FAMILIARISATION EXPERIMENT

MATLAB AND LTSPICE

RITU ANN ROY GEORGE

B170106EC

S7 ECE B BATCH

MATLAB

MATLAB is a programming platform designed specifically for engineers and scientists. The heart of MATLAB is the MATLAB language, a matrix-based language allowing the most natural expression of computational mathematics.

Using MATLAB, we can:

- Analyze data
- Develop algorithms
- Create models and applications

MATLAB finds applications in many domains such as data analytics, computational Biology, computational finance, wireless communications and Internet of Things.

MATLAB: WINDOWS

Command Window

Workspace Window
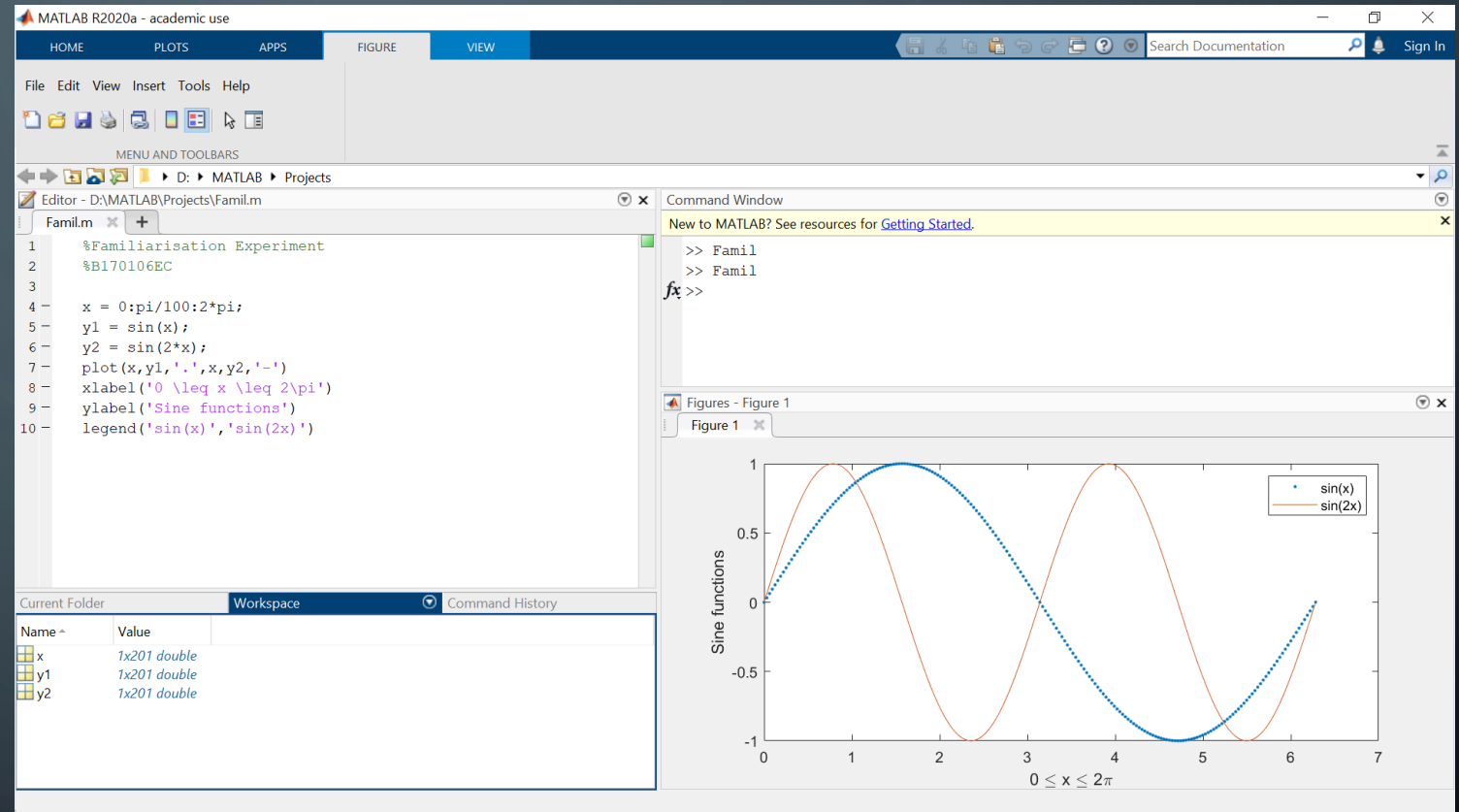
Current Folder Window

Command History Window

TOOLS

Toolboxes
- Control Systems Toolbox
- DSP Toolbox
- Image Processing Toolbox

Inbuilt Commands and Functions
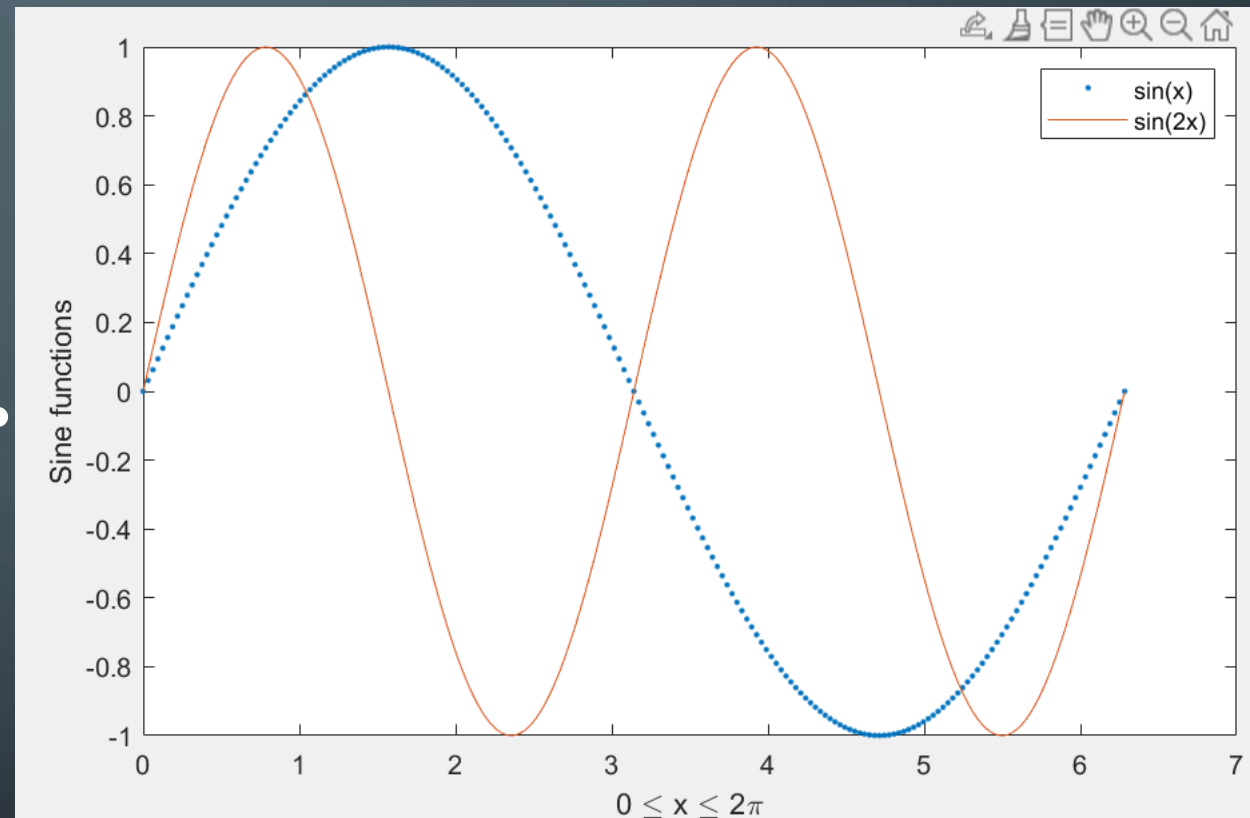- clc, help
- zeros(a,b), cos(x)

MATLAB PROGRAM

AIM: To plot a sine curve on MATLAB

THEORY: MATLAB has an inbuilt trigonometric function sin(x) that generates the curve in the specified range for x

ALGORITHM:

1. Specify range for input x

2. Generate two sine curves sin(x) and sin(2x)

3. Plot both the figures on the same plot.

OBSERVATION

LTSPICE

LTspice is a SPICE-based analog electronic circuit simulator computer software, produced by Analog Devices (originally by Linear Technology). It is the most widely distributed and used SPICE software in the industry.

Advantages of Ltspice:

- Outperforms many simulation solutions in the market
- Not hobbled by any arbitrary limits - Stable circuit simulation with unlimited number of nodes
- Freeware

LTSPICE SIMULATION

AIM: To simulate a resistive current divider circuit

THEORY: For 2 resistors in parallel, current through each resistor is divided in the ratio:

$$\frac{I_2}{I_1} = \frac{R_1}{R_2}$$

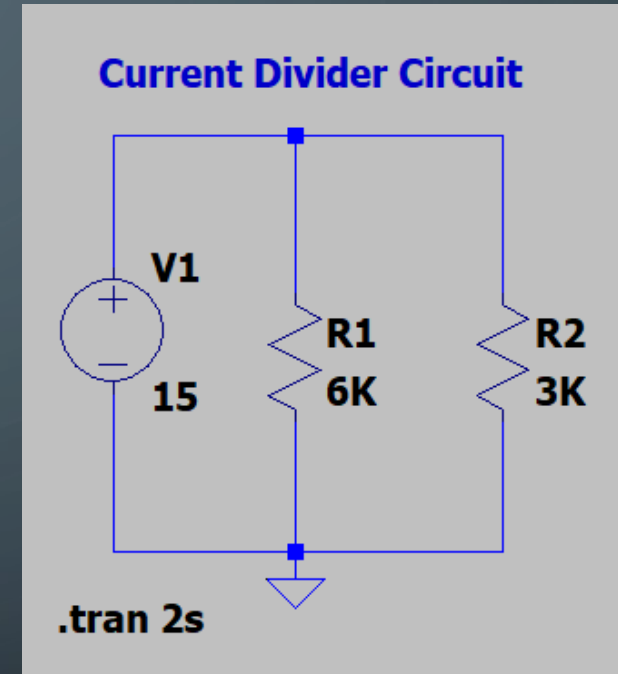$$I_{(R_1 \, or \, 2)} = I * \frac{R_{2 \, or \, 1}}{R_1 + R_2}$$

For a voltage source of 15V and

$$if \ R_1 = 6K\Omega \ and \ R_2 = 3K\Omega, then$$

$$R_{net} = 3K\Omega, net \ current \ I = 7.5mA$$

$$I_1 = 2.5mA \ and \ I_2 = 5mA$$

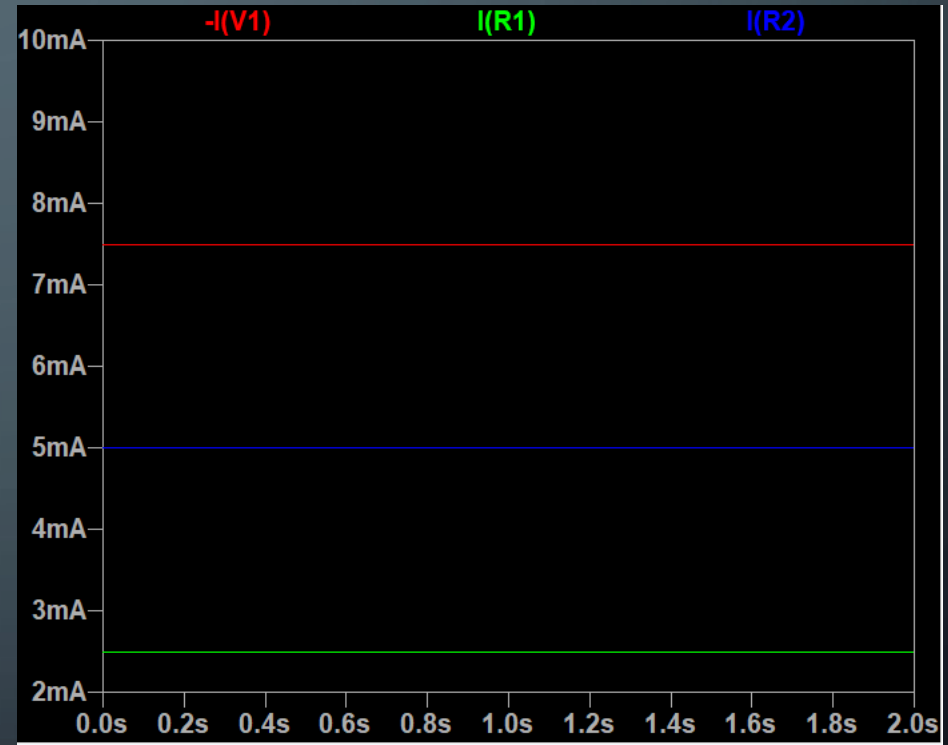Current Divider Circuit

V1
+
−
15

R1
6K

R2
3K

.tran 2s

LTSPICE SIMULATION

ALGORITHM:

1. Connect the components.

2. Switch on the voltage supply

3. Measure the current flowing through each resistor and plot the figures on the same plot.

RESULT: The simulation values agree with the calculated current values.

OBSERVATION

CONCLUSION

INFERENCE:

1. The current through the voltage source is shown to be of negative value since LTspice assumes energy used by loads as positive and that used by sources as negative.

2. LTspice produces fast, accurate simulation results.

3. Both MATLAB and LTspice are immensely helpful for circuit analysis.

RESULT:

Familiarised with MATLAB and LTspice by performing simple simulation experiments.

ADDITIONAL MATERIAL: MATLAB CODE:

```
1       %Familiarisation Experiment
2       %Ritu-B170106EC
3 —     clear; close all; clc;
4 —     x = 0:pi/100:2*pi;
5 —     y1 = sin(x);
6 —     y2 = sin(2*x);
7 —     plot(x,y1,'.',x,y2,'-')
8 —     xlabel('0 \leq x \leq 2\pi')
9 —     ylabel('Sine functions')
10 —    legend('sin(x)','sin(2x)')
```