

RoboJackets Electrical Training Week 3 Lab Guide

Jackie Mac Hale

October 11, 2023
v2.0

Contents

1	Background	2
2	Objective	2
2.1	Task 1	2
2.2	Task 2	2
2.3	Task 3	2
3	Materials	2
4	Relevant Information	2
4.1	Adding libraries to KiCAD	2
4.2	Adding parts to your schematic	2
4.3	Adding global labels	3
5	Guided Lab	4
5.1	Add a pull-down resistor to the SW1 Buttons circuit	4
5.2	Add a PTS645 button to the SW2 Buttons circuit	5
5.3	Add a 5V supply for the pull-up resistor in the "RESET" circuit	5
5.4	Create a connection for the signal "D2_CTRL" using global labels	6
6	Troubleshooting	7

1 Background

This week's lecture topic was on designing schematics with KiCAD. A majority of RoboJackets will use KiCAD to design printed circuit boards (PCBs) for their robots. For this reason, it's important to feel comfortable using this software and learn about the many different features KiCAD offers for PCB design.

In this lab, you will be given a partially built schematic for a PCB. Your objective is to complete this PCB using the parts list given below. You will need to use some RoboJackets libraries to add these parts to the schematic.

2 Objective

2.1 Task 1

- Find the desired parts in the RoboJackets library. This step will be required for the parts of the lab that ask you to insert parts into the schematic

2.2 Task 2

- Place parts from the parts list into their respective places in the schematic

2.3 Task 3

- Create connections between different components in the schematic where they're needed

3 Materials

- KiCAD
- Cloned repository of [electrical-training](#)

4 Relevant Information

4.1 Adding libraries to KiCAD

To add the symbol library to KiCAD, make sure you have cloned the electrical-training repository linked above under Materials. Upon opening KiCAD, go to Preferences > Manage Symbol Libraries. Click on the folder icon which says "Add existing library to table". Locate the .kicad_sym in the kicad-libraries directory of the electrical-training directory and open it. Make sure this library is added to the "Global Libraries" section of your symbol libraries to be able to use it with any KiCAD project you create.

4.2 Adding parts to your schematic

You can click the "Add a symbol" button located in the right menu of the schematic editor or press "A" to open the part selection menu.

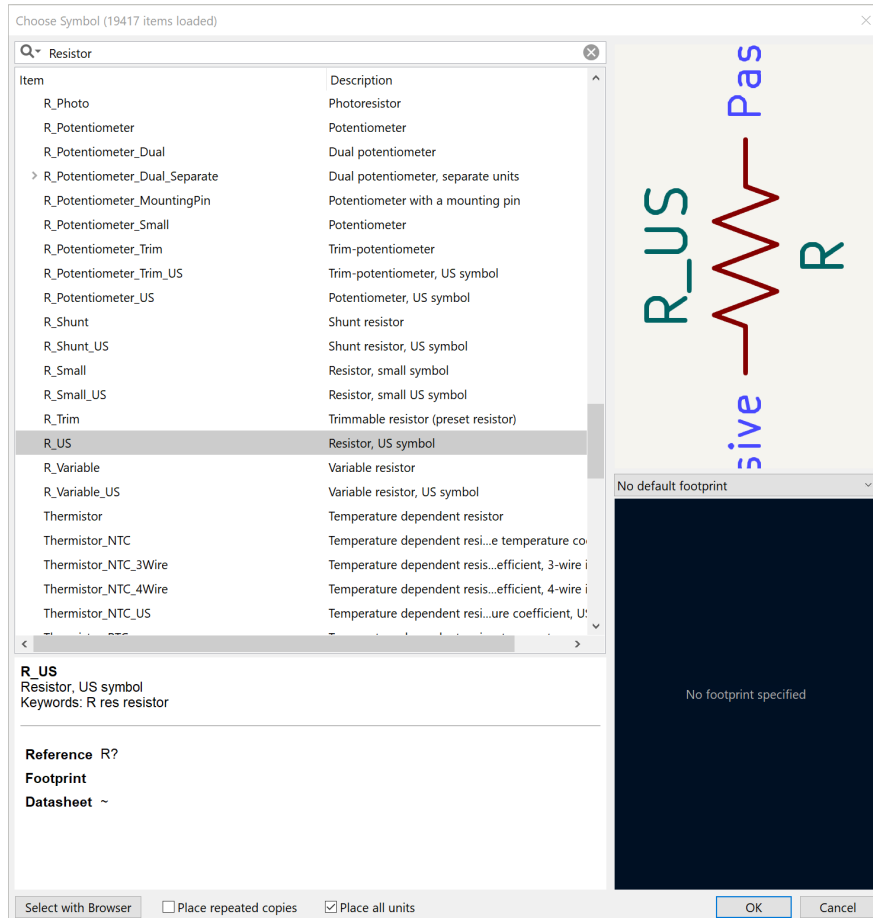


Figure 1: Add Symbol Menu

4.3 Adding global labels

If there are connections that need to be made far away from each other, you can use the “Add a global label” button in the right menu to do this.

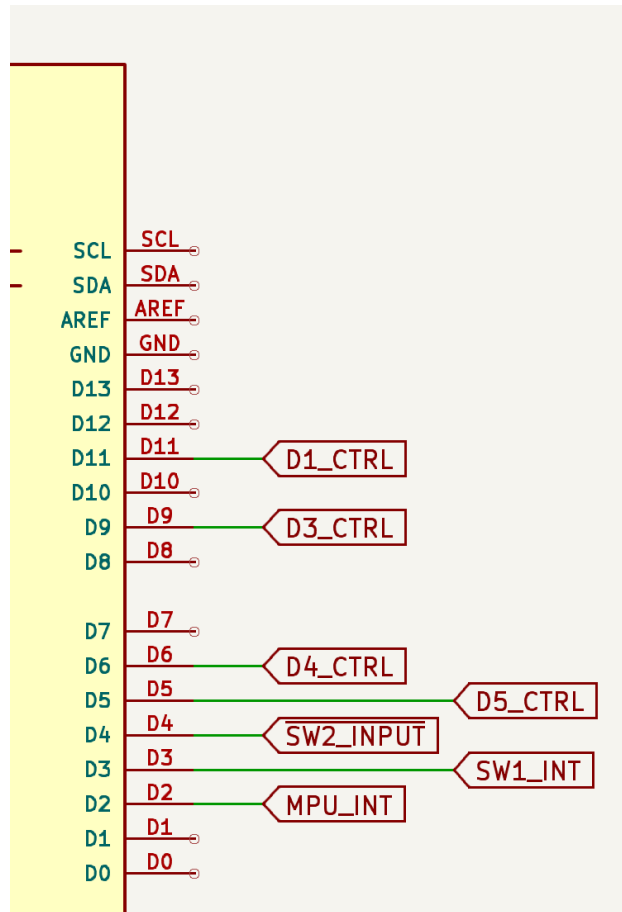


Figure 2: Adding labels

5 Guided Lab

5.1 Add a pull-down resistor to the SW1 Buttons circuit

- Select “Add a symbol” in the right bar and search for a resistor in the Device library. It should be called R_US.
- Place the resistor in the correct place in the circuit. Orientation of resistors is not important.
- Create connections on both sides of the resistor.
- Give the resistor the value “10K” (10,000 ohms). This can be done by clicking on the part, pressing “E”, and typing the value into the “Value” field.

Purpose: This resistor is needed in the circuit to ensure that when switch “SW1” is open, the “SW1_INT” signal is not floating but 0V.

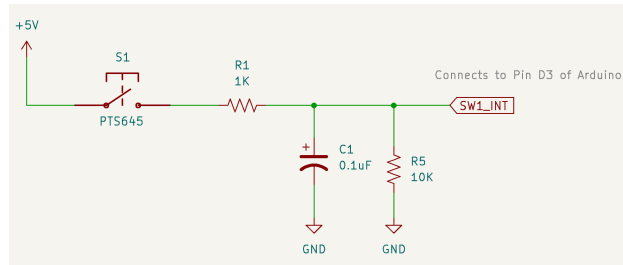


Figure 3: Adding resistor

5.2 Add a PTS645 button to the SW2 Buttons circuit

- Select “Add a symbol” and search for PTS645 button in the RoboJackets library. This part is a normally open pushbutton. When pushed, the circuit is closed, otherwise it’s open.
- Place the button in the correct place in the circuit.
- Create connections on both sides of the button as shown in the picture.

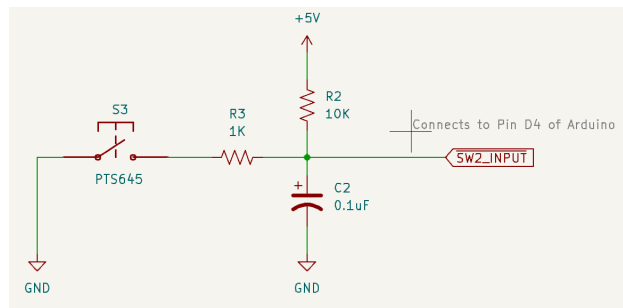


Figure 4: Adding switch

5.3 Add a 5V supply for the pull-up resistor in the ”!RESET” circuit

- Select “Add a symbol” and search for a +5V supply. This will signify a connection to the 5V source on the board to this point of the circuit.
- Place the part in the correct place in the circuit.
- Create a connection from the +5V supply to resistor R4.

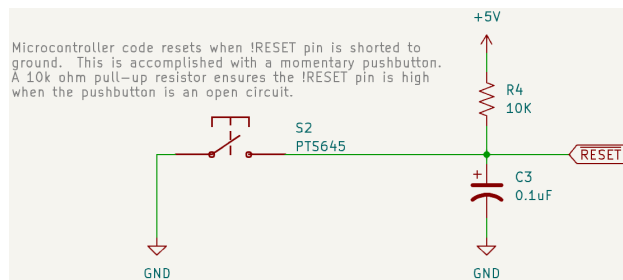


Figure 5: Adding 5V supply

5.4 Create a connection for the signal “D2_CTRL” using global labels

- Create a net from pin D10 on the Arduino Uno Microcontroller to a label.
- Name the label “D2_CTRL”. The name you give a label corresponds to the name of its signal. In this case, the D10 pin is controlling the D2 LED, so naming the signal “D2_CTRL” is a good way to signify what the signal controls.
- Create another label and net that connect to the second resistor from the top in the “LEDs” section. Give this label the same name as the signal from D10.

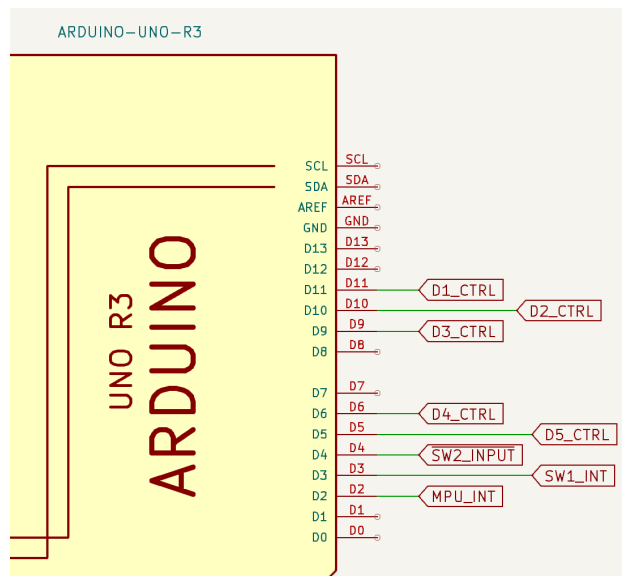


Figure 6: Adding D2_CTRL label on D10

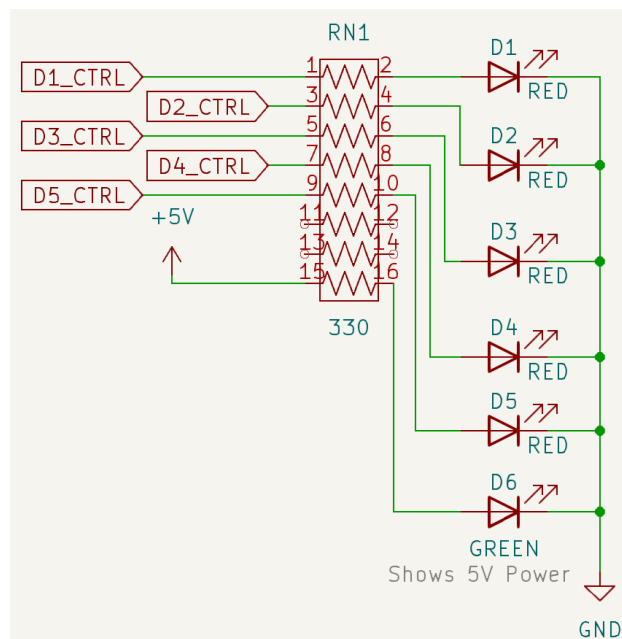


Figure 7: Adding D2_CTRL label on second LED

Purpose: This method of connecting different parts that are far apart will make the schematic cleaner and easier to read. But, a physical connection will need to be made when you are designing the board itself. This will be the topic of next week's training.

6 Troubleshooting

- [KiCAD Getting Started Guide](#) - Includes how to use the schematic editor