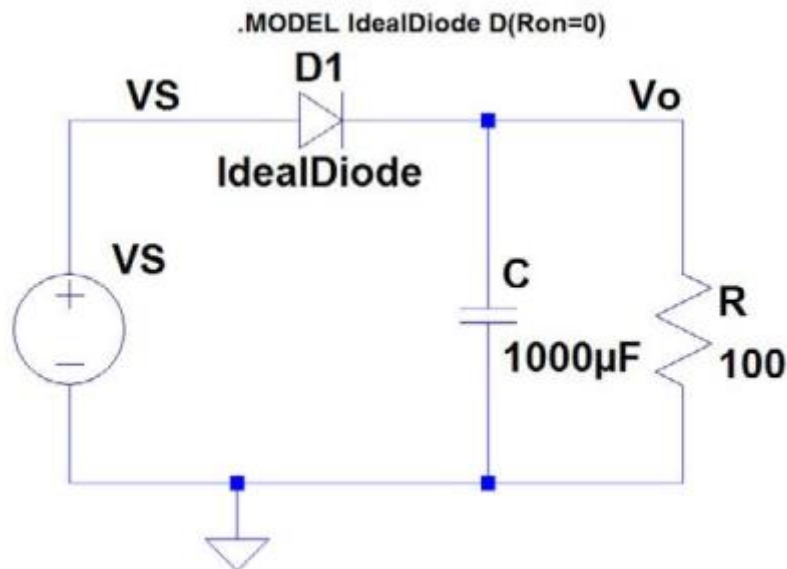


## EE 311 LT-SPICE HW#1

**Q-1)** Consider the following circuit as illustrated in Fig.1. Apply a sinusoidal voltage supply to the input with  $10V_{p-p}$  and 60Hz. Select time domain analysis for 50ms.

- Construct this circuit in LT-SPICE for “Ideal Diode” model with zero on resistance. Plot input voltage, diode current and output voltages.
- For ideal diode model in a), calculate ripple voltage value ( $V_r$ ) and maximum diode current ( $i_{Dmax}$ ).
- Construct this circuit with “diode.txt” of CMOS Spice Lib Files at the website.
- Repeat calculations in b) for construction in step of c) with “diode txt” model.
- Compare results of ideal model and “diode.txt” and explain differences.



**Fig.1:** Half Wave Rectifier With a Smoothing Filter

Q-2) Consider the Fig.2.

- In this circuit, calculate  $\tau$  comes from capacitor and resistors.
- Calculate capacitor voltage at the time of  $\tau$ .
- Construct the circuit in LT-SPICE with PWL source as depicted in Fig.2. Check the capacitor voltage that you calculated in b). **Hint:** Utilize from cursor attachment. Show the results. Comment on the differences between your calculation and SPICE's result.

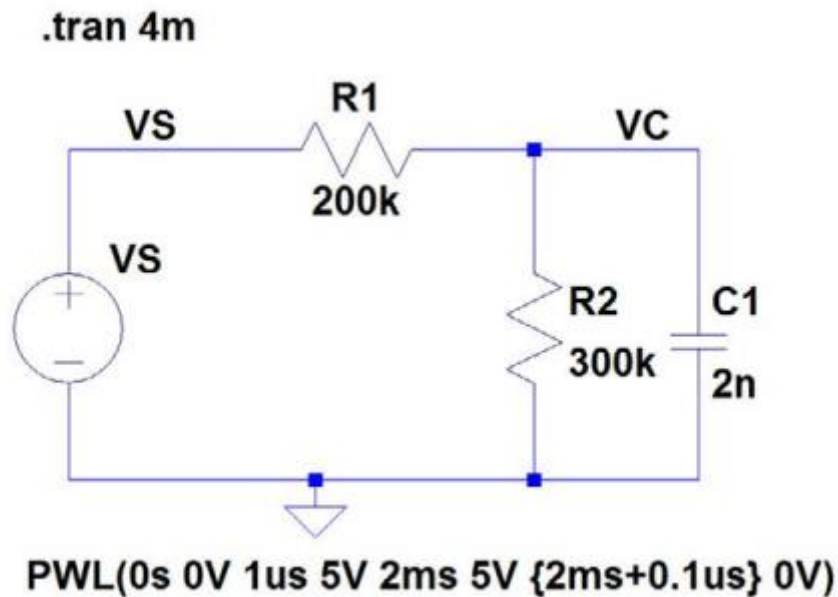


Fig.2: A RC network with PWL source

**NOTE:** Please, show all your calculations step by step for partially credit.