“grlc3x3x2.mat” contains the MNA formulation of the circuit represented in the SPICE netlist “grlc3x3x2.sp”. After you load the MATLAB file “grlc3x3x2.mat”, you should see variables such as “G”, “C”, “B”, “u” in your MATLAB workspace, representing the MNA formulation of the circuit as
\[ Gx(t) + Cdx(t)/d(t) = Bu(t) \]

There are two voltage sources, both with DC voltage value 1v, as shown in “u” variable or in SPICE netlist. There is only one current source, with changing current values. The current values are shown in the SPICE netlist as piecewise linear (PWL) representation, and they have already been transformed into “u” variable also: “u{3, 1}” contains two columns, with the first column as time points, and the second column as current values at the corresponding time points. You need to do interpolation to handle the current waveform in PWL style.

Please write a code using Trapezoidal (Trap) rule to do transient simulation of the circuit. Your code should be able to plot one figure, showing the voltage waveform of the current source in time domain, from time 0ns to 200ns. You can freely choose the time step \( h \), but make sure the accuracy is kept.

Using MATLAB for this project is recommended. The code for this project is usually less than 40 lines in MATLAB.

Hint: you can use \( y(t) = B^T x(t) \) to get the outputs at all three sources, but you have to find out which one is the voltage of the current source (the other two are currents of the two voltage sources). You can also check your results by comparing with SPICE (PSPICE, HSPICE, Cadence Spectre, and whatever you SPICE level simulator installed in your computer or in your lab server), but this is not required.

The deadline for your code upload is 23:59, October 25th (Wednesday).