University of Illinois at Urbana-Champaign
Department of Electrical and Computer Engineering

ECE 343: Electronic Circuits Lab

Project : ***Design of AC-DC power supply + Voltage regulator***

Learning Objectives:

- Circuit design using nonlinear circuit elements.
- Making and justifying design choices based on requirements.
- Simulate designed power suppy on LTSpice
- Evaluate performance parameters and compare with design requirements
- Build a AC-DC power supply
- Solder circuit on a PCB
- Compute DC-DC conversion efficiency of three circuits- voltage dividers, Zener diode based DC-DC conversion (ECE 110), Power supply designed in this lab (ECE 343)
- Compare DC-DC conversion efficiency of designed voltage regulator with DC-DC conversion using a boost/buck converter (ECE 469)

1 Components Required

- **Breadboard**
- **Resistors**, **Capacitors:** Based on design.
- **Diodes:** D1N750 (Zener Diode), D1N 4001/4002 (Rectifier Diodes).
- **BJT:** 2N3055, **OpAmp:** LM741

2 Introduction

In this lab we will design an AC/DC power supply. This project will be completed in three lab sessions and will call upon your knowledge of nonlinear circuit elements like diodes and the ability to make justified design choices. The design makes use of two different kinds of diodes - the basic silicon PN junction device, used as a rectifier in this case, and the zener diode, used to regulate the output DC voltage. Figure 1 shows the block diagram of a basic DC power supply.

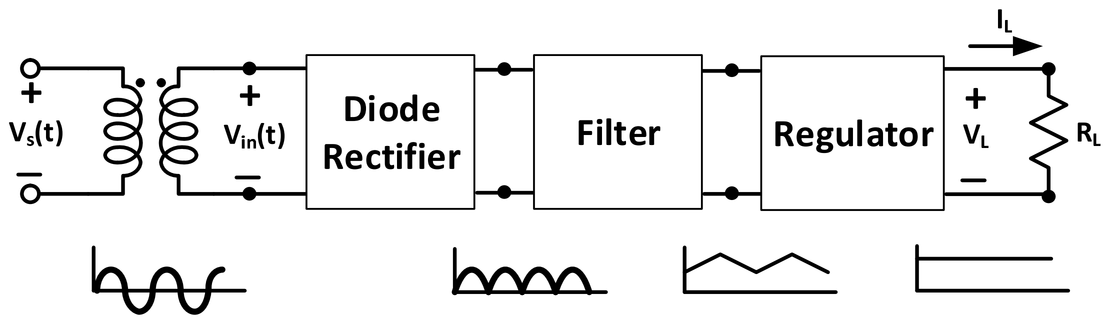


Figure 1: Block diagram of DC power supply - (Adapted from Microelecronic Circuits, Sedra & Smith)

The various components of the DC power supply are:

- **Transformer:** Transfers electrical energy from one circuit to another. It consists of two coils (primary and secondary) wound on a magnetic core. The primary winding is connected to the **120** V (RMS) AC mains and the secondary winding is connected to the rectifier circuit.
- **Diode Rectifier:** The rectifier circuit converts the input AC signal V_s to a one sided signal **(Lab 2)**.
- **Filter:** The output of rectifier is passed through a filter block to reduce ripple in the signal.
- **Zener-diode based regulator:** Used to further reduce the ripple in the voltage and to give a stable DC output voltage.

3 Design Specifications:

The power supply will be designed to meet the following specifications,

- **DC open circuit output voltage = 4.7V .**
- **Open circuit output voltage stays within 2 % of the desired voltage as AC line voltage varies from 115 Vrms to 125 Vrms.**
- **Ripple voltage at output is less than 2 % of open circuit output DC voltage.**
- **The output current can vary from 0 to 20 mA.**

4 Design Walk-through

In this section we will design the different blocks shown in Fig. 1. For each section of the power supply circuit you have some decisions to make. Your choice of filter capacitor, rectifier circuit, and transformer configuration are all affected directly or indirectly by the choice of output voltage (2V in this case).

4.1 AC power source and rectifier

For this design we will use the sine wave source for LTSpice simulations. The actual circuit will be tested using a transformer connected to AC mains. The input sine wave to the power supply is rectified using a rectifier as shown below:
The rectifier circuit has two possibilities: (1) Half-wave rectifier (Fig. 2 (a)) and Full-wave bridge rectifier(Fig. 2 (b)).

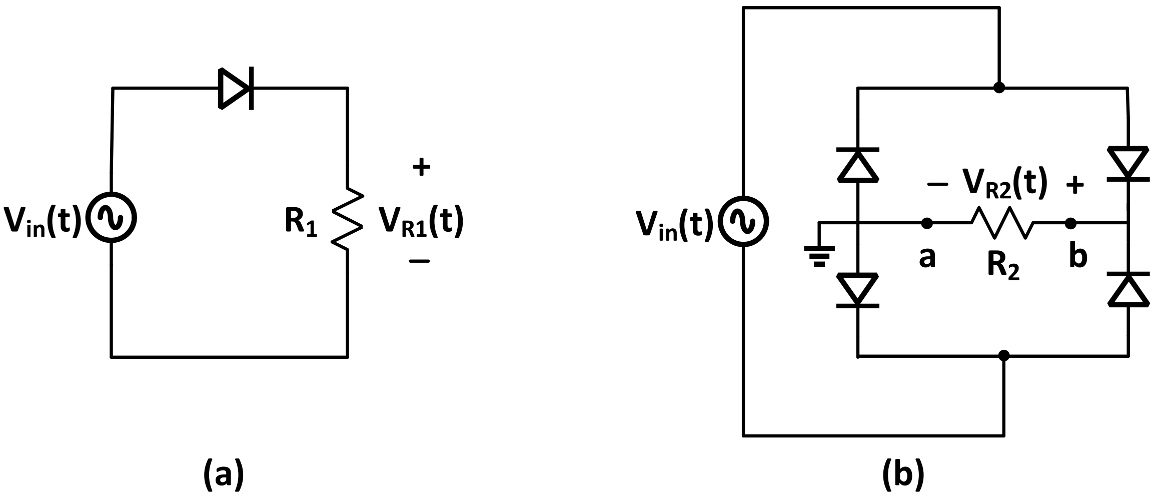
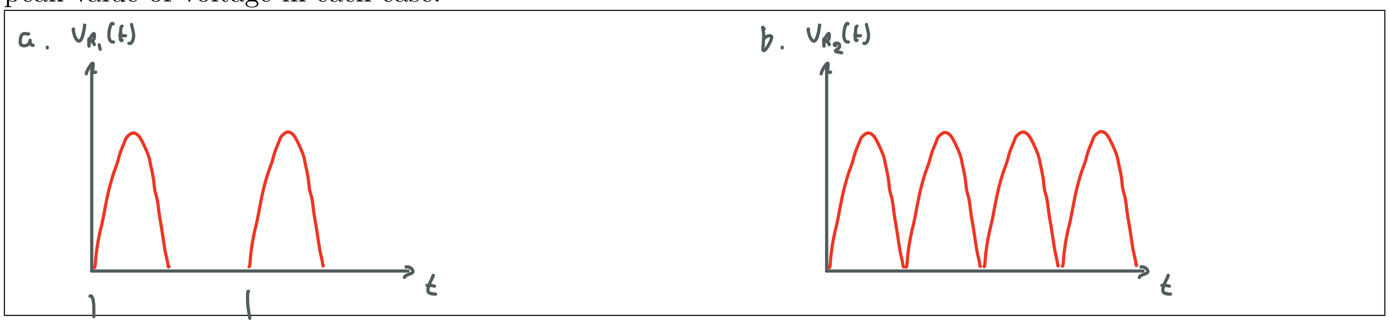


Figure 2: Output of the rectifier circuit (Bridge Rectifier).

- Give a sketch of voltage across resistors $\mathbf{R_1}$ and $\mathbf{R_2}$. Assume $\mathbf{V_{in}(t) = 25\sqrt{2}sin(2\pi 60t)V}$. Clearly indicate the peak value of voltage in each case.



- What are the advantages/disadvantages of each circuit?

(a) takes less area while (b) is continuous and doesn't "waste" energy.

- The average voltage value, $< \mathbf{V} >$, of voltage $\mathbf{V(t)}$ is given by,

$$< \mathbf{v} > = \frac{\mathbf{1}}{\mathbf{T}} \int_{\mathbf{0}}^{\mathbf{T}} \mathbf{v(t)dt}$$

  Compute the average value of input voltage $\mathbf{Vin(t)}$, half wave rectified output $\mathbf{V_{R1}(t)}$, and full wave rectified output $\mathbf{V_{R2}(t)}$.You may assume the rectification is Ideal and there is not voltage drop across the diodes.

a.

$f = 60\,Hz$

$25\sqrt{2}\sin(2\pi 60 t)$; $T = \frac{1}{60}\,s$

$$\boxed{\langle v\rangle_a =} 60\int_0^{60} V_{R_1}(t)\,dt$$

$$= 60\left[\int_0^{1/120} V_{R_1}(t)\,dt + \cancelto{0}{\int_{1/200}^{1/60} V_{R_1}(t)\,dt}\right]$$

$$= 60\int_0^{1/120} 25\sqrt{2}\sin(2\pi 60 t)\,dt$$

$$= \frac{\cancel{60}\cdot 25\sqrt{2}}{2\pi \cancel{60}}\left(-\cos(2\pi 60 t)\Big|_0^{1/120}\right)$$

$$= \frac{25\sqrt{2}}{2\pi}\left(-\cos\left(2\pi \frac{60}{120}\right) + \cancelto{1}{\cos(0)}\right)$$

$$= \frac{25\sqrt{2}}{2\pi}\left(-\cancelto{-1}{\cos(\pi)} + 1\right)$$

$$= \boxed{\frac{25\sqrt{2}}{\pi}\ [V]}$$

b.

$f = 120\,Hz$ $T = \frac{1}{120}\,s$

$$\boxed{\langle v\rangle_b =} 120\int_0^{1/120} V_{R_2}(t)\,dt$$

$$= 120\int_0^{1/120} 25\sqrt{2}\sin(2\pi 60 t)\,dt$$

$$= \frac{120\cdot 25\sqrt{2}}{2\pi 60}\left(-\cos(2\pi 60 t)\Big|_0^{1/120}\right)$$

$$= \frac{25\sqrt{2}}{\pi}\left(-\cancelto{-1}{\cos(\pi)} + \cancelto{1}{\cos(0)}\right)$$

$$= \boxed{\frac{50\sqrt{2}}{\pi}\ [V]}$$

4.2 Regulator

The regulator is simply a resistor and zener-diode (Fig. 3), whose function is to hold the output voltage, V_0, constant against variations in the load current and against variations in the AC line voltage. It also serves to reduce the ripple voltage that appears across the filter capacitor and would otherwise appear at the power supply output V_0 as well.

4.3 Regulator Design

The regulator design mainly involves picking an appropriate **operating point** for the zener diode. The basic configuration of regulator is shown in Fig. 3.
While designing the following considerations must be kept in mind:

1. Too much current and the diode overheats.

2. Too little and you run onto the knee of the I-V curve and lose regulation.

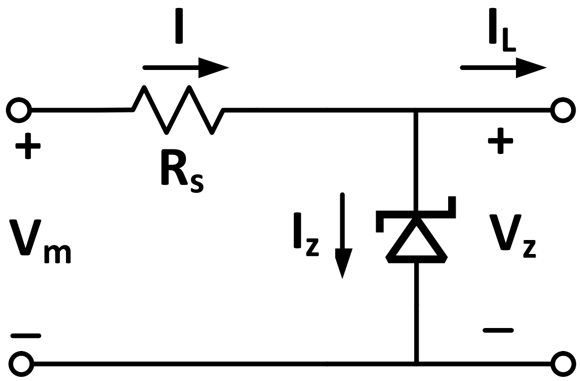
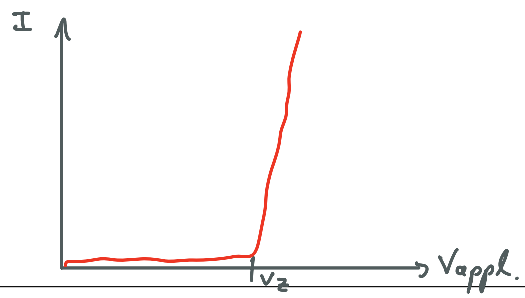


Figure 3: Regulator circuit.

3. The resistor $\mathbf{R_s}$ limits maximum current that can flow through the zener diode. This condition occurs wnen the load current $\mathbf{I_L = 0}$.

Answering the following questions will help you design the regulator circuit:

- Sketch the $\mathbf{I - V}$ characteristic of a reversed biased zener diode.



- Based on your sketch give a short explanation of how reversed biased zener diode can function as a regulator. Also mention which part of the $\mathbf{I - V}$ curve should the diode be operated to operate as a regulator.

If $V_{appl} \geq V_z$, then the voltage across the diode Vd will remain constant to a point. Therefore, if we are sufficiently above V_z such that negative noise doesn't cause $V_m < V_z$ and positive noise doesn't cause overheating, then $V_d = V_z$ constantly.

- The zener voltage $\mathbf{V_z}$ is fixed by the DC supply output voltage. What is the value of $\mathbf{V_z}$ for this design.

4.7V

- The part number of zener diode used in this lab is **D1N750**. Refer to the device data sheet and note down the max power rating of the diode. Use **75%** of the maximum power to compute the maximum current that the diode can tolerate (This condition occurs when load current $\mathbf{I_L = 0}$).

Max Power: 500 mW

75% Max Power: 375 mW

$P = I V_z \Rightarrow \boxed{I_z = \frac{P}{V_z} = 79.8\,mA}$

- What is the value of $\mathbf{R_s}$ required to ensure that current through the diode does not exceed the maximum allowed value. Assume $\mathbf{V_m = 25\sqrt{2}V}$.

$V_m - 4.7 = I R_s \Rightarrow \boxed{R_s} = \frac{V_m - 4.7}{I} \boxed{= 384.15\,\Omega}$

- The current $\mathbf{I_z}$ though will vary based on the load current ($\mathbf{I_L}$) that is drawn from the regulator. What is the minimum value of current $\mathbf{I_z}$.

$I = I_z + I_L$

$I_z = I - I_L = 79.8\,mA - 20\,mA = 59.8\,mA$

- At this point you must verify that the zener diode operates in the correct region of operation. Perform a DC sweep simulation of the circuit shown in Fig. 4 You can sweep voltage $\mathbf{V_{in}}$ from $\mathbf{4.4 - 4.8V}$. Plot $\mathbf{I_z}$ **vs** $\mathbf{V_{in}}$ and verify that the minimum current through the zener diode is above the knee of the $\mathbf{I_z}$ **vs** $\mathbf{V_{in}}$ curve.

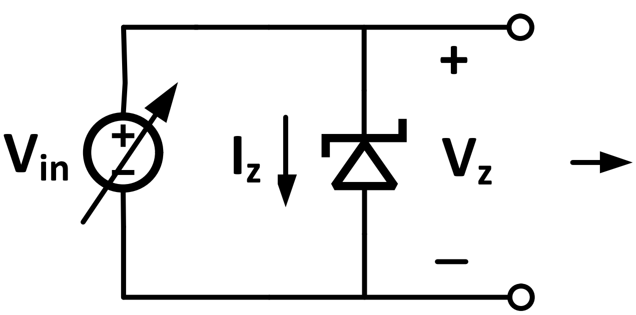


Figure 4: Zener Diode Simulation Circuit

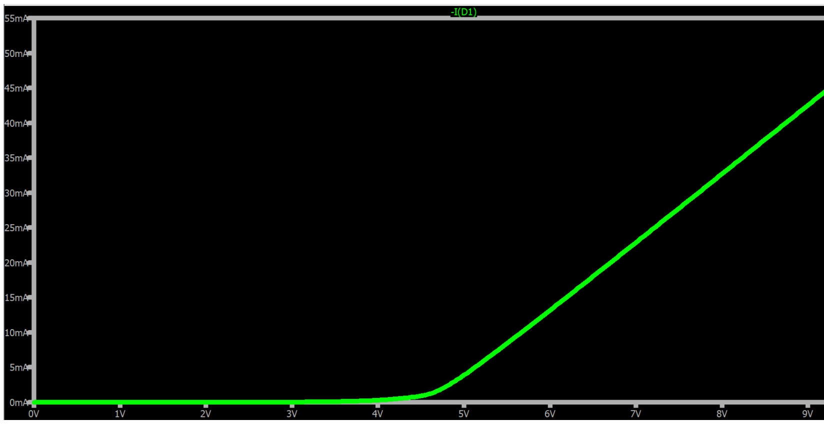


Figure 5: Simulation Output

4.4 Filter

The filter block is used to reduce the ripple in the rectifier output. The filter section is **only a single capacitor** connected from the output of the rectifier circuit to ground (Fig. 6). The following points illustrate the role of the capacitor:

- The capacitor provides current to the load between peaks of rectified output.
- Between peaks of rectified voltage, the filter capacitor is discharging at a rate that depends upon the amount of current delivered to the regulator
- The **ripple voltage**, ($V_{ripple,C}$) **across the capacitor (C)** can related to the value of the capacitance as follows,

$$\mathbf{V_{ripple,C} = \frac{I_L}{fC}},$$

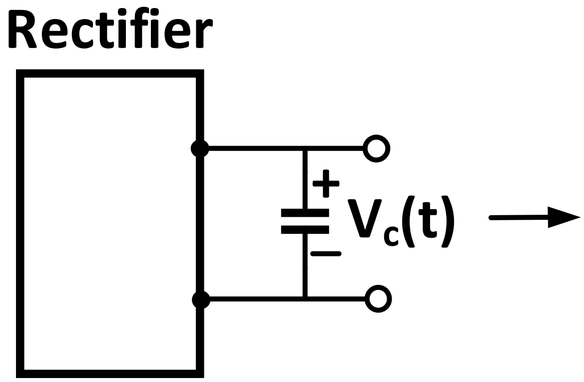


Figure 6: A capacitor filter

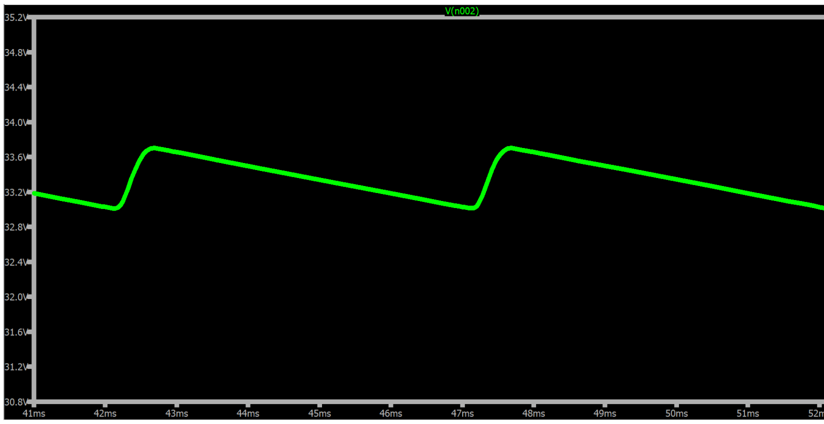


Figure 7: Filter Output

where, $\mathbf{I_L}$ is the DC component of the load current and $\mathbf{f}$ is the frequency of the filter output.

- **In order to use the expression to compute C, the output ripple voltage must be converted to the ripple across capacitor**.

4.4.1 Picking the Capacitor

Using the maximum allowable **output ripple voltage** (from the specifications), we can work backwards to the maximum allowable ripple voltage at the filter.This is done by replacing the diode by its incremental model as shown below:

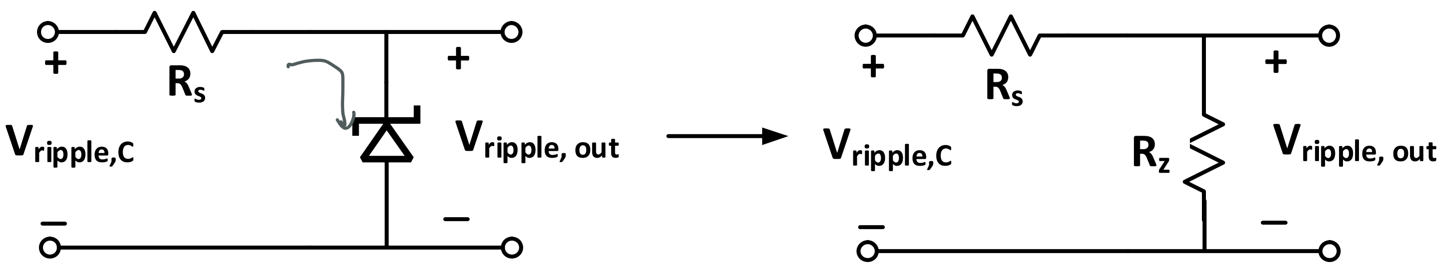


Figure 8: Circuit for determining the ripple voltage across capacitor.

- Use the device data-sheet to obtain the incremental resistance of the diode.

$R_z = 19\,\Omega$

- Use the maximum allowable **output ripple voltage** to obtain the maximum allowable ripple across the capacitor.

Max: 4.94 V

$I_z = \frac{4.94}{R_z} = \frac{V_{c,max} - 4.94}{R_s}$

$\Rightarrow V_{c,max} = 4.94 \frac{R_s}{R_z} + 4.94$

$= 104.82$ V

Min: 4.47 V

$\Rightarrow V_{c,min} = 4.47 \frac{R_s}{R_z} + 4.47$

$= 94.85$ V

$V_{c,ripple} =$

$V_{out,ripple} = 4.7 \times 0.02$

$= 0.094$ V

$0.094 = V_{c,ripple} \frac{19}{19+384.15}$

$\Rightarrow V_{c,ripple} \approx 2$ V

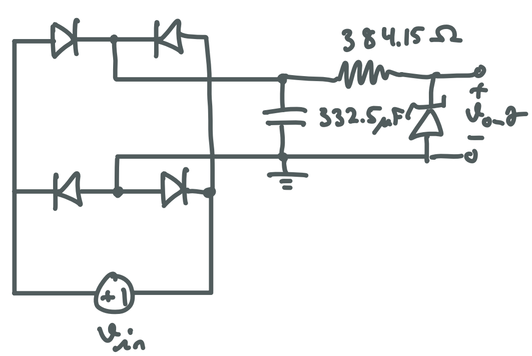
- Compute the capacitor value. Also mention the voltage rating of the capacitor. **Hint:** This is obtained from the maximum voltage difference across the capacitor.

$$V_r = \frac{I_L}{fC}$$

$$\Rightarrow C = \frac{I_L}{f V_r} = \frac{59.8\,mA}{120(2V)} = 332.5\,\mu F$$

4.5 Complete Design

Sketch the complete power supply circuit (based on bridge rectifier) you have designed. Include all the components and their values. For now you can replace the transformer circuit with sinusoidal input of appropriate value.



5 Simulations

- We will now verify the designed power supply using LTSPICE. You are to construct the power supply and run simulations that will test the performance of the power supply over a variety of realistic situations like variable loading and dirty power.

Table 1: Power Supply- Simulation Results

$\mathbf{V_{in}}$ (Volts)	$\mathbf{I_L}$ (mA)	Ripple across C (V)	Output voltage ripple (V)	Output DC voltage (V)
$\mathbf{24\sqrt{2}}$	0	1.439 V	0.003V	4.77 V
$\mathbf{26\sqrt{2}}$	0	1.589 V	0.003V	4.776V
$\mathbf{25\sqrt{2}}$	0	1.517 V	0.003V	4.773 V
$\mathbf{25\sqrt{2}}$	20	1.44V	0.004 V	4.749 V

- **Also save the output plots for each case.**

6 Bench Test - To be done in Lab

- Build the circuit you have tested in simulations. Complete the following table to verify its operaiton. **Please Note:** Ask your TA to verify your circuit before you test it.

Table 2: Power Supply Bench Test Results

$\mathbf{V_{in}}$ (Volts)	$\mathbf{I_L}$ (mA)	Ripple across C (V)	Output voltage ripple (V)	Output DC voltage (V)
$\mathbf{115\sqrt{2}}$				
$\mathbf{125\sqrt{2}}$				
$\mathbf{120\sqrt{2}}$				
$\mathbf{120\sqrt{2}}$	20			

7 Design of a complete power supply

In this part of the lab you will improve the output voltage and load current capabilities of the power supply you designed. The new power supply will have the following specifications:

- **DC open circuit output voltage = 9.5V – 10V .**
- **Open circuit output voltage stays within 2 % of the desired voltage as AC line voltage varies from 115 Vrms to 125 Vrms.**
- **Ripple voltage at output is less than 2 % of open circuit output DC voltage.**
- **The output current can vary from 0 to 80 mA.**

The new specifications will be met by building on the power supply you have already designed and tested.

7.1 Design Walk-through

In this section will design the additional components that will improve the specifications of the designed DC power supply.

7.1.1 Increasing the supply voltage

The first step is to increase the output voltage of the power supply to approximately **10** V. Consider the circuit shown below:

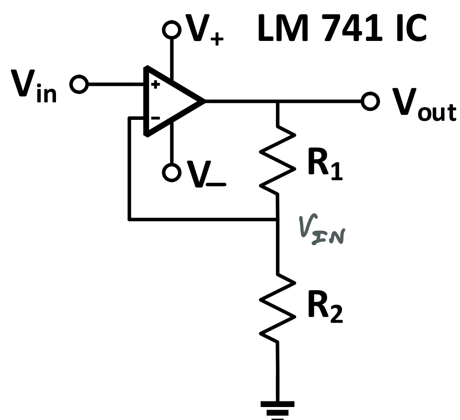


Figure 9: Opamp in negative feedback.

- Compute gain of the circuit $\mathbf{A_v} = \dfrac{\mathbf{Vout}}{\mathbf{V_{in}}}$ and fix the ratio $\dfrac{\mathbf{R_1}}{\mathbf{R_2}}$ to obtain $\mathbf{A_v} \approx \mathbf{2}$.

$$V_{IN} = V_{OUT} \frac{R_2}{R_1 + R_2}$$

$$\Rightarrow A_V = 2 = \frac{V_{out}}{V_{IN}} = \frac{R_1 + R_2}{R_2} = \frac{R_1}{R_2} + 1$$

$$\Rightarrow \boxed{\frac{R_1}{R_2} = 1}$$

- Compute the maximum power dissipated in the resistors $\mathbf{R_1}$ and $\mathbf{R_2}$. While performing bench test of the designed the resistors you pick must be rated for this power. **Hint:** Determine the condition under which max power dissipation occurs.

$V_{IN} = 4.7V \Rightarrow V_{OUT} = A_v V_{IN} = 9.4V$

$\approx 5V$ $\approx 10V$

$\boxed{P_{max} = IV}$

$= 80mA \cdot 5V$

$\boxed{= 400\,mV}$

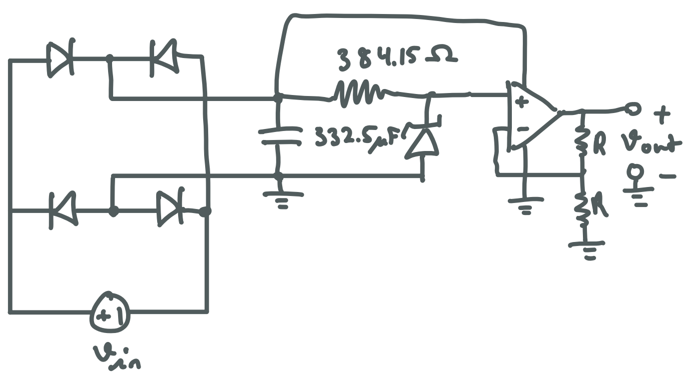
$I = 80mA$

- The opamp circuit shown in Fig. 9 will be used to increase the output voltage of the DC power supply you designed. **The rail voltages for the opamp can be taken from the terminals of filter capacitor**. You must ensure that this satisfies the maximum allowed rail voltages for LM 741 IC (ref: Datasheet). Note down the maximum allowed rails voltages from the device datasheet and then justify if voltage across the plates of capacitor satisfies this requirement.

Table 3: Rail voltage specifications

$\mathbf{V_{max}}$ (V) (From datasheet)	$\mathbf{V_{c,max}}$ (V) (Capacitor)
±22V	$25\sqrt{2} \pm 1.517$

- Sketch the new circuit that includes opamp in negative feedback configuration to increase the power supply output. **Note:** The rail voltage $\mathbf{V_+}$ can be taken from the terminals of filter capacitor and $\mathbf{V_-}$ can be at ground.



7.1.2 LTSpice Simulation

- Simulate your circuit using LTSpice. Complete the following table to verify its operation. **Please Note:** Ask your TA to verify your circuit before you test it.

Table 4: Power Supply with Opamp- Simulation Results

V_{in}	I_L (mA)	Ripple across C (V)	Output voltage ripple (V)	Output DC voltage (V)
$24\sqrt{2}$	0	1.439 V	0.007 V	9.542
$26\sqrt{2}$	0	1.588 V	0.007 V	9.554
$25\sqrt{2}$	0	1.514 V	0.007 V	9.548
$25\sqrt{2}$	5	1.514 V	0.007 V	9.548
$25\sqrt{2}$	20	1.514 V	≈ 0 V	8.671
$25\sqrt{2}$	80	1.622 V	≈ 0 V	0.184

- Is the output DC voltage maintained at the designed value for different load currents? Justify your answer. **Hint: Device Datasheet can give you a clue!**

No it didn't as the max short circuit is 25 mA.

- Note down the maximum power dissipated by resistors $\mathbf{R_1}, \mathbf{R_2}, \mathbf{R_s}$ under no load conditions and $\mathbf{V_{in}} = \mathbf{25\sqrt{2}}$.

$P_1 = P_2 = \frac{5^2}{1000} = 0.025\,W$

$P_3 = \frac{(25\sqrt{2} - 5)^2}{384.15} = 2.4\,W$

- **Save simulation plots showing DC output voltage and ripple.**

8 Increasing output current rating

In this section you will make further modifications to your circuit to meet the output current specifications of the power supply. Consider the BJT shown below:

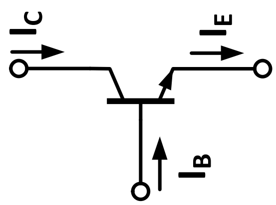
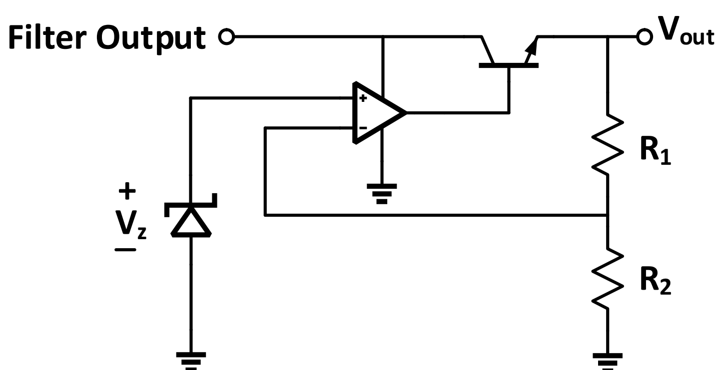


Figure 10: BJT for current gain.

- Give a brief explanation of how the BJT can be used to provide DC current gain.

In the forward active mode, a BJT follows: $I_C = \beta I_B$ & $I_E = \beta I_B$ where $\beta >> 1$ typically. This means for a little I_B, we can get a greater I_E.

- Consider now the circuit shown in Fig. 11 below. Compute the maximum power dissipated by the BJT in the



$P_{BJT} = \Delta V I_E$
$= (25\sqrt{2} - 10)(80mA)$
$= 2.03 \text{ W}$

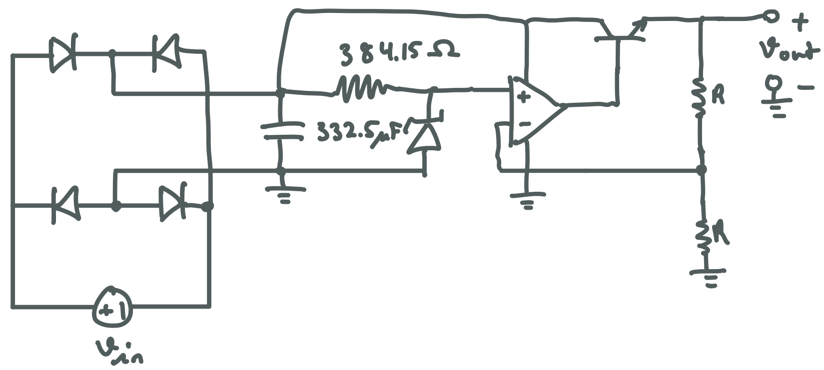
Figure 11: Power Supply with current gain element.

circuit shown in Fig. 11. Compare with the specifications in device data sheet.

Table 5: BJT Power handling capacity

$\mathbf{P_{max}}$ (W) (From datasheet)	$\mathbf{P_{max}}$ (W) (calculated)
115W	2.03W

- Sketch the complete power supply circuit.



8.0.1 LTSpice Simulation

- Simulate your circuit using LTSpice. Complete the following table to verify its operation.

Table 6: Complete Power Supply- Simulation Results

$\mathbf{V_{in}}$ (Volts)	$\mathbf{I_L}$ (mA)	Ripple across C (V)	Output voltage ripple (V)	Output DC voltage (V)
$\mathbf{24\sqrt{2}}$	0	1.626 V	0.008 V	9.541 V
$\mathbf{26\sqrt{2}}$	0	1.796 V	0.008 V	9.554 V
$\mathbf{25\sqrt{2}}$	0	1.716 V	0.008 V	9.548 V
$\mathbf{25\sqrt{2}}$	5	1.815 V	0.008 V	9.547 V
$\mathbf{25\sqrt{2}}$	20	2.109 V	0.01 V	9.547 V
$\mathbf{25\sqrt{2}}$	80	3.242 V	0.015 V	9.543 V

- Note down the power dissipated by the transistor ($\mathbf{P_T}$) and power delivered to the load ($\mathbf{P_L}$) under maximum load conditions and $\mathbf{V_{in}} = \mathbf{25\sqrt{2}}$.

$$P_T = (25\sqrt{2} - 9.543)\left(0.08 - \frac{9.543/2}{1000}\right) = 1.941\,W$$

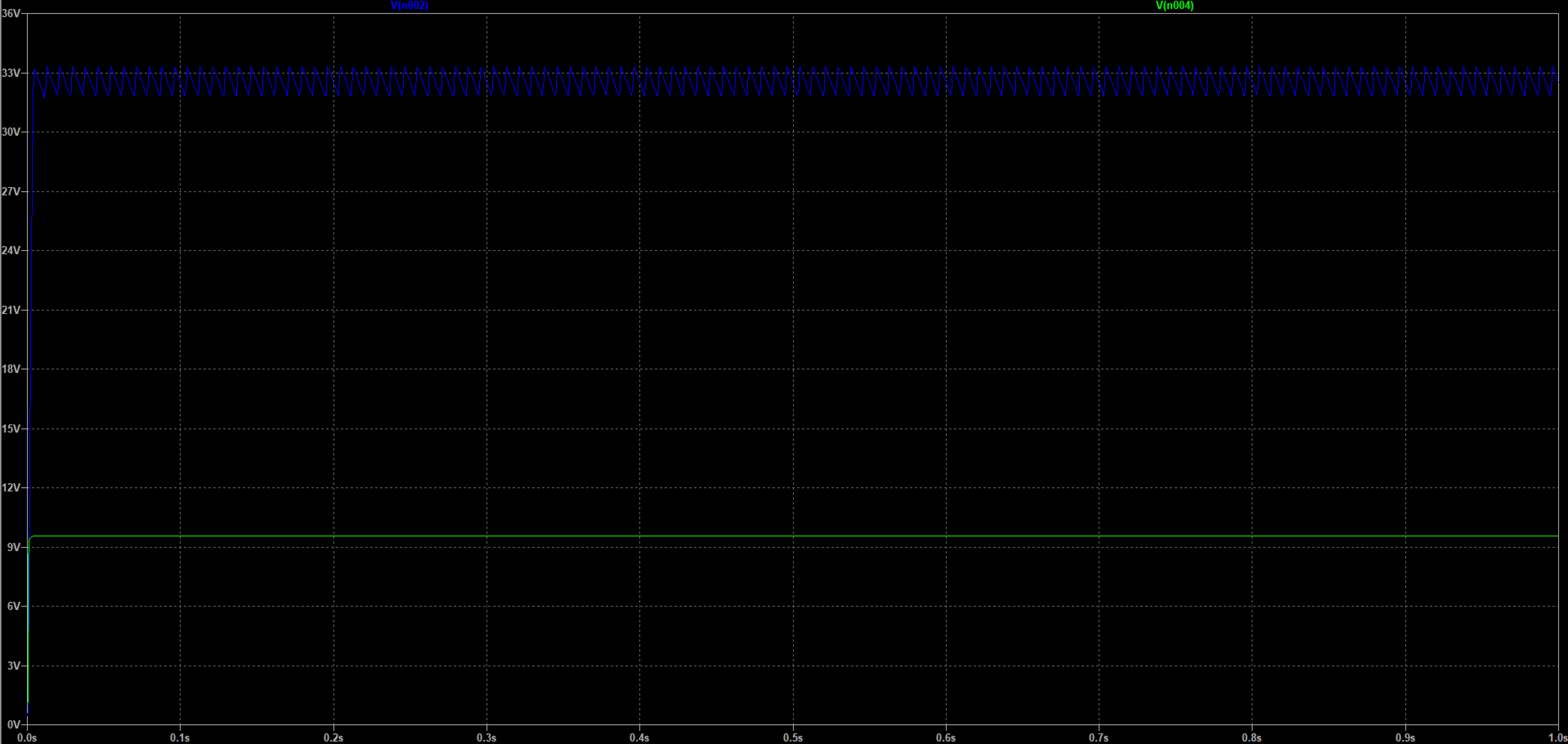
$$P_L = 80mA \cdot 9.543 = 76.34\,mW$$

9 Bench Test - To be done in Lab

1. Build the circuit you have tested in simulations. Complete the following table to verify its operation. **Please Note:**. Ask your TA to verify your circuit before you test it.

Table 7: Power Supply Bench Test Results

$\mathbf{V_{in}}$ (Volts)	$\mathbf{I_L}$ (mA)	Output DC voltage (V)
$\mathbf{115\sqrt{2}}$		
$\mathbf{125\sqrt{2}}$		
$\mathbf{120\sqrt{2}}$		
$\mathbf{120\sqrt{2}}$	80	


V(n002)
V(n004)
36V
33V
30V
27V
24V
21V
18V
15V
12V
9V
6V
3V
0V
0.0s
0.1s
0.2s
0.3s
0.4s
0.5s
0.6s
0.7s
0.8s
0.9s
1.0s

University of Illinois at Urbana-Champaign
Department of Electrical and Computer Engineering

ECE 343: Electronic Circuits Lab

Project : ***Design of AC-DC power supply + Voltage regulator***

Learning Objectives:

- Circuit design using nonlinear circuit elements.
- Making and justifying design choices based on requirements.
- Simulate designed power suppy on LTSpice
- Evaluate performance parameters and compare with design requirements
- Build a AC-DC power supply
- Solder circuit on a PCB
- Compute DC-DC conversion efficiency of three circuits- voltage dividers, Zener diode based DC-DC conversion (ECE 110), Power supply designed in this lab (ECE 343)
- Compare DC-DC conversion efficiency of designed voltage regulator with DC-DC conversion using a boost/buck converter (ECE 469)

---

1 Components Required

- **Breadboard**
- **Resistors**, **Capacitors:** Based on design.
- **Diodes:** D1N750 (Zener Diode), D1N 4001/4002 (Rectifier Diodes).
- **BJT:** 2N3055, **OpAmp:** LM741

10 Soldering

1. Solder the DC power supply components on the PCB provided to you! Check if the soldered PCB gives the correct output.

11 DC-DC Conversion

In this section we will explore the idea of DC-DC conversion in some more detail. DC-DC conversion is an important operation in electronic circuits. The power supply in a desktop personal computer (PC) converts an AC input to a DC voltage voltage value that is then used to power different components on the PC. Circuits also use multiple DC voltage levels. The pinout of an ATX power supply used in a PC is shown below in Fig. 12. Note that the pinouts indicate several DC voltage values. The power supply we designed can be viewed as a DC-DC converter (after the rectification step). The final regulator circuit designed in this lab used a reference voltage of $\mathbf{V_{ref} = 4.7\ V}$.

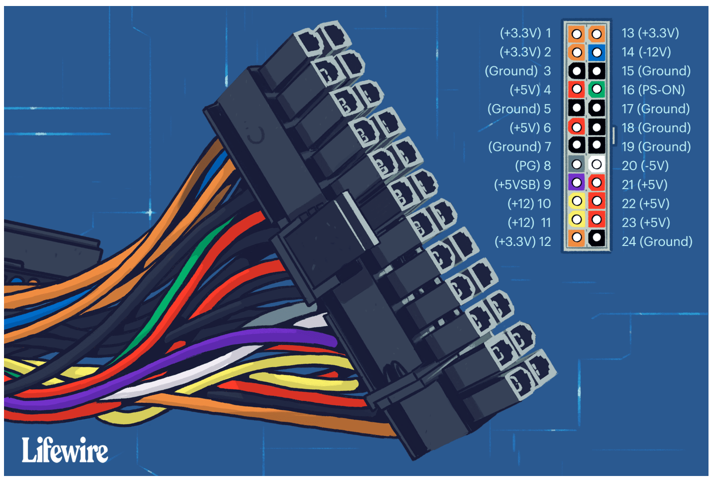


Figure 12: ATX power supply pinout. (ATX)

1. Consider a common method for DC-DC conversion shown in Fig. 13. The circuit in Fig. 13 is a voltage divider. Assume $\mathbf{V_1 = 25\sqrt{2}\ V}$. Compute the efficiency η of the voltage divider circuit for the values of $\mathbf{V_{out}}$ shown in table 8. You may assume $\mathbf{R_1 = 1\ k\Omega}$. ($\eta = \frac{\text{output power}}{\text{input power}}$)

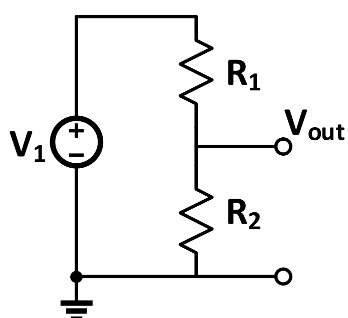


Figure 13: voltage Divider Circuit.

$$v_{out} = v_1 \frac{R_2}{R_1 + R_2}$$

$$v_{out}R_1 + v_{out}R_2 = v_1 R_2$$

$$v_{out}R_1 = R_2(v_1 - v_{out})$$

$$\to R_2 = \frac{v_{out}R_1}{v_1 - v_{out}}$$

Table 8: DC-DC Conversion using voltage divider

$\mathbf{V_{out}}$ (Volts)	$\mathbf{R_2}$ **(kΩ)**	Efficiency, η
8	.293	0.23
4.7	.153	0.13
2	.06	0.06

2. Consider now a circuit similar to the DC-DC conversion circuit you saw in ECE 110 **(Introduction to Electronics)**. We designed a similar circuit in Phase 1 of this project. Compute the efficiency of the circuit shown below under full load conditions. Assume $\mathbf{V_{in} = 25\sqrt{2}\ V}$, $\mathbf{R_1 = 375\ \Omega}$, $\mathbf{I_L(max) = 20\ mA}$, $\mathbf{V_{out} = 4.7\ V}$.

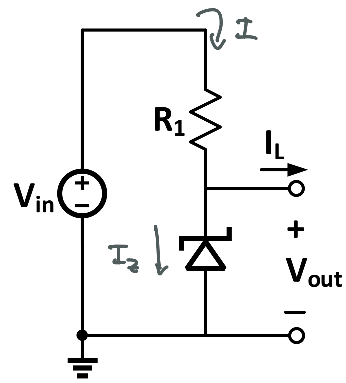


Figure 14: Zener diode based DC-DC conversion.

$$I = \frac{25\sqrt{2} - 4.7}{375} = 81.75\ mA$$

$$\Rightarrow \eta = \frac{V_{out} I_L}{V_{in} I} = 0.033$$

3. Consider the DC-DC conversion step that we implemented using the circuit shown in Fig. 15 below.

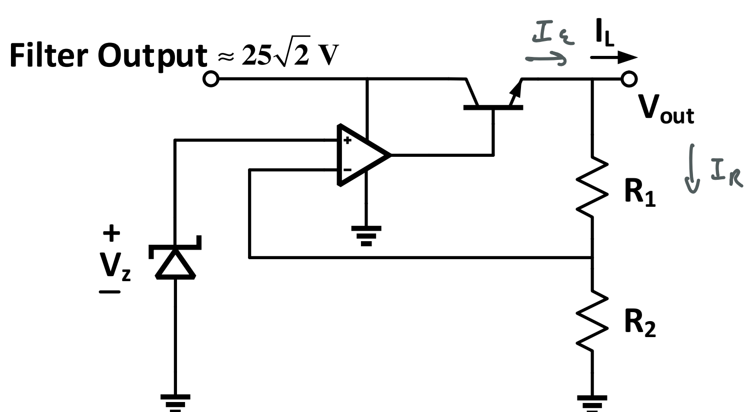


Figure 15: DC-DC conversion .

Assuming that power consumed by the transistor is the main source of power loss in the converter shown in Fig. 15, compute the efficiency of the DC regulator you designed under full load conditions ($\mathbf{I_L = 80\ mA}$). Assume $\mathbf{V_z = 4.7\ V}$, $\mathbf{R_1 = 10\ k\Omega}$, $\mathbf{R_2 = 10\ k\Omega}$, and $\mathbf{V_{out} = 9.54\ V}$.

$I_R = \frac{V_{out}}{R_1 + R_2} = 4.77\times10^{-4}$

$\Rightarrow I_E = 80.5 mA$ $\quad\quad \Rightarrow \eta = \frac{V_{out} I_L}{V_{in} I_C} = 0.268$

$\Rightarrow I_C \approx I_E$

4. Consider the circuit shown in Fig. 16. The switches $\mathbf{S_1}$ and $\mathbf{S_2}$ operate in complimentary fashion. The figure also shows the switching function $\phi_1(t)$ of switch $\mathbf{S_1}$. Note that $\phi_1(t)$ is periodic with period $\mathbf{T}$. The switch $\mathbf{S_1}$ is turned on for a duration $\mathbf{DT}$ during every cycle. The quantity $\mathbf{D}$ is called duty ratio and is given by,

$$\mathbf{D} = \frac{\text{Time switch } \mathbf{S_1} \text{ is on}}{\mathbf{T}}.$$

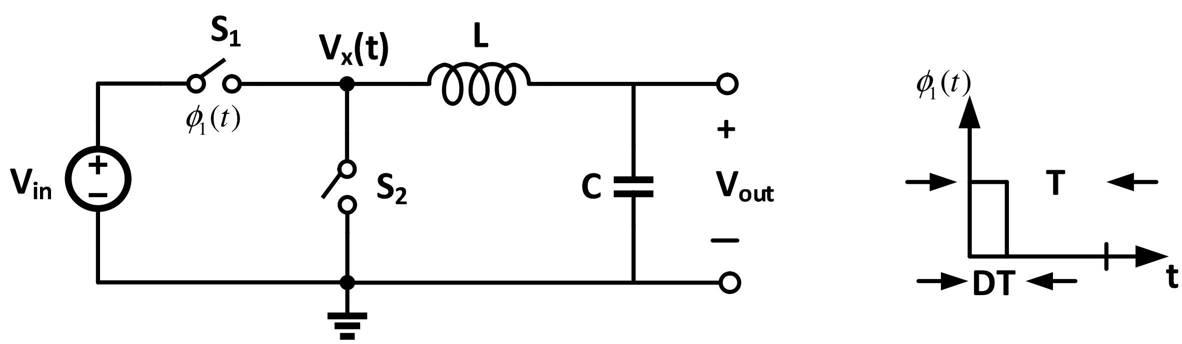


Figure 16: Switched Circuit.

The circuit above represents the idea behind buck converter circuit that you will study in detail in ECE 464/469 **(Power Electronics/Power Electronics Lab)**. The average value, $< \mathbf{V_x} >$, of voltage $\mathbf{V_x(t)}$ is given by,

$$< \mathbf{V_x} > = \frac{1}{\mathrm{T}} \int_0^{\mathrm{T}} \mathrm{V_x(t)dt}.$$

Compute the value of $< \mathbf{V_x} >$ in terms of duty ratio $\mathbf{D}$.

$$\langle V_x \rangle = \frac{1}{T}\int_0^{DT} V_x(t)\,dt = \frac{1}{T}\left(DT\,V_x(t)\right) = \boxed{D V_x(t)}$$

5. The inductor and capacitor in Fig. 16, perform lowpass filtering on voltage $\mathbf{V_x(t)}$. Assuming that all the components in Fig. 14 are ideal, what would be DC-DC conversion efficiency (in theory) of the converter in Fig. 16

$$\eta = 1$$

6. Observe demo of the DC-DC conversion using a boost/buck converter (based on the circuit shown in Fig. 16) used in ECE 469. Note down the input power and output power of the converter, and compute the efficiency of the converter.

$$\eta = \frac{\frac{7.18^2}{10}}{0.706 \cdot 20} = 0.84$$

12 Reflections

1. What are the advantages and disadvantages of each of the DC-DC converters discussed in Section 11

Voltage Divider:
Simple to make, but not steady voltage when load attached.

Zener Circuit:
Steady voltage irrespective of load, low efficiency.

Circuit 3:
Steady voltage and high I_L. Higher efficiency. Low noise.

Buck Converter:
Higher noise but far more efficient than circuit 3.

2. Identify one application for each type circuit we discussed in section 11.

Voltage Divider:
Good for sensing applications as attaching a load would change voltage.

Zener Circuit:
Good for a reference voltage and could be used as DC-DC converter but very inefficient.

Circuit 3:
Better version of Zener Circuit as higher I_L will greater efficiency. Good for low-noise applications.

Buck Converter:
Higher noise but far more efficient than circuit 3. Better for power transfer.

3. You may have noticed several solar panels on the roof of ECE building. Some of solar panels (60 in number!) are used for research purposes. One of the goals is to use these sixty panels to supply power back to the power grid. This will require converters to be designed for each solar panel. Assuming that the goal is to transfer **21kW** of power back to the power grid, which one of the converters will you pick. State your reasons.What would be the main constraint that will influence your decision? Would any additional information help you make your decision?

The buck converter as this is more efficient allowing to make it easier to transfer power.

4. Figure 17 shows various components in an IPhone.Identify some analog components in Fig. 17.

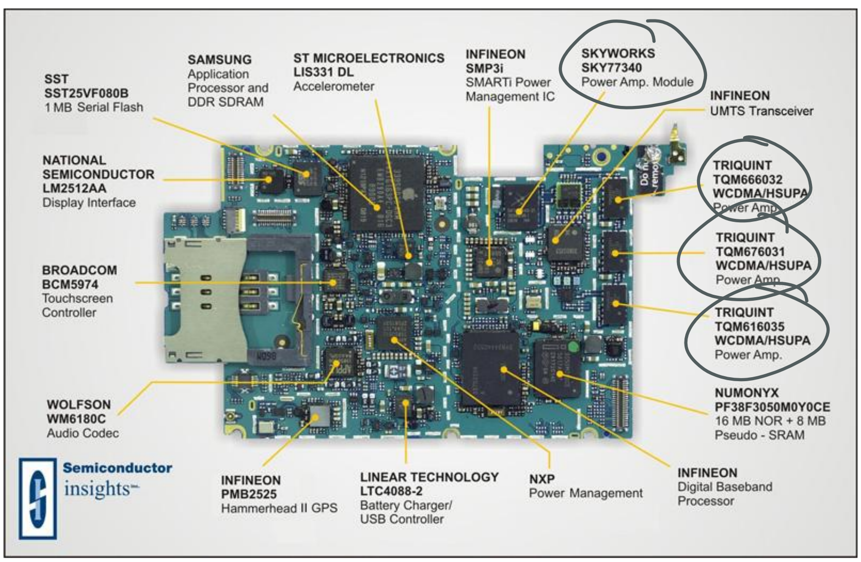


Figure 17: Iphone under the hood.

Circled them above.

5. Lithium-Ion batteries used in cell phones are rated at **3.7 V**. The analog components usually operate in the range **1.2 V – 1.8 V**, depending on the process technology (**65nm – 180nm**). Analog components are also sensitive to noise. Would any of the methods discussed in section 11 work? Justify your answer. If none of the above methods work, propose a method that can be applied in this case.

We would need to use a low noise option like what we designed in the previous section or in part 3, but an even lower noise option may be needed.