

# ENGR 098 — Lab 1: PSpice Installation and Practice

Yixu Liu (Vincent)

August 28, 2025

## Objective

- Install and launch PSpice 9.1 (student/demo) successfully.
- Build a simple test circuit in Schematics (e.g., a DC source, resistors, LEDs).

## Materials & Tools

- Windows PC (VDI).
- PSpice 9.1 (Student/Demo).

## Procedure (Summary)

1. **Install / Access PSpice**
2. **Create a New Schematic.** Place source, resistors, and LEDs. Add ground.
3. **Set Values.** Example:  $R = 220\Omega$ ,  $V_{DC} = 5\text{ V}$ .

# Circuit Diagram

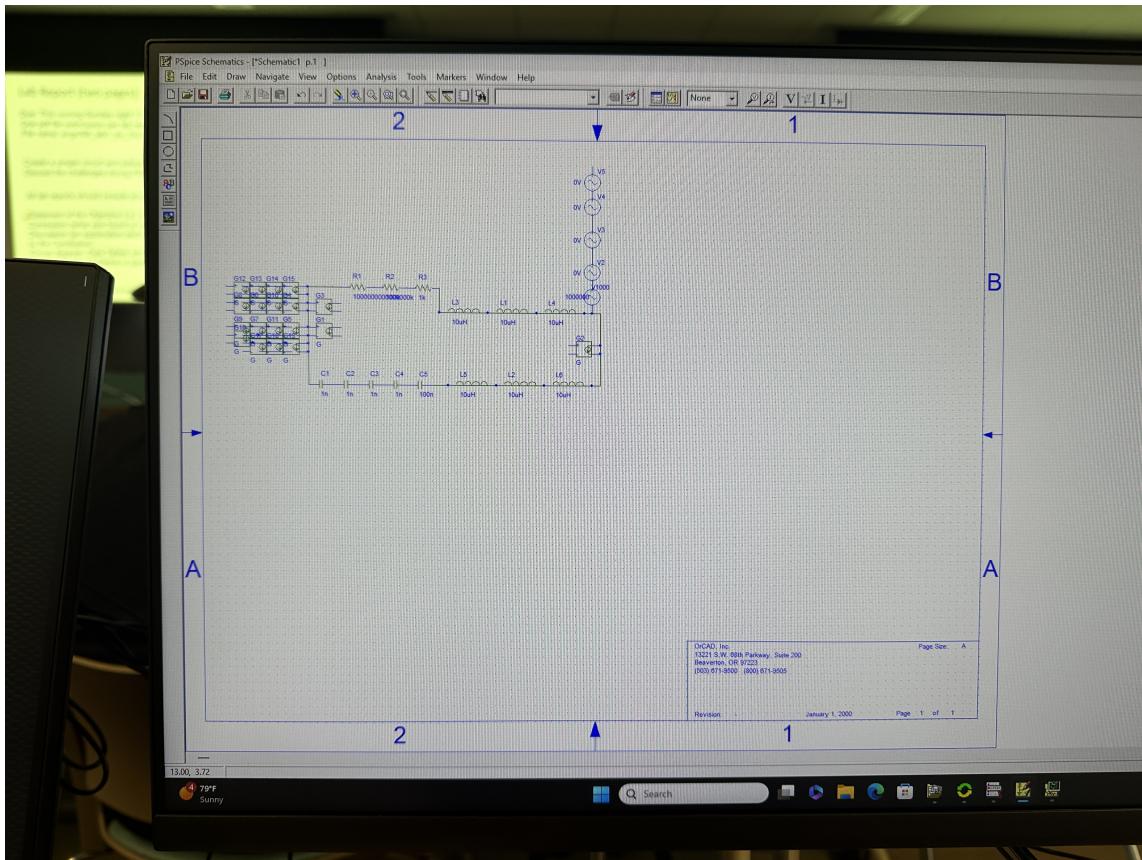


Figure 1: PSpice used for Lab 1.