Familiarization Experiment

EC4091D Communication Engineering Lab II

sayed_b170011ec@nitc.ac.in

Department of Electronics and Communication Engineering
National Institute of Technology, Calicut

September 21, 2020

AIM

Matlab

LTspice

AIM

▶ The objective of this experiment is to get familiarized with the MATLAB environment, it'slanguage and LTspice circuit simulation tools.

Matlab

- ▶ Matlab is a computing environment that comes with a programming language.

- ▶ Matlab can be used to perform numerical computations and graphical visualizations

- ▶ The Matlab environment generally consists of

  1. Command Window : Commands to be executed are entered here

  2. Directory Explorer : Current Directory files are shown here

  3. Editor : Program files are editted here

  4. Workspace : Current program variables can be viewed here

Matlab (cont.)

(a) Command
Window
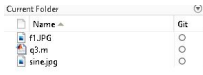
(b) Directory
Explorer

(c) Workspace

(d) Editor

Figure 1: MATLAB Environment

Matlab (cont.)

▶ Here we have a MATLAB generated plot of a cosine wave $cos(2\pi ft)$, with frequency f $= 1000Hz$

▶ Below it is the same cosine wave sampled at a frequency $f_s = 200Hz$, the samples(red) are plotted over the unsampled cosine wave(green)
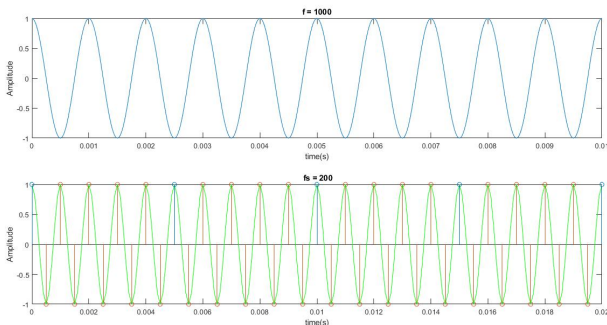

Figure 2: A cosine wave of frequency $1000Hz$ (Top); A cosine wave ($f = 1000Hz$) sampled at $f_s = 200Hz$

LTspice

- ▶ LTspice is a SPICE-based analog electronic circuit simulator computer software, produced by semiconductor manufacturer Analog Devices.

- ▶ It is the most widely distributed and used SPICE software in the industry.

- ▶ LTspice provides the following features

  1. Schematic Capture : Design circuits by dragging and connecting components from a component library

  2. Simulator : Simulate circuit behavior after giving inputs and specifying circuit component parameters

  3. Waveform Viewer : View results of simulation graphically, can view waveforms at multiple nodes in a circuit

LTspice (cont.)

- Figure 3(a) shows the schematic of an RC Low pass filter, with an input sine wave of frequency $10KHz$ and amplitude $1V$.

- Figure 3(b) shows the corresponding waveforms under transient analysis. As expected, the output sine wave (green) has a lower amplitude than the input sine wave (blue) since it's frequency is high ($10KHz$)
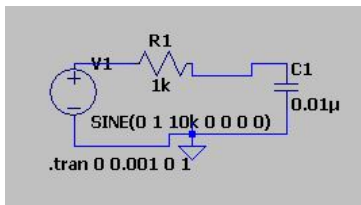
(a) RC Low Pass Filter Schematic
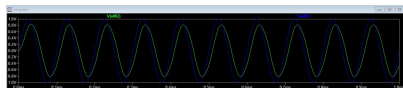
(b) Output Waveform to an Input sine wave of frequency $10KHz$

Figure 3: Transient Simulation of an RC low pass filter