Dylan Falzone
9/12/2023
4539-8897

EE Design 1 Technical Report
Power Supply Design

Introduction

In our daily lives we are surrounded by electronics. They are in our homes, offices, restaurants and even our pockets. Most of these electronics serve different purposes, seemingly having nothing in common. For example, a phone is nothing like a crock pot, and a crock pot is nothing like a car. Yet, despite the multitude of differences between them, every single electronic device has one thing in common. Somewhere inside of it is some type of power supply circuitry.

These power supply circuits are essential to the functioning of electronics, as they allow for consistent and reliable flow of energy into the circuit in the form of electricity. Without them, circuits would face a number of problems. Battery powered electronics would be unreliable and would be impossible to recharge. And even more severe, electronics that draw power from wall outlets would need larger, more expensive components designed to work with 120 volts. As such, in order to see why these power supply circuits are so essential, in this lab I will construct a simple power supply circuit capable of stepping down a 120V AC source to a stable 5V DC voltage.

Methods

Stages of a Power Supply Circuit

While the exact nature and composition of a power supply circuit may vary, our method and the method that is most common involves 4 key stages.

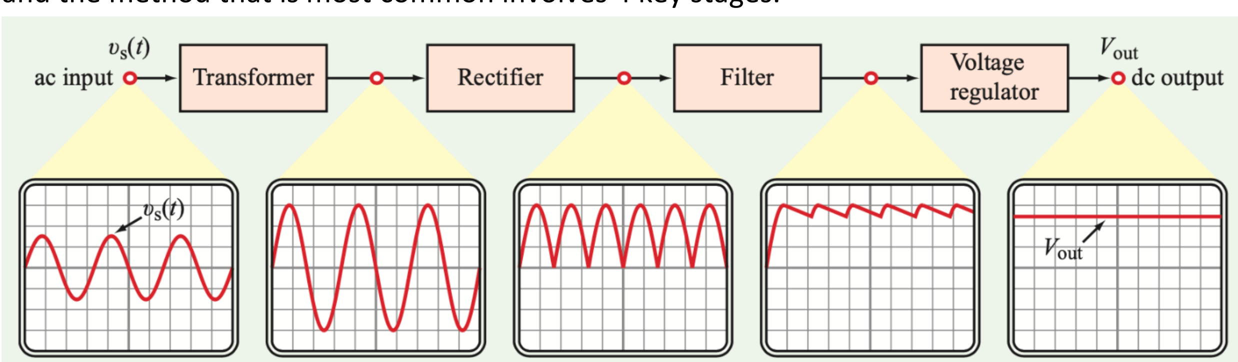


Figure 1: The 4 stages of the power supply system. [1]

As shown in figure 1, we begin with the transformer. Transformers are a marvel of engineering and physics that allow for voltages to be stepped down substantially while maintaining power conservation. With this stage, the 120V wall outlet becomes a mere 12.7V. This massive step down gives us a voltage that is in the right ballpark but must still be processed before use.

Stage 2 is the rectifier. What happens here is simpler. The sinusoidal voltage, rapidly changing from positive to negative, must be forced into a solely positive state. Dips in the negative direction must be made opposite, to mimic those in the positive direction. This is done by a full bridge rectifier, comprised of 4 diodes arranged in a diagonal pattern, end to end [2]. After this stage, we are left with an always positive, but still high and varied voltage.

Stage 3 is the filter. This is perhaps the most complex of the stages. This stage removes the majority of the ripples from the signal, outputting something much more linear, but not totally flat. This is achieved by a capacitor (C) and a resistor (Rs). The capacitor charges and discharges, reducing the variation in the voltage, while the resistor forms a voltage divider with the load, bringing down our overall voltage to the desired level. Due to the importance of this stage, the values of the components had to be chosen precisely.

$$V_r = \frac{[(V_{s1} - 1.4) - V_z]T_{rect}}{R_s C} * \frac{(R_z||R_L)}{R_s + (R_z||R_L)}$$

$Equation\ 1: An\ expression\ for\ the\ peak - to - peak\ ripple\ voltage. [1]$

The above equation was referenced, but not strictly used to gain understanding of the relationship between the capacitor and the resistor of the filter and their impact on ripple voltage, which was to be minimized. Combined with some guess-and-check, I was able to correctly determine working values for the components, which will be discussed in the following questions. These values brought down the ripple voltage significantly, readying the voltage for the final step.

The final stage is the rectifier. This part of the circuit aims to totally flatten out the voltage over the load, giving us a nearly perfect DC 5V. This can be accomplished in a variety of ways, but for this lab a reverse-biased Zener diode and an IC were used. The diode was chosen so that it would go into breakdown at 5.1V, activating whenever the voltage got too high. The IC functioned by other means that are beyond the scope of this report, though the difference in its capabilities will be discussed in a later section. Each of these methods completed the process, regulating the voltage to a clean 5V DC.

Zener Diode Regulated Circuit

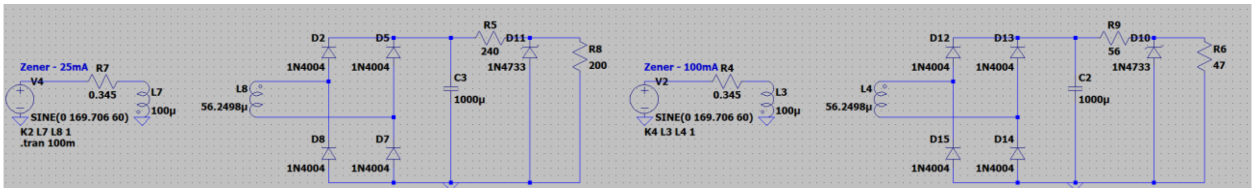


Figure 2: LTSpice schematic of the circuit using a Zener Diode as the regulator. The circuit on the left produces a 25mA load current while the right produces a 100mA load current (in theory).

Figure 2 shows the final schematic in all of its power-supplying glory. From left to right, I will quickly go over component specifics and why they were chosen. Firstly, the transformer values were found by using the following equation.

$$V2 = \sqrt{\frac{L2}{L1}} * V1$$

Equation 2: Transformer voltage equation. [3]

This allowed us to find the correct inductances for our desired input and output voltages. The resistor was mostly done by guess-and-check, since finding the internal resistance of the real transformer was beyond the scope of this lab. These transformer values were used for all other circuits in this lab so this explanation will not be repeated.

Next, the values for the filter were chosen as discussed previously. For the 25mA load current circuit, they were determined to be 1000uF and 240 Ohms respectively. Since the resistor formed a voltage divider with the load, the ratio between the two was mostly preserved for the 100mA load current circuit as well. Applying this ratio, Rs was found to be 56 Ohms while the capacitor remained the same. The only real difference between the two circuits can be easily explained by V=I*R. In order to achieve a larger load current, we simply dropped the resistance.

IC Regulated Circuit

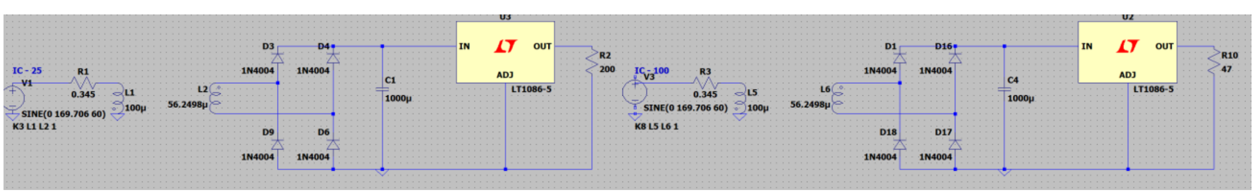


Figure 3: LTSpice schematic of the circuit using an IC as the regulator. The circuit on the left produces 25mA load current while the right produces a 100mA load current.

The practical circuit used LM7085T IC, however, for the sake of the schematic a LT1086-5 was used. This component was placed into the circuit instead of the Zener diode, in the same place, with a connection to the filter, the load, and ground. Much of the circuit remained the same, but notably, the filter stage did change. Due to the internal resistance of the IC, keeping the Rs Resistor proved to be too much, lowering the voltage significantly below its intended value. This was simply removed, giving us the circuits above. The load resistances in each configuration also remained the same as their diode counterparts.

Results

Zener Regulated Circuit Outputs

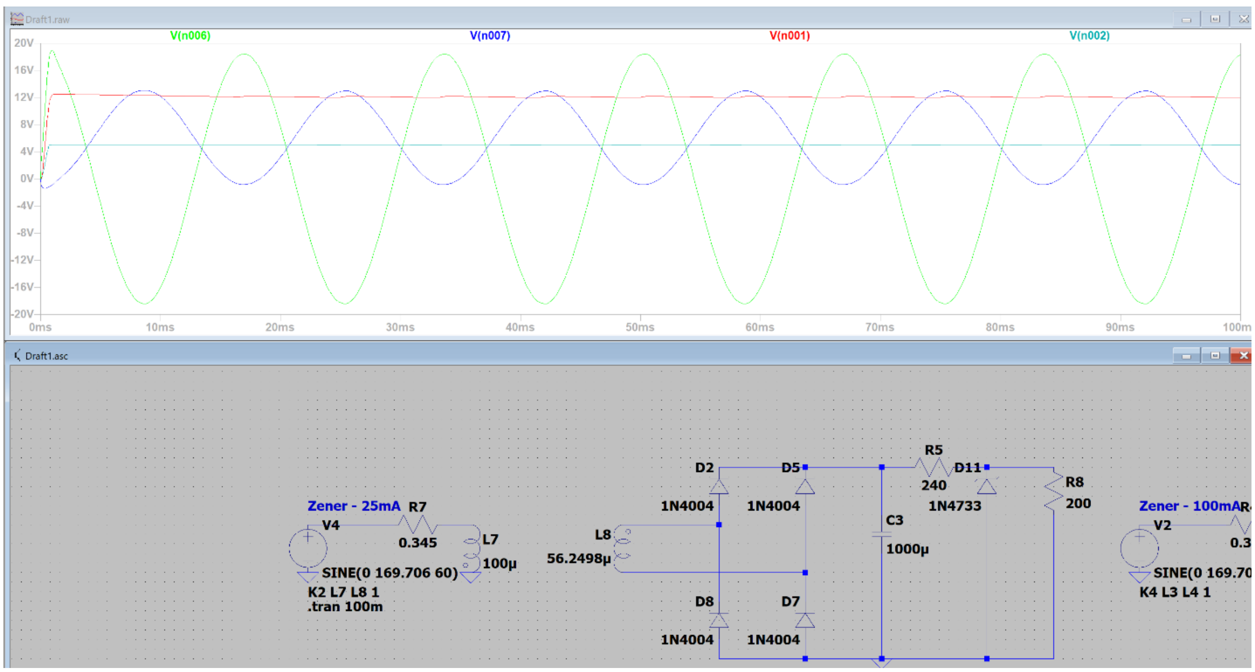
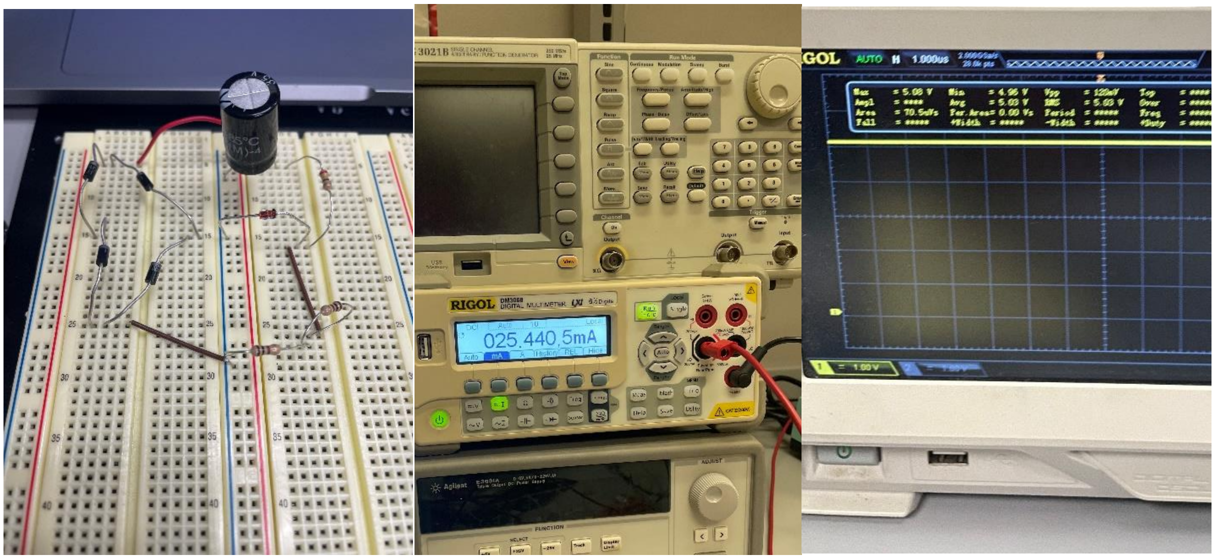


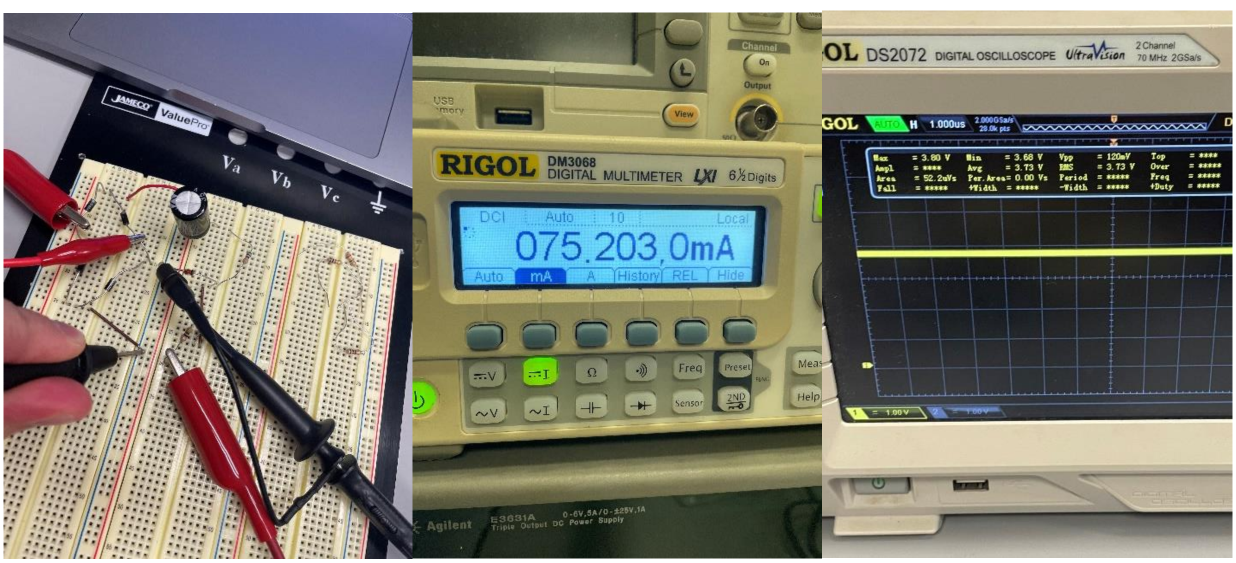
Figure 4: Zener 25mA spice circuit voltage output at each stage.

The spice schematic for the 25mA Zener behaved just as expected. Voltages at each stage corresponded to what would be expected, following the stages in figure 1.



Figures 5-7: Zener 25mA breadboard: circuit, load voltage, load current.

The breadboard implementation of the 25mA Zener also behaved just as expected. Voltages at the output of each node matched their spice counterparts and the output voltage overall was a nearly perfect 5V dc with a 25mA line current. Notably, since there were no 200ohm resistors in the lab kit, I used two 100ohm resistors in series to simulate the correct resistance.



Figures 8-10: Zener 100mA breadboard: circuit, load voltage, load current.

This is where the need for an IC became clear. The Zener was unable to reach 5V 100mA on the load. This was because as the current through the load increased, the current through the Zener became too great, exceeding its capabilities. It lost its ability to regulate the voltage effectively, causing the load voltage to decrease overall. This was not a problem with the spice schematic and was only seen in the breadboard implementation. It is for this reason that an IC needed to be used in this lab so that we could reach 100mA load current.

IC Regulated Circuit Outputs

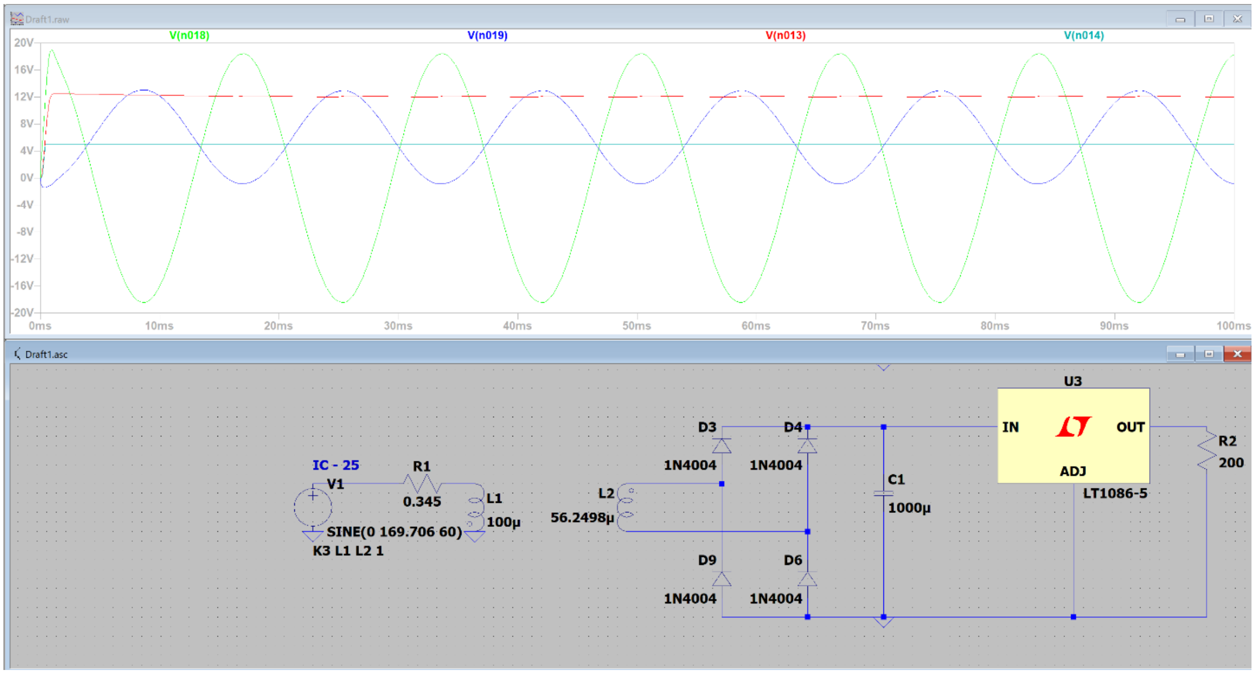
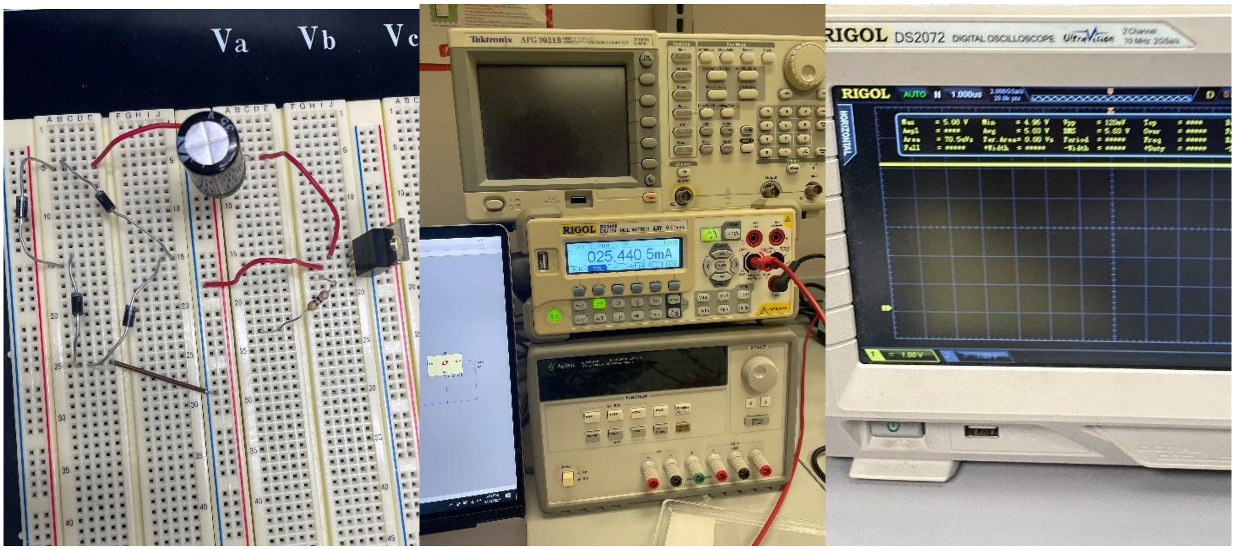


Figure 8: IC 25mA circuit voltage output at each stage.

The spice schematic for the IC behaved as expected, matching the stages exactly as in the previous configuration and as in figure 1.



Figures 11-13: IC 25mA breadboard: circuit, load voltage, load current.

The breadboard implementation of the 25mA IC also worked as expected. The voltages at each node very closely matched those in spice, and the outputs were perfect. Despite the fact that the spice component was not the same as the one used in the breadboard, it was a close enough match to achieve the exact same effect.

Figures 14-16: IC 100mA breadboard: circuit, load voltage, load current.

The 100mA circuit also worked as expected, achieving the correct voltages and currents throughout and on the outputs as pictured. However, there was one difference with this circuit. There was a substantial amount of movement on the voltage at the output. It varied between 4.6-5V instead of maintaining 5V. These oscillations were incredibly rapid, as seen in the image of the oscilloscope (Figure 16). In order to fix this, I added a secondary 10uF capacitor to the output of the circuit, as pictured in Figure 14. This did not fully eliminate the oscillation, but it did reduce it to a range of 4.8-5V instead. Attempts to add more capacitance on the input or output of the circuit did not help achieve a cleaner output, so I kept the single additional capacitor, as I felt its impact was enough to display that the circuit worked as intended.

Conclusion

Through first-hand planning, simulation, and experimentation, we were able to see why power supply circuits are so crucial to modern circuitry. Stepping down a voltage by massive amounts, rectifying it, changing its shape, and turning it into something we can use to power any circuit we desire. We have learned to manipulate energy, the lifeblood of electric circuits, bending it into whatever shapes we desire by using math, physics, and the occasional guess-and-check. That is the power of the power supply circuit.

References

[1] F. T. Ulaby, M. M. Maharbiz, and C. Furse, P450,432 in Circuit analysis and Design, Ann Arbor, MI: Michigan Publishing, 2018

[2] Last Minute Engineers, “The full-wave bridge rectifier,” Last Minute Engineers, https://lastminuteengineers.com/the-full-wave-bridge-rectifier/ (accessed Sep. 13, 2023).

[3] E. Haustveit, “Transformer and inductor design,” Switchcraft, https://www.switchcraft.org/learning/2016/12/10/transformer-and-inductor-design (accessed Sep. 13, 2023).