# EE 233 Circuit Theory

## Spring 2024

# LTspice

*Circuit Simulation Software Tool*

*DC operating point, transient analysis, parametric sweep*

**Department of Electrical and Computer Engineering**

**University of Washington, Seattle WA**

**Mahmood A. Hameed**

# Download

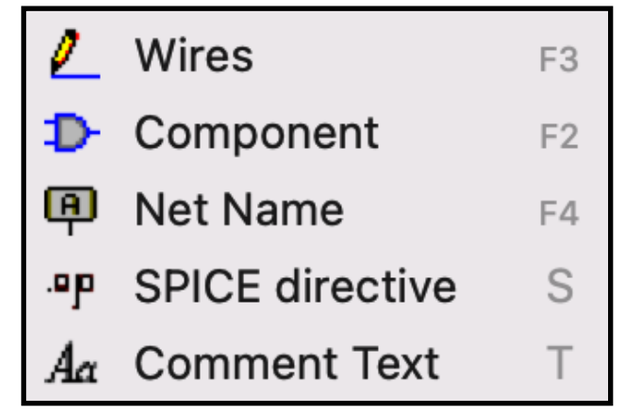
- Google “ltspice download”
  - From analog devices website
- Use OS specific download option
- Complete installation process
- Windows users will have a better usage experience

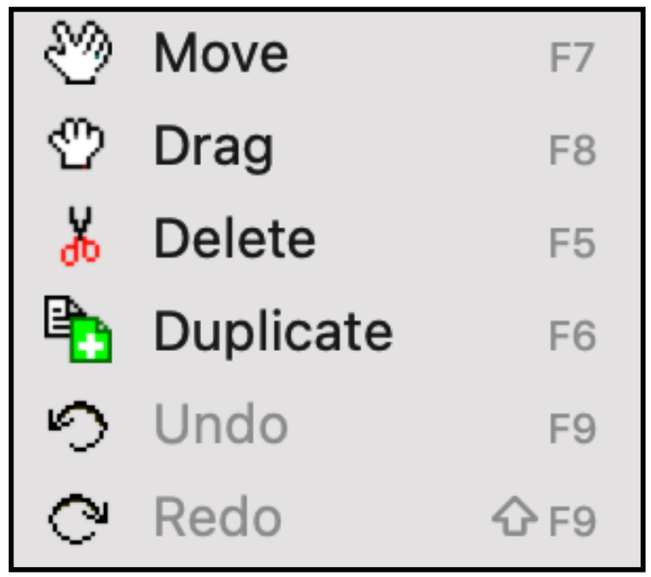
# Shortcuts - useful for MacOS

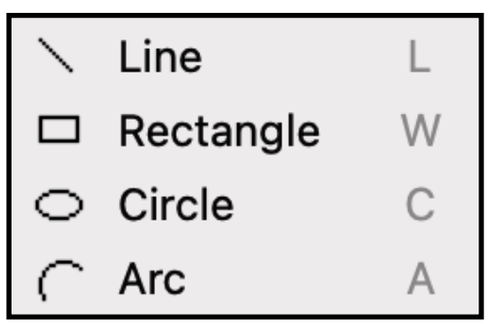
- **voltage source**: v
- **Resistor**: r
- **Capacitor**: c
- **Inductor**: l
- **GND**: g
- **Zoom to fit**: space
- **add text**: t
- **rotate**: command r
- grid points on/off

**SPICE directives:**

- .op
- .tran 100u 5m
- .ac dec 50 1 100Meg
- .step param X .1u .3u .1u
  - {X}



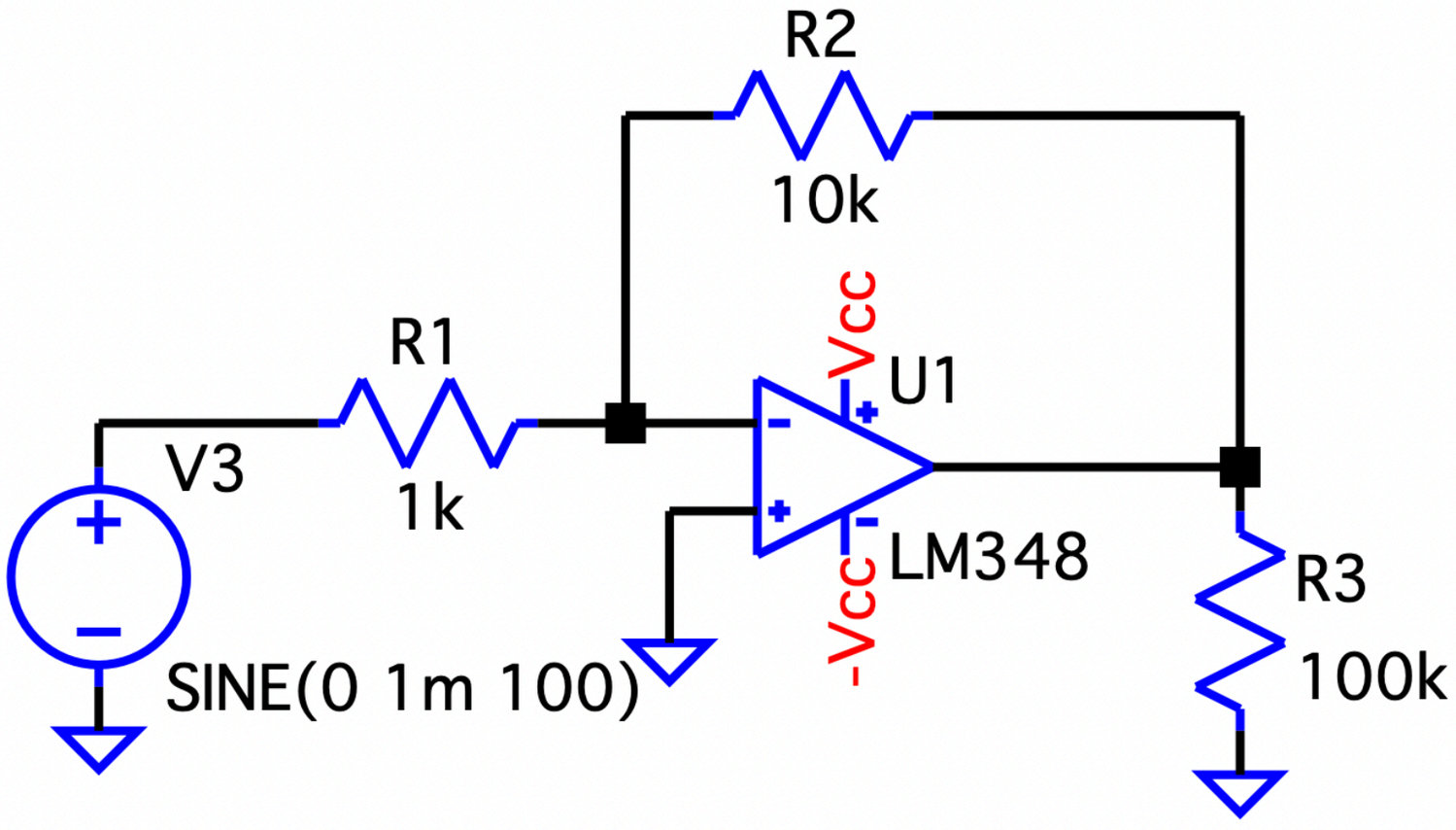



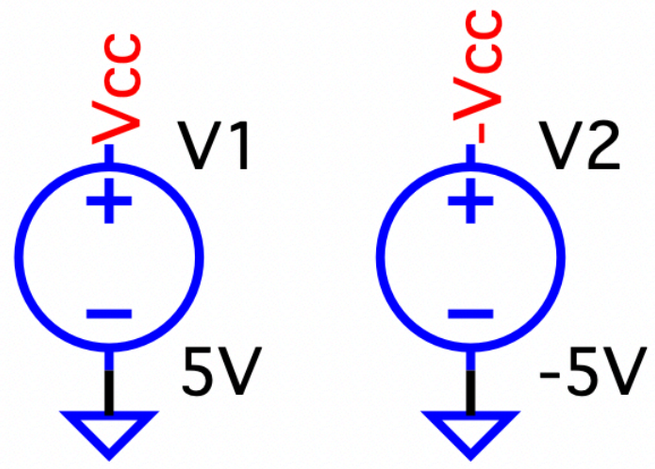



# Demo

- Voltage divider, DC operating point simulation
- RC circuit, transient analysis, parametric sweep, AC analysis
- Inverting amplifier, adding a model to LTspice library

# Inverting amplifier using LM348




.tran 40m

.lib LM348.301

# Options

- Change color preferences
- Export data