Class:	EE221 Circuits II - 1001	Semester:	Fall 2023

Points		Document author:	INarek Kalikian
		Author's email:	kalikn1@unlv.nevada.edu
		Document topic:	Prelab & Postlab 8

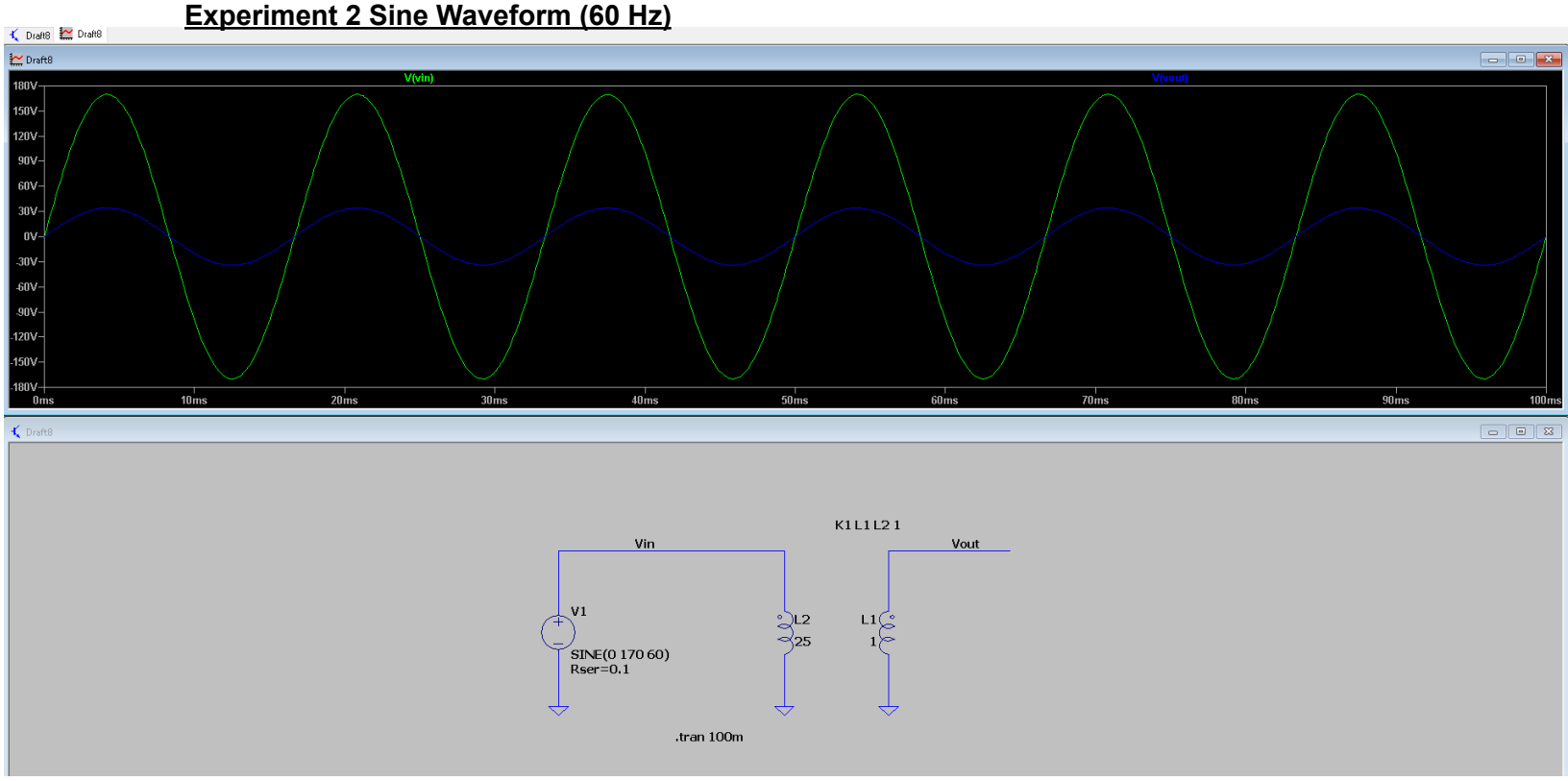
Instructor's comments:

Introduction / Theory of operation

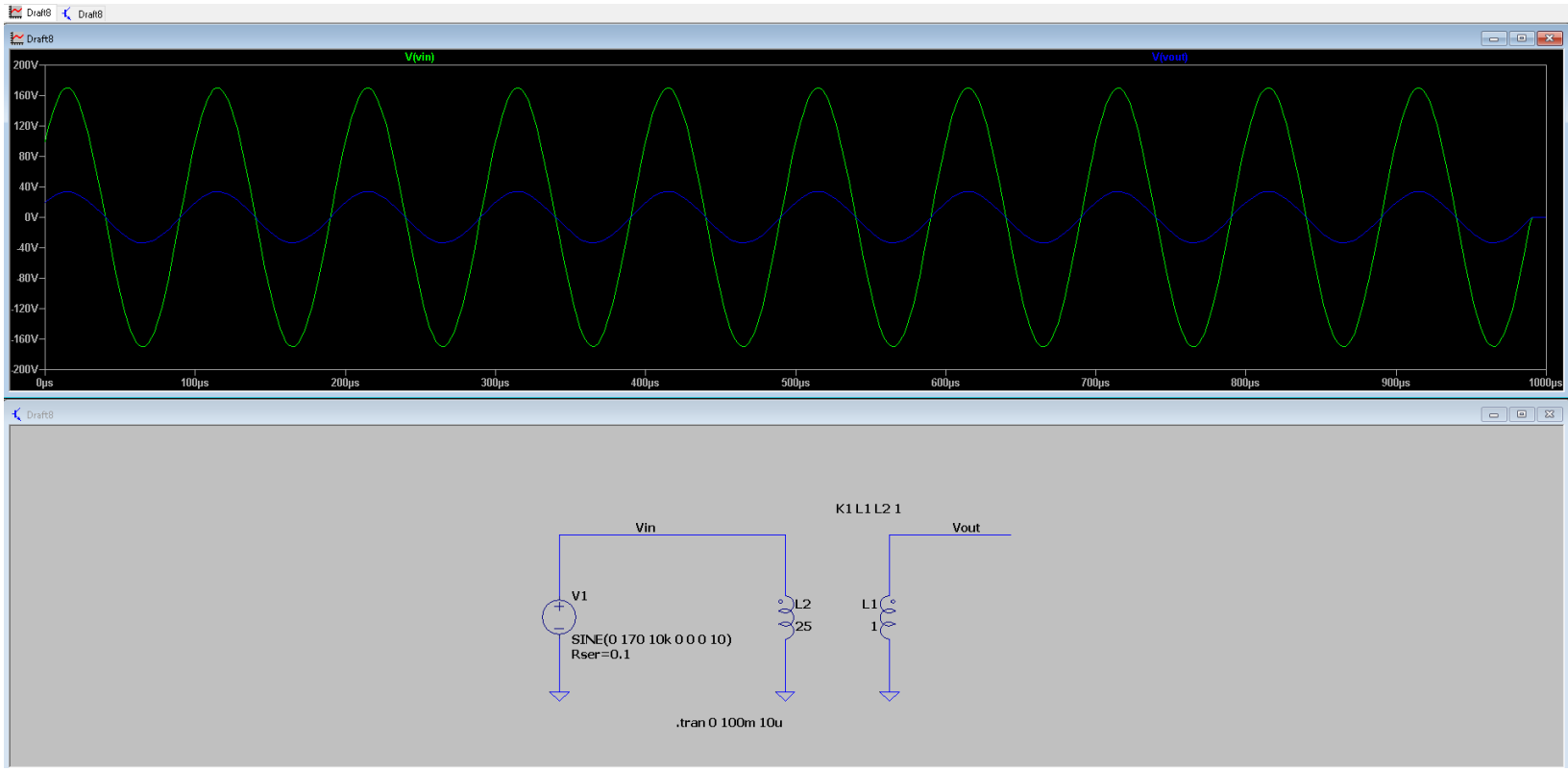
In this lab I learned how transformers work, the ratios between them, and how to calculate voltages at specific nodes in these types of magnetically-coupled circuits. We weren't able to apply these in-person so we simulated all the experiments using LTspice instead.

Prelab main content

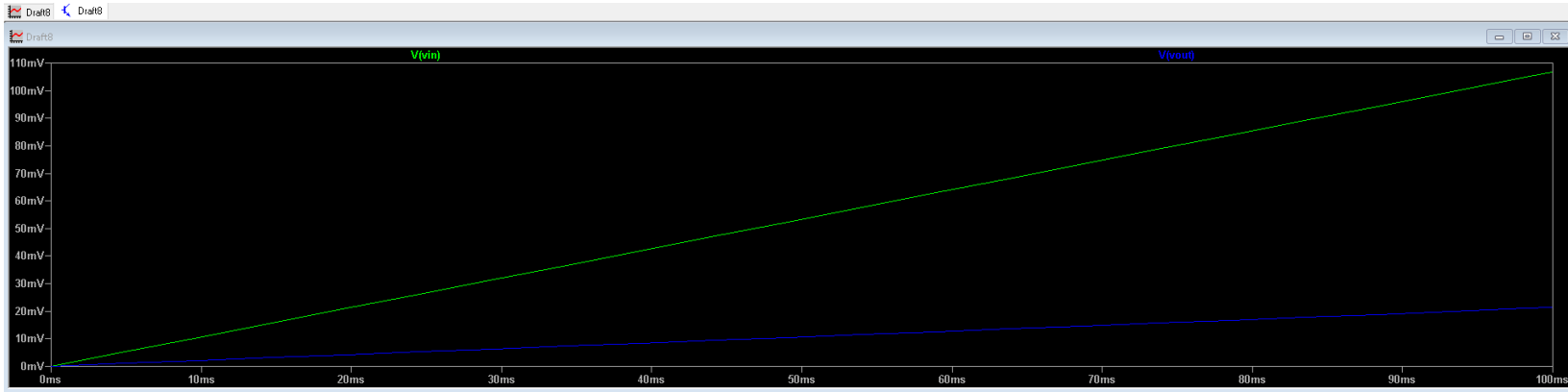
Experiment 2 Sine Waveform (60 Hz)

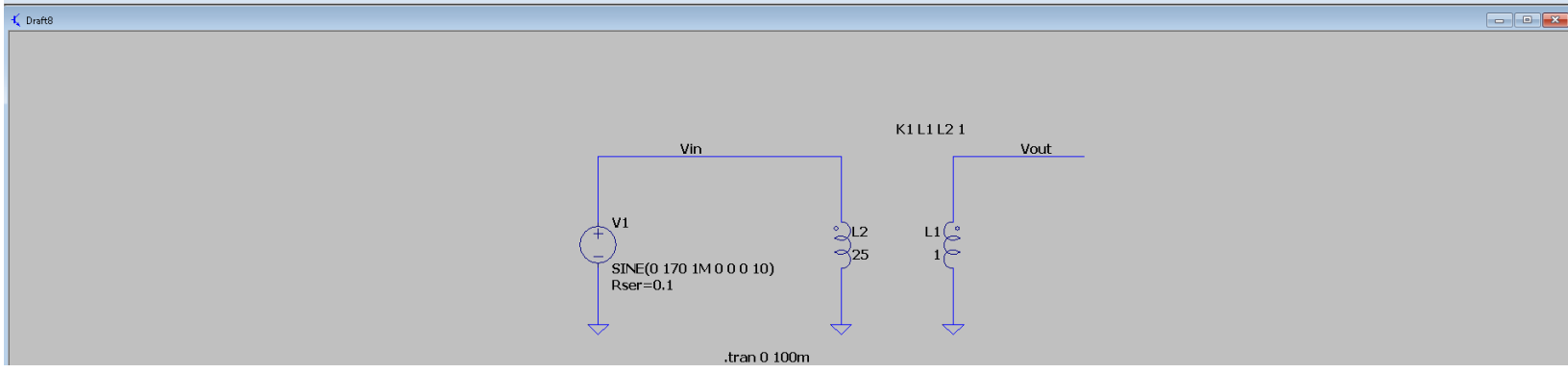


Experiment 2 Sine Waveform (10k Hz)

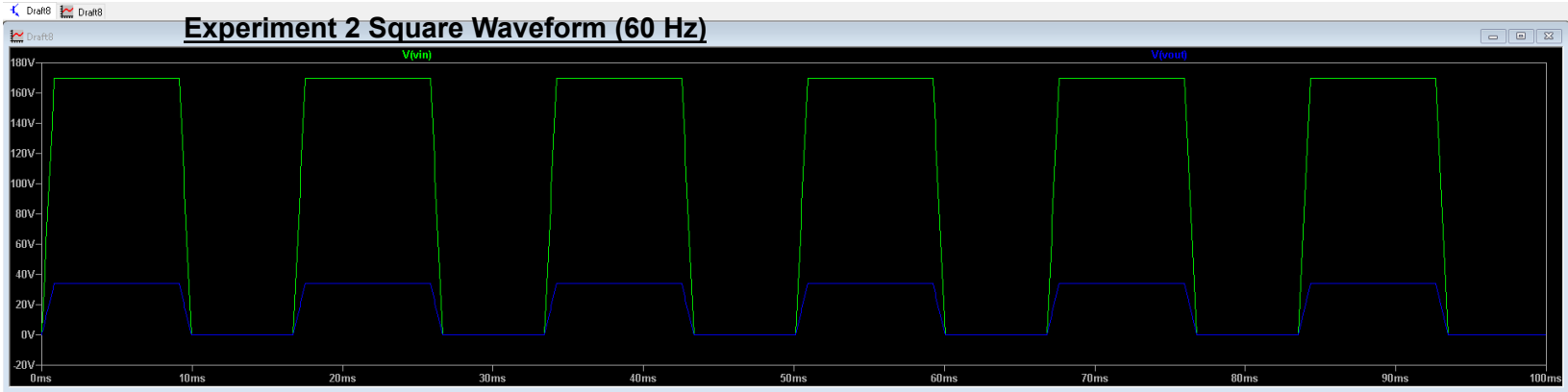


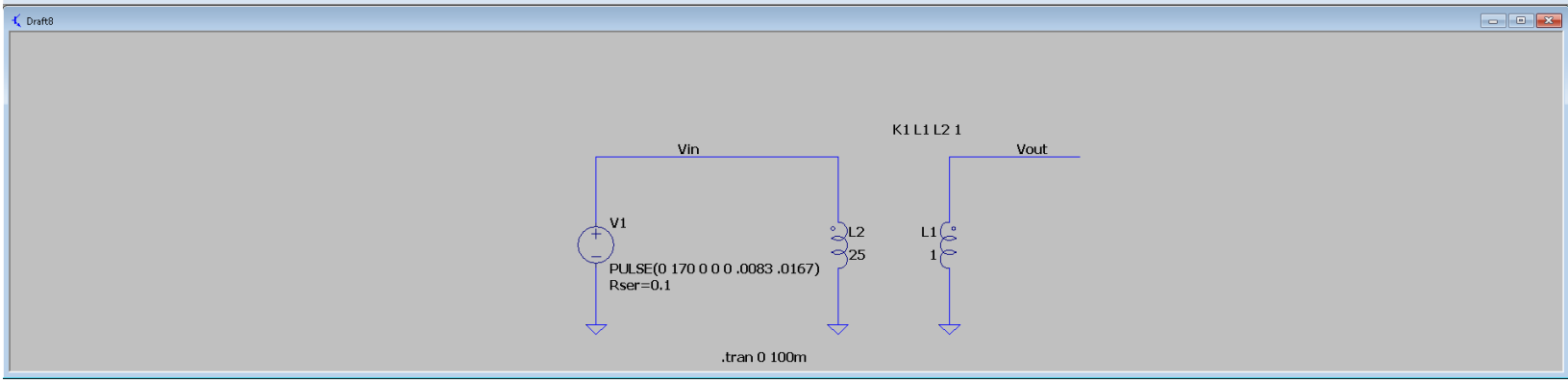
Experiment 2 Sine Waveform (1M Hz)



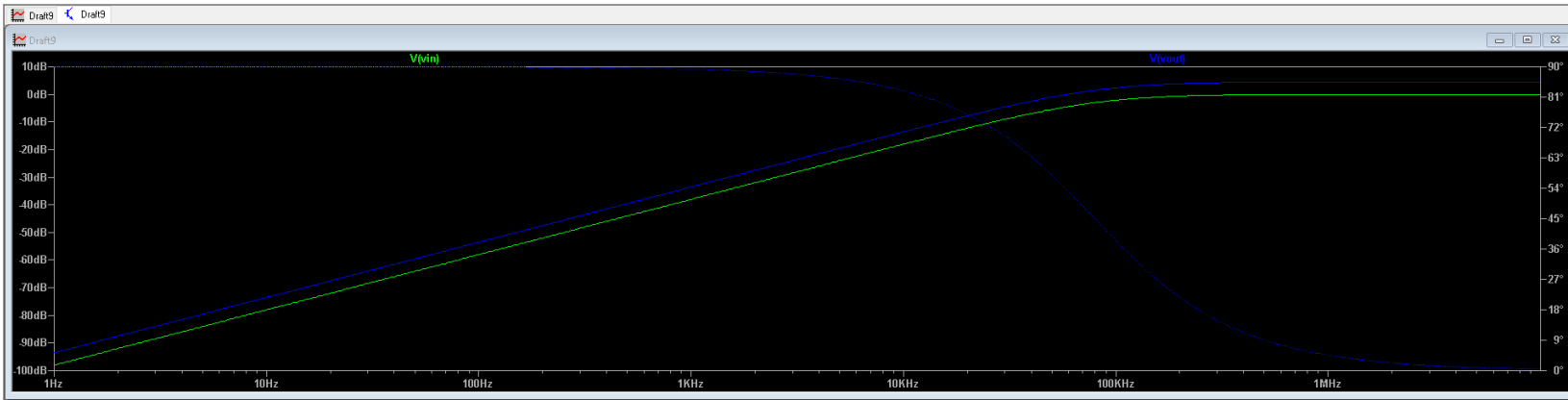


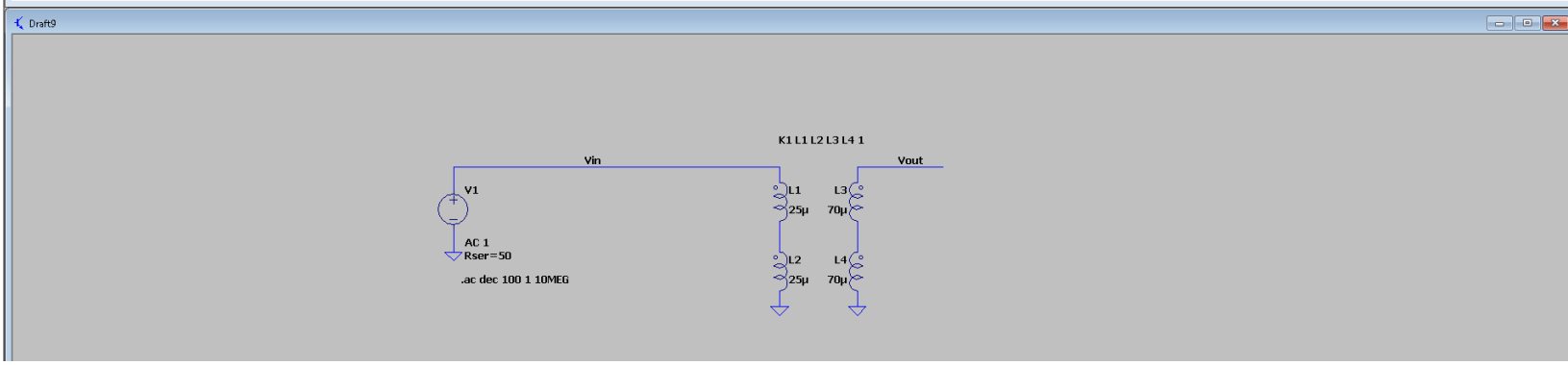
Experiment 2 Square Waveform (60 Hz)





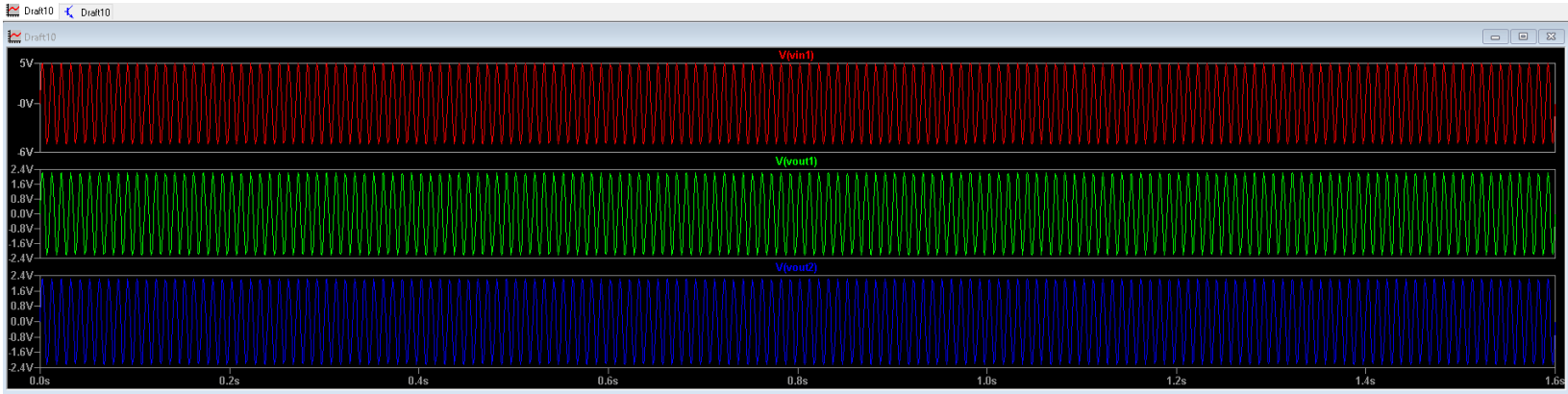
Experiment 3 AC Waveform

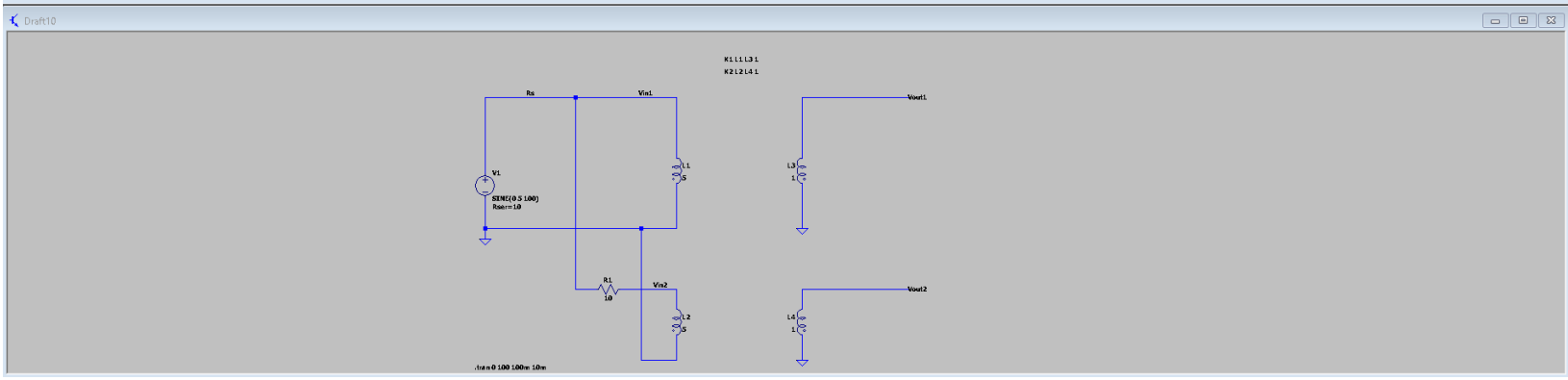




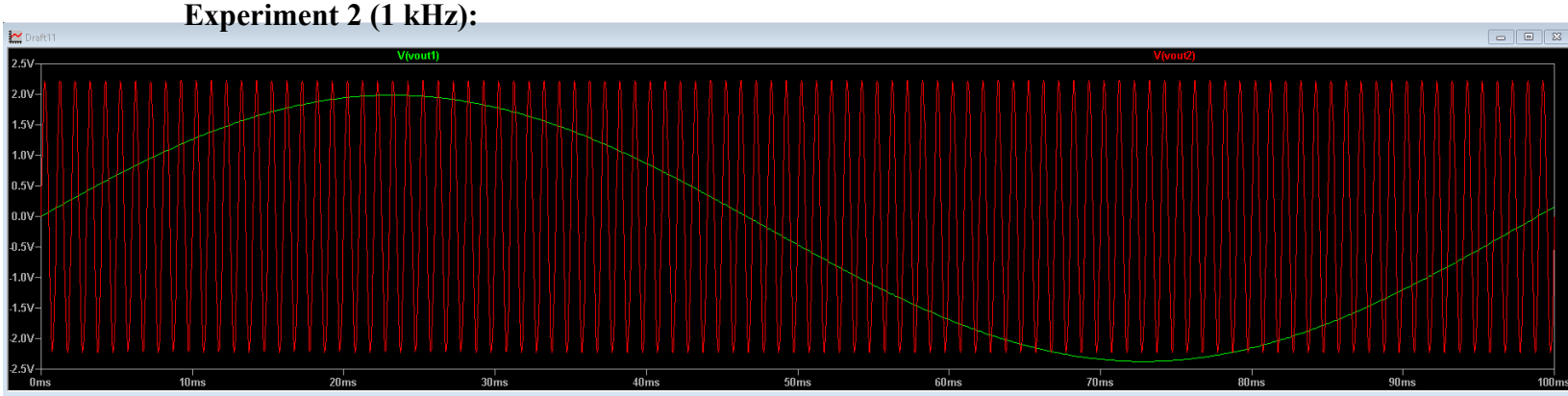
3. Experiment Results

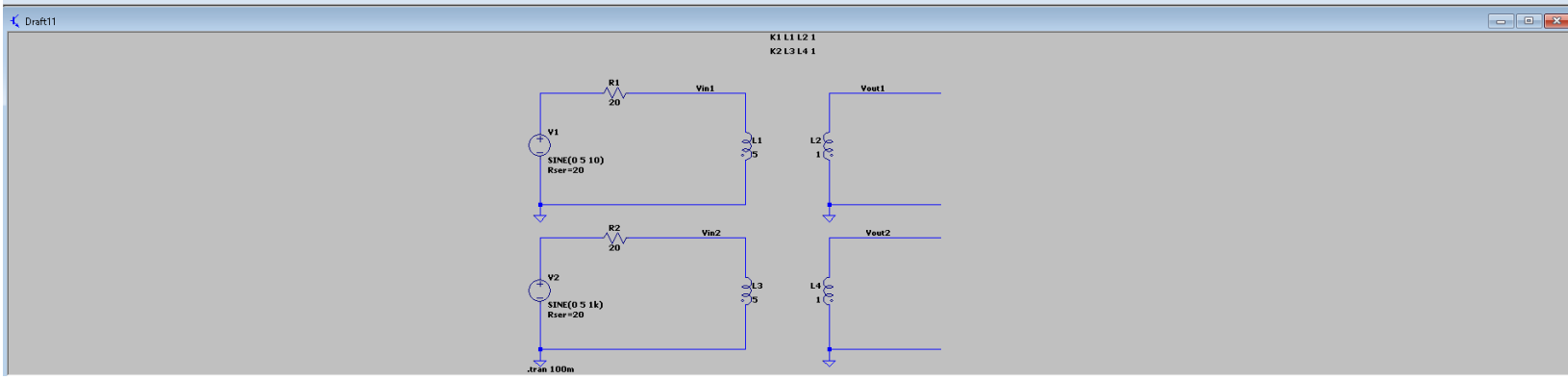
Experiment 1 Waveforms:



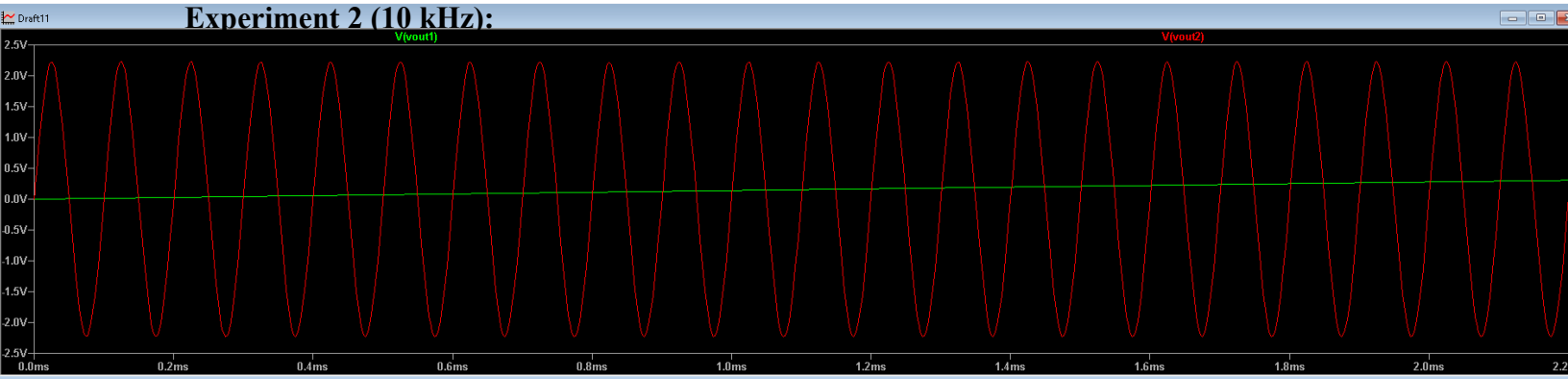


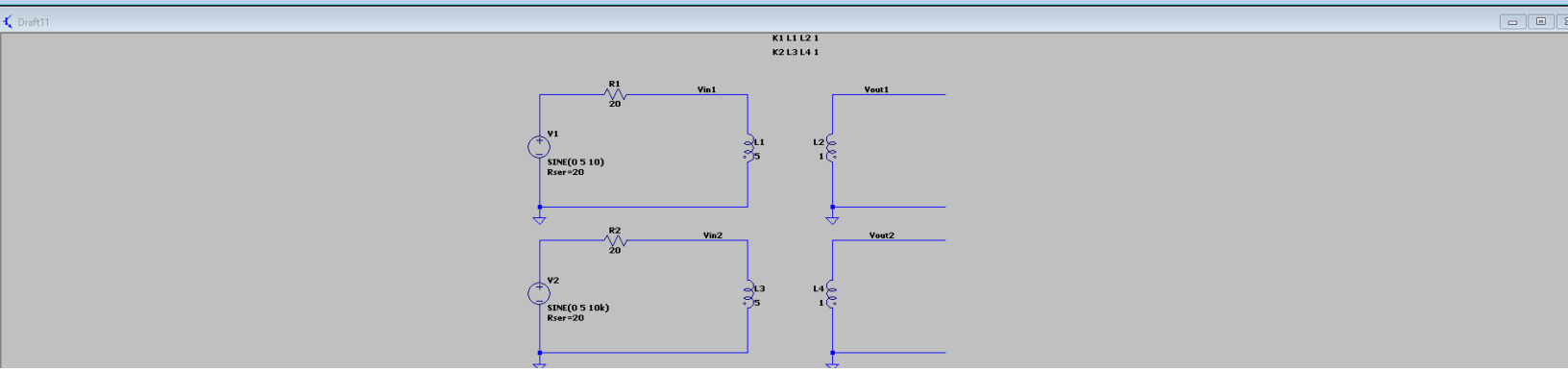
Experiment 2 (1 kHz):



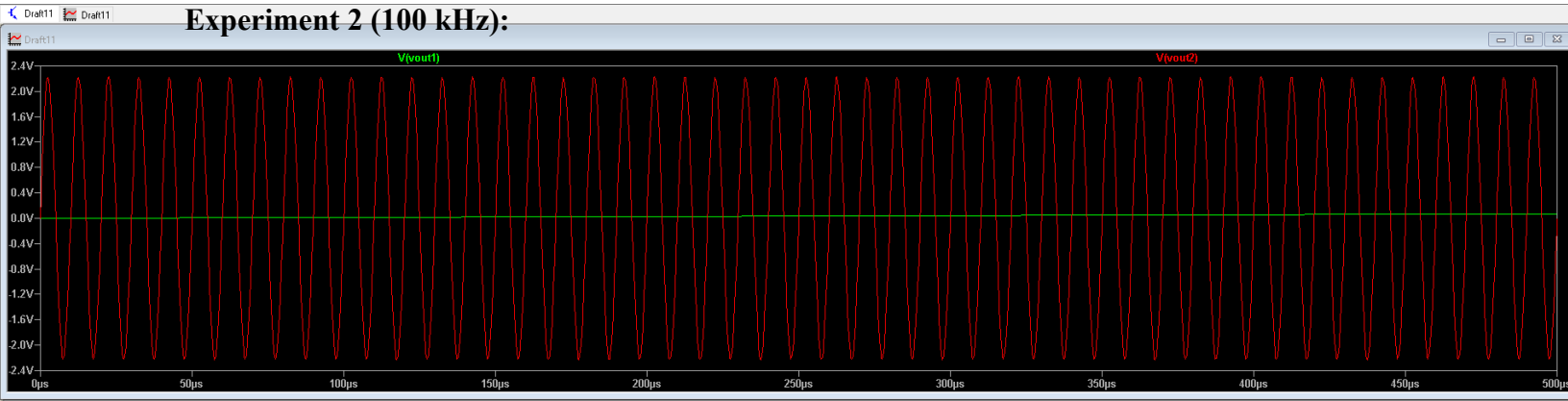


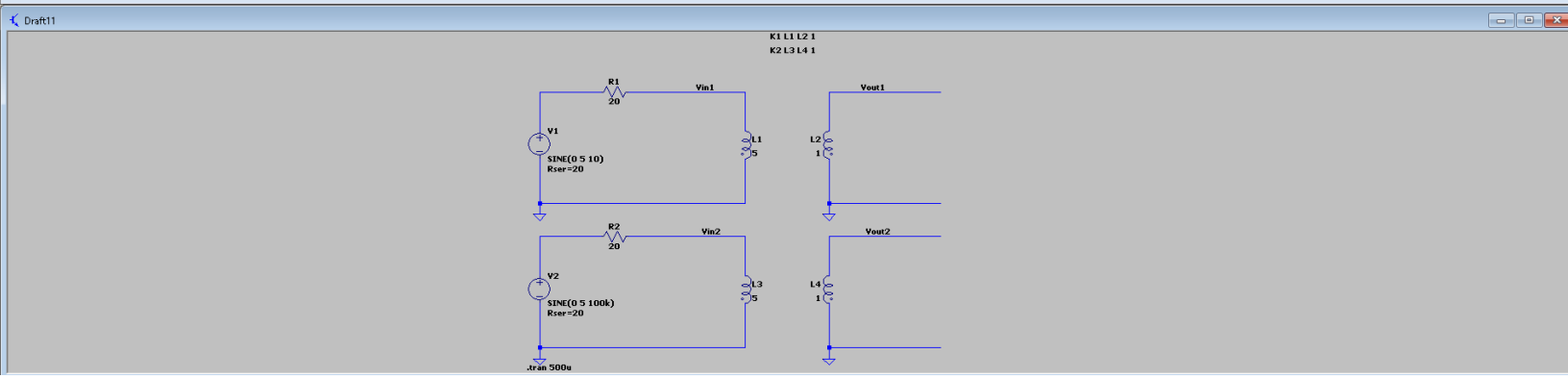
Experiment 2 (10 kHz):



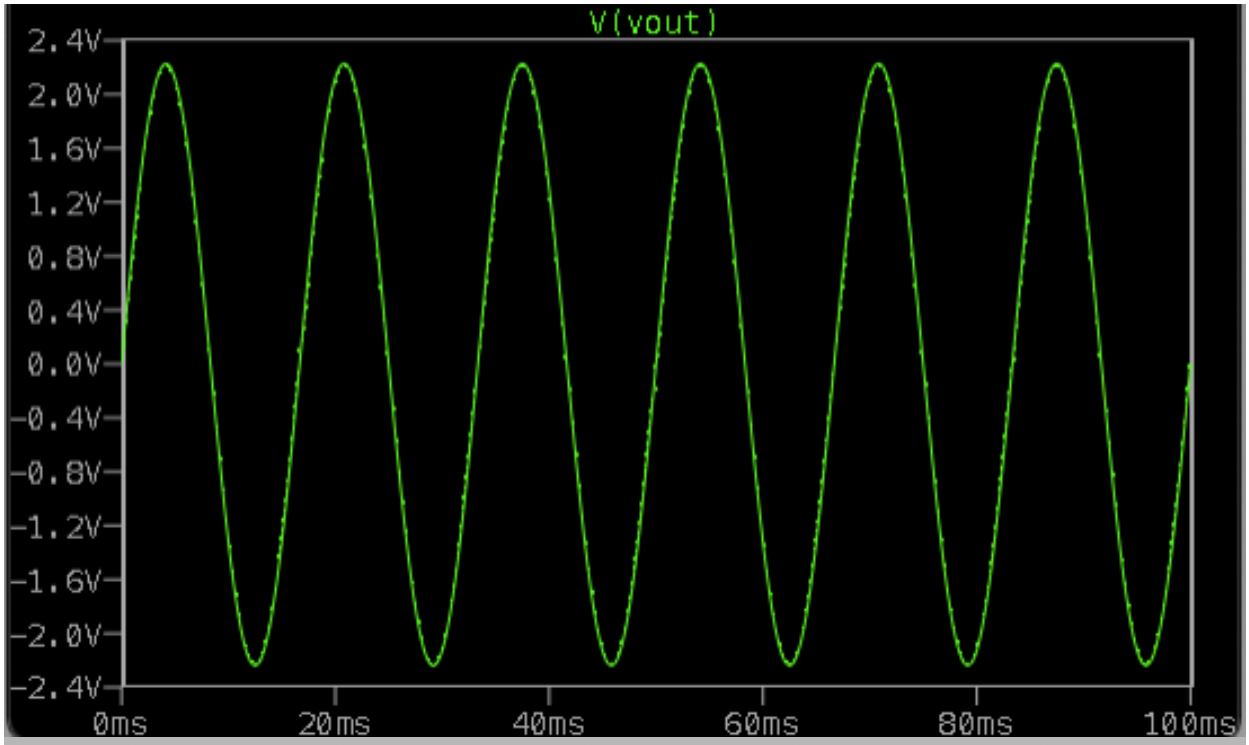


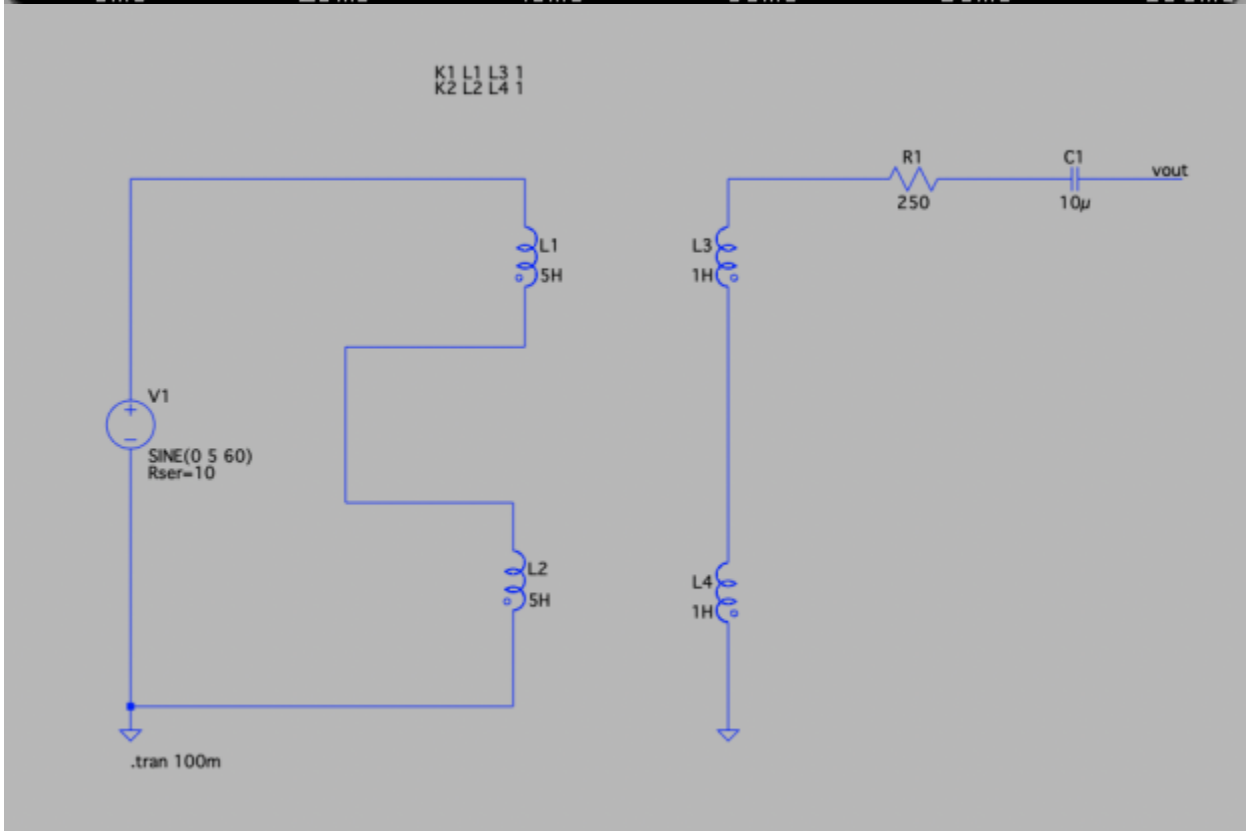
Experiment 2 (100 kHz):

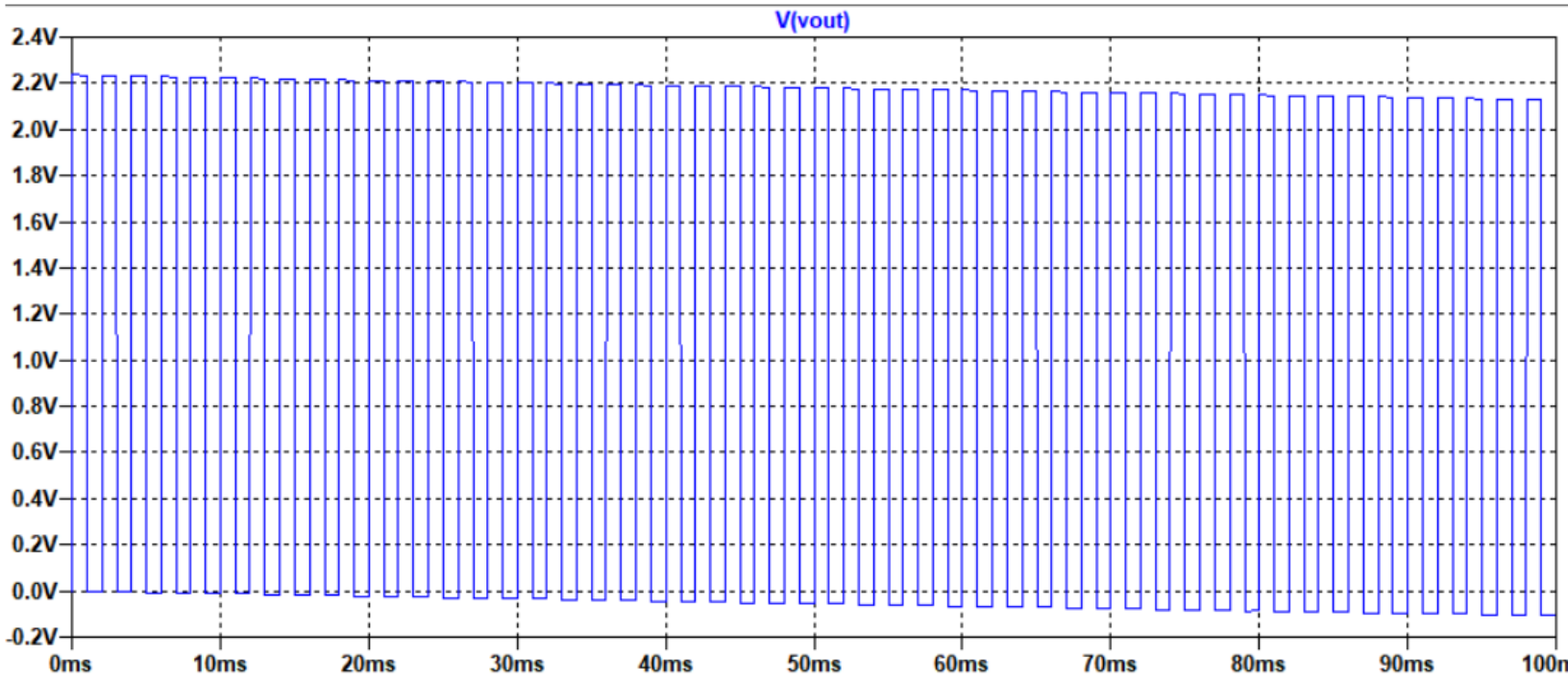




Experiment 3:







4. Questions

Question 1: We can see that both output voltages share the same value by looking at the sine waveform results from experiment 1. This is because the circuits themselves and their corresponding voltages started as pretty much the same. The difference in the measured values came from switching the frequencies.

Question 2: In the second experiment, we can see that the output voltages became closer to sinusoidal waves the more the frequency was increased. From this we can conclude that the amplitude being output is very closely related to the frequency.

5. Conclusions

The objective of this lab was to learn about transformers, how they work, and what their implementation and practical application into a real circuit looks like. Unfortunately, we were not able to see the real-life implementation due to external factors so we had to settle for simulating these circuits exclusively on LTspice. We didn't face many issues because the LTspice simulations were pretty straightforward and we had experience using the magnetically-coupled inductors in the prelab.