

**G.PULLA REDDY ENGINEERING COLLEGE (AUTONOMOUS): KURNOOL**

**ELECTRICAL & ELECTRONICS ENGINEERING DEPARTMENT**

**III Semester B.Tech. - E.E.E. (Scheme-2020)**

**BASIC SIMULATION LABORATORY (BS (P))**

**List of Experiments**

1. Write a program for basic operations on matrices using MATLAB.
2. Write a program for determining the Eigen values and Eigen vectors using MATLAB.
3. Write a program for generation on various signals and Sequences (periodic and a periodic), such as unit impulse, unit step, square, saw tooth, triangular, sinusoidal, ramp using MATLAB.
4. Write a program for operations on signals and sequences such as addition, multiplication, scaling, shifting, folding, computation of energy and average power using MATLAB.
5. Write a program for finding the real and imaginary of signal using MATLAB.
6. Write a program for finding the Fourier transform of a given signal and plotting its magnitude and phase spectrum using MATLAB.
7. Write a program to perform convolution between signals and sequences using MATLAB.
8. Write a program to find the loop currents and Node voltages of the given circuit in MATLAB.
9. Verify KVL and KCL of the given circuit Using PSim.
10. Verify Reciprocity and Superposition theorem Using PSim.
11. Simulate Transient Analysis of a Linear Circuit using MATLAB simulink.
12. Simulate Steady-State Analysis of a Linear Circuit using MATLAB simulink.
13. Model a circuit for the nodal analysis to compute voltages  $v_1$  and  $v_2$  for a given circuit using MATLAB simulink.
14. a) Simulation of Half wave Rectifier.  
b) Simulation of Full wave Rectifier.
15. Verification of Locus diagrams through simulation.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for basic operations on matrices using MATLAB**

**GPRECD/EEE/EXPT-BS(P)-1**

**Date: 18-10-2021**

**AIM:** To study all basic operations on matrices.

**SOFTWARE:** MATLAB software

**THEORY:** Matrix operations are most fundamental to MATLAB. You can enter matrices in several different ways

- Enter an explicit list of elements
- Load matrices from external data files using load command
- Generate matrices using built in functions
- Separate the elements of a row with blanks or commas. Use a semicolon to indicate the end of each row. Surround the entire list of elements with square brackets [ ].

**PROCEDURE:**

1. Open MATLAB.
2. Open newM-file.
3. Initialize matrix A and matrix B.
4. Perform various operations such as addition, subtraction, multiplication (A+B,A-B,A\*B,A./B).
4. Save in current directory.
5. Compile and Run the program.
6. For the output see command window\ Figure window.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for basic operations on matrices using MATLAB**

**GPRED/EEE/EXPT-BS(P)-1**

**Date: 18-10-2021**

**RESULT:**

**VIVA QUESTIONS:-**

1. Expand MATLAB? And importance of MATLAB?
2. What is clear all and close all will do?
3. What is disp() and input()?
4. What is the syntax to find the inverse of Matrix?
5. What is the syntax to find the rank of the matrix?

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for determining the Eigen values and Eigen Vectors using MATLAB**

**GPRECD/EEE/EXPT-BS(P)-2**

**Date: 18-10-2021**

**AIM:** To write a program for determining the Eigen values and Eigen vectors on matrices using MATLAB.

**SOFTWARE:** MATLAB software

**THEORY:** Matrix operations are most fundamental to MATLAB. You can enter matrices in several different ways

- Enter an explicit list of elements
- Load matrices from external data files using load command
- Generate matrices using built in functions
- Separate the elements of a row with blanks or commas. Use a semicolon to indicate the end of each row. Surround the entire list of elements with square brackets [ ].

**PROCEDURE:**

1. Open MATLAB.
2. Open newM-file.
3. Enter matrix A of order nXn.
4. Compute Eigen values using function eig(A) and also find Eigen vectors.
4. Save in current directory.
5. Compile and Run the program.
6. For the output see command window\ Figure window.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for determining the Eigen values and Eigen Vectors using MATLAB**

**GPRECD/EEE/EXPT-BS(P)-2**

**Date: 18-10-2021**

**RESULT:**

**VIVA QUESTIONS:-**

1. What is the syntax to find square of matrix.
2. What is command window?
3. What is the syntax to find the determinant of a matrix?
4. What is the syntax to find the Eigen values and eigenvectors of the matrix?
5. What is the syntax to find the trace of the matrix?

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE:** Write a program for generation on various signals and Sequences (periodic and aperiodic), such as unit impulse, unit step, square, saw tooth, triangular, sinusoidal, ramp using MATLAB

**GPRED/EEE/EXPT-BS(P)-3**

**Date: 18-10-2021**

**AIM:** To generate various signals unit impulse, unit step, Ramp, sine, sinc, triangular, sawtooth using MATLAB.

**SOFTWARE:** MATLAB software

**THEORY:**

- The unit impulse signal is the simplest of all and one of the most useful signals in signal processing applications. The impulse signal is given by

$$\delta(t) = \begin{cases} 1 & \text{for } t = 0 \\ 0 & \text{for } t \neq 0 \end{cases}$$

- The unit step signal which is frequently used in signal processing applications

$$u(t) = \begin{cases} 1 & \text{for } t \geq 0 \\ 0 & \text{for } t < 0 \end{cases}$$

- The Ramp signal is given by

$$r(t) = \begin{cases} t & \text{for } t \geq 0 \\ 0 & \text{for } t < 0 \end{cases}$$

- The sine wave of frequency  $f$  and phase angle  $\theta$  can be expressed by the equation

$$X = \sin(2\pi ft + \theta)$$

- The triangular signal is defined as

$$X(t) = 1 - t/a \quad \text{for } t \leq a$$

- The MATLAB code is self explanatory and generates a square wave of 50 Hz frequency. It calls a built in MATLAB function 'square'.

The MATLAB code calls the 'sawtooth' built-in function and the sampling frequency is specified externally for more clarity.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for generation on various signals and Sequences (periodic and a periodic), such as unit impulse, unit step, square, saw tooth, triangular, sinusoidal, ramp using MATLAB**

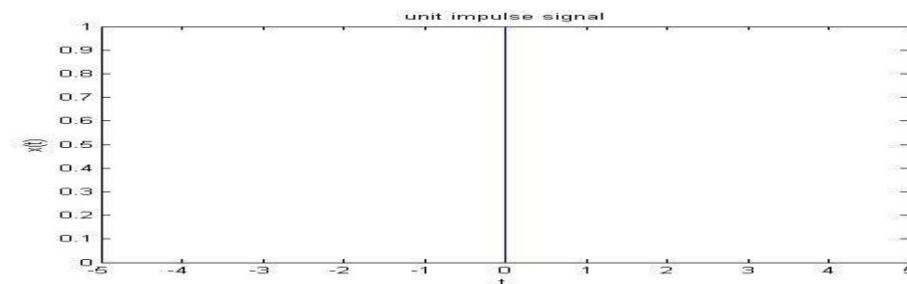
**GPRECD/EEE/EXPT-BS(P)-3**

**Date: 18-10-2021**

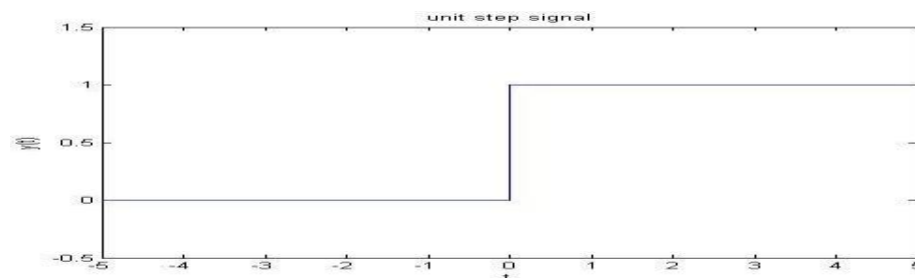
**PROCEDURE:**

1. Open MATLAB
2. Open new M-file
3. Initialize the time limits for the impulse signal.
4. plot the signal taking time on x-axis and  $x(t)$  amplitude on y axis using plot function.
5. Lable the x-axis with time and y axis with amplitude
4. Save in current directory
5. Compile and Run the program
6. For the output see command window\ Figure window
7. Repate the step 3 to 6 for generation of unit step, square, sawtooth, triangular, sinusoidal and Ramp signal.

**EXPECTED OUTPUT: IMPULSE SIGNAL**



**UNIT STEP SIGNAL:**



1.

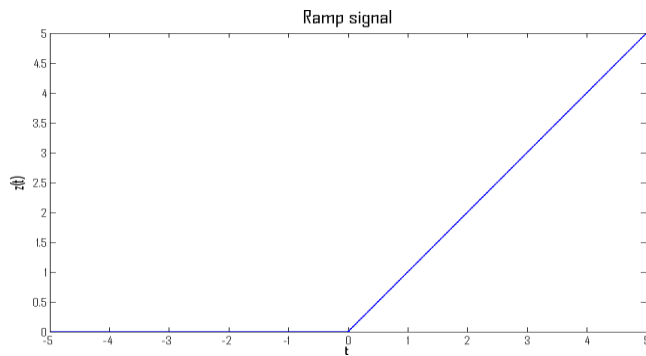
**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for generation on various signals and Sequences (periodic and a periodic), such as unit impulse, unit step, square, saw tooth, triangular, sinusoidal, ramp using MATLAB**

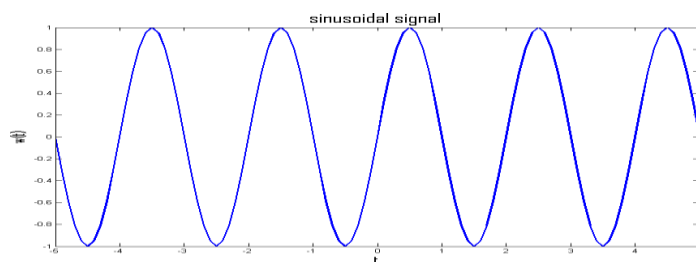
**GPRECD/EEE/EXPT-BS(P)-3**

**Date: 18-10-2021**

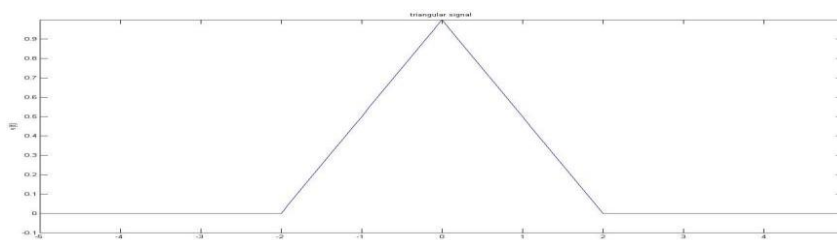
**RAMP SIGNAL:**



**Sinusoidal signal:**



**Triangular signal:**





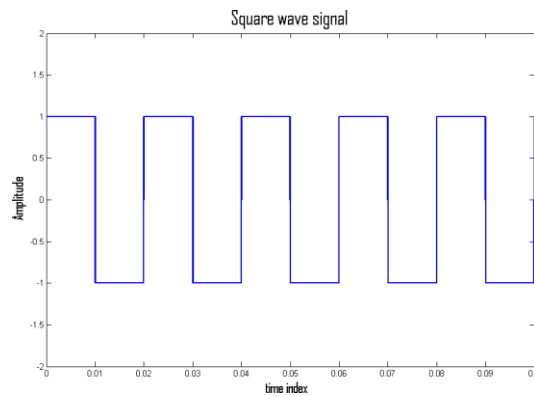
**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE:** Write a program for generation on various signals and Sequences (periodic and a periodic), such as unit impulse, unit step, square, saw tooth, triangular, sinusoidal, ramp using MATLAB

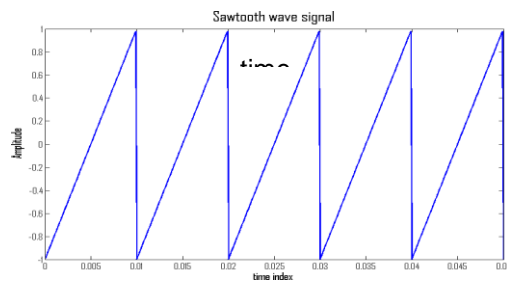
**GPRED/EEE/EXPT-BS(P)-3**

**Date: 18-10-2021**

**Square signal:**



**Sawtooth signal:**



**RESULT:**

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for generation on various signals and Sequences (periodic and aperiodic), such as unit impulse, unit step, square, saw tooth, triangular, sinusoidal, ramp using MATLAB**

**GPRECD/EEE/EXPT-BS(P)-3**

**Date: 18-10-2021**

**VIVA QUESTIONS: -**

1. Define Signal?
2. Define Deterministic and Random Signal?
3. Define ramp Function?
4. What is Signal Modeling?
5. Define Periodic and Aperiodic Signal

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE:** Write a program for operations on signals and sequences such as addition, multiplication, scaling, shifting, folding, computation of energy and average power

**GPRECD/EEE/EXPT-BS(P)-4**

**Date: 18-10-2021**

**AIM:** To perform all the operations on signals like shifting, folding, scaling, addition, multiplication

**SOFTWARE:** MATLAB software

**THEORY:** A discrete time system is a device that operates on a discrete time input signal  $x(n)$  to produce another discrete time signal  $y(n)$  at its output. **Signal Scaling:** The scalar or constant multiplier block produces a new signal sequence by multiplying each sample of the input signal  $x(t)$  by a scalar quantity 'A'. If the scale factor is greater than unity  $x(t)$  is amplified 'A' times whereas the signal  $x(t)$  gets attenuated or suppressed, if A is a positive fraction. Mathematically the signal scaling operation is expressed as  $y(t)=Ax(t)$

**Signal Folding:** When a given signal  $x(t)$  is flipped from left to right about the time origin  $n=0$ , the signal is said to be folded or reflected. The folding operation creates the mirror image of  $x(t)$ .  $y(t) = x(-t)$

**Signal shifting:** A signal  $x(t)$  is shifted in time by replacing the variable  $t$  by  $t-k$  where  $k$  is an integer. If  $k$  is a positive integer the signal  $x(t)$  is delayed by  $k$  units in time as it gets shifted towards right. However if  $k$  is negative the signal  $x(t)$  is time advanced by  $k$  units and it gets shifted towards left.

**Signal Addition:** The addition operation performs addition of two signals. Such a device is called as Adder.

$$y(t) = x_1(t)+x_2(t)$$

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE:** Write a program for operations on signals and sequences such as addition, multiplication, scaling, shifting, folding, computation of energy and average power

**GPRED/EEE/EXPT-BS(P)-4**

**Date: 18-10-2021**

**Signal Multiplication:** The multiplication operation performs multiplication of two signals. Such a device is called as multiplier or modulator.  $y(t) = x_1(t)*x_2(t)$

**PROCEDURE:**

1. Open MATLAB
2. Open newM-file
3. Define the signal  $x(t)$  say  $x(t)=1+t$  for  $(t \geq 0 \& t \leq 2)$
4. Plot the signal taking time on x-axis and  $x(t)$  amplitude on y axis using plot function.
5. Label the x-axis with time and y axis with amplitude using label command.
6. Perform signal addition, signal multiplication, scaling, shifting, folding, computation of energy and average power
6. Save in current directory
7. Compile and Run the program
8. For the output see command window\ Figure window

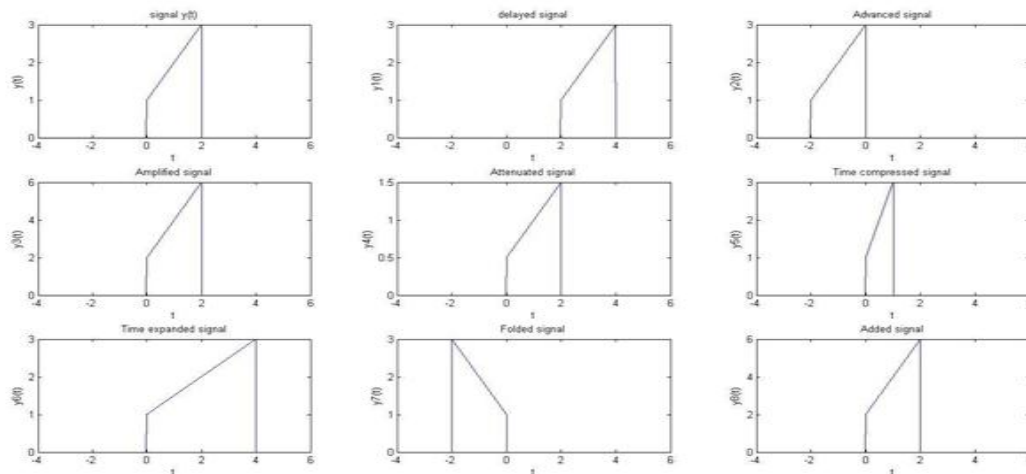
**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE:** Write a program for operations on signals and sequences such as addition, multiplication, scaling, shifting, folding, computation of energy and average power

**GPRED/EEE/EXPT-BS(P)-4**

**Date: 18-10-2021**

**EXPECTED OUTPUT:** for the taken input signal  $x(t)=1+t$ ;



**RESULT:**

**VIVA QUESTIONS:**

1. What is amplitude scaling of a signal.
2. What is time scaling of a signal.
3. What is time shifting of signal.
4. What is time reversal of signal.
5. Define energy and power of a signal.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for finding the real and imaginary of signal using MATLAB**

**GPRECD/EEE/EXPT-BS(P)-5**

**Date: 18-10-2021**

**AIM:** To find the Real and imaginary parts of the given signal .

**SOFTWARE:** MATLAB software

**THEORY:** An Arbitrary real signal which doesn't have any sort of symmetry property can be expressed as the sum of two signal components which is the even signal and the other an odd signal. Any signal  $x(t)$  be expressed as the sum of  $x_e(t)$  and  $x_o(t)$ .

$$x(t) = x_e(t) + x_o(t)$$

$x_e(t)$  is the even part of the signal.

$x_o(t)$  is the odd part of the signal.

A complex signal consists of two real signals - one for the real and one for the imaginary part. The linear processing of a complex signal, such as filtration with a time-invariant linear filter, corresponds to applying the processing both to the real and the imaginary part of the signal.

**PROCEDURE:**

1. Open MATLAB
2. Open new M-file
3. Define any complex signal say  $x(t)$
4. Plot real and imaginary parts of signal using real and imaginary function in MATLAB.
4. Save in current directory
5. Compile and Run the program
6. For the output see command window\ Figure window

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for finding the real and imaginary of signal using MATLAB**

**GPRECD/EEE/EXPT-BS(P)-5**

**Date: 18-10-2021**

**RESULT:**

**VIVA QUESTIONS:-**

1. What is the formula to find odd part of signal?
2. What is Even Signal?
3. What is Odd Signal?
4. What is the formula to find even part of signal?
5. What is the difference b/w stem & plot?

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE:** Write a program for finding the Fourier transform of a given signal and plotting its magnitude and phase spectrum using MATLAB

**GPRED/EEE/EXPT-BS(P)-6**

**Date: 18-10-2021**

**AIM:** To find the Fourier transform of a signal and plotting its magnitude and phase spectrum.

**SOFTWARE:** MATLAB software

**PROCEDURE:**

1. Open MATLAB
2. Open newM-file
3. Initialize the time limits for the impulse signal.
4. Plot the signal taking discrete time on x-axis and x(n) amplitude on y axis using plot function.
5. Determine the number of samples.
6. Label the x-axis with time and y axis with amplitude.
7. Using the stem function plot magnitude and phase plot.
8. Save in current directory
9. Compile and Run the program
10. For the output see command window\ Figure window

**EXPECTED OUTPUT:** when taken input signal is  $x=[1\ 1\ 1\ 0\ 0\ 0\ 0\ 0]$

x =

1     1     1     0     0     0     0     0

y = 3.0000

1.7071 - 1.7071i

0 - 1.0000i

0.2929 + 0.2929i

1.0000

0.2929 - 0.2929i



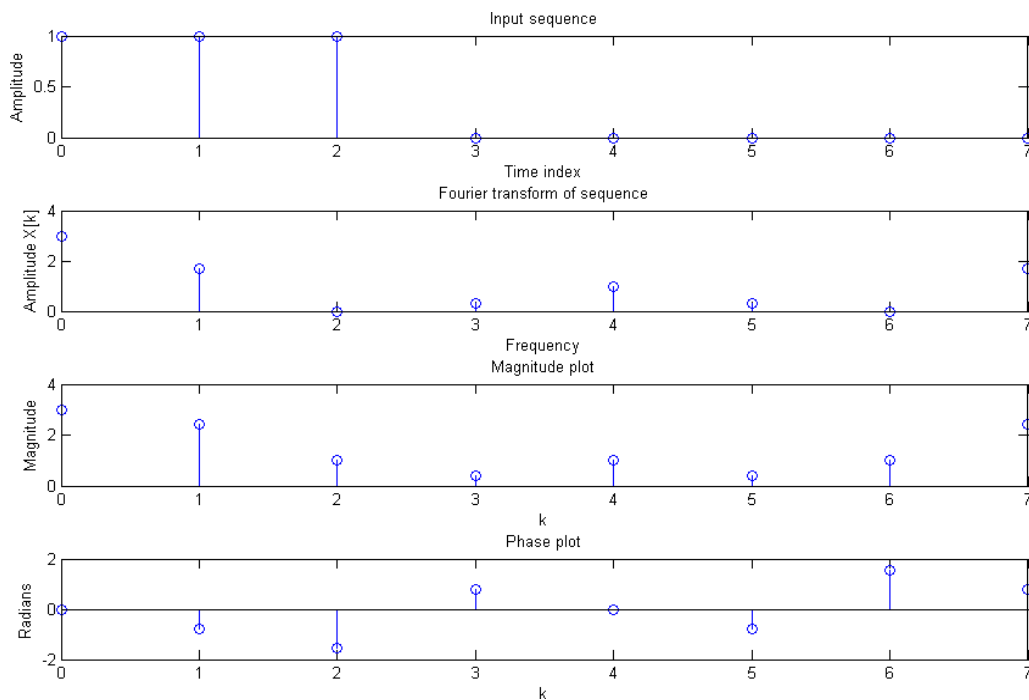
**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program for finding the Fourier transform of a given signal and plotting its magnitude and phase spectrum using MATLAB**

**GPRED/EEE/EXPT-BS(P)-6**

**Date: 18-10-2021**

$0 + 1.0000i$   
 $1.7071 + 1.7071i$



**RESULT:**

**VIVA QUESTIONS:-**

1. Define Fourier series?
2. What are the properties of Continuous-Time Fourier Series?
3. What is the sufficient condition for the existence of F.T?
4. Define the F.T of a signal?
5. What is the difference b/w F.T&F.S?

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program to perform convolution between signals and sequences using MATLAB**

**GPRED/EEE/EXPT-BS(P)-7**

**Date: 18-10-2021**

**AIM:** To perform convolution between two continuous time signals using MATLAB.

**SOFTWARE:** MATLAB software

**THEORY:** The convolution operation is the back bone for signal processing and it is also applicable for discrete time signals.

The symbol \* is used to denote the convolution operation. The convolution operation is expressed mathematically by the following equation

$$y(t) = x_1(t) * x_2(t)$$

**PROCEDURE:**

1. Open MATLAB
2. Open newM-file
3. Type the program
4. Define two signals say  $x_1(t)=1$  for  $(t \geq 1$  and  $t \leq 10)$  and  $x_2(t)=( t \geq 2$  and  $t \leq 10)$
4. Plot the signal taking time on x-axis and  $x(t)$  amplitude on y axis using plot function.
5. Label the x-axis with time and y axis with amplitude using label command.
6. Perform convolution using convolution function.
5. Compile and Run the program
6. For the output see command window\ Figure window

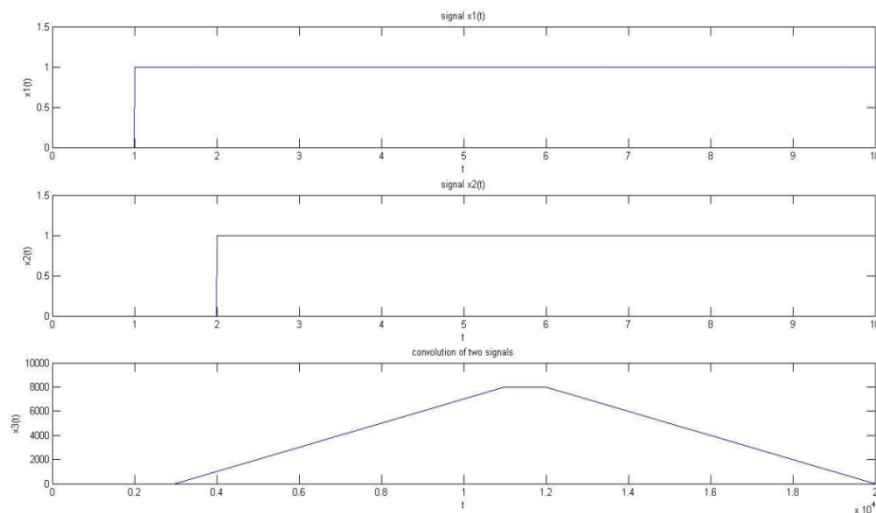
**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program to perform convolution between signals and sequences using MATLAB**

**GPRED/EEE/EXPT-BS(P)-7**

**Date: 18-10-2021**

**EXPECTED OUTPUT:** for the given input signal are  $x_1(t)=1$  for ( $t \geq 1$  and  $t \leq 10$ ) and  $x_2(t)=1$  ( $t \geq 2$  and  $t \leq 10$ )



**RESULT:**

**VIVA QUESTIONS:-**

1. Define Convolution?
2. Define Properties of Convolution?
3. What is the Difference between Convolution & Correlation?
4. What are Dirichlet Condition?
5. What is Half Wave Symmetry?

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program to find the loop currents and Node voltages of the given circuit in MATLAB**

**GPRED/EEE/EXPT-BS(P)-8**

**Date: 18-10-2021**

**AIM:** Write a program to find the loop currents and Node voltages of the given circuit in MATLAB.

**SOFTWARE:** MATLAB software

**THEORY:**

**NODAL ANALYSIS:**

Kirchhoff's current law states that for any electrical circuit, the algebraic sum of all the currents at any node in the circuit equals zero. In nodal analysis, if there are n nodes in a circuit, and we select a reference node, the other nodes can be numbered from  $V_1$  through  $V_{n-1}$ . With one node selected as the reference node, there will be n-1 independent equations. If we assume that the admittance between nodes i and j is given as  $Y_{ij}$ , we can write the nodal equations:

$$\begin{aligned} Y_{11} V_1 + Y_{12} V_2 + \dots + Y_{1m} V_m &= I_1 \\ Y_{21} V_1 + Y_{22} V_2 + \dots + Y_{2m} V_m &= I_2 \text{ and so on} \end{aligned}$$

From this we have  $I=[Y][V]$

**LOOP ANALYSIS:**

Loop analysis is a method for obtaining loop currents. The technique uses Kirchhoff voltage law (KVL) to write a set of independent simultaneous equations. The Kirchhoff voltage law states that the algebraic sum of all the voltages around any closed path in a circuit equals zero.

In loop analysis, we want to obtain current from a set of simultaneous equations. The latter equations are easily set up if the circuit can be drawn in planar fashion. This implies that a set of simultaneous equations can be obtained if the circuit can be redrawn without crossovers.

For a planar circuit with n-meshes, the KVL can be used to write equations for each mesh that does not contain a dependent or independent current source. Using KVL and writing equations for each mesh, the resulting equations will have the general form:

$$\begin{aligned} Z_{11}I_1 + Z_{12} I_2 + Z_{13} I_3 + \dots + Z_{1n} I_n &= V_1 \\ Z_{21}I_1 + Z_{22} I_2 + Z_{23} I_3 + \dots + Z_{2n} I_n &= V_2 \text{ and so on} \end{aligned}$$

From this we have  $V=[Z][I]$

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program to find the loop currents and Node voltages of the given circuit in MATLAB**

**GPRECD/EEE/EXPT-BS(P)-8**

**Date: 18-10-2021**

**PROCEDURE:**

**I.NODAL ANALYSIS**

1. Open MATLAB
2. Open new M-file
3. Initialize the Admittance matrix Y and current vector I of the given circuit.
5. Find the Nodal voltages using  $V=Inv(Y)I$ .
4. Save in current directory
5. Compile and Run the program
6. For the output see command window\ Figure window

**II.LOOP ANALYSIS**

1. Open MATLAB
2. Open new M-file
3. Initialize the impedance matrix Z and voltage vector V of the given circuit.
4. Solve for loop currents using  $I=Inv(Z)*V$ .
5. Save in current directory.
6. Compile and Run the program
7. For the output see command window\ Figure window

**RESULT:**

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Write a program to find the loop currents and Node voltages of the given circuit in MATLAB**

**GPRED/EEE/EXPT-BS(P)-8**

**Date: 18-10-2021**

**VIVA QUESTIONS:-**

1. What is impedance matrix?
2. What is admittance matrix?
3. State KCL.
4. State KVL.
5. What is difference between nodal analysis and loop analysis.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Verify the KVL and KCL of the Given Circuit using PSim**

**GPRED/EEE/EXPT-BS(P)-9**

**Date: 18-10-2021**

**AIM:** To Verify KVL and KCL of a given circuit using PSIM

**SOFTWARE:** PSIM

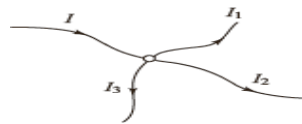
**THEORY:**

**KCL:** Kirchhoff's current law states that the algebraic sum of currents at a node or junction is equal to zero. It can also be stated as the sum of currents entering a node is equal to sum of currents coming out of that node.

This law is also called **Kirchhoff's first law**, **Kirchhoff's point rule**, or **Kirchhoff's junction rule** or **nodal rule**.

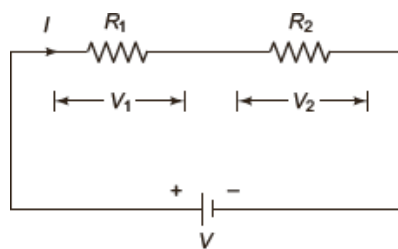
Consider the circuit shown in Fig.1 below. It can be seen that the currents flowing through the branches are  $I_1$ ,  $I_2$ ,  $I_3$  whereas the net current is given by  $I$ . So, using Kirchhoff's current law, we can write

$$I = I_1 + I_2 + I_3$$



**KVL:** Kirchhoff's voltage law (KVL) states that the algebraic sum of voltages in a closed loop is equal to zero.

As current flows through resistor there will be a voltage drop. This voltage drop of all resistors along with the source voltage should be zero. Consider the circuit shown below figure. Two resistors  $R_1$  and  $R_2$  are connected to each other and are supplied from a voltage source of  $V$  volts. Assume that a current of  $I$  amps flows through the circuit.



**TITLE: Verify the KVL and KCL of the Given Circuit using PSim**

**GPRED/EEE/EXPT-BS(P)-9**

**Date: 18-10-2021**

The value of current  $I$  flowing through the resistors is the same in the above circuit. Due to the flow of current there will be a voltage drop in both the resistors. Let the voltage drops in both the resistors be  $V_1$  and  $V_2$ . So, in order to write the voltage equations along with the polarities the following notation can be considered.

(i) Starting at one end of the circuit (preferably the ground terminal) follow the current direction and note down the first sign that comes in that direction along with that value of voltage.

(ii) Follow the same steps for the remaining elements also.

Using the above methodology, the voltage equations may be written as

$$-V + V_1 + V_2 = 0$$

But  $V_1 = IR_1$  and  $V_2 = IR_2$ , which implies

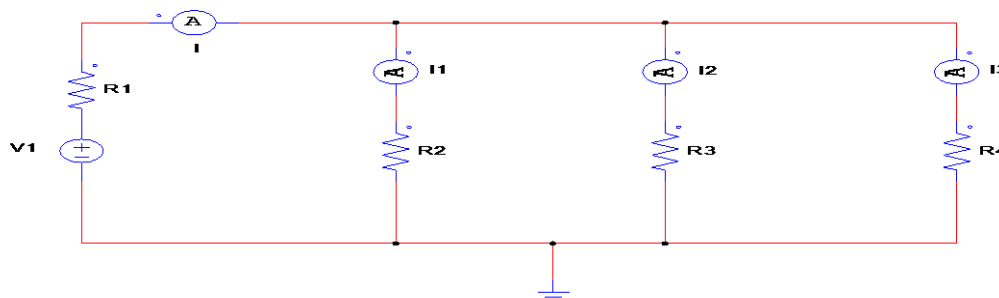
$$-V + IR_1 + IR_2 = 0$$

Or

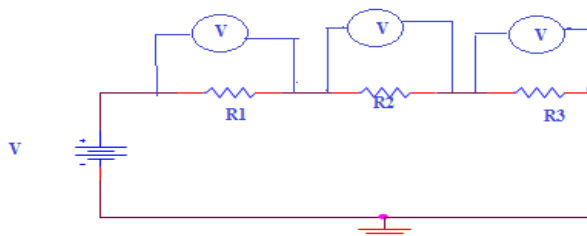
$$IR_1 + IR_2 = V$$

**CIRCUIT DIAGRAM:**

**1. KCL**



**2. KVL**





**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Verify the KVL and KCL of the Given Circuit using PSim**

**GPRECD/EEE/EXPT-BS(P)-9**

**Date: 18-10-2021**

**PROCEDURE:**

- 1) Start the PSIM software.
- 2) Launch PSIM. Select **File >> Open**, and load the schematic file from the sub-folder "examples\kvl".
- 3) Drag & drop the components from library.
- 4) Choose **Simulate >> Run PSIM Simulation** to start the simulation. Component that can be plotted into simulink or design window.
- 5) After the simulation is complete, the waveform processing program SIMVIEW will appear.

**TABULAR COLUMN:**

**FOR KCL**

Applied Voltage V (volts)	I (A)		I <sub>1</sub> (A)		I <sub>2</sub> (A)		I <sub>1</sub> +I <sub>2</sub> (A)	
	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical

**For KVL**

Applied Voltage V (volts)	V <sub>1</sub> (volts)		V <sub>2</sub> (volts)		V <sub>3</sub> (volts)		V <sub>1</sub> +V <sub>2</sub> +V <sub>3</sub> (volts)	
	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical	Theoretical	Practical

**RESULT:**

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Verify the KVL and KCL of the Given Circuit using PSim**

**GPRECD/EEE/EXPT-BS(P)-9**

**Date: 18-10-2021**

**VIVAQUESTIONS:**

1. State KCL.
2. State KVL.
3. What is the difference between PSIM and MATLAB software.
4. Nodal analysis is based on which Kirchhoff's law.
5. Mesh analysis is based on which Kirchhoff's law

**TITLE: Verify Reciprocity and Super Position Theorem using PSIM**

**GPRED/EEE/EXPT-BS(P)-10**

**Date: 18-10-2021**

**AIM:** To verify Reciprocity Theorem and Super position Theorem using PSIM

**SOFTWARE:** PSIM

**THEORY:**

**RECIPROCITY THEOREM:**

**STATEMENT:** *In a linear bilateral network, the ratio of excitation to response is equal in the case even though the positions of excitation and response are interchanged. However, if the excitation is a voltage source, the response must be a current and vice-versa.*

The reciprocity theorem can be illustrated using this network. Considering the network has  $k$  loops and the network is excited by  $V_s$  (voltage source) at  $s_{th}$  loop and current should be determined at  $r_{th}$  loop then

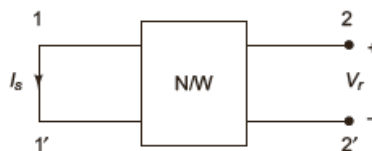
$$I_r = \frac{A_{sr}}{D_z} V_s \text{ which implies } \frac{V_s}{I_r} = \frac{D_z}{A_{sr}} \quad (a)$$

where  $A_{sr}$  is a cofactor

$D_z$  is the impedance of the network

$\frac{A_{sr}}{D_z}$  is the admittance

As we interchange the positions of response and excitation, the circuit is



**Fig. 2**

$$I_s = \frac{A_{rs}}{D_z} V_r \text{ which implies}$$

**TITLE: Verify Reciprocity and Super Position Theorem using PSIM**

**GPRED/EEE/EXPT-BS(P)-10**

**Date: 18-10-2021**

$$\frac{V_r}{I_s} = \frac{D_z}{A_{rs}} \quad (b)$$

From equations a and b

$$\frac{V_s}{I_r} = \frac{D_z}{A_{sr}} \quad \text{and} \quad \frac{V_r}{I_s} = \frac{D_z}{A_{rs}}$$

If  $A_{sr}$  and  $A_{rs}$  are equal we can have a symmetric network with RLC elements and no controlled sources

$$\frac{V_s}{I_r} = \frac{V_r}{I_s}$$

**SUPER POSITION THEOREM:**

**STATEMENT:** *In any linear network with several independent sources and dependent sources, the overall response in any part of the network is equal to the sum of the individual responses due to each independent source with all other independent sources reduced to zero*

Superposition theorem is valid only for linear systems. Superposition theorem is not applicable to power. This theorem can be applicable to voltages and currents. The above statement can be proved by the following explanation.

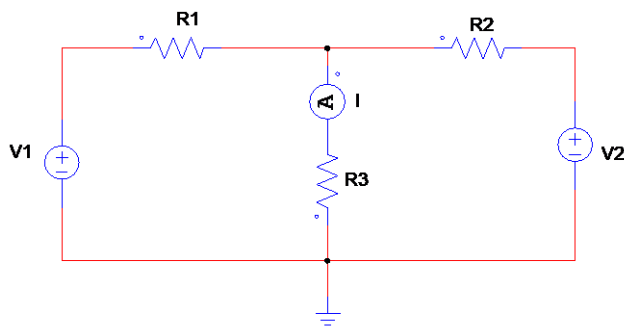
In linear systems, currents are propositional to the voltages  $I \propto V$

In this theorem, we add individual currents of a branch due to each source taking at a time to get the resultant current. This is similar to voltages.

∴ We can have  $I = I_1 + I_2 + \dots$

**CIRCUIT DIAGRAM: Super Position**

Fig:1



**TITLE: Verify Reciprocity and Super Position Theorem using PSIM**

**GPRED/EEE/EXPT-BS(P)-10**

**Date: 18-10-2021**

Fig:2

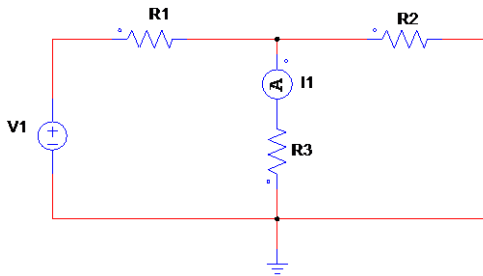


Fig:3

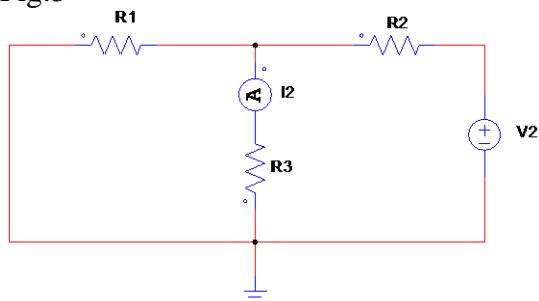


Fig:4

**CIRCUIT DIAGRAM: Reciprocity Theorem**

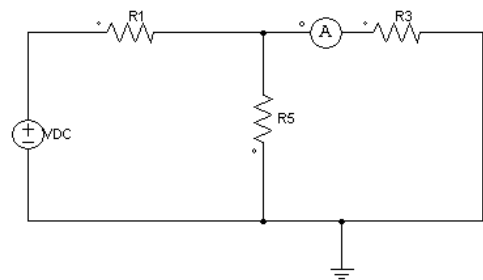
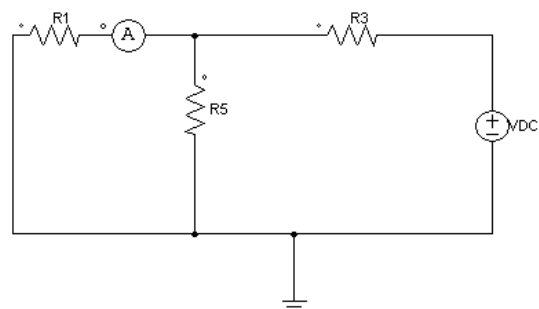


Fig:5



**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Verify Reciprocity and Super Position Theorem using PSIM**

**GPRECD/EEE/EXPT-BS(P)-10**

**Date: 18-10-2021**

**PROCEDURE:**

- 1) Start the PSIM software.
- 2) Launch PSIM. Select **File >> Open**, and load the schematic file from the sub-folder “examples\reciprocity”.
- 3) Drag & drop the components from library.
- 4) Choose **Simulate >> Run PSIM Simulation** to start the simulation. Component that can be plotted into simulink or design window.
- 5) After the simulation is complete, the waveform processing program SIMVIEW will appear

Tabular Column: 1. Super position Theorem:

Table: 1 for fig 2

Sl.no.	V <sub>1</sub> (V)	I(A)	
		Th.	Pr.

Table:2 for fig 1

Sl.no.	V <sub>1</sub> (V)	V <sub>2</sub> (V)	I(A)		I=( I <sub>1</sub> + I <sub>2</sub> ) (A)	
			Th.	Pr.	Th.	Pr.

Table: 3 for fig 3

Sl.no.	V <sub>2</sub> (V)	I <sub>2</sub> (A)	
		Th.	Pr.

**2. Reciprocity Theorem: Table 4 for fig 4**

S.No.	V	I		V / I	
		Th.	Pr.	Th.	Pr.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Verify Reciprocity and Super Position Theorem using PSIM**

**GPRED/EEE/EXPT-BS(P)-10**

**Date: 18-10-2021**

Tabular Column 5 for fig 5

S.No.	V	I		V / I	
		Th.	Pr.	Th.	Pr.

**RESULT:**

**VIVA QUESTIONS:**

1. State Superposition Theorem.
2. State Reciprocity Theorem.
3. What are the applications of Reciprocity Theorem.
4. What are the applications of Superposition Theorem.
5. State Nortons Theorem.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Simulate Transient Analysis of a Linear Circuit using MATLAB simulink**

**GPRECD/EEE/EXPT-BS(P)-11**

**Date: 18-10-2021**

**AIM:** To Simulate Transient Analysis of a Linear Circuit using MATLAB simulink

**SOFTWARE:** MATLAB software

**THEORY:**

This circuit is a simplified model of a 230 kV three-phase power system. Only one phase of the transmission system is represented. The equivalent source is modeled by a voltage source (230 kV rms/sqrt(3) or 187.8 kV peak, 60 Hz) in series with its internal impedance ( $R_s$   $L_s$ ) corresponding to a 3-phase 2000 MVA short circuit level and  $X/R = 10$ . ( $X = 230e3^2/2000e6 = 26.45$  ohms or  $L = 0.0702$  H,  $R = X/10 = 2.645$  ohms). The source feeds a RL load through a 150 km transmission line. The line distributed parameters ( $R = 0.035$ ohm/km,  $L = 0.92$  mH/km,  $C = 12.9$  nF/km) are modelled by a single pi section (RL1 branch 5.2 ohm; 138 mH and two shunt capacitances C1 and C2 of 0.967 uF). The load (75 MW - 20 Mvar per phase) is modelled by a parallel RLC load block. A circuit breaker is used to switch the load at the receiving end of the transmission line. The breaker which is initially closed is opened at  $t = 2$  cycles, then it is reclosed at  $t = 7$  cycles. Current and Voltage Measurement blocks provide signals for visualization purpose.

**PROCEDURE:**

1. Open a new simulation window by clicking on simulink button.
2. Window that appears in simulink library browser the window contains component that can be plotted into simulink or design window.
3. Drag & drop the components from library.
4. Start the simulation and observe line voltage and load current transients during load switching and note that the simulation starts in steady-state.
5. Open the Powergui block and select "Steady State Voltage and Currents" to measure the steady-state voltage and current phasors. Using the Powergui select now "Initial States Setting" to obtain the initial state values (voltage across capacitors and current in inductances).



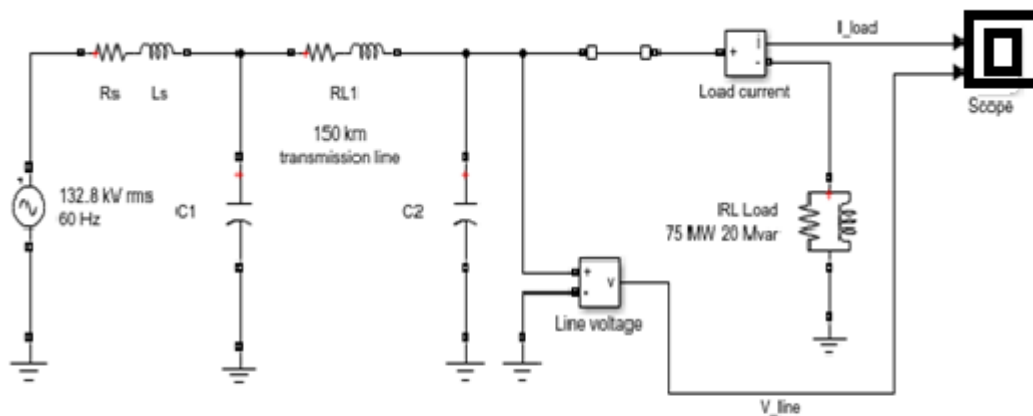
**TITLE: Simulate Transient Analysis of a Linear Circuit using MATLAB simulink**

**GPRED/EEE/EXPT-BS(P)-11**

**Date: 18-10-2021**

Now, reset all the initial states to zero by clicking the "to zero" button and then "Apply" to confirm changes. Restart the simulation and observe transients at simulation starting. Using the same Powergui window, you can also set selected states to specific values.

**SIMULINK DIAGRAM:**



**Transient Analysis of a Linear Circuit**

**RESULT:**

**VIVA QUESTIONS:**

1. What is transient analysis.
2. Why transient analysis is required.
3. What is power GUI
4. What is scope.
5. Differentiate transient and steady state.

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Simulate Steady state Analysis of a Linear Circuit using MATLAB simulink**

**GPRED/EEE/EXPT-BS(P)-12**

**Date: 18-10-2021**

**AIM:** To Simulate Steady state Analysis of a Linear Circuit using MATLAB simulink

**SOFTWARE:** MATLAB software

**THEORY:**

A 5th harmonic filter is connected at a bus bar fed by a 60 Hz, 100 V inductive sources. A 5th harmonic (300 Hz, 1 A) current is injected at the bus bar. This linear system consists of 3 states (2 inductor currents and 1 capacitor voltage), 2 inputs ( $V_s$ ,  $I_s$ ) and 2 outputs (Current and Voltage Measurement). An Impedance Measurement block is used to compute the impedance versus frequency of the circuit.

**PROCEDURE:**

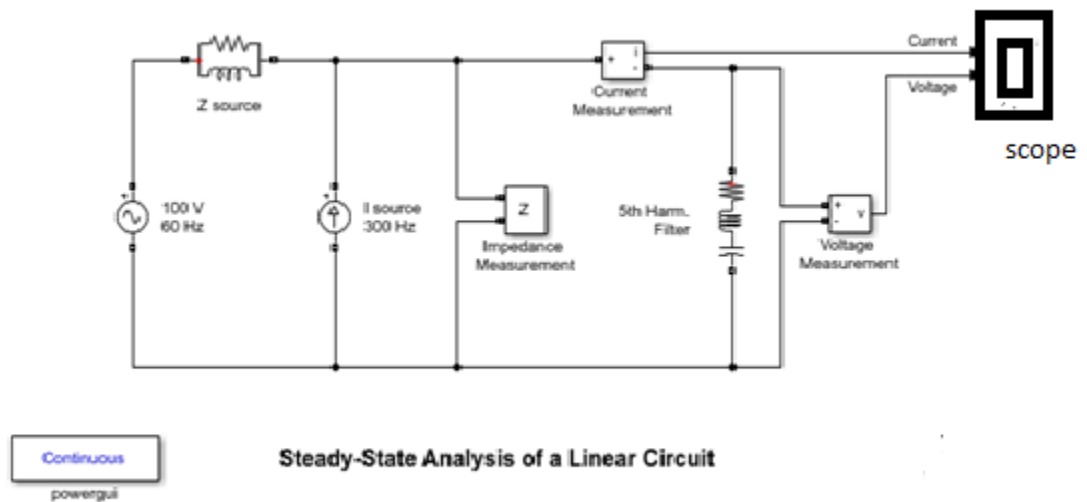
1. Open a new simulation window by clicking on simulink button.
2. Window that appears in simulink library browser the window contains component that can be plotted into simulink or design window.
3. Drag & drop the components from library.
4. Use the Powergui block to find the steady-state 60Hz and 300 Hz components of voltage and current phasors. The values of the 3 states (phasors and initial values) can be also obtained from the powergui block.
5. Open the scope and start the simulation from the Simulation/Start menu. Notice that the simulation starts in steady-state. Using the Powergui block, select Impedance vs Frequency Measurement. A new window opens.
6. The measurement will be performed for the specified frequency range vector [0: 2:1000] (0 to 1000 Hz by steps of 2 Hz). Click on the Display button. The impedance is displayed in a graphic window. Notice the series resonance at 300 Hz corresponding to the tuned frequency of the filter.

**TITLE: Simulate Steady state Analysis of a Linear Circuit using MATLAB simulink**

**GPRED/EEE/EXPT-BS(P)-12**

**Date: 18-10-2021**

**SIMULINK DIAGRAM:**



**RESULT:**

**VIVA QUESTIONS:**

1. What is steady state analysis.
2. Why steady state analysis is required.
3. What is impedance
4. What is admittance
5. Differentiate transient and steady state

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE:** Model a circuit for the nodal analysis to compute voltages v1 and v2 for a given circuit using MATLAB simulink

**GPRED/EEE/EXPT-BS(P)-13**

**Date: 18-10-2021**

**AIM:** To model a circuit for the nodal analysis to compute voltages v<sub>1</sub> and v<sub>2</sub> on MATLAB

**SOFTWARE:** MATLAB software

**THEORY:**

**NODAL ANALYSIS**

Kirchhoff's current law states that for any electrical circuit, the algebraic sum of all the currents at any node in the circuit equals zero. In nodal analysis, if there are n nodes in a circuit, and we select a reference node, the other nodes can be numbered from V<sub>1</sub> through V<sub>n-1</sub>.

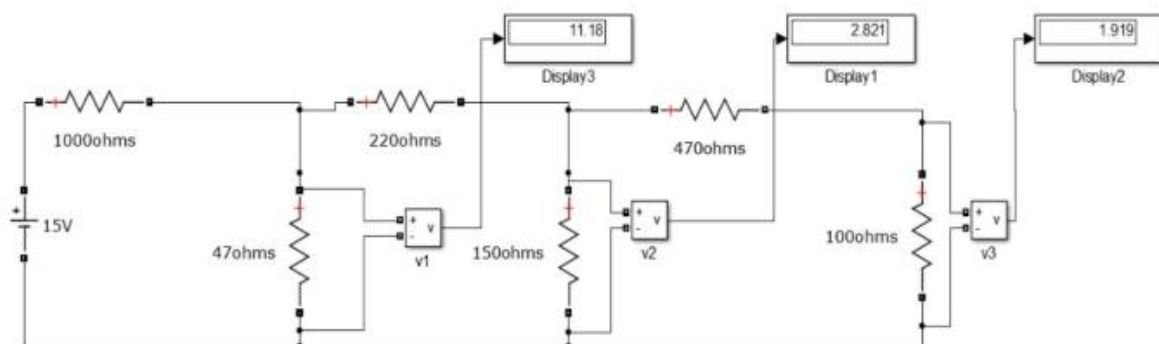
**Steps Used in Nodal Analysis**

1. Select a node as the reference node. Assign voltages V<sub>1</sub>, V<sub>2</sub>... V<sub>n-1</sub> to the remaining nodes. The voltages are referenced with respect to the reference node.
2. Apply KCL to each of the non reference nodes.
3. Use Ohm's law to express the branch currents in terms of node voltages. Node Always assumes that current flows from a higher potential to a lower potential in resistor. Hence, **current** is expressed as follows

$$I = \frac{V_{high} - V_{low}}{R}$$

4. After the application of Ohm's Law get the 'n-1' node equations in terms of node voltages and resistances.

**SIMULINK DIAGRAM:**



**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Model a circuit for the nodal analysis to compute voltages v1 and v2 for a given circuit using MATLAB simulink**

**GPRED/EEE/EXPT-BS(P)-13**

**Date: 18-10-2021**

**PROCEDURE:**

- 1) Start the Matlab software.
- 2) Open a new simulation window by clicking on simulink button.
- 3) Window that appears in simulink library browser the window contains component that can be plotted into simulink or design window.
- 4) Drag & drop the components from library.
- 5) Run the simulation.

**TABULAR COLUMN:**

Applied Voltage V (volts)	Node voltage( $V_1$ )		Node voltage ( $V_2$ )	
	Theoretical	Practical	Theoretical	Practical

**RESULT:**

**VIVA QUESTIONS:**

1. What is transient analysis.
2. Why transient analysis is required.
3. What is power GUI.
4. What is scope.
5. Differentiate transient and steady state

**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: a) Simulation of Half wave Rectifier**  
**b) Simulation of Full wave Rectifier**

**GPRECD/EEE/EXPT-BS(P)-14**

**Date: 18-10-2021**

**AIM:** Simulation of single phase half wave and full wave diode rectifier with R load in MATLAB

**SOFTWARE:-** MATLAB

**THEORY:**

**Half Wave Rectifier:-**

A rectifier is a device that converts alternating current (AC) to direct current (DC). It is done by using a diode or a group of diodes. Half wave rectifiers use one diode, while a full wave rectifier uses multiple diodes. When a standard AC waveform is passed through a half-wave rectifier, only half of the AC waveform remains. Half-wave rectifiers only allow one half-cycle (positive or negative half-cycle) of the AC voltage through and will block the other half-cycle on the DC side, during the positive half cycle of the AC voltage, the diode will be forward biased and the current flows through the diode. During the negative half cycle of the AC voltage, the diode will be reverse biased and the flow of current will be blocked

**Single Phase Full wave Rectifier:-**

Primary function of full wave diode rectifier simulation is to establish a dc level from a sinusoidal input voltage that has zero voltage. Single phase supply, is a fully controlled bridge- circuit .In the bridge circuit, diagonally opposite pairs of diodes are made to conduct, and are commutated, simultaneously.

During the first positive half-cycle, diodes D1 and D2 are forward biased and if they are triggered simultaneously, then current flows through the path L-D1-R-D2-N. Hence, in the positive cycle, diodes D1 and D2 are conducting.

During the negative half cycle of the a.c. input, diodes D3 and D4 are forward biased and if they are triggered simultaneously, current flows through the path N-D3-R-D4-L. Diodes D1, D2 and D3, D4 are triggered at the same firing angle  $\alpha$  in each positive and negative half-cycles of the supply voltage, respectively.

When the supply voltage falls to zero, the current also goes to zero. Hence, diodes D1 and D2 in positive half cycle and D3, D4 in negative half cycle turn off by natural commutation.

**TITLE: a) Simulation of Half wave Rectifier**  
**b) Simulation of Full wave Rectifier**

**GPRED/EEE/EXPT-BS(P)-14**

**Date: 18-10-2021**

The related voltage and current waveforms for the circuit are shown in the figure given below the circuit diagram;

**PROCEDURE:**

- 1) Start the Matlab software.
- 2) Open a new simulation window by clicking on simulink button.
- 3) Window that appears in simulink library browser the window contains component that can be plotted into simulink or design window.
- 4) Drag & drop the components from library.
- 5) Run the simulation.

**SIMULINK DIAGRAM: 1. HALF WAVE RECTIFIER**

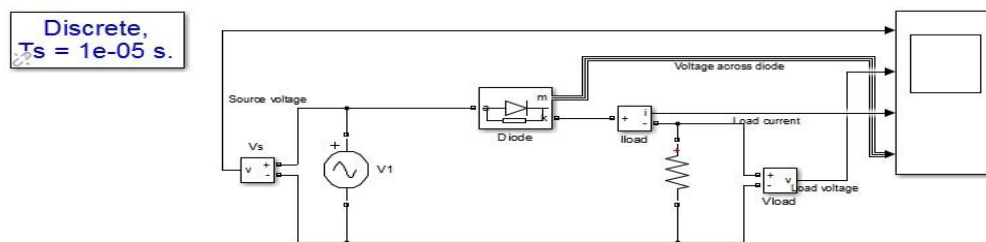


Figure1: simulink model of half wave rectifier

**EXPECTED OUT PUT:**

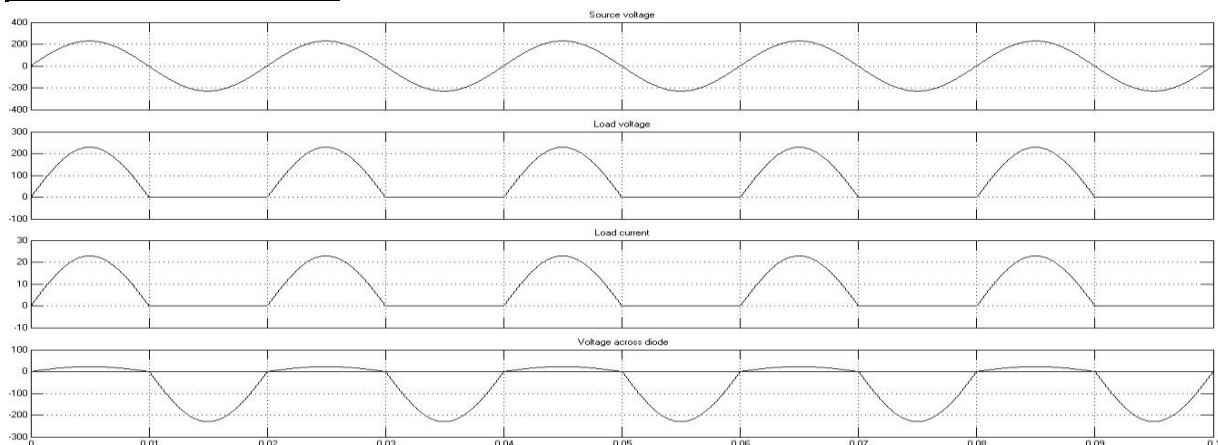


Figure 2: wave forms of Half wave rectifier in terms of input voltage, load voltage, load current and voltage across diode

**TITLE: a) Simulation of Half wave Rectifier**  
**b) Simulation of Full wave Rectifier**

**GPRED/EEE/EXPT-BS(P)-14**

**Date: 18-10-2021**

**SIMULINK DIAGRAM: 2. FULL WAVE RECTIFIER**

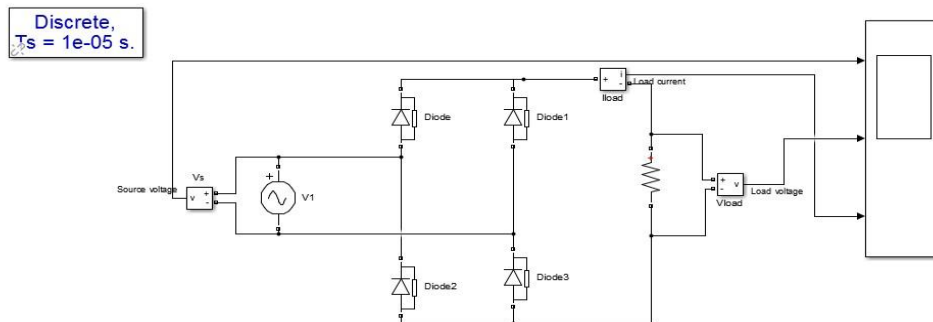


Figure 3: simulink model of Full wave rectifier

**EXPECTED OUTPUT :**

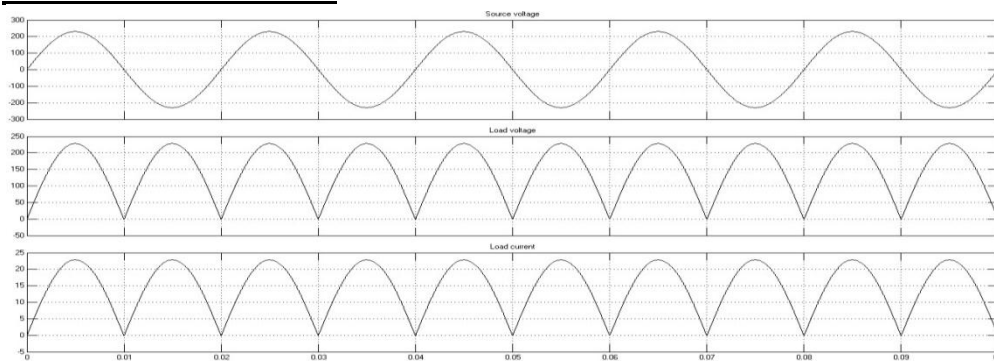


Figure 4: wave forms of Full wave rectifier in terms of input voltage, load voltage ,load current and voltage across diode

**RESULT:**

**VIVA QUESTIONS:**

1. What is a rectifier.
2. Give the expression for output voltage of half wave rectifier with R-Load.
3. Give the expression for output voltage of full wave rectifier with R-Load.
4. What is the difference between half wave and full wave rectifier.
5. What do you mean by reverse blocking state of diode.



**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL**  
**ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT**  
**BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Verification of Locus diagrams through simulation**

**GPRED/EEE/EXPT-BS(P)-15**

**Date: 18-10-2021**

**AIM:** To verify the Locus diagram through simulation

**SOFTWARE:** MATLAB

**THEORY:**

Locus diagrams are useful in determining the behaviour or response of an RLC circuit, when one of its parameters is varied while the frequency and voltage are kept constant. The magnitude and phase of the current vector in the circuit depend upon the values of R, L and C and frequency at the fixed source voltage. The path travelled by the tip of the current vector when the parameters R, L or C are varied while frequency and voltage are kept constant is called the locus diagram.

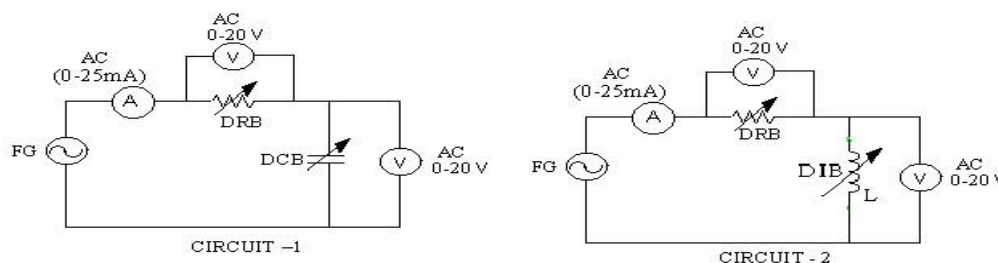
**R-Varying:**

If  $R = 0$ , then  $I = V / X_L$  or  $V / X_C$  and has maximum value. It will lag or lead the voltage by  $90^\circ$  depending on whether the reactance is inductive or capacitive. The angle  $\theta$  represents the phase angle ( $\theta = \tan^{-1}(VX / VR)$ ). As R is increased from zero value, I and  $\theta$  decrease. In the limiting case when  $R = \infty$ , then  $I = 0$  and  $\theta = 0^\circ$ . The locus of end-point qitI is a sem-circle of radius  $V / X$ .

**R-Fixed:**

If  $X = 0$ , then  $I = V / R$  and has maximum value. The current will be in phase with Voltage as it is a purely resistive circuit i.e., the phase  $\theta$  is zero. As X is increased depending on whether the reactance is inductance or capacitance the current starts lagging or leading.

**CIRCUIT DIAGRAM:**



**G.PULLA REDDY ENGINEERING COLLEGE (Autonomous): KURNOOL  
ELECTRICAL AND ELECTRONICS ENGINEERING DEPARTMENT  
BASIC SIMULATION LAB (Scheme-2020)**

**TITLE: Verification of Locus diagrams through simulation**

**GPRED/EEE/EXPT-BS(P)-15**

**Date: 18-10-2021**

**PROCEDURE:**

- 1) Start the MATLAB software.
- 2) Open a new simulation window by clicking on simulink button.
- 3) Window that appears in simulink library browser the window contains component that can be plotted into simulink or design window.
- 4) Drag & drop the components from library.
- 5) Run the simulation.

**RESULT:**

**VIVA QUESTIONS**

1. Define locus diagram.
2. Define impedance.
3. How does the voltage varies with respective frequency across inductor.
4. What is the current expression for series RL circuit.
5. What is the current expression for series RC circuit.