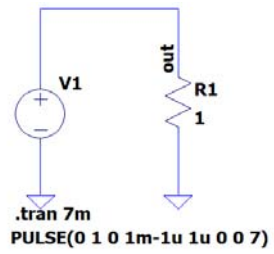
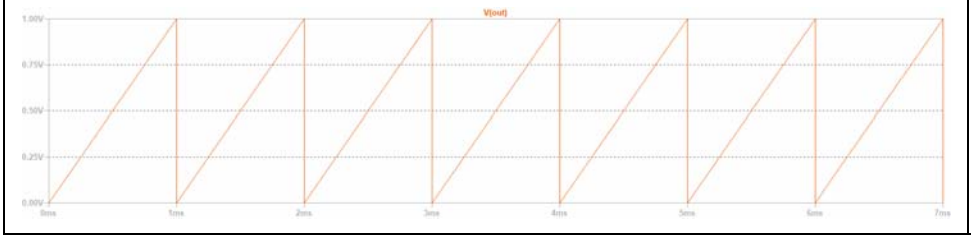
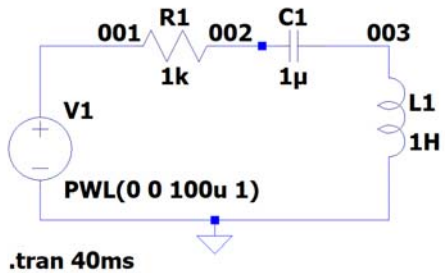
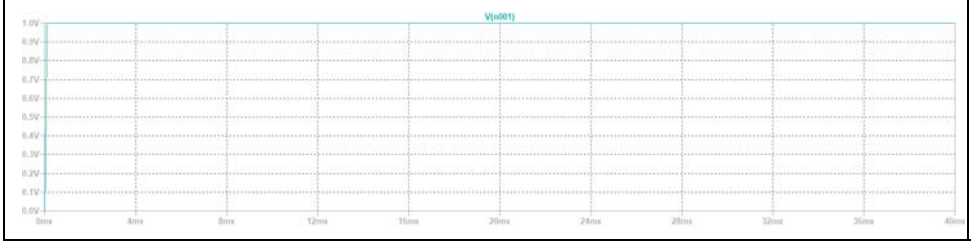
1.6.

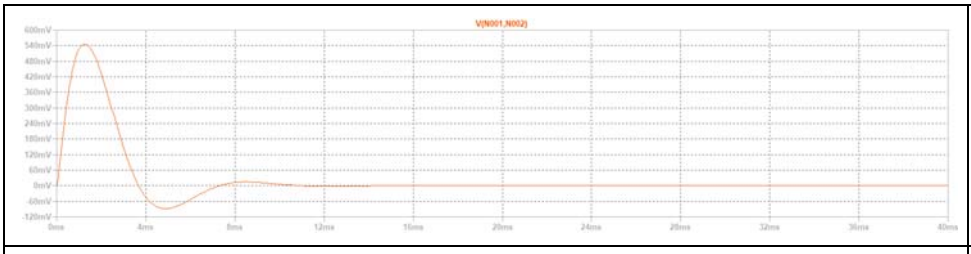
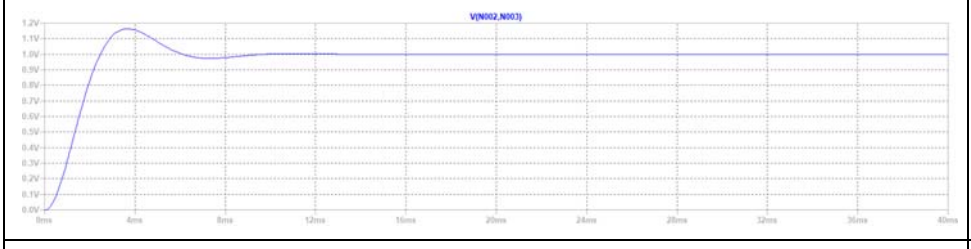
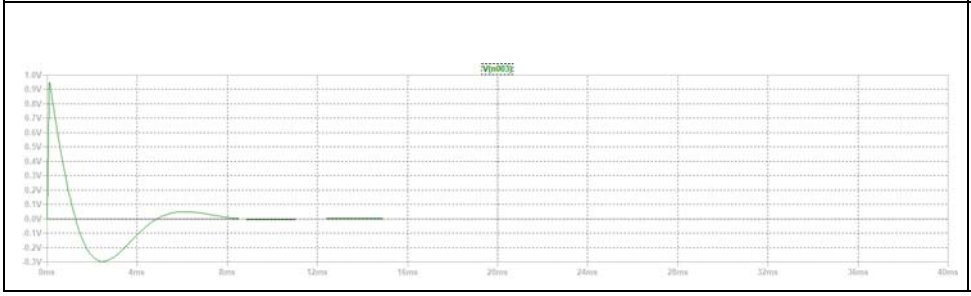
Using the PULSE source statement of LTSpice, together with the circuit setup shown in Fig.P1.3, generate the saw-tooth voltage waveform shown in Fig. P1.6. Verify your results by plotting the voltage across the 1-ohm resistor for at least 6 cycles of its waveform.

	1V saw-tooth voltage waveform of 1ms period, the voltage drop for 1us and increased for 1ms.
	The voltage across the 1-ohm resistor for 7 cycles.

1.17.

For the second order RLC circuit shown in Fig. P1.17 subject to a 1 V step input, simulate the transient behavior of the circuit and plot the voltage waveform that appears across each element for about 40 ms. Use a time step of no more than 100 us.

	1 V step input with a time step of 100 us.
	Input voltage

	Resistor voltage vR
	Capacitor voltage vC
	Inductor voltage vL

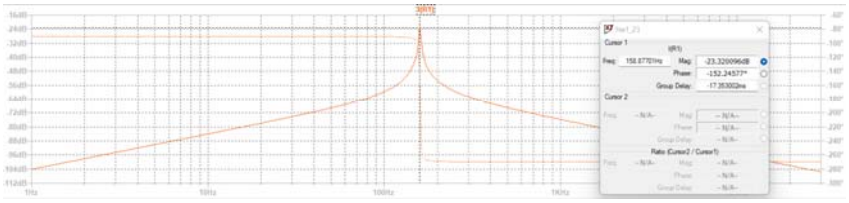
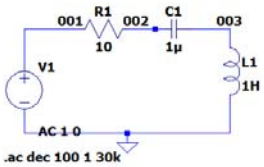
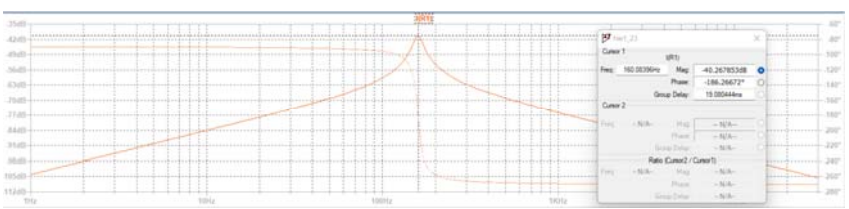
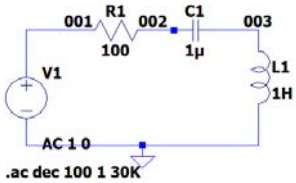
1.23.

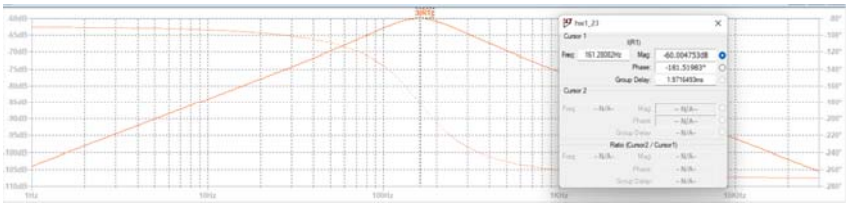
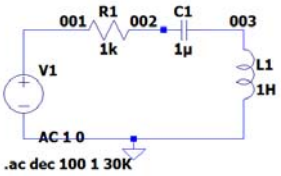
Compute the frequency response behavior of the RLC circuit shown in Fig.P1.17 with R having values of 10, 100 and 1 k-ohm.

Plot the magnitude and phase response of each case and compare them. Select an appropriate frequency range and number of points that best illustrate your results.

Frequency range: (1,30k) ; Number of points: 100

Resonance frequency (fr) = 1/2*pie*SQRT(LC) = 159.15(1/s)

10ohm	Resonance frequency = 158.87701(1/s) 	
100ohm	Resonance frequency = 160.08396(1/s) 	

1kohm	Resonance frequency = 161.28082(1/s) 	

Reflection

This is our first time using LTSpice, I found it hard to conduct and simulate circuit, so I spent a lot of time reviewing and search for information; however, LTSpice is indeed really useful to present and simulate the experiment result. There are still lots of places for us to be familiar with, and I believe that those tools in LTSpice will help me a lot in the future, too.