Name: Aryaman Mishra
Reg. no: 19BCE1027
Date: 23-08-2021

EXPERIMENT NO: 4

Design and Troubleshooting of Bridge Rectifiers with RC filters

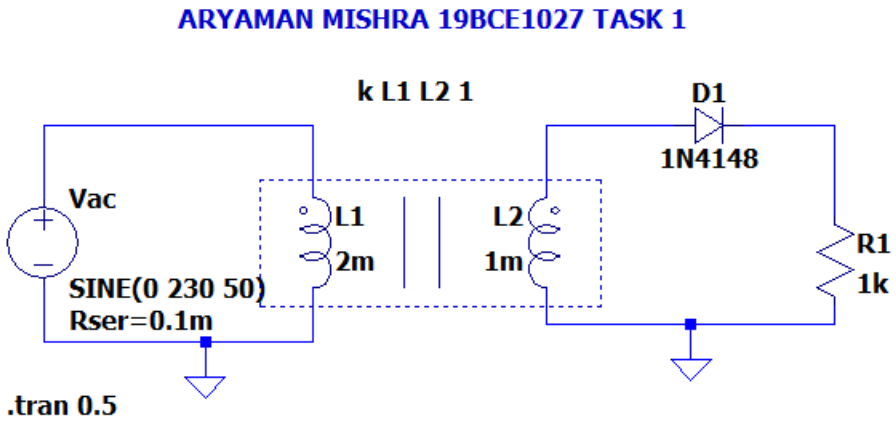
Aim: Design Half wave rectifier and plot input voltage and output voltages

Software used: LTSpice

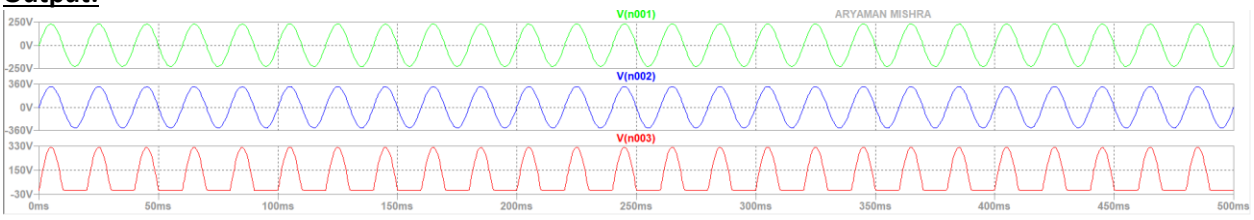
Components required: Resistors, voltage source, inductors, diode.

Task 1.1: In phase windings (2m:1m)

Circuit:

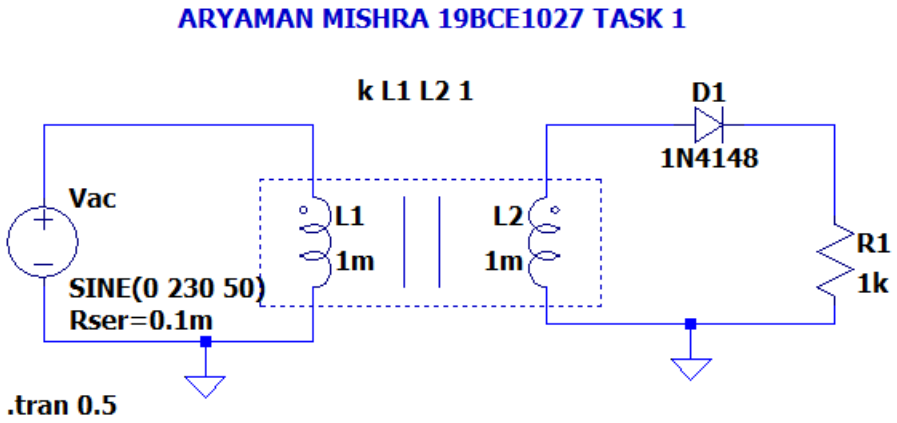


Output:

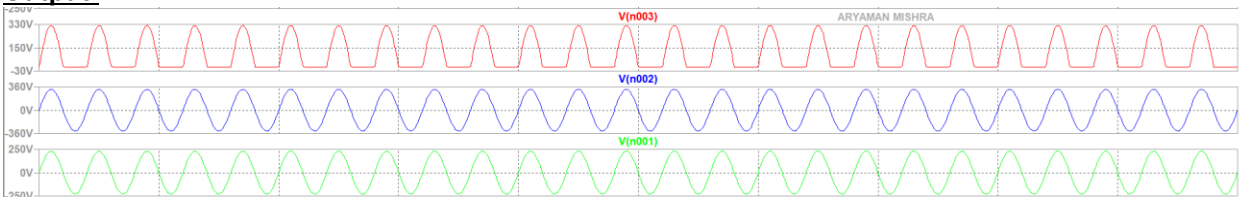


Task 1.2: In phase windings (1m:1m)

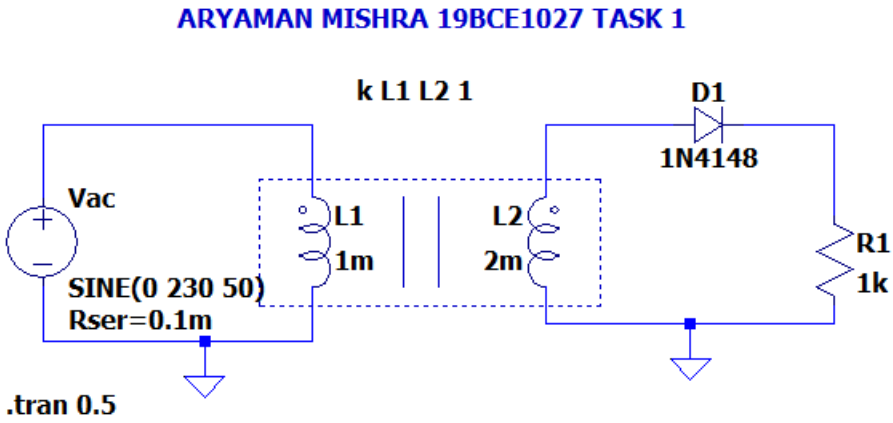
Circuit:



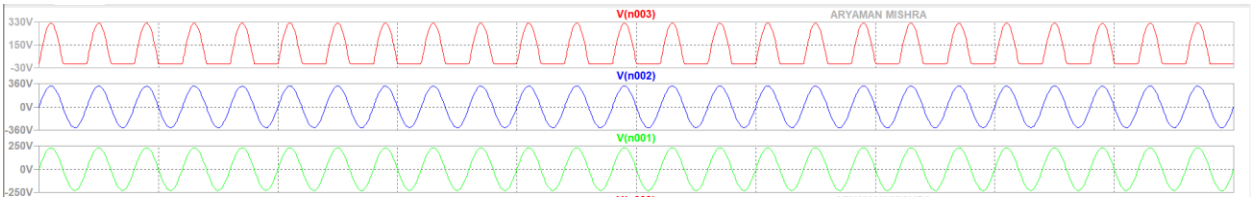
Output:



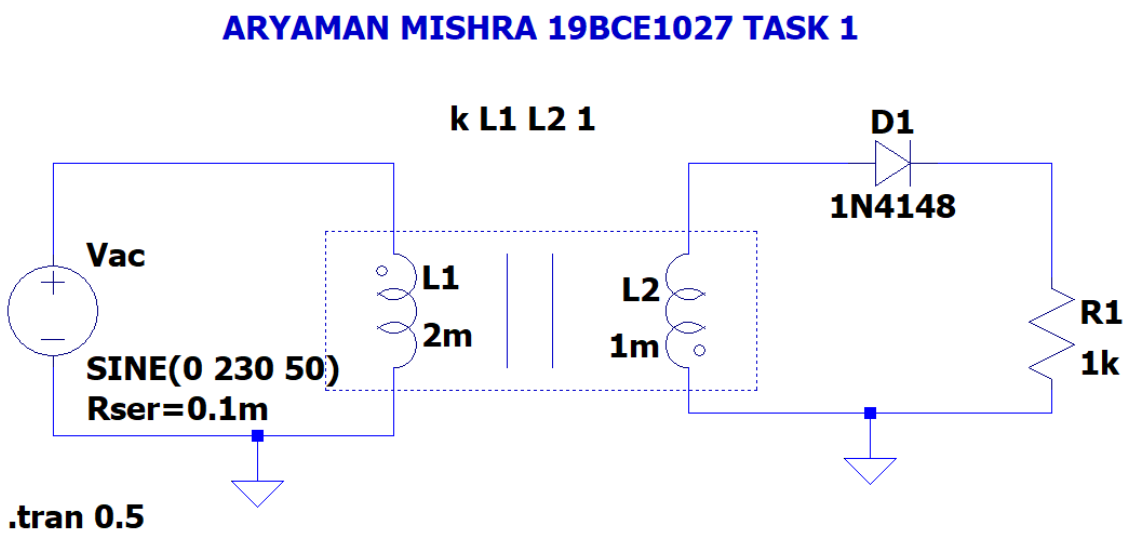
Task 1.3: n phase windings (1m:2m)



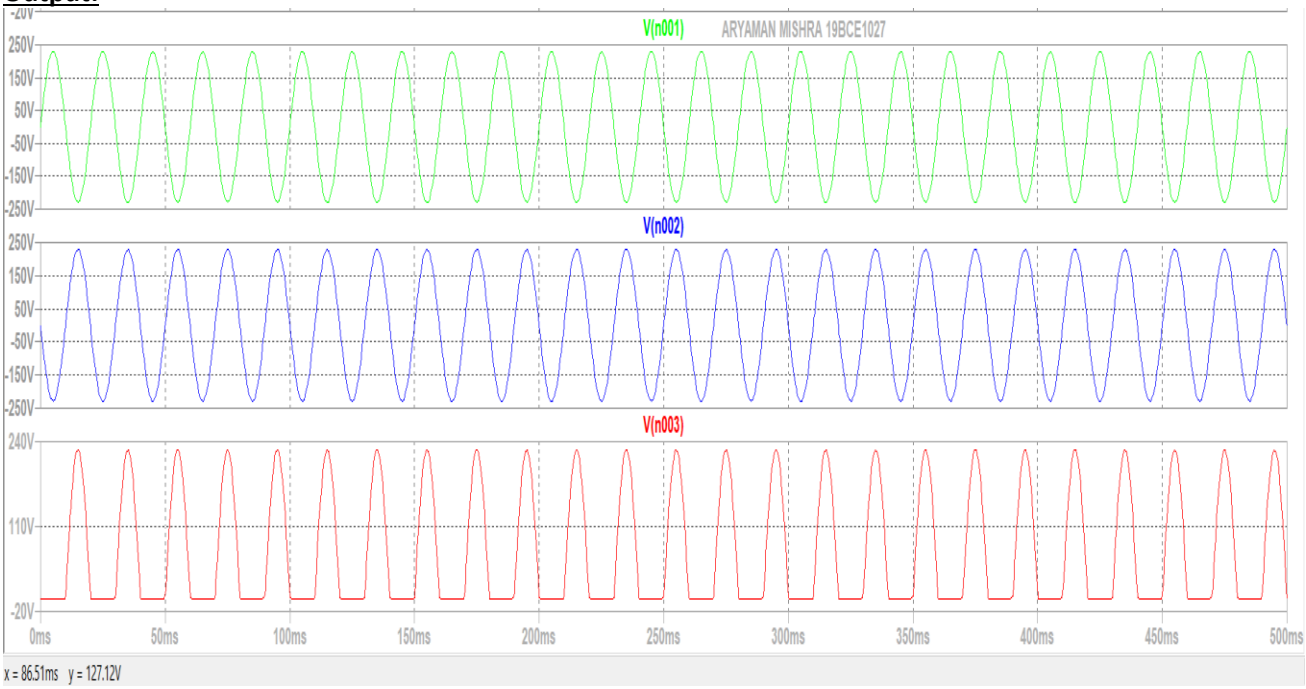
Output:



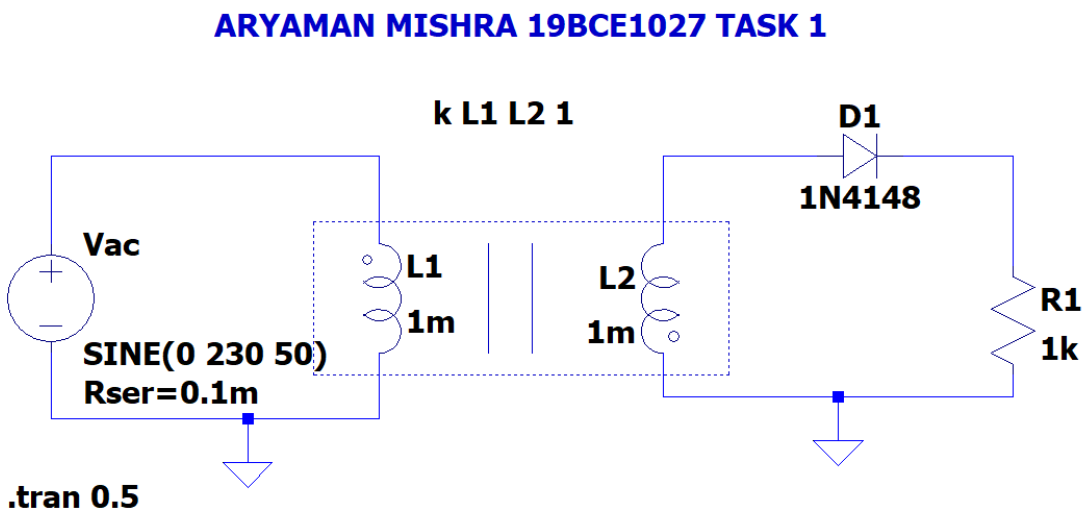
Task 1.4: 180° out of phase windings (2m:1m) 1.5



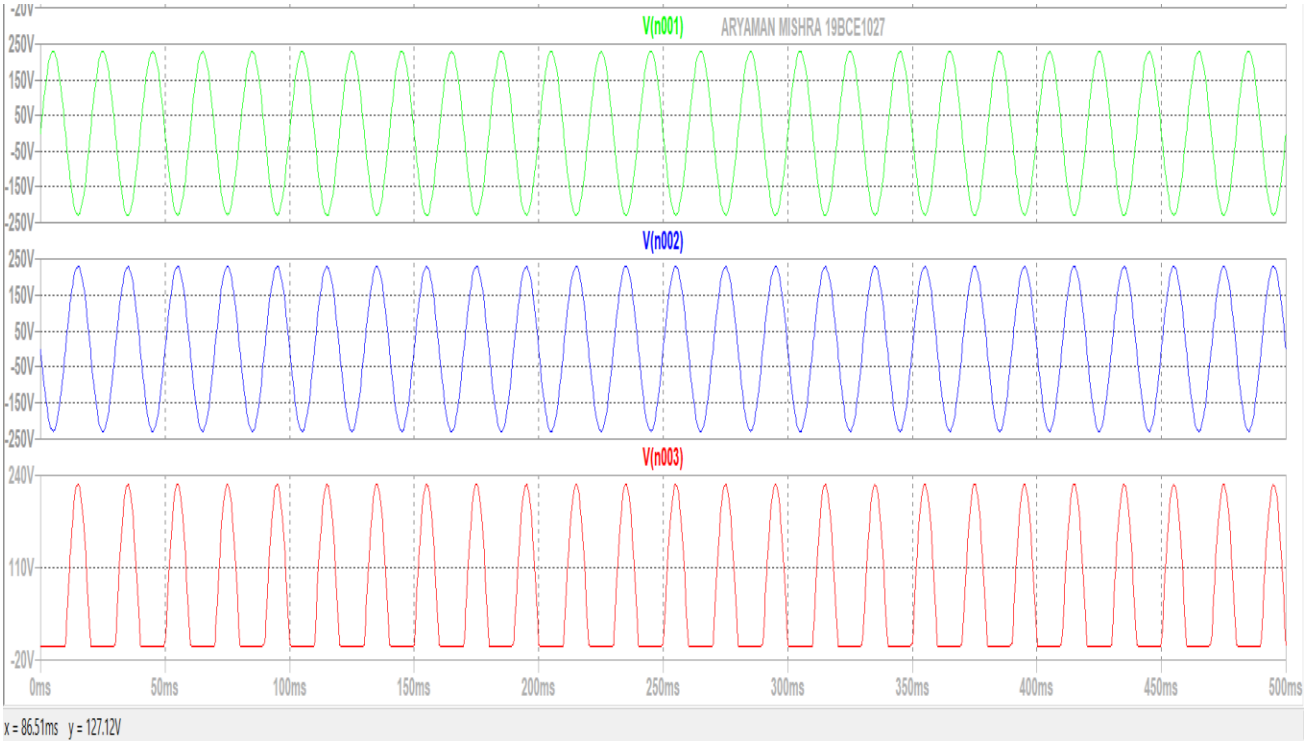
Output:



Task 1.5: 180° out of phase windings (1m:1m)



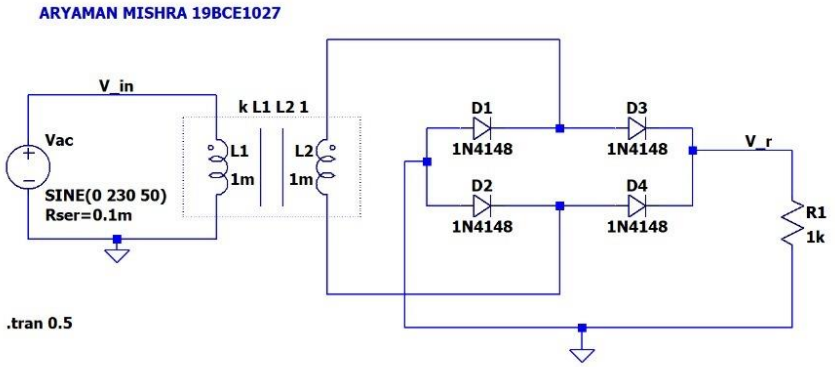
Output:



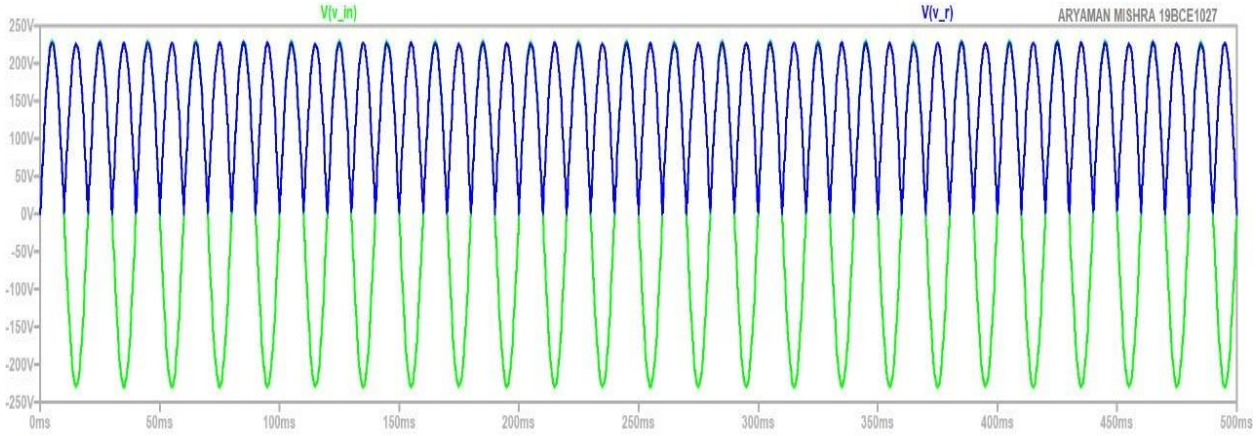
Task#02: Full wave rectifier Design

Full wave rectifier and plot input voltage and output voltages : (across the load – resistor) with

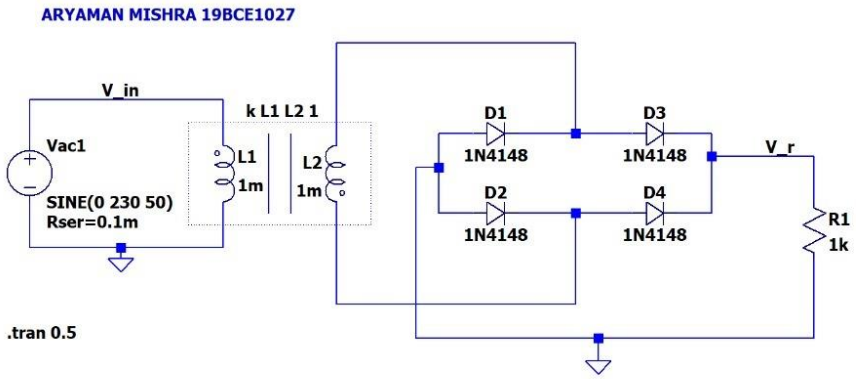
2.1 In phase windings (1m:1m)



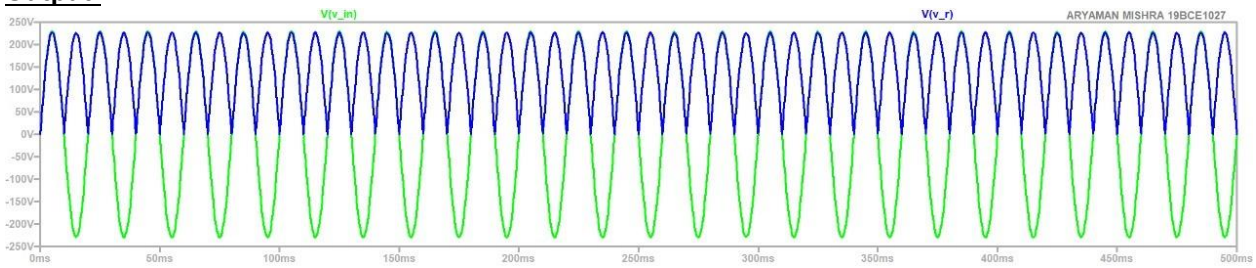
Output:



2.2 180° out of phase windings (1m:1m)



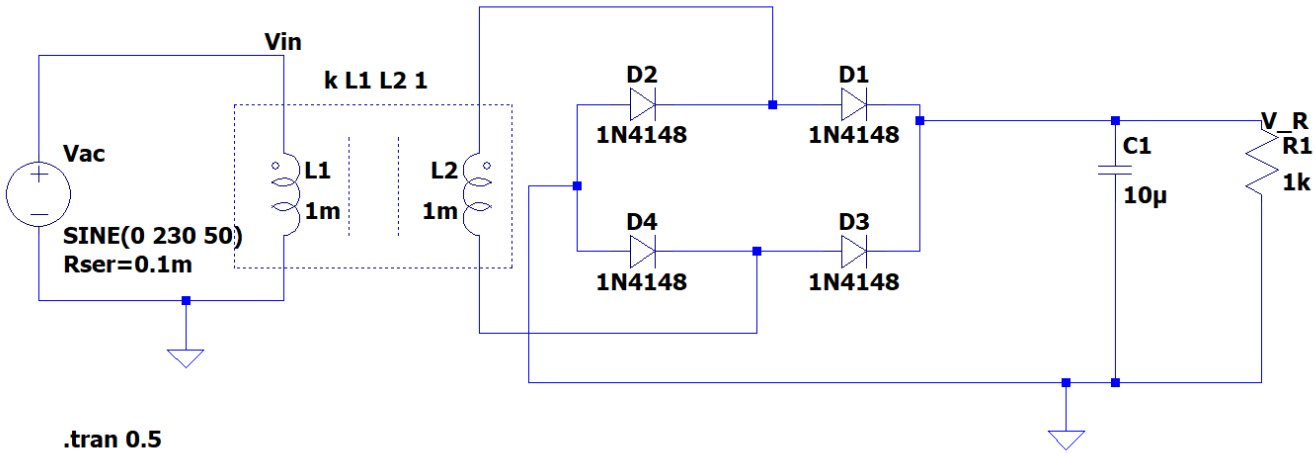
Output:



Task#03: DC Power supply

Obtain output voltage plot for the tasks
3.1 to 3.4 given in table below and enter observation

ARYAMAN MISHRA 19BCE1027



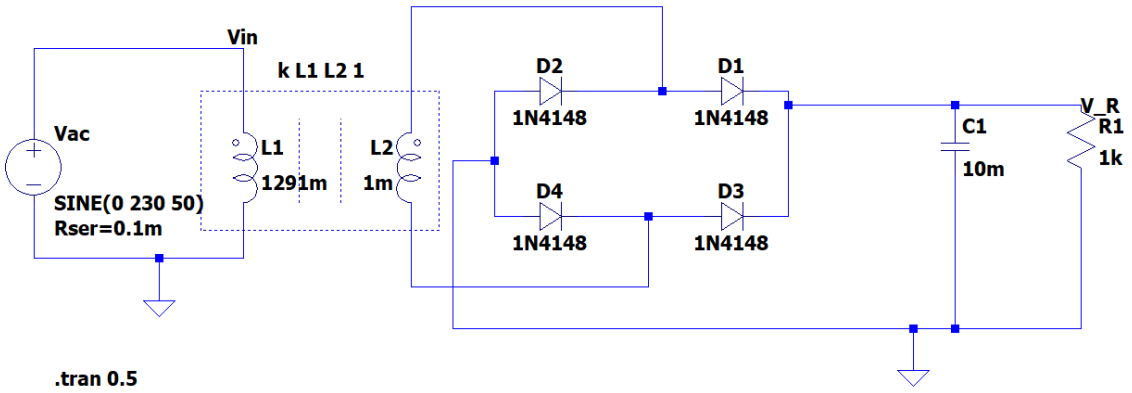
10uf = 122.498V, 226.589V
1m = 223.709V, 225.405V
100uf = 209.498 V, 227.85 V
10m = 224.23V, 224.23V

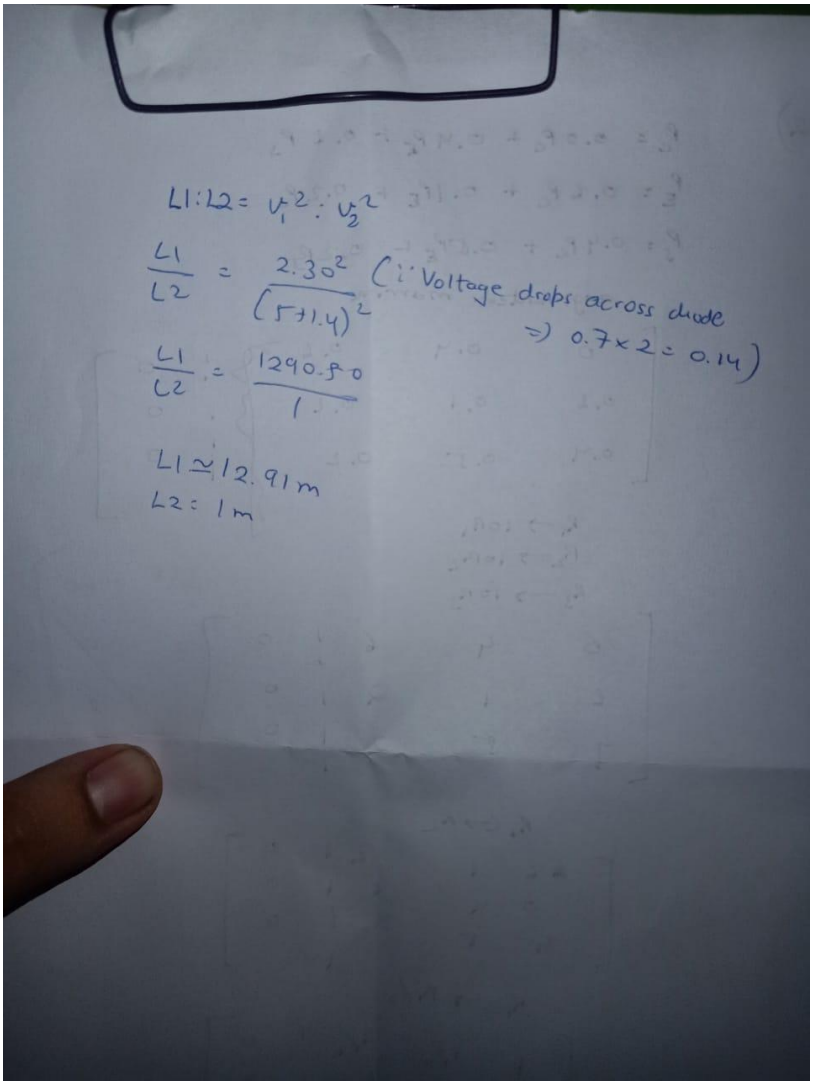
Task	Capacitor	Comment on charging	Comment on discharging	Comment on attainment of 100% of peak of input and saturation (constant) of output.
3.1	10uF	Increased voltage	Decreased voltage	No voltage gets passed
3.2	100uF	Increased voltage	Decreased voltage	No voltage gets passed
3.3	1mF	Voltage remains same	Voltage remains same	No voltage gets passed
3.4	10mF	Voltage remains same	Voltage remains same	No voltage gets passed

Conclusion:We designed and simulated the clippers and clampers circuits using LTspice tool. Thus experiment is successfully completed.

Task#04: Design a DC Power supply of 5V

ARYAMAN MISHRA 19BCE1027





Conclusion:We designed and simulated the clippers and clampers circuits using LTspice tool. Thus experiment is successfully completed.