

SECTION ONE

Lab Equipment: Using Your Tools

SECTION OVERVIEW

This section will be a review on lab equipment usage and safety precautions. The first step in validating any new design should be an accurate simulation. In this course, PSPICE will be used to simulate all the designs that will be used. In this section proper usage of the variable power supply, function generator and oscilloscope will be reviewed.

Objectives

- Be able to complete each lab in a safe manner.
- Be able to utilize the oscilloscope settings appropriately.
- Understand how to setup a lab power supply for single and dual channel supply.
- Be able to use PSPICE to simulate semiconductors and integrated circuits.

Materials

- Oscilloscope probe, BNC to alligator clips cable, test leads.
- Solder-less breadboard and jumpers.
- Your textbook, *Sedra and Smith*, for reference: Pages 174-182.

Design constraints

Box 1 lists the design constraints for the power supply. Keep these in mind as you complete each section of each lab. Each criterion must be met. On the Web-page, you can find the full project specification document. As you progress through the process of building your power supply, you need to use the specification document to build your system. Just like in any engineering job, you need to build to the specification. The project requirements are listed and explained in more detail in the specification.

- Two channels, one positive and the other negative, when referenced ground.
- Each channel must be able to supply at least 900mA per channel continuously, at voltages between ± 2 and ± 12 V.
- Current limiting circuitry for each channel, protecting above 1 Amp +/- 10%.
- Voltage ripple less than $0.75V_{p-p}$ in amplitude per channel, with both channels fully loaded to 900mA.
- External connections for leads and voltage adjustment.
- Easily accessible AC power switch.
- A clearly visible power indication light.
- Safely equipped with a cooling fan that should not normally be running at 70°F , but should reach rated speed around 95°F .
- Power supply assembled safely and with no electrical hazards, and utilizes a safety fuse.

Box 1: Power Supply Design Constraints

PRE-LAB

Nothing will be required for turn-in in Section One. If you would like to get started, start creating your net list for Part Six, (which appears later in this section). If you are not familiar with the PSPICE commands listed in Part Six, you can find examples in the [PSPICE reference document](#).

In future sections of this lab, you will be asked to design the circuit element prior to coming to lab. You should submit these sections in the format, as indicated by the project specification document. At the end of the entire lab, you will be asked to submit a fully-completed project specification document.

LAB

This lab is divided into six parts: **lab safety rules, cost analysis of parts, understand the oscilloscope and signal generator settings, power supply block diagram, and SPICE simulation of a full-wave rectifier.**

Part One – Lab safety rules

The most important component in any lab work is the ability to complete it safely. This means not only preventing harm to yourself or others, but also not damaging the lab equipment. The TA may ask you to leave the lab if your actions are not safe.

Make a note of the following lab safety rules:

1. **Power supply:** The power supply in this lab interfaces with the electrical system of the building. Any connections that are energized to 120 VAC must be covered with shrink wrap before plugging in the power cord. This is especially important in Section Two.



The TA may ask that to see that the circuit is safe before it is energized.



Even though later labs will use low voltage DC, these may still contain shock hazards.

2. **Filter Capacitors:** The filter capacitors used in Section Two are only charged to 18 VDC or less. However, their high capacity gives the potential for a significant source of current long after the power supply is unplugged.



Always treat a circuit as if it were energized! Never work on a power supply while it is plugged in.

Lab safety rules (Contd.)

3. **One-Hand Rule:** A very important rule to follow is often called the “one hand” rule. This means you never work on an electrical system with more than one hand. Only using one hand precludes the possibility of a path of current from one hand to the other, by way of your heart. Your TA will remind you of this rule if you do not follow it carefully.

Causing parts to get hot; emit smoke, or even cause loud “bang” noises are all a part of learning. Some mistakes will happen and you may even destroy a transistor. This is all part of the learning process.

Part Two – Cost analysis of parts

An important aspect of an engineer’s job is being able to create a design that not only meets the design constraints, but also does so with minimal cost.

In this course, you will be required to turn in schematics of your final design, along with a tally of the parts and the cost of these parts. Any circuits built for extra credit will not count towards the total cost of your design. The cost analysis is part of your project specification document and should be neatly integrated into the document under Section Eight.

Groups that meet all the design constraints and have the cheapest power supplies will be awarded extra points.

Part Three – Understand the oscilloscope and signal generator settings

It is imperative that you understand how to use an oscilloscope properly and in a timely manner. Familiarize yourself with: AC and DC coupling modes, “Measure” features, and probe compensation. In this section, you will prove that you know how to calibrate probes, and measure AC and DC components properly as well.

In order to understand the oscilloscope settings, follow these steps:

1. Plug in the probe, compensate it, and make sure you are in the proper gain mode (10x, 100x, etc.).
2. Create a 60 Hz sinusoidal AC signal of 200 mV amplitude, with DC offset of 1V with your signal generator.
3. Use the oscilloscope to measure the signal generator output. Measure the offset and AC signal separately, (i.e. AC and DC coupling modes).
4. Save the output with TekVisa (OpenChoice) to your computer. Plot the AC and DC recorded data separately. Show the peak and frequency (period) clearly.
5. Each lab member should calibrate their probe.

<http://classes.engr.oregonstate.edu/engr/winter2002/engr201/engr201tds210.html>



It is important that you understand the operation of an oscilloscope and be able to use it properly. You will frequently use an oscilloscope in *Signals and Systems II* (ECE 352), *Microprocessor System Design* (ECE 473) and in the Senior Design Project.

Part Four – Power supply block diagram

The power supply that is built in this course can be described using a functional block diagram, such as the one shown in Figure 1. In any complex engineering design, functional diagrams are important in breaking up the overwhelming design into small manageable pieces. Figure 1 is a simplified block diagram of the power supply project.

With respect to Figure 1, note the following:

- This is an example of how one would specify each of the signals entering or leaving such a block.
- The design starts at the power cord and ends at the voltage regulators.
- Internal to the supply, there are many different sub-systems that perform a variety of different tasks.
- During the course of this term, you will be designing circuits that fulfill the requirements for all of these blocks.

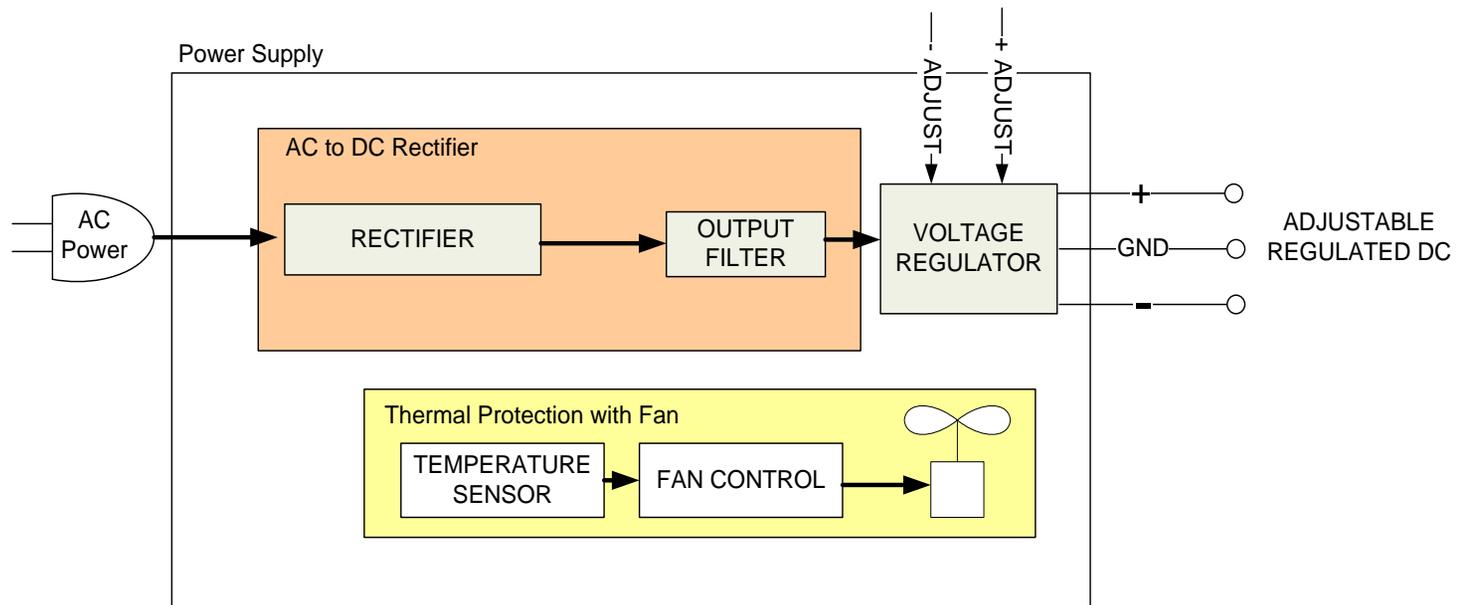


Figure 1: Simplified Block Diagram for the Power Supply

Section One: Lab Equipment (Using Your Tools)

The following table (Table 1) shows the various external connection specifications for the power supply.

Name	Direction	Description
AC Input	Input	Frequency: 60Hz Voltage: 120VAC Maximum Current: 0.5 Amps
Positive Voltage Output	Output	Voltage Range: 2 to 12V Max Current @ All Voltages: 0.9 Amp Current Limit: 1 Amp +/- 10% Maximum Ripple: 0.75 V _{PP}
Negative Voltage Output	Output	Voltage Range: -2 to -12V Max Current @ All Voltages: 0.9 Amp Current Limit: 1 Amp +/- 10% Maximum Ripple: 0.75 V _{PP}
Positive Supply Over Current Indicator (optional)	Output	Turn On: 1 Amp +/- 10% Indication: Visual or Audible
Negative Supply Over Current Indicator (optional)	Output	Turn On: 1 Amp +/- 10% Indication: Visual or Audible
Positive Voltage Adjustment	Input	Adjustment Method: Dial/ Potentiometer Full Scale: Single Turn
Negative Voltage Adjustment	Input	Adjustment Method: Dial/ Potentiometer Full Scale: Single Turn

Table 1: External Connection Specifications for the Power Supply

Part Five – SPICE simulation of a full-wave rectifier

In the *Introduction to Electrical and Computer Engineering: Concepts* class, (ECE 112), you would have learned to *create SPICE net lists*. In this lab, you will need to be able to *create and read those net lists*. Refer to *Appendix A* for information on PSPICE net lists, models, and simulations.



The name “SPICE” is a generic term to describe a circuit simulation program. Many types of SPICE programs exist, among which PSPICE and HSPICE are most commonly used. In this lab, you will be using PSPICE.

You could create a .cir file by hand and not depend on the schematic entry tool. Open the .cir file with the “PSPICE AD Student” program to simulate your circuit.

As you know, a diode only has two terminals, making it an easy device for which to create a net list entry. Simulate the circuit, as shown in Figure 2, using the 1N4004 model for the diodes.

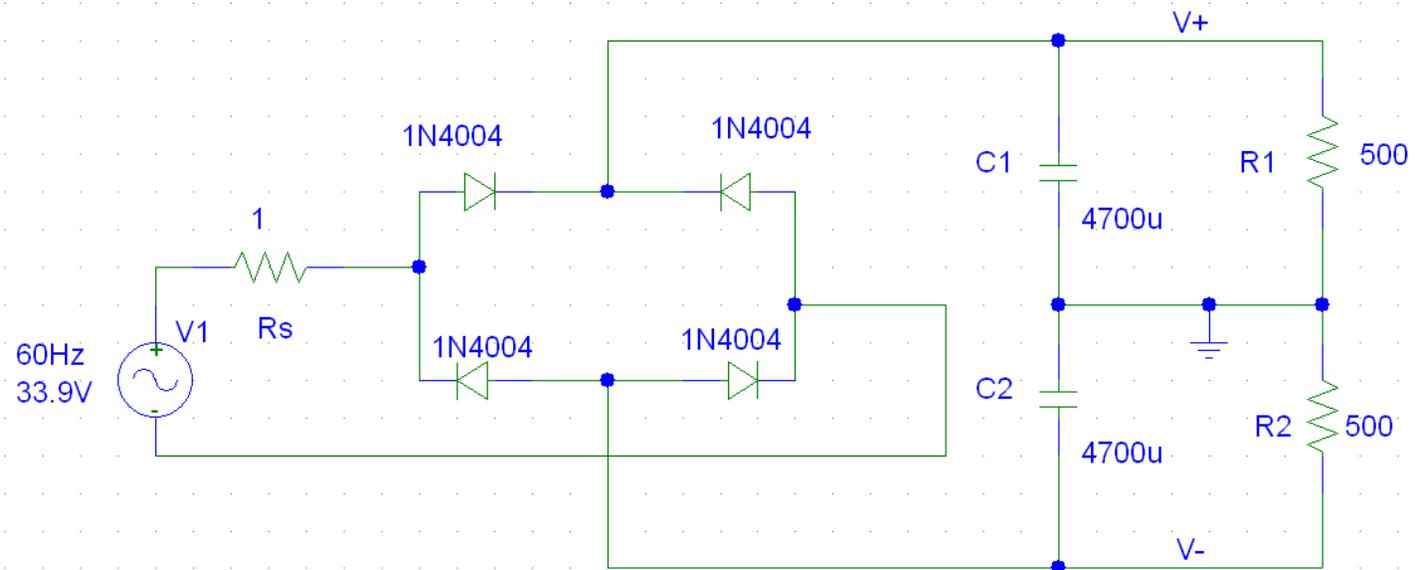


Figure 2: Full-Wave AC-DC Voltage rectifier with ripple filter.

This circuit is a full-wave AC-DC rectifier with capacitors acting to smooth the output. Resistors R1 and R2 are models of a load. Each load is one channel of the rectifier. Varying their resistance will provide different currents through the loads, (i.e. R1 and R2).

Section One: Lab Equipment (Using Your Tools)

Using the circuit in Figure 2, measure/ analyze the following:

1. Measure the output voltage (V_+ to ground and V_- to ground) for load resistance of 100, 500, 10K and 100K Ω . (The output will not be exactly DC, but will have some imperfection, which is the ripple.)
2. Report the ripple amplitude and frequency under the different load conditions. (Note that resistor R_s represents an output resistance of the source of 1 Ω . The SIN stimulus simulates a 24VAC_{rms} source).
3. Vary the size of the filter capacitors C1 and C2. What is the effect on the ripple?
4. Perform a transient analysis on the output, using the following guidelines:

SIN *<offset> <ampl> <freq>* AC transient sinusoidal stimulus.

.TRAN *<time_step> <time_stop>* Transient analysis specifier.

.PRINT TRAN *output_variables* Allows the results from the transient analysis to be an output in the form of tables, and referred to as print tables in the output file.



In PSPICE AC stimulus, the **amplitude** is specified, **not the RMS** value. Be very careful when referring to AC values. It is best to just always specify whether you are referring to RMS, amplitude (0-peak), or peak-to-peak voltages. Use this convention: VAC_{rms} , VAC_{0-peak} , VAC_{p-p} .

Also note that for sinusoidal waveforms, $V_{rms} \times \sqrt{2} = V_{0-p}$. Therefore the wall outlet is: 120 VAC, 169.7 VAC_{0-peak}, or 339.4 VAC_{p-p}. All forms are equivalent.

5. Note that you are performing a transient analysis. It takes some finite time for the output to stabilize. Which circuit components cause this? A common mistake is to run the transient analysis for too long or not long enough.



If you need help with PSPICE commands, refer to the PSPICE documentation on the CDROM (in the back of your textbook). It is located at <D:\Orcad\document\PSpcRef.pdf>, as well as on the [TekBots Web-page](#) for this course. Use the web to search for examples as well.



In engineering problems, it is often useful to consider limiting situations. For example, what happens to the output as C1 and C2 are very large, i.e. infinite? PSPICE cannot simulate infinite, instead use 1 Farad, (which is very large for a capacitor).

Similarly, what happens to the output, as R1 and R2 become very large, (i.e. no load attached)? This can be simulated by making R1 and R2 very large, i.e. 1M Ω . Be careful, however, to allow sufficient time for the circuit to stabilize. Also, be mindful of the RC time constant.

Measure/ analyze using Figure 2 (Contd.)

6. Use simulation results to determine the voltage across both load resistors, (from V- to V+).
7. Determine the minimum and maximum voltage, (keeping in mind that there should be a ripple in the voltage). Also, determine the frequency.

Note: You will need to use an AC voltage source (using the SIN stimulus) as well as the “.TRAN *time_step time_stop*” command, in order to have your simulation run for several milliseconds.

You may also want to use “.PRINT TRAN *output_variables*” to get a printing of the output voltage.

8. Repeat these simulations for a load resistor of 100, 10K, and 100K Ω .

STUDY QUESTIONS/ TURN-IN

The post-lab/ report for each week will vary slightly; so please pay attention to what the questions are, prior to starting your lab. The post-labs will consist of either questions to be answered separately, or having you update your project specification document to match your results. Sometimes, it will be a combination of the two.

The following need to be turned in, **at the beginning of the next lab session (Section Two)**:

1. An Excel graph of the oscilloscope measurements in AC and DC coupling modes from Part 4. Show the amplitude and frequency of the AC component.
2. A copy of your simulation results from Part Six. Include measurements of the ripple voltage and frequency with each load current. Comment your PSPICE net list as much as possible.
3. Lastly, turn in your **answers to the study questions** below. Please write your answers to the following set of study questions on a different sheet of paper. You may type your answers, or write them neatly by hand. Keep your answers clear and concise.

Section One: Lab Equipment (Using Your Tools)

- How is voltage ripple affected by load current (size of load resistors) and filter capacitors? What is the frequency of the voltage ripple and how is it related to the stimulus?
- How significant is the output resistance of the source? For what load currents is it non-negligible, (>5% change in output)?

✓ TA Signature: _____
(Lab work readable/ turned-in).

CHALLENGE

There will be several challenges throughout this course. Full credit for each challenge question will require a neatly drawn (or printed) schematic, along with a detailed report on your solution to the challenge problem.

- Simulate the circuit in Figure 3 using PSPICE and the models that you have downloaded. Run the simulation with various values for R4. (**Hint:** You will need to use a sub-circuit). What is the maximum current through R4 before the circuit fails to operate properly? Once you have a feel for the output of the circuit, describe in detail how the circuit operates to create this output.

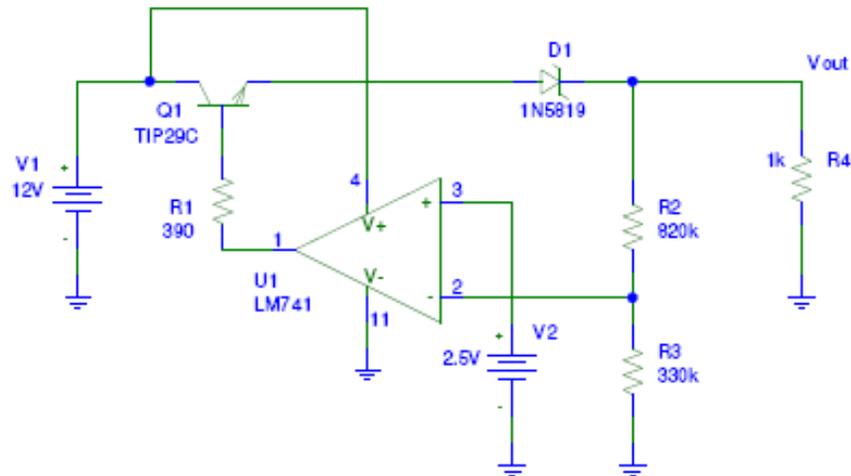


Figure 3: Challenge problem circuit