

EEL 3111C Lab Manual - Spring 2022

Written by Michael Mitchell, Edited by Wenhsing Wu and Baker Herrin

January 10, 2022

Contents

0 Lab Rules and Policies	6
0.1 Requirements	6
0.2 Structure	7
0.3 Tentative Schedule	9
0.4 Grading	10
0.5 Makeup Policy	10
0.6 Copying	10
0.7 Safety	11
1 Lab 1 - Introduction to the Digilent Analog Discovery	12
1.1 Objective	12
1.2 Materials	12
1.3 Introduction	12
1.4 Big Picture	14
1.5 Pre-Lab Requirements	14
1.6 In-Lab Requirements	15
1.7 Write Up	16
2 Lab 2 - LTSpice	17
2.1 Objective	17
2.2 Materials	17
2.3 Introduction	17
2.4 Pre-Lab Requirements	18
2.4.1 Modeling Simple Circuits	18
2.4.2 LTspice Variables	19
2.4.3 Monte Carlo Simulations	20
2.5 In-Lab Requirements	21
2.5.1 Equivalent Resistance	21
2.5.2 Monte Carlo with Different Tolerances	22
2.5.3 Resistor Costs	22
2.6 Write Up	23

3 Lab 3 - Resistive Circuits	24
3.1 Objective	24
3.2 Materials	24
3.3 Introduction	24
3.3.1 Resistor Combinations	24
3.3.2 Voltage and Current Dividers	25
3.3.3 Resistor Ladders	26
3.3.4 Measuring Voltage and Current	26
3.3.5 Resistor Color Code	26
3.4 Big Picture	27
3.5 Pre-lab Requirements	27
3.5.1 Resistor Ladders	27
3.5.2 R-2R Ladder	28
3.6 In-Lab Requirements	29
3.6.1 Resistor Tolerances	29
3.7 Write Up	30
4 Lab 4 - Controlled Sources	31
4.1 Objective	31
4.2 Materials	31
4.3 Introduction	31
4.3.1 Voltage Controlled Voltage Source	31
4.3.2 Feedback	32
4.4 Pre-Lab Requirements	33
4.4.1 Simulating a Controlled Source	33
4.4.2 Monte Carlo Simulations	34
4.4.3 Feedback	35
4.5 In-Lab Requirements	36
4.5.1 Worst Case Analysis	37
4.6 Write Up	38
5 Lab 5 - Operational Amplifiers	39
5.1 Objective	39
5.2 Materials	39
5.3 Introduction	39
5.3.1 The Ideal Op Amp	39
5.3.2 Common Configurations	40
5.3.3 Op Amp Pins	41
5.4 Big Picture	42
5.5 Pre-Lab Requirements	43
5.5.1 LTspice TLV272 Op Amp Model	43
5.5.2 Op Amp Construction	44
5.6 In-Lab Requirements	45
5.6.1 Experimental Op Amp Measurements	45

5.7 Write Up	46
6 Lab 6 - Operational Amplifiers Applications	47
6.1 Objective	47
6.2 Materials	47
6.3 Introduction	47
6.4 Big Picture	49
6.5 Pre-Lab Requirements	49
6.5.1 Buffering	49
6.5.2 Level Shifting	50
6.5.3 Differential to Single Ended Conversion	52
6.6 In-Lab Requirements	54
6.6.1 Construction	54
6.7 Write Up	55
7 Lab 7 - Capacitor Applications	56
7.1 Objective	56
7.2 Materials	56
7.3 Introduction	56
7.3.1 Time Domain Response	56
7.3.2 555 Timer	57
7.4 Big Picture	59
7.5 Pre-Lab Requirements	59
7.5.1 LTspice Simulations	59
7.5.2 Breadboard Implementation	60
7.6 In-Lab Requirements	61
7.6.1 Construction	61
7.7 Write Up	61
8 Lab 8 - Diode Applications	62
8.1 Objective	62
8.2 Materials	62
8.3 Introduction	62
8.3.1 Diode Operation	62
8.3.2 Applications	63
8.3.3 Potentiometers	65
8.4 Big Picture	66
8.5 Pre-Lab Requirements	66
8.5.1 LTspice Simulations	66
8.5.2 Breadboard Implementation	67
8.6 In-Lab Requirements	68
8.6.1 Breadboard Implementation	68
8.7 Write Up	69

9 Lab 9 - Filters	70
9.1 Objective	70
9.2 Materials	70
9.3 Introduction	70
9.3.1 Passive First Order RC Filters	71
9.3.2 Active RC Filters	72
9.4 Big Picture	73
9.5 Pre-Lab Requirements	73
9.5.1 LTspice Simulations	74
9.5.2 Breadboard implementation	74
9.6 In-Lab Requirements	75
9.6.1 Practical Implementations	75
9.7 Write Up	75
10 Lab 10 - Final Project	77
10.1 Objective	77
10.2 Materials	77
10.3 Introduction	77
10.3.1 Audio Amplifier	78
10.3.2 Recommendations	79
10.4 Pre-Lab Requirements	81
10.5 In-Lab Requirements	81
10.5.1 Wavegen Settings and Speakers	81
10.6 Write Up	82

Chapter 0

Lab Rules and Policies

The purpose of this laboratory is to provide a hands on experience for students in conjunction with the recorded lectures and problem sessions. While solving nodal equations and drawing circuits has a lot of utility, there is a great amount of understanding to be gained from building circuits, seeing them work, and then understanding why they work. The objectives of the laboratories are as follows:

- To verify concepts presented in the course.
- To enable students to gain skills in using basic electrical laboratory equipment such as meters, power supplies, and oscilloscopes, which will be need in subsequent electrical engineering laboratories and in industry.
- To make the students aware of the limits of the operation of components and equipment.
- To have the students gain skills in recording data and in reporting experimental results to effectively communicate what they have done and observed.
- To give students self-confidence in electrical laboratories.
- To have the students become safety conscious in a laboratory environment.

0.1 Requirements

Each student is required to have the following:

- A laptop with Waveforms installed
- If using a Macbook with the M1 chip (2020 or newer Macbooks) a parallel booting system is required
- Digilent Analog Discovery (1/2/NI)

- Breadboard
- Wiring kit
- Lab parts kit

A Digilent Analog Discovery kit for this course can be purchased from the UF bookstore. It includes an Analog Discovery 2, Breadboard, wiring kit, and assorted parts.

For the wiring kit, I recommend this one. You want to use "solderless breadboard/jumper" wires like at this link because the wires with other kits are more difficult to debug. https://www.amazon.com/REXQualis-Breadboard-Assorted-Prototyping-Circuits/dp/B081H2JQRV/ref=sr_1_1_sspa?dchild=1&keywords=solderless+breadboard+wires&qid=1610048019&sr=8-1-spons&psc=1&smid=AX8SR0V05IQ2E&spLa=ZW5jcnlwdGVkUXVhbGlmaWVyPUE

The lab manual is subject to change as errors are corrected, students are responsible for the newest version of the lab manual.

0.2 Structure

The laboratory portion of the course is made up of nine labs and a final project and takes place in NEB 250. Every laboratory will be composed of an objective, list of materials, introduction, big picture, pre-lab requirements, in-lab requirements, and a write up.

1. **Objective:** States the goals for a given lab.
2. **Materials:** List the materials used for a lab.
3. **Introduction:** Introduces the overall theme of the lab and the supporting background information.
4. **Big Picture:** A block diagram of the final project showing how the circuits in this lab fit together in the final project.
5. **Pre-Lab Requirements:** Most of the lab is accomplished by the student at home. The requirements for the pre-lab will list out what needs to be completed by students prior to their entry in to the lab.
 - Pre-lab quizzes are available on Canvas for 24 hours before the start of lab and must be completed by 8:30am on the day of your lab.
 - Pre-lab submissions to Canvas: tables, circuit diagrams and their resulting outputs, etc., must be compiled in to a document and submitted to Canvas by 8:30am on the day of your lab. No late submissions are accepted.
 - Pre-lab circuit demonstrations must be ready at the start of lab.

- Failure to complete the pre-lab on time, either the quiz, submission, or demonstration, with result in a zero for a given lab and the student will be barred from completing the lab.
6. **In-Lab Requirements:** Additional lab tasks that build on the work done in the pre-lab. Students arriving later than 5 minutes to lab will not be admitted.
7. **Write Up:** The conclusion of a lab requires a short write up, the project will require a formal lab report (more on that later) that summarizes the in-lab work. A typical write must have an introduction, summary of a results, discussion covering the work done in-lab (not the pre-lab), and a conclusion. See the template for details. Write ups due time will be posted on Canvas, typically due in one week, by 8:30am on the day of the next lab.

Consider the following when writing your lab write up.

- For every value listed, include the appropriate units.
- Be careful of the number of significant digits in the calculated results. This number should not be greater than the least number of significant digits in the data used for the calculations.
- Be objective and do not use colloquial phrases.
- Always give numerical results when applicable. Do not just say, “the results were as expected.” Specify the actual results, the expected results, any errors, and explain them.
- Be concise. A short direct write up is more desirable than a long rambling one. But at the same time, don’t give one word answers. The reader should clearly be able to tell that the writer understands the subject material.
- Keep everything neat and label everything clearly.
- Remember to number the equations and use figure numbers and table numbers for any drawings or tables. Also include captions for all figures and tables.
- Percent error is defined as $\delta = \left| \frac{\text{Theoretical} - \text{Experimental}}{\text{Theoretical}} \right| \times 100\%$
- Use the file "Circuits 1 prelab and write up template" in the "Lab related files" folder as a template for both Pre-Lab and Write-up reports.
- The write up must be typed, including tabled values, no pictures.
- All figures and tables must have a number and a caption.
- Include section labels, if it’s unclear which section is which, you’re more likely to lose points for not having a section.

- Don't make large unreadable tables and avoid weird table placement. All values must be appropriately tabled.
- Don't use large spacing, single spaced is fine.
- Don't include figures from the lab manual or equations unless they're asked for. You also don't need to show how you calculated voltage or percent error. Percent error is absolute value.
- Avoid rephrasing or copying the objective from the lab manual. The introduction and conclusion are worth a lot in the write up because the expectation is that you'll put time in to them. This is a cap of four sentences but that doesn't mean you can get away with one or two weak sentences.
- The discussion section should be a discussion on the results of lab and not discussing how you did the lab. What do the results show? You should aim for a few paragraphs.

0.3 Tentative Schedule

Subject to change.

- Lab 1 - Week 3
- Lab 2 - Week 4
- Lab 3 - Week 5
- Lab 4 - Week 6
- Lab 5 - Week 7
- Lab 6 - Week 8
- Lab 7 - Week 9
- Lab 8 - Week 10
- Lab 9 - Week 11
- Lab Project - Week 12-13

0.4 Grading

The grade for the lab section is made up labs and a project. Each lab is composed of a pre-lab, in-lab, and a write up. Students must complete the pre-lab, online submission and circuit demo, in order to complete the in-lab portion, failure to complete the pre-lab will result in a zero for the entire lab.

- Each lab is worth 10% and the project is worth 20% of the total lab grade. There is a total possible score of 110% for the lab. A minimum score of 70% in lab is required to pass the course, subject to change based on overall student performance. Completing the project is also required to pass the course.
- There is an on-line prelab quiz in Canvas required to be done, due at the same time with the prelab report (8:30am of the lab day). It is a few basic concepts required for the lab and worth 10 points (except Lab 1). Prelab report is worth 40 points, including the 10 points prelab demo circuits if required. Students are required to attend the lab to complete their in-lab items to get 20 points. Incomplete lab will get zero points for the In-lab. No partial points. In-lab results are required to be checked for completion. Absence will result in zero points for both In-Lab (20 points)and Write-Up (30 points), because there is no proof of individual work performed.
- Each individual lab is graded as follows and will be accompanied by a rubric when appropriate:
 - Pre-lab - 50% = 10% pre-lab quiz + 40% pre-lab report (this includes the demo points)
 - In-Lab - 20% You either get these points or you don't. There is no partial credit
 - Write Up - 30% If absent, this is automatically a 0, since there is no proof of personal work

0.5 Makeup Policy

Students are only allowed to make up missed labs with a legitimate absence and must produce documentation for their absence.

0.6 Copying

Students are bound by the University of Florida honor code and every student is expected to produce their own work. Any students who are found copying will receive a zero for the lab in question.

0.7 Safety

Although unlikely, it is possible to become injured in this laboratory. Emergency contacts by phone are 911 and 395-0050, the latter is for the Emergency Room at Shands Hospital. On some university phones it may be necessary to dial 9 first.

When working in any electrical laboratory, always keep electrical safety in mind. Following are some safety rules that each student should be aware of before beginning a laboratory experiment. Some of the rules do not apply to this laboratory, but to other electrical laboratories.

1. Never work totally alone in the laboratory. Someone else, and preferably the laboratory TA, should be present in the case of an emergency.
2. Use only the equipment provided. Do not use other equipment unless the laboratory TA approves the use.
3. Turn off the power before handling any wires. Never use damaged items, whether they are leads, components, equipment, or any other item.
4. To decrease the chances of being shocked, wear dry shoes and do not stand on metal or wet concrete. Also, do not wear any metal or jewelry. Moreover, do not handle wires, components, or equipment with wet hands.
5. In a laboratory in which soldering irons are used, keep an attentive eye on a hot soldering iron. Also, place it in the proper holder when not in use. Never leave a hot soldering iron unattended.
6. Make no connection to the power supply until the very last step. This practice will ensure that a student who is handling leads will not be shocked. Also, the circuit being built will not be harmed during construction.
7. Should someone become incapacitated because of electric shock, no one should touch that person until the power is turned off. Otherwise, the rescuer could also be shocked. Call for emergency services as soon as possible since resuscitation is likely if treatment is applied quickly. If breathing has stopped, begin CPR immediately and continue until qualified medical assistance has arrived.
8. If a piece of equipment is not working properly, attach a note and report the problem to the lab TA. The malfunction could possibly lead to a life-threatening situation.

Chapter 1

Lab 1 - Introduction to the Digilent Analog Discovery

1.1 Objective

The purpose of this laboratory is to familiarize students with the Digilent Analog Discovery and its associated software suite.

1.2 Materials

- Laptop with Waveforms required
- If using a Macbook with the M1 chip (2020 or newer Macbooks) a parallel booting system is required
- Digilent Analog Discovery
- Breadboard
- Wiring kit
- Lab parts kit
- Multimeter

1.3 Introduction

The Analog Discovery combines a variety of tradition test bench tools in to at simple embedded device that fits in the palm of your hand. This lab focuses on bringing students up to speed on how to use the variety of tools associated with the Analog Discovery.

There are a variety of tutorials and how-to examples on getting started with the Analog Discovery and its associated software. A good place to start is the Digilent Getting Started Guide: <https://reference.digilentinc.com/learn/instrumentation/tutorials/analog-discovery-2-getting-started>. There are also a variety of tutorials for working with the specific tools:

- Using the Waveform Generator - <https://reference.digilentinc.com/learn/instrumentation/tutorials/ad2-waveform-generator/start>
- Using the Oscilloscope - <https://reference.digilentinc.com/learn/instrumentation/tutorials/ad2-oscilloscope/start>
- Using the Power Supplies - <https://reference.digilentinc.com/learn/instrumentation/tutorials/ad2-power-supplies/start>

Digilent also has the same tutorials but in video form:

- Analog Discovery 2 Quick-Start: Video1 - Unboxing and Software Download - https://www.youtube.com/watch?v=2nAvh28o-t4&list=PLSTiCUIiN_BoLtf_bWtNzhb3VUP-KDvv91
- – Analog Discovery 2 Quick-Start: Video 2a - Installing WaveForms on Windows - https://www.youtube.com/watch?v=SzOnDa8TVYw&list=PLSTiCUIiN_BoLtf_bWtNzhb3VUP-KDvv91&index=2
- Analog Discovery 2 Quick-Start: Video 2b - Installing WaveForms on Mac - https://www.youtube.com/watch?v=4-06-vTMIHg&list=PLSTiCUIiN_BoLtf_bWtNzhb3VUP-KDvv91&index=3
- Analog Discovery 2 Quick-Start: Video 2c - Installing WaveForms on Linux - https://www.youtube.com/watch?v=uYc8-HwGNCA&list=PLSTiCUIiN_BoLtf_bWtNzhb3VUP-KDvv91&index=4
- Analog Discovery 2 Quick-Start: Video 4 - Oscilloscope Tool - https://www.youtube.com/watch?v=ln1ETnKmKk8&index=6&list=PLSTiCUIiN_BoLtf_bWtNzhb3VUP-KDvv91
- vAnalog Discovery 2 Quick-Start: Video 5 - Waveform Generator Tool - https://www.youtube.com/watch?v=0hRMF2jn8co&index=7&list=PLSTiCUIiN_BoLtf_bWtNzhb3VUP-KDvv91
- vAnalog Discovery 2 Quick-Start: Video 9 - Power Supplies - https://www.youtube.com/watch?v=EL5u7xVUBho&index=11&list=PLSTiCUIiN_BoLtf_bWtNzhb3VUP-KDvv91

Additionally, a previous student, David Munzer, has created a few tutorials in power point that can be accessed through Canvas.

- Analog Discovery Waveforms Tutorial
- Breadboarding Tutorial
- Debugging Tutorial

1.4 Big Picture

This portion of the lab usually shows how the circuits in this lab fit in to the final project but for the introductory labs, it's left out.

1.5 Pre-Lab Requirements

An Analog Discovery, breadboard, wiring kit, and a laptop with Waveforms 2015 (or later) installed is required to be admitted to the lab.

Work through the various tutorials to get a feel for the Analog Discovery, Waveforms 2015, and breadboarding. Please use wires from a wiring kit as pictured below. They can be found on Amazon. Search "solderless breadboard wires".

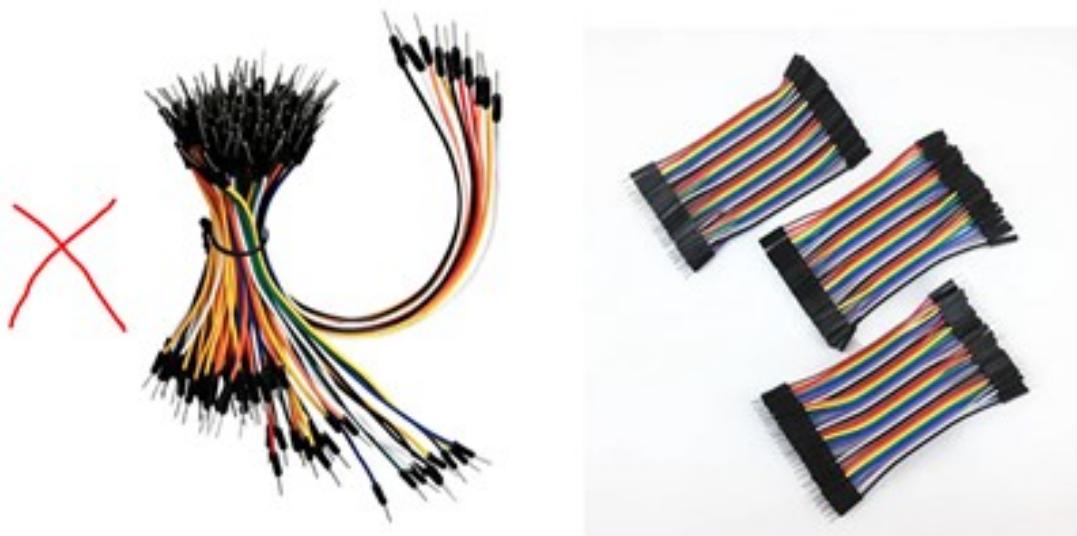


Figure 1.1: Example of what the wires you use should look like.

1.6 In-Lab Requirements

To be accomplished in lab:

1. Pre-lab check
2. Review of Lab Rules and Policies
3. Hand out lab kits
4. Live tutorial

1.7 Write Up

Typically this section would details the requirements for the write up but there is no write up for this lab.

Chapter 2

Lab 2 - LTSpice

2.1 Objective

The objective of the laboratory is to introduce students to SPICE (Simulation Program with Integrated Circuit Emphasis) and illustrate the differences between real and ideal components.

2.2 Materials

- Laptop with LTSpice

2.3 Introduction

For the purposes of this class, LTspice (current version is XVII) will be used. It is a free version of spice that's supported by Linear Technology (recently acquired by Analog Devices).

It can be downloaded from Linear Technologies' website: <http://www.linear.com/design-tools/software/#LTspice>

There are a variety of tutorials and videos associated with LTspice that gives a good background to the software. Linear Technologies offers a getting started guide and an overview video but they're a bit more advanced and covers the variety of advanced features offered by LTspice. There's also a variety of videos that are more topic specific. The best advice for using the software is to use on Windows and if you have to use a Mac, put all your documents on the desktop. The Mac version looks bare bones compared to the Windows version but all of the same functionality exists, it's just hidden in sub menus.

- LT - Getting Started Guide - <http://cds.linear.com/docs/en/software-and-simulation/LTspiceGettingStartedGuide.pdf>
- LT - LTspice IV Overview - https://www.youtube.com/watch?v=j_PScPJYNYQ

- LT - Schematic Editor - https://www.youtube.com/watch?v=aHhQnIFyDyE&index=14&list=PL4vooS_8RnzE4EoE27QssuxsccFmspbRP
- LT - Waveform Viewer - https://www.youtube.com/watch?v=xFLD5wb1X7U&index=13&list=PL4vooS_8RnzE4EoE27QssuxsccFmspbRP
- LT - Stepping Parameters - https://www.youtube.com/watch?v=hH74uZvEm6I&index=10&list=PL4vooS_8RnzE4EoE27QssuxsccFmspbRP

2.4 Pre-Lab Requirements

Install LTspice, work through the tutorials, and complete the following.

2.4.1 Modeling Simple Circuits

A resistor ladder is shown in Figure 2.1.

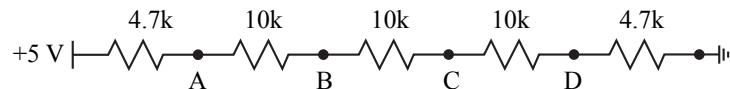


Figure 2.1: Resistor ladder example.

1. Using LTspice, layout the circuit in Figure 2.1 and run a DC operating point simulation to generate the node voltages at nodes A, B, C, and D. Table the node voltages.
2. When measuring the node voltages in the previous lab, the oscilloscope was used. The scope probes have a finite input resistance of $1\text{ M}\Omega$. Re-simulate the node voltages with a $1\text{ M}\Omega$ resistor at nodes A through D, Figure 2.2 (a) through (d). There should be a total of 4 simulations with node voltages A through D for each simulation. Table the values.
3. Save the tabled values to submit to Canvas as part of the pre-lab.

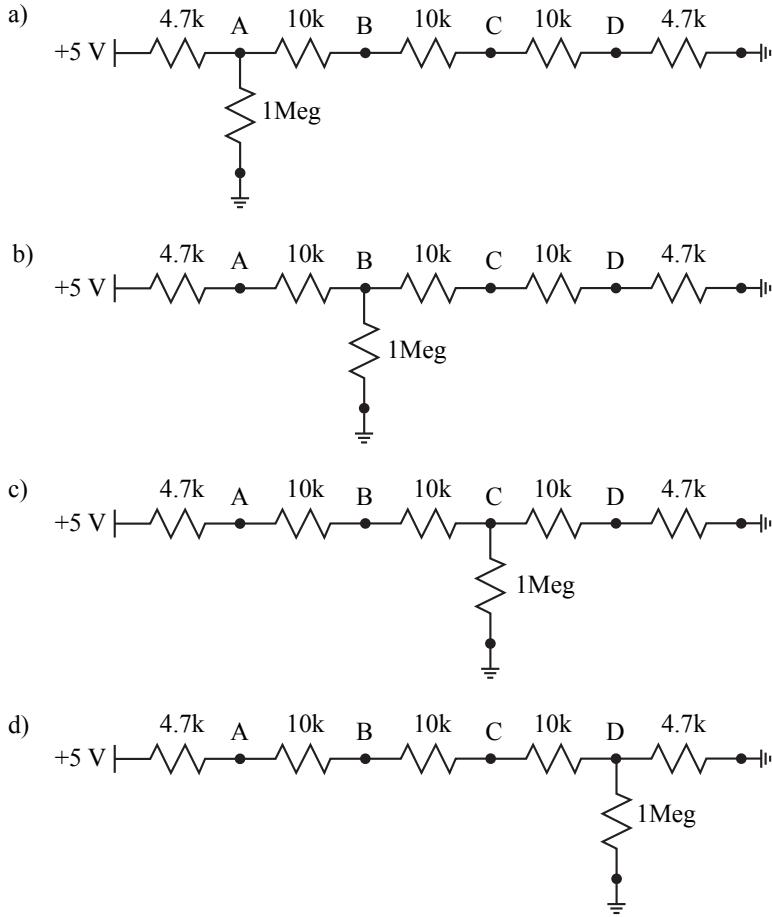


Figure 2.2: Resistor ladder with a $1\text{ M}\Omega$ resistor attached at nodes A through D ((a) through (d)).

2.4.2 LTspice Variables

There is clearly an effect caused by the input resistance of the oscilloscope. Using LTspice, quantify the effect as a function of resistance.

1. Layout the circuits in Figure 2.3 in LTspice on the same schematic. Note the resistor values are a variable $\{R\}$, the source voltage is 5 V DC and each output node is labeled.
2. Use the following spice directive in conjunction with a DC operating point simulation to sweep the resistors in both voltage dividers.

```
.step param R list 10 100 1k 10k 100k 1Meg 10Meg
```

3. Plot the voltages at V_{out1} and V_{out2} and set the x axis to logarithmic. Save the

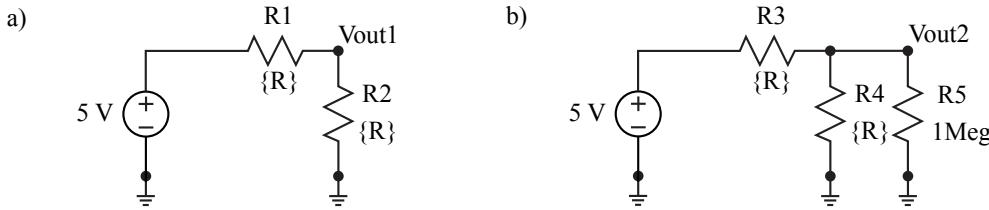


Figure 2.3: Two circuits to test the loading effect of one of the o-scope probes. The first circuit is a simple voltage divider that is not loaded (a) while the second circuit has a $1\text{ M}\Omega$ resistor in parallel with R_4 (b).

results in to a document, include your circuit and simulation results, which will be submitted to Canvas as part of the pre-lab.

2.4.3 Monte Carlo Simulations

Resistors have tolerances for their values which can vary from 10% for generic through hole resistors to as low as 0.005% for laser trimmed resistors. There is never a guarantee that a resistor has a resistance exactly equal to its value.

LTspice has a Monte Carlo function, $\{\text{mc}\{x,y\}\}$, that will randomize a number x with a tolerance of y between $x^*(1+y)$ and $x^*(1-y)$ with a uniform distribution.

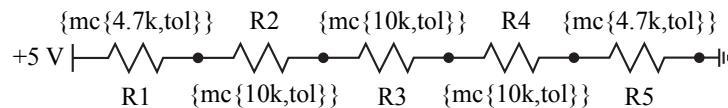


Figure 2.4: Resistor ladder with the values replaced by the LTspice Monte Carlo function.

1. Layout the circuit in Figure 2.4, setting the resistor values to call mc function.
2. Use the following spice directive in conjunction with a DC operating point simulation to randomize all the resistor values for 1000 trials.

```
.param tol=0.05
.step param run 1 1k 1
```

3. Plot all of the node voltages, A-D, on separate plot planes to view how the voltage at each nodes varies for each trial. Table the voltage ranges for each node.
4. Save the results in to a document, include your circuit and simulation results, which will be submitted to Canvas as part of the pre-lab. Use ".meas" spice directive to measure the voltage range (V_{\max} and V_{\min}) of nodes. Below is an example of using this to find the maximum voltage at node A:

```
.meas VAmmax max V(A)
```

VAmmax is the new variable, max is the command to find the max of V(node name)

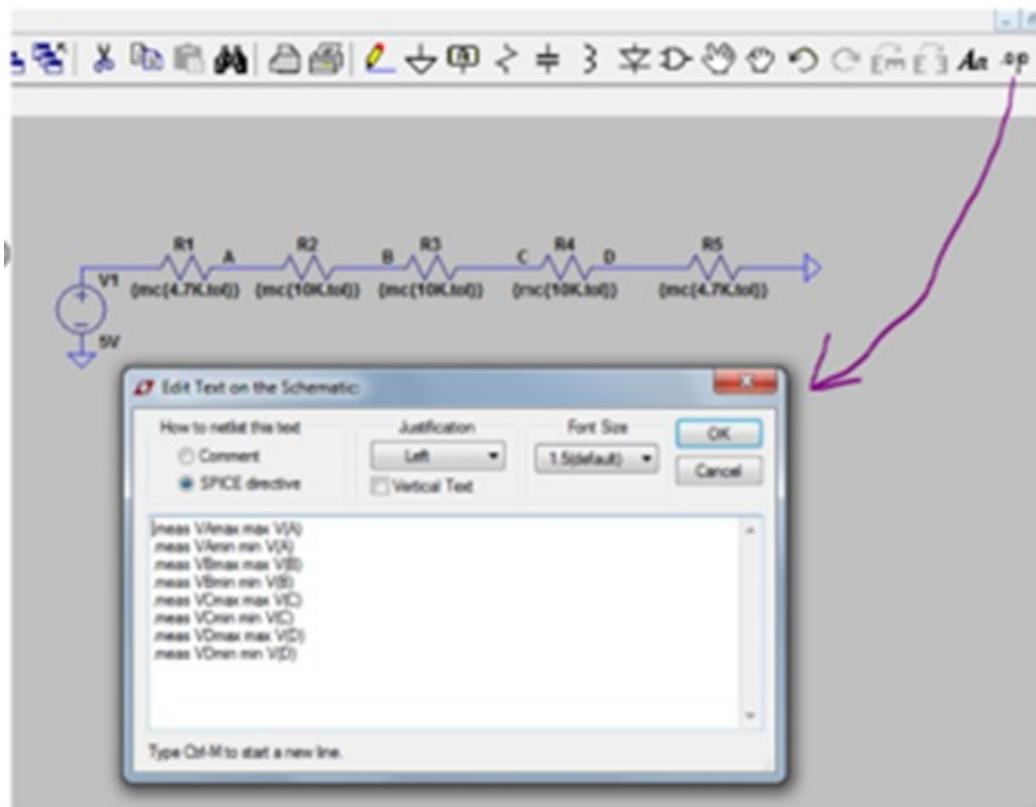


Figure 2.5: Example of using the max measure directive.

2.5 In-Lab Requirements

The following must be completed and submitted to Canvas before the start of lab.

1. Section 2.4.1 - Tabled voltages.
2. Section 2.4.2 - Simulation results.
3. Section 2.4.3 - Tabled voltages and simulation results.

Complete the following items in lab.

2.5.1 Equivalent Resistance

The equivalent resistance for an R-2R ladder with all the digital bits tied to ground is R . However, the R-2R ladder used in the previous lab didn't have perfect branches

(0.47k and 1k). So, the calculation for the equivalent resistance isn't trivial. When building the below circuit in spice, instead of assigning each resistor value to 470, give them the value " Ra " and create a spice directive ".param Ra=470 ". This will allow you to then change the directive's value from 470 to 500 afterwards and then rerun it to complete the experiment. This is a much more effective method.

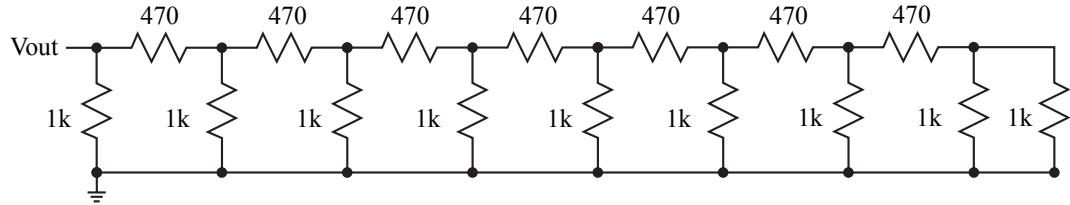


Figure 2.6: An 8 bit R-2R ladder with all of the digital bits tied to ground.

1. Using a 1 V test source at V_{out} run a DC operating point simulation and calculate the equivalent resistance. Save the equivalent resistance value.
2. Change 470Ω resistors to 500Ω resistors and confirm that the equivalent resistance is 500Ω .

2.5.2 Monte Carlo with Different Tolerances

Repeat the Monte Carlo simulations from Section 2.4.3 using the following tolerances. Table the voltage ranges at each node.

1. 1%
2. 0.1%
3. 0.05%
4. 0.01%
5. 0.005%

2.5.3 Resistor Costs

While having a tighter resistor tolerance is desired, the often overlooked problem is cost. Using the following Digikey link (<http://digikey.com>) search Through Hole Resistors, find the maximum and minimum cost of in-stock, 0.25 Watt, Carbon Film for 10 K ohm resistors for the tolerance listed below: (If the required item is out of stock, just make a note of it in your result.)

1. 5%

2. 1%
3. 0.1%
4. 0.05%
5. 0.01%
6. 0.005%

2.6 Write Up

Include the following in the write up.

1. Two equivalent resistor values from Subsection 2.5.1.
2. Table of voltage ranges from Subsection 2.5.2.
3. Table of cost ranges from Subsection 2.5.3.

Discuss the effects of resistor variation, the effect it had on the resistor ladder and its associated node voltages, and the cost of resistors with a tighter tolerance. Mention the two equivalent resistor values only briefly.

Chapter 3

Lab 3 - Resistive Circuits

3.1 Objective

The objective of the laboratory is to introduce basic resistive circuits, the effect of resistors in series and parallel, applications of simple resistor ladders, and the effects of real world resistors.

3.2 Materials

- Laptop with Waveforms
- Digilent Analog Discovery
- Breadboard
- Wiring kit
- Lab parts kit

3.3 Introduction

Resistors are a fundamental part of all electronic circuits and understanding how they work and how to apply them is a basic requirement for any electronics course.

3.3.1 Resistor Combinations

Resistors in series will always add together

$$R_{Total} = R_1 + R_2 \dots R_N \quad (3.1)$$

while resistors in parallel combine as the addition of reciprocals.

$$\frac{1}{R_{Total}} = \frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N} \quad (3.2)$$

Series resistances are fairly straight forward, add one resistor to the other, but the parallel combination leads to different results based on the values of the resistors in parallel. For instance, a large resistor in parallel with a small resistor gives an equivalent resistance approximately equal to the small resistor. Similarly, two resistors of the same value in parallel gives an equivalent resistance equal to half of either resistor.

3.3.2 Voltage and Current Dividers

Resistors can be used to make both voltage and current dividers. The use of either depends on the application but often times it is required to generate a small voltage from a large voltage or a smaller current from a larger current.

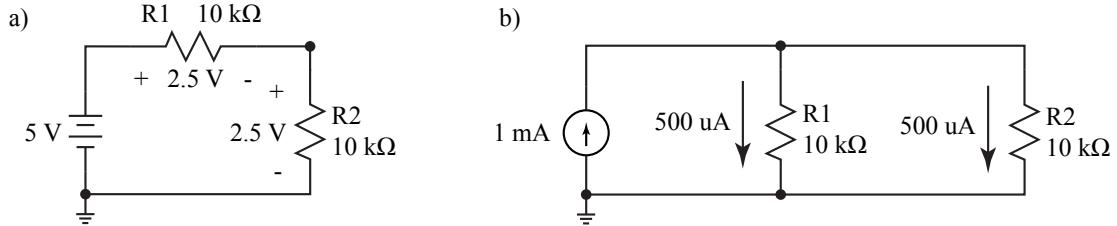


Figure 3.1: Examples of a voltage divider (a) and a current divider (b).

Figure 3.1 shows examples of a simple voltage divider (a) and a simple current divider (b). The output of a voltage divider can be simply calculated as

$$V_{out} = \frac{R2}{R1 + R2} V_{in} \quad (3.3)$$

In the example shown, the output voltage is 2.5 V, or the voltage across R2, which follows simply from plugging in to the equation. Because of the values for R1 and R2 are the same, the voltage drop across R1 is also 2.5 V.

$$\left(\frac{10 \text{ k}\Omega}{10 \text{ k}\Omega + 10 \text{ k}\Omega} \right) 5 \text{ V} = 2.5 \text{ V} \quad (3.4)$$

Similarly, a current divider is calculated in the same way where

$$I_{out} = \frac{R2}{R1 + R2} I_{in} \quad (3.5)$$

and the resulting current in each resistor is 500 μA which is again found by plugging in to the equation. Because the values for R1 and R2 are the same, the current through R1 is also 500 μA.

$$\left(\frac{10 \text{ k}\Omega}{10 \text{ k}\Omega + 10 \text{ k}\Omega} \right) 1 \text{ mA} = 500 \mu\text{A} \quad (3.6)$$

Note that a voltage divider applies to resistors in series while a current divider applies to resistors in parallel.

3.3.3 Resistor Ladders

Resistors are often used in a string or ladder configuration in order to generate reference voltages with a specific spacing. Typical applications of a resistor ladder are analog to digital conversion where the input voltage is compared to voltages on the resistor ladder or digital to analog conversion where digital bits (1s and 0s) are converted to an analog voltage.

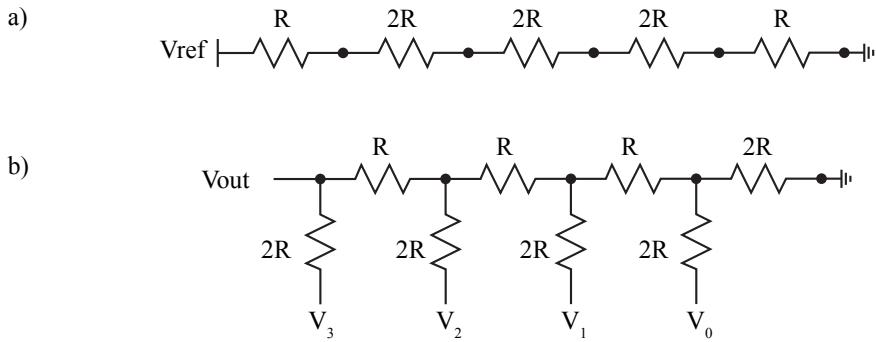


Figure 3.2: A simple resistor ladder where the resistor values set the voltages at each node (a) and a 4 bit R-2R ladder where digital bits, represented as voltages V_0 through V_3 are converted to an analog voltage, V_{out} .

3.3.4 Measuring Voltage and Current

There was a time when voltages and currents had to be measured with separate meters. But today, analog voltage and current meters have been supplanted by Digital Multimeters (DMM). A typical DMM has the ability to make several measurements such as AC and DC voltage, AC and DC current, resistance, and continuity. For the purpose of this experiment the measurements will focus on DC voltage and current.

3.3.5 Resistor Color Code

There are a variety of resistor in the course kit and while it's possible to measure all of the values individually, learning how to read the color codes will make identifying individual resistors much easier. See the look up table below.

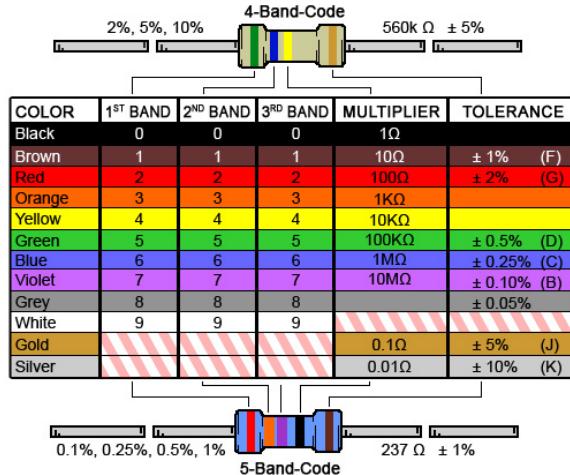


Figure 3.3: Resistor color code look up table.

3.4 Big Picture

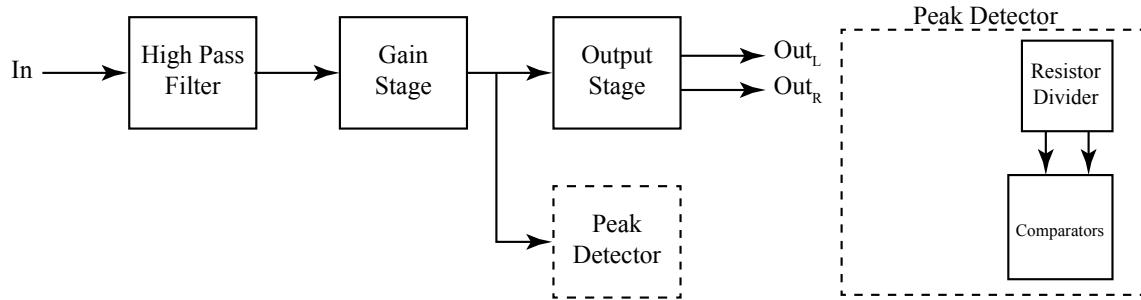


Figure 3.4: Big picture with emphasis on the peak detector.

This lab focuses on basic resistor circuits which will help to form the resistor divider used to set the threshold voltages in the peak detector for the final project.

3.5 Pre-lab Requirements

Complete the following before coming to lab.

3.5.1 Resistor Ladders

1. Using the circuit in Figure 3.5, calculate the ideal voltages for nodes A, B, C, and D. Table the values for future comparison.
2. Construct the circuit in Figure 3.5 and measure the actual voltages using the oscilloscope. Table the values.

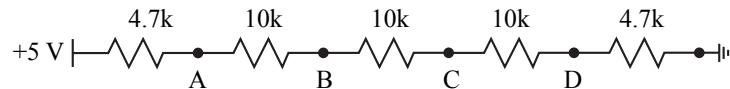


Figure 3.5: Resistor ladder example

3.5.2 R-2R Ladder

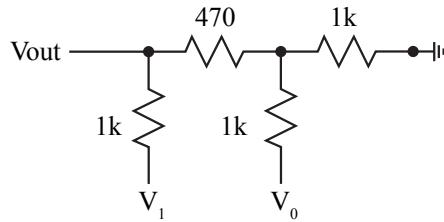


Figure 3.6: Two bit R-2R ladder.

1. Using the circuit in Figure 3.6, calculate the ideal output voltages for the following combination of voltages and table the values:
 - (a) $V_0 = 0 \text{ V}$ and $V_1 = 0 \text{ V}$
 - (b) $V_0 = 5 \text{ V}$ and $V_1 = 0 \text{ V}$
 - (c) $V_0 = 0 \text{ V}$ and $V_1 = 5 \text{ V}$
 - (d) $V_0 = 5 \text{ V}$ and $V_1 = 5 \text{ V}$
2. Construct the circuit in Figure 3.6 and measure the output voltage using the oscilloscope, table the values. In order to generate the changes in voltages, set Wavegen 1 (V_0) to a 1 Hz square wave with a 2.5 V amplitude, a 2.5 V offset, and a 180 phase offset. Set Wavegen 2 (V_1) to a 500 mHz square wave with a 2.5 V amplitude, a 2.5 V offset, and a 180 phase offset.
3. Save the circuit and Wavegen settings to demo to your lab instructor.
4. The output should look like the Scope image in Figure 3.7

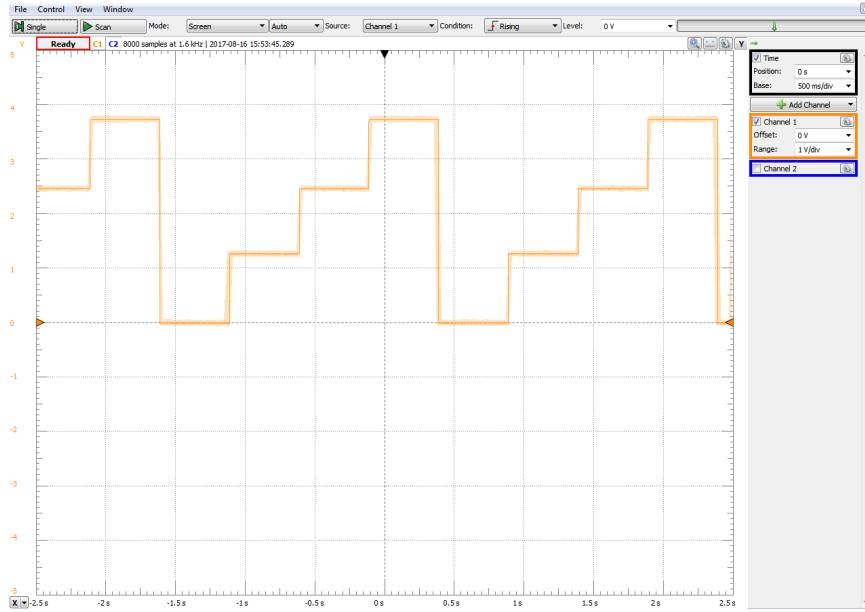


Figure 3.7: The desired output for the R-2R ladder circuit with the appropriate inputs.

3.6 In-Lab Requirements

The following items must be ready at the start of lab and students who do not have all items completed and ready to show at the beginning of lab will receive a zero and denied access to the lab.

- Submit the table of voltages for the resistor ladder and the R-2R ladder to Canvas in a document.
- Demo the output of the R-2R ladder as V_0 and V_1 change.

3.6.1 Resistor Tolerances

There is no such thing as a perfect resistor. All resistors have tolerance about their specified value. For example, a $10k\ \Omega$ resistor with a $\pm 5\%$ tolerance will have a value between $10k\ \Omega - 5\% \leq R \leq 10k\ \Omega + 5\%$.

- Using the multimeter, measure the resistance of the resistors in the ladder from Subsection 3.5.1. Remember to remove the resistors from the breadboard so that they can be measured independently.
- Re-calculate the ideal voltages using the measured resistances from the previous step and table the values.

- Using the multimeter, measure the resistance of the resistors used in the R-2R ladder from Subsection 3.5.2. Remember to remove the resistors from the breadboard so that they can be measured independently.
- Re-calculate the ideal voltages using the measured resistances from the previous step and table the values.
- Show the calculated values to your lab instructor before leaving the lab.

3.7 Write Up

Detail the effects of the resistance tolerances on the two different resistor ladders. Include the tabled values for both circuits and calculate the percent error using the experimentally measured values from the pre-lab and the ideal values calculated from the measured resistors.

For all lab write up submissions and reports the backgrounds for LTSpice and Waveforms should be changed from dark background to light background to make them readable for grading.

If the prelab simulation is incorrect, do not compare the in lab results to wrong values. Correct numbers or circuit schematics should be used in the write-up report.

Chapter 4

Lab 4 - Controlled Sources

4.1 Objective

The purpose of this lab is gain an understanding of controlled voltage sources and the effects of input resistance, output resistance, and feedback.

4.2 Materials

- Laptop with LTSpice

4.3 Introduction

A controlled source references a voltage or current and in turn generates a voltage or current, often amplifying the reference in the process. Four possible variations exist:

- Voltage Controlled Voltage Source (VCVS)
- Voltage Controlled Current Source (VCCS)
- Current Controlled Voltage Source (CCVS)
- Current Controlled Current Source (CCCS)

For this course, the primary concern is a VCVS because it serves as part of the model for an amplifier.

4.3.1 Voltage Controlled Voltage Source

As the name implies, a VCVS generates a voltage controlled by a different voltage. Usually this voltage is across a resistance but for the ideal case, the resistance is infinite and appears as an open circuit. Similarly, the output is simply the gain, A , multiplied by the control voltage, V_i , when the output resistance is zero and appears

as a wire. Consider the models for a VCVS show in Figure 4.1 (a) through (c). For the ideal case, (a) and (b), the voltage drop V_i provides the control voltage for the controlled source which generates a voltage output of AV_i . Practically, the input and output resistances must be considered as shown in (c).

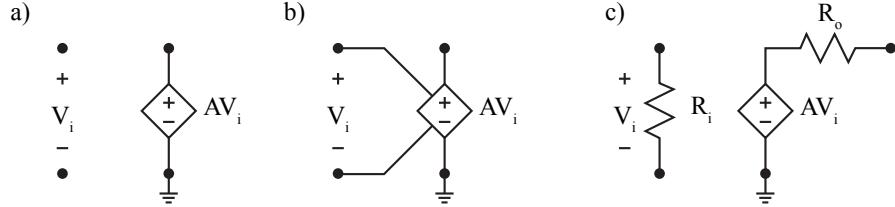


Figure 4.1: Representations of a voltage controlled voltage source, ideal model (a), variation used in LTspice (b), and a model with an input and output resistance (c).

4.3.2 Feedback

The combination of an amplifier with feedback can be represented many ways. One of the generic ways is a signal flow diagram as shown in Figure 4.2.

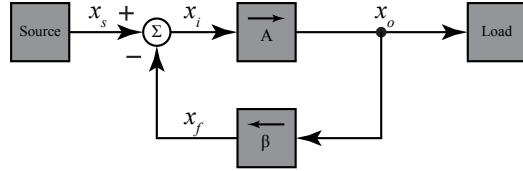


Figure 4.2: A signal flow diagram for an amplifier with feedback. Each voltage is represented by some value x , with an amplifier of gain A , and a feedback factor of β .

The amplifier has a gain, A , which is called the open-loop gain and represents the gain from the input, x_i , of the amplifier to the output, x_o . This is described by the equation

$$x_o = Ax_i \quad (4.1)$$

In this example, the output is feedback through a feedback network with a feedback factor, β . Which gives the feedback voltage, x_f , equal to the output voltage, x_o , times the feedback factor, β .

$$x_f = \beta x_o \quad (4.2)$$

The feedback voltage, x_f , is then subtracted from the source signal, x_s , which gives the input to the amplifier, x_i .

$$x_i = x_s - x_f \quad (4.3)$$

Here it's important to recognize that the loading effects of the load and source have been neglected as this is a generic case with finite gain. The total gain of the system, with feedback, then becomes

$$A_f \equiv \frac{x_o}{x_s} = \frac{A}{1 + A\beta} \quad (4.4)$$

This equation represents the closed loop gain of the system which is smaller than the open loop gain by a factor of $1 + A\beta$, called the amount of feedback. The quantity $A\beta$ is the loop gain and must be positive. Often the loop gain is large, $A\beta \gg 1$, or simply A is large, which simplifies the closed loop gain to

$$A_f \simeq \frac{1}{\beta} \quad (4.5)$$

Which gives a result entirely dependent on the feedback factor. This is an important because if the gain, A , is large, the closed loop gain will depend entirely on elements in the feedback network.

4.4 Pre-Lab Requirements

Complete the following using LTspice.

4.4.1 Simulating a Controlled Source

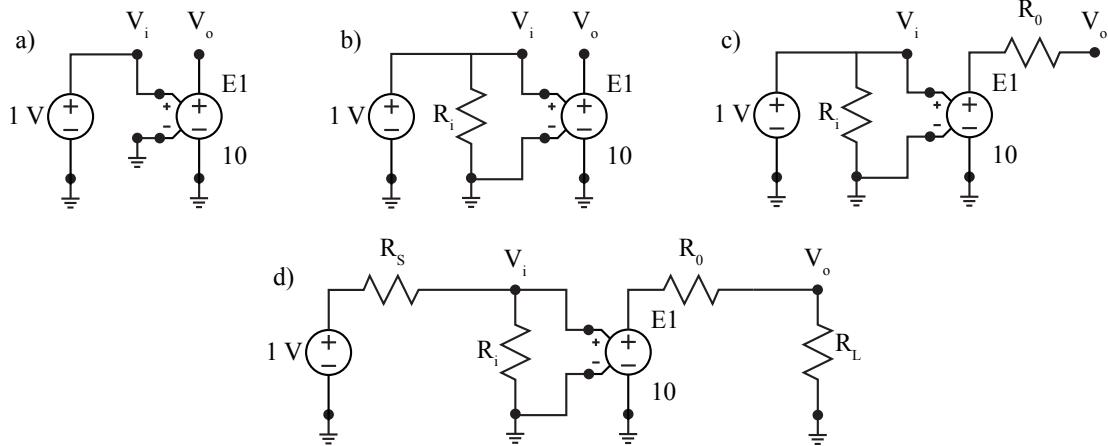


Figure 4.3: Different configurations for a VCVS in LTspice: the ideal case (a), with an input resistance (b), with an input resistance and output resistance (c), and with a source and load resistance (d).

1. Setup a simple voltage controlled voltage source in LTspice with a gain of 10 as shown in Figure 4.3 (a). Use the e component in LTspice, voltage dependent voltage source, and set the gain, E, to 10 and run a DC operating point solution to confirm that the output of the controlled source is 10 V. Table the input, V_i , and output, V_o , voltages.
2. Connect an input resistance from the voltage input to ground as shown in Figure 4.3 (b). Set the resistance to $10\text{ k}\Omega$ and re-run the DC operating point simulation, table the values for V_i and V_o .
3. Connect an output resistance form the controlled source output as shown in Figure 4.3 (c). Set the resistance to $10\text{ k}\Omega$ and re-run the DC operating point simulation, table the values for V_i and V_o .
4. Connect the circuit as shown in Figure 4.3 (d) with a source, R_s , and load, R_L , resistance. Set both values to $50\text{ }\Omega$ and re-run the DC operating point simulation, table the values for V_i and V_o .
5. Step through different values of R_i from 0.1 to 1Meg in powers of 10 and determine the best value for the input resistance. Plot the input voltage, V_i , on a log scale and save an image of the plot and circuit.
6. Using the value of R_i from the previous step, step through different values of R_o from 0.1 to 1Meg in powers of 10 and determine the best value for the output resistance. Plot the output voltage, V_o , on a log scale and save an image of the plot and circuit.

4.4.2 Monte Carlo Simulations

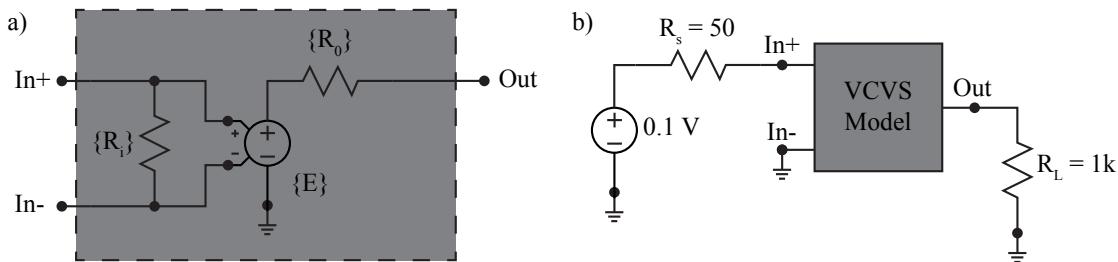


Figure 4.4: A general model for a voltage controlled voltage source with input and output resistance (a) and model used in conjunction with a DC source and a source and load resistance (b). Note that each of the values for the two resistors and the gain are LTspice variables, denoted by the curly brackets, {}.

1. Create a sub-circuit of the VCVS from Subsection 4.4.1 as shown in Figure 4.4 (a). Set the input resistance and output resistance to the previously determined values and the gain to 10. The second half of this video, <http://www.linear.com/solutions/7723>, is a helpful tutorial for creating a sub-circuit (symbol) from an already created circuit. Note that default parameters are defined in the subcircuit. To quickly make a subcircuit, in LTspice navigate to the Hierarchy tab, and select "Open this sheet's symbol". This will prompt you to generate a new symbol file, select yes. You can now access this symbol file under the component selection option. All you need to do is change the top directory (the drop down list at the top of the page). Your top directory will be wherever on your computer you created your original LTspice .asc file, and the symbol file will have been generated and must remain in that same directory/folder.
2. Connect a 0.1V DC voltage source in series with a source resistance of $50\ \Omega$ to the positive input terminal, connect the negative input terminal to ground, and a $1k\ \Omega$ resistor to ground at the output as shown in Figure 4.4 (b).
3. Run a Monte Carlo simulation for the gain, E, of the VCVS. Set the tolerance to 0.1 (10%) and run the simulation for 1000 iterations. The Monte Carlo function call can be passed in as a variable to the VCVS model where $E = \{mc\{10,tol\}\}$ in the Value field for the Value attribute. Plot the output voltage and save an image of the circuit and plot.
4. Repeat the Monte Carlo simulation for the input resistance, R_i . Save an image of the output voltage plot.
5. Repeat the Monte Carlo simulation for the output resistance, R_o . Save an image of the output voltage plot.

4.4.3 Feedback

The VCVS model developed in previous sections has been re-drawn using the more common symbol for an amplifier as shown in Figure 4.5 (a). However, the model is still the same and it's not required to redraw your model.

1. Construct the circuit as shown in Figure 4.5 (b). The resistors R_S and R_L remain $50\ \Omega$ and $1\ k\Omega$ respectively.
2. Run a Monte Carlo simulation for the gain, E, of the VSVS model. Set the gain to $10E5$, tolerance to 0.3 (30%), and run the simulation for 1000 iterations. Plot the output voltage and save an image of the circuit and plot.
3. Construct the circuit as shown in Figure 4.5 (c). Set R_2 to $2\ k\Omega$ and R_1 to $1\ k\Omega$. The resistors R_S and R_L remain $50\ \Omega$ and $1\ k\Omega$ respectively.

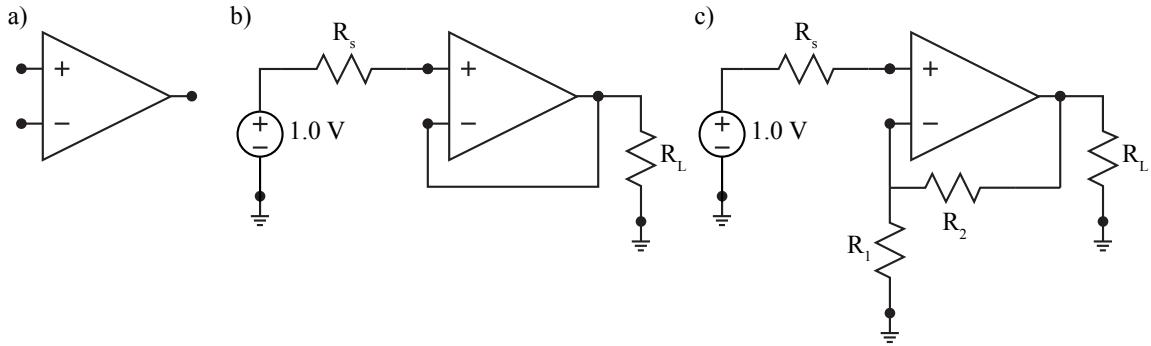


Figure 4.5: The VSCS model redrawn as the more common amplifier model symbol (a), unity gain configuration (b), and non-inverting gain configuration (c).

4. Run a Monte Carlo simulation for the gain, E , of the VSVS model. Set the gain to $10E5$, tolerance to 0.3 (30%), and run the simulation for 1000 iterations. Plot the output voltage and save an image of the circuit and plot.

4.5 In-Lab Requirements

The following must be completed and submitted to Canvas before the start of lab.

1. Subsection 4.4.1:
 - (a) Items 1 - 4: Table of input and output voltages.
 - (b) Item 5: Image of circuit and plot of the input voltage.
 - (c) Item 6: Image of circuit and plot of the output voltage.
2. Subsection 4.4.2:
 - (a) Item 3: Image of circuit and plot of the output voltage.
 - (b) Item 4: Plot of the output voltage.
 - (c) Item 5: Plot of the output voltage.
3. Subsection 4.4.3:
 - (a) Item 2: Image of circuit and plot of the output voltage.
 - (b) Item 4: Image of circuit and plot of the output voltage.

Complete the following items in lab.

4.5.1 Worst Case Analysis

While Monte Carlo analysis is useful for looking at a range of possible values, often times it's important to see the potential worst case values given a specified tolerance. Use the VCVS model that was previously developed with the values of R_i and R_o previously defined and a gain of 10E5.

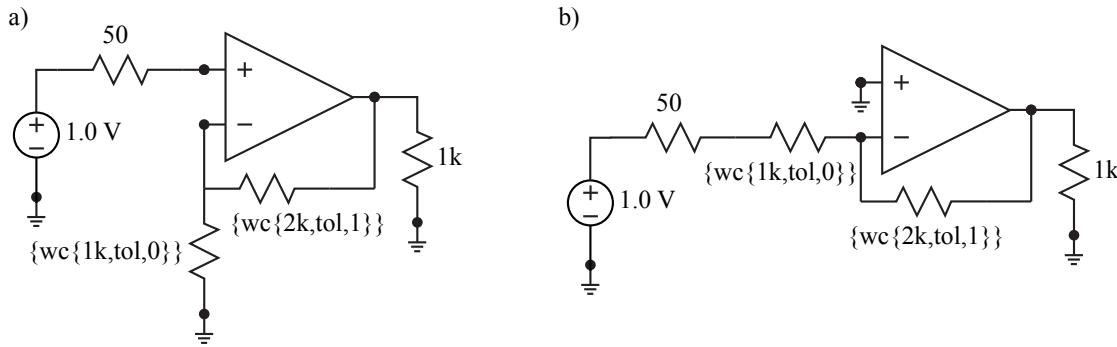


Figure 4.6: Non-inverting configuration with the necessary resistor values for a worst case simulation (a) and inverting configuration with the necessary resistor values for a worst case simulation (b).

1. Construct the circuit as shown in Figure 4.6 (a)
2. Run a DC operating point simulation with the following additional parameters:

```
.func binary(run,index) floor(run/(2**index))-2*floor(run/(2**index+1))
.func wc(nom,tol,index) if(run==numruns,nom,if(binary(run,index),nom*(1+tol),nom*(1-tol)))
.param numruns=4
.step param run 0 4 1
.param tol=0.05
```

3. Plot the output voltage and save an image of the plot. Determine the worst case results, which resistor tolerances, that gives the highest and lowest output voltages.
4. Repeat the simulation for a tolerance of 0.01 (1%), 0.005 (0.5%), 0.001 (0.1%), 0.0005 (0.05%), and 0.0001 (0.01%). For each case, table the high, low output voltages, and determine the cost of the components. Use the following Digikey link to determine pricing: <http://digikey.com>. Search through hole resistors, select the ones in-stock for 0.25 Watt, Carbon Film according to the required tolerance.
5. Construct the circuit as shown in Figure 4.6 (b)
6. Run a DC operating point simulation with the parameters previously stated.

7. Plot the output voltage and save an image of the plot. Determine the worst case results that gives the highest and lowest output voltages.
8. Repeat the simulation for a tolerance of 0.01 (1%), 0.005 (0.5%), 0.001 (0.1%), 0.0005 (0.05%), and 0.0001 (0.01%). For each case, table the high, low output voltages, and determine the cost of the components.

4.6 Write Up

Include the following in the write up.

1. Subsection 4.5.1
 - (a) Item 3: Output voltage plot for a 5% tolerance and the highest and lowest output voltage cases.
 - (b) Item 4: Tabled output voltage range and pricing for different resistor tolerances.
 - (c) Item 7: Output voltage plot for a 5% tolerance and the highest and lowest output voltage cases.
 - (d) Item 8: Tabled output voltage range and pricing for different resistor tolerances.

Discuss the benefits of feedback, worst case analysis for different resistor tolerances, and the cost/benefit trade off for different resistor tolerances.

For all lab write up submissions and reports the backgrounds for LTSpice and Waveforms should be changed from dark background to light background to make them readable for grading.

If the prelab simulation is incorrect, do not compare the in lab results to wrong values. Correct numbers or circuit schematics should be used in the write-up report.

Chapter 5

Lab 5 - Operational Amplifiers

5.1 Objective

The purpose of this lab is to introduce operational amplifiers through the ideal op amp model, spice op amp model, and practical applications of op amps in common configurations.

5.2 Materials

- Laptop with LTSpice
- Analog Discovery
- Breadboard
- Wiring kit
- Lab parts kit with TLV272

5.3 Introduction

There is a large variety of op amp and specialized amplifiers. For this lab the more recent TLV272 is used, a dual op amp, which contains two op amps, or channels, in a single chip.

5.3.1 The Ideal Op Amp

Recall that an ideal op amp will have the following properties.

- Zero input current, often called the input bias current and represented as I_B or I_{IB} .

- Zero input offset voltage, represented as either V_{IO} or V_{OS} .
- Infinite input resistance, represented as R_{in} or r_i .
- Zero output resistance, represented as R_{out} or r_o .
- Infinite gain, represented as A or α and often called the open loop gain.

These quantities are represented in Figure 5.1.

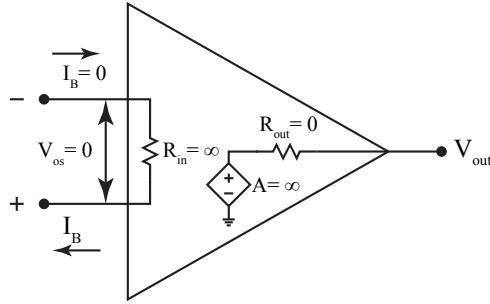


Figure 5.1: Basic op amp model.

However, as should be expected, practical op amp do not have these ideal quantities. For example, the Texas Instrument's TLV272 has finite values for all of the ideal properties. Note that these values vary and depend on production variation, temperature, and supply voltage.

- Finite input bias current, typically 1 pA.
- Finite input offset voltage, typically 0.5 mV.
- Finite input resistance, typically 1,000 GΩ differential input resistance.
- Finite gain, typically 115 dB (563k)

5.3.2 Common Configurations

There are a variety of configurations op amps can be used for but there are a few common configurations that form the basis of more complicated op amp circuits.

Inverting Amplifier

An inverting configuration, Figure 5.2 (a), takes its name from the inversion that happens from input to output, giving an output voltage proportional to R_2 and R_1 independent of the actual op amp properties.

$$V_{out} = -\frac{R_2}{R_1}V_{in} \quad (5.1)$$

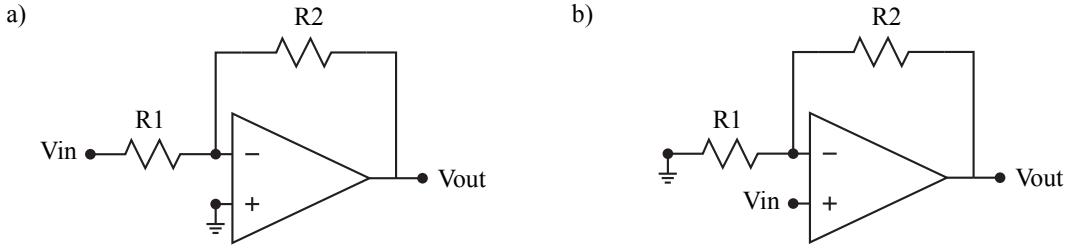


Figure 5.2: Different op amp configurations: inverting amplifier (a) and non-inverting amplifier (b)

Non-Inverting Amplifier

Similar to the inverting case, Figure 5.2 (b), this configuration results in an output voltage proportional to R_2 and R_1 but without an inversion and always has a gain of at least 1.

$$V_{out} = \left(1 + \frac{R_2}{R_1}\right)V_{in} \quad (5.2)$$

There is also a special case of the non-inverting amplifier where it is connected as a voltage follower or unity gain amplifier as shown in Figure 5.3.

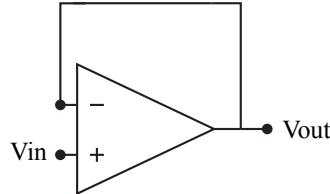


Figure 5.3: A non-inverting amplifier configured as a unity gain buffer.

The output is related directly to the input where

$$V_{out} = V_{in} \quad (5.3)$$

and while this configuration may seem redundant, it has the advantage of providing a large input impedance to the source, V_{in} , and a small output impedance to the load, V_{out} .

5.3.3 Op Amp Pins

A pinout is a connection diagram for an integrated circuit and shows which pins connect to which op amp terminals internally, not to be confused with the pins in a spice simulation. Op amps have a fairly common pinout resulting in chips that can

be swapped out easily. Figure 5.4 shows the pinout for the TLV272 with the op amp diagram drawn within.

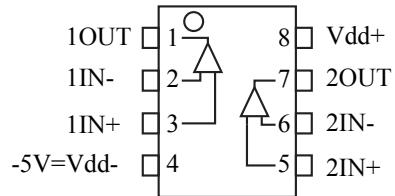


Figure 5.4: Pinout for TI's TLV272. Note that two op amps are contained, often called a dual op amp, within a single chip.

Name	Number	Description
1OUT	1	Output of the first op amp
1IN-	2	Inverting input of the first op amp
1IN+	3	Non-inverting input of the first op amp
Vdd-	4	Negative supply, connected to ground for single supply operation
2IN+	5	Non-inverting input of the second op amp
2IN-	6	Inverting input of the second op amp
2OUT	7	Output of the second op amp
Vdd+	8	Positive supply

Table 5.1: Pin descriptions from the TLV272 datasheet.

The pins are numbered from 1 to 8 and count counter-clockwise starting in the upper left and the individual pin descriptions are in Table 5.1. Most of the pins should be familiar from class.

Polarity or the orientation of the chip is determined by a circle or indentation on the chip itself, the TLV272 has a small circle indentation in the upper left hand corner to indicate polarity and the position of pin 1.

5.4 Big Picture

This lab focuses on op amps which will be used in the gain and outputs stages of the final project.

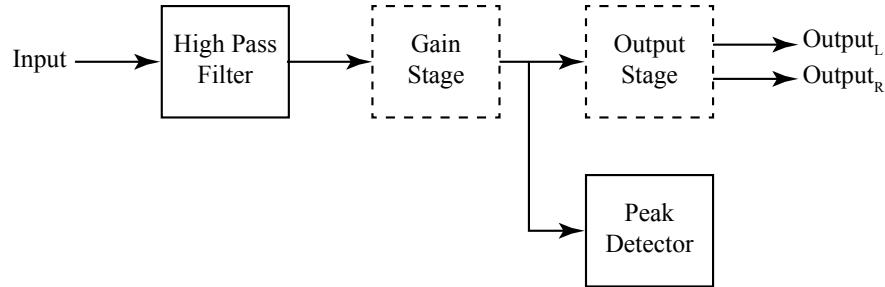


Figure 5.5: Big picture with emphasis on the gain and output stages.

5.5 Pre-Lab Requirements

Complete the following before coming to lab.

5.5.1 LTspice TLV272 Op Amp Model

Complete the following using the TLV271 spice model, always power the op amp with ± 5 V for V_{dd+} and V_{dd-} .

1. Obtain the TLV2 spice model from TI's website: <http://www.ti.com/lit/zip/slom249>. Note that this is the model for the TLV271 which is the same op amp that's duplicated in the TLV272. If you have trouble accessing the zip file using the link, the file can also be accessed from the "lab related files" folder on canvas.
2. Import the model in to LTspice using the instructions here: <http://www.linear.com/solutions/4678>. Make sure that the file, TLV271.lib, is extracted from the zip before opening and making the model, otherwise LTspice won't be able to reference the sub circuit file. Also, the pin numbers in the spice model correspond to the numbers in the sub circuit for the TLV272 and not the numbers in Table 5.1.
3. Read the TLV272 datasheet. <http://www.ti.com/lit/ds/symlink/tlv272.pdf>. Make sure note that pin 4 should be connected to -5 V, NOT ground. It is also recommended to use the schematic on page 42 rather than the schematic in the datasheet to avoid confusion when constructing your physical circuit.
4. Using the voltage follower configuration in Figure 5.3, run a DC operating point simulation (.op) and determine the V_{out} when V_{in} is zero volts; effectively calculating the offset voltage, table the value.
5. With the same voltage follower configuration, Figure 5.3, set V_{in} to a 1 V amplitude pulse. Vinitial(V): 0, Von(V): 1, Tdelay(s): 0.5m, Ton(s): 10s, Ncycle: 100. Leave any remaining variables blank. Run a transient analysis with a stop

time of 1m (.tran 1m). Plot the input and output voltage from 495 us to 505 us and then find then the time it takes for the output to reach 1 V, the time should be on the order of micro seconds. This is a measure of slew rate in V/us, table the value. Save an image of the circuit and the plot of the input and output voltage for submission to canvas.

6. Configure an inverting amplifier, Figure 5.2 (a), with a gain of 10 by setting $R_2 = 10\text{k}$ and $R_1 = 1\text{k}$. Set the input voltage to a sine wave with the following parameters, DC offset(V): 0, Amp(V): 1, Freq(Hz): 1k, and the remaining variables blank. Run a transient analysis with a stop time of 1m (.tran 1m). Plot the input and output voltage and record the max and min values of V_{out} , this is a measure of the maximum output swing, table the value. Save an image of the circuit and the plot of the input and output voltage for submission to canvas.
7. Using the same inverting configuration, Figure 5.2 (a), set V_{in} to a 0.1 V DC source and run a transient analysis with a stop time of 1m (.tran 1m). Plot $-V(vout)/V(vin)$, this is a measure of the effective gain, table the result. Save an image of the circuit and the plot of effective gain for submission to canvas.

5.5.2 Op Amp Construction

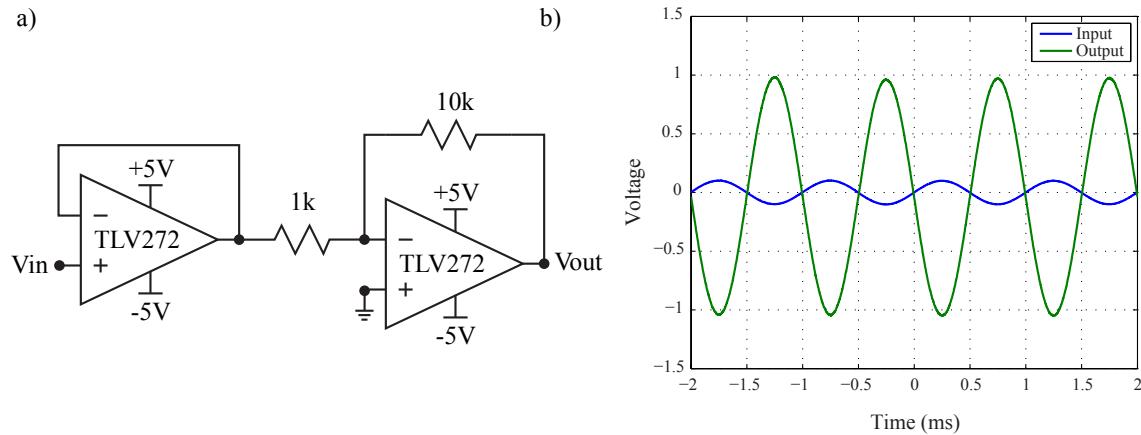


Figure 5.6: Schematic for the demo circuit, a voltage follower followed by an inverting amplifier with a gain of 10 (a) and the resulting output (b)

1. Using only one of the TLV272 op amp in your lab parts kit, build the circuit in Figure 5.6 (a) on a breadboard. Set the voltage supplies to +/- 5 V and scope channel 1 to Vin and scope channel 2 to Vout. Configure the Wavegen to supply a 1 kHz sine wave with a 0.1 V amplitude.

5.6 In-Lab Requirements

The following must be completed at the start of lab.

1. All of the theoretical parameters from Subsection 5.5.1 tabled:
 - (a) Offset voltage.
 - (b) Slew rate.
 - (c) Maximum output swing.
 - (d) Effective gain.
2. Images of the output and circuits from Subsection 5.5.1 tabled:
 - (a) Item 5: Image of the circuit and the plot of the input and output voltage.
 - (b) Item 6: Image of the circuit and the plot of the input and output voltage.
 - (c) Item 7: Image of the circuit and the plot of effective gain.
3. Op amp configurations from Subsection 5.5.2 built on a breadboard and working.

5.6.1 Experimental Op Amp Measurements

Using the pre-built op amp configurations, complete the following.

1. Using the voltage follower, Figure 5.3, input a 0 V DC voltage from the Wavegen and record the offset voltage. Choose an appropriate voltage scale, the voltage is on the order of mV. Save a screenshot of the waveform scope.
2. Using the voltage follower again, Figure 5.3, input a 0.5 V amplitude square wave with a 0.5 V offset from the Wavegen. Set the appropriate trigger and determine the slew rate in V/us. Save a screenshot of the waveform scope.
3. Using the inverting amplifier, Figure 5.2 (a), input a 1k Hz sine wave with a 1 V amplitude from the Wavegen. Determine the maximum output voltage swing.
4. Using the inverting amplifier again, Figure 5.2 (a), input a 0.1 V DC from the Wavegen. Determine the effective gain. Choose an appropriate voltage scale for the input and output voltages.
5. Table all of values calculated in this subsection.

5.7 Write Up

Include the following in the write up.

1. Table of experimental parameters.
2. Screenshots of scope output plots.
3. Percent error, using the pre-lab values as the theoretical and the in-lab values as experimental.

Discuss the differences between the ideal op amp model, spice TLV272 model, and the physical op amp focusing on the quantities calculated for this lab: offset voltage, slew rate, output voltage range, and effective gain. Touch on the following in your discussion.

- When is it appropriate to use one model over the other?
- How accurate should the spice model for an op amp be?

For all lab write up submissions and reports the backgrounds for LTSpice and Waveforms should be changed from dark background to light background to make them readable for grading.

If the prelab simulation is incorrect, do not compare the in lab results to wrong values. Correct numbers or circuit schematics should be used in the write-up report.

Chapter 6

Lab 6 - Operational Amplifiers Applications

6.1 Objective

The purpose of this lab is to delve in to the applications of operational amplifiers. There is an almost endless number of applications, the possibilities are often not intuitively obvious for students seeing them for the first time.

6.2 Materials

- Laptop with LTSpice
- Analog Discovery
- Breadboard
- Wiring kit
- Lab parts kit with TLV272

6.3 Introduction

There are a variety of configurations op amps can be considered standard and while the simple inverting and non-inverting cases were show in Lab 5, here the pool is expanded to include a summing amplifier and a difference amplifier.

Summing Amplifier

A summing amplifier, Figure 6.1 (c), is effectively the same configuration as an inverting amplifier but has multiple inputs that allow for a weighted output voltage.

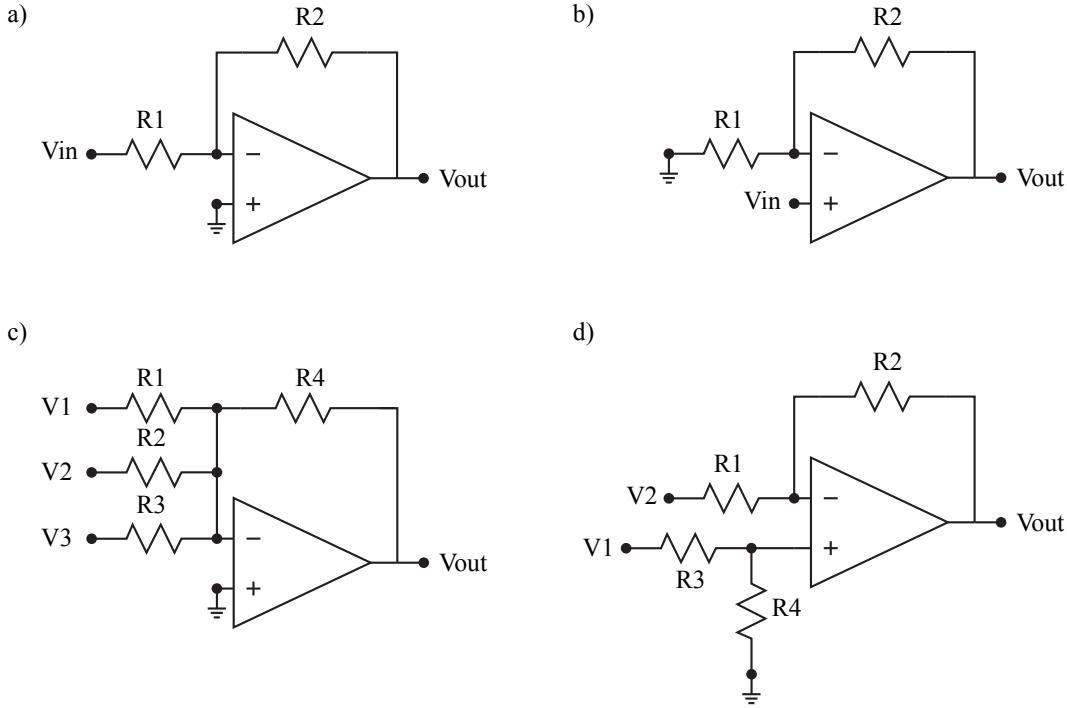


Figure 6.1: Different op amp configurations: inverting amplifier (a), non-inverting amplifier (b), summing amplifier (c), and difference amplifier (d).

$$V_{out} = -\left(\frac{R_4}{R_1}V_1 + \frac{R_4}{R_2}V_2 + \frac{R_4}{R_3}V_3\right) \quad (6.1)$$

When the resistors R_1 , R_2 , R_3 , and R_4 are all equal, the output voltage is an equal combination of the input voltages but the resistors can be chosen to weight the input voltages different based on the application.

Difference Amplifier

Difference amplifiers, Figure 6.1 (d), as the name implies takes the difference between the two input voltages resulting in that depends on the ratios of resistor values.

$$V_{out} = V_1 \left(\frac{R_2}{R_1 + R_2} \right) \left(\frac{R_3 + R_4}{R_3} \right) - V_2 \frac{R_4}{R_3} \quad (6.2)$$

When the resistors $R_1 = R_3$ and $R_2 = R_4$ the output voltage simplifies to

$$V_{out} = (V_1 - V_2) \frac{R_2}{R_1} \quad (6.3)$$

6.4 Big Picture

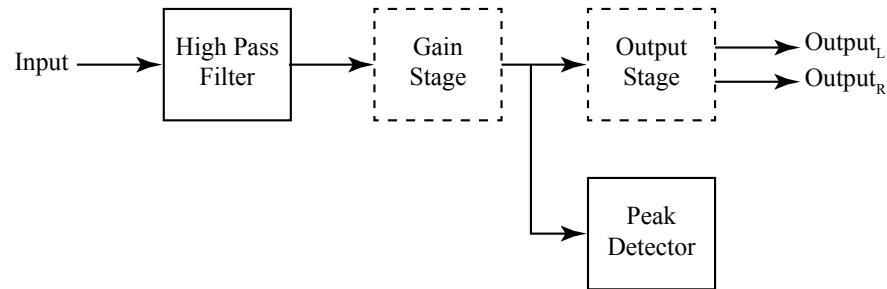


Figure 6.2: Big picture with emphasis on the gain and output stages.

This lab again focuses on op amps but op amps used as different building blocks to compose more complicated circuits. The final project circuit is about as complicated as the final circuit in this lab.

6.5 Pre-Lab Requirements

Complete the following before coming to lab.

6.5.1 Buffering

Buffering is an important op amp application because it solves a problem, loading effects, that can't easily be solved with purely resistive circuits.

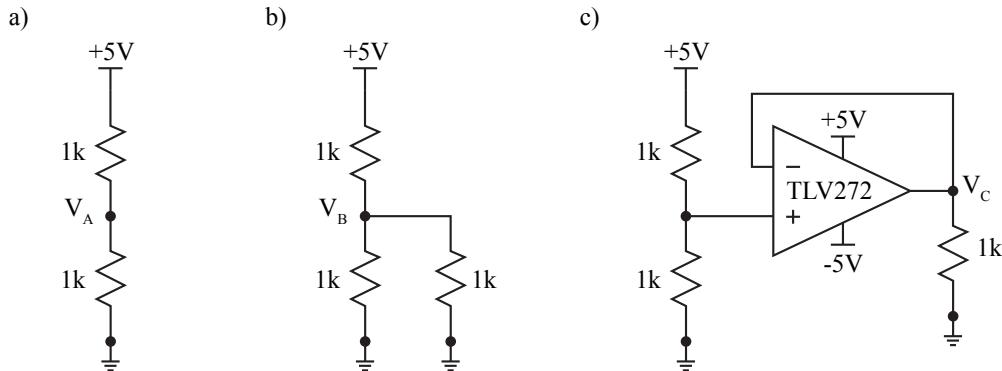


Figure 6.3: A simple voltage divider (a), voltage divider with a load (b), and a voltage divider with a voltage follower between the output and the load (c).

1. Simulate all three circuits in Figure 6.3 using a DC operating point (.op). Keep all the circuits on the same schematic and table the voltages V_A , V_B , and V_C .

Save an image of the circuit and submit the image and table in a document to Canvas.

2. Discuss why two of the voltages are approximately 2.5 V but one is not. Submit your answer as part of the document submitted to Canvas.

6.5.2 Level Shifting

Level shifting in electronic circuits is the process of change the supply rails from one range to another. For example, when preparing a signal for analog to digital conversion, the signal may be bi-polar (+/-) and must be converted to a single voltage rail. So, instead of a sine wave that swings to +2.5V and -2.5V, 2.5V is added so that the sine wave swings from +5V to 0V with the same amplitude.

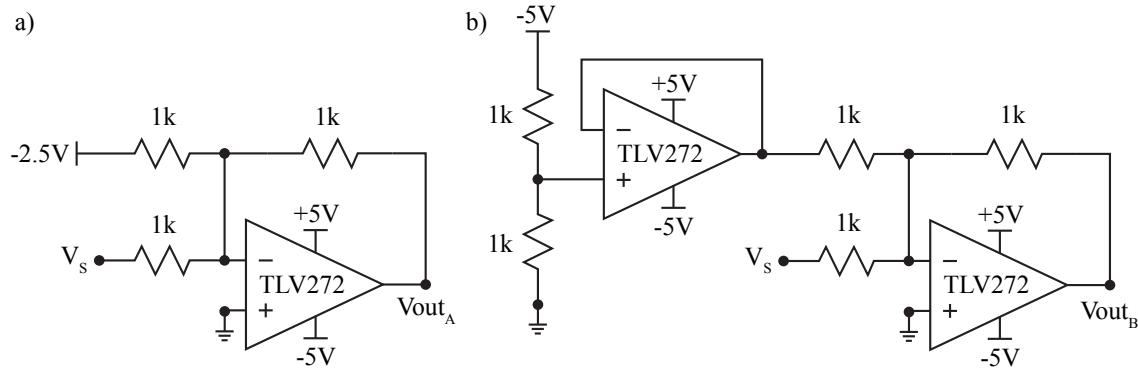


Figure 6.4: A summing amplifier used to add 2.5V to $-V_{IN}$ (a) and the same configuration where the -2.5V source has been replaced with a buffered voltage divider (b).

1. Simulate the circuit in Figure 6.4 (a) using a transient analysis with a stop time of 1m (.tran 1m). Set V_S to a sine wave with a 0 V DC offset, 2 V amplitude, and a 1 kHz frequency. Save an image of the circuit and a plot of the input and output voltage for submission to Canvas.
2. Explain why the resistor divider in Figure 6.4 (b) uses -5V instead of 5V. Submit your answer as part of the document submitted to Canvas.
3. Practically, a 2.5V reference may not be available for use and instead must be self generated. Simulate the circuit in Figure 6.4 (b) using a transient analysis with a stop time of 1m (.tran 1m). Set V_S to a sine wave with a 0 V DC offset, 2 V amplitude, and a 1 kHz frequency. Save an image of the circuit and a plot of the input and output voltage for submission to Canvas.
4. Construct the circuit in Figure 6.4 (b) on a breadboard. Configure the Wavegen to generate a sine wave at 1 kHz with a 2 V amplitude and a 0V offset. Plot the

input and output on the Scope and save the circuit to demo at the start of lab.

5. In the more general case, the full range of input must be accounted for. Simulate the circuit in Figure 6.5 using a transient analysis with a stop time of 1m (.tran 1m). Set V_{IN} to a sine wave with a 0 V DC offset, 5 V amplitude, and a 1 kHz frequency. Save an image of the circuit and a plot of the input and output voltage for submission to Canvas.

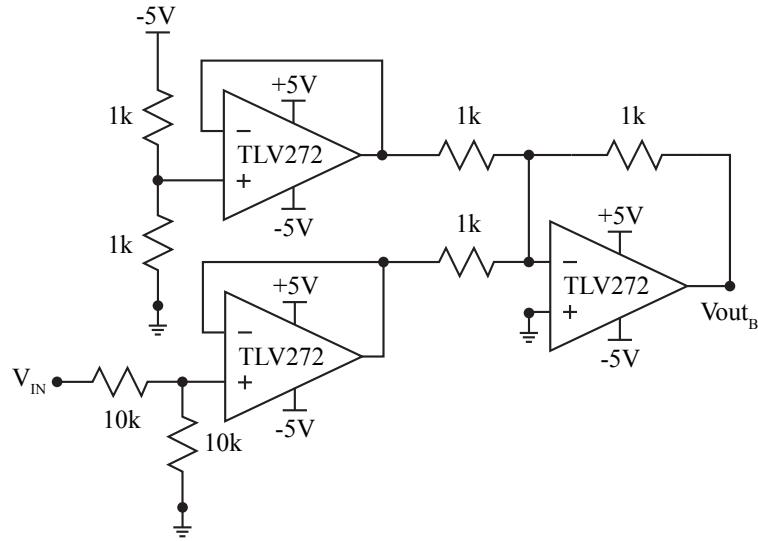


Figure 6.5: A DC level shifter configured to convert the full range of +/-5 to +5/0.

6.5.3 Differential to Single Ended Conversion

Differential signals, two signals where one signal is 180 degrees out of phase with the other signal, have a variety of applications. The primary benefit of differential signals is increased noise performance but at the cost of extra hardware, you need two of everything. Converting from a differential signal to a single ended signal can be accomplished fairly simply with a difference amplifier.

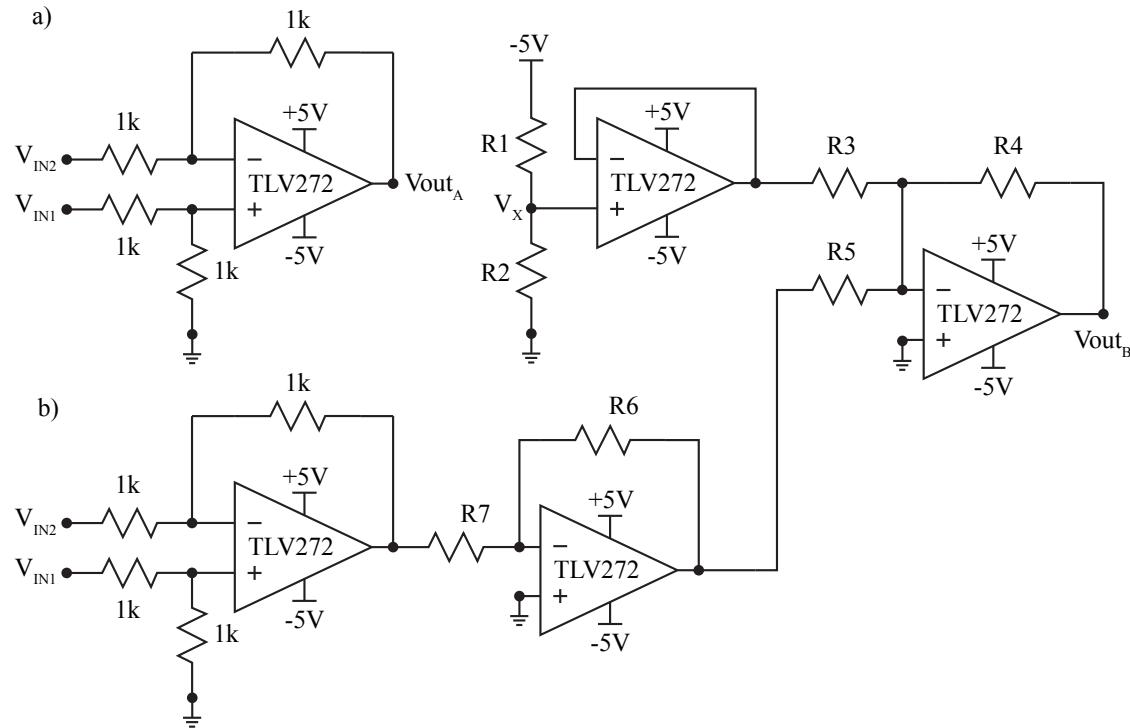


Figure 6.6: A difference amplifier that converts a differential signal to single ended (a) and a combination of the level shifter and differential to single ended converter (b).

1. Simulate the circuit in Figure 6.6 (a) using a transient analysis with a stop time of 1m (.tran 1m). Set V_{IN1} to a sine wave with a 0 V DC offset, 0.5 V amplitude, and a 1k Hz frequency. Set V_{IN2} to a sine wave with a 0 V DC offset, 0.5 V amplitude, a 1k Hz frequency, and a phase of 180 degrees. Save an image of the circuit and a plot of the input (both inputs) and output voltages for submission to Canvas.
2. Determine the resistor values for $R1$ through $R7$, recall that there are limited quantities in the lab kit. Choose $R1$ and $R2$ so that V_x is 2.5V, $R3$, $R4$, and $R5$ so that both voltages are added equally, and $R6$ and $R7$ so that the output, V_{out_B} utilizes as much of the full range, 0 to 5V, as possible without clipping.

3. Simulate the circuit in Figure 6.6 (b) using a transient analysis with a stop time of 1m (.tran 1m). Set V_{S1} and V_{S2} to their previously defined values. Save an image of the circuit and a plot of the input (both inputs) and output voltages for submission to Canvas.
4. Explain how the circuit in Figure 6.6 (b) works in a few sentences. Submit your answer as part of the document submitted to Canvas.

6.6 In-Lab Requirements

The following must be completed at the start of lab.

1. Section 6.5.1
 - (a) Item 1: Image of the circuit and a table of output voltages.
 - (b) Item 2: Answer to question.
2. Section 6.5.2
 - (a) Item 1: Image of the circuit and a plot of the input and output voltages.
 - (b) Item 2: Answer to question.
 - (c) Item 3: Image of the circuit and a plot of the input and output voltages.
 - (d) Item 4: Plot of the input and output voltages and a circuit demonstration.
 - (e) Item 5: Image of the circuit and a plot of the input and output voltages.
3. Section 6.5.3
 - (a) Item 1: Image of the circuit and a plot of the input and output voltages.
 - (b) Item 3: Image of the circuit and a plot of the input and output voltages.
 - (c) Item 4: Answer to question.

Complete the following in lab.

6.6.1 Construction

1. Construct the circuit from Section 6.5.2 Item 5. Set Wavegen1, V_{IN} , to a sine wave with a 5 V amplitude, 0 V offset, and a frequency of 1 kHz.
2. Plot the input and output voltages, save an image of the plot.
3. Construct the circuit from Section 6.5.3 Item 3. Set Wavegen1 to V_{IN1} and Wavegen2 to V_{IN2} . Synchronize wavegen1 and wavegen2 using the dropdown in the wavegen tab of waveforms. It is located two tabs to the right of "Run all" and is defaulted to "No synchronization".
4. Plot the input (differential) and the output voltages, save an image of the plot.

6.7 Write Up

Include the following in the write up.

1. Section 6.6.1
 - (a) Item 1: Schematic of the circuit, do not copy the schematic from the lab manual, an image from LTspice is fine.
 - (b) Item 2: Plot of the input and output voltages.
 - (c) Item 3: Schematic of the circuit, do not copy the schematic from the lab manual, an image from LTspice is fine.
 - (d) Item 4: Plot of the input and output voltages.

Discuss the two circuits constructed in lab. Explain how each circuit works at the block level and how resistor tolerance could or could not have an effect on the output. Also explain the design choices for the circuit, Section 6.5.3 Item 3, constructed in lab. Which resistors were chosen and why?

Chapter 7

Lab 7 - Capacitor Applications

7.1 Objective

The objective of this lab is to introduce another fundamental circuit element, the capacitor, and their applications within certain circuits. Specifically for this lab, the focus is on a capacitor used in conjunction with a 555 timer.

7.2 Materials

- Laptop with LTSpice
- Analog Discovery
- Breadboard
- Wiring kit
- Lab parts kit

7.3 Introduction

There are a variety of applications of capacitors which could fill an entire class. The focus for this lab will be the use of a capacitor and its associated RC time constant in conjunction with a 555 timer.

7.3.1 Time Domain Response

While the operation of a capacitor can be analyzed in both the time and frequency domains, the focus here is on the time domain.

A simple RC circuit with a switch as shown in Figure 7.1 will charge and discharge as the 1 V source is connected and disconnected. Charging to the source voltage takes the form of

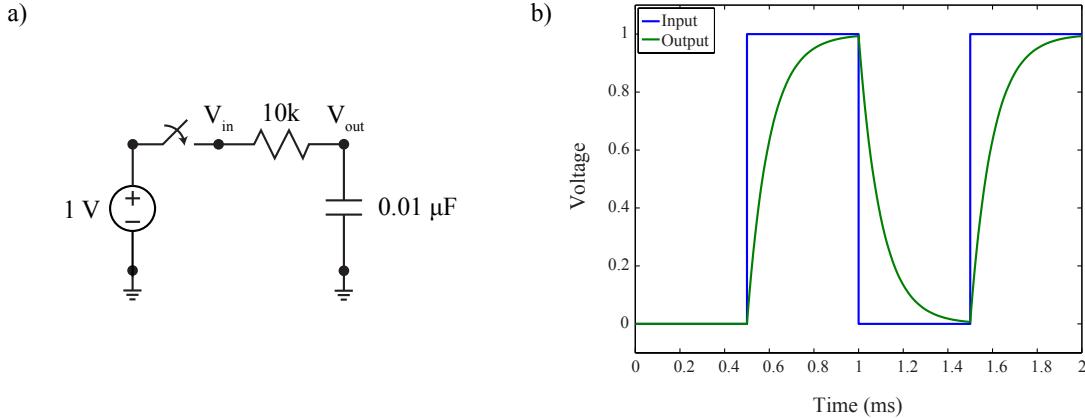


Figure 7.1: A simple RC circuit (a) and the associated input and output (b). Note that the input is the node after the switch in (a).

$$V_{out} = 1 - V_{in}e^{-t/\tau} \quad (7.1)$$

where $\tau = RC$ and discharging takes the form

$$V_{out} = V_{in}e^{-t/\tau} \quad (7.2)$$

While the result is novel and opens up a lot of possibilities, there isn't a clear path to the actual application of the capacitor. Sure, it can be used in the previously mentioned simple RC circuit, but there isn't a clear use. Enter the 555 timer, an early chip that became incredibly popular due to the ability to generate various timing signals using external components to set an RC time constant.

7.3.2 555 Timer

A 555 timer can operate in two modes, mono-stable and a-stable. Mono-stable operating is when the circuit will only output once when triggered and will wait for future triggers to output again. A-stable operating requires no input and will internally trigger to perpetually produce output.

For clarification, there is a distinction between the pulse duration, the time a signal is high (highest positive voltage, usually the positive voltage rail) and the period. Figure 7.2 shows the difference between the pulse duration and period for a square wave. The pulse duration is the time the square wave is at its maximum value, 1 V in this case, for a total of 0.25 ms. While the period is the time it takes for the signal to repeat, 0.5 ms.

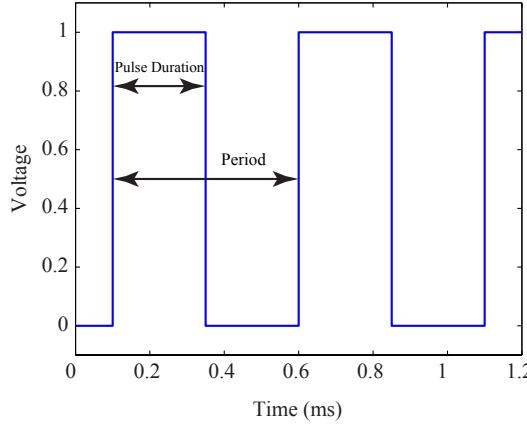


Figure 7.2: A plot of a square wave with the pulse duration and period labeled.

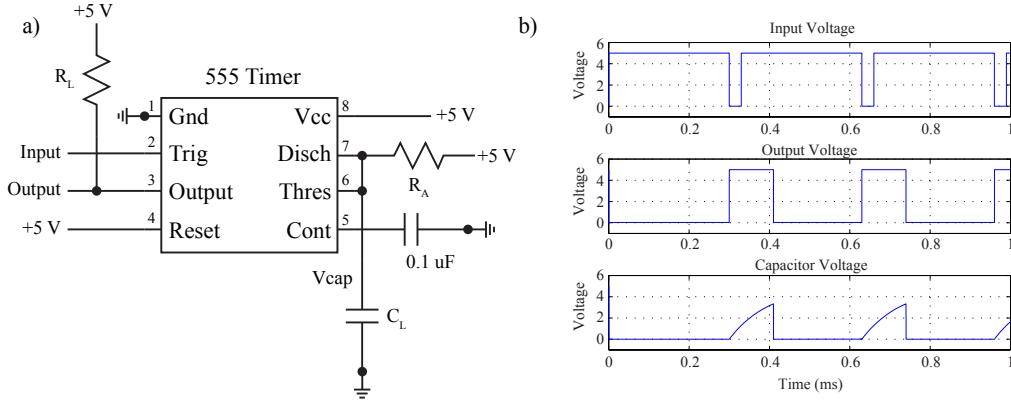


Figure 7.3: The circuit for mono-stable operation (a) and the output voltages (b). V_{cc} is 5 V, $R_L = 1\text{k }\Omega$, $C_L = 0.01\mu\text{F}$, and $R_A = 10\text{k }\Omega$. Note that wires that cross are only connected if there is a dot at the junction.

Mono-Stable Configuration

The mono-stable configuration, Figure 7.3, requires an input and only triggers when that input passes below a specific threshold (active low). Once the input falls below the threshold voltage, the output goes to 5 V for $t = 1.1R_AC_L$ and the capacitor starts to charge. Once the capacitor voltage reaches a threshold, the output and the capacitor voltage both go to zero and the circuit waits for the input voltage to go zero again.

A-Stable Configuration

The a-stable configuration, Figure 7.4, does not require an input and utilizes the charging and discharging of the capacitor to trigger the input. With the addition of a second resistor, R_B , the capacitor will charge through resistors R_A and R_B but the

capacitor will only discharge through R_B , leading to two different time constants.

As the capacitor discharges, the output is zero until the capacitor voltage reaches the lower threshold voltages and then the output goes to 5 V until the capacitor voltage reaches the upper threshold and the process repeats.

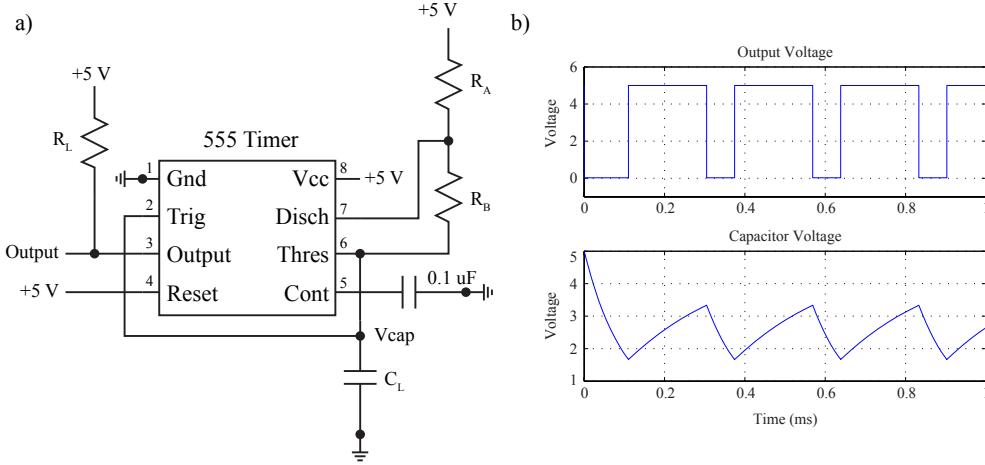


Figure 7.4: The circuit for a-stable operation (a) and the output voltages (b). V_{cc} is 5 V, $R_L = 1k \Omega$, $C_L = 0.01\mu F$, $R_A = 10k \Omega$, and $R_B = 10k \Omega$. Note that wires that cross are only connected if there is a dot at the junction.

The time that the output is high is set by the two resistors and capacitor:

$$t_H = 0.693(R_A + R_B)C_L \quad (7.3)$$

While the time the output is low is only set by R_B and the capacitor:

$$t_L = 0.693(R_B)C_L \quad (7.4)$$

7.4 Big Picture

While capacitors are used in the finale project, they're used for filtering purposes as opposed to use in the time domain with the 555 timer.

7.5 Pre-Lab Requirements

Complete the following before coming to lab.

7.5.1 LTspice Simulations

Using LTspice, complete the following. Use 5 V for V_{cc} , $R_L = 1k \Omega$, and the NE555 model for the 555 timer found in the Misc component section.

1. Read the NE555 datasheet, <http://www.ti.com/product/NE555/datasheet>. Please note that a mono-stable allows for the pulse width to change as resistor and capacitor values are varied, while an a-stable configuration allows for the period to be varied as resistor and capacitor values are varied.
2. Using the mono-stable configuration, determine the output when $R_A = 15k \Omega$ and $C_L = 0.01\mu F$. Set the input to a pulse source with the following settings: 0 5 0 1n 1n 0.2m 0.25m 5 and run a transient simulation with a stop time of 1m (.tran 1m). Table the output voltage pulse duration, then save an image of the circuit and a plot of the input and output voltage for submission to Canvas.
3. Using the mono-stable configuration, choose R_A and C_L so that the output pulse duration is approximately 200 μs ($f \approx 5$ kHz) for a 100 kHz input; the solution should be within 4 μs , 200 μs +/- 4 μs (5 kHz +/- 100 Hz). Use only the parts in your lab kit. Set the input to a pulse source with the following settings: 0 5 0 1n 1n 8u 10u 50 and run a transient simulation with a stop time of 0.5m (.tran 0.5m). Table the output voltage pulse duration, then save an image of the circuit and a plot of the input and output voltage for submission to Canvas.
4. Using the a-stable configuration, determine the output when $R_A = 10k \Omega$, $R_B = 10k \Omega$, and $C_L = 0.01\mu F$. Run a transient simulation with a stop time of 1m (.tran 1m). Table the output voltage period, then save an image of the circuit and a plot of the output voltage for submission to Canvas.
5. Using the a-stable configuration, choose R_A , R_B , and C_L so that the frequency of the output signal is 10 kHz, there is an exact solution. Use only the parts in your lab kit. Run a transient simulation with a stop time of 0.5m (.tran 0.5m). Table the output voltage period, then save an image of the circuit and a plot of the output voltage for submission to Canvas.

7.5.2 Breadboard Implementation

1. Build a mono-stable configuration from Section 7.5.1 Item 2. Set Vcc to 5 V supply and use a 5 kHz square wave as the input with a 2.5 V amplitude and a 2.5 V offset. You can determine the capacitor values by their quantity or the labels: 102J - 0.001 μF , 103J - 0.01 μF , 333J - 0.033 μF , 473J - 0.047 μF , 104J - 0.1 μF , and 105K - 1 μF .
2. Display the output voltage and the capacitor voltage on the o-scope to demonstrate the circuit to your lab instructor at the start of lab. In this case it helps to ground the negative terminals for the o-scope probes.

7.6 In-Lab Requirements

The following must be completed at the start of lab.

1. Subsection 7.5.1
 - (a) Item 2: An image of the circuit and a plot of the input and output voltage.
 - (b) Item 3: An image of the circuit and a plot of the input and output voltage.
 - (c) Item 4: An image of the circuit and a plot of the output voltage.
 - (d) Item 5: An image of the circuit and a plot of the output voltage.
 - (e) Table of both theoretical pulse durations and periods.
- (a) Section 7.5.2: Mono-stable configuration implemented and working on a breadboard.

7.6.1 Construction

For all four cases, determine the experimental pulse durations/periods for the output voltage and table the resulting values.

1. Save an image of the output voltage and capacitor voltage on the o-scope for the circuit from Subsection 7.5.2.
2. Construct the circuit from Section 7.5.1 Item 3 on a breadboard and save an image of the output voltage and capacitor voltage on the o-scope.
3. Construct the circuit from Section 7.5.1 Item 4 on a breadboard and save an image of the output voltage and capacitor voltage on the o-scope.
4. Construct the circuit from Section 7.5.1 Item 5 on a breadboard and save an image of the output voltage and capacitor voltage on the o-scope.

7.7 Write Up

Include the following in the write up.

1. All four plots from the in-lab section detailing the output voltage and capacitor voltage.
2. Tabled experimental pulse durations/periods for the output voltages and percent error when compared to the theoretical values in the pre-lab. Pre-lab values and pre-lab plots are not required.

Discuss the differences between the theoretical outputs from LTspice and the experimental outputs when implementing the circuits on a breadboard. Why would they be different and do the scope probes adversely affect the results?

Chapter 8

Lab 8 - Diode Applications

8.1 Objective

The objective of this lab is to introduce diodes and a few of their applications.

8.2 Materials

- Laptop with LTSpice
- Analog Discovery
- Breadboard
- Wiring kit
- Lab parts kit

8.3 Introduction

Diodes are a fundamental circuit element whose one-directional current flow has a variety of applications. Diodes can be used as indicators (light emitting diodes (LEDs)), voltage protection elements, and rectification.

8.3.1 Diode Operation

Each diode has an anode and cathode, Figure 8.1 (a), and conducts current one-way once the voltage across the diode has reached the forward voltage, often called forward bias, $V_F \approx 0.7$ V. A diode can also conduct current in the opposite direction when the voltage across the diode is negative, often called reverse bias, where the breakdown voltage, V_{BR} , is large, usually -50 V or less. There is often a small current that flows when a diode is reverse bias, but we will neglect that effect for the purposes

of this lab. The schematic symbol for an LED is similar, Figure 8.1 (b), but indicates that light is generated when the diode is operating with a forward bias. And the I-V curve for a typical diode, Figure 8.1 (c).

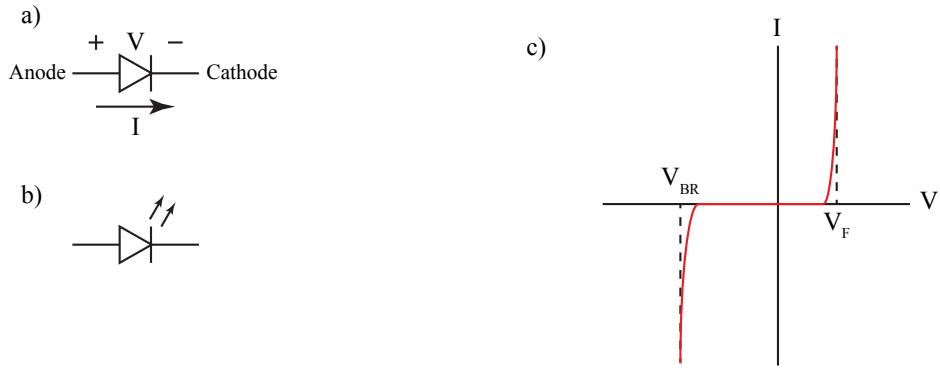


Figure 8.1: Diode schematic symbol with the anode and cathode labeled (a), schematic symbol for an LED (b), and the I-V curve for a diode (c).

8.3.2 Applications

There are a variety of applications for diodes and this lab will focus on only a few, rectifications and clip detection.

Rectifiers

Signal rectification is the process of converting an alternating signal in to its positive or negative components. Figure 8.2 is an example of a simple half-wave rectifier. After the input sine wave passes the diodes forward bias voltage, ≈ 0.7 V, the diode conducts and passes the positive half of the sine wave but doesn't pass the negative half of the sine wave because the diode is reverse bias and does not conduct.

Simply adding a capacitor to the half-wave rectifier results in an AC to DC converter, Figure 8.3. The circuit initially behaves as it did previously, once the input sine wave reaches the forward bias voltage, the output follows the input. However, once the sine wave starts to decrease, the output then discharges as a function of an RC time constant, the process then repeats. For higher input frequencies and capacitances, the output can be approximated as a DC source.

Clip Detection

There are often cases where the input or output of a system isn't easily accessible for test equipment. A variety of dials, indicators, and display panels are often used to report on the status of a system, examples include current draw, internal voltages, and clip detection.

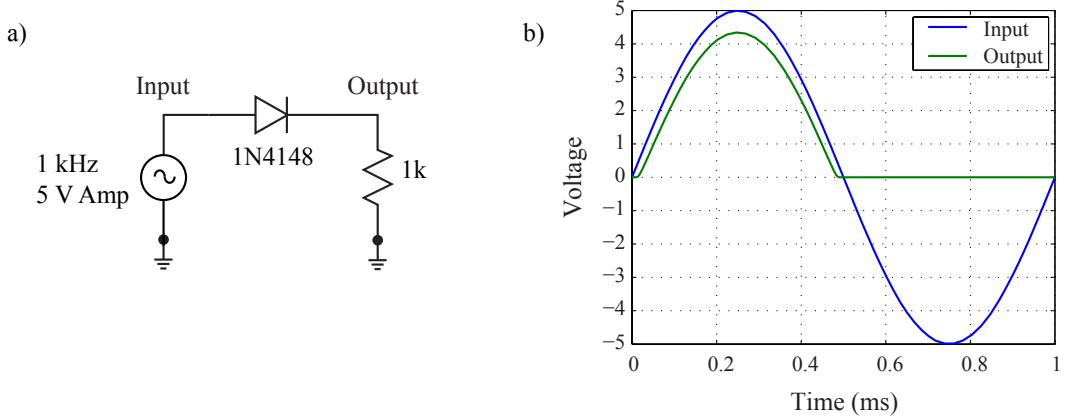


Figure 8.2: A simple half wave rectifier (a) and the resulting input and output signals (b).

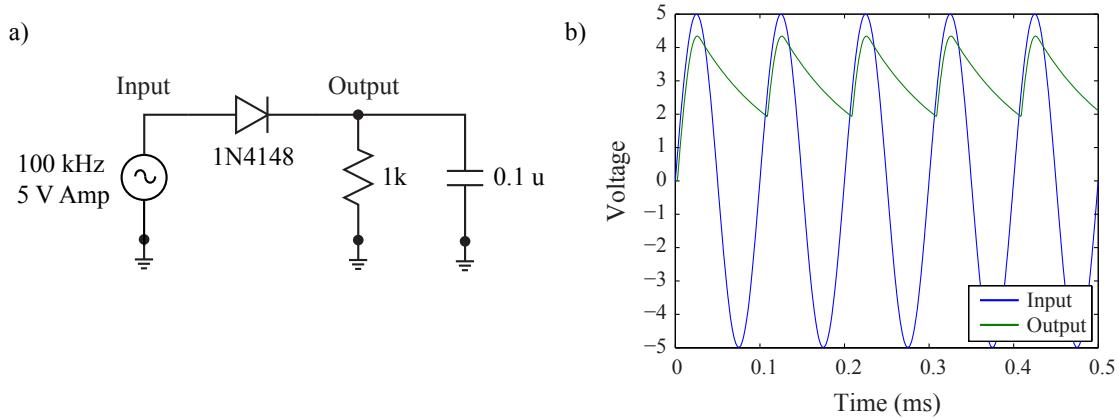


Figure 8.3: A simple half wave rectifier but with a capacitor at the output (a) and the resulting input and output signals (b).

Clip detection is the process of determining if a signal is clipping or not. A signal that is clipped, unless desired, can degrade the performance of the system. A clipping detection circuit is able to indicate that the output of the op amp is clipping without viewing the output on an o-scope, but instead through an LED.

Figure 8.4 is an example of a clipping detector. If the input is higher, or lower, than the reference voltage V_A , or V_B , the LED D1, or D2, illuminates indicated that the input voltage is above, or below, the reference voltage. Note that this circuit does not use op amps, but comparators. Comparators, as the name implies, compares the input at both terminals and produces a digital output, either a 1 (5 V) or 0 (0V), if the positive terminal input is larger than the negative terminal input.

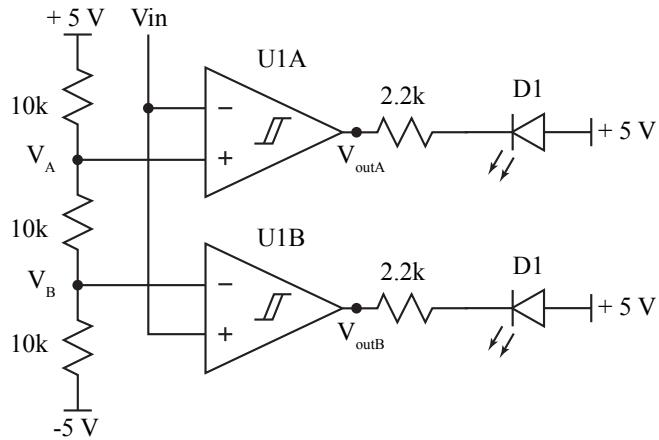


Figure 8.4: A clipping detection circuit. The input is compared to fixed voltages, V_A and V_B , by U1A and U1B respectively. If the input voltage is greater, or less, than the reference voltage, the output is pulled low and LED is illuminated.

8.3.3 Potentiometers

A potentiometer (or “pot”) is a three terminal resistor as shown in Figure 8.5. The resistance between terminals 1 and 3 is fixed and is equal to the rated value for the potentiometer. Terminal 2 is connected to a movable contact called the arm or wiper and the resistance between terminals 2 and 1 or between 2 and 3 can be varied by moving the arm. If terminals 1 and 3 are connected across a voltage source, then the voltage between terminals 2 and 1 or between 2 and 3 can be varied by moving the arm. In some cases, terminal 2 is connected to either terminal 1 or 3 so that the resistance from terminals 1 to 3 can be varied.

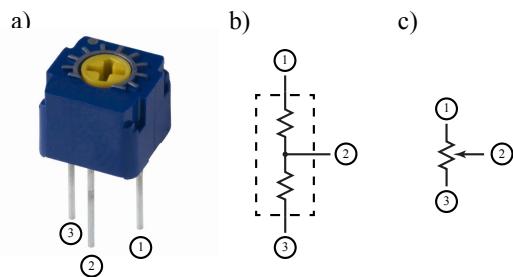


Figure 8.5: A picture of a practical potentiometer with the appropriate pins labeled (a), an equivalent circuit diagram of a potentiometer with the wiper in a fixed position (b), and the common circuit schematic representation for a potentiometer (c).

The potentiometer model in spice is as simple a single resistor, with a resistance equal to the total resistance or less than the total resistance, where the values changes simulation to simulation to the desired resistance.

8.4 Big Picture

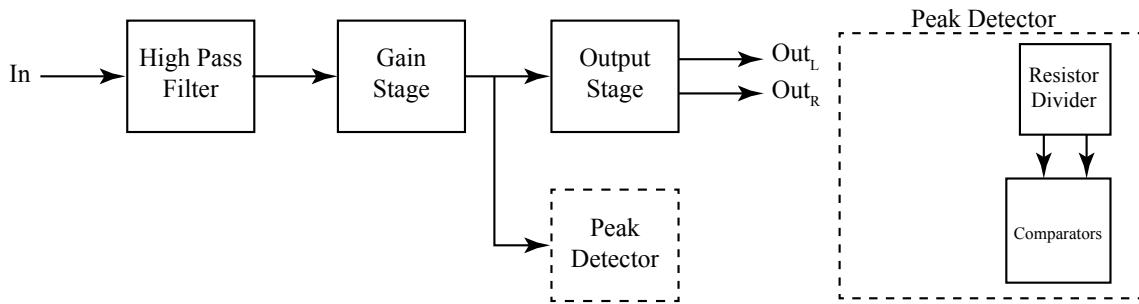


Figure 8.6: Big picture with emphasis on the peak detector.

This lab focuses on diode circuits which will help to form the comparator circuits used in the peak detector for the final project.

8.5 Pre-Lab Requirements

Complete the following before coming to lab.

8.5.1 LTspice Simulations

Using LTspice, complete the following.

1. Build the circuit in Figure 8.2 (a). Set the input to a 1 kHz, 5 V amplitude sine wave and run a transient simulation with a stop time of 1m (.tran 1m) and plot the input and output. To use the 1N4148 diode model, right click on the diode after placing it, Pick New Diode, and then choose the 1N4148 model (Mfg. OnSemi). Save an image of the circuit and the plot of the input and output voltage for submission to canvas.
2. Build the circuit in Figure 8.3 (a). Set the input to a 100 kHz, 5 V amplitude sine wave and run a transient simulation with a stop time of 50u (.tran 50u) and step through possible capacitor values using the following spice directive:

```
.step param C list 0.001u 0.01u 0.1u 1u
```

Save an image of the circuit and the plot of the input and output voltage for submission to canvas.

3. Build the circuit in Figure 8.4. Use the default LED model in LTSpice, and download "slcj016.zip" file from Canvas "Lab Related Files" folder for the LM393 comparator model. Don't use the "slcj016b.zip" for the comparator model from

TI because it's a newer model that doesn't work for input higher than (Vcc-2V). Power the LM393 with +/-5 V. Set the input voltage to a 1 kHz, 5 V amplitude sine wave and run a transient solution with a stop time of 2m (.tran 2m). Plot the input voltage and the current through each diode. Save an image of the circuit and the plot for submission to canvas.

- Build the circuit in Figure 8.7. Power the LM393 and TLV272 with +/-5 V. Set the input voltage to a 1kHz, 0.1 V amplitude sine wave and run a transient solution with a stop time of 2m. Choose the value for R1, 10k pot, so that the diodes conduct current and illuminate. Plot the input voltage, output of the op amp, positive and negative inputs to the comparator, and the current through both LEDs. (6 items total). Save an image of the circuit and the plot for submission to canvas.

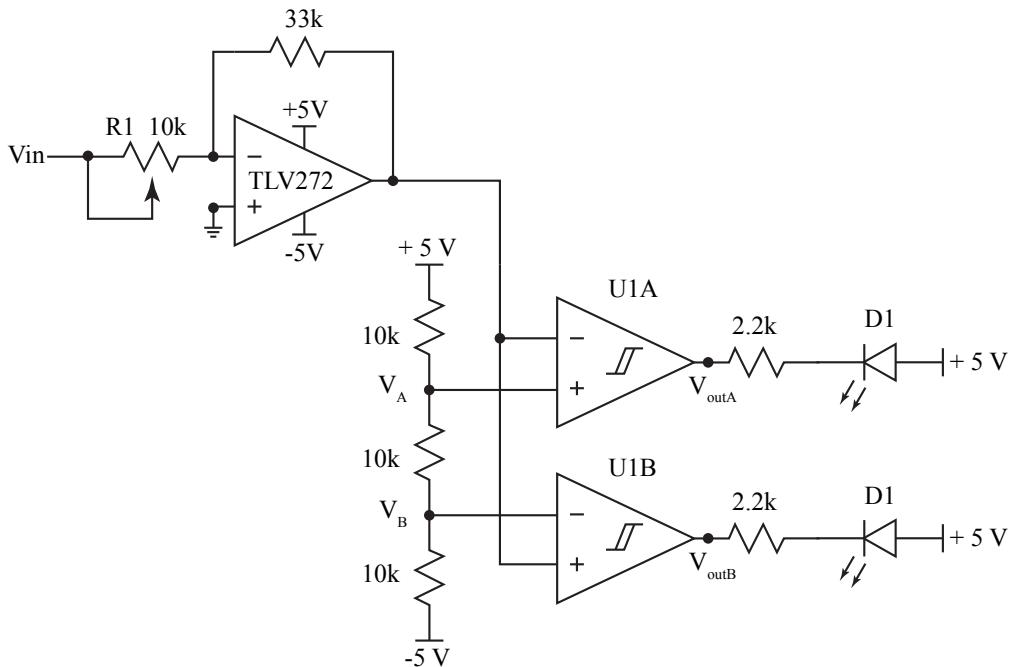


Figure 8.7: A variable gain amplifier, R1 is a potentiometer that can be varied, where the output is checked for clipping.

8.5.2 Breadboard Implementation

- Read the datasheet for the LM393, <http://www.ti.com/product/LM393/technicaldocuments>.
- Build the circuit in Figure 8.4 on a breadboard. It is important to note that the pinout for the comparator chip is the same as the op amp. It works differently, but should be connected the same as the op amp. This means you should still

connect pin 8 to +5V and pin 4 to -5V (not GND) just as you have done with the opamp. Set the input to a 0.5 Hz, 5 V amplitude sine wave. Use the two red LEDs in your lab kit, they're bagged separately. The anode for the LEDs is the longer of the two leads. The diodes will illuminate whenever the input voltage is above or below the reference voltage and should appear to blink in turn. Save the circuit to show your lab instructor at the start of lab.

8.6 In-Lab Requirements

The following must be completed at the start of lab.

1. Section 8.5.1
 - (a) Item 1: Half wave rectifier, image of the circuit and the plot of the input and output voltage.
 - (b) Item 2: Half wave rectifier with output smoothing, image of the circuit and the plot of the input and output voltage.
 - (c) Item 3: Clipping circuit, image of the circuit and the plot.
 - (d) Item 4: Variable gain amplifier with clip detector, image of the circuit and the plot.
2. Section 8.5.2
 - (a) Item 2: Operational clipping circuit.

8.6.1 Breadboard Implementation

1. Plot the input voltage and the voltage at the output of either comparator, Figure 8.4, on the o-scope and save an image of the display.
2. Build the circuit from Section 8.5.1 - Item 1. Plot the input and output on the o-scope and save an image of the display.
3. Build two version of the circuit from Subsection 8.5.1 - Item 2, one with where the capacitance is 0.1 uF and the other is 0.01 uF. Plot the outputs of both circuits on the o-scope and save an image of the display.
4. Build the circuit from Subsection 8.5.1 - Item 4. Set the input to a 0.5 Hz, 0.1 V amplitude sine wave. Vary the potentiometer, increase the gain, so that the LEDs illuminate when the op amp output is greater than the reference voltages. Plot the output of the gain amplifier and the output of either comparator on the o-scope and save an image of the display.

8.7 Write Up

This write up is different than previous write ups. Instead of comparing ideal and non-ideal values, discuss the operation and limitations of each circuit. For example, the first circuit is limited by forward voltage which limits how much of the signal can be rectified. Backup your discussion with images, all four, from the in-lab portion. Also, include diagrams for each circuit, typically this is not required but for this lab circuit diagrams are required. Do not copy the circuit diagrams from the lab manual, create your own, either digitally or by hand.

Chapter 9

Lab 9 - Filters

9.1 Objective

The objective of this lab is to introduce the concept of filters and explore their applications through examples of passive and active filters.

9.2 Materials

- Laptop with LTSpice
- Analog Discovery
- Breadboard
- Wiring kit
- Lab parts kit

9.3 Introduction

Laboratories so far have focused on the time domain. Given a signal with a specific frequency, there is a resulting output that can be viewed with an appropriate time base on the o-scope. Filters are analyzed primary through the frequency domain, where the output is a function of frequency and not time.

Filters, as the name implies, filters out certain frequencies from passing to the output. There are several types of filters, such lowpass, highpass, bandpass, notch (bandreject), and allpass. While there are applications for all the filter types, there is only enough time to focus on a few, in this case, lowpass and highpass. Additionally, the math associated with filters can become complicated quickly and so only the results will be represented here.

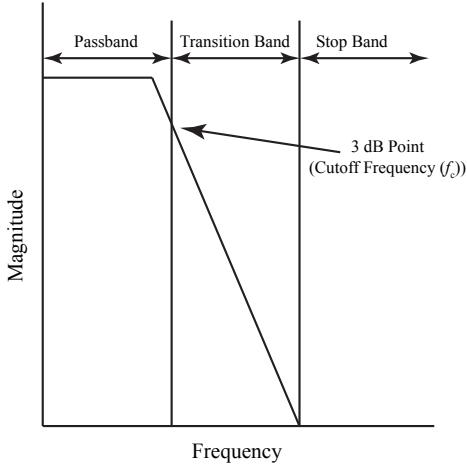


Figure 9.1: Example of response for a generic lowpass filter. The filter passes low frequencies frequencies, thus the name, and attenuates higher frequencies.

Figure 9.1 is an example of a generic lowpass filter response in the frequency domain. The filter, as the name implies, will only pass low frequency signals, higher frequency signals are attenuated through the transition and stop bands. A filter's 3 dB point, cut off frequency, is set by design and is often chosen to obtain a specific attenuation at a certain frequency. Similarly, a high pass filter will behave in the opposite fashion, low frequency signals will be attenuated while high frequency signals will be passed.

9.3.1 Passive First Order RC Filters

Figure 9.2 shows two simple RC filter configurations and their resulting frequency responses. These filters are considered to be passive, no active elements that require power, and of the first order, there's only one combination of resistors and capacitors.

While the configurations may seem puzzling at first, their results come easily when considering the effects at low and high frequencies. At low frequency, a capacitor acts as an open circuit, $\frac{1}{j\omega C}$ is large when $\omega = 2\pi f$ is small, while at high frequencies, it acts as a short circuit, $\frac{1}{j\omega C}$ is small when ω is large. So, for a lowpass, it makes sense that low frequencies are passed while high frequencies are attenuated. Similarly, for a highpass filter, low frequencies are attenuated while high frequencies are passed.

There are a variety of equations relating the transfer function of a filter, but instead the focus is on the important parameters. In the case of a simple RC filter, the 3 dB point, or the frequency where the output has dropped to half of the input, is governed by the following equation:

$$f_c = \frac{1}{2\pi RC} \quad (9.1)$$

For the examples in Figure 9.2 (a) and (c), the 3 dB point is calculated to be

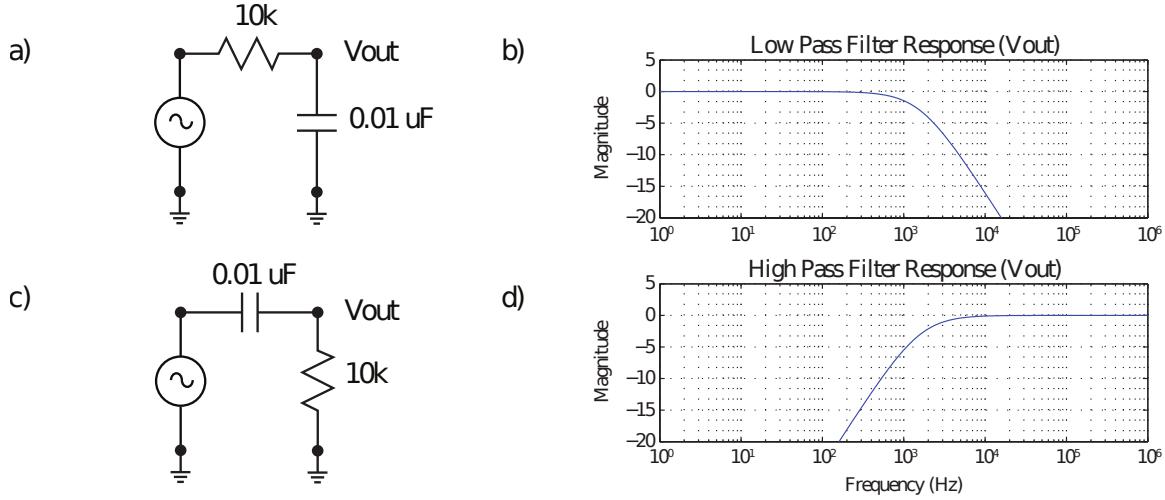


Figure 9.2: An example of a simple RC low pass filter (a) and the resulting frequency response (b). Also, a simple RC highpass filter (c) and the resulting frequency response (d).

$$\frac{1}{2\pi 10k \cdot 0.001\mu} = 15.916 \text{ kHz. Which matches the frequency response for both configurations.}$$

9.3.2 Active RC Filters

There are a variety of active filter topologies but a simple implementation is to use an appropriately placed capacitor in conjunction with a typical inverting amplifier configuration, Figure 9.3 (a) and (b).

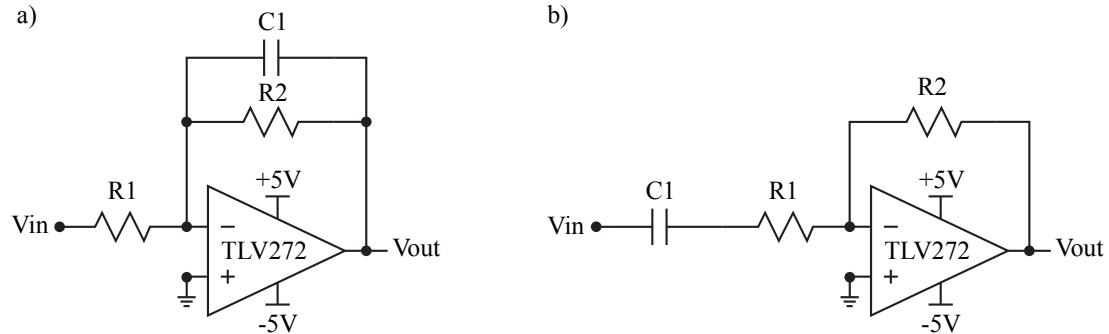


Figure 9.3: An example of an active RC low pass filter (a) and an active RC high pass filter (b).

The design of either the active RC low pass or high pass is also fairly simple for the first order case. A low pass filter has the following equations

$$\frac{V_{OUT}}{V_{IN}} = -\frac{R_2}{R_1} \frac{1}{j\omega C_1 R_2 + 1}$$

$$H_0 = -\frac{R_2}{R_1}$$

$$f_0 = \frac{1}{2\pi C_1 R_2}$$

where H_0 is the passband gain and f_0 is the 3dB frequency. Note that unlike the passive RC case, filters using op amps can also have gain. Similarly, the high pass filter has the following equations

$$\frac{V_{OUT}}{V_{IN}} = -\frac{R_2}{R_1} \frac{j\omega C_1 R_1}{j\omega C_1 R_1 + 1}$$

$$H_0 = -\frac{R_2}{R_1}$$

$$f_0 = \frac{1}{2\pi C_1 R_1}$$

9.4 Big Picture

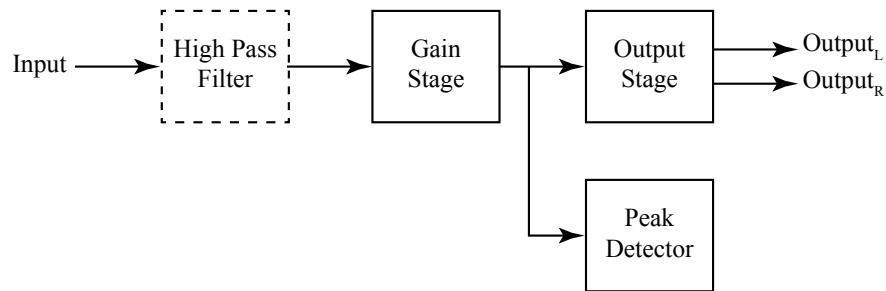


Figure 9.4: Big picture with emphasis on the high pass filter.

This lab focuses on filter circuits which will help to create the high pass filter used in the final project used to remove the large DC offset in the source signal.

9.5 Pre-Lab Requirements

Complete the following before coming to lab.

9.5.1 LTspice Simulations

Complete the following using the TLV271 spice model, always power the op amp with ± 5 V for V_{dd+} and V_{dd-} .

1. Review the LTspice tutorial for AC Analysis, <https://www.youtube.com/watch?v=fziUQaVQxA4&t=1s>.
2. Build a simple lowpass filter, Figure 9.2 (a), but set $R = 10 \text{ k}\Omega$ and $C = 0.001 \mu\text{F}$. Set the voltage source to an AC amplitude of 1 and run an AC analysis with the following settings: Decade, 100, 1, 1Meg. Save an image of the circuit, a plot of the output, and table the 3 dB frequency for submission to Canvas.
3. Build a simple highpass filter, Figure 9.2 (c), but set $R = 15 \text{ k}\Omega$ and $C = 0.01 \mu\text{F}$. Set the voltage source to an AC amplitude of 1 and run an AC analysis with the following settings: Decade, 100, 1, 1Meg. Save an image of the circuit, a plot of the output, and table the 3 dB frequency for submission to Canvas.
4. Build an active RC lowpass filter, Figure 9.3 (a). Set $C_1 = 0.1\mu\text{F}$ and determine the resistor values for a gain of -1 V/V (0 dB) and a $f_0 = 1.59 \text{ kHz}$ using only values in your lab kit. Set the voltage source to an AC amplitude of 1 and run an AC analysis with the following settings: Decade, 100, 1, 1Meg. Save an image of the circuit, a plot of the output, and table the 3 dB frequency for submission to Canvas.
5. Build an active RC highpass filter, Figure 9.3 (b). Set $C_1 = 0.1\mu\text{F}$ and determine the resistor values for a gain of -10 V/V (20 dB) and a $f_0 = 482.3 \text{ Hz}$ using only values in your lab kit. Set the voltage source to an AC amplitude of 1 and run an AC analysis with the following settings: Decade, 100, 1, 1Meg. Table the 3 dB frequency. Save an image of the circuit, a plot of the output, and table the 3 dB frequency (3 dB from the passband) for submission to Canvas.

9.5.2 Breadboard implementation

1. Review the Digilent tutorial for the Network Analyzer tool: https://www.youtube.com/watch?v=31tq_A_2TcY.
2. Build the active RC lowpass filter from Section 9.5.1 - Item 4 on a breadboard. Use a TLV272 op amp and power it with ± 5 V rails and wire up the Analog Discovery to use the network analyzer tool.
3. Set the start frequency to 100 Hz, stop frequency to 100 kHz, and the samples to 100. Connect CH1+ and W1 to the input and CH2+ to the output. Uncheck the box next to channel 1 and leave the remaining settings in their default configuration. Click run once to generate the filter's frequency response. Save the circuit to show your lab instructor at the start of lab.

9.6 In-Lab Requirements

The following must be completed at the start of lab.

1. Section 9.5.1
 - (a) Table of 3 dB frequencies (four values).
 - (b) Item 2: Simple RC lowpass filter - image of the circuit and a plot of the output.
 - (c) Item 3: Simple RC highpass filter - image of the circuit and a plot of the output.
 - (d) Item 4: Active RC lowpass filter - image of the circuit and a plot of the output.
 - (e) Item 5: Active RC highpass filter - image of the circuit and a plot of the output.
2. Section 9.5.2
 - (a) Item 3: Active RC lowpass filter - image of the circuit and a plot of the output.

9.6.1 Practical Implementations

For all active circuits, use a TLV272 op amp and power it with ± 5 V rails.

1. Set the number of samples to 1000 and re-run the network analyzer. Save an image of the output, and measure the 3 dB frequency.
2. Build the circuit from Section 9.5.1 - Item 2, run the network analyzer, measure the 3 dB frequency, and save an image of the output.
3. Build the circuit from Section 9.5.1 - Item 3, run the network analyzer, measure the 3 dB frequency, and save an image of the output.
4. Build the circuit from Section 9.5.1 - Item 5, run the network analyzer (set the amplitude to 100 mV from 1 V and the top to 30 dB for this case), measure the 3 dB frequency, and save an image of the output.

9.7 Write Up

Include the following in the write up.

- Table of experimental 3 dB frequencies and their associated percent errors when compared to the pre-lab.

- Network analyzer images for each circuit.

Discuss the operation of each filter and the difference between the theoretical and experimental 3 dB frequencies and touch on the following topics.

- Besides component variation, what other effects continue to shifting the 3 dB frequency from its ideal value?
- What are the advantages and disadvantages for each filter type, passive and active?

Chapter 10

Lab 10 - Final Project

10.1 Objective

The objective of the final project is to bring together concepts taught in previous labs in to a single circuit.

10.2 Materials

- Laptop with LTSpice
- Analog Discovery
- Breadboard
- Wiring kit
- Lab parts kit with TLV272 and LM393P
- Speakers with 3.5mm connector

10.3 Introduction

The nature of the lab will be different to accommodate the final project. With a total score that accounts for 20% of the entire lab grade, the final project is required to pass the course. There is no quiz for the final project and the breakdown is as follows:

- 20% Pre-lab
- 40% In-lab Demonstration (This includes building your final circuit on a provided printed circuit board)
- 40% Write up

The pre-lab simply requires a working spice schematic with the correct outputs, this is much simpler than it sounds. Source files for the spice simulation are available on Canvas under the Labs folder, Jingle4.wav and Jingle19.wav. The wav files can be imported in to LTspice using the following instructions, <http://www.linear.com/solutions/6087>. The final simulation, the simulation that's submitted to Canvas, must use the Jingle19.wav file and a transient simulation that's 19 seconds long (.tran 19). Because of the total time to complete the simulation, several minutes, a shorted file has been provided, Jingle4.wav, which runs for four seconds.

Failure to complete the pre-lab will result in a zero for 20% of the final project and being barred from the first lab sessions. A student will not however receive a zero for the entire lap project.

Lab sessions for the final project will span two weeks and exist purely for the student to demo their working breadboard circuit using the in-lab speakers and connector. Because there isn't enough connectors or speakers for students to take them home, both items must remain in the lab. The deadline to have a circuit checked off is approximately the end of the second lab session. Students are not allowed to demo their circuit during lab sessions that are not their own or during office hours.

The write up for the project is also different, see the template in the files section, and the due time for the final report will be announced in Canvas.

10.3.1 Audio Amplifier

The final project is a basic audio amplifier with a block diagram shown below in Figure 10.1. A simple jingle with a small amplitude, mV, and a large DC offset, 0.5 V, will serve as the input. The system functions as follows.

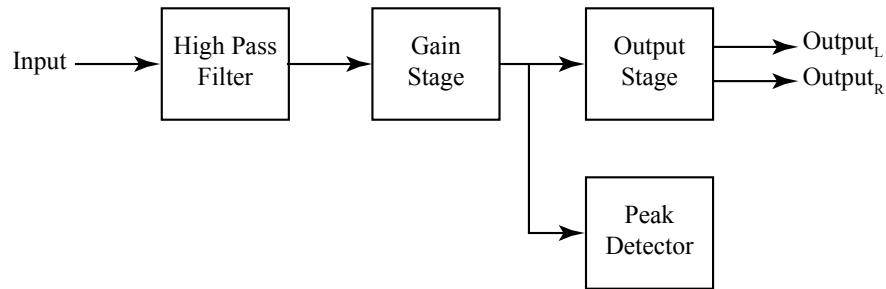


Figure 10.1: Audio amplifier block diagram.

- The high pass filter removes the DC offset from the jingle, effectively AC coupling the signal in to the system.
- A gain stage is used to amplify the jingle to a range of volts from mV so that it can be heard from the speakers.

State	LED 1	LED 2
Input < Threshold 1 and 2	Off	Off
Threshold 1 < Input < Threshold 2	On	Off
Threshold 1 and 2 < Input	On	On

Table 10.1: Different states for the peak detector

- An output stage must buffer the output to the speakers, the outputs do not need to be offset in phase.
- A peak detector is also required in order to indicate when the output is in the right range, defined by two thresholds. When the output passes the first threshold (LED 1 turns on), the output is operating in the normal linear range with an output on the order of Volts, and the second threshold (LED 2 turns on), where the output is about to start clipping. See the different LED states in Table 10.1. Note that this is a slight variation on the circuit used in the diode lab.

10.3.2 Recommendations

A few words on some of the individual sections.

- As always, you're limited to the components in your lab kit, not the Digilent student kit, choose your values wisely.
- Choose the 3dB frequency of the high pass filter carefully. The audio band is 20 Hz to 20 kHz and a 3 dB frequency in the kHz will attenuate (weaken/shrink) important parts of the jingle. Good rule of thumb is if you are over 1 KHz then you are too high. However don't choose it too low, aim for higher than 50 Hz.
- The required gain is high, and amplification should occur in two stages. One stage will have a fixed gain. This will be your active high pass filter, both accomplishing the needed filtering and providing gain. The next stage will be another amplifier with variable gain (using a potentiometer as the input resistor). This will allow for a variable voltage on the output signal.
- The output stage should be trivial.
- Choose the voltage thresholds for the peak detector so that at a reasonable gain, one LED is always on and the other only turns on when the output starts, or is about to start, to clip or distort.
- Use of the active high pass filter for your first stage eliminates the loading effect.

- The circuit should pass a demo using sine wave (10mV amplitude, 1KHz, 0.5V DC offset) as input and the peak detection circuit's Threshold 1 and Threshold 2 voltages should be 1V and 4V. Both LED should be off when the input is <1V, one LED on when the input is between 1V and 4V for OPAMP's linear range operation, then both LEDs should be on when the input is >4V to indicate OPAMP's output is getting into saturation range. The comparator's input configuration is different than Lab 8's, don't copy it directly.
- Output stage should have two buffers (for channels L and R).
- After you finish the demo and verification of your final project circuits, you need to continue to perform more measurement using scope, spectrum analyzer and network analyzer functions. Follow the file "Final Project Write Up Template" to complete your final report.
- Due time for the final report will be announced in Canvas.

And some more general advice.

- Build everything in LTspice before doing any work on a breadboard.
- The spice simulation of the 19 second jingle file will take a long time, several minutes. The 4 second jingle will also take a few minutes. Start instead with a 10 mV amplitude 1k Hz sine wave with a 0.5 V DC offset and a transient simulation of 1m (.tran 1m). Once you have that working, start using the jingle files.
- Use virtual net nodes instead of making various +5 V and -5 V supplies.
- All of the active components are powered by +5 V and -5 V supplies.
- When it's time to build the circuit on a breadboard, build the circuit in a linear fashion. The input should start at one end with the output at the other, don't make it a maze.
- Use color coded wires to make debugging easier and avoid creating extra nodes whenever possible. The more nodes in the circuit, the less likely it's going to work.
- Build stages one at a time to confirm that they're working as opposed to building everything at once and then trying to debug everything at once.
- Complete the project in the first week, there's a test the second week and it's better to simply get the project out of the way. During the second week you will also need to make time to solder the parts from the project on to a provided PCB for the final demo.
- Once you've demoed your circuit, don't take it apart, you'll still need it for items in the write up. It is recommended to complete all measurements required by the write up before soldering to the PCB.

10.4 Pre-Lab Requirements

This following must be completed and submitted to Canvas by the start of lab.

1. Image of the spice schematic and a plot of the input and all subsequent outputs (outputs of each individual stage) for a transient simulation for Jingle19 submitted to Canvas. Include the plot of the output and a circuit schematic, nothing else. The schematic and plot should be appropriately labeled. Images that are unclear or vague will receive little to no points. Your plot must include the following in **different plot planes**: input voltage, output of the high pass filter, output of the variable gain amplifier, output of the buffer stage (at least one), and the current of both LEDs. The gain doesn't need to be high enough to induce clipping in order to show that both LEDs turn on but the first LED should turn on.

10.5 In-Lab Requirements

Unlike previous labs, the final project lab spans two weeks (two lab sessions). The final project in-lab portion is only complete once a student has demoed a working circuit. There are two steps to demonstration that a circuit is working:

1. Using an input of a 10 mV amplitude sine wave with a 0.5 V DC offset, demonstrate that that the output can be varied and the peak detectors functions properly.
2. Using the 19 second jingle file as an input, demonstrate that the output can be heard on both speakers and the peak detector functions properly.

A student can demonstrate their working circuit during either of the two lab sessions, but only in their lab session, not during another lab session or office hours. There are a limited number of connectors and speakers, they will only be handed out to a student once they have demonstrated the first step of the required demonstration.

10.5.1 Wavegen Settings and Speakers

In order to play the sound file, Jingle19.wav, it must be imported in to the Wavegen's player. Instead of simple, choose play from the pulldown. Click import and then select the Jingle19.wav file. You'll be given a list of options and a plot of the signal, leave the settings to their default and select ok. Hit run to start the jingle playing.

The 3.5mm connector has three right angle pins soldered to it which allow it to be used with a breadboard. The center pin is ground and the other two pins connect to a speaker. Because the outputs are simply buffered, there's no need for a distinction between left and right.

10.6 Write Up

The write up is also different for the lab project, it takes the form of a formal report. See the template for details. The write up due time will be posted to Canvas, and is typically due around the end of the semester.