Circuit Analysis

Department of Computer Engineering

Khwaja Fareed University of Engineering and
Information Technology
Rahim Yar Khan, Pakistan

Draft Copy – 18 July 2017

Contents

Chapter 1

Lab 1: Introduction to the Lab

1.1 Introduction

The following lab is designed to give a brief description of how to use the necessary equipment to perform most of the labs you will encounter. You will use primarily four pieces of equipment; the multimeter (DMM), the oscilloscope, the function generator, and the trainer. The following is a brief description of your purpose for using each piece of equipment:

- The multimeter measures both alternating current (AC) and direct current (DC) voltage (V), current (I), and resistance (R). The multimeter also measures frequency in Hertz (cycles per second.)
- The digital oscilloscope displays and measures voltage waveform and frequency; it also has several other useful, yet convenient, functions.
- The function generator generates sine, triangle, and square waves at the selected frequency and amplitude. The function generator could be regarded as your AC power supply.
- The trainer contains the nodes for your circuit layout. It also contains variable DC power supplies, its own function generator, LED outputs, an 8 speaker, two variable resistors, and various switches.

  You will not use all of the options on the equipment, some control descriptions have been left off of the following equipment drawings.

1.2 THE DIGITAL MULTIMETER (DMM)

1.2.1 Measuring DC Voltage

- Insert the appropriate red lead into the top right opening (the + end of the V);
- Insert the appropriate black lead into the middle opening (the âĂŞ end of the V.)

- Next, press the DC V button.
- Then, apply the clips at the end of the leads as shown in Figure 1.1 (In the example below, the voltage is being measured across the resistor.
- The rest of the circuit has been excluded for simplicity.)
- The display will read the DC voltage across the resistor.

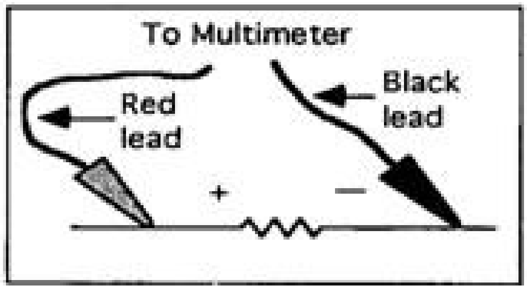

Figure 1.1: Measuring Voltage

1.2.2 Measuring DC Current

- Insert the appropriate red lead into the bottom right opening (the + end of the I);
- Insert the appropriate black lead into the middle opening (the âĂŞ end of the I.)
- Next, press the Shift button and then the DC V button. Then, apply the clips at the end of the leads as shown in Figure 1.2
- The rest of the circuit has been excluded for simplicity. The display will read the DC current through the resistor.

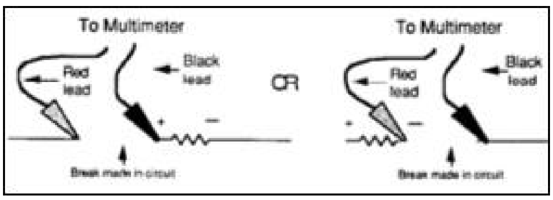

Figure 1.2: Measuring Voltage

1.2.3 Measuring AC Voltage

- Insert the appropriate red lead into the top right opening (the + end of the V); insert the appropriate black lead into the middle opening (the âĂŞ end of the V.)
- Next, press the AC V button.
- Then, apply the clips at the end of the leads as shown in Figure 1.1.
- The display will read the AC voltage (RMS) across the resistor.

1.2.4 Measuring AC Current

- Insert the appropriate red lead into the bottom right opening (the + end of the I); insert the appropriate black lead into the middle opening (the âĂŞ end of the I.)
- Next, press the Shift button and then the AC V button. Then, apply the clips at the end of the leads as shown in Figure 1.2.
- The display will read the AC current (RMS) through the resistor.

1.2.5 Measuring Resistance

- Insert the appropriate red lead into the top right opening (the + end of the V);
- Insert the appropriate black lead into the middle opening (the âĂŞ end of the V.) After doing the above, press the2 W button.
- Remove at least one of the ends of the resistor from the circuit (the correct resistance of the resistor cannot be measured while the resistor is in the circuit.)
- Next, place the clips across the resistor to be measured.
- The display will read the resistance of the resistor.

1.2.6 Measuring Frequency

Follow the steps for measuring voltage, but press the Freq. Button instead of a voltage button. The display will give the frequency of the voltage or current.

1.2.7 Measuring Time Period

Follow the steps for finding frequency except press the Shift button and then the Freq.Button. The display will give the period of the voltage or current.

1.3 The Oscilloscope

Unlike the multimeter, the oscilloscope can display two channels of voltage (such as the voltage across two resistors); however, the black clip of both channels must be connected to the ground of the circuit. In other words, if you were measuring the voltage across a resistor, one of its ends must be connected to ground in the circuit (with the black clip connected to the grounded end); otherwise, you will not get a correct reading and may damage the oscilloscope and/or the circuit! The oscilloscope is able to display both a picture of the voltage over time, called a waveform, and measured information about the waveform such as frequency or amplitude.

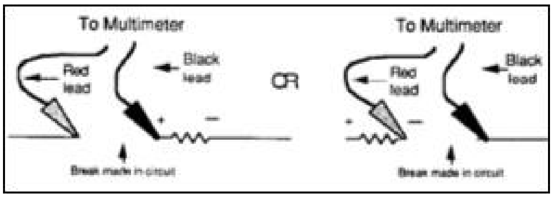

Figure 1.3: Oscilloscope in Lab

1.3.1 Measuring DC Voltage

Generally, the oscilloscope is not used to measure DC voltages; however, it is possible to do so with the oscilloscope.

- First, connect the cable leads making sure the âĂŞ end is connected to ground. The cable is connected to the oscilloscope at the BNC connector labeled 1 X (channel 1) or 2 Y (channel 2).
- After doing this, turn the corresponding Volts/Div knob until a signal is observed.
- You may also need to turn the Time/Div knob if the oscilloscope is slowly displaying your signal.
- Besides the signal, additional information will be present on the screen.
- The channel gain settings (in volts/division) are displayed in the upper left corner of the screen.
- The channel gain determines the Y-axis scale.
- The time/division (X-axis scale) is displayed at the top-middle portion of the screen.
- The time/div setting is not important for DC measurements.
- Once a signal is observed on the screen, the channel gain can be used to calculate the DC voltage.

- Count the number of graticule divisions (grid lines) between the channel ground reference (the small ground symbol on the right edge of the screen) and the signal you are measuring.
- The voltage is calculated by multiplying the channel gain (Volts/div) by the number of divisions.
- In order to simplify the division counting, you may wish to move the position of the channel ground reference onto a graticule line by adjusting the corresponding position knob.

1.3.2 Measuring AC Voltage

- Connect the leads of the channel(s) of the oscilloscope making sure the negative (-) end is connected to ground.
- After doing so, press the auto-scale button (AS).
- The waveform(s) should be displayed on the screen.

  Note: if you are only measuring one waveform, you should disconnect the leads of the channel you are not using, otherwise the oscilloscope may have a difficult time displaying the signal after pressing auto-scale.
- In addition to the signal, other information will be present on the screen.
- The channel gain settings (in volts/division) are displayed in the upper left corner of the screen.
- The channel gain determines the Y-axis scale.
- The time/division (X-axis scale) is displayed at the top-middle portion of screen.

1.3.3 Adjusting the Display

- After pressing auto-scale, you may want to modify the display.
- For example, you may want to show more periods of the waveform.
- This can be accomplished by increasing the amount of Time/div.
- This is achieved by turning the Time/div knob counter-clockwise.
- Similarly, you may want to adjust the amplitude display of the signal.
- Turn one of the Volts/Div knobs to change the display scale (there is one knob for each channel.) You also may want to move one of the waveforms up of down the screen.
- Turn one of the position knobs to move the waveforms.
- Note that a ground reference point for each displayed waveform is shown on the right side of the screen.

1.3.4 Time Measurement

- Although you can manually make time measurements by multiplying the time/-div setting by the number of divisions, the oscilloscope will do this for you.
- After displaying the waveform on the screen press the time button (T) which is located in the Measurement section of the control faceplate.
- You will now see various voltage options (frequency, period, duty cycle, and channel #) located on the screen above the hot keys.
- Press the desired key, and measurement will be listed at the bottom of the screen.
- You can also measure the time difference between two waveforms by pressing the Cursors buttons located in the Measurement section of the control faceplate.
- You will now see various cursor options (V1, V2, t1, and t2) located on the screen above the hot keys.
- Press the t1 key, and move the horizontal dashed line using the control knob just below the Cursors button to the desired position.
- Next press the t2 key and do the same for that dashed line.
- You will then see the location of each of the two cursors displayed on the bottom of the screen as well as the difference in voltage, t.

1.4 Function Generator

The function generator produces sine, triangle, and square waveforms of various amplitudes and frequencies. Although there are additional functions listed on the faceplate, you will only need to concern yourself with those mentioned above.

1.4.1 Waveform Selection

The three waveforms listed have corresponding buttons located approximately in the center of the faceplate (the fourth button is a one-time pulse which you will not use.) To select the desired waveform, simply press the button corresponding to the desired waveform. An LED should be lit inside the selected waveformâĂŹs button.

1.4.2 Frequency Selection

- The frequency is selected by first pressing the frequency button until both the FRQ wording is lit as well as the FRQ button LED.
- Once these are all lit, the verniers may be pressed to change the display.

- Note that the three buttons each operate on one of the digits in the display.
- The range of the frequency is viewed by viewing the appropriate LEDs (those having Hz in the unit.)
- The range may be changed by pressing the range button up or down.

1.4.3 Signal Output

- The appropriate leads should be connected to the output terminal. Note: NEVER apply a voltage input to the OUTPUT terminal.
- Before the function generator will supply the appropriate output, the DISABLE LED should be off.
- If it is not off, press the DISABLE button.

1.5 Activities & Excercise

1.5.1 DMM

- Measure the voltage of the AC line.
- Measure the frequency of the line AC voltage.
- Measure the resistance of the given resistors.

1.5.2 Oscilloscope

- Look over the oscilloscope at your bench. Find the major controls, as described in your manual.
- Each bench is equipped with a signal generator. Look it over, and set it up to produce a triangular waveform output at about 1 kHz.
- Connect the output of the signal generator to the oscilloscope and get a stable display. Use Normal Trigger. Do not use AUTO triggering. Set it up to trigger on the channel connected to the signal generator. Set the trigger polarity to rising (+). Adjust the output of the signal generator to 1V peak to peak. Adjust the oscilloscope sensitivity so that the waveform is as large as possible, using standard settings only. Adjust the time scale so that at least 2 and no more that 5 complete cycles are displayed.
- Experiment with the trigger level control. Notice how it sets where in the waveform the display begins. You should be able to move the starting point of the trace back and forth along the ramp in the triangular wave. Notice how the trace disappears as the level is set above or below the top or bottom of the waveform.

- Set the trigger polarity to falling (-), and observe that the trace now begins on the falling ramp of the wave.
- Connect both probes of the oscilloscope to the oscilloscope probe calibration connection on the front of the scope. Adjust the oscilloscope so that both waveforms are displayed simultaneously.
- Use the RUN/STOP button to freeze an image of the signals. Use the cursors to determine the frequency, period, and amplitude of the signal being measured.
- Change the signal generator waveform to a different type, and experiment with viewing it. There is no substitute for familiarity when it comes to operating an oscilloscope.

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 1.1: Lab Rubric

Chapter 2

Lab 2 : Introduction to PSpice

2.1 Introduction

Electronic circuit design requires accurate methods for evaluating circuit performance. Because of the enormous complexity of modern integrated circuits, computer-aided circuit analysis is essential and can provide information about circuit performance that is tedious to obtain with laboratory prototype measurements. Computer-aided analysis makes possible the following procedures:

- Evaluation of the effects of variations in such elements as resistors, transistors, and transformers.
- Assessment of performance improvements or degradations.
- Evaluation of the effects of noise and signal distortion without the need for expensive measuring instruments
- Sensitivity analysis to determine the permissible bounds determined by the tolerances of all element values or parameters of active elements
- Fourier analysis without expensive wave analyzers
- Evaluation of the effects of nonlinear elements on circuit performance
- Optimization of the design of electronic circuits in terms of circuit parameters SPICE (Simulation Program with Integrated Circuit Emphasis) is a general-purpose circuit program that simulates electronic circuits. It can perform analyses on various aspects of electronic circuits, such as the operating (or quiescent) points of transistors, time-domain response, small-signal frequency response, and so on. SPICE contains models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is a versatile program and is widely used in both industry and academic institutions.

2.2 Background

PSpice is a member of the SPICE family of circuit simulators, all of which originate from the SPICE2 circuit simulator, whose development spans a period of about 30 year. During the mid-1960s, the program ECAP was developed at IBM [1]. In the late 1960s, ECAP served as the starting point for the development of the program CANCER at the University of California (UC) at Berkeley. Using CANCER as the basis, SPICE was developed at Berkeley in the early 1970s. During the mid-1970s, SPICE2, which is an improved version of SPICE, was developed at UC Berkeley. The algorithms of SPICE2 are robust, powerful, and general in nature, and SPICE2 has become an industry-standard tool for circuit simulation. SPICE3, a variation of SPICE2, is designed especially to support computer-aided design (CAD) research programs at UCâĂŞ Berkeley. As the development of SPICE2 was supported using public funds, this software is in the public domain, which means that it may be used freely by all U.S. citizens.

SPICE2, referred to simply as SPICE, has become an industry standard. The input syntax for SPICE is a free-format style that does not require data to be entered in fixed column locations. SPICE assumes reasonable default values for unspecified circuit parameters. In addition, it performs a considerable amount of error checking to ensure that a circuit has been entered correctly. PSpice, which uses the same algorithms as SPICE2, is equally useful for simulating all types of circuits in a wide range of applications. A circuit is described by statements stored in a file called the circuit file. The circuit file is read by the SPICE simulator. Each statement is self-contained and independent of every other statement, and does not interact with other statements. SPICE (or PSpice) statements are easy to learn and use. A schematic editor can be used to draw the circuit and create a Schematics file, which can then be read by PSpice for running the simulation.

2.3 Circuit Analysis Using PSPICE

PSpice allows various types of analysis. Each analysis is invoked by including its command statement. The types of analysis and their corresponding dot commands are described in the following text.

DC analysis is used for circuits with time-invariant sources (e.g., steady-state DC sources). It calculates all node voltages and branch currents for a range of values, and their quiescent (DC) values are the outputs. The dot commands and their functions are:

- DC sweep of an input voltage or current source, a model parameter, or temperature over a range of values (.DC)
- Determination of the linearized model parameters of nonlinear devices (.OP)
- DC operating point to obtain all node voltages

- Small-signal transfer function with small-signal gain, input resistance, and output resistance (Thevenin s equivalent; .TF)

- DC small-signal sensitivities (.SENS)

  Transient analysis is used for circuits with time-variant sources (e.g. AC sources and switched DC sources). It calculates all node voltages and branch currents over a time interval, and their instantaneous values are the outputs. The dot commands and their functions are:

- Circuit behavior in response to time-varying sources (.TRAN)

- DC and Fourier components of the transient analysis results (.FOUR)

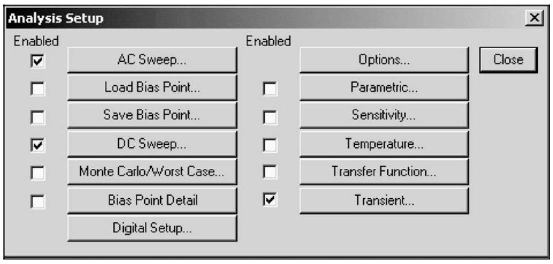

Figure 2.1: Analysis Setup in PSpice Schematics version

AC analysis is used for small-signal analysis of circuits with sources of variable frequencies. It calculates all node voltages and branch currents over a range of frequencies, and their magnitudes and phase angles are the outputs. The dot commands and their functions are:

- Circuit response over a range of source frequencies (.AC)

- Noise generation at an output node for every frequency (.NOISE) In Schematics versions, the commands are invoked from the setup menu, as shown in the figure 2-1.

2.4 CONSTRUCTING AND SIMULATING A DC CIRCUIT

Lets draw the simple DC circuit in Figure 2-2 using Schematics. Lets get Schematics using the Start/Programs/PSpice Student/Schematics sequence of pop-up menus. When Schematics opens, your screen will change to Schematic Editor window in Figure 2-3. Notice that the window bar across the top informs us that this is a new schematic. Also, note the Help menu – it will get some use. A quick look under the Draw menu, Figure 2-4, shows the options weâĂŹll need to create our schematic – namely, Get New Part and Wire. Lets get started.

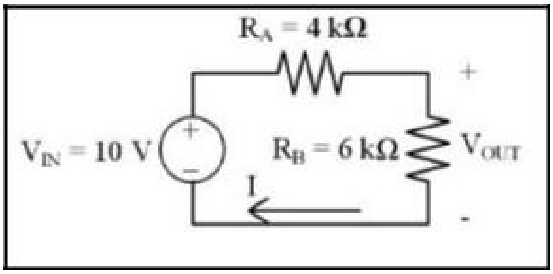

Figure 2.2: Basic Circuit

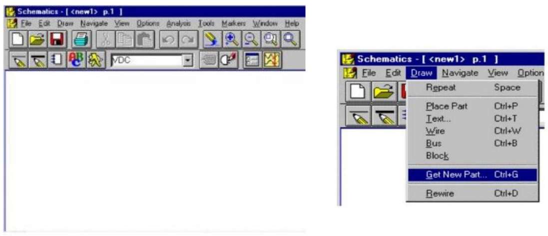

Figure 2.3: Getting Parts

2.4.1 Getting Parts

First, lets get the voltage source. This is done by left clicking the Draw menu and selecting Get New Part. The Part Browser dialog box in Figure 2-5 appears listing all the parts in the evaluation version. Since we do not know the Schematics name for a dc voltage source, we select Libraries, and the Library Browser window in Figure 2-5 appears listing the various libraries the parts are divided into. For example, resistors, capacitors and inductors are in the ANALOG.slb library. Integrated circuit parts in the EVAL.slb library. Of interest to us is the SOURCE.slb library, which has already been selected in Figure 2-5. Select the part VDC and OK. The box in Figure 2-5 should reappear. Select Place & Close and you are back to the schematic page with one difference: the mouse pointer is now a dc voltage source! Just point it any place inside the drawing area and left click ONCE. The result is a voltage source in our schematic. Since we need only one source, we click the right mouse button to stop placing sources and the mouse pointer returns to its original status as shown in Figure 2-6. Notice that Schematics calls the source V1.If we had continued to left click, we would have created V2, V3, etc.. Also, note the grid dots. Schematics always align whatever you draw to the grid dots even if you place the part between dots. Finally, the two leads extending from the source will be used to make wiring connections to the resistors RA and RB. This, real world feeâĂŹ is one feature that makes Schematics so popular. Now we'll place the resistors. We repeat the process of Draw/Get New Part except this time, when the Part Browser Basic dialog box appears; we type in R and select OK. Now the mouse pointer is a resistor. We left click once to place R1, MOVE THE MOUSE SOME DISTANCE, and left click to place R2. To stop adding resistors, click once.

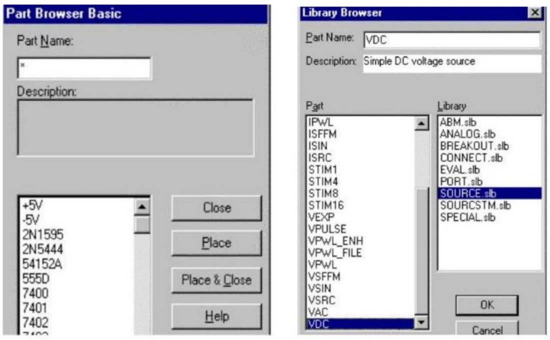

Figure 2.4: Getting Parts

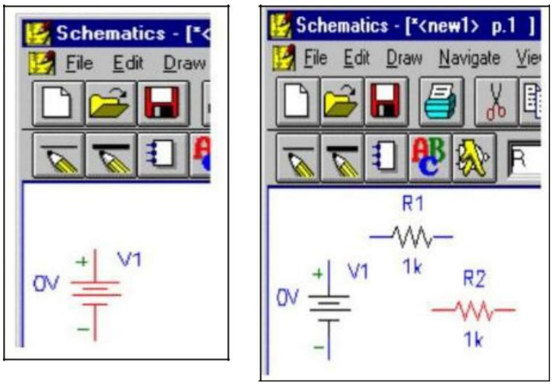

Figure 2.5: Getting Parts

Moving the mouse between part placements keeps them from stacking atop one another in the diagram. Figure 2-6 shows the present condition. Although you may not be able to see it in this black and white print, R2 is red meaning it is still selected and may be edited. Note that the resistors are automatically assigned values of 1 kâĎę.

2.4.2 Changing Parts Attributes

To change the voltage sources attributes (its name and value), simply double click on the name V1 and the Edit Reference Designator window in Figure 2-7 appears. Type in the new name, in this case Vin, and select OK. To change the sources value, double click on the value in Figure 2-6, to open the Attributes window in Figure 2-7. Enter 10V and select OK. Do likewise for the resistors, renaming them Ra and Rb and setting their values at 4 kâĎę and 6 kâĎę respectively.

2.4.3 Arranging Parts and Pin Numbers

Every pin (external connection) on every part in every Schematics library is numbered. When arranging parts in Schematics, it is vital to understand the relationship

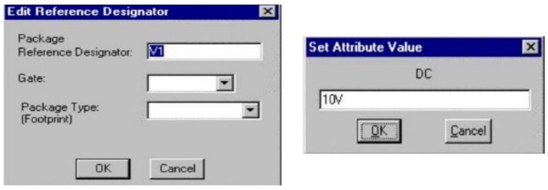

Figure 2.6: Getting Parts

between the passive sign convention in Figure 2-2, current directions in PSpice and the partâĂŹs pin numbers. When a part is placed, it is automatically oriented either vertically or horizontally. As shown in Figure 2-8, vertically placed parts have pin 1 at the top and pin 2 at bottom, while horizontally oriented parts have pin 1 on left and pin 2 on right. Now consider R2 in our circuit in Figure 2-9. We need to rotate it

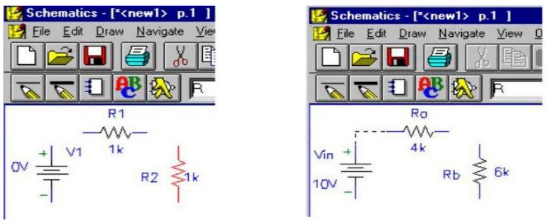

Figure 2.7: Getting Parts

for appearance sake. Also, the current of interest, I, flows downward through Rb in Figure 2-2. Selecting Rotate from the Edit menu (or CNTL-R) will spin Rb 90 counterclockwise, putting pin 1 at the bottom. Should we ask for the current in Rb, we will get the negative of I. It is better to rotate Rb 270, leaving pin 1 at the top. Now the current in Rb as PSpice sees it is the same as I. Of course, a part can be dragged to a new position. To make our schematic match Figure 2-2, weâĂŹll rotate R2 three times and drag it at the right end of the diagram. A diagram similar to Figure 2-9 should result.

2.4.4 Wiring parts

To connect the parts, go to the Draw menu and select Wire. Now the mouse pointer turns into a symbolic pencil. To connect the top of the source to Ra, use the mouse to place the pencil at the end of the wire stub from Vin, click once and release. Next, move the mouse up and over to the left end of Ra. The wire is drawn up and over at 90Âř angles as the dashed lines in Figure 2-9. DASHED LINES ARE NOTYET WIRES!

Left click once to complete cut the connection. The dashed lines become solid and the connection is made as shown in Figure 2-10. Any leftover wire fragments (dashed lines) can be removed by selecting Redraw from the Draw menu. As for drawing the two remaining wires, here a Schematics shortcut. To reactivate the wiring pencil, if you need to, double right click. This shortcut reactivates whatever the most recent mouse use happened to be. Simply repeat the steps listed above to complete the wiring. Figure 2-10 shows the wired schematic. A note of caution – be patient when wiring. This is the area of Schematics where you are most likely to err, so be careful.

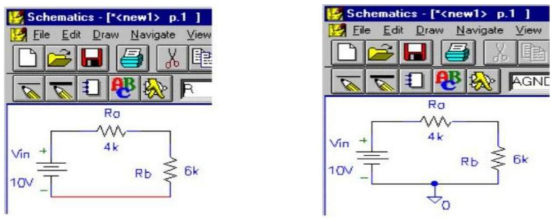

Figure 2.8: Getting Parts

2.4.5 The Ground

PSpice requires that all schematics have a ground, or reference terminal. The voltage there will be zero and all other node voltages are referenced to it. The part you need is either the analog ground (AGND) or the earth ground (EGND) in the PORT.slb library. In this example, we will get the AGND part and place it at the bottom of the schematic as shown in Figure 2-10. If the connection dot does not appear in your schematic, simply select the AGND part and move until it touches the bottom wire in the diagram. Your schematic is finished and ready for saving.

2.4.6 Saving Schematics

Simply go to File menu and select Save. PSpice will try to save your schematics in the My Documents directory. Its much better to save your work in a different directory. Schematic files are given the extension .sch.

2.5 Getting Results

The netlist is the old-fashioned Spice code listing for the circuit you have drawn in Schematics. To create the netlist, go to the Analysis menu and select Create Netlist. At the bottom of the Schematic Editor Window, is the either message Netlist Created or a dialog box appears informing you of netlist errors. Assuming your netlist is fine; you can view it by returning to the Analysis menu and selecting Examine Netlist. The Windows utility Notepad opens the netlist file shown in Figure 2-11. Notice that all three of our elements are here along with the proper values. The text $N0001 $N0002 etc., are the node numbers that Schematics made up when it converted our diagram to the netlist. Those of you familiar with Spice will recognize each line of the netlist as nothing more than the proper data statements for dc voltage sources and resistors. By tracing our way through the node numbers we can assure ourselves that our circuit is properly connected. The source is 10 volts positive at node 2 and with respect to 0. AGND is always node number zero. Ra connects 2 to 1 and Rb finished the loop between 0 and 1 âĂŞ itâĂŹs perfect. Remember, the order of the node numbers is very important! Later on, when we ask Schematics for the current through Ra for example, Schematics will give us the current from node 2 to node 1. This is consistent with the passive sign convention we discussed earlier. So, while the particular node numbers are not vital, their order is: and we find that order in the netlist.

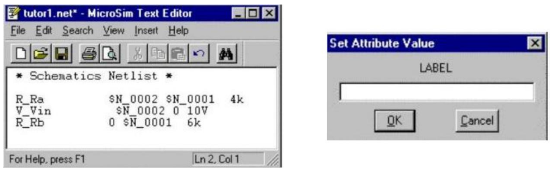

Figure 2.9: Results

2.6 Naming Nodes

As for we Schematics users, this is about the only use we have for the netlist. After all, the whole idea of the Schematics program is to get away from Spice formats; node numbers and data statements. Here the rub. We will eventually want to know the node voltages in our circuit. We need a way to tell Schematics which nodes are of interest to us. The old Spice way was to use the node number but we are trying our best to avoid those. Schematics allow us to give any node a unique name. Lets call the output node Vout. Simply double click on wire at the output node and the dialog in Figure 2-11 will appear. Type in Vout as shown and select OK. The schematic should look like Figure 2-12. You can drag the Vout label wherever you like.

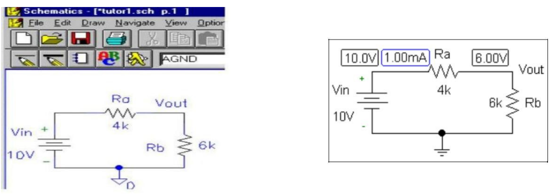

Figure 2.10: Results

2.7 Displaying Voltages and Current

On the Schematic In the Analysis menu, you will find the Display Results on Schematic option. When enabled, PSpice will print all dc node voltages and branch currents directly on the schematic. This is very convenient since both the circuit and the results are on one screen. To delete an undesired result, click on it and press the Delete key. The final circuit is shown in Figure 2-12 where results for Vin, Vout and I are included.

2.8 Activities and Excercise

- Draw a series circuit and find out the voltage drop across each resistor and total current passing through the circuit.

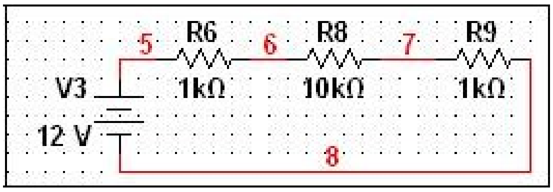

Figure 2.11: Question 1

Table 2.1: Question 1

Resister	R6	R8	R9	REQ	Voltage	Vs	VR6	VR8	VR9
Theoretical					Theoretical				
Measured					Measured				
% Error					% Error				

- Draw a parallel circuit and find out the currents through each resistor and voltage.

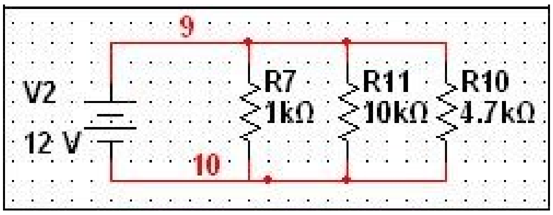

Figure 2.12: Question 2

- Draw a series and parallel combination circuit and find out the voltages and currents through each resistor.

Table 2.2: Question 2

Resister	R6	R8	R9	REQ	Voltage	Vs	VR6	VR8	VR9
Theoretical					Theoretical				
Measured					Measured				
% Error					% Error				

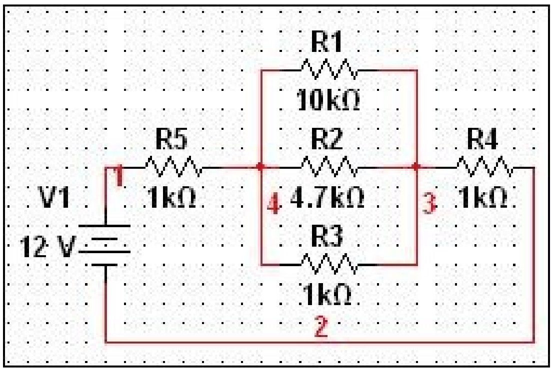

Figure 2.13: Question 2

Table 2.3: Question 3

Resister	R6	R8	R9	REQ	Voltage	Vs	VR6	VR8	VR9
Theoretical					Theoretical				
Measured					Measured				
% Error					% Error				

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 2.4: Lab Rubric

Lab 3

3.1 Objectives

The objective of this experiment is to analyze simple resistive circuits in DC. The circuits considered here are: resistors in series, resistors in parallel, series-parallel combination voltage divider, and current divider. This experiment will allow the experimental verification of the theoretical analysis.

3.2 Introduction

The theoretical analysis of the circuits under study is based on Ohm and Kirchhoffs laws.

3.2.1 Ohm Law

The voltage V (in Volts, V) across a resistor is directly proportional to the current I (in Amperes, A) flowing through it. The constant of proportionality is the resistance R (in Ohms).

3.2.2 Resistor in Series

The current through elements in series is the same for all of them. The voltage across the i_th element is The sum of the voltages across each element is equal to the voltage applied to the entire series combination. The equiva1ent resistance of the series combination is the sum of the individual resistances.

3.2.3 Resistor in Parallel

The voltage across N elements in parallel is the same for all of them. The current through the ith element is The sum of the currents through each element is equal to the current provided to the entire parallel combination.

3.2.4 Series-Parallel Combination

An example of a series-parallel combination circuit is shown in Figure 3-3. The analysis of this type of circuit is accomplished by substituting the series (or parallel) combinations by their equivalent resistances, such that the circuit is transformed into a pure parallel (or series) circuit. Once the electrical parameters (voltage and/or current) have been determined for the equivalent resistances, the voltages and/or currents for the individual resistors in the series or parallel combinations can be obtained by using these parameters as Vs and Is for the corresponding combination.

3.2.5 Voltage Divider

A series circuit with two resistors will divide the applied voltage into two voltages and across each resistor. Notice that is the output of the voltage divider (see figure 3-4), as it is referenced to ground. The proportion in which the input voltage is divided is given by In order for this circuit to operate as a voltage divider, the output current must be zero or very small compared with the current through.

3.2.6 Current Divider

A parallel circuit with two resistors will divide the applied current into two currents and through each resistor (see figure 3-5). The proportion in which the input current is divided is given by

3.3 Activities and Excercise

- Draw a series circuit and find out the voltage drop across each resistor and total current passing through the circuit.

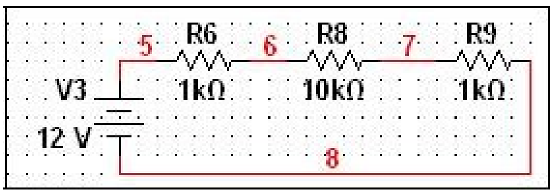

Figure 3.1: Question 1

Table 3.1: Question 1

Resister	R6	R8	R9	REQ	Voltage	Vs	VR6	VR8	VR9
Theoretical					Theoretical				
Measured					Measured				
% Error					% Error				

- Resistors in series: Assemble the circuit in figure 3-1 with component values (as provided in lab). Take measurements to complete the entries corresponding to the experimental values in table 3-1.

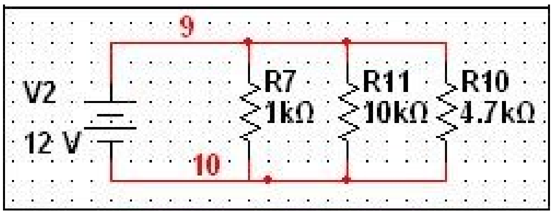

Figure 3.2: Question 2

Table 3.2: Question 1

Resister	R6	R8	R9	REQ	Voltage	Vs	VR6	VR8	VR9
Theoretical					**Theoretical**				
Measured					**Measured**				
% Error					**% Error**				

- Resistors in parallel: Assemble the circuit in figure 3-2 with component values (as provided in lab). Take measurements to complete the entries corresponding to the experimental values in table 3-2.

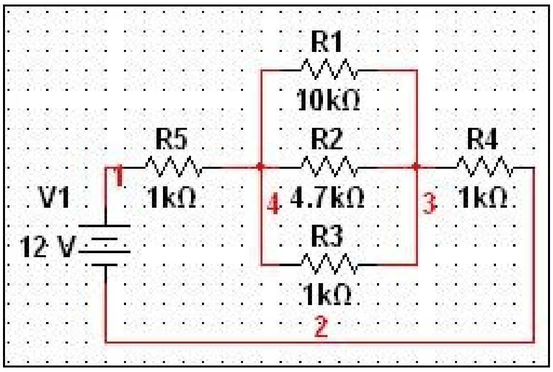

Figure 3.3: Question 2

Table 3.3: Question 3

Resister	R6	R8	R9	REQ	Voltage	Vs	VR6	VR8	VR9
Theoretical					Theoretical				
Measured					Measured				
% Error					% Error				

3.4 Analysis

- Calculate the error percentage between measure and theoretical data and complete all the corresponding entries in table 3-1 through 3-5. The error percentage is given by:

  From the above results comment on the three parameters with the highest error percentage and the three with the lowest error percentages. Discuss the possible causes for error and comment on the overall agreement between the measured and theoretical data.

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 3.4: Lab Rubric

Chapter 4

Lab 4: DC Circuit in Matlab

4.1 MATLAB Fundamentals

MATLAB is numeric computation software for engineering and scientific calculations. The name MATLAB stands for MATRIX LABORATORY. MATLAB is primarily a tool for matrix computations. MATLAB was originally written to provide easy access to the matrix computation software packages LINPACK and EISPACK.

MATLAB is a high-level language whose basic data type is a matrix that does not require dimensioning. There is no compilation and linking as is done in high-level languages, such as C or FORTRAN. Computer solutions in MATLAB seem to be much quicker than those of a high-level language such as C or FORTRAN. All computations are performed in complex-valued double precision arithmetic to guarantee high accuracy.

MATLAB has a rich set of plotting capabilities. The graphics are integrated in MATLAB. Since MATLAB is also a programming environment, a user can extend the functional capabilities of MATLAB by writing new modules.

MATLAB has a large collection of toolboxes in a variety of domains. Some examples of

MATLAB toolboxes are control system, signal processing, neural network, image processing, and system identification. The toolboxes consist of functions that can be used to perform computations in a specific domain.

4.1.1 Basic Operations

When MATLAB is invoked, the command window will display the prompt >>. MATLAB is then ready for entering data or executing commands. To quit MATLAB, type the command

exit or quit

MATLAB has on-line help. To see the list of MATLABâĂŹs help facility, type help

The help command followed by a function name is used to obtain information on a specific MATLAB function. For example, to obtain information on the use of fast

Fourier transform function, fft, one can type the command

help fft

The basic data object in MATLAB is a rectangular numerical matrix with real or complex elements. Scalars are thought of as a 1-by-1 matrix. Vectors are considered as matrices with a row or column. MATLAB has no dimension statement or type declarations. Storage of data and variables is allocated automatically once the data and variables are used. MATLAB statements are normally of the form:

$>> variable = expression$

Expressions typed by the user are interpreted and immediately evaluated by the MATLAB system. If a MATLAB statement ends with a semicolon, MATLAB evaluates the statement but suppresses the display of the results. MATLAB is also capable of executing a number of commands that are stored in a file. Consider the matrix Z:

$$M = \begin{bmatrix} 1 & 2 & 3 \\ 4 & 5 & 6 \\ 7 & 8 & 9 \end{bmatrix}$$

This matrix may be entered in MATLAB as follows:

$$M = \begin{bmatrix} 1 & 2 & 3 & ;4 & 5 & 6; & 7 & 8 & 9 \end{bmatrix}$$

Note that the matrix entries must be surrounded by square brackets

with row elements separated by blanks or by commas. The end of each row, with the exception of the last row, is indicated by a semicolon. Consider a row vector B with five elements:

$$M = \begin{bmatrix} 1 & 2 & 3 & 4 & 5 \end{bmatrix}$$

This vector can be entered in MATLAB as:

$$M = \begin{bmatrix} 1, & 2, & 3, & 4, & ,5 \end{bmatrix}$$

For readability, it is better to use spaces rather than commas between the elements. MATLAB is case sensitive in naming variables, commands and functions. Thus b and B are not the same variable. If you do not want MATLAB to be case sensitive, you can use the command

Command	Description
%	Comments. Everything appearing after % command is not executed.
demo	Access on-line demo programs
length	Length of a matrix
clear	Clears the variables or functions from workspace
clc	Clears the command window during a work session
clg	Clears graphic window
diary	Saves a session in a disk, possibly for printing at a later date

Figure 4.1: Question 2

4.1.2 Matrix Operation

The basic matrix operations are addition (+), subtraction (-), multiplication (), and conjugate transpose ′ of matrices. In addition to the above basic operations, MATLAB has two forms of matrix division: the left inverse operator or the right inverse operator /. Matrices of the same dimension may be subtracted or added. Thus if E and F are entered in MATLAB as

$$E = \begin{bmatrix} 7 & 2 & 3 & 4 & 3 & 6 & 8 & 1 & 5 \end{bmatrix}$$

$$F = \begin{bmatrix} 1 & 4 & 2 & 6 & 7 & 5 & 1 & 9 & 1 \end{bmatrix}$$

$G = E - F$

$$G = \begin{bmatrix} 6 & -2 & 1 \\ -2 & -4 & 1 \\ 7 & -8 & 4 \end{bmatrix}$$

$H = E + F$

$$G = \begin{bmatrix} 8 & 6 & 5 \\ 10 & 10 & 11 \\ 9 & 10 & 6 \end{bmatrix}$$

A scalar (1-by-1 matrix) may be added to or subtracted from a matrix. In this particular case, the scalar is added to or subtracted from all the elements of another matrix.

$J = H + 1$

$$J = \begin{bmatrix} 9 & 7 & 6 \\ 11 & 11 & 12 \\ 10 & 11 & 7 \end{bmatrix}$$

Matrix multiplication is defined provided the inner dimensions of the two operands are the same. Thus, if X is an n-by-m matrix and Y is i-by-j matrix, XY is defined provided m is equal to i. Since E and F are 3-by-3 matrices, the product $Q = EF$

$$Q = \begin{bmatrix} 22 & 69 & 27 \\ 28 & 91 & 29 \\ 19 & 84 & 26 \end{bmatrix}$$

Any matrix can be multiplied by a scalar. For example,

2 Q

$$ans = \begin{bmatrix} 44 & 138 & 54 \\ 56 & 182 & 58 \\ 38 & 168 & 52 \end{bmatrix}$$

Note that if a variable name and =sign are omitted; a variable name ans is automatically created. Matrix division can either be the left division operator $\backslash$ or right division operator/. The right division a/b, for instance, is algebraically equivalent to a/b, while the left division$a\backslash b$ is algebraically equivalent to b/a. If $I = Z^-1$ and Z is non-singular, the left division, Z/V is equivalent to the MATLAB expression $I = inv(Z) * V$
Note that if a variable name and $=$ sign are omitted; a variable name ans is automatically created. Matrix division can either be the left division operator$\backslash$ or right division operator/. The right divisiona/b, for instance, is algebraically equivalent to, while the left division a$\backslash$ b is algebraically equivalent tob/a. If= and Z is non-singular, the left division ,$Z\backslash V$ is equivalent to the MATLAB expression

$I = V * inv(Z)$

There are MATLAB functions that can be used to produce special matrices.

Function	Description
ones(n,m)	Produces n-by-m matrix with all the elements being unity
eye(n)	gives n-by-n identity matrix
zeros(n,m)	Produces n-by-m matrix of zeros
diag(A)	Produce a vector consisting of diagonal of a square matrix A

Figure 4.2: Question 2

4.2 Example

The voltage, v, across a resistance is given as (OhmâĂŹs Law), $V = iR$, where i is the current and R the resistance. The power dissipated in resistor R is given by the expression $P = i^2R$. If R = 10 Ohms and the current is increased from 0 to 10 A with increments of 2A, write a MATLAB program to generate a table of current, voltage and power dissipation.

%Voltage and power calculation

$R = 10$; % Resistance value

$i = 0 : 2 : 10$; % Generate current values

$v = i.R$; % array multiplication to obtain voltage

$p = (i.^2) * R$; % power calculation

4.2.1 M-Files

Normally, when single line commands are entered, MATLAB processes the commands immediately and displays the results. MATLAB is also capable of processing a sequence of commands that are stored in files with extension .m . MATLAB files with extension m are called m-files. The latter are ASCII text files, and they are created with a text editor or word processor. To list m-files in the current directory on your disk, you can use the MATLAB command what. The MATLAB command, type, can be used to show the contents of a specified file. M-files can either be script files or function files. Both script and function files contain a sequence of commands. However, function files take arguments and return values.

4.2.2 Script Files

Script files are especially useful for analysis and design problems that require long sequences of MATLAB commands. With script file written using a text editor or word processor, the file can be invoked by entering the name of the m-file, without the extension. Statements in a script file operate globally on the workspace data. Normally, when m-files are executing, the commands are not displayed on screen. The MATLAB echo command can be used to view m-files while they are executing. To illustrate the use of script file, a script file will be written to simplify the following complex valued expression z.

4.2.3 Function Files

Function files are m-files that are used to create new MATLAB functions. Variables defined and manipulated inside a function file are local to the function, and they do not operate globally on the workspace. However, arguments may be passed into and out of a function file. The general form of a function file is

$$functionvariable(s) = function_name(arguments)$$

% help text in the usage of the function

.

.

End

4.3 Actiities and Exercise

- Using Loop analysis to find out the current in the given circuit and write code in

  MATLAB:

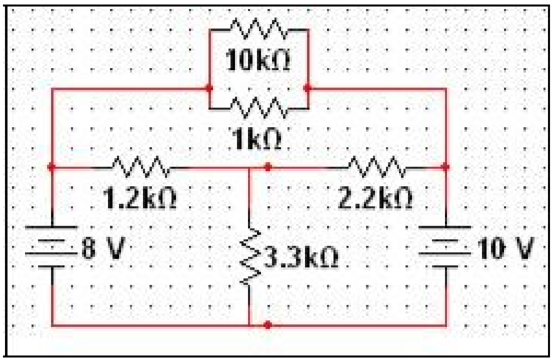

Figure 4.3: Question 2

- Find node voltages in the given network and write code in MATLAB for given network:

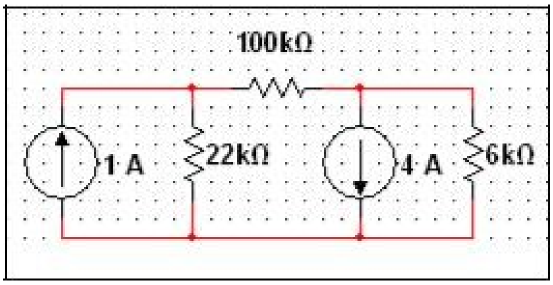

Figure 4.4: Question 2

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 4.1: Lab Rubric

Lab 5: Y-Δ Transformation

5.1 Introduction

In many circuits, resistors are neither in series nor in parallel. For these circuits, it may be necessary to convert from one circuit form to another to simplify the solution. Two typical circuit configurations that often have these difficulties are the Wye (Y) and Delta (D) circuits. The transformation is used to establish equivalence for networks with three terminals. Where three elements terminate at a common node and none are sources, the node is eliminated by transforming the impedances. For equivalence, the impedance between any pair of terminals must be the same for both networks. The equations given here are valid for complex as well as real impedances.

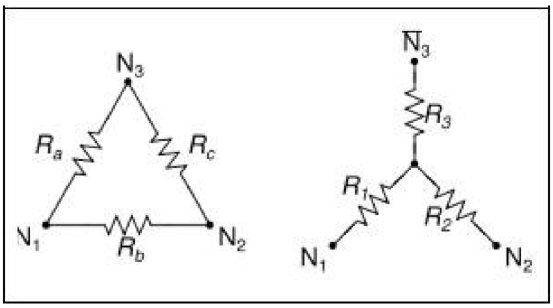

Figure 5.1: Question 2

5.2 Δ to Y Transformation

The general idea is to compute the impedance Ry at a terminal node of the Y circuit with impedances R′, R″ to adjacent nodes in the Δ circuit by

$$R_y = R'R''/\sum R_\Delta$$

where R_Δ are all impedances in the Δ circuit. This yields the specific formulae

$$R_1 = R_aR_b/R_a + R_b + R_c$$
$$R_2 = R_bR_c/R_a + R_b + R_c$$
$$R_3 = R_aR_c/R_a + R_b + R_c$$

5.3 Y to Δ Transformation

The general idea is to compute the impedance R_y at a terminal node of the Y circuit with impedances R′, R″ to adjacent nodes in the Δ circuit by

$$R_\Delta = R_p'/R_o$$

where $R_P = R_1R_2 + R_2R_3 + R_3R_1$ is the sum of the products of all pairs of impedances in the Y circuit and R_o is the impedance of the node in the Y circuit which is opposite the edge with R_Δ. The formula for the individual edges are thus

$$R_a = (R_1R_2 + R_2R_3 + R_3R_1)/R_2$$
$$R_b = (R_1R_2 + R_2R_3 + R_3R_1)/R_3$$
$$R_c = (R_1R_2 + R_2R_3 + R_3R_1)/R_2$$

5.4 Activities and Exercise

- Calculate the total resistance of the given circuit and also calculate the loop currents.
- Make connection as shown in given diagram.
- Measure total resistance and loop currents.
- Write all the values in the given table.

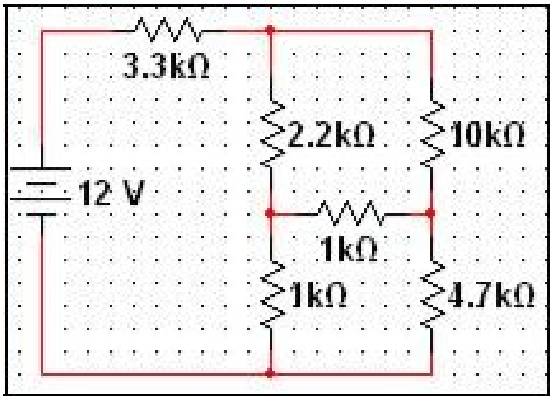

Figure 5.2: Exercise 1

Table 5.1: Add caption

Parameter	RT	I1	I2	I3
Theoretical				
Experimental				
% Error				

- Calculate the total resistance of the given circuit and also calculate all the voltage drops across the each resistor.
- Make connection as shown in given diagram.
- Measure total resistance and loop currents.
- Write all the values in the given table.

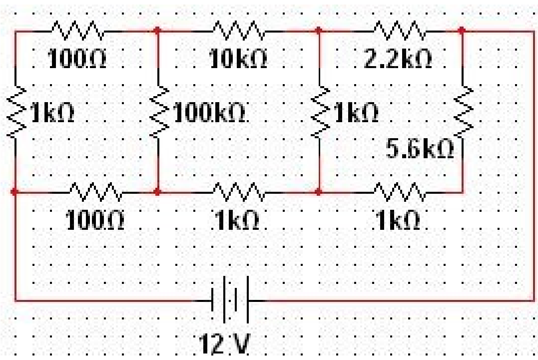

Figure 5.3: Exercise 2

Table 5.2: Add caption

Parameter	RT	I1	I2	I3
Theoretical				
Experimental				
% Error				

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 5.3: Lab Rubric

Chapter 6

Lab 6: Simulating Op-Amps in Multisim

6.1 Objectives

The purpose of this lab is to introduce the different features of MultiSim. Begin by first opening up MultiSim. For Windows users the default location can be found by clicking:

$Start- > AllPrograms- > ElectronicsWorkbench- > DesignSuiteFreewareEdition9- > MultiSim9$

You should see a screen similar to figure 6.1. This is called as a âĂIJCapture and SimulateâĂİ environment because you âĂIJCaptureâĂİ your schematic by drawing it in MultiSim and then you âĂIJSimulateâĂİ it. Following figure shows the different parts of the MultiSim workspace. Note that the location of the toolbars on your MultiSim window may be different.

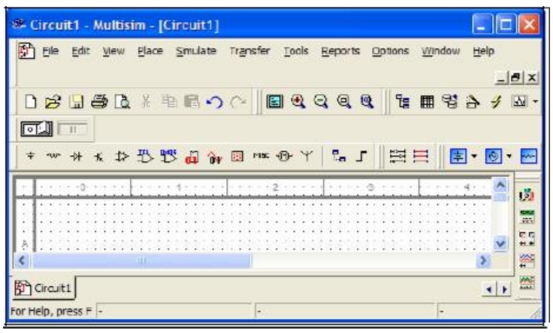

Figure 6.1: âĂIJCapture and SimulateâĂİ environment

6.2 Basic Elements

The MultiSim user interface consists of the following basic elements

- Menus are where you find commands for all functions.

- The system toolbar contains buttons for commonly performed functions.
- The instruments toolbar contains buttons for each instrument available with your edition of MultiSim.
- The link to Edaparts.com launches your Web browser loaded with the Electronic Workbench Edaparts.com website. You can then navigate the site to download parts.
- The zoom toolbar allows you to zoom in and out on the active circuit.
- The MultiSim design bar is an integral part of MultiSim, and is explained in more detail below.
- The in use list lists all the components used in the active circuit, for easy re-use.
- The component toolbar contains Parts Bin buttons that let you open Parts Bins (which, in turn, contain buttons for each family of components).
- The circuit window is where you build your circuit designs.
- The status line displays useful information about the current operation and a description of the item the cursor is currently pointing to.
- The simulate switch is an easy way to run/stop/pause simulation of your circuit.

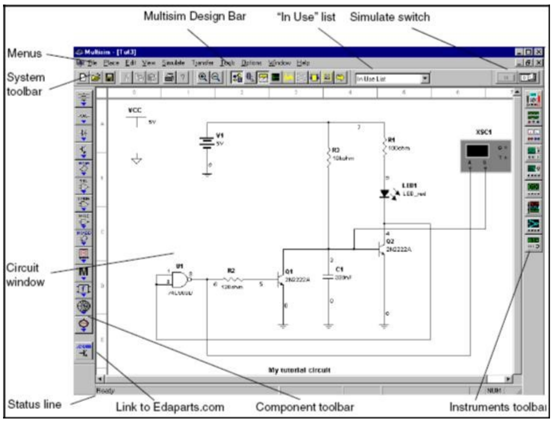

Figure 6.2: Basic Elements in MultiSim

6.3 Design Bar

The Design Bar is a central component of MultiSim, allowing you easy access to the sophisticated functions offered by the program. The Design Bar guides you through the logical steps of building, simulating, analyzing and, eventually, exporting your design. Although Design Bar functions are available from conventional menus, this manual assumes you are taking advantage of the ease of use offered by the Design Bar. This manual explains the use of those Design Bar buttons that are necessary to create and simulate the circuit described in the manual.

	The Component Design button is selected by default, since the first logical activity is to place components on the circuit window.
	The Component Editing button lets you modify the components in MultiSim, or add components.
	The Instrument Design button is selected by default and lets you attach instruments to your circuit.
	The Simulate button runs/pauses/stops the simulation. The green sine wave line moves while simulation is running.
	The Analysis button lets you choose the analysis you want to perform on your circuit.
	The Postprocessor button lets you perform further operations on the results of your simulation.
	The VHDL/Verilog HDL button allows you to work with VHDL modeling (not available in all editions). For details on the functions associated with this button, see Chapter 10, "HDLs and Programmable Logic".
	The Reports button lets you print reports about your circuits (list of components, component details, instruments). For details on the functions associated with this button, see Chapter 11, "Reports".
	The Transfer button lets you communicate with and export to PCB layout programs, such as Ultiboard, also from Electronics Workbench. You can also export simulation results to programs such as MathCAD and Excel. For details on the functions associated with this button.

Figure 6.3: Design Bar Button

6.4 Eaxample: OPAMP

- Start the MultiSim Electronic Workbench program.
- You will create a new schematic and simulation so choose$File- > SaveAs$, navigate to or create a directory where you can save this schematic and simulation then fill in the filename as shown below. (I called this one tut1.) Click SAVE when you have navigated to the proper directory and entered a name for the project.

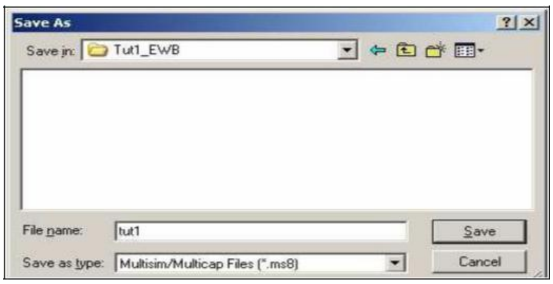

Figure 6.4: Example1

- For reference, a picture of the circuit we will build and simulate appears below:

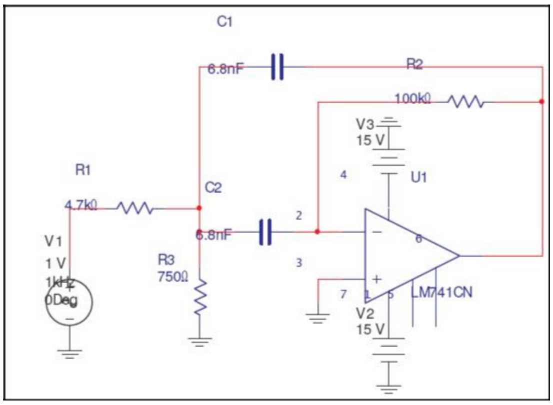

Figure 6.5: Example1

- You should now have a blank schematic. Start placing components by selecting $Place- > Component$ from the menu bar. You will see another dialog box as shown. Start with the OpAmp: pick Analog Components in the Group drop down menu:

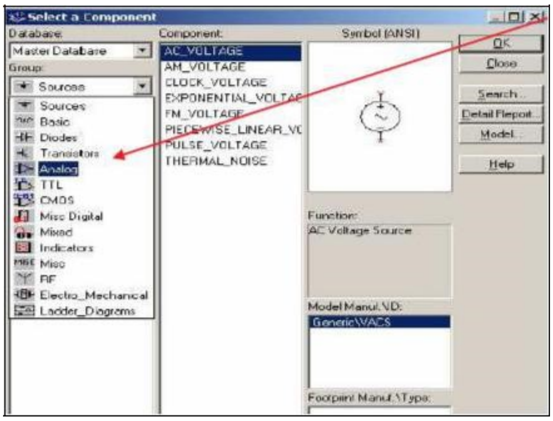

Figure 6.6: Example1

- You should see a new dialog box. Select OPAMP then LM741CN as shown below then click OK. (Some versions of the 741 do not simulate well. This one does.)
- Place the OpAmp on the schematic and flip it vertically to get the minus terminal on top.To flip the OpAmp, make sure it is selected (surrounded by a dashed blue line).Then right click once on the OpAmp and choose Flip Vertical from the pull down menu as shown:
- Next add a couple of power supply connections. You will add a DC source by again selecting $Place- > Component$. Only this time pick SOURCES from the drop down menu.Then select $POWER_SOURCES$ and DC_POWER as shown below then click OK.

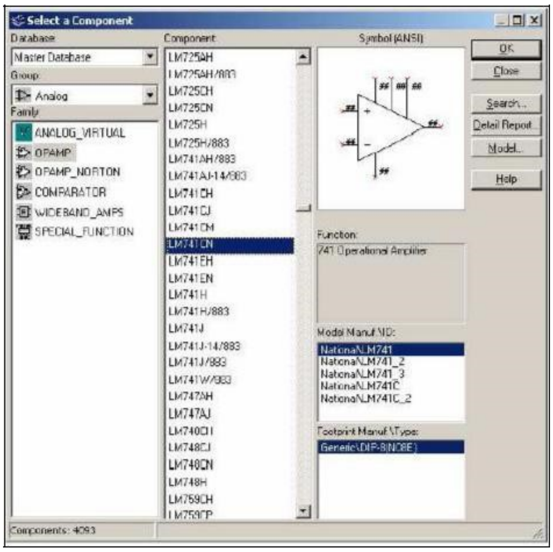

Figure 6.7: Example1

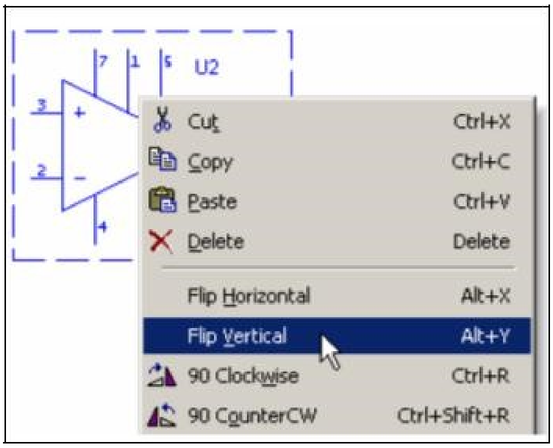

Figure 6.8: Example1

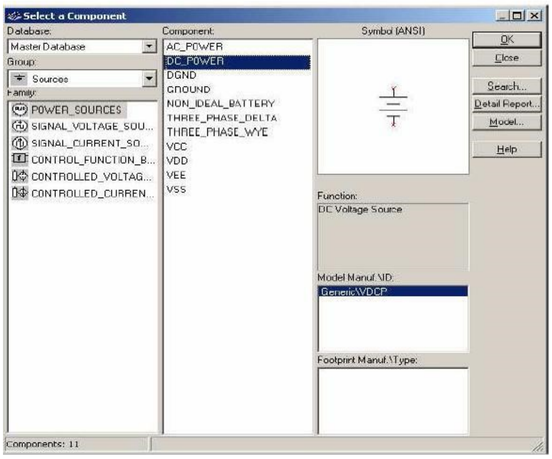

Figure 6.9: Example1

- Put the battery (DC power source) on the schematic somewhat below the OpAmp. Align the top battery terminal with the power input (pin 7) on the OpAmp and raise the battery until it just makes a connection as shown. Then add a second battery for the negative supply above the OpAmp. Double click on the âĂIJ12VâĂİ and change it to 15V for each battery. Pick up a GROUND symbol (also in Source) and put one at the bottom of the positive battery. Then flip one and put it at the top of the negative battery. Your circuit should now look like:

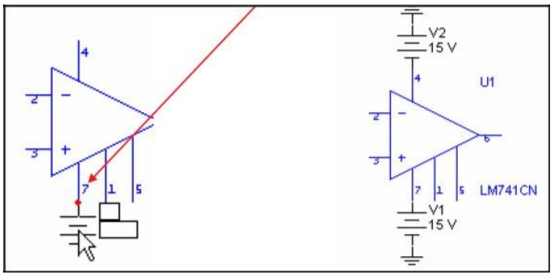

Figure 6.10: Example1

- Place the rest of the components on the schematic as shown below. Resistors and capacitors are under the Basic components drop down menu. The source V1 is under Sources->Signal Voltage. Your populated schematic should look like:

- Wiring circuits in MultiSim is easy. Simply place the cursor over the terminal you wish to wire, left click once and start moving the mouse. Everywhere you want to âĂIJpinâĂİ the wire, left click once. When you reach the place where you want to terminate the wire, left-click once if it is on a terminal or twice if it is not connected to anything. See the figures below:

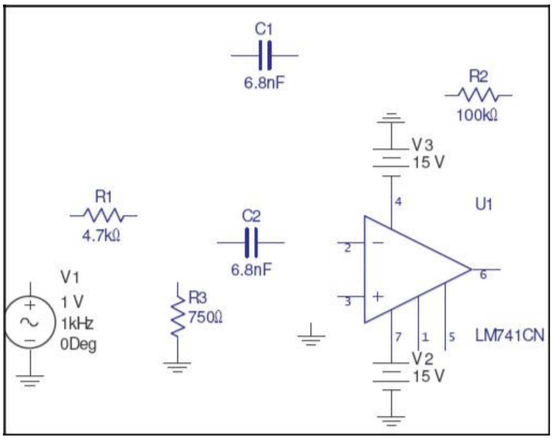

Figure 6.11: Example1

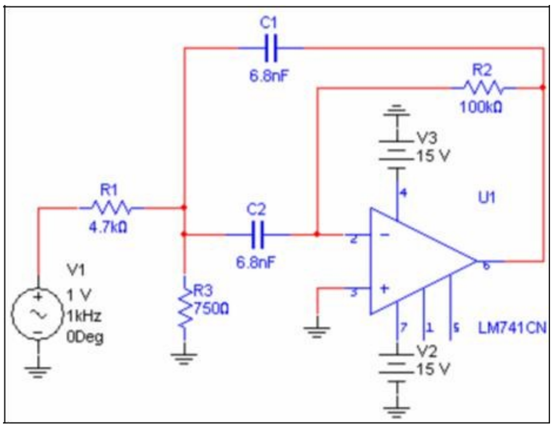

Figure 6.12: Example1

- Make sure all connections are made as shown above. Then you have one last thing before simulation. You will give a name to the output node so you can easily find it for simulations. Left click once on the red wire going up from the output of U1.
- Now you are ready to run some simulations. MultiSim is quite powerful âĂŞ you can add virtual instruments to your circuit and run simulations that way. However we are going to be a little more sophisticated and let MultiSim produce Bode plots for us.In order to do that, we have to setup what we want on the analysis. Note our 1V, 1kHz input voltage source. MultiSim can sweep it to produce Bode plots automatically.
- You will now run a simulation. First select$Simulate- > Analyses- > ACAnalysis$ from the top menu bar. Make the Frequency Parameters page look as shown. (Be sure to change the vertical scale to decibels for a Bode plot.)

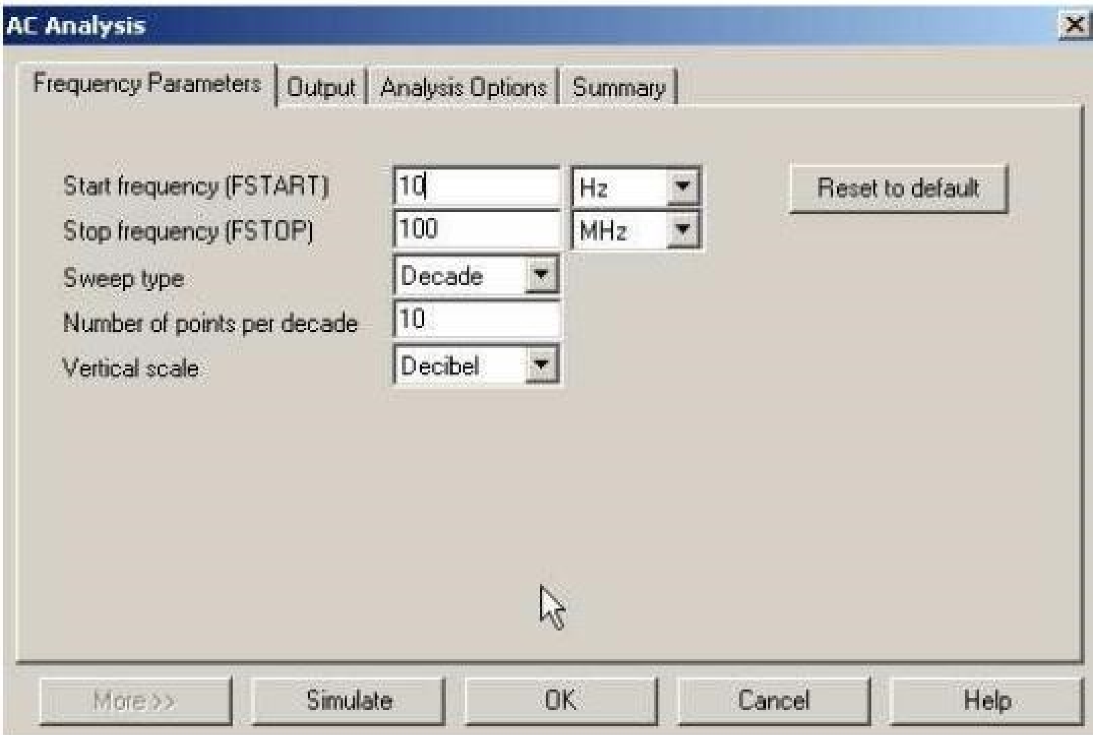

Figure 6.13: Example1

- You should get a resulting Bode plot that looks like the one below.

6.5 Activities and Exercise

- Simulate the Inverting Amplifier circuit and calculate the output voltage and compare the result with simulation result?

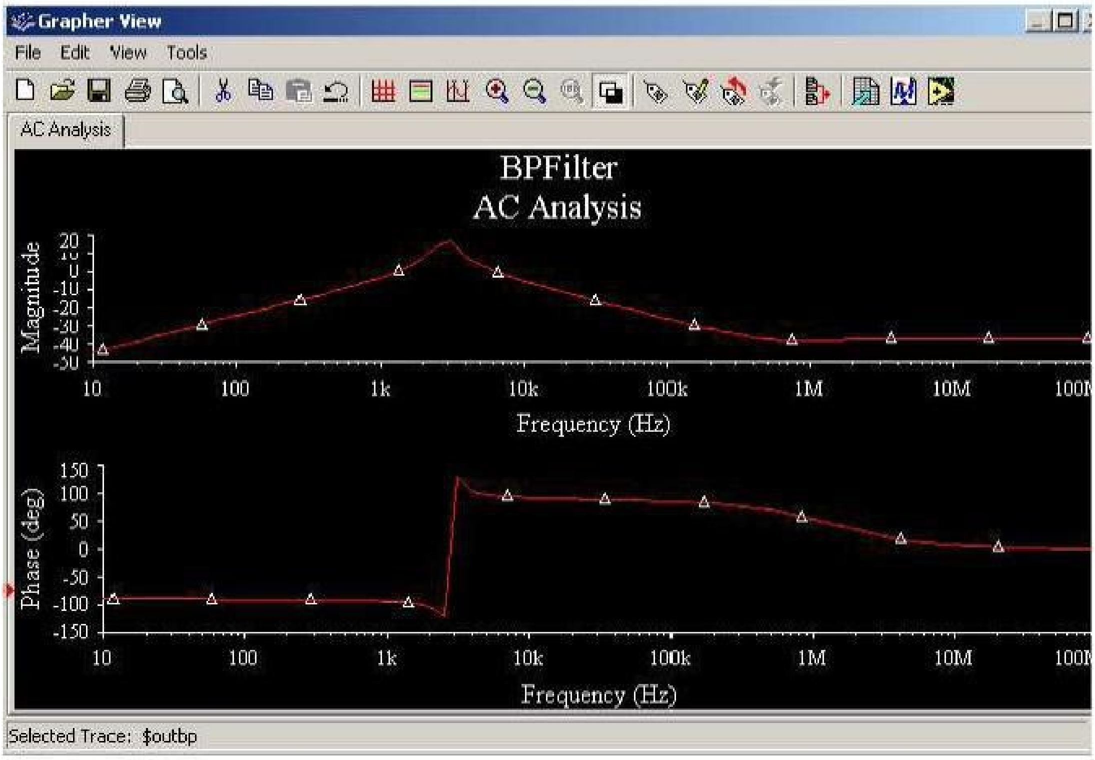

Figure 6.14: Example1

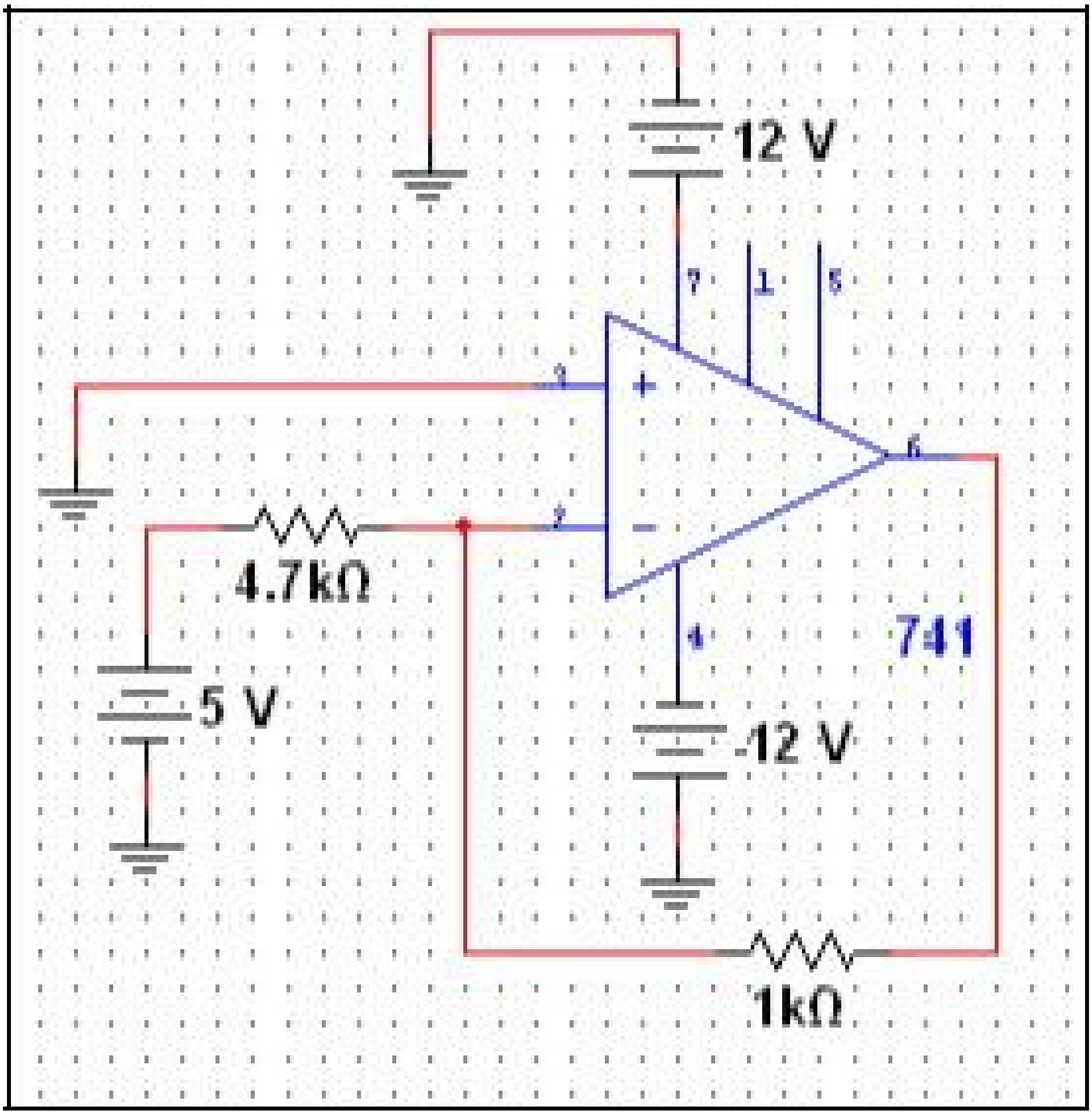

Figure 6.15: Exercise 1

- Simulate the Non-inverting Amplifier and calculate the output voltage and compare the result with simulation result?

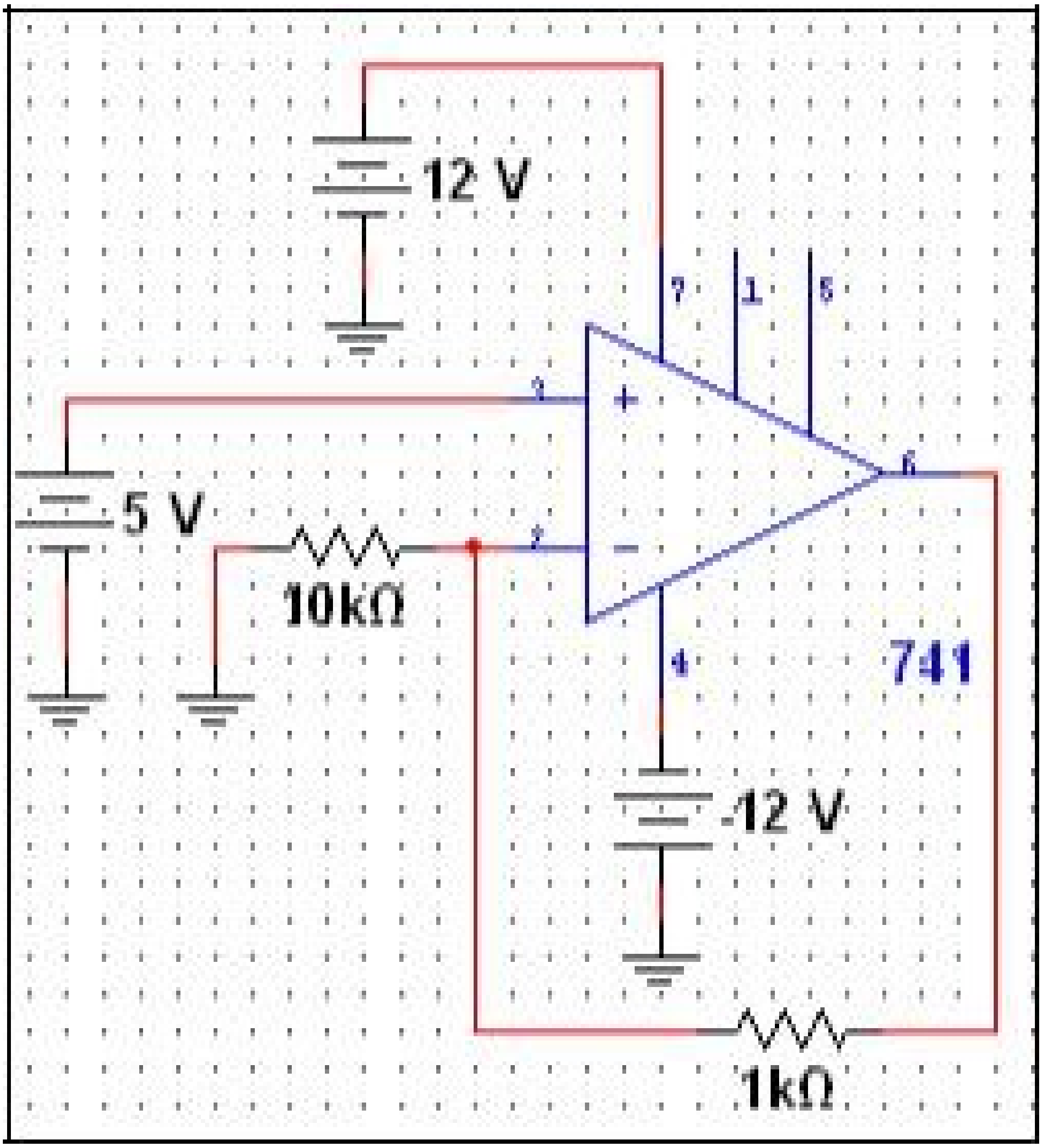

Figure 6.16: Exercise2

6.6 Summary

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 6.1: Lab Rubric

Chapter 7

Lab 7 : Operaational Amplifier

7.1 Objectives

This experiment will provide exposure to operational amplifiers. We will be simulating and building some basic op-Amp configuration.

7.2 Introduction

Operational Amplifiers have become basic building blocks in electronic circuit design because of their reliability, compact nature, and relative ease of use. When used properly the overall transfer characteristic can be precisely controlled by stable passive components such as resistors, capacitor and diodes. Using feedback techniques, suppression of any non-ideal performance characteristic can be readily achieved, resulting in circuit with highly predictable performance.

7.3 Theory

Be sure to read over the appropriate appendices in your lab manual so you remember how to use the breadboard, the equipment and how to read component values of the components themselves, as well as how to use the op-amp. The amplifier we will be using in this experiment is μ741. We will be simulating four basic configuration using op-amps.

7.3.1 Inverting Amplifier

- An inverting amplifier is shown in figure 7.1. The principal features of this circuit are that the output voltage is the negative of the input voltage and is amplified by a gain that depends only on the feedback resistance between the source and the input terminal.
- Be sure to note that the positive terminal of the operational amplifier is grounded.
- The gain of this amplifier is given by,

$G = -R_F/R_i$

7.3.2 Non-Inerting Amplifier

- A non-inverting amplifier is shown in figure 7.2. The input is connected to the positive terminal. The output has the same polarity as the input, but is again multiplied by a gain that is determined by the feedback resistance and the resistance between the negative terminal and ground.
- The input resistance for this configuration is usually very large.
- The gain of this amplifier is given by:
  $G = (R_F + R_i)/R_i$

7.3.3 Differential Amplifier

- A differential amplifier is shown in figure 7.3. This amplifier is a combination of the two latter configurations.
- The output voltage is an amplified version of the difference between the two inputs notes that changing the resistor values in this configuration we can have an amplifier which amplifies with different gains to each of its inputs and then subtract one input from the other.
- The gain for this particular set of values is given by,

  $G = -R_F/R_i(V_1 - V_2)$

7.3.4 Summing Amplifier

- The summing amplifier is shown in figure 7.4 and is characterized by a set of N parallel input voltages, each in series with a resistor of a specific value.
- The currents produced by each source are summed at node A.
- The output voltage is the negative of the sum of the input voltages, each multiplied by the ratio of the feedback resistance to the resistor value in series with, the respective voltage source,
- By varying the resistance that is in series with each source, almost any set of weighting factors can be achieved in the summation; In essence, the output voltage is a linear combination of the input voltages.
- The gain for each input is given by gain of inverting amplifier, where we take R_i replaceR_L instead. R_n is the resistance in series with n^th source.

7.4 Activities and Exercise

Assemble each of the four amplifier configurations you simulated in the preparation part and verify that each configuration operates as described in that section and in your simulations.

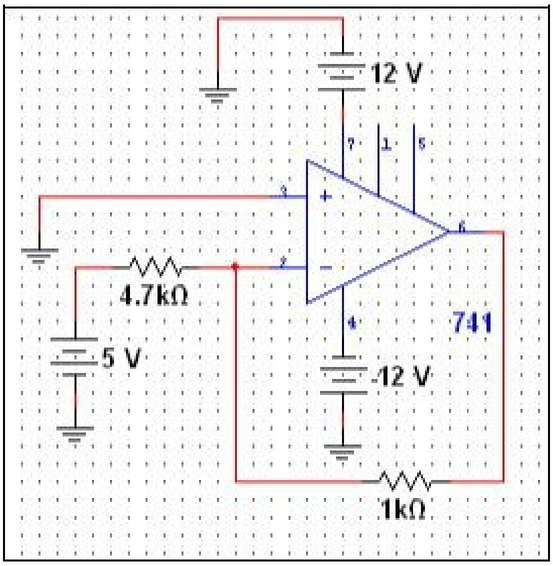

Figure 7.1: âĂIJExercise 1

Table 7.1: Exercise 1

	VRi	VRf	Vi	V0	Av
Theoretical					
Experimental					
% Error					

Table 7.2: Exercise 1

	VRi	VRf	Vi	V0	Av
Theoretical					
Experimental					
% Error					

7.5 Summary

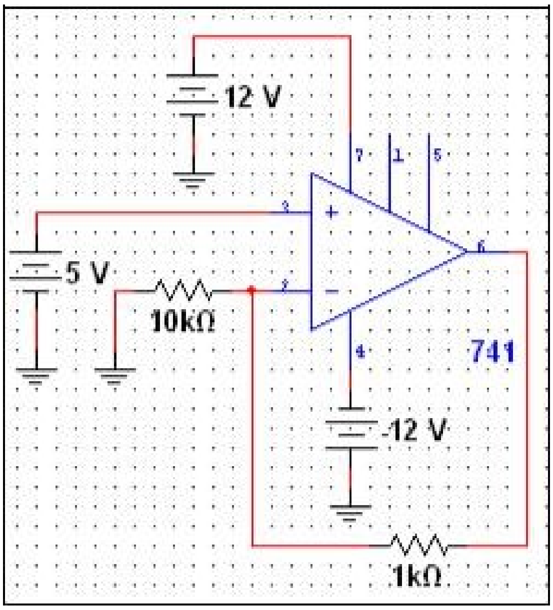

Figure 7.2: âĂIJExercise 1

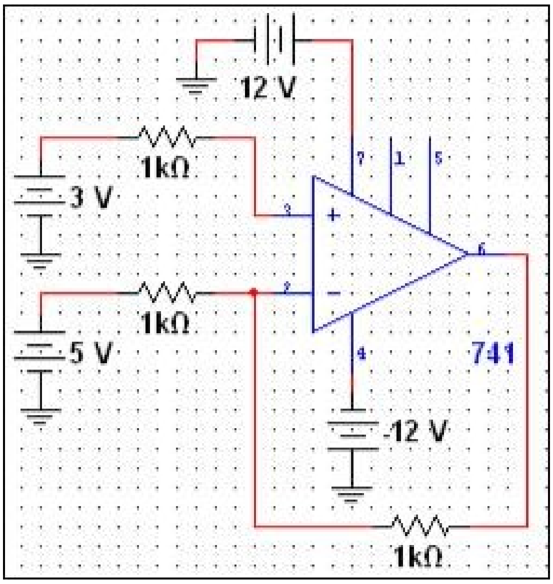

Figure 7.3: âĂIJExercise 1

Table 7.3: Exercise 1

	VRi	VRf	Vi	V0	Av
Theoretical					
Experimental					
% Error					

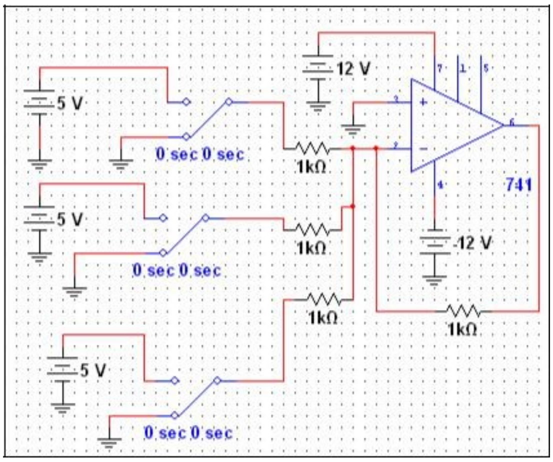

Figure 7.4: âĂIJExercise 1

Table 7.4: Add caption

S.No.	Vi1	Vi2	Vi3	Vo(Theoratical)	Vo(Eperimental)	% Error	Av
1	0	0	0				
2	0	0	1				
3	0	1	0				
4	0	1	1				
5	1	0	0				
6	1	0	1				
7	1	1	0				
8	1	1	1				

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 7.5: Lab Rubric

Chapter 8

Lab 8: Circuit Analysis Techniques âĂŞ I

8.1 Objectives

The objective of this experiment is to analyze DC resistive circuits employing the node-voltage method, the mesh current method. Experimental results will allow the verification of the theoretical analysis.

8.2 Techniques in Circuit Analysis

Some circuit configurations cannot be solved by reduction according to series/parallel circuit rules mentioned in the previous lab, due to multiple unknown values. Mathematical techniques to solve for multiple unknowns (called simultaneous equations or systems) can be applied to basic laws of circuits (introduced in previous lab) to solve such complex circuits. A brief description of each method is presented below:

8.2.1 Node-Voltage Method

- In circuit analysis, nodal analysis also known as node-voltage method, is a method of determining the unknown voltage between nodes (points where circuit elements or branches connect) in an electric circuit in terms of the branch currents using current law (KCL).
- Select one node in an N-node circuit as the ground reference. The choice does not affect the result and is just a matter of convention. Choosing the node with most connections can simplify the analysis.
- Assign a variable for each node whose voltage is unknown. If the voltage is already known, it is not necessary to assign a variable.
- For each unknown voltage, form an equation based on Kirchoff's current law (KCL). Basically, add together all currents leaving from the node and mark the sum equal to zero.

- Solve the system of simultaneous equations for each unknown voltage.

8.2.2 Mesh-Current Method

- The loop analysis, also known as the mesh-current method, uses Kirchhoff's Voltage Law (KVL) to determine unknown currents in an electric circuit.
- One loop current is assigned to each independent loop in a circuit that contains N independent loops.
- Write N independent KVL equations for each loop expressing the voltage across elements in the mesh in terms of loop current.
- Solve the equation system for unknown mesh currents. If any solution is negative, then the assumed current direction is wrong!
- All the branch currents can be calculated from the mesh currents.

8.3 Activities and Exercise

Node-voltage method: Assemble the circuit in Figure with the component values shown in Table. Take measurements to complete the entries corresponding to the experimental values.

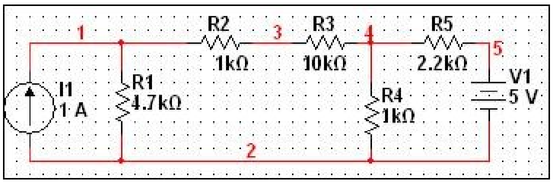

Figure 8.1: âĂIJCapture and SimulateâĂİ environment

Table 8.1: Add caption

Parameter	Theoratical	Experimental	% Error

Mesh-current method:

Assemble the circuit in Figure with the component values shown in Table. Take measurements to complete the entries corresponding to the experimental values.

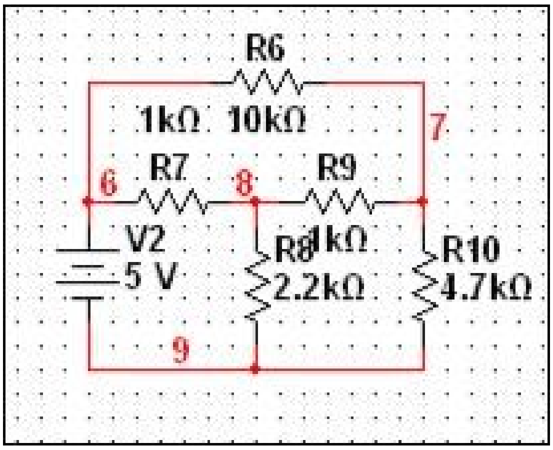

Figure 8.2: âĂIJCapture and SimulateâĂİ environment

Table 8.2: Add caption

Parameter	**Theoretical**	**Experimental**	**% Error**

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 8.3: Lab Rubric

8.4 Summary

Chapter 9

Lab 9:Circuit Analysis Techniques âĂŞ 2

9.1 Objectives

The objective of this experiment is to analyze DC resistive circuits employing the source transformation, Thevenin and Norton theorems. Experimental results will allow the verification of the theoretical analysis.

9.2 TECHNIQUES FOR CIRCUIT SIMPLIFICATION

In this section, source transformations and Thevenin and Norton equivalent circuits are discussed as additional methods to simplify the circuit analysis.

9.2.1 Source transformations

- A source transformation, shown in Fig. 9-1, allows the replacement of a voltage source in series with a resistor by a current source in parallel with the same resistor, or vice versa. In order for these two circuits to be equivalent, the voltage drop and the current drawn by any load at nodes a and b must be the same. The relationship between and is simply:

$I_s = V_s/R$

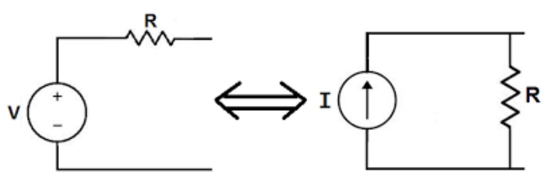

Figure 9.1: Source Transformation

9.3 Thevenin and Norton equivalents

For purpose of analysis, it is often desirable to replace, at a specific pair of terminals (nodes) of a circuit, the set of interconnected elements (resistors and power sources) behind the pair of terminals, by a single resistor and a single power source. Thevenin and Norton equivalents are techniques of circuit simplification that describe the behavior of a circuit at a specific pair of terminals.

9.3.1 Thevenin equivalent

- A Thevenin equivalent circuit is composed of an independent voltage source V_th in series with a resistor R_th, replacing an interconnection of sources and resistors.
- The value of V_th is open circuit voltage at the pair of terminal in the original circuit.
- R_th is given by the ratio of V_th and the short circuit current I_sc observed at the pair of terminals when a short circuit is placed across them. Thus, $R_th = V_th/I_sc$
- Determination of V_th and R_th can be simplified by applying source transformations.

9.3.2 Norton Equivalent

- A Norton equivalent circuit is composed by an independent current source I_N in parallel with a resistorR_N .
- The Norton equivalent can be derived from the Thevenin equivalent simply by making a source transformation.
- Therefore, the Norton current equals the short circuit current at the terminal of interest,$I_N = I_sc$, and the Norton resistance is identical to the Thevenin resistance. $R_N = R_th$
- Notice that source transformation can be applied to calculate I_N and R_th.

9.4 Activities and Exercise

- Source transformations: Assemble the circuit in Figure with the component values shown in Table. Take measurements to complete the entries corresponding, to the experimental values. Replace$R_1 by R_1 and R_3 by R_3 to measure V_o$.
- Thevenin equivalent: Assemble the circuit in Figure with the component values shown in Table. Take measurements to complete the entries corresponding to the experimental values. Connect the DMM in current measurement mode across terminals a and b in order to measure I_sc.

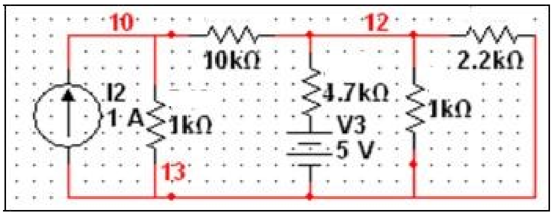

Figure 9.2: Source Transformation

Table 9.1: Add caption

Param	Vs	Is	R1	R2	R3	R4	R5	R1	R3	Param	Unit	Theor	Exper	% Error
Theor										Vo	V			

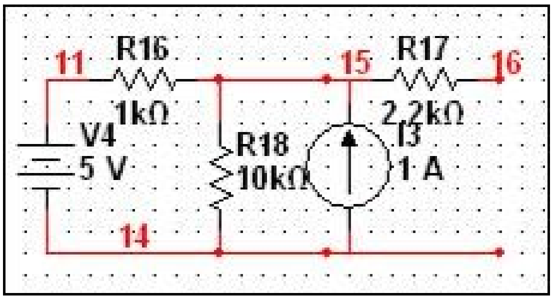

Figure 9.3: Thevenin equivalent

Table 9.2: Add caption

Param	Vs	Is	R1	R2	R3	Param	Theor	Exper	% Error
Values						Vth			
						Rth			

- Norton equivalent: Assemble the circuit in Figure with the component values shown in Table. Connect the DMM in current measurement mode across terminals a and b in order to measure In. Measure V_th in order to determine R_n.

Table 9.3: Add caption

Param	**Vs**	**Is**	**R1**	**R2**	**R3**	**Param**	**Theor**	**Exper**	**% Error**
Values						**Vth**			
						Rth			

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 9.4: Lab Rubric

Chapter 10

Lab 10: DC Power Supply

10.1 Introduction

Power supply is a supply of electrical power. A device or system that supplies electrical or other types of energy to an output load or group of loads is called a power supply unit or PSU. The term is most commonly applied to electrical energy supplies, less often to mechanical ones, and rarely to others. A power supply may include a power distribution system as well as primary or secondary sources of energy such as:

- Conversion of one form of electrical power to another desired form and voltage, typically involving converting AC line voltage to a well-regulated lower-voltage DC for electronic devices. Low voltage, low power DC power supply units are commonly integrated with the devices they supply, such as computers and household electronics; for other examples, see switched-mode power supply, linear regulator, rectifier and inverter (electrical).
- Batteries
- Chemical fuel cells and other forms of energy storage systems
- Solar Power
- Generators or alternators

Constraints that commonly affect power supplies are the amount of power they can supply, how long they can supply it without needing some kind of refueling or recharging, how stable their output voltage or current is under varying load conditions, and whether they provide continuous power or pulses. A regulated power supply or stabilized power supply is one that includes circuitry to tightly control the output voltage and\ or current to a specific value. The specific value is closely maintained despite variations in the load presented to the power supply's output, or any reasonable voltage variation at the power supply's input.

10.2 POWER SUPPLY TYPES

Power supplies for electronic devices can be broadly divided into linear and switching power supplies. (It is customary to refer to a simple unregulated supply as linear, although in fact is has no active electronic devices, either in linear or switched mode.) The linear supply is usually a relatively simple design; it becomes increasingly bulky and heavy for high-current equipment due to the need for large mains-frequency transformers and heat-sinked electronic regulation circuitry. Linear voltage regulation circuitry reduces voltage by dissipating it, making efficiency low. A switched-mode supply of the same rating as a linear supply will be smaller, is usually more efficient, but will be more complex.

10.2.1 Battery power supply

A battery is a type of power supply that is independent of the availability of mains electricity, suitable for portable equipment and use in locations without mains power. A battery consists of several electrochemical cells connected in series to provide the voltage desired. Batteries may be primary (able to supply current when constructed, discarded when drained) or secondary (rechargeable; can be charged, used, and recharged many times) The primary cell first used was the carbon-zinc dry cell. It had a voltage of 1.5 volts; later battery types have been manufactured, when possible, to give the same voltage per cell. Carbon-zinc and related cells are still used, but the alkaline battery delivers more energy per unit weight and is widely used. The most commonly used battery voltages are 1.5 (1 cell) and 9V (6 cells). Various technologies of rechargeable battery are used. Types most commonly used are NiMH, and lithium ion and variants.

10.2.2 Unregulated power supply

An AC powered unregulated power supply usually uses a transformer to convert the voltage from the wall outlet (mains) to a different, nowadays usually lower, voltage. If it is used to produce DC, a rectifier is used to convert alternating voltage to a pulsating direct voltage, followed by a filter, comprising one or more capacitors, resistors, and sometimes inductors, to filter out (smooth) most of the pulsation. A small remaining unwanted alternating voltage component at mains or twice mains power frequency (depending upon whether half or full wave rectification is used) ripple is unavoidably superimposed on the direct output voltage.

For purposes such as charging batteries the ripple is not a problem, and the simplest unregulated mains-powered DC power supply circuit consists of a transformer driving a single diode in series with a resistor. Before the introduction of solid-state electronics equipment used valves (vacuum tubes) which required high voltages; power supplies used step-up transformers, rectifiers, and filters to generate one or more direct voltages of some hundreds of volts, and a low alternating voltage for filaments. Only the most advanced equipment used expensive and bulky regulated power supplies.

10.2.3 Linear regulated power supply

The voltage produced by an unregulated power supply will vary depending on the load and on variations in the AC supply voltage. For critical electronics applications a linear regulator may be used to set the voltage to a precise value, stabilized against fluctuations in input voltage and load. The regulator also greatly reduces the ripple and noise in the output direct current. Linear regulators often provide current limiting, protecting the power supply and attached circuit from overcurrent. Adjustable linear power supplies are common laboratory and service shop test equipment, allowing the output voltage to be adjusted over a range. For example, a bench power supply used by circuit designers may be adjustable up to 30 volts and up to 5 amperes output. Some can be driven by an external signal, for example, for applications requiring a pulsed output.

10.2.4 AC\ DC Supply

In the past, mains electricity was supplied as DC in some regions, AC in others. Transformers cannot be used for DC, but a simple, cheap unregulated power supply could run directly from either AC or DC mains without using a transformer. The power supply consisted of a rectifier and a filter capacitor. When operating from DC the rectifier was essentially a conductor, having no effect; it was included to allow operation from AC or DC without modification.

10.2.5 Programmable power supply

Programmable power supplies allow for remote control of the output voltage through an analog input signal or a computer interface such as RS232 or GPIB. Variable properties include voltage, current, and frequency (for AC output units). These supplies are composed of a processor, voltage/current programming circuits, current shunt, and voltage/current read-back circuits. Additional features can include overcurrent, over-voltage, and short-circuit protection, and temperature compensation. Programmable power supplies also come in a variety of forms including modular, board-mounted, wall-mounted, floor-mounted or bench top.

Programmable power supplies can furnish DC, AC, or AC with a DC offset. The AC output can be either single-phase or three-phase. Single-phase is generally used for low-voltage, while three-phase is more common for high-voltage power supplies. Programmable power supplies are now used in many applications. Some examples include automated equipment testing, crystal growth monitoring, and differential thermal analysis.

10.2.6 Uninterruptible power supply

An uninterruptible power supply (UPS) takes its power from two or more sources simultaneously. It is usually powered directly from the AC mains, while simultaneously charging a storage battery. Should there be a dropout or failure of the mains, the battery instantly takes over so that the load never experiences an interruption. Such a scheme can supply power as long as the battery charge suffices, e.g., in a computer installation, giving the operator sufficient time to effect an orderly system shutdown without loss of data. Other UPS schemes may use an internal combustion engine or turbine to continuously supply power to a system in parallel with power coming from the AC mains. The engine-driven generators would normally be idling, but could come to full power in a matter of a few seconds in order to keep vital equipment running without interruption. Such a scheme might be found in hospitals or telephone central offices.

10.2.7 Voltage multipliers

Voltage multipliers, as the name implies, are circuits designed to multiply the input voltage. The input voltage may be doubled (voltage doubler), tripled (voltage tripler), quadrupled (voltage quadrupler), etc. Voltage multipliers are also power converters. An AC input is converted to a higher DC output. These circuits allow high voltages to be obtained using a much lower voltage AC source. Voltage multipliers, as the name implies, are circuits designed to multiply the input voltage. The input voltage may be doubled (voltage doubler), tripled (voltage tripler), quadrupled (voltage quadrupler), etc. Voltage multipliers are also power converters. An AC input is converted to a higher DC output. These circuits allow high voltages to be obtained using a much lower voltage AC source. Typically, voltage multipliers are composed of half-wave rectifiers, capacitors, and diodes. For example, a voltage tripler consists of three half-wave rectifiers, three capacitors, and three diodes (see Cockroft Walton Multiplier). Full-wave rectifiers may be used in a different configuration to achieve even higher voltages. Also, both parallel and series configurations are available. For parallel multipliers, a higher voltage rating is required at each consecutive multiplication stage, but less capacitance is required. The voltage capability of the capacitor limits the maximum output voltage.

Voltage multipliers have many applications. For example, voltage multipliers can be found in everyday items like televisions and photocopiers. Even more applications can be found in the laboratory, such as cathode ray tubes, oscilloscopes, and photomultiplier tubes.

10.3 Activities and Exercise

5V DC Power Supply

- Make a connection as shown in circuit diagram.
- Connect all the components as given.
- Measure the input and output voltages.
- Also measure the pulsated and pure dc on oscilloscope.

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 10.1: Lab Rubric

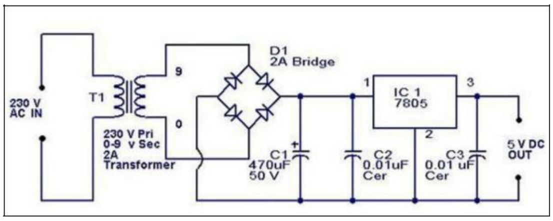

Figure 10.1: DC Power Supply

10.4 Summary

Chapter 11

Lab 11 : Variable Regulated Power Supply

11.1 Introduction

There are many types of power supply. Most are designed to convert high voltage AC mains electricity to a suitable low voltage supply for electronic circuits and other devices. A power supply can by broken down into a series of blocks, each of which performs a particular function. For example, consider a 5V regulated supply: Each

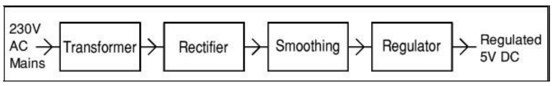

Figure 11.1: Block Diagram of a Regulated power supply

of the blocks is described in more detail below:

- Transformer steps down high voltage AC mains to low voltage AC.
- Rectifier converts AC to DC, but the DC output is varying.
- Smoothing smoothes the DC from varying greatly to a small ripple.
- Regulator eliminates ripple by setting DC output to a fixed voltage.

Power supplies made from these blocks are described below with a circuit diagram and a graph of their output:

11.1.1 Transformer + Rectifier

The varying DC output is suitable for lamps, heaters and standard motors. It is not suitable for electronic circuits unless they include a smoothing capacitor.

11.1.2 Transformer + Rectifier + Smoothing

The smooth DC output has a small ripple. It is suitable for most electronic circuits.

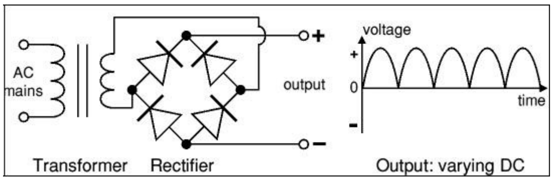

Figure 11.2: Transformer + Rectifier

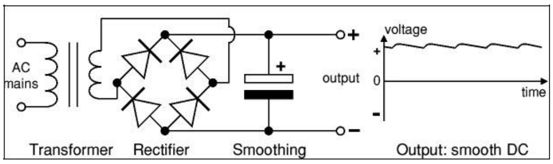

Figure 11.3: Transformer + Rectifier

11.2 Activities and Exercise

- Make connections as shown in circuit diagram.
- Adjust the voltage with varying potentiometer and Regulator.
- Regulate the different voltages.
- Meausre all voltages with DMM.

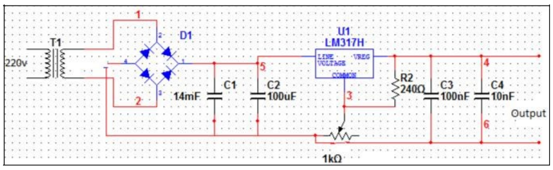

Figure 11.4: Transformer + Rectifier

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 11.1: Lab Rubric

Chapter 12

Lab 12 : First Order RL and RC Circuits

12.1 Introduction

Inductors and capacitors have the ability to store energy. In circuit analysis, we determine the voltages and currents that arise in circuits composed of resistors, and either inductors or capacitors, when energy is released or acquired by the inductor or capacitor as a consequence of an abrupt change in the DC voltage or current in the circuit. The description of the voltages and currents in this type of circuits is done in terms of first order differential equations.

12.2 Natural Response

The currents and voltages that arise when the energy stored in an inductor or capacitor is suddenly released to the resistors in the circuit are referred to as the natural response of the circuit. The behavior of these currents and voltages depends only on the nature of the circuit, and not on external sources of excitation.

12.2.1 Natural response of an RL circuit

- In an RL circuit, the natural response is described in terms of the voltage and current at the terminals of the resistor when the external source of power stops delivering energy to the circuit.

- The expressions for the current and voltage across the resistor are:

  $$i(t) = I_o e^(- t/T) \; V(t) = I_o R e^(- t/T)$$

  Where I_o is the initial current through the inductor before the power source goes off and the inductor starts releasing energy to the circuit. The symbol T represents the time constant of the circuit, which determines the rate at which the current or voltage decays to zero.

$$T = L/R$$

- An RL circuit is shown in Fig. 12-1. Here, provides a square signal with a DC offset voltage such that the bottom part of waveform is aligned with 0 Volts level.

- R_1 is the internal resistance of the voltage source, hence when the square wave takes the value of zero Volts (and can be viewed as a short circuit) the energy of the inductor L is released through the combination of R_1andR_2.

- In order to observe the natural response of the circuit, the period T of the square wave must be long enough to allow complete charge and discharge of the inductor.

  Usually $T = 20T$ is appropriate for this purpose.

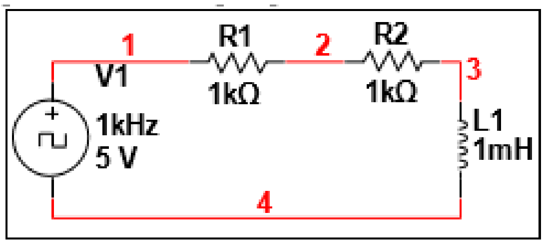

Figure 12.1: RL Circuit

12.2.2 Natural response of an RC circuit

- The natural response of an RC circuit is analogous to that of an RL circuit.

- An RC circuit is shown in figure 12.2

- The expressions for the current and voltage across the resistor are:

$$v(t) = V_o e^(- t/T)\ i(t) = V_o/Re^(- t/T)$$

- Where V_o the initial voltage is across the (fully charged) capacitor before the power source is switched off and the capacitor starts releasing energy to the circuit.

- The time constant for RC circuit is given by:

$$T = RC$$

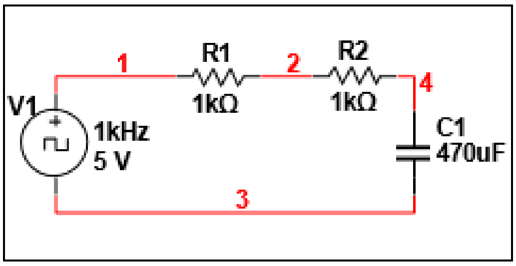

Figure 12.2: RC Circuit

12.3 Activities and Exercise

12.3.1 Natural Response of RC circuit

- Set the frequency of the function generator to about 0.05/T Hz.
- The oscilloscope probe across the capacitor. Press Autoscale.
- Press Main/Delayed and select Lft under Time Ref from the soft menu in the oscilloscope screen.
- Press Slope/Coupling and select the second slope in the soft menu (going from high to low).
- Adjust the Time/Div knob to the closest value to T.
- The amount of time per division is shown on top of the oscilloscope display. For example, 20.0 ms / means the time span displayed in the screen is (20 ms) x 10 equal to 200 ms.
- At this point the natural response of the RC circuit should be displayed.
- Record your results and show them to your instructor.

Table 12.1: Exercise1

Parameters	Theoretical	Measured	% Error
T			
Vt			
It			

12.3.2 Natural response of RL circuit

- Repeat above procedure (with necessary modifications) to obtain natural response for RL circuit.
- Record your results and show them to your instructor.

Table 12.2: Exercise2

Parameters	Theoretical	Measured	% Error
T			
Vt			
It			

12.4 Step Response

The response of a circuit to the sudden application of a constant voltage or current source is referred to as the step response of the circuit. This case .presents the opposite conditions of the natural response. Now, in RL or RC circuits, the inductor or capacitor (assumed to be completely discharged) begins acquiring energy after a sudden application of an external power source. The voltages and currents that arise in the circuit under these conditions are discussed next.

12.4.1 Step response of an RL circuit

- In an RL circuit the initial conditions to determine the step response are assumed to be $I_o = 0$
- The expressions for the current in the circuit and the voltage across the inductor after the voltage source is applied are:

$$v(t) = V_s e^(- t/T)\ i(t) = (V_s/R)(1 - e^(- t/T))$$

- Notice that the current increase from zero to a find value of V_s/R at a rate determined by the time constant $T = L/R$

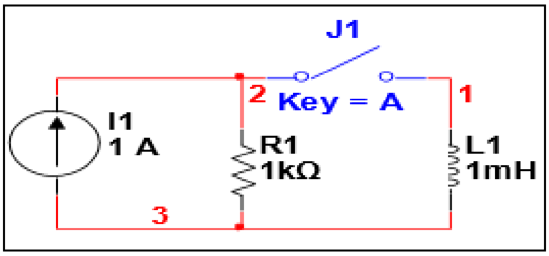

Figure 12.3: RL Step Response

Table 12.3: Exercise 3

Parameters	Theoretical	Measured	% Error
T			
Vt			
It			

12.4.2 Step response of an RC circuit

- In an RC circuit (see Fig; 12-4) the initial voltage across the capacitor is assumed to be $V_o = 0$
- The expressions for the current and voltage in the capacitor after the current source is applied are given below:

$$i(t) = I_o e^(- t/T) \; i(t) = (I_o R)(1 - e^(- t/T))$$

- Notice that Eq. indicates that the voltage increases from zero to a final value of $I_O R$ at a rate determined by the time constant $T = RC$.

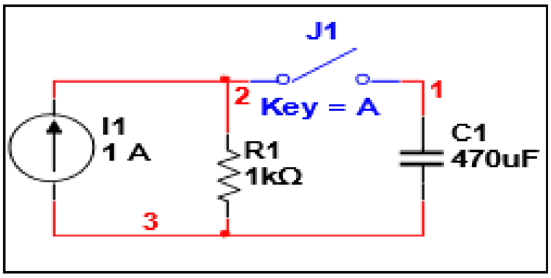

Figure 12.4: RC Step Response

Table 12.4: Exercise4

Parameters	Theoretical	Measured	% Error
T			
Vt			
It			

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 12.5: Lab Rubric

Chapter 13

Lab 12 : First Order RL and RC Circuits

13.1 Introduction

Natural responses of RLC circuits are generated by the release of energy by inductor or capacitor or both as a consequence of an abrupt change in DC voltage or current in the circuit. Similarly, the step response of RLC circuits is generated when the inductor or capacitor or both acquire energy after a sudden application of DC voltage or current to the circuit. The description of the voltages and currents in this type of circuits is done in terms of differential equations of second order. For example, by applying Kirchhoff's current law to the parallel RLC circuit, the following expression is found in terms of the voltage v:

$$\frac{d^2v}{dt^2} + \frac{1dv}{RCdt} + \frac{v}{LC} \tag{13.1}$$

Similarly, for the series RLC circuit, application of Kirchhoffs voltage law and some mathematical manipulation yield, in terms of the current i:

$$\frac{d^2i}{dt^2} + \frac{Rdi}{Cdt} + \frac{i}{LC} = 0 \tag{13.2}$$

These are second-order ordinary differential equations whose general solution can be found from the roots of the characteristic equation:

$$s^2 + 2\alpha s + \omega^2 = 0 \tag{13.3}$$

Where α is known as the neper frequency (in radians/sec). For parallel RLC circuits, its value is given by $\alpha = 1/2RC$. And for series RLC circuit, it is given by $\alpha = R/2L.\omega^2$ is the resonant radian frequency (in rad/s), given by $\omega = 1/sqrtLC$ The values of $\alpha and \omega$ determine the form of the natural (or step) response of a second order parallel (or series) RLC circuit. The form of the response can be categorized as:

- Over-damped: The response is over-damped when $\alpha > \omega^2$. In this case the natural (or step) response approaches its final value without oscillation.

- Under-damped: In this case $\alpha < \omega^2$ and response oscillates about its final value.
- Critically-damped: In this case $\alpha = \omega^2$, and the response is on the verge of oscillating about its final value.

13.2 NATURAL RESPONSE

- Determining the natural response of a second-order circuit consists in finding the type of damping and applying appropriate set of equations.
- Notice that in order to determine the coefficient values, a set of two linear equations needs to be solved.
- Here,x(0) and dx/dt(0) represent the initial values of the variable representing the response (voltage or current) and its derivative evaluated at $t = 0$, respectively.

Damping	Natural Response Equations	Coefficient Equations
Over Damped	$x(t) = A_1e^{s_1t} + A_2e^{s_2t}$	$x(0) = A_1 + A_2$ $\left.\frac{dx}{dt}\right\|_{t=0} = A_1s_1 + A_2s_2$
Under Damped	$x(t) = (B_1cos(\omega_dt) + B_2sin(\omega_dt))e^{-\alpha t}$	$x(0) = B_1$ $\left.\frac{dx}{dt}\right\|_{t=0} = -\alpha B_1 + \omega_dB_2$ $\omega_d = \sqrt{\omega_0^2 - \alpha^2}$
Critically Damped	$x(t) = (D_1t + D_2)e^{-\alpha t}$	$x(0) = D_2$ $\left.\frac{dx}{dt}\right\|_{t=0} = D_1 - \alpha D_2$

Figure 13.1: RC Circuit

13.2.1 Natural response of a parallel RLC circuit

- The determination of the natural response of a parallel RLC circuit consists of finding the voltage V generated across the parallel branches by the release of energy stored in the inductor or capacitor or both.
- The initial conditions of voltage across the capacitor and current through the inductor are represented by $V_0 and I_0$ respectively.
- V_s provides a square signal with a DC offset voltage such that the bottom part of the waveform is aligned with the zero Volts level,R_1 is the internal resistance of the voltage source, andR_2 is the resistance in RLC circuit.
- A test circuit is shown in figure 13.2.
- The natural response of the circuit can be generated in a periodic fashion providing that the period T of the square wave is long enough to allow the charge and discharge of the respective elements.

13.2.2 Natural response of a series RLC circuit

- The determination of the natural response of the series RLC circuit consists of finding the current I generated through the series element by the release of energy store in the inductor or capacitor or both.
- A test circuit is shown in Fig. 13.2.
- The period of the square signal must last long enough to allow the charge and discharge of the respective elements.

13.3 Activities and Exercise

13.3.1 Natural response of a parallel RLC circuit.

- Consider the circuit shown in figure 13.1 and the component values (as provided in lab). Assume that the signal provided by the function generator switches from 10 to 0 Volts at $t = 0$.
- Determine $\alpha and \omega$ the type of damping of this circuit and find the expression for the voltage v(t).
- Generate a graph of v(t) versus time in the interval from 0 to 2 ms on oscilloscope.

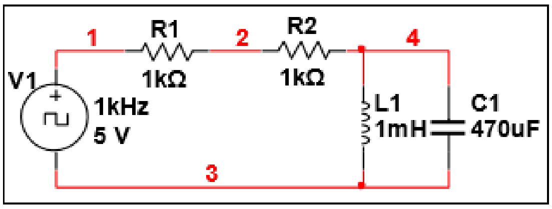

Figure 13.2: Parallel RLC Circuit

13.3.2 Natural response of a series RLC circuit

- Consider the series RLC circuit shown in Fig, 13-3.
- Assume that the signal provided by the function generator switches from 10 to 0 Volts t = 0.
- Determine$\alpha and \omega$ the type of damping of this circuit and find the expression for the current i(t).
- Generate a graph of i(t) versus line in the interval from 0 to 1 ms on the oscilloscope.

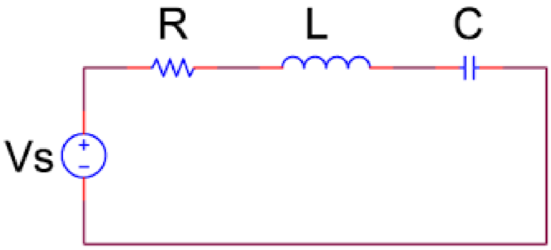

Figure 13.3: Series RLC circuit

Microprocessor Systems Lab Rubric						
	PLOs	Excellent	Good	Satisfactory	Poor	Score
		3.0	2.0	1.0	0	
Apparatus Usage	PLO-5	Can independently setup, operate and handle the apparatus	Can setup and handle the apparatus with minimal help	Can setup and handle the apparatus with some help	Cannot setup or handle the apparatus	
Data Acquisition	PLO-5	All required data is recorded and presented accurately and completely in the required format	Data recorded and presented is complete but accuracy is not as per requirement. The required format is followed.	Data recorded and presented is partially complete. The required format is followed but with few deficiencies.	Data recorded only with considerable assistance from instructor	
Design	PLO-3	Specifications, parameters and constraints of design are completely present	Specifications, parameters and constraints of design are partially present	Some specifications, parameters and constraints of design are present	No specifications, parameters and constraints of design are present	
Data Interpretation	PLO-4	Analyzes and interprets data correctly for all tasks/ experiments in the lab	Analyzes and interprets data correctly for few tasks/ experiments in the lab	Analyzes data correctly however unable to interpret it for tasks/ experiments in the lab	Unable to analyze and interpret data for any tasks/ experiments in the lab	
	Total Score in Lab (Out of 12)					

Table 13.1: Lab Rubric