Faculty of Engineering and Information Sciences
UOWD

ECTE202
Circuits and Systems

Laboratory Notes

UNIVERSITY OF
WOLLONGONG
IN DUBAI

Version: 1.1

UNIVERSITY OF WOLLONGONG IN DUBAI

Document Owner: Nidhal Abdulaziz, SECTE

Author: Abigail Copiaco, Nidhal Abdulaziz

Approved By:

Approval Date:

Document Control:

Version	Date	Author	Reason
1.0	09/10/2019	Abigail Copiaco Nidhal Abdulaziz	First version of the document. Designed all the lab experiments and formatted to the standard template.
1.1	06/09/2020	Abigail Copiaco Nidhal Abdulaziz	Minor updates on experiments based on Autumn 2019 comments

Table of Contents

Introduction

The aim of these laboratories is to provide practical investigation into circuits and systems through Multisim, and MATLAB. The laboratory program consists of an introductory set of exercises and a project with two parts. You have 7 scheduled laboratory classes, however students are **strongly advised to commence work on the project early and make use of the open access laboratory times (subject to approval to attend the campus)**.

Schedule and Format

Laboratory classes are held every week (including week 1) and each lab session is 2 hours.

Laboratory Notebooks, Demonstrator Verification and Progress Marks

When performing experimental work or design work, it is important to keep notes on what is being done at each step of the process. Each student should maintain an A4-sized notebook and use it to record all work completed, analyse results and record comments for the lab tasks. Once you have completed each task, your notebook should be presented to the demonstrator who will sign the laboratory verification sheet at the end of these lab notes. A copy of this verification sheet will be provided in your first class and should be pasted into the front cover of your laboratory notebook. *In the case of online labs, the tutor will keep notes of each students' work and participation in the online lab sessions.*

Three progress marks will be assigned for this lab as indicated in below Table 1.

Table 1. Progress Component and Due Date

Progress Component	Due Date
Lab 1	Week 1
Lab 2	Week 2
Lab 3	Week 3
Lab 4	Week 4
Project 1 Work	Week 5
Project 1 Submission	Week 6
Lab 5	Week 7
Lab 6	Week 8
Lab 7	Week 9
Project 2 Work	Week 10
Project 2 Submission	Week 11

Table 2. Assessment criteria for lab progress

Laboratory Progress (Mark out of 4 for the logbook) Soft copy submission of the logbook for the online lab sessions.				
1: Very few details provided in the log book.	2: Only a few parts complete and described in the logbook. Unable to demonstrate working code.	2.5: Most lab exercises complete but little results or details of methodology (including Matlab code) are included. Code successfully runs.	3: Most lab exercises complete and some details of results and methodology (including Matlab code) are described. Code successfully runs.	4: All lab exercises are complete and all methodology and all results and methodology (including Matlab code) are described. Code successfully runs.

Introduction to Multisim

What is Multisim?

Multisim is an industry-standard software used to simulate, design, and verify analog, digital, and power electronics circuits. The parts of the Multisim software can be seen in Figure 1.
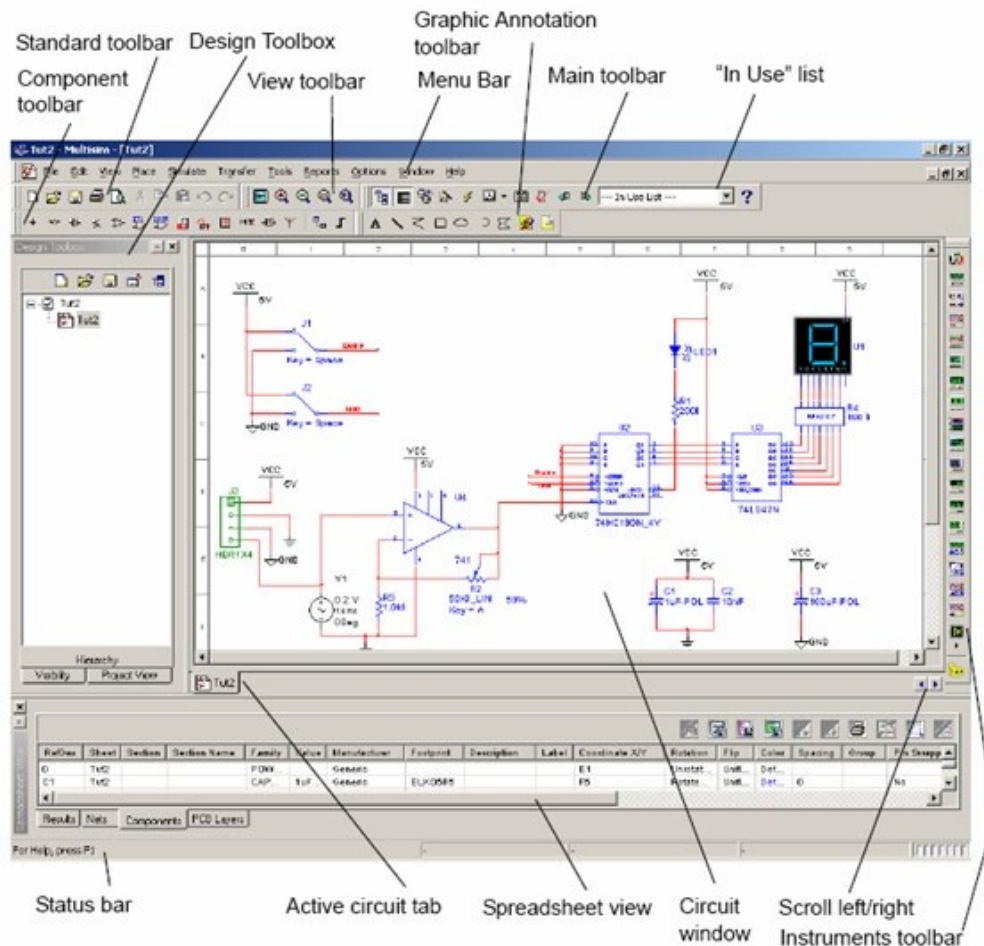
Figure 1. Parts of Multisim [1]

Note: For the labs, we are using Multisim NI version 12.0. All computers in the laboratory are equipped with the same. In case you would like to acquire the software for home-use, kindly coordinate with the lab engineer in order to get the software installed in your laptop

Link to Multisim Installation: *http://www.ni.com/tutorial/14581/en/*

Further instructions for the software activation will be provided soon.

How to use Multisim?

1. Placing Components

To place components onto your circuit design, upon opening the Multisim software, simply go to:

Place -> Component

This should lead you to the following interface, as per Figure 2, where you can search and add components depending on your requirement. Note that components are segregated according to different groups and families. However, in case you are not certain regarding the group of the component you are looking for, simply search for the name of the component, using the *"Search…"* button on the right hand side.
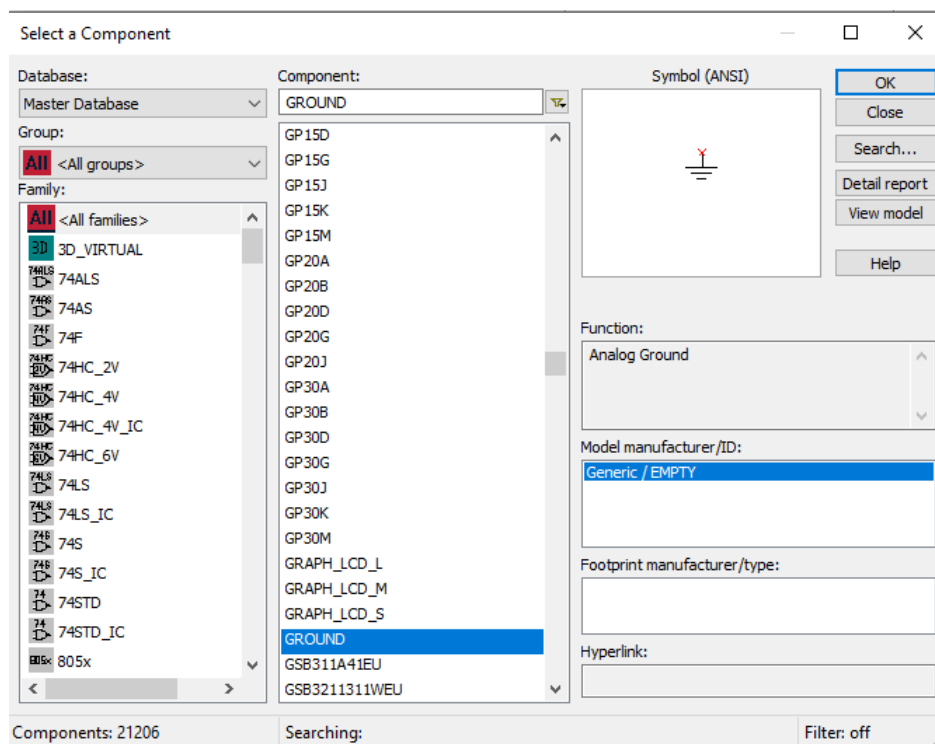
Figure 2. Component Selection Interface

Important Points:

1. To connect components, click on the starting point. Once the line appears, connect it to the destination connection by just clicking on the end point.

2. Once you are done making the connections, run the simulation using the ▶ button.

Note:

For more details regarding NI Multisim, please refer to the software manual, accessible through the following link: http://www.ni.com/pdf/manuals/374483d.pdf

PART A: Multisim Simulation

Lab 1: Fundamental Concepts and Basic Laws

Fundamentals of DC Circuits

(a) Resistors connected in series   (b) Resistors connected in parallel

(a) R = R1+ R2 + R3 + R4

(b) R = (1/R1 + 1/R2 + 1/R3 + 1/R4)^ -1

Kirchhoff's Voltage Law (KVL)

The algebraic sum of all voltages in a loop must equal to zero.

Kirchhoff's Current Law (KCL)

The algebraic sum of all currents entering and exiting a node must equal zero.
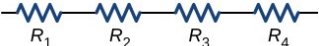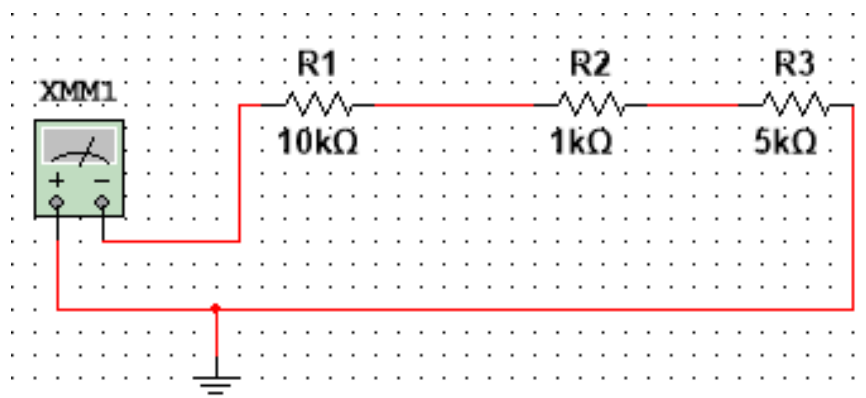
Ohm's Law

$$E = IR \qquad I = \frac{E}{R} \qquad R = \frac{E}{I}$$

Joule's Law

$$P = IE \qquad P = \frac{E^2}{R} \qquad P = I^2 R$$

Where,

$E =$ Voltage in volts
$I =$ Current in amperes (amps)
$R =$ Resistance in ohms
$P =$ Power in watts

Task 1: Simple Circuit in Series

Combining the knowledge you gained about the operation of Multisim with the fundamentals of DC circuits, construct Circuit 1.1, and answer the following questions accordingly in your logbooks.

Circuit 1.1 Resistors in Series

1. Calculate the total resistance manually, and write the result in your logbook.

2. After constructing the circuit on Multisim, double-click on the multimeter, and select the ohmmeter option (indicated by the Ohms notation). What is the value of the total resistance? Does it match your calculation?

3. Replace the multi-meter in Circuit 1.1 with a DC Source of 5V. Following this, place a multi-meter along R1, R2, and R3. Record the voltages and currents flowing through each resistor (fill in the following table in your logbooks). Finally, on the fourth row of the table, measure the total Voltage of the entire circuit. For this, you have to place the multi-meter from R1 to R3.

Note: For measuring voltages, the multimeter has to be parallel to the resistor, but for measuring currents, it has to be in series with the resistor in question.

Table 1.1 Readings for Circuit 1.1

Resistors	Resistance (Ohms)	Voltages (V)	Currents (I)
R1	10 kohm		
R2	1 kohm		
R3	5 kohm		
Total:			

Given that we know the voltage source value, and the total resistance value from the previous calculation, calculate for the total current using Ohm's law. Does this match the total current gathered from your readings in Table 1.1?

4. What can you observe with the individual and total voltage and current readings for series connections? Write down the KVL proof notation.

Task 2: Simple Circuit in Parallel

Combining the knowledge you gained about the operation of Multisim with the fundamentals of DC circuits, construct Circuit 1.2, and answer the following questions accordingly in your logbooks.
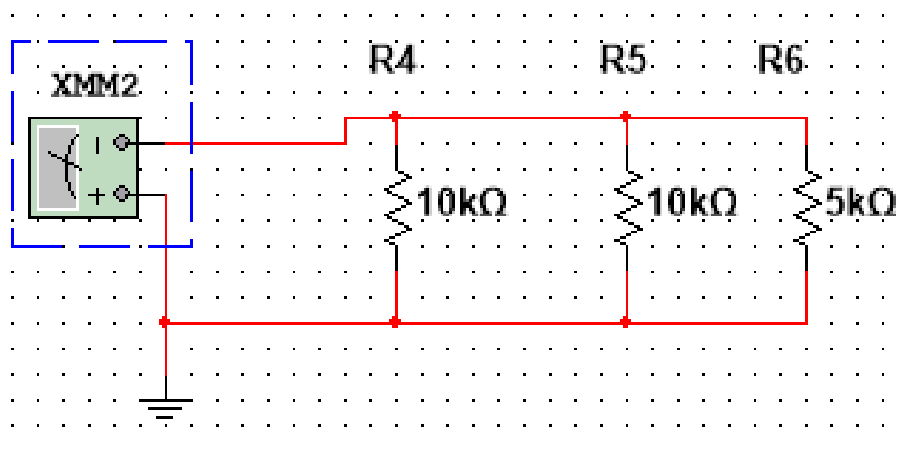
Circuit 1.2 Resistors in Parallel

1. Calculate the total resistance manually, and write the result in your logbook.

2. After constructing the circuit on Multisim, double-click on the multimeter, and select the ohmmeter option (indicated by the Ohms notation). What is the value of the total resistance? Does it match your calculation?

3. Replace the multi-meter in Circuit 1.2 with a DC Source of 5V. Following this, place a multi-meter along R4, R5, and R6. Record the voltages and currents flowing through each resistor (fill in the following table in your logbooks). Finally, on the fourth row of the table, measure the total Voltage of the entire circuit. For this, you have to place the multi-meter from R4 to R6.

Note: For measuring voltages, the multimeter has to be parallel to the resistor, but for measuring currents, it has to be in series with the resistor in question.

Table 1.2 Readings for Circuit 1.2

Resistors	Resistance (Ohms)	Voltages (V)	Currents (I)
R4	10 kohm		
R5	1 kohm		
R6	5 kohm		
Total:			

Given that we know the voltage source value, and the total resistance value from the previous calculation, calculate for the total current using Ohm's law. Does this match the total current gathered from your readings in Table 1.2?

4. What can you observe with the individual and total voltage and current readings for parallel connections? Write down the KCL proof notation.

Task 3: Series and Parallel Combination

Combining the knowledge you gained about the operation of Multisim with the fundamentals of DC circuits, construct Circuit 1.3, and answer the following questions accordingly in your logbooks.
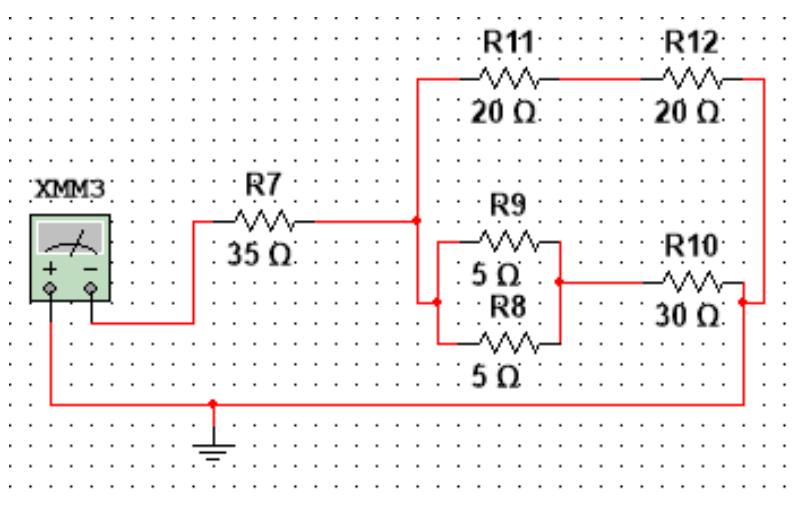
Circuit 1.3 Combination of Series and Parallel Resistors

1. Using your knowledge about series and parallel resistors, calculate the total resistance manually, and write the result in your logbook (along with the computation).

2. After constructing the circuit on Multisim, double-click on the multimeter, and select the ohmmeter option (indicated by the Ohms notation). What is the value of the total resistance? Does it match your calculation?

3. Replace the multi-meter in Circuit 1.3 with a DC Source of 5V. Following this, place a multi-meter along every resistor. Record the voltages and currents flowing through each resistor (fill in the following table in your logbooks). Finally, on the fourth row of the table, measure the total Voltage of the entire circuit. For this, you have to place the multi-meter from R7 to ground.

Note: For measuring voltages, the multimeter has to be parallel to the resistor, but for measuring currents, it has to be in series with the resistor in question.

Table 1.3 Readings for Circuit 1.3

Resistors	Resistance (Ohms)	Voltages (V)
R7	35 ohm	
R8	5 ohm	
R9	5 ohm	
R10	30 ohm	
R11	20 ohm	
R12	20 ohm	
Total:		5 V

4. From the individual voltage readings, prove how the total voltage is 5 V using KCL and KVL theories.

Lab 2: Nodal and Mesh Analysis

Nodal Analysis

- Is a method of determining the voltage (potential difference) between "nodes" (points where elements or branches connect) in an electrical circuit in terms of the branch currents.

The basic procedure for solving **Nodal** Analysis equations is as follows:

1. Write down the current vectors, assuming currents into a node are positive
   Eg. A (N x 1) matrices for "N" independent nodes.
2. Write the admittance matrix [Y] of the network where:

   Y_{11} = the total admittance of the first node.

   Y_{22} = the total admittance of the second node.

   R_{JK} = the total admittance joining node J to node K.

3. For a network with "N" independent nodes, [Y] will be an (N x N) matrix and that Ynn will be positive and Yjk will be negative or zero value.
4. The voltage vector will be (N x L) and will list the "N" voltages to be found.

Read more about the Nodal Analysis in the following link:
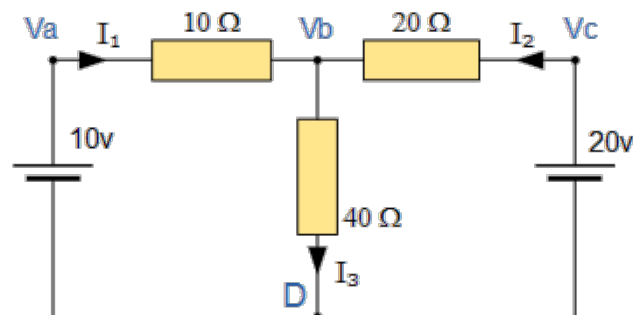https://www.electronics-tutorials.ws/dccircuits/dcp_6.html

Example:

Figure 2.1 Nodal Analysis Example

In the above circuit, node D is chosen as the reference node and the other three nodes are assumed to have voltages, Va, Vb and Vc with respect to node D. For example:

$$I_1 + I_2 = I_3$$

Using the Ohm's law, this can be converted to:

$$\frac{(V_a - V_b)}{10} + \frac{(V_c - V_b)}{20} = \frac{V_b}{40}$$

As Va = 10v and Vc = 20v , Vb can be easily found by:

$$\left(1 - \frac{Vb}{10}\right) + \left(1 - \frac{Vb}{20}\right) = \frac{Vb}{40}$$

$$2 = Vb\left(\frac{1}{40} + \frac{1}{20} + \frac{1}{10}\right)$$

$$Vb = \frac{80}{7} V$$

$$\therefore I_3 = \frac{2}{7} \ or \ 0.286 Amps$$

Mesh Analysis

- a method that is used to solve planar circuits for the currents (and indirectly the voltages) at any placein the electrical circuit. Planar circuits are circuits that can be drawn on a plane surface with no wires crossing each other.

The basic procedure for solving **Mesh** Analysis equations is as follows:

1. Label all the internal loops with circulating currents. (I_1, I_2, …I_L etc)
2. Write the [L x 1] column matrix [V] giving the sum of all voltage sources in each loop.
3. Write the [L x L] matrix, [R] for all the resistances in the circuit as follows:
    R_{11} = the total resistance in the first loop.
    R_{nn} = the total resistance in the Nth loop.
    R_{JK} = the resistance which directly joins loop J to Loop K.
4. Write the matrix or vector equation [V] = [R] x [I] where [I] is the list of currents to be found.

Read more about the Mesh Analysis in the following link:
https://www.electronics-tutorials.ws/dccircuits/dcp_5.html

Example:

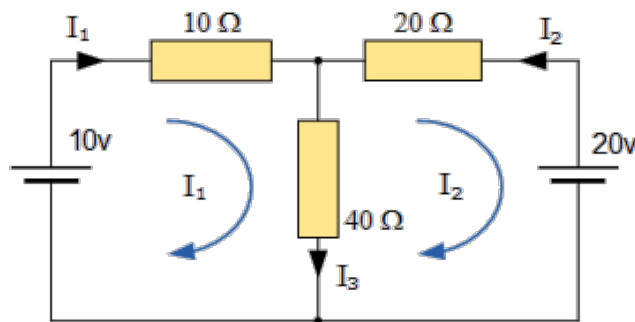Following the same example as we had before, define the three main equations by going around each loop:

Figure 2.2 Mesh Analysis Example

We have:

$$-10 + 10I_1 + 40I_3 = 0$$
$$-20 + 20I_2 + 40I_3 = 0$$
$$I_1 + I_2 = I_3$$

Applying the third equation into the first two equations gives us:

$$10I_1 + 40(I_1 + I_2) = 10$$
$$20I_2 + 40(I_1 + I_2) = 20$$

Simplifying these equations would give us:
$$50I_1 + 40I_2 = 10$$
$$40I_1 + 60I_2 = 20$$

From the second equation, we can derive I_1 to be:

$$I_1 = \frac{20 - 60I_2}{40}$$

Substituting this to the first equation will give us I2 to be 0.4286 A. Plugging this onto the previous equation for I1 would give -0.1429 A.

Thus, $I_3 = I_1 + I_2$ = -0.1429 + 0.4286 **= 0.2857 A** ,

The result is the same as the answer gotten for Nodal analysis.

Dependent Sources

-   a dependent source is a voltage source or a current source whose value depends on a voltage or current elsewhere in the network. Dependent sources are useful, for example, in modelling the behavior of amplifiers.

Task 1: Application of the Nodal Analysis

Consider the following circuit.
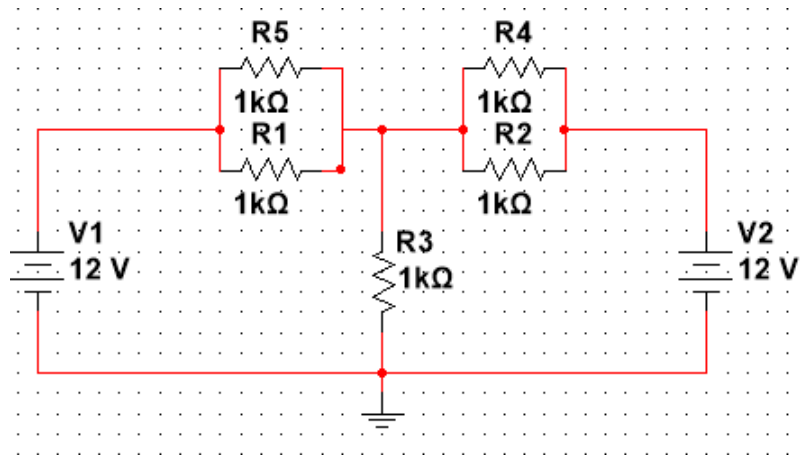
Circuit 2.1 Nodal Analysis Exercise

1. Simplify the circuit using the rules of series and parallel resistors. Draw the simplified circuit in your logbook, along with the resistor values.

2. Construct the simplified circuit in Multisim.

3. Find the values of I1, I2, and V3 (along R3) using probes or the multi-meter. However, probes are more advisable so that we do not have to alter with the circuit connections.

    Note: Probes can be found by clicking on the double arrow at the bottom of the Instruments Toolbar. Go to:
    Measurement Probe -> From dynamic probe settings

    Fill in the following table.

Table 2.1 Nodal Analysis Readings

I1	
I2	
I3	
V3	

4. Looking at the simplified circuit, change the values of the resistors by dividing each resistance by 10 (eg. 1kohm will become 100 ohm). Note down the updated probe readings. Did the values change?

Table 2.2 Updated Nodal Analysis Readings

I1	
I2	
I3	
V3	

5. Using the circuit in point number 4, use Nodal Analysis to find the value of V3 manually. Is it the same as the one founded through Multisim?

Task 2: Application of the Mesh Analysis

Construct the following circuit in Multisim. Find the values of I1, I2, and I3 using probes or the multi-meter. However, probes are more advisable so that we do not have to alter with the circuit connections.

Note: Probes can be found by clicking on the double arrow at the bottom of the Instruments Toolbar. Go to: *Measurement Probe -> From dynamic probe settings*
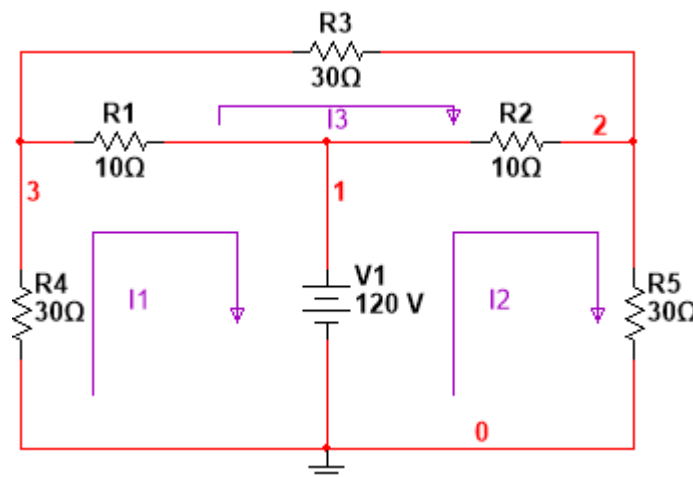
Circuit 2.2 Mesh Analysis Exercise

1. Fill in the following table in your logbook:
2.

Table 2.3 Current Readings

I1	
I2	
I3	

3. Prove your readings by solving the problem via Mesh Analysis. The three starting point equations are provided below. Solve for I1, I2, and I3.

$$120 = 10I_3 - 40I_1$$

$$120 = 40I_2 - 10I_3$$

$$I_3 = \frac{10I_1 + 10I_2}{40}$$

4. Change the value of the resistors by multiplying each by 10. (For example, 30 ohms becomes 300 ohms, and so on.) Note down the updated values of the currents in the following table. What do you observe?
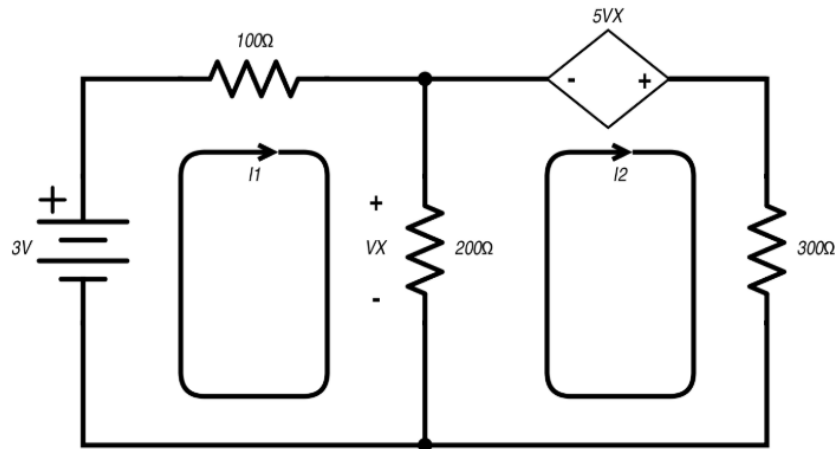
Table 2.4 Current Readings

I1	
I2	
I3	

5. Change the value of the voltage source by multiplying it by 10. What do you notice with the current values?

Task 3: Mesh Analysis with Dependent Sources

Consider the following circuit, with a dependent voltage source.

Circuit 2.3 Mesh Analysis with Dependent Sources

1. Solve the problem via Mesh Analysis. The three starting point equations are provided below. Solve for I1, I2, and Vx.

$$-3 + (100 + 200)I_1 - 200I_2 = 0$$

$$200 (I_2 - I_1) - 5V_x + 300I_2 = 0$$

$$V_x = 200(I_1 - I_2)$$

2. Construct the circuit in Multisim. In Multisim, the dependent voltage source can be represented by a Voltage-controlled Voltage Source. This should be connected normally as it is on the circuit diagram. However, the top voltage control part should be connected to the source by which it is dependent upon. In this case, it should be connected across the 200 ohm resistor (since Vx is across that resistor).

   For more information about how to connect dependent sources on Multisim, consider the following link: https://www.youtube.com/watch?v=UWYUTr5yFko

3. Read I1, I2, and Vx in Multisim through the use of Probes. Verify if they are the same as the results of your calculations.

   Note: Probes can be found by clicking on the double arrow at the bottom of the Instruments Toolbar. Go to: *Measurement Probe -> From dynamic probe settings*
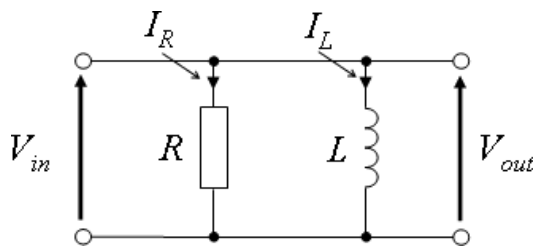
Lab 3: First Order Circuits

What are First Order Circuits?

First order circuits are circuits that contain only one energy storage element, which can either be a capacitor or an inductor. Thus, the voltages and currents of these circuits can be described using only a first order differential equation. The two possible types of first-order circuits are:

1. RC (resistor and capacitor)
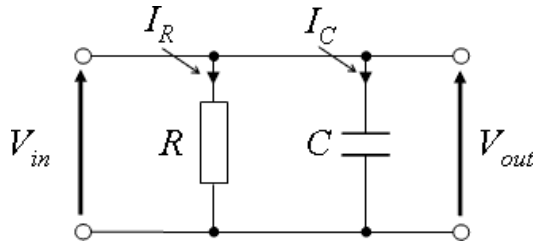
2. RL (resistor and inductor)

First order circuits can be reduced into a less complex, easier to understand circuits via their Thevenin or Norton equivalents.

Thevenin's Theorem

- States that any collection of storage elements and resistances with two terminals can be replaced by a single voltage source and a single series resistor (Thevenin resistance). The value of the voltage source is the open circuit voltage at the terminals, while the value of the Thevenin resistance is the voltage source divided by the current with the terminals short circuited.

The Thevenin voltage e is the open circuit voltage at terminals A and B.

The Thevenin resistance r is the resistance seen at AB with all voltage sources replaced by short circuits and all current sources replaced by open circuits.
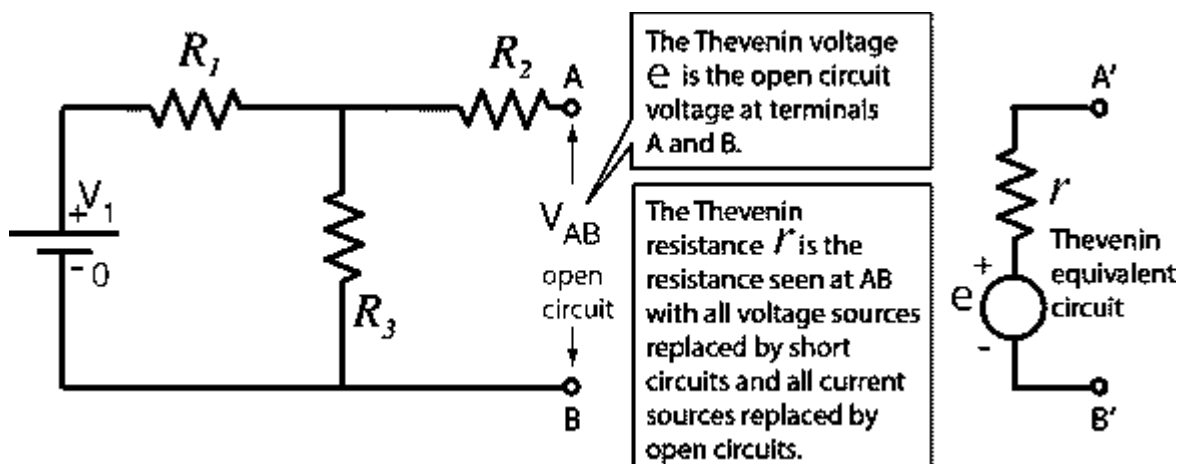
Figure 3.1 Thevenin's Theorem [2]

Norton's Theorem

- States that any collection of storage elements and resistances with two terminals is electrically equivalent to an ideal current source in parallel with a single resistor. The value of the Norton resistance is the same as that of the Thevenin equivalent. The current, on the other hand, can be found by dividing the open circuit voltage by the Norton resistance.
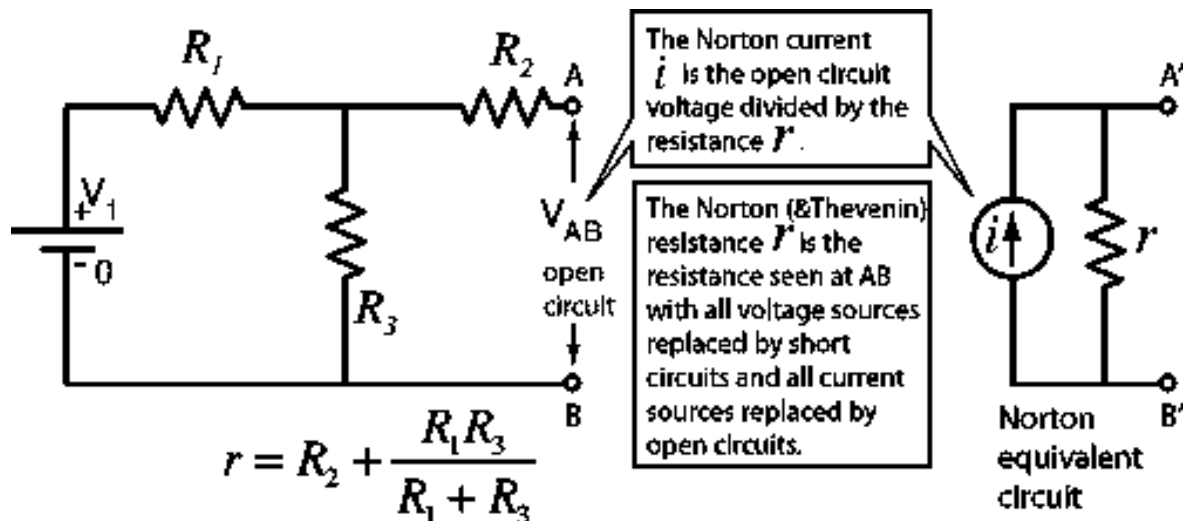

$$r = R_2 + \frac{R_1 R_3}{R_1 + R_3}$$

Figure 3.2 Norton's Theorem [2]

Task 1: The RC Circuit

In this exercise, we will observe the behavior of an RC circuit through transient analysis. (Reference: [3])

1. Download the relevant Multisim file uploaded in Moodle. (Lab3_T1)

2. Upon opening the file in Multisim, you will observe the following circuit.
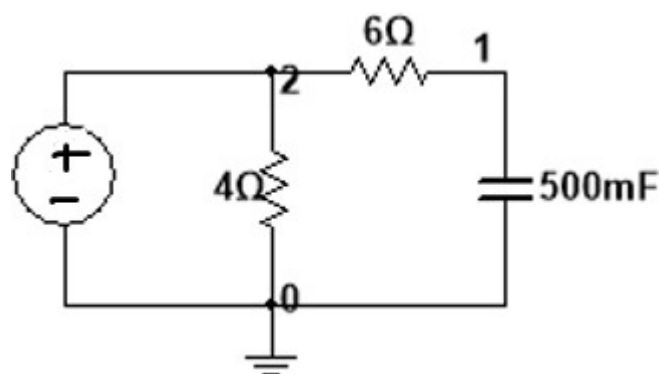
Circuit 3.1 RC Circuit

Problem Statement:
We will observe the behavior of v(t) *(V(1) – across the inductor)*, via a graph extracted through transient analysis.

3. Choose menu "Simulate | Analyses | Transient Analysis". You should get an Analyses Parameters tab, similar to Figure 3.3
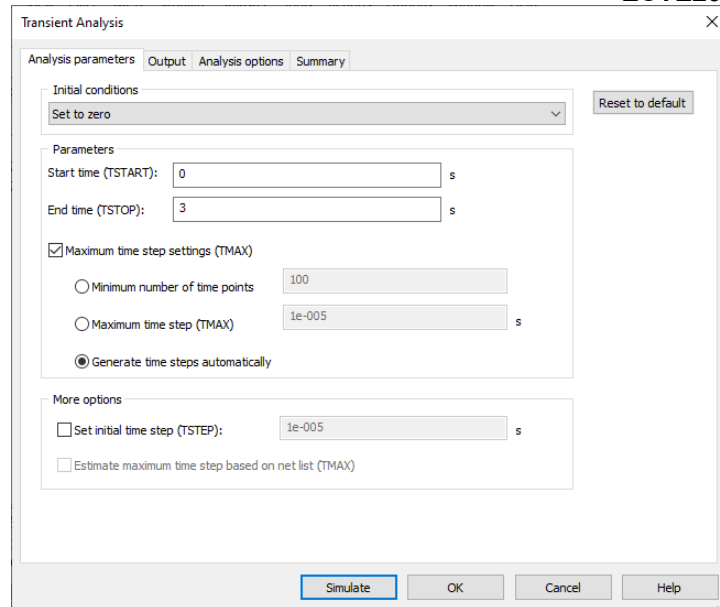
Figure 3.3 Transient Analyses Parameters Tab

4. On the Analyses Parameters tab we need to configure some settings. According to [3], we need to assume *v*(0) = 0.
   o We need to set the Initial Conditions drop down menu to "Set to zero".
   o Set the Parameters Start time (TSTART) and Stop time (TSTOP) to 0 and 3 respectively.

   Note: Setting the stop time to 3 allows us to see the behavior of the circuit after the given signal has been applied and passed.

5. On the Output tab we need to highlight V(1) and select the "Add" button. This adds that variable to the analysis.
6. Run the simulator by pressing the Simulate button.

7. Analyze the graph to determine *v*(t). What type of graph does the plot signify? Draw the graph on your logbook.

8. On the Grapher View, click on Tools -> Export to Excel. You should get the points of Y (the voltage) versus X (time).
   o What is the maximum voltage value? When does it occur?
   o What is the minimum voltage value? When does it occur?

Task 2: The RL Circuit

In this exercise, we will observe the behavior of an RL circuit through transient analysis. (Reference: [3])

1. Construct the following circuit in Multisim.
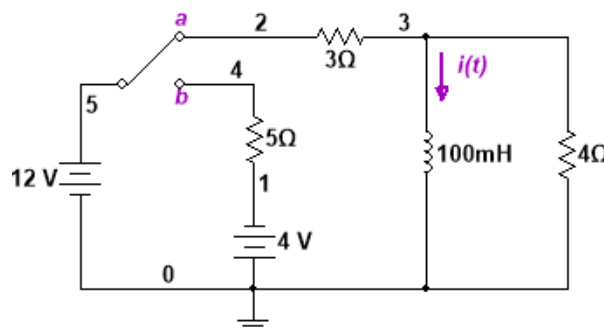
Circuit 3.2 The RL Circuit

Problem Statement:

We will determine i(t) if the switch was set to *a* for a long time and switched instantaneously to *b* at t=0.

2. We need to determine the inital value of i before t=0. To do this place a probe next to the i arrow and be sure switch is in position a.

3. Ensure the probe direction matches the defined current direction (if necessary, right-click on the probe and choose "Reverse Probe Direction").

4. Press the green arrow. Record the value of I in your logbook, and stop the simulation.

5. We now need to transition the switch to position b and initialize the inductor to the current measured in step 6. (right-click the inductor and go to Properties. Under the Value tab check the box next to Initial Conditions and enter the value recorded. Click OK to exit the properties window)

6. From here Transient Analysis will be needed to obtain the graph of current i(t).(choose menu "Simulate | Analyses | Transient Analysis").

7. On the Analyses Parameters tab we need to configure the settings.
    1. We need to set the Initial Conditions drop down menu to "User Defined".
    2. The stop time (TSTOP) set to 1.

8. On the Output tab we need to highlight I(Probe1) and select the "Add" button. This adds that variable to the analysis.

9. Run the simulator by pressing the Simulate button.

10. Analyze the graph to determine i(t). Draw the graph in your logbook. What does the graph suggest?

11. Go to Tools -> Export to Excel. What time does the graph stabilize? (From the excel points, select the time before the current drops from 0).

12. What type of graph does the current signify?

Lab 4: Second Order Circuits

What are Second Order Circuits?

Circuits that include an inductor, capacitor, and resistor connected in series or in parallel are second-order circuits. To describe their voltages and currents, one must use second-order differentiation. An example of a second-order circuit can be seen in Figure 4.1.
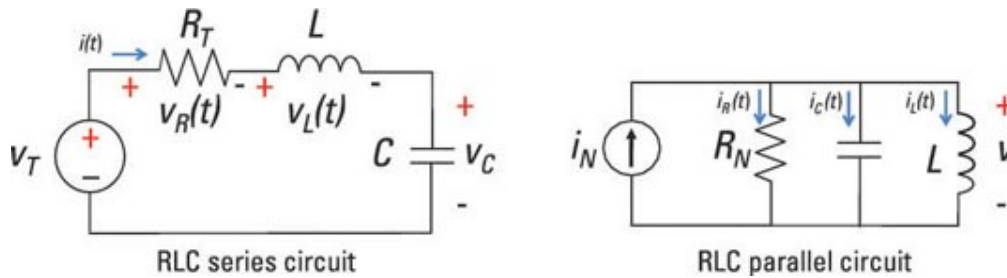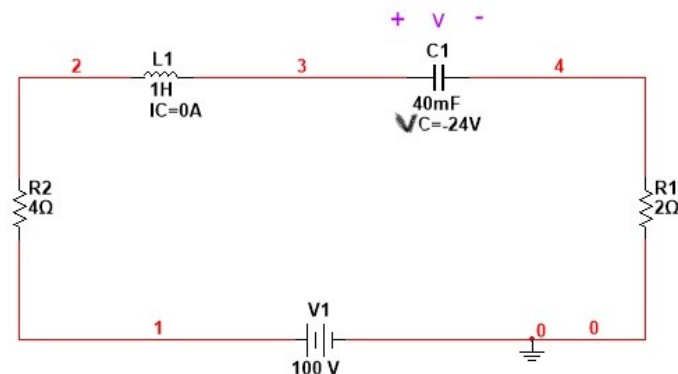

Figure 4.1 RLC circuits in (a) Series, and (b) Parallel

Task 1: RLC Series Circuit

In this exercise, we will observe the behavior of a series-connected RLC circuit through transient analysis. (Reference: [3])

1. Download the relevant Multisim file uploaded in Moodle. (Lab4_T1)
2. Upon opening the file in Multisim, you will observe the following circuit.

Circuit 4.1 Series-connected RLC Circuit

Problem Statement:
Because of previous charging from a current source, the capacitor now has an initial voltage of -24V. Use Multisim to find V for 0 < t < 10s.

3. Open the Transient Analysis Simulator
    (choose menu "Simulate | Analysis | Transient Analysis...")

4. Under the "Analysis Parameters" tab set "Initial Conditions" to "User-Defined"

5. Under the "Analysis Parameters" tab within the "Parameters" section set:
    *   "Start time (TSTART)" to 0.
    *   "End time (TSTOP)" to 10.

6. Under the "Output" select "Add Expression".

7. Under "Variables" select V(3) and click "Copy Variable to Expression"

8. Then under "Functions" select "-" and click "Copy Function to Expression"

9. Then under "Variables" select V(4) and click "Copy Variable to Expression"

10. Click "OK"

11. Select "Simulate" at the bottom of the "Transient Analysis" window to run the simulation.

12. Draw the Transient Analysis on your Logbook. Export the data to excel, and answer the following questions:

- At what value (in Volts) did the voltage stabilized at?
- At what time did this stabilization occur?
- How many samples was taken?
- Observe the graph and comment on it.
- Include calculations for the damping. (Is it critically damped, etc.)?

Task 2: RLC Parallel Circuit

In this exercise, we will observe the behavior of a parallel-connected RLC circuit through transient analysis. (Reference: [3])

1. Download the relevant Multisim file uploaded in Moodle. (Lab4_T2)
2. Upon opening the file in Multisim, you will observe the following circuit.

Circuit 4.2 Parallel-connected RLC Circuit

Problem Statement:
Given circuit as shown, find Vl and Vc. Assume that the capacitor voltage and inductor current at t = 0 are both zero.
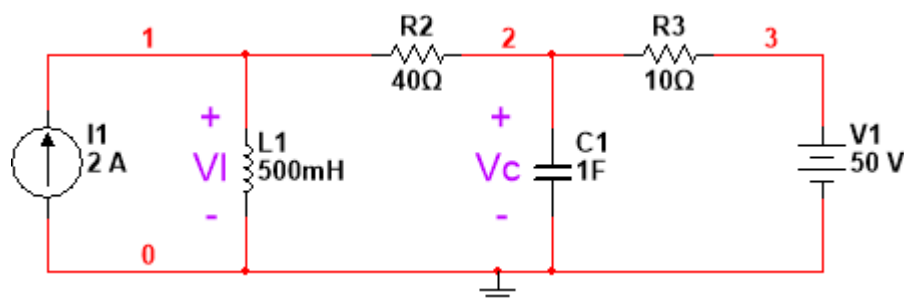
3. Open the Transient Analysis Simulator (choose menu "Simulate | Analysis | Transient Analysis...")

4. Under the "Analysis Parameters" tab set "Initial Conditions" to "User-Defined"

5. Under the "Analysis Parameters" tab within the "Parameters" section:
   - Set "Start time (TSTART)" to 0.
   - Set "End time (TSTOP)" to 0.1.

6. Under the "Output" tab verify that the Variable "V1" and "V2" are selected for analysis.

7. Select "Simulate" at the bottom of the "Transient Analysis" window to run the simulation. Draw the transient analysis plots in your logbook.

8. With the time base from 0 to 0.1s you will be able to see the voltage spike in the inductor (V1). What is the value reached by the voltage spike?

9. What type of graph does the inductor have (V(1))? How about the capacitor (V(2))?

10. Repeat steps 5-7 with a time base of 0 to 50s to see the voltage increase in the capacitor. Draw the graph in your logbook. What is the steady state value of the capacitor's voltage? What type of graph does it show for V(2)? How about for V(1)?

11. Include calculations for the damping. (Is it critically damped, etc.)?

Task 3: RLC Series-Parallel Circuit

In this exercise, we will observe the behavior of a combination of series and parallel-connected RLC circuit through transient analysis. (Reference: [3])

1. Construct the following circuit in Multisim.
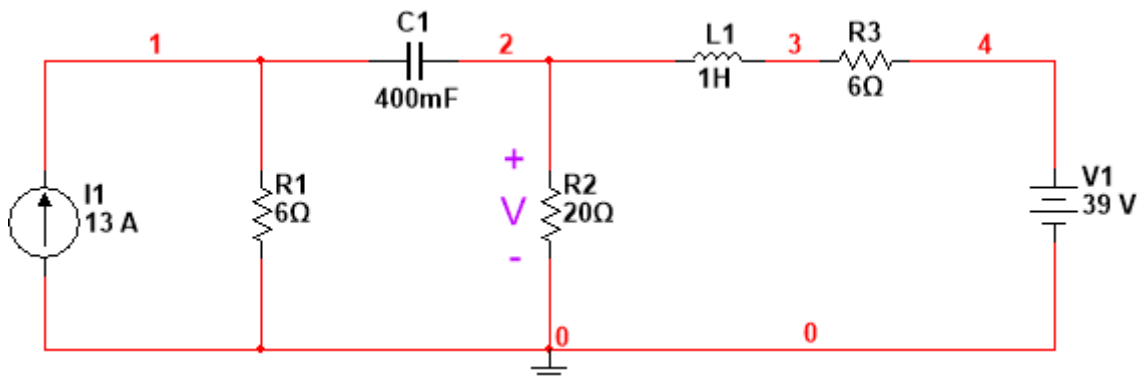
Circuit 4.3 Series-Parallel RLC Circuit

Problem Statement:

Given circuit as shown, find v(t) for 0 < t < 4 s. Assume that the capacitor voltage and inductor current at t = 0 are both zero.

2. Open the Transient Analysis Simulator (choose menu "Simulate | Analysis | Transient Analysis...")

3. Under the "Analysis Parameters" tab set "Initial Conditions" to "User-Defined"

4. Under the "Analysis Parameters" tab within the "Parameters" section set "Start time (TSTART)" to 0.

5. Under the "Analysis Parameters" tab within the "Parameters" section set "End time (TSTOP)" to 4.

6. Under the "Output" tab verify that the Variable "V2" is selected for analysis.

7. Select "Simulate" at the bottom of the "Transient Analysis" window to run the simulation. Draw the graph on your logbook. What is the initial voltage value? Is it decreasing or increasing?

Introduction to MATLAB

Matlab is an abbreviation of Matrix Laboratory. It is a popular mathematical programming environment and it is used extensively in education as well as in industry. It is a high-level matrix/array language and has a vast collection of computational algorithms. Code developed in Matlab can be converted into C, C++ and other languages.

You can get help on any Matlab function by typing help <function> or you could use the help browser. You can use the Help browser to search and view documentation and demonstrations for MATLAB functions. To open the Help browser, click the Help button in the desktop toolbar, type help browser in the Command Window, or use the Help menu in any tool. There are two panes: The Help Navigator, on the left, for finding information, includes Contents, Index, Search, and Demos tabs. The display pane, on the right, is for viewing documentation and demos.

Start MATLAB and use the help function to learn more about the following:

MATLAB Commands	1. Whos 2. Help 3. Clear 4. Close all 5. Cd 6. Dir pwd
Matrix Declaration	1. simple declaration 2. null matrix matrix 3. with ones matrix
Arithmetic	1. Addition 2. Subtraction 3. Multiplication 4. Division
Matrix Manipulation	1. Addressing of individual element 2. Complete Row and Column addressing 3. Transpose 4. Saving and Loading data 5. Concept of Function and m-files

MATLAB Installation Link: https://www.mathworks.com/downloads/

Activation Key: MyUOWD → Notices → search for Matlab

PART B: MATLAB Simulation

Lab 5: Transfer Functions and Bode Plots

What is a Transfer Function?

The Transfer function is a mathematical function relating the output or response of a system such as a filter circuit to the input or stimulus. The formula for finding the transfer function is given by:

$$H(s) = \frac{Y(s)}{X(s)}$$

As observed, the transfer function is defined to be the ratio of the output of a system to the input of a system, considering its initial conditions and equilibrium point to be zero. This assumption is relaxed for systems observing transience. The Transfer function can also be represented by the following block diagram (Figure 5.1):
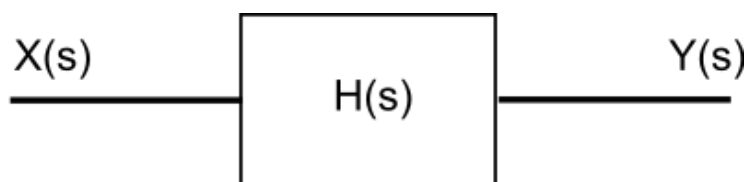

Figure 5.1 Transfer Function Block Diagram

Steps of Calculating the Transfer Function

1. Considering a first or second order circuit, perform the representation of storage elements (capacitors and inductors). Such that:
   For capacitors:
   $$C \rightarrow \frac{1}{sC}$$
   For inductors:
   $$L \rightarrow sL$$
2. Use KVL and KCL to extract the circuit equation/s.
3. Simplify the equations until you get it in the following form:
$$H(s) = \frac{Y(s)}{X(s)} = \frac{V_o}{V_i}$$

Bode Plots

- Are graphical ways of representing the transfer function in the frequency domain.
- A Bode plot is a plot of the frequency response of a system. It shows the magnitude and phase of a transfer function or other complex-valued quantity, versus frequency.
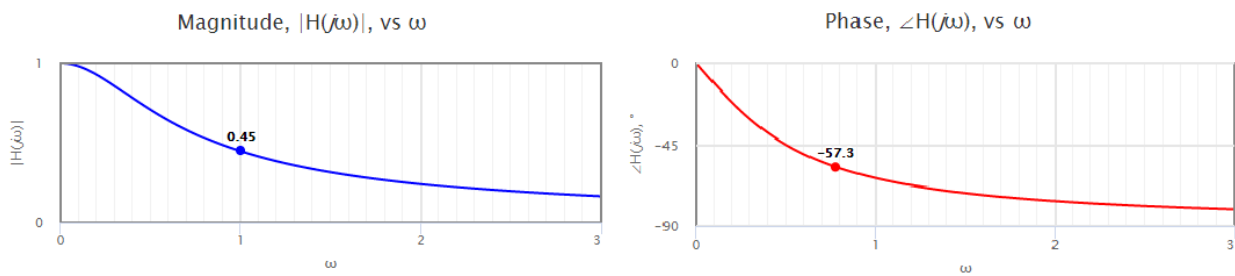- An example of a bode plot can be seen in Figure 5.2

Figure 5.2 Bode Plot Examples of (a) Magnitude and (b) Phase [4]

**Refer to the following link to access the log paper for plotting the Bode plots manually:**
https://uowd.box.com/s/ojls9nibtigjb1j94411xj8wphn9q42h

Task 1: Transfer Function of a First Order Circuit

Consider the following RC Circuit in Circuit 5.1, where R = 5 ohms and C = 2 F. Consider its initial conditions and equilibrium point to be zero.
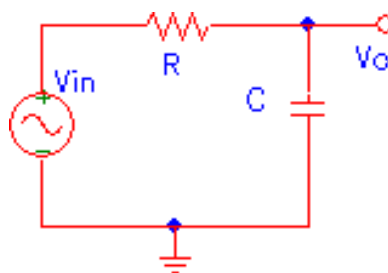
Circuit 5.1 First Order RC Circuit

1. In your logbooks, compute for the transfer function using the steps mentioned earlier as a guide. Note that Vo is along C.

2. Write the coefficients of the transfer function's numerator and denominator in a MATLAB script (using arrays).

3. Use the MATLAB command 'tf' in order to save it into a system. Use the MATLAB help option to get more information about the commands.

4. Plot the system's magnitude and frequency using the bode plot. Use the MATLAB command 'bode' in order to do this. Draw the magnitude and frequency plots roughly in your logbook.

5. With the bode plot figure open, go to Tools, and check the Data Cursor option. What is the magnitude in dB and frequency in rad/s by which the magnitude plot starts to decline / rise?

6. What is the magnitude in dB and frequency in rad/s of the -3dB point? What type of filter is this?

7. Get the magnitude and phase values through the 'bode' command.

8. Repeat steps 1-6 with the following RL circuit, where Vo is indicated along L1. Consider its initial conditions and equilibrium point to be zero.
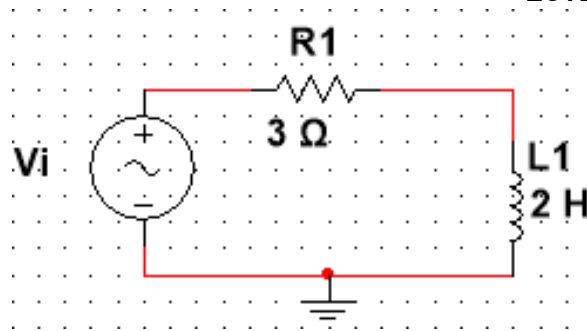
Circuit 5.2 First Order RL Circuit

What can you notice about the magnitude of the bode plot? How does this compare with that of the RC circuit? What is the magnitude in dB and frequency in rad/s of the -3dB point? What type of filter is this?

Task 2: Transfer Function of a Second Order Circuit

Consider the following RLC circuit given in Circuit 5.3. Consider its initial conditions and equilibrium point to be zero.
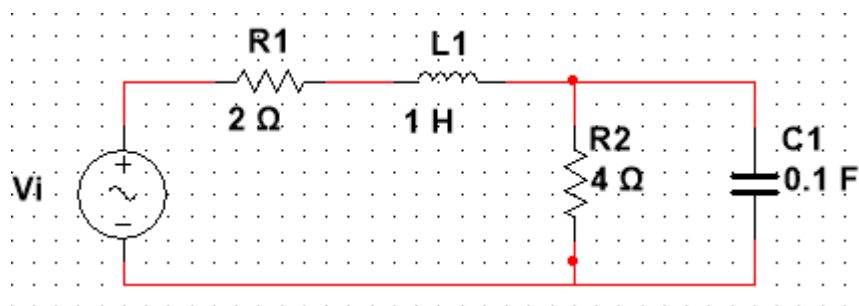
Circuit 5.3 Second Order RLC Circuit

1. In your logbooks, compute for the transfer function using the steps mentioned earlier as a guide. Note that Vo is along C1.

2. Write the coefficients of the transfer function's numerator and denominator in a MATLAB script (using arrays).

3. Use the MATLAB command 'tf' in order to save it into a system. Use the MATLAB help option to get more information about the commands.

4. Plot the system's magnitude and frequency using the bode plot. Use the MATLAB command 'bode' in order to do this. Draw the magnitude and frequency plots roughly in your logbook.

5. With the bode plot figure open, go to Tools, and check the Data Cursor option. What is the magnitude in dB and frequency in rad/s by which the magnitude plot starts to decline?

6. What is the magnitude in dB and frequency in rad/s of the -3dB point? What type of filter is this?

7. This is a second order RLC circuit. What is the reason why the bode magnitude plot declines instead of rises?

8. Get the magnitude and phase values through the 'bode' command.

Lab 6: Laplace Transforms

Laplace Transform

The transform allows equations in the "time domain" to be transformed into an equivalent equation in the Complex S Domain. The mathematical definition of the Laplace transform is defined by:

$$F(s) = \mathcal{L}\left\{f(t)\right\} = \int_{0^-}^{\infty} e^{-st} f(t)\, dt$$

As a general rule the transform of a function f(t) is written as F(s). Time-domain functions are written in lower-case, and the resultant s-domain functions are written in upper-case. However, functions in the Laplace representation can be converted back to time domain using the Inverse Laplace Transform.

The Inverse Laplace transform converts a function in the complex S-domain to its counterpart in the time-domain. Its mathematical definition is as follows:

$$f^{-1}\{F(s)\} = \frac{1}{2\pi} \oiint_{c-ii\infty}^{c+ii\infty} e^{st} F(s)\, ds = ff(t)$$

Task 1: Application of Laplace and Inverse Laplace Transforms in MATLAB

MATLAB has special functions called 'laplace', and 'ilaplace', which can aid you in performing laplace transforms. For this first task, we are going to test MATLAB's capability in performing this transform and its inverse.

1. Compute the Laplace transform of the following equations via MATLAB. Crosscheck this with the table in Appendix B to verify if correct. Show the code to your instructor.

   Tips:
   o First, you have to declare the symbolic variable names that you will be using via the MATLAB command `syms`. Use the MATLAB help to know more about this command and its syntax.
   o Write the function.
   o Perform the laplace transform on the function. Note down the answers in your logbook.

   Question 1: $x(t) = \frac{1}{4}\sin(t)$

   Question 2: $x(t) = \frac{5 - e^{-t}}{4\sqrt{\pi t^3}}$

   Question 3: $x(t) = \dfrac{\cos\left(\frac{1}{2t}\right)}{\sqrt{\pi t}}$

2. Compute the Inverse Laplace transform of the following equations via MATLAB. Crosscheck this with the table in Appendix B to verify if correct. Show the code to your instructor.

   Tips:
   - First, you have to declare the symbolic variable names that you will be using via the MATLAB command `syms`. Use the MATLAB help to know more about this command and its syntax.
   - Write the function.
   - Perform the inverse laplace transform on the function. Note down the answers in your logbook.

   Question 1: $F(s) = \dfrac{1}{(s-a)^2}$

   Question 2: $F(s) = \dfrac{e^{-2*s}}{\sqrt{s^2+1}}$

   Question 3: $F(s) = \dfrac{2}{5}(\ln(s))^2$

Task 2: Inverse Laplace Transform Using Partial Fraction Expansion

Find the partial fraction expansion of the following Laplace transforms.

Note: Use MATLAB command 'residue' to get the partial-fraction expansion. This command finds the residues, poles, and direct term of a partial fraction expansion of the ratio of two polynomials B(s)/A(s). The syntax is as follows:

$$\frac{B(s)}{A(s)} = \frac{R(1)}{s-P(1)} + \frac{R(2)}{s-P(2)} + \cdots + \frac{R(n)}{s-P(n)} + K(s)$$

Use the MATLAB help for the residue command to get the syntax on how to use it on functions. Write down the partial fraction expansions on your logbook.

Question 1: $Y(s) = \dfrac{3s^2+3s+1}{s^2(s^2+3s+2)}$

Question 2: $Y(s) = \dfrac{2s+1}{s^2(2s+2)}$

Question 3: $Y(s) = \dfrac{s^3+4s^2+s+2}{s^2+3s+1}$

Lab 7: Introduction to MATLAB App Designer

MATLAB App Designer

The MATLAB App Designer lets you design applications and simple user interfaces through MATLAB.

Before starting the lab, have a read through the following links in order to gather more information about the app designer:
- Create a Simple UI
  https://www.mathworks.com/help/matlab/creating_guis/ways-to-build-matlab-guis.html
- App Components
  https://www.mathworks.com/help/matlab/creating_guis/choose-components-for-your-app-designer-app.html
- Callbacks for Components
  https://www.mathworks.com/help/matlab/creating_guis/write-callbacks-for-gui-in-app-designer.html

Task 1: Creating a Bode Plotter for a First Order Circuit

1. To use the App Designer, launch MATLAB, and on the toolbar, go to **Apps -> Design App**

2. Select **Blank App**. This should lead you to the App designer's design view, where you can see all the possible components that you can use. Construct the user interface as per Figure 7.1 (the components can simply be dragged and dropped to the Design Window)
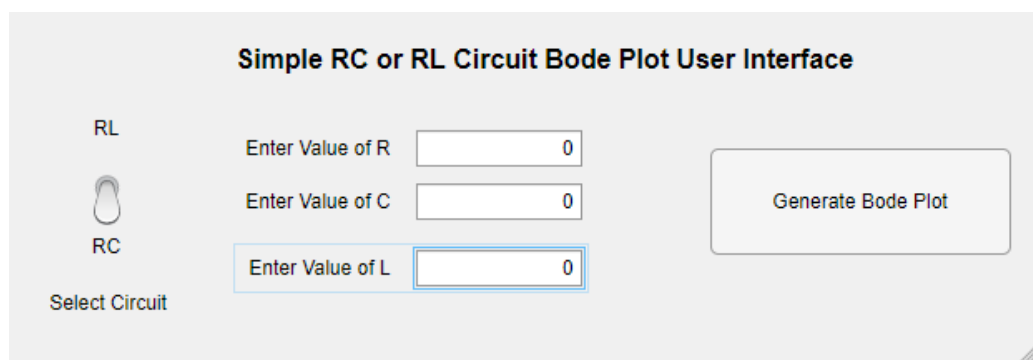

Figure 7.1 MATLAB App Designer User Interface

The UI observed in Figure 7.1 tells the user to choose whether the circuit in question is an RC or RL series circuit of the following connectivity (as per Figure 7.2). It then asks the user for the value of the Resistor, Capacitor, or Inductor. It then generates the bode plot accordingly (for any value of R, C, or L).
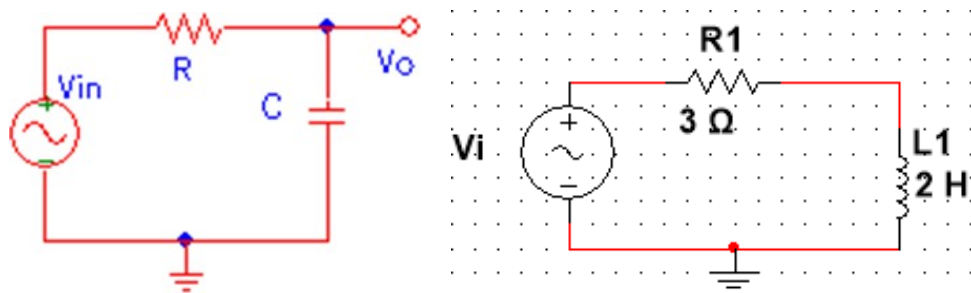

Figure 7.2 Series RC (a) and RL (b) circuits considered in the UI

Note that the texts and titles can be changed by double-clicking on the text you want to change.

3. After constructing the look of the UI interface,

   Right click on the button -> Callbacks -> Add a Callback

   This will create a function that takes into consideration the value of the Toggle Switch, as well as the edit functions.

4. Switch from Design View into **Code View**.

   From here, anything highlighted in white are editable code, while anything highlighted in gray are non-editable code that are automatically generated by the App Designer.

5. Under the Callback function for the button that you just created, write a code that runs our desired functionality. Follow the guidelines given below when writing the code:

   o Check if the value of the Circuit Switch is at 'RL' or 'RC'

   o If the value is at 'RL'

      ▪ Get the value of R
      ▪ Get the value of L
      ▪ Construct the system function following the transfer function equation:

   $$H(s) = \frac{sL}{sL + R}$$

      This resulted from computing the general transfer function from the circuit.

   o If the value is at 'RC'

      ▪ Get the value of R
      ▪ Get the value of C
      ▪ Construct the system function following the transfer function equation:

   $$H(s) = \frac{1}{1 + sRC}$$

      This resulted from computing the general transfer function from the circuit.

   o Plot the bode plot for the system using `bode`

   o Under the Editor toolbar, click on Save.

   o Run the Application.

When writing your code, consider the following guide:

- To get the value of an Edit Field, follow the syntax below:

```
val = app.NameofEditField.Value;
```

- To check the value of the Circuit switch, follow the syntax below:

```
strcmp(app.NameofCircuitSwitch.Value, 'Name of option');
```

References

[1] Natasha B. (2010), Multisim Basics Course: Two Tips for Schematic Capture, National Instruments forum, Accessible at: https://forums.ni.com/t5/National-Instruments-Circuit/Multisim-Basics-Course-Two-Tips-for-Schematic-Capture/ba-p/3482056?profile.language=en

[2] R. Nave, Hyperphysics, Accessible at: http://hyperphysics.phy-astr.gsu.edu/hbase/electric/Norton.html

[3] C.K. Alexander and M.N.O. Sadiku, Fundamentals of Electric Circuits: Fourth Edition, PSpice Simulation Problems.

[4] E. Cheever, (2005), What Bode Plots Represent: The Frequency Domain, Accessible at: https://lpsa.swarthmore.edu/Bode/BodeWhat.html

NOTE: A link to these resources and two sample sentences for the project are also available from the subject website.

Appendix A

<u>Demonstrator's Verification Sheet</u>

Upon completion of each section of the laboratory, each student should have the demonstrator sign to verify they have completed each part. A copy of this sheet will be provided in your first class and should be pasted into the front cover of your laboratory notebook.

Name:

Student Number:

Lab 1 completion date:	Demonstrator's Signature:

Lab 2 completion date:	Demonstrator's Signature:

Lab 3 completion date:	Demonstrator's Signature:

Lab 4 completion date:	Demonstrator's Signature:

Lab 5 completion date:	Demonstrator's Signature:

Lab 6 completion date:	Demonstrator's Signature:

Lab 7 completion date:	Demonstrator's Signature:

Appendix B

ID	$\hat{f}(s)$	$f(t)$
$F01$	$\dfrac{1}{1+s^2}$	$\sin t$
$F02$	$\dfrac{1}{\sqrt{s}+\sqrt{s+1}}$	$\dfrac{1-e^{-t}}{2\sqrt{\pi t^3}}$
$F03$	$\dfrac{1}{\sqrt{s(s+2)}}$	$e^{-t}I_0(t)$
$F04$	$\dfrac{1}{\sqrt{s}\left(1+\sqrt{s}\right)}$	$e^t erfc\left(\sqrt{t}\right)$
$F05$	$\exp\left(-2\sqrt{s}\right)$	$\dfrac{e^{-1/t}}{\sqrt{\pi t^3}}$
$F06$	$\dfrac{\exp\left(-\sqrt{s}\right)\cos\sqrt{s}}{\sqrt{s}}$	$\dfrac{\cos\left(1/2t\right)}{\sqrt{\pi t}}$
$F07$	$\dfrac{\exp(-1/s)}{\sqrt{s^3}}$	$\dfrac{\sin 2\sqrt{t}}{\sqrt{\pi}}$
$F08$	$\dfrac{-\ln(s)}{s}$	$\ln(t)+\gamma$
$F09$	$\dfrac{e^s K_1(s)}{s}$	$\sqrt{t(t+2)}$
$F10^*$	$\dfrac{1}{2}\left(\ln(s)\right)^2$	$\dfrac{\ln(t)+\gamma}{t}$
$F11^*$	$s^3\ln(s)$	$\dfrac{6}{t^4}$