Q3 MATLAB and Simulink Basics

Prove your skill set in using tools for analytical calculations.

Q3.6 Importing LTSpice Data to MATLAB

I can import simulation data (from LTSpice or equivalent) to MATLAB and plot the function.

I used my schematic from Q1.3 of the circuit simulation proof of skills (shown in the image of circuit and LTspice graph below (A0)) and created a graph of the voltage across R1 by running the simulation. I then exported the simulation as a text file and used the import data function in MATLAB. I then changed the output type from table to column vector and plotted the voltage versus time using the plot() function. This then created the graph shown below (A1).

A0:
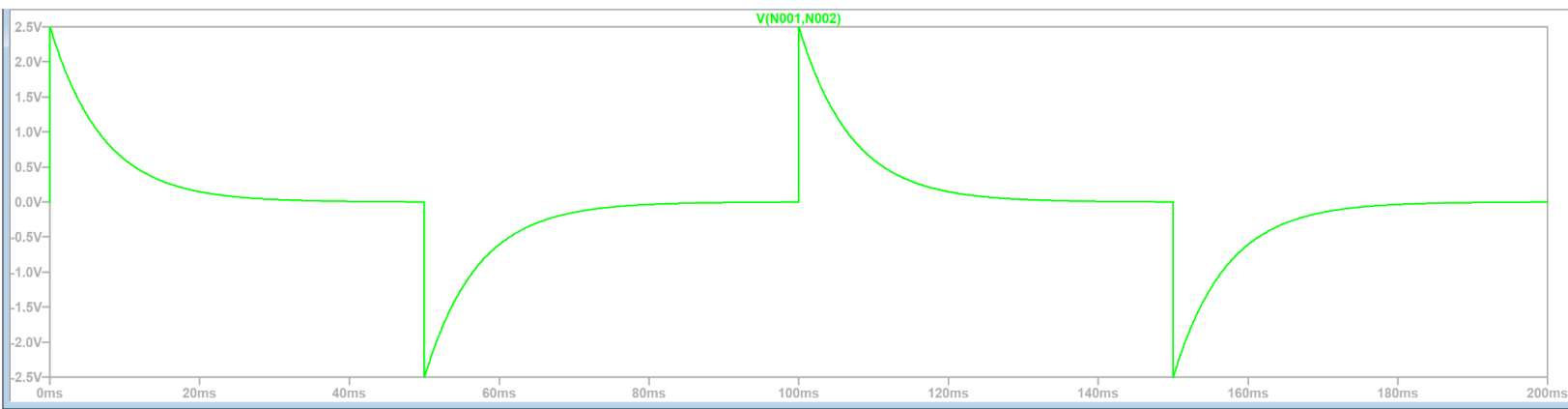
.tran 0 200m 0 0.1u
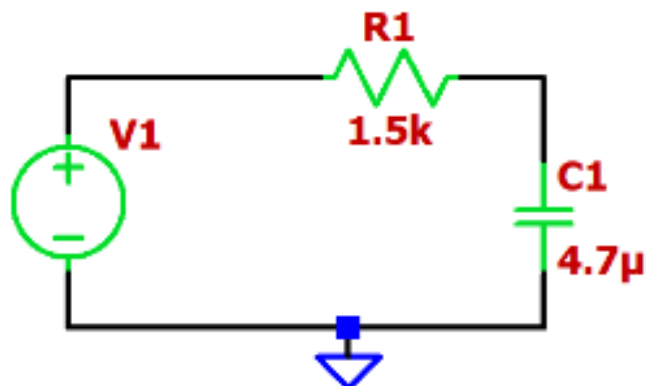
PULSE(0 2.5 0.1p 0.1p 0.1p 50m 100m 2)

(A1):

(line thickness was increase for better visibility)

Editor - C:\Users\martif5\Documents\MATLAB\GraphofR1VolatageVSTime.m

| GraphFromADALM1000.m | × | RegresionLine.m | × | MatrixPart4.m | × | MatrixP |

```
1    plot(time, VN001N002, "LineWidth",4);
2    xlabel("Time (MS)")
3    ylabel("Voltage (V)")
4    grid on
5
```

Command Window

```
>> GraphofR1VolatageVSTime
fx >>
```