

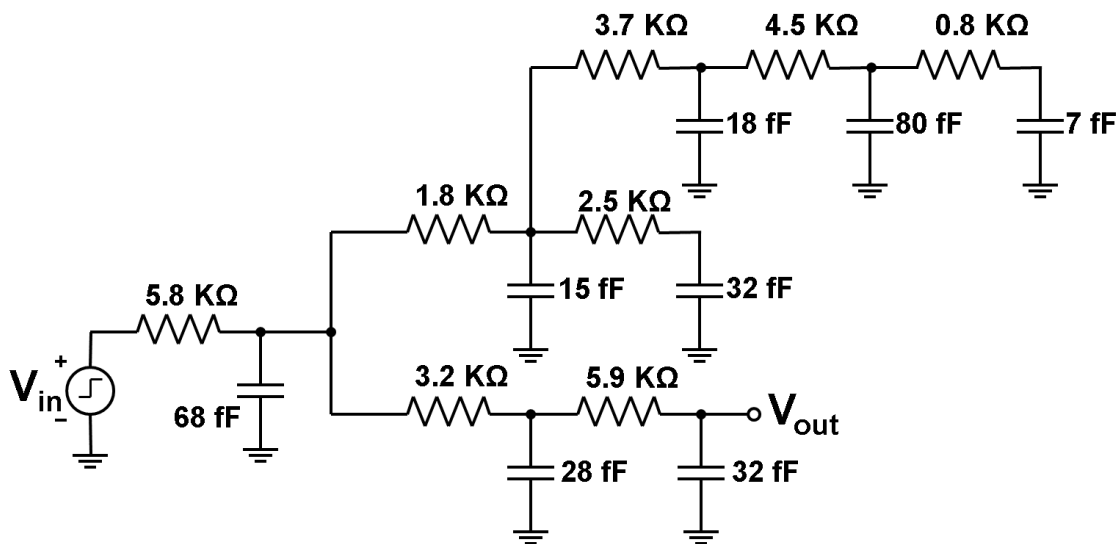
ECE 520 - VLSI Design (spring 2026)

Homework #9

Due in class on Thursday April 9, 2026

This homework is to prepare you for the class project. It requires working with SPICE tool.

1. Feel free to use any version of SPICE tool for this homework. A comprehensive [user's guide for PSPICE](#) can also be helpful.
2. After you familiarize yourself with the tool, write a SPICE file to do transient analysis on the following RC circuit.



3. After successful simulation, measure the rise and fall propagation delays (t_{pLH} and t_{pHL}) from the input to the output (V_{out}).
4. Using the Elmore delay model, hand calculated the t_{pLH} and t_{pHL} in this circuit.
5. Determine the percentage of error between Elmore model and exact SPICE simulation for the propagation delays.