

Chapter 1

Introduction to Laboratory Tools

1.1 Overview

The purpose of this lab is to learn some of the fundamentals about your oscilloscope that can be used throughout your engineering career. Here is a list of the different tasks to be completed during the lab.

- View a single waveform and take measurements using an Oscilloscope.
- View multiple waveforms and take measurements.
- Explore the concept of triggering.
- Use the oscilloscope's measurement functions.

1.2 Pre-lab

In future labs, you will be required to complete a pre-lab for each lab. This pre-lab will cover concepts that will make the lab much easier to complete. The pre-labs are graded and are due at the beginning of your lab section each week. If you do not have your pre-lab finished at the beginning of the lab period, you will receive no credit for the pre-lab. Be sure to make two copies of the pre-lab, one for your TA, and the other for yourself.

1.3 Preparation

A lab notebook is suggested so you can keep all your information together in one place. This information will be useful for writing the lab reports and is an industry practice often required at various companies.

Suggestions of what to bring to lab each week:

- Your ENGR 202 kit of parts. (Handed out in first lab)
- Mini Grabber to banana plug. (See Figure 1.1(a), Purchased at Bookstore).
- BNC cables with Mini Grabber. (See Figure 1.1(b), Purchased at Bookstore).
- Oscilloscope Probe (See Figure 1.1(c), Handed out in first lab).
- Your lab notebook.



((a)) Mini Grabber to Banana Plug



((b)) BNC to Mini Grabber



((c)) Oscilloscope Probe

Figure 1.1: Tools for Lab

1.4 Procedure

1.4.1 Formation of Teams

Each team will have two group members. If the lab has an odd number of people, there will be one group of three for that lab. You will be spending your entire term in these groups, so be sure that you trade information about how to contact each other. As a member of a team, everyone needs to contribute towards the teams success. Please review the questions below and decide how you will divide your responsibilities.

- How will you divide the task of writing the final turned in report?
- How will you divide the task of conducting the lab?
- How will you decide when something is "finished well enough" to turn in?
- How will you handle problems when they show up?

1.5 Using the Oscilloscope

This objective outlines the following: how to set-up the oscilloscope (oscope), information about the MEASURE function of the oscilloscope, and finally, how to trigger the oscilloscope.

Oscilloscope Basics

Although the oscilloscope may appear to be overwhelming because of its many buttons and knobs, there are only a few things needing to be understood in order to successfully view a waveform. An oscilloscope displays voltage signals versus time. The time is the X-axis and the voltage is the Y axis. The oscilloscope can be adjusted to different ranges of each axis as well as the 'zoom' on each. This is very similar to what might be done on a graphing calculator in a mathematics course. This can be seen in Figure 1.2.

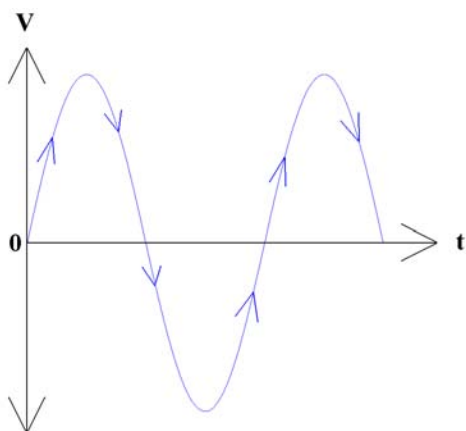


Figure 1.2: Voltage vs. Time Graph for Oscilloscope

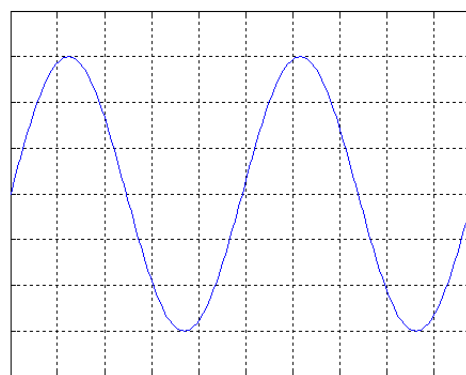


Figure 1.3: Example Waveform as Seen on oscilloscope

To adjust the 'zoom' on the oscilloscope so the signals displayed can be better observed, the VOLTS/DIV and SEC/DIV knobs are used. Some scopes will have multiple VOLTS/DIV knobs. These oscilloscopes have multiple channels that can be used to take measurements. When each channel is turned on, it will be displayed on the screen. This is useful when comparing different voltage signals to each other. Each knob will adjust each channel independently. As the names suggest they determine the number of volts and seconds that will be displayed per division, i.e. it adjusts the

CHAPTER 1. INTRODUCTION TO LABORATORY TOOLS

scale of the Y axis. Even if an oscilloscope has multiple voltage channels, the time axis will always be the same for every channel, hence there is only one knob for SEC/DIV.

In figure 1.3 a series of grid lines in both the horizontal and vertical directions can be seen. Each one of the grid lines makes a 'division' on the screen. For example this waveform has 6 vertical 'divisions' from its top peak to its bottom peak. Comparing either the two of the lower peaks or two of the upper peaks, it can be seen that there are approximately 5 horizontal 'divisions' between them.

If the 'volts per division' (VOLTS/DIV) was set at 1 VOLTS/DIV then this waveform would be 6V peak-to-peak. If the 'seconds per division' (SEC/DIV) was set at $1\mu\text{s}$ on the scope then this waveform has a period of about $5\mu\text{s}$ or a frequency of 200 kHz.

1. If figure 1.3 has a VOLTS/DIV of .5V, what would the Peak to Peak voltage be?

_____ V (1)

2. If the SEC/DIV was set to $250\mu\text{s}$, what would the Frequency be? Assume the period is 5 divisions. A useful relationship is $T = 1/f$, where T is the period in seconds and f is the frequency in Hz.

_____ Hz (2)

Setting up the Oscilloscope

In order to set-up and use the oscilloscope, follow these steps:

1. Find the oscilloscope on your lab bench. Using your oscilloscope probe, connect the BNC end to 'Channel 1'. See Figure 1.4.
2. Connect the probe onto the PROBE COMP, next to CH 1 on the oscilloscope. The black grabber (Ground) should be connected to the lower of the two connections. The probe tip should be clipped to the upper connection. See Figure 1.5.

It is very important to note that all of the ground cables on all of the oscilloscope channels are connected together inside of the oscilloscope. Use caution to always connect all of the ground cables to the same circuit node.

3. Press the CH 1 MENU button. This changes the display so that options for adjusting the properties of Channel 1 are displayed. If you press the button again, Channel 1 will be removed from the display. Press one more time to have it reappear. The options for Channel 1 are displayed along the right hand side of the screen. These are adjusted by using the unmarked buttons next to each option. See Figure 1.6.
 - Set the coupling to AC. This centers your waveform on the reference line. Remember when the channel is AC coupled, it will not display DC voltages, since the DC component of a signal is not displayed.
 - Set the probe to $10x$. Most general-use oscilloscope probes are $10x$, while very sensitive probes maybe $100x$ or even $1000x$.
4. As previously mentioned, an oscilloscope measures voltage with respect to time. The X-axis is therefore the time axis and the Y-axis is the voltage axis. However, just like graph paper, the oscilloscope can be set to whatever scale you want. In the lower left corner of the display, CH1 X.XX V is displayed. This is the VOLTS/ DIV of Channel 1. Similarly, values for CH2 and the time axis are displayed if that channel is turned on.
5. To view the 'PROBE COMP' signal, the oscilloscope must be set to the correct scales. The PROBE COMP wave outputs a 5V peak-to-peak square wave. Set the CH1 voltage scale using the VOLTS/DIV knob for CH1 so that you can see the entire waveform.
6. The frequency of the PROBE COMP waveform is 1000 Hz. Using the SEC/DIV knob, adjust the time scale so that about five periods of the wave are displayed on the screen. Remember that the period is the inverse of the frequency.

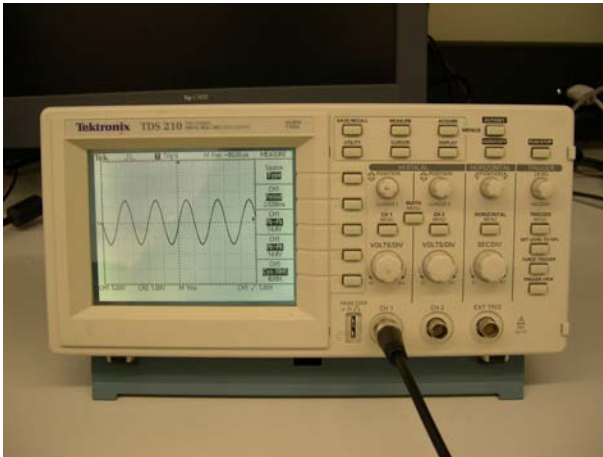


Figure 1.4: Tektronix TDS 210 Oscilloscope



Figure 1.5: Oscoprobe attached to PROBE COMP

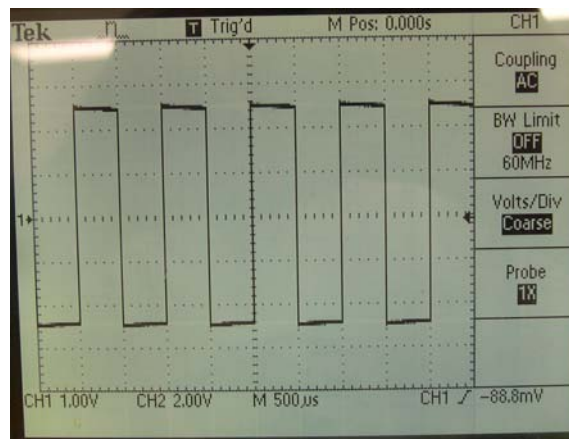


Figure 1.6: PROBE COMP waveform

1.5.1 Oscilloscope Built-in Tools

The CURSOR Function

The cursor can measure voltage and time differences. By using the buttons on the right hand side of the screen either voltage or time measurements can be selected. Once the type and source of the measurement have been selected use the position knobs to place the cursors where desired. Using the probe comp square wave as seen in figure 1.6 calculate the following using the cursor function. To make the task easier, press the 'Run/Stop' button to freeze the image on the screen first.

1. Find the V_{peak} of the Waveform _____ **V** (3)
2. Find the Period _____ **s** (4)
3. Find the Frequency _____ **Hz** (5)

Cursors can be used for a variety of functions. One that will be done in this course is to measure phase shift between two waveforms.

CHAPTER 1. INTRODUCTION TO LABORATORY TOOLS

The MEASURE Function

The oscilloscope can also display measurements of waveforms such as the period, frequency, or peak-to-peak voltage. Pressing the MEASURE button on the oscilloscope, will display the measurements on the right side of the screen. The button next to each box changes the channel as well as which type of measurement the box displays. For each measurement two things can be adjusted, Source and Measurement Type. One of the buttons will change whether Source or Type is highlighted. The other 4 buttons modify the selected item for each of the 4 measurements. Try displaying Frequency and Peak-to-Peak voltage for Channel 1.

The TRIGGER Function

The trigger level on the oscilloscope sets the voltage that triggers the oscilloscope. The trigger prevents horizontal drift of the trace. Find the TRIGGER LEVEL knob on the oscilloscope. Notice it moves an arrow on the right of the screen. Observe what happens if the arrow is moved outside the range of the waveform.

1.5.2 Using the Function Generation

Find the function generator on the top shelf of your lab bench. See figure 1.7 . To connect to the function generator output, a BNC cable will be required.



Figure 1.7: Tektronix CFG253 function generator

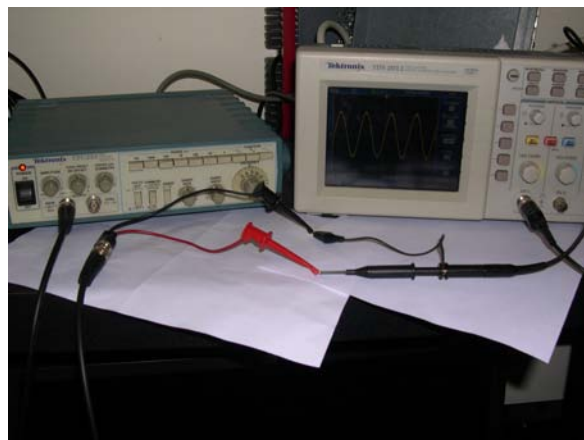


Figure 1.8: Connecting Oscilloscope to Function Generator

1. Connect the BNC cable into the function generator's MAIN output and then to the oscilloscope probe. The red mini-grabber should be connected to the tip of the probe while the black mini-grabber connects to the ground lead of the oscilloscope probe, as shown in figure 1.8
2. Using the oscilloscope and the function generator, the next step will be to create a sine wave with the same characteristics (V_{peak} and frequency) as the PROBE COMP wave. The only difference is that the new wave will be a 'sine' wave not a 'square' wave. During this lab only sine wave will be used, but it is good to understand that there are other wave shapes used in engineering.

Point of Interest: Sine waves are considered 'pure' wave and only have a frequency at a single point. All other repeating waveforms are composed of collections of sine waves at different frequencies.
3. Since the desired sine wave is known, the oscilloscope can be properly configured ahead of time to display it. Please note at this point the frequency generator should be turned off.
4. Using the CH1 VOLTS/DIV knob, set the voltage per division so that the wave will fit on the screen but not be too small. Then, using the SEC/DIV knob, set the seconds per division so that at least one period will fit on the screen, but not more than about ten periods.

5. Turn the function generator on, and push the sine wave button under FUNCTION.
6. The amplitude knob closest to the power switch on the function generator allows you to adjust the voltage output, and you adjust the frequency using the multiplier buttons and the fine adjustment knob. However, the oscilloscope must be used to measure the frequency and amplitude; ***the knobs on the function generator cannot be trusted to be accurate.***
7. Get your waveform as close as possible using the grid markers on the oscilloscope and the cursor function to find more precise numbers.
It may help to move the horizontal position of your wave so that the zero crossing of the wave goes through a grid line. Use the HORIZONTAL POSITION knob above the SEC/DIV knob to adjust the horizontal position.
8. Have your TA sign off that the waveforms are correct.

TA Signature: _____

1.6 Measuring Real Signals

Now that the oscilloscope basics have been covered, inspection of a real signal is next. Using the lab supplied audio cables and the .wav files on the lab webpage, generate and display a waveform of a stereo signal. The TA in each lab will assign two files for each group. Each group must answer the following question for the two assigned waveforms only!

File name: _____ (6)

Right channel: Frequency: _____ (7)

Right channel: Amplitude: _____ (8)

Left channel: Frequency: _____ (9)

Left channel: Amplitude: _____ (10)

Time Shift of Right Channel compared to Left Channel: _____ (11)

File name: _____ (12)

Right channel: Frequency: _____ (13)

Right channel: Amplitude: _____ (14)

Left channel: Frequency: _____ (15)

Left channel: Amplitude: _____ (16)

Time Shift of Right Channel compared to Left Channel: _____ (17)

1.7 LTspice Basic Tutorial

1.7.1 Getting Started

LTspice will be a program used at OSU for simulating and designing analog circuits. It is highly suggested that students learn how to use LTspice. There are many online tutorials that can make a great addition to this basic introduction to LTspice.

CHAPTER 1. INTRODUCTION TO LABORATORY TOOLS

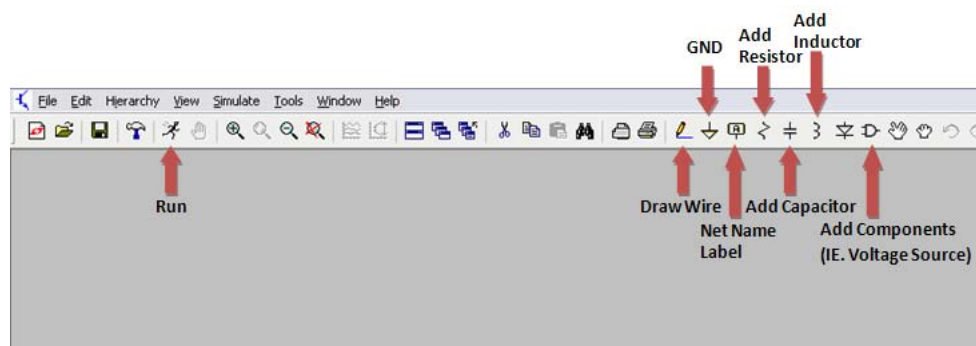


Figure 1.9: LTspice Top Menu

1. Open LTspice and start a new schematic.
2. Look at Figure 1.9 to get a basic idea of the different menu icons.
3. Start by adding *TWO* resistor's to the schematic by clicking the 'add resistor' button in the top menu. The placement of the resistor does not matter. The circuit to be designed is a voltage divider. Right-clicking the mouse will return the cursor to a normal cursor.
4. Then add a voltage source by clicking on the 'add component' button in the top menu. Search through the list of parts and select the one labeled 'voltage'.
5. To set the voltage and resistance of the components right click on the element and enter the value. One of the resistor's should be 10Ω and the other 25Ω . The voltage source should be set to 5V.
6. Once all the component values have been set they can be wired together. Use the 'draw wire' to connect them. See the schematic Figure 1.10 .
7. It is very useful to give certain important nodes (often called Nets) distinct meaningful names. Name the net between the two resistors.
8. After connecting the components **MAKE SURE TO ADD A GROUND REFERENCE**. Use the 'GND' button in the top menu. Where you place the ground in the circuit will dramatically effect the outputs. Remember that 'ground' is only a concept as it acts as a reference point in the circuit to compare other things to. It is recommended that ground is connected to the negative terminal of the Voltage source.
9. Before simulating the circuit selecting the correct 'type' of simulation is important. Based on the type of information the user would like about the circuit, LTspice can simulate in different ways. Some common simulations are DC operating point, AC Analysis, and Transient Analysis. A the DC operation point analysis will be performed first. The output of this simulation is a list of all the DC node voltages and currents in the circuit. Select 'simulate' from the tool bar. Then select 'edit simulation cmd'. See figure 2.10 Select the tab that says 'DC op pnt'. Click 'OK' and place the '.op' on the schematic.
10. The final step is to run the analysis. Select the the 'Run' button in the top menu. A file will then open with the node voltages and current through the elements.

Print out the results and turn them in with your study questions.

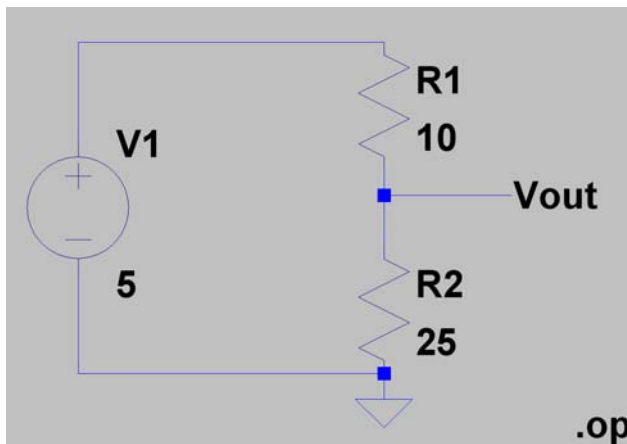


Figure 1.10: Voltage Divider Schematic

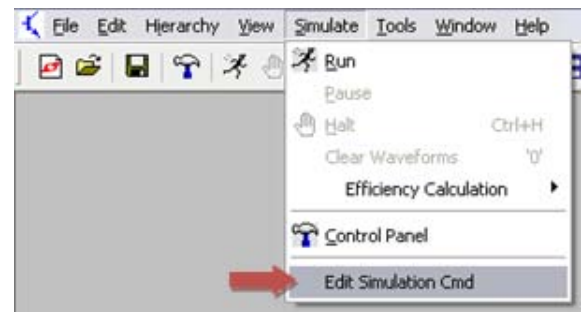


Figure 1.11: Editing Simulation Command

1.8 Study Questions

For this laboratory turn in a copy of the lab with values filled in as well as answers to the following questions. Each person should turn in their own work.

1. In your own words, describe DC and AC coupling mode on the Oscopce. Identify a reason to use each when inspecting a circuit.
2. In your own words, describe and draw the differences between a Magnitude voltage and a Peak-to-peak Voltage. Label your axes.
3. What is the frequency range of the function generators in the lab? What does the DC offset knob do?
4. Describe a method of determining time shift.

Chapter 2

Characterizing Components Using Lab Tools and LTSpice

2.1 Pre-Lab

The answers to the following questions are due at the beginning of the lab. If they are not done by the beginning of the lab, no points will be awarded. Bring two copies as one will be turned in to the teaching assistant.

1. In an inductive circuit, which leads, voltage or current?
2. Explain and fill in the table below with how V_{OUT} differs between the two circuits in figures 2.1 and 2.2 as frequency approaches DC and ∞ .

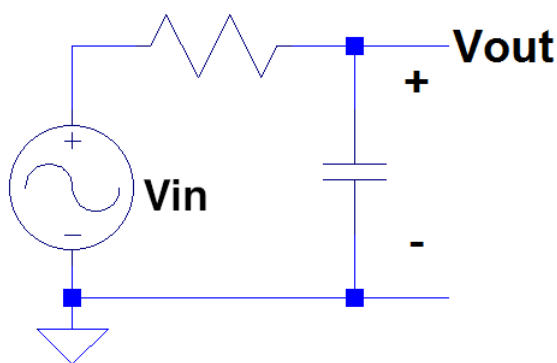


Figure 2.1: Pre-lab Circuit A

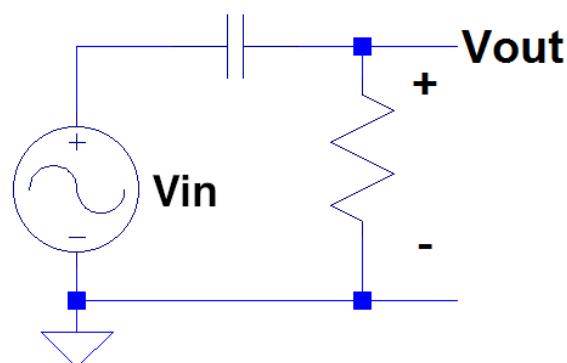


Figure 2.2: Pre-lab Circuit B

	DC	∞
Circuit A	V_{in} ?	
Circuit B		

2.2 Section Overview

In this lab, differences between simulated values and real values from circuits will be explored in addition to practice using the o-scope to measure current and phase angle.

2.3 Determining Current through a Simple Circuit

Begin by setting the function generator to generate a $5V_{PP}$ sine wave oscillating at 5kHz. Record what this source voltage is in phasor notation. Since this is the source for the circuit, a 0° phase angle can be assumed.

Source Voltage Amplitude: _____ V (1)

Recall from lecture the behavior of current and voltage in capacitive and inductive circuits. In an inductive circuit current lags voltage, as in Figure 2.3. In a capacitive circuit, voltage lags current, Figure 2.4.

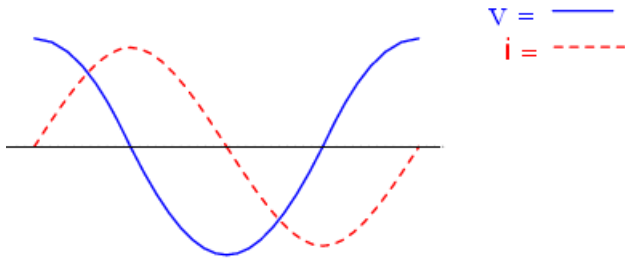


Figure 2.3: Inductive Circuit

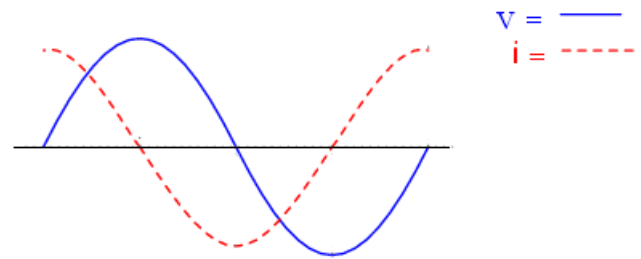


Figure 2.4: Capacitive Circuit

To demonstrate this, construct the circuit shown in Figure 2.5 using a $5V_{PP}$ sine wave from the function generator as V_S .

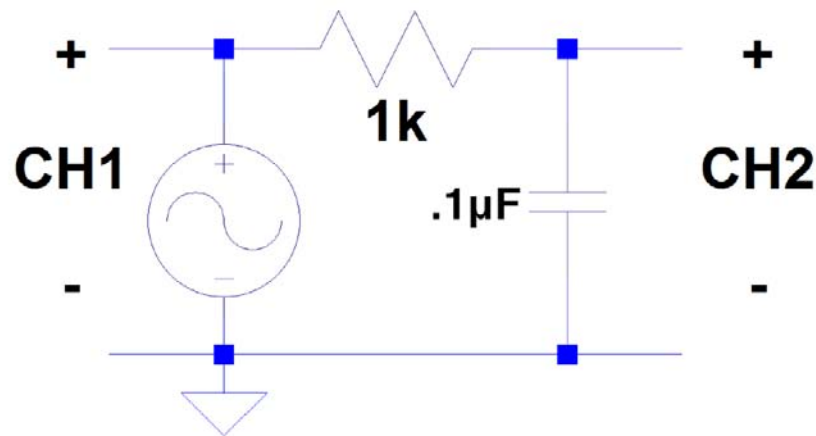


Figure 2.5: Schematic of Capacitive Circuit

To measure the voltage and current in the circuit using the o-scope both channel 1 and 2 need to be connected to the circuit. See figure 2.5 for how to connect the different channels.

1. CH1 of the oscilloscope will measure the voltage across the source (V_S).
2. CH2 of the oscilloscope to measure the voltage across the capacitor.
3. Connect the ground of the probe for CH2 to the side of the capacitor not connected to the resistor. This will be considered the ground reference from now on.
4. Ground should always be the same, so connect all the ground leads to the same node.

In order to solve for the phase angle, the two signals on the o-scope must be investigated. Use the oscilloscope to solve for the phase shift between the voltage and current and use this to determine the phase shift. To do this, follow these steps:

- (a) Zoom in until only one period is visible. This allows for the cursors to be accurately adjusted giving the best measurement.
- (b) Press the cursor button to make two cursors appear on the screen (one for each waveform).

- (c) Press the type button so that the cursors can measure small differences in time.
- (d) Move each cursor to adjacent 'zero crossings' of the two corresponding waves. This step is very important because you are trying to measure the difference in time between the two waves. In order to do this, the cursors must be on the same parts of their own waves.
- (e) Record the time difference between the two cursor locations from the o-scope(delta)._____ (2)
- (f) Find the period of the waveforms ($T = 1/f$)_____ (3)
- (g) Find the fraction of a period that the wave forms are separated by (delta/T)_____ (4)
- (h) Multiply this value by 360 to get the number of degrees that the two waveforms are different_____ (5)

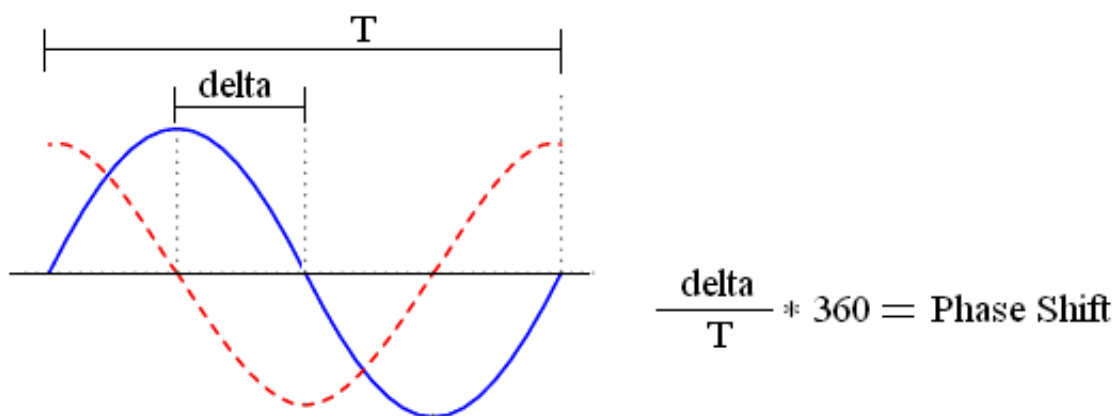


Figure 2.6: Calculating Phase Shift

- 5. Since the resistor is $1k\Omega$, according to Ohm's Law ($V = IR$), the current can be easily calculated.
NOTE: The voltage across the resistor needs to be calculated from V_S and V_{CAP}
- 6. View both of these waves on the oscilloscope and answer the following questions.
 - What is V_{CAP} in phasor notation _____ (6)
 - Calculate V_R in phasor notation _____ (7)
 - Using the **MATH** function on the oscilloscope measure and record V_R ._____ (8)
 - What is the difference between calculated and measured? _____ (9)
 - Calculate the current phasor. _____ (10)
NOTE: Use Ohm's Law.
 - Is the voltage lagging the current, or is the current lagging the voltage? (11)
 - Is this consistent with what you learned in lecture? (12)

2.4 Determining Impedance

Recall that the impedance Z of a circuit is a measure of the resistance and reactance for that particular circuit, measured in ohms. You can use impedance along with phasor voltages and currents to come up with an updated version of Ohm's Law. It is interesting to note that ohms law always applies to every circuit. If the circuit is being investigated strictly as a DC circuit, R is used. If the circuit is looked at in the AC domain, Z is used. The same rules even hold when analysis in the transient (S) domain occurs. The same equations work, just with a few different terms.

$$Z = \frac{V}{I}$$

Recall that impedance for capacitors and inductors is dependent on frequency according to the table below (where $\omega = 2\pi f$).

Element	Impedance(Ω)
R	$Z = R$
L	$Z = j\omega L$
C	$Z = 1/j\omega C$

Use the same process for the inductive circuit in figure 2.7. Choose a frequency that allows for a phase shift of at least 10 degrees.

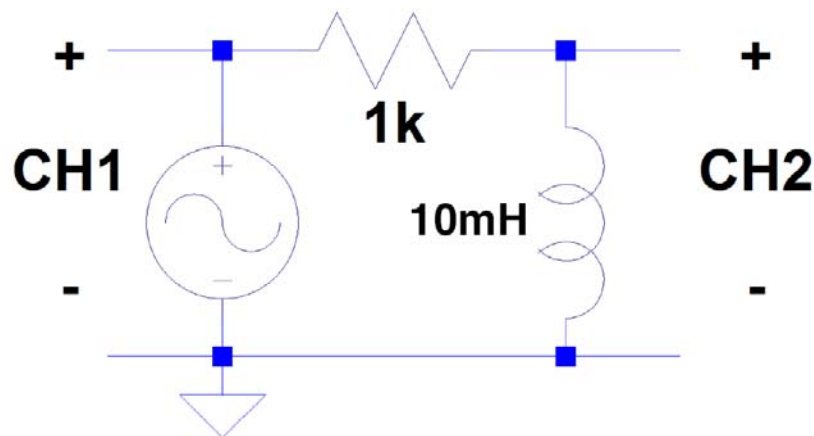


Figure 2.7: Schematic of Inductive Circuit

Connect the oscilloscope probes as explained in the last circuit. Take the needed measurements and calculate the requested values.

Frequency: _____ (13)

Magnitude of Voltage Across Circuit: _____ (14)

Magnitude of Voltage Across Resistor: _____ (15)

Now use Ohm's Law and the voltage across the resistor to calculate the magnitude of the current through the entire circuit. With the magnitudes of both the voltage and current, you can now calculate the magnitude of the impedance

CHAPTER 2. CHARACTERIZING COMPONENTS USING LAB TOOLS AND LTSPICE

using the modified version of Ohm's Law.

Magnitude of Current Through Circuit_____ (16)

Magnitude of the Impedance:_____ (17) Hint: $Z = \frac{V}{I}$

Now that you have both the magnitude and phase angle of the total impedance, you can break it up into the resistance (impedance due entirely to the resistor) and the reactance (impedance due entirely to the inductor). Draw the impedance of the circuit (Imaginary and Real components) in figure 2.8. Be sure to include labels on the axis and actual values.

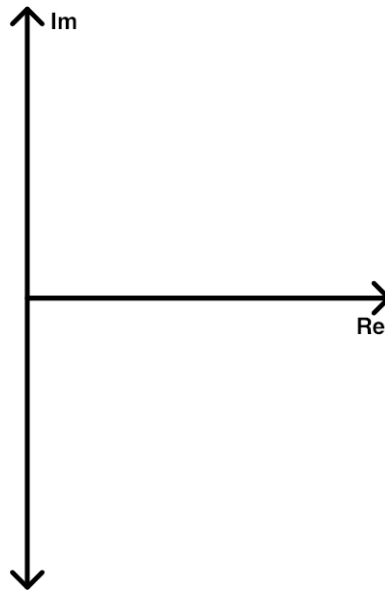


Figure 2.8: Magnitude of Impedance

Impedance of Inductor:_____ (18)

Do not use given value of inductor to calculate impedance. Instead, use the known resistance, magnitude of impedance, and trigonometry

Finally, you can calculate the actual inductance of the inductor, not just the labeled value, using the formula that relates impedance to inductance: $Z = j\omega L$ where $\omega = 2\pi f$.

Inductance of Inductor:_____ (19)

Is this consistent with the known value of the inductor? If not, why are they different? (20)

2.5 LTSpice Analysis

Now that you have measured some basic inductive and capacitive circuits for yourself, it's time to simulate one of them using LTSpice.

1. Begin by opening LTSpice and starting a new schematic.
2. Begin by adding an AC voltage source. To do this, press F2 on the keyboard and select "voltage" from the list of options. Click OK to place the source on the new schematic. Once the voltage source has been placed on the schematic, right click on it and then click "advanced" to bring up the options shown in Figure 2.9.
3. On this screen, choose the "SINE" option with an amplitude of $5V_{PP}$ and a frequency of 5000Hz. Finally click OK to finish.
4. Next, press the "R" key to bring up a resistor and place it on the diagram in the correct location. Again, right click on the resistor to set the resistance to 1K ohm.
5. Finally, add an inductor to complete the circuit. To add an inductor press "L". Place it and set the value of 10mH.
6. Once all of the components are placed, it's time to add the wires that connect them. To add wires press F3 and place them with the mouse. Right click when finished.
7. Before running the simulation designate a ground node, by pressing the "G" key and placing the symbol.

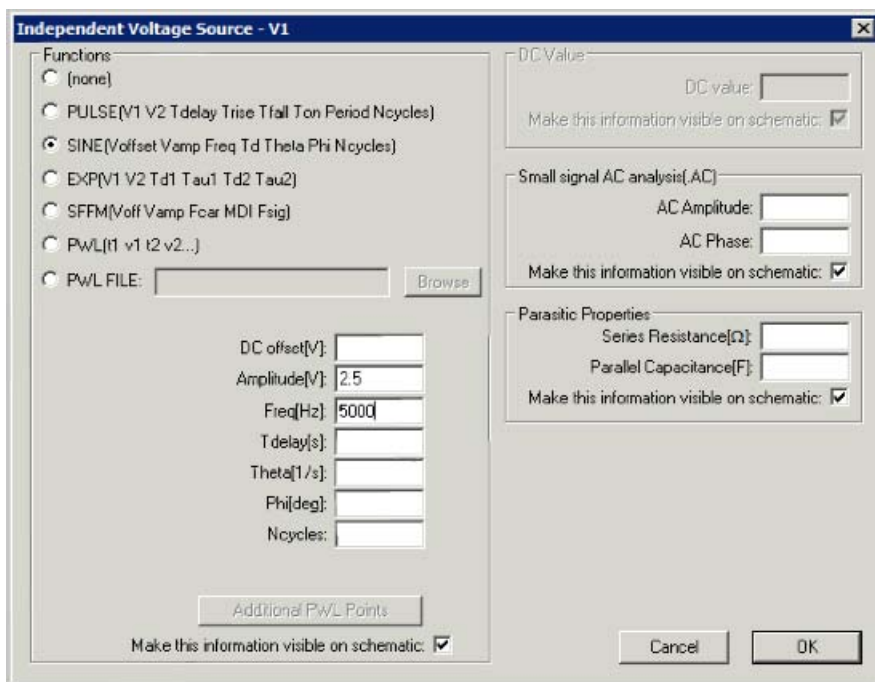


Figure 2.9: LTSpice Voltage Source Options

8. To run the simulation, go to Simulate and then Run to bring up the run settings window. Since the voltage source is alternating at such a high frequency only a very small window of time is needed to see the waveform. To begin, set the Start time at 1 and the Stop time at 1.01 and leave the Max Timestep blank. Select OK when ready.

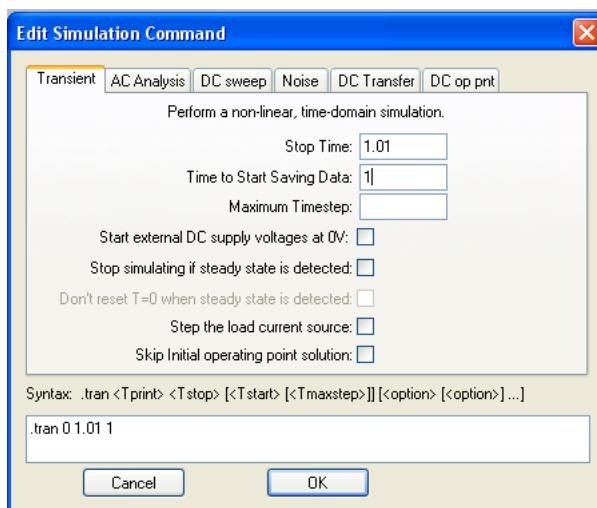


Figure 2.10: Edit Simulation Command Window

9. Now the simulation should have run, and a blank chart will be displayed. To see the voltage and current graphs use the mouse as a probe on the circuit. By moving the mouse over any component an ammeter probe will appear to measure current. Place the mouse over the resistor and select to view the waveform of the current going through it.
10. To view voltage at any node, move the mouse over the node and select when the voltmeter probe appears. Place the mouse over the node connected to the positive terminal of the voltage source and select.
11. There should be a couple of waveforms, one for voltage and one for current displayed. Are the waveforms you see consistent with what you saw on the oscilloscope earlier?
12. To measure phase difference use the same techniques used with the oscilloscope. Right select on the trace labels at the top of the graph and assign a cursor to each waveform.

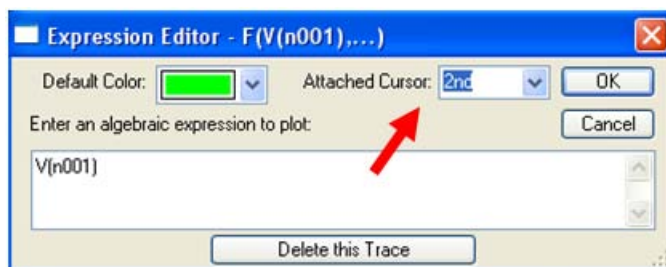


Figure 2.11: Assigning Cursors

13. Drag each cursor to adjacent peaks of the waveforms. To get the amplitude and time difference between the two waves, look at the window with the cursor information (pictured in Figure 2.12).

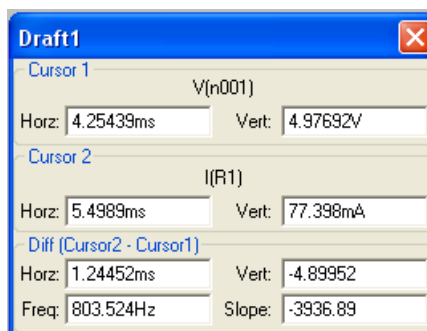


Figure 2.12: Reading Amplitude and Time Differences

14. The "Vert" sections show the value of the waveform at that point in time. With the cursors at the peaks of the waves, they will be showing the full amplitude of each wave.
- Amplitude of Voltage Waveform:** _____ (21)
- Amplitude of Current Waveform:** _____ (22)
15. Under the Diff(Cursor2 & Cursor1) section you can read the time difference between the two waves under the "Horz" section.
- Time Difference Between the Waves:** _____ (23)
16. Is this the same value that you got for your delta value from the oscilloscope? If not, why not? (24)
17. Print out simulation results and schematic and turn in with study questions. *Make sure simulation results are NOT printed with black background and the different signals can be differentiated*

2.6 Study Questions

1. If you had used a voltage source with 100kHz frequency instead of 1kHz, would your current waveform have been shifted more or less in the inductor circuit, figure 2.7? How about the capacitor circuit, figure 2.5?
2. What value capacitor would you have to add in series with the inductor in figure 2.7 to shift the current waveform until it is back in phase with the voltage and a frequency of 5 KHz?
3. Explain the process for solving for the inductor value in Section 2.4

Chapter 1

Simulation of Non-Ideal Components in LTSpice

1.1 Pre-Lab

The answers to the following questions are due at the beginning of the lab. If they are not done at the beginning of the lab, no points will be awarded. Bring two copies as one will be turned into the teaching assistant.

1. How do you expect the frequency generator output voltage to change with increasing load (decreasing impedance)?
2. Explain the meaning of component tolerance. (E.g. Tolerance of a resistor)
3. Is a real inductor completely reactive? Why?

1.2 Section Overview

The engineering method always begins with an initial design that is implemented and tested. Once the design is complete and before the circuit is ever implemented, the engineer will simulate its behavior using a tool such as LTSpice. LTSpice is a great tool for simulating circuits so that you can know how they will respond before you build them. However, simulating ideal circuit components is not the complete picture. Circuits in the real world are not ideal; they contain parasitic components. In this lab the non-ideal characteristics of components will be explored. LTSpice will be used to account for these slight differences between real and ideal parts.

1.3 Non-Ideal Resistors

Reality is messy. Most lecture material covers electronics from an ideal point of view. Only in lab when using real components does it matter that parts have tolerances. Even resistors have an accuracy tolerance that can vary their value by as much as 10%. To demonstrate this property, start by calculating V_{out} in the ideal circuit in figure 1.1 and enter it below.

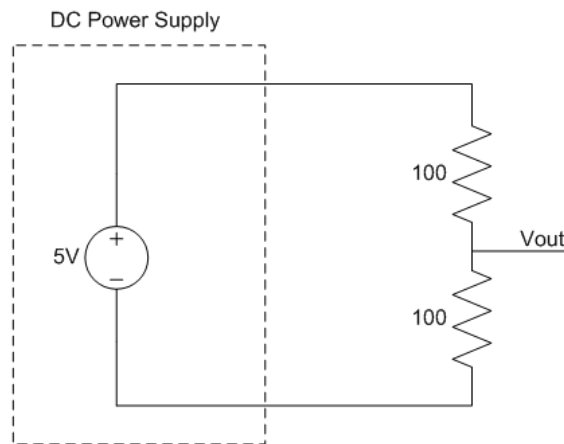


Figure 1.1: Voltage Divider

What is the expected (calculated) voltage at V_{out} ? _____ (1)

All resistors have an associated tolerance. The through-hole resistors included in your kits have colored bands to represent their value as well their tolerance. Look at the last color band on the resistor to determine the accuracy tolerance according to table 1.1.

4 th Band Color	Accuracy Tolerance
Brown	1%
Red	2%
Orange	3%
Yellow	4%
Gold	5%
Silver	10%

Table 1.1: Resistor Tolerance

This percentage is representative of the percent resistance that the resistor might be off from its stated value. For instance a $1\text{k}\Omega$ resistor with a tolerance of 3% could possibly be $(1000\Omega)(.03) = 30\Omega$ off from $1\text{k}\Omega$. As you can imagine this could change your expected results by a substantial amount.

Calculate the possible difference in resistance due to tolerance of the resistors used in the circuit:
 _____ (2)

Using an ohmmeter, measure the actual values of the resistors and record them on the circuit in figure 1.2. Now construct the circuit from figure 1.2. Measure the voltage being supplied by the power supply and record it on the circuit. Finally, measure the voltage at V_{out} and record it as well.

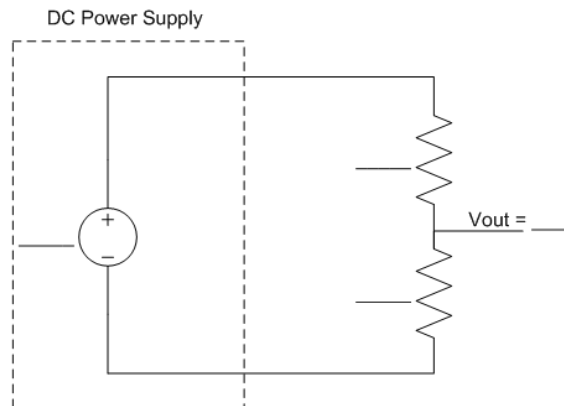


Figure 1.2: Fill in Voltage Divider

Compare the two circuits (ideal and non-ideal) and see how there are slight differences. In most cases these small differences don't have a big effect on a circuit, but in many cases (e.g. AC sources at very high frequencies) the non-ideal properties of components have to be considered.

1.4 Non-Ideal Voltage Source

An ideal voltage source is one with no internal resistance; where the current supplied is only dependent on the external circuit. As the resistance of this circuit approaches zero, the ideal voltage source is able to supply a current approaching infinity according to ohm's law $V = IR$. Of course, this cannot ever be the case since then the source would be supplying an infinite amount of power ($I * V$).

In reality, every voltage source has some internal impedance or resistance commonly referred to as output impedance. This impedance prevents the function generator from being able to break the laws of physics. Such non-ideal voltage sources can be modeled with some impedance in series as shown in figure 1.3. This becomes

CHAPTER 1. SIMULATION OF NON-IDEAL COMPONENTS IN LTSPICE

more important as an attached circuit draws an increasing amount of current. The voltage output of the function generator will drop as more and more current is drawn.

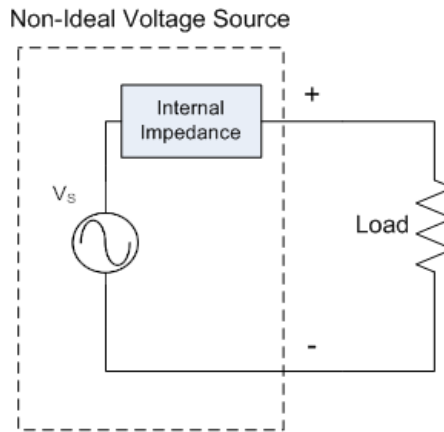


Figure 1.3: Non-Ideal Source

To accurately simulate a circuit, it is sometimes necessary to know the internal impedance of a source. In this lab calculating the internal resistance and reactance of the function generator is the next step.

1. Using the oscilloscope, ensure that the signal generator is producing a $9V_{pk-pk}$, 1kHz sine wave. The impedance of the oscilloscope probe is very large, so the voltage observed is at what is called 'no load.' It is important to realize that the voltage will change when the circuit is loaded if nothing else is done. Since there is essentially no current flowing, there will be no voltage drop across the internal impedance and therefore the value you measure will be equal to V_s .
2. Record your measured value of V_s _____ (3)
3. Now, using the breadboard, connect a $1k\Omega$ load resistor to the function generator and measure the magnitude of the voltage across this resistor. Do not adjust the function generator in any way.
4. Record the voltage across the $1k\Omega$ resistor using the oscilloscope. _____ (4)
5. Now replace the $1k\Omega$ resistor with a 220Ω load resistor.
6. Record the voltage across the 220Ω resistor. _____ (5)

Since the value of the load resistor and the voltage across it is known, the impedance can be solved for using a basic voltage divider equation.

$$|V_{load}| = \frac{R_{load}}{|R_{load} + Z_{out}|} |V_s|$$

Separating the internal impedance into resistance and reactance gives:

$$|V_{load}| = \frac{R_{load}}{|R_{load} + R_{out} + jX_{out}|} |V_s| = \frac{R_{load}}{\sqrt{(R_{load} + R_{out})^2 + X_{out}^2}} |V_s|$$

In this simplified equation there are two unknowns, R_{out} and X_{out} , corresponding to the resistive and reactive parts of the internal impedance. With two unknowns another equation is needed to form a system of equations that can be

solved simultaneously. The second equation can be found by using the measurements from the second resistor. Calling the 1kΩ resistor R1 and the 220Ω resistor R2 with the corresponding load voltages V1 and V2 the above equation can be adjusted to give both of the equations below.

$$\sqrt{(R_1 + R_{out})^2 + X_{out}^2} = R_1 \frac{|V_s|}{|V_1|}$$

$$\sqrt{(R_2 + R_{out})^2 + X_{out}^2} = R_2 \frac{|V_s|}{|V_2|}$$

Squaring both equations and then subtracting the second from the first, yields the following:

$$(R_1 + R_{out})^2 - (R_2 + R_{out})^2 = (R_1 \frac{|V_s|}{|V_1|})^2 - (R_2 \frac{|V_s|}{|V_2|})^2$$

1. This equation with only one unknown, R_{out} , can be solved. Using the measured values, solve for R_{out} and record it _____ (6)
2. Using the calculated value for R_{out} solve for X_{out} using either of the above equations and record it _____ (7)
Note: X_{out} and R_{out} can be positive or negative. Assume positive.
3. Now record the total internal impedance of the function generator in both rectangular and polar form.

Rectangular (Resistance and Reactance):

Polar:

_____ + j _____ Ω (8)

_____ + ∠ _____ ° (9)

1.5 Other Non-Ideal Components

Non-ideal aspects of inductors, capacitors, and resistors also come into play in circuit simulation and construction. Ideally, inductors and capacitors are made up entirely of reactance and do not dissipate any real power. In reality both components feature some parasitic qualities which include series resistances, capacitances and inductances for each element. These parasitics typically only become a problem at high frequencies currents.

One good example of this is the long windings of wire around the core of an inductor, which create a measurable resistance commonly referred to as equivalent series resistance (ESR). To see this, try using a digital multimeter to measure the resistance of a 10mH inductor.

Record the resistance here. _____ (10)

The measured resistance is relatively small, but can certainly have an effect depending on the frequency of a circuit. To simulate a circuit completely, a model of an inductor with an ESR would be needed. This could be done by using a resistor in series with the inductor as shown in figure 1.4. However most simulation tools allow for the addition of DC resistance in the component properties for inductors (and capacitors).



Figure 1.4: ESR Inductor

CHAPTER 1. SIMULATION OF NON-IDEAL COMPONENTS IN LTSPICE

Similar to inductors, capacitors also exhibit some ESR although it is much harder to measure. A couple common ESR values for capacitors are in table 1.2. Non-ideal capacitor's can also be modeled using a small resistor in series with the capacitor.

Capacitance	Equivalent Series Resistance
$.1\mu F$	$.08\Omega$
$10\mu F$	$.03\Omega$

Table 1.2: Capacitor Resistance Values

1.6 Modeling Non-Ideality in LTSpice

As this lab has shown, there are no completely ideal electrical components. Every component has some parasitic properties that prevent it from behaving exactly as expected. In some cases, it can be important to accurately model these properties before constructing the circuit. LTSpice has mechanisms for accounting for all of the non-idealities in all of the basic components. To show this, let us create a non-ideal version of one of the circuits from lab 2, shown in Figure 1.5.

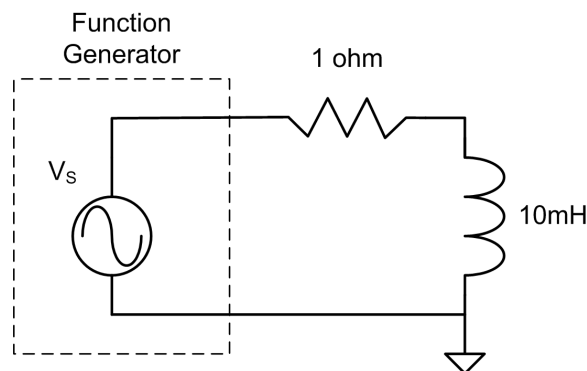


Figure 1.5: Non-Ideal Circuit

Begin by placing a $5V_{Peak-Peak}$ 1kHz voltage source, 1Ω resistor and 10mH inductor on a new LTSpice circuit and connect them to match the above circuit. This is the ideal version of the circuit. Make a copy of the circuit on the same page. This will be used for the non-ideal version of this circuit. Now add parasitics to the circuit by adding components to simulate their presence.

First, right click on the voltage source to bring up the advance settings. On the right part of the window there is a section for parasitic properties of the source. Under the section titled "Series Resistance", fill in the output resistance of the function generator that was solved for earlier.

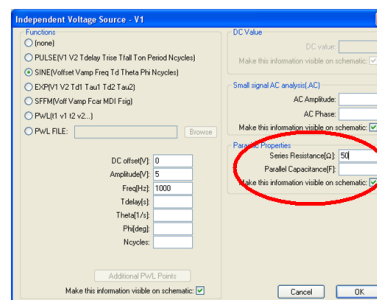


Figure 1.6: Using LTSpice for Non-Ideality

To be 100% accurate, the input reactance of the function generator should be identified as inductive or capacitive and converted to a parallel capacitance to be entered into LTSpice. For the purposes of this lab, the reactive portion of the input impedance will be ignored.

Next, right click on the non-ideal resistor to bring up the resistor settings. Under tolerance, enter 5 percent to model a real resistor.

Finally, right click on the non-ideal inductor and enter the value of ESR that was measured earlier.

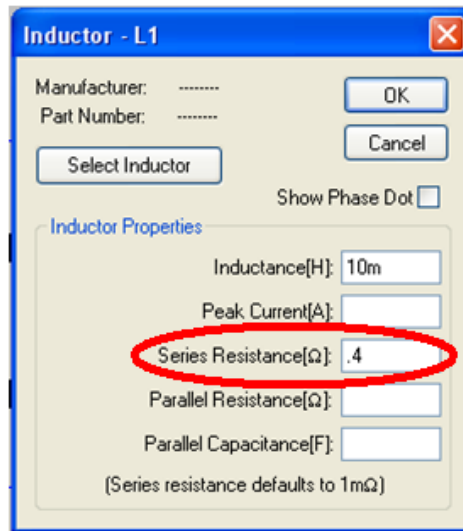


Figure 1.7: Series Resistance in LTSpice

Add a ground node to each circuit and then select Simulate → Run to bring up the simulation settings window. Enter 1.01 for the Stop Time and 1 for the Start Time and click OK to run the simulation.

Using the digital probes in LTSpice, view the waveforms of the current through the inductor on both the ideal and non-ideal circuit. There should be noticeable differences. Spend a few minutes looking at the differences between the two simulations in LTSpice. If the differences are not obvious, ask the course teaching assistant for help.

Print out Schematic and Simulation results to turn in with lab report

Make sure output graphs of each waveform is distinguishable with correct labels.

1.7 Study Questions

1. Calculate the equivalent series inductance of the function generator from the X_{out} determined earlier.
2. Calculate the peak instantaneous and RMS power dissipated through a 220Ω resistor supplied by a $9V_{peak-peak}$, 1Khz signal.
3. List a few examples of when it would be important to account for the non-ideal characteristics in your circuit design.

Chapter 1

Mystery Box Challenge

1.1 Pre-Lab

The answers to the following questions are due at the beginning of the lab. If they are not done at the beginning of the lab, no points will be awarded. Bring two copies as one will be turned into the teaching assistant.

1. Using the data sheets provided on the lab webpage, find the typical value for DC resistance or equivalent series resistance of the following components; Inductor, Ceramic Capacitor, and Electrolytic Capacitor.
2. List the impedance equations for an inductor and a capacitor. What is the simplified equation for a capacitor and an inductor in series? In parallel? Do these equations include the DC resistance or equivalent series resistance?

1.2 Section Overview

In this lab the electrical contents of several mystery boxes will be found. The possible components inside the mystery box could include a resistor, a capacitor, an inductor or a combination of these elements in series or parallel. First a simple mystery box (one component) will be researched. Then one additional mystery box will be explored. Finally an LTSpice simulation will be created to validate the contents.

IMPORTANT: *The boxes need to be handled with care and if taken home or tampered with, it will result in immediate failure of lab 4.*

1.3 Mystery Box 1

1.3.1 Overview

There is one component in mystery box 1. The boxes are numbered and it is important to record which box you are using. The value you find for the component must be within 5% of the actual value. The strategies in the following section will aid you with this level of accuracy. **Make sure your mystery box is labeled with a car manufacturer.**

1.3.2 Strategies for Solving Mystery Box 1

1. When analyzing reactive components, it is important to look at the components using a variety of frequencies. This is important since it is difficult to tell the difference between a resistor and a capacitor at only a single frequency. To perform the analysis make sure measurements are taken at DC and 5 AC frequencies between 100Hz and 100KHz. Try to space these frequencies throughout the domain. Record these frequencies as they will be necessary later.
2. When performing some of the analysis steps, an external resistor may need to be used. To ensure as much accuracy as possible, measure the resistance exactly of the resistor rather than just assuming it is the value indicated on it. Be aware that the value of the resistor also should be 'with in range' of the black box component. Several different resistors may need to be used based on the frequency used for analysis.
3. Using a multimeter will measure DC characteristics while using the oscilloscope and function generator will allow for measuring the AC characteristics at different frequencies. Comparing the differences between the DC measurements and the AC measurements will help to identify impedances due to real components versus reactive components.

1.3.3 Procedure for Mystery Box 1

1. Using what you have learned in previous labs, identify the component inside the mystery box, draw the schematic of the component include its value. For each frequency record the appropriate information about the waveform
2. Use LTSpice to draw the schematic for your test setup and simulate the results. Refer to lab 2 for help with simulation in LTSpice.
3. Compare the results with what is observed in real life with the 5 different input frequencies between 100Hz and 100Khz used earlier.

1.4 Mystery Box 2

1.4.1 Overview

Mystery Box 2 contains 2 elements. Any of the elements listed above could be inside. The task is to find out what the elements are, their values and simulate the results. Another task is to create a plan of action to quickly determine the elements. The layout of mystery box 2 is in figure 1.1. Use the layout along with prior knowledge obtained throughout this lab to find out what the components are and their values. **Make sure your mystery box is labeled with a band or singer.**

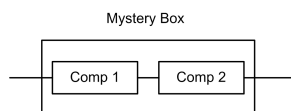


Figure 1.1: Mystery Box 2 Layout

1.4.2 Strategies for Mystery Box 2

1. Using the methods from mystery box 1, determine the contents of mystery box 2. The internal components will be in a series combination, meaning the impedance of the box must be something of the form shown in figure 1.2. To solve the box, it is a matter of figuring how much resistance and reactance each element has.

$$Z_T = Z_1 + Z_2$$

Figure 1.2: Series Impedance Equation

2. Remember to take measurements at DC and 5 AC frequencies between 100Hz and 100KHz. Try to space these frequencies throughout the domain. Record these frequencies as they will be necessary later.

1.4.3 Procedure for Mystery Box 2

1. Using what you have learned in previous labs, identify the component inside the mystery box, draw the schematic of the component include its value. For each frequency record the appropriate information about the waveform
2. Use LTSpice to draw the schematic for your test setup and simulate the results. Refer to lab 2 for help with simulation in LTSpice.
3. Compare the results with what is observed in real life with the 5 different input frequencies between 100Hz and 100Khz used earlier.

1.5 What to Turn In

Turn in a well written lab report including all schematics, calculations, and simulation results. A report template can be found on the webpage. In addition, your report must answer the following questions:

1. In lab, you investigated a two component mystery box. If these boxes had three or more components, could you have determined the value of each component? Explain your answer in detail.
2. For both boxes, comparisons with simulations were made at multiple frequencies. Why were all of these comparisons made? There will be slight variations between simulation and real measurements. Do these differences change depending on the frequency observed? If they do why? If they do not, why not?

Chapter 5

Basic Filters

5.1 Pre-Lab

The answers to the following questions are due at the beginning of the lab. If they are not done at the beginning of the lab, no points will be awarded. Bring two copies as one will be turned into the teaching assistant.

1. What is the final equation for V_{out} in figure 5.1 and sketch V_{out} vs. Frequency.
2. What is the final equation for V_{out} in figure 5.2 and sketch V_{out} vs. Frequency.

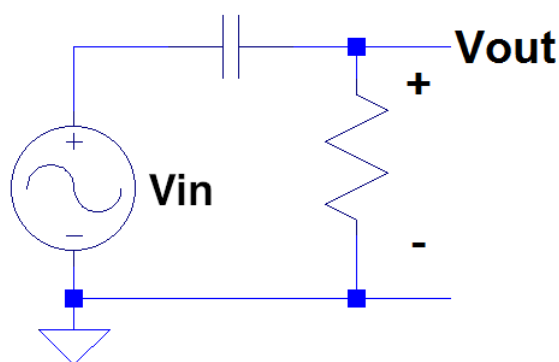


Figure 5.1: Pre-lab Circuit A

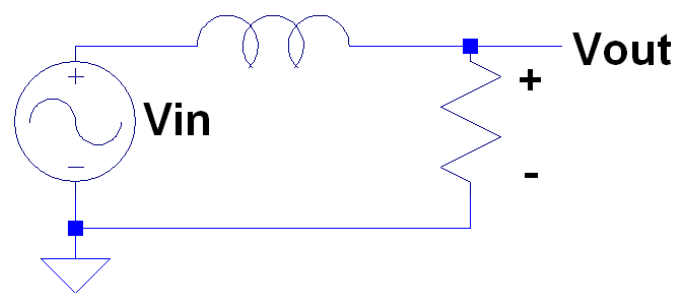


Figure 5.2: Pre-lab Circuit B

5.2 Section Overview

Each group will examine three filters and characterize their functions. A filter is a circuit designed to change its impedance over a range of frequencies. This intentional design allows for less energy in certain frequency ranges to be 'passed through' to the output. For this lab, you will build a low-pass, high-pass, and band-pass filter.

5.3 Three Filter Types

5.3.1 Low-Pass Filter

A low pass filter is a circuit that passes low frequency signals. As you can see from figure 5.3 the lower frequencies are passed while the upper frequencies are attenuated, or reduced. The point at which frequencies stop getting passed and start getting attenuated is known as the corner frequency. The corner frequency is the point where the output voltage is 70.7% of the input voltage ie. $(0.707 \times V_{IN})$. This point is sometimes called the 'cutoff frequency' or the '-3dB' point. Remember -3dB is in decibels and is the log scale equivalent to 70.7% of the input voltage.

5.3.2 High-Pass Filter

A high pass filter is a circuit that attenuates, or blocks, low frequencies, while passing higher frequencies. Just like with the low pass filter the cutoff point is where the output is -3dB of the input. See Figure 5.4

5.3.3 Band-Pass Filter

A band pass filter is a circuit that passes a certain range of frequencies. One way of thinking about it is that you have a high pass and a low pass at the same time, so only frequencies between the two filters are allowed to pass through. The center point of the frequencies being passed, or the 'pass band', is called the center frequency. Since a

band pass filter has two sides, there is technically two corner frequencies, one for the high pass and one for the low pass. For the filters we will be using, we want to make a symmetric response, so that each cut-off frequency will be equidistant from the center frequency. See figure 5.5

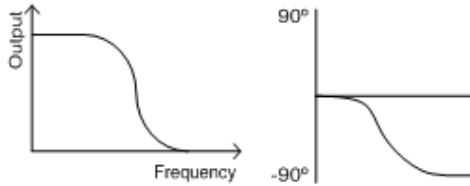


Figure 5.3: Low pass filter response/phase response

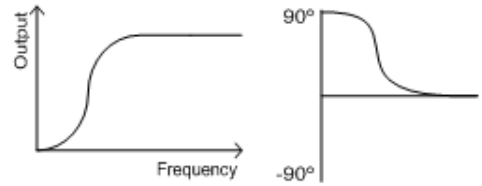


Figure 5.4: High pass filter response/phase response

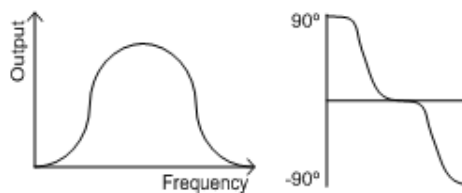


Figure 5.5: Band pass filter response/phase response

5.4 Guess the Filter

1. Connect the circuit in figure 5.6 and measure both the input voltage, V_{IN} , and the output voltage, V_{OUT} , on the oscilloscope. For V_{IN} use an 8VPP sine wave at a starting frequency of 10 Hz. This data should be recorded in table 5.1.

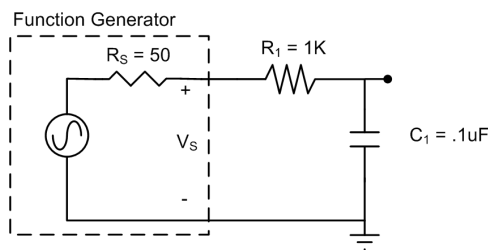


Figure 5.6: Mystery Filter One

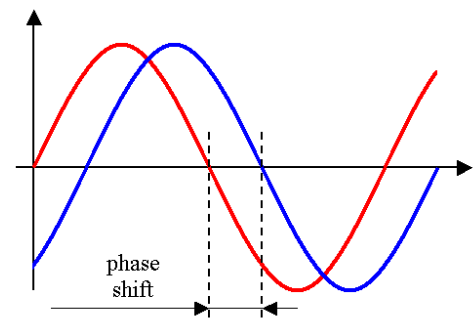


Figure 5.7: A Sample Phase Shift

2. Slowly vary the frequency from 10 Hz up to 300 kHz. You should see that the output voltage changes over this frequency range. Please find the corner frequency and record it below. Calculate the phase shift of V_{IN} and V_{OUT} at this frequency and determine what type of filter this is. Remember that the corner frequency is where $V_{OUT} = (0.707) \times V_{IN}$.

F_{3db} Both of Them: _____ Phase Shift: _____
 Filter Type: _____

Hint: To find the phase shift, keep in mind that a full revolution of a sinusoidal wave is 360 degrees. Since you will need to find the phase shift between V_{IN} and V_{OUT} , calculate that value using time shift and period measurements from the oscilloscope. See lab 2 for more information.

Table 5.1: Classification of Filter One

VIN (V)	Freq (Hz)	VOUT (V)	Period (sec)	Time Shift	Phase Shift
8	10	8	.1	0	0°
8	30				
8	100				
8	300				
8	1k				
8	3k				
8	10k				
8	30k				
8	100k				
8	300k				

- Once you have found the corner frequency of the circuit in Figure 5.6, plot the magnitude of V_O and phase response of the range of frequencies from 10Hz to 300kHz. Fill in table 5.1 with your calculations then plot your results on a separate sheet of paper. Use logarithmic scale for frequency axis.
- Connect the circuit in figure 5.8 and measure both the input voltage, V_{IN} , and the output voltage, V_{OUT} , on the oscilloscope. For V_{IN} use an 8VPP sine wave at a starting frequency of 10 Hz.
NOTE: At low frequencies the DC resistance of the inductor is much larger than the reactance, so it appears to be a resistor not an inductor.

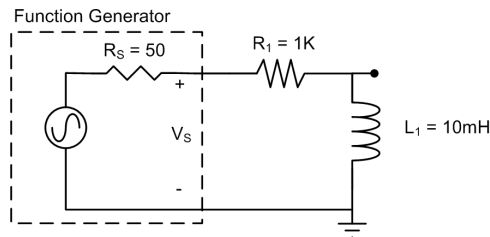


Figure 5.8: Mystery Filter Two

- Slowly vary the frequency from 10 Hz up to 300 kHz. You should see that the output voltage changes over this frequency range. Please find the corner frequency and record it below. Calculate the phase shift of V_{IN} and V_{OUT} at this frequency and determine what type of filter it is.

F_{3db} Both of Them: _____ Phase Shift: _____
Filter Type: _____

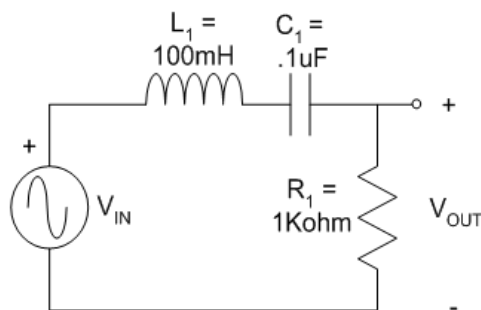


Figure 5.9: Mystery Filter Two

6. Once you have found the corner frequency of the circuit in Figure 5.8, plot the magnitude of V_{OUT} and phase response of the range of frequencies from 10Hz to 300kHz. Fill in table 5.2 with your calculations then plot your results on a separate sheet of paper. Use logarithmic scale for frequency axis.

Table 5.2: Classification of Filter Two

VIN (V)	Freq (Hz)	VOUT (V)	Period (sec)	Time Shift	Phase Shift
8	10	0	.1	.025	90°
8	30				
8	100				
8	300				
8	1k				
8	3k				
8	10k				
8	30k				
8	100k				
8	300k				

7. Connect the circuit in figure 5.9 and measure both the input voltage, V_{IN} , and the output voltage, V_{OUT} , on the oscilloscope. For V_{IN} use an 8VPP sine wave at a starting frequency of 10 Hz.
8. For this circuit you will end up with two frequencies where $V_{OUT} = (0.707) \times V_{IN}$. Because of this you will need to take extra measurements.

F_{3db} Both of Them: _____ Phase Shift: _____
 Filter Type: _____

9. Once you have found the corner frequencies of the circuit in Figure 5.9, plot the magnitude of V_{OUT} and phase response of the range of frequencies from 10Hz to 300kHz. Fill in table 5.3 with your calculations then plot your results on a separate sheet of paper. Use logarithmic scale for frequency axis.

Table 5.3: Classification of Filter Three

VIN (V)	Freq (Hz)	VOUT (V)	Period (sec)	Time Shift	Phase Shift
8	10	0	.1	.025	90°
8	30				
8	100				
8	300				
8	1k				
8	3k				
8	10k				
8	30k				
8	100k				
8	300k				

5.5 Study Questions

1. Be sure to include the completed tables, along with computer (Excel or Matlab) plots of the magnitude and phase response based on your table.
2. For each filter, calculate the 'real' value of your components from the (L and C) assuming the value of R is correct and using your known frequency. These must be within .5 percent of the real values, not the values written on the parts.

Chapter 1

Cross Over Design

1.1 Pre-Lab

The answers to the following questions are due at the beginning of the lab. If they are not done at the beginning of the lab, no points will be awarded. Bring two copies as one will be turned into the teaching assistant.

1. No Pre-lab this time!

1.2 Section Overview

Most home and car audio systems contain multiple speakers such as a tweeter, mid-range and woofer. This is because each speaker has a specific range of frequencies it was designed to output. In fact, tweeters may break if powered with lower frequencies and woofers can not move fast enough to generate higher frequencies. To deal with this, speaker systems have crossovers. These crossovers are filters designed to prevent unwanted frequencies to each speaker. Crossovers are commonly designed to be either two-way or three-way based on the number of speakers, two and three speakers respectively. In this lab, each group will be designing and building a three way crossover network to drive a multiple speaker system. Speakers are provided, but these speakers too have optimal output frequencies. The following graph, figure 1.1, shows what the frequency response of a crossover network response might look like. Note that the solid lines are magnitude and the dashed lines are phase shifts.

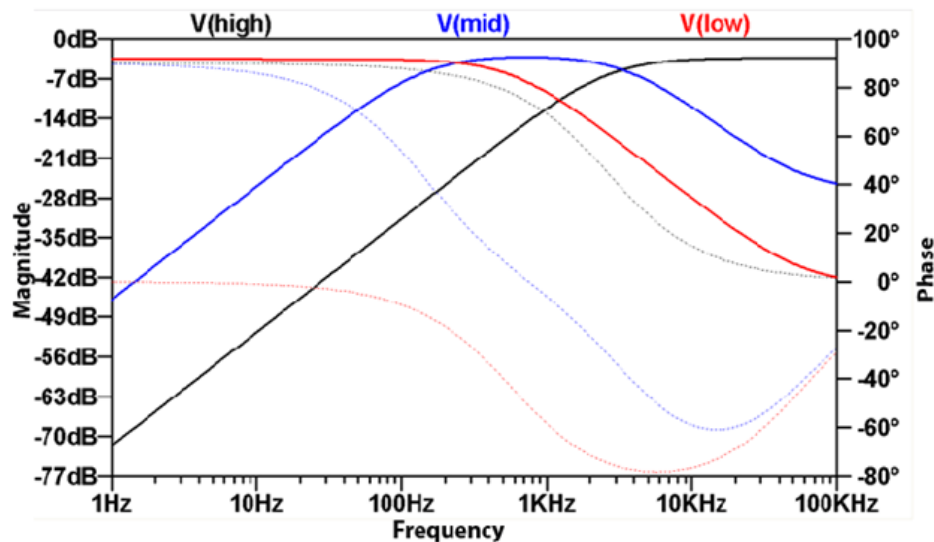


Figure 1.1: Three-way Crossover Frequency Responses

1.3 Procedure

The final product of this lab is a functional crossover network comprised of three filters and an amplifier for each filter. When creating crossovers, there are two methods of filtering the audio. The first is to filter after amplification, and the second is to filter before amplification. Some simple, low cost, and low quality systems will filter after amplification, but this wastes power and requires bulky components to work well. Most designs have the cross over first then amplify the signal for each speaker separately. This is the recommended design. A block diagram is shown in figure 1.2

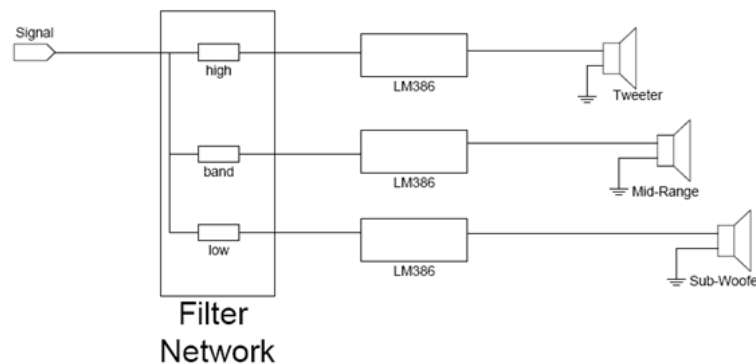


Figure 1.2: Block Diagram for Three-Way Crossover

The procedure for this lab can be broken up into three main steps as listed below.

1. Design a bandpass, low pass, and high pass filters with appropriate pass frequencies. The design will be simulated in LTspice using an AC analysis to ensure expected behavior.
2. Construct a physical prototype of each filter and test the filters over a range of frequencies using the frequency generator and o-scope.
3. Solder the working filters and amplifiers onto a protoboard and prepare for final testing. A schematic for the amplifier will be provided.

At the end of this lab, each group will perform a final test of their speakers and crossover using a frequency sweep, such as the *Ultimate Sound Test* on YouTube. Grading will be based on how well each filter passes the expected frequencies and stops all others.

1.4 Design and Simulation

The first step in the design phase for this lab is to decide on appropriate pass and stop frequencies for each filter. The tweeter speaker can produce frequencies greater than 5Khz, and the sub box can produce frequencies less than 200Hz. For proper sound, the midrange should produce all frequencies not produced by the subwoofer or tweeter. To verify that the mid-range can produce these frequencies, refer to the datasheet for the midrange on the webpage.

The final implementation of the design will be driven by an audio jack on a computer. In order for this to be successful the filter can not be designed so that it 'overloads' the audio jack. If this was to happen, no sound would come from the computer and it could even be damaged. When designing the system assume that the audio jack on the computer cannot source more than 10mA of current. As a rule of thumb, avoid using any resistors less than 100 Ohms.

Based on the frequencies that should be filtered, create filters similar to those from Lab 5. The corner frequencies will need to be modified, which can be done by changing component values. It is recommended that the values for the inductors be selected first from the possible values included in the lab kits (1mH and 10mH) as it is easier to find capacitors than it is to find inductors. Starting from those values, design the filters. Pay attention to the output voltages of your filters. The input will typically be at most 1 Vpp and the amplifiers included in your kits have a fixed gain of 20. You will be supplying the amplifiers with 12 volts. This means to be safe that the maximum output from your filters can be about 0.6 Vpp. If your output voltage is too great, you can simply turn down the volume.

HINT: To best design the filter, the output impedance is also needed. This can be found on the datasheet for the LM386 part. Be sure to account for it.

CHAPTER 1. CROSS OVER DESIGN

To simulate the filters, use an *AC Analysis* in LTspice. To perform an AC analysis, draw the circuit and set the voltage source to a DC value of **AC 1**. Under **Simulate**, choose **Edit Simulation Cmd** and select the **AC Analysis tab**. Choose **Decade** for the type of sweep, **100** points per decade, a start frequency of **10Hz**, and a stop frequency of **100Khz**. Running the simulation will now give a graph of output voltage relative to input voltage on a log scale over the selected range of frequencies. Note: the solid line is the magnitude response, and the dashed line is the phase shift.

1.5 Prototype and Test

Once the filters are designed and verified, construct the filters on a breadboard and test the response over a range of frequencies. Use the frequency generator and oscilloscope to fill in the table 1.1 below.

Table 1.1: Crossover Measurements

Freq (Hz)	VIN	VOUT Low	VOUT Mid.	VOUT High
10Hz				
30Hz				
100Hz				
300Hz				
1KHz				
3KHz				
10KHz				
30KHz				

1.6 Construction and Final Testing

The final step is to solder the circuit onto a protoboard. Solder the filters to one side of the board, conserving as much space for amplifier as possible. For soldering the amplifier, refer to the demo board and the schematic provided. Be sure to connect wires to the filter input, output, and the amplifier output for testing and verification.

Table 1.2: Final Testing

Filter	Pass Bands Acceptable?	Stop Bands Acceptable?
Low Pass		
High Pass		
Band Pass		

1.7 Study Questions

Turn in a well written report including all schematics, calculations, and simulation results. Be sure to describe how you determined the component values for each filter.