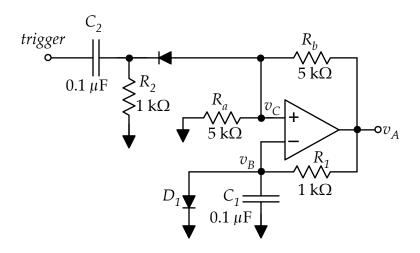a) Use SPICE to simulate the one-shot circuit shown below. (Same as figure from the text.)

   Trigger the one-shot by introducing a *negative-going* pulse at the input, and use a transient analysis to make plots of v_A, v_B, and v_C as functions of time.

   Notes

   - For the trigger, you can use a VPULSE source that starts at +10 V, then drops to -10 V for 10 μs or so, and then goes to +10 V for the rest of the period. Make the period of the VPULSE long enough so that that there is only one input pulse per run.

   - For the diodes in SPICE, use the generic diode model — it is not necessary to change any aspects of the model to use it here.

   - In SPICE, adjust the op amp model parameters and power supplies so that the output saturation levels will be ±10 V.

b) Re-design the one-shot (i.e. pick a new value of the R_1C_1 product) so that the output is exactly 100 μs long. Confirm your re-design with SPICE.