each component type), moreover, the design needs to re-adjust holes that fit with the pin of each component, for instance, 30 mil holes for jumpers and resistor and 45 mil holes for diodes, and capacitors.
- Suitable alloy solder wire (small hole is soldered with thinner wire; big hole works with thicker wire).
- Solder splashes: Avoid moving the iron lead when soldering. Wipe the splashes with acetone solution.

Figure 7. Common PCB Soldering Problems to Avoid. [8]

4. **PCB Components Soldering Process - Results**

We first receive the soldering iron from the instructor and strictly follow the steps below:

- Preparation: Unbox and setup the soldering iron on the workstation.
- Step 1: Inspection of screws tightness and loose part(s). A throughout check before heating is required to avoid the extreme heat hazard from the soldering iron.
- Step 2: The soldering person is well-equipped with Safety Glasses, Face Mask(s), Safety Gloves as well as Smoke Absorbers to avoid Smoke Hazard to the body.
- Step 3: Plug in and Pre-heat. The task-assigned person will plug in first, well placed the plug and the soldering iron wires in order to keep it away from the extreme heat of the soldering iron. The given iron has an adjustable temperature feature. It is also recommended by our instructor to set it at around $400°C$ for the optimal result.
- Step 4: It is also important to wet the sponge before the soldering session start. Once the iron is heated and the sponge is wet, all debris and residues of solder remaining on the iron tip from previous sessions must be wiped away.
- Step 5: When the previous steps are done, it is recommended that we picked up the soldering iron with our dominant hand and hold the solder with the other hand to best handle the tools for PCB soldering. DO NOT touch the metal part of the iron once it's heated.
- Step 6: Once all components are in place, flip the circuit board/ PCB over, and bend the components leads to 45°, hold the iron almost like holding a pen. Apply heat to the copper pad and the leads for 3-4 seconds; now use the other hand to bring the solder and apply contact with the heated joint.
- Step 7: Once the soldering process is completed, remove the solder and soldering iron to allow the solder to cool off naturally and get ready to repeat for the next components.

After finishing the PCB soldering process, Figure 8-15 below shows the different sides of the completed PCB.

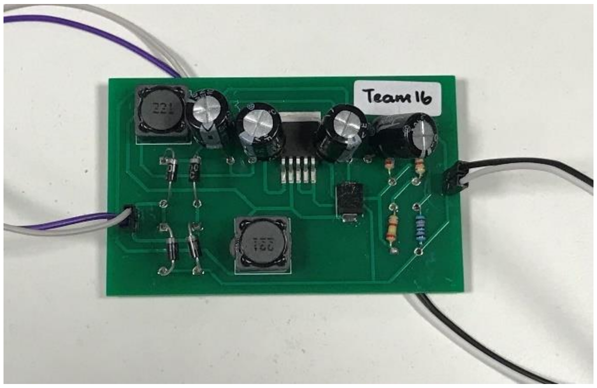

Figure 8. Completed PCB - TOP view

Figure 9. Completed PCB - BOTTOM view

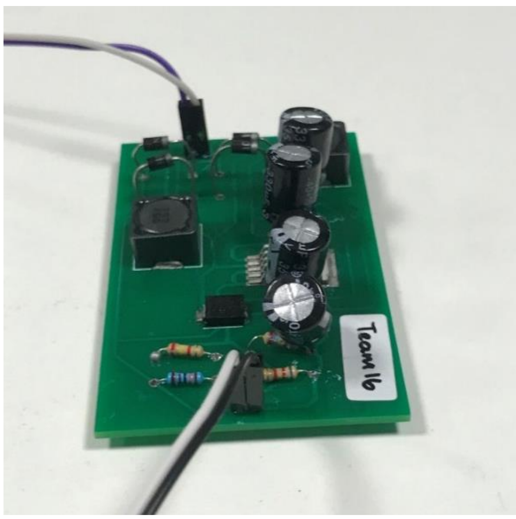

Figure 10. Completed PCB - Right TOP view

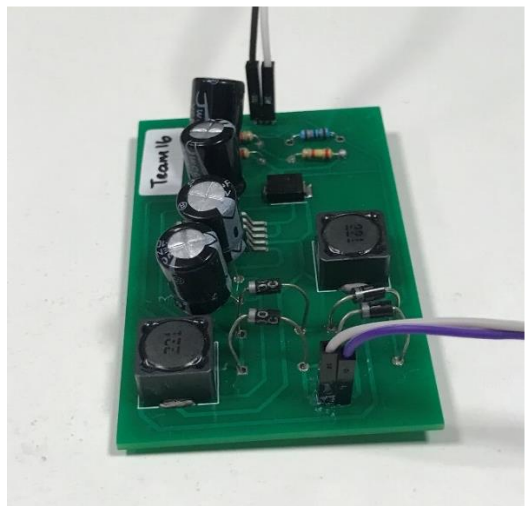

Figure 11. Completed PCB - Left TOP view

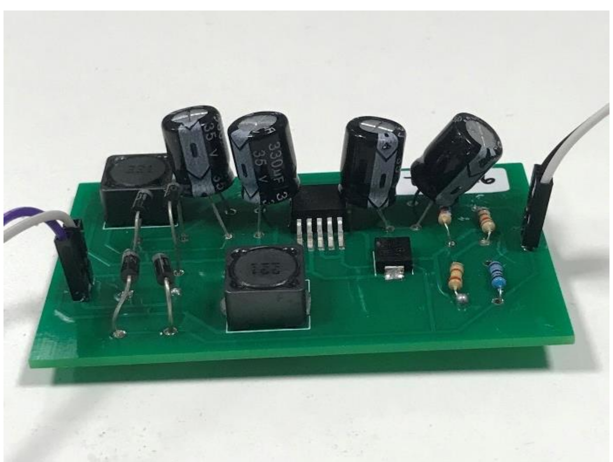

Figure 12. Completed PCB - Front TOP view

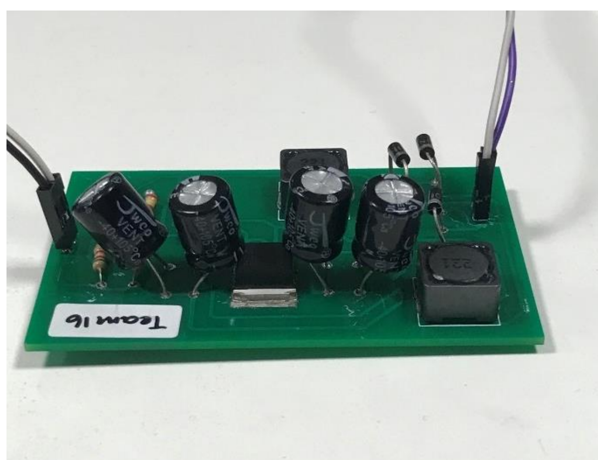

Figure 13. Completed PCB - Back TOP view

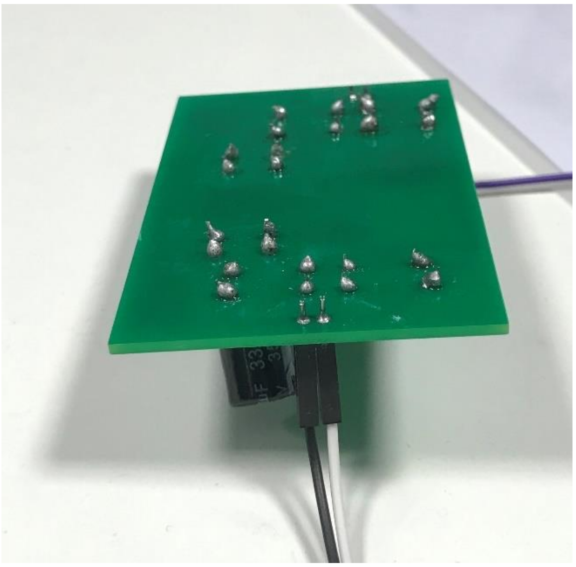

Figure 14. Completed PCB - Right BOTTOM view

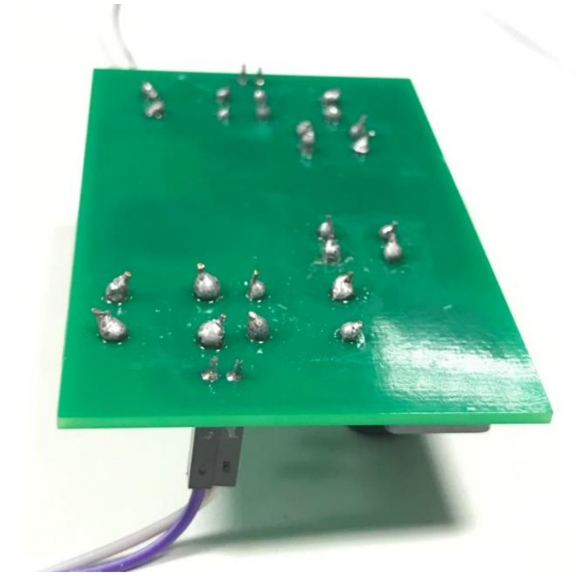

Figure 15. Completed PCB - Left BOTTOM view

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 a 298-Ohm resistor. The circuit was setup based on the schematic shown in Figure 1. The setup of the circuit test is shown in 16.

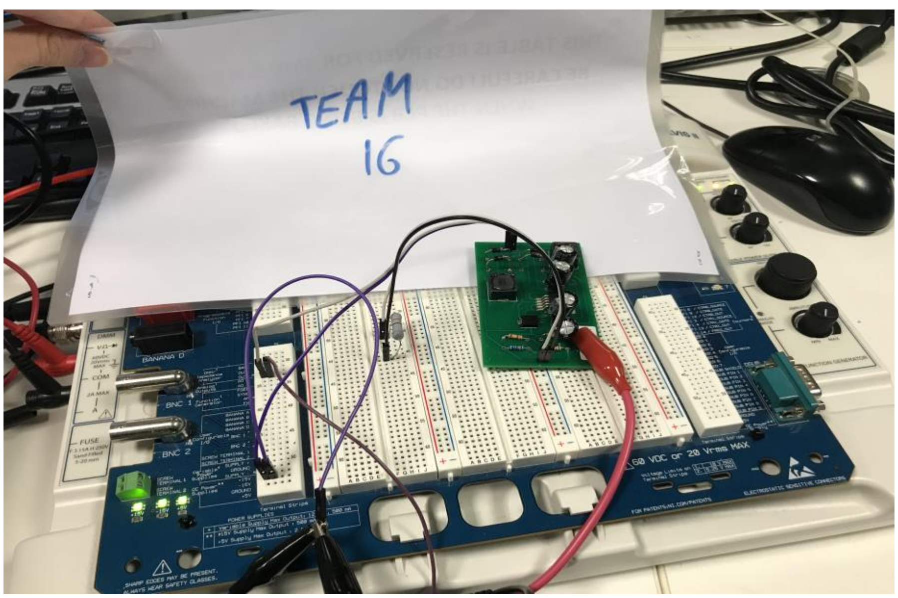

Figure 167. Actual circuit test setup

Circuit Test Data Collection

The actual tested results are shown in 17.

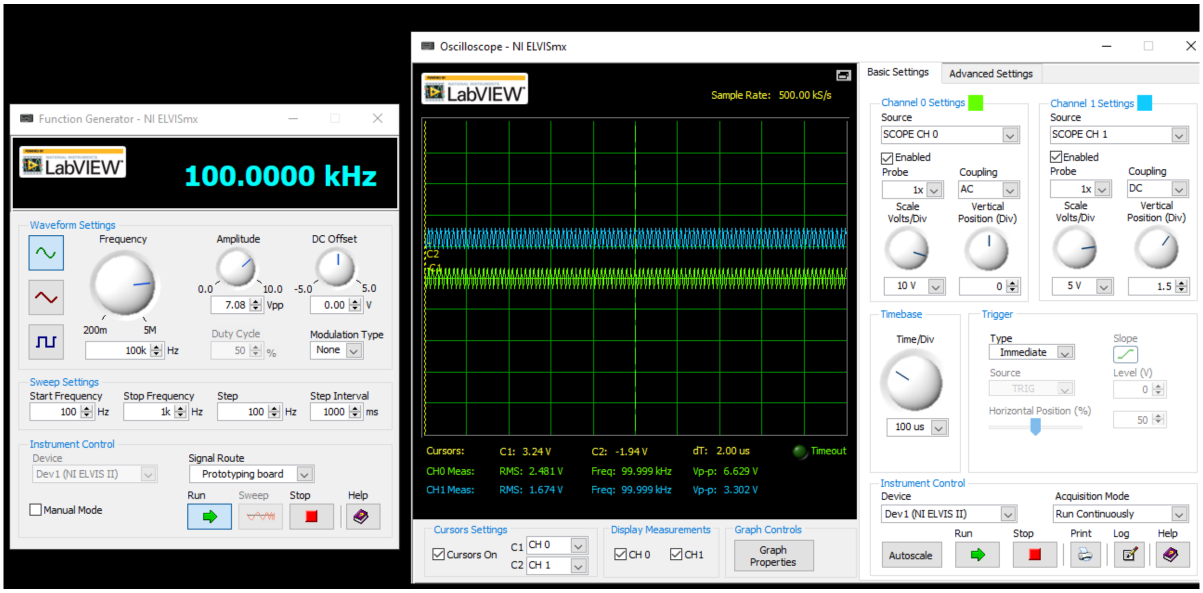

Figure 178. Actual measured waveform

Data from actual experiment was recorded into Table 3.

Table 3. Actual Measured Data Summary

Data name	Value	Unit
Output voltage	3.302	V
Ripple voltage	0.912	V
Output current	11.081	mA

The ripple voltage is measured as follows [2]:

Explain VIC in is not enough 10 => fail to produce 5V VIC out

$V_{rec,\ min} \geq V_{IC\ drop,\ LM2576} + V_{IC\ out} = 0.600\ V + 3.257\ V = 3.857\ V$

$V_{rec,\ max} = V_P - 2V_D = 6.629\ V - 2 * 0.930\ V = 4.769\ V$

$V_r \leq V_{rec,\ max} - V_{rec,\ min} = 4.769\ V - 3.857\ V = 0.912\ V$

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 difference in the IC chips for the cases, whereas LM2592HVS was used in the simulation while LM2576 was soldered on the PCB board. These 2 IC chips are different in their dropout voltage and fixed frequency to produce ideal voltage output.

Moreover, compared to what the team designed in Assessment 2, the similarity between the circuits can be listed as both designs generate a linear DC waveform of the load voltage via the implementation of the input capacitor and the diode bridge. These 2 implementations convert the AC voltage from the input source to the DC-like voltage (figure 4).

While there are several differences between them, such as the electronic devices used to regulate the rectified voltage which reduces the ripple voltage. For instance, the NPN junction BJT transistor and Zener diode were used to maintain a constant voltage output. On the other hand, IC chip, inductors, output capacitors, and Schottky diode are applied in this design. Unlike the BJT Zener voltage regulator design, the declining pattern of the load voltage is observed while carrying the simulation in the linear IC regulator circuit. Moreover, the linear regulator design works with higher frequency compared to the BJT Zener diode regulator circuit. Overall, the linear voltage regulator circuit is more complex than the BJT Zener diode circuit

Those differences can be explained as the linear IC regulator design witness heat dissipation that causes sharp voltage ripple as the output voltage is dropped. Therefore, applications of inductors work as energy storage devices to cover the loss in voltage while output capacitors stabilize the voltage by reducing the voltage ripple and filtering out the noises [4]. The larger frequency in the linear regulator circuit compared to the BJT is a concern as it affects the performance of the overall converter, whereas determining the values and sizes of capacitors and inductors, as well as the transient response from the load. Thus, referring to the frequency and component selections to the datasheet of the IC chip is vital to adjust the desired performance of the design circuit.

References

[1] Texas Instruments, "LM2592HV Simple Switcher Power Converter 150-kHz 2-A Step-Down Voltage Regulator", Texas Instruments Incorporated, Dallas, Texas, 2001. [Online]. Available at: https://pdf1.alldatasheet.com/datasheet-pdf/view/837299/TI1/LM2592HVS-ADJ/NOPB.html (Accessed: 19 Sep 2022)

[2] Texas Instruments, "LM2576/LM2576HV Series Simple Switcher 3A Step-Down Voltage Regulator", Texas Instruments Incorporated, Dallas, Texas, 1999. [Online]. Available at: https://pdf1.alldatasheet.com/datasheet-pdf/view/543767/TI1/LM2576.html (Accessed: 19 Sep 2022)

[3] Texas Instruments, “AN-1973 Benefits and Challenges of High-Frequency Regulators”, Texas Instruments Incorporated, Dallas, Texas, 2009. [Online]. Available at: https://www.ti.com/lit/an/snva399a/snva399a.pdf (Accessed: 18 Sep 2022)

[4] C. Glaser, “Power Management - Does Inductor Ripple-Current Percentage Still Matter in Low Power Step-Down Converters?”, Electronic Design.com. Available at: https://www.electronicdesign.com/power-management/article/21801119/does-inductor-ripplecurrent-percentage-still-matter-in-lowpower-stepdown-converters (Accessed: 18 Sep 2022)

[5] “Supplement - Selection of Inductors and Capacitors for DC/DC Converters”, *TechWeb.com*. Available at: https://techweb.rohm.com/knowledge/dcdc/dcdc_pwm/dcdc_pwm02/8267 (Accessed: 18 Sep 2022)

[6] Texas Instruments, “Input and Output Capacitor Selection”, Texas Instruments Incorporated, Dallas, Texas, 2006. [Online]. Available at: https://www.ti.com/lit/an/slta055/slta055.pdf?ts=1663512764994&ref_url=https%253A%252F%252Fwww.google.com%252F (Accessed:18 Sep 2022)

[7] V. Manev, H. Visser, P. Baltus and H. Gao, "A Comparison of Tunnel Diode and Schottky Diode in Rectifier at 2.4 GHz for Low Input Power Region," 2019 IEEE Wireless Power Transfer Conference (WPTC), 2019, pp. 274-277, doi: 10.1109/WPTC45513.2019.9055615. [Online]. Available at: https://ieeexplore.ieee.org/stamp/stamp.jsp?tp=&arnumber=9055615&isnumber=9055510 (Accessed: 19 Sep 2022)

[8] Helen, "13 Common PCB Soldering Problems to Avoid," Seeedstudio, 2021. [Online]. Available: https://www.seeedstudio.com/blog/2021/06/18/13-common-pcb-soldering-problems-to-avoid/. (Accessed: 19 Sep 2022)