Name: Student ID:

Lab Section: Date:

# **Prelab**

1. How many 2-to-1 muxes were used to implement the 8-bit bus mux in lab 3?
2. Go to this webpage <https://www.build-electronic-circuits.com/kicad-tutorial/> and read the article “Make Your First Printed Circuit Board” Part 1, which covers how to draw schematics. Afterwards, answer the following questions:
   1. Why do we need to add footprints to the schematic?
   2. The article describes two types of resistor footprints: SMD and THT. What is the meaning of these acronyms?
3. Watch the tutorial video for this lab, which is linked on Canvas. Then, answer these questions:
   1. How do you create input buses?
   2. What is the name of the pin-equivalent multiplexer chips that we are using in KiCad for this lab?
4. List three advantages of creating a PCB implementation for a circuit?

# 

# 

# 

**Lab**

**4.2** Verify that you placed your chips, connectors, power and ground wires correctly. Show your progress on the schematic to the TA before you proceed. (25 pts)

TA Initials:

**4.4** Verify that all of the components are present and connected correctly. Show your progress on the schematic to the TA before you proceed. (25 pts)

TA Initials:

**4.6** Verify that power and ground are added in all of the appropriate places. Then, make sure to label your schematic. Show your progress to the TA before you proceed. (25 pts)

TA Initials:

**4.8** Verify that your schematic is annotated. Also verify that the components are assigned to the correct footprints. Show your final product to the TA. (25 pts)

TA Initials: