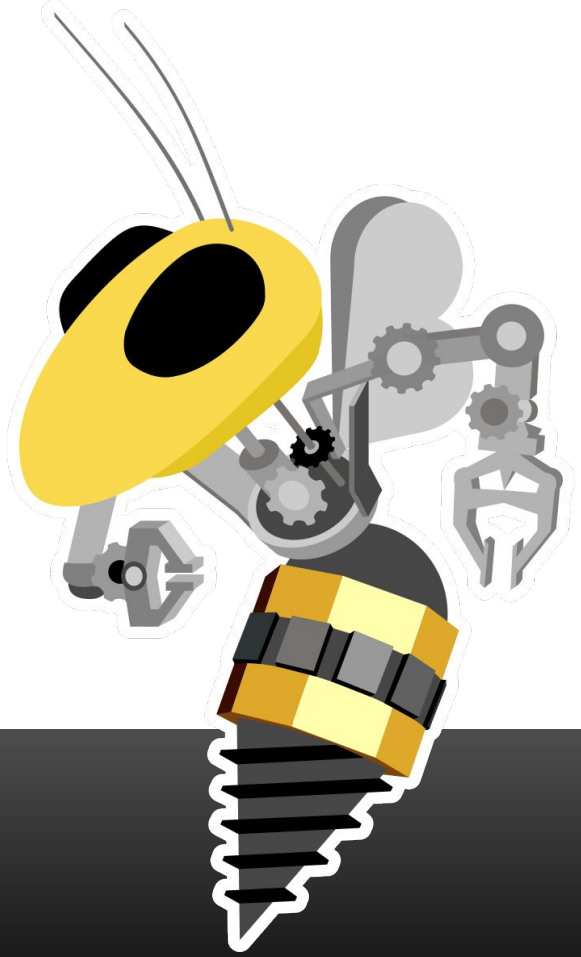
Welcome!

Electrical Training
Week 4



RoboJackets

Competitive Robotics at Georgia Tech

www.robojackets.org

Last Week!

- Communication Systems
- KiCAD Schematics

This Week!

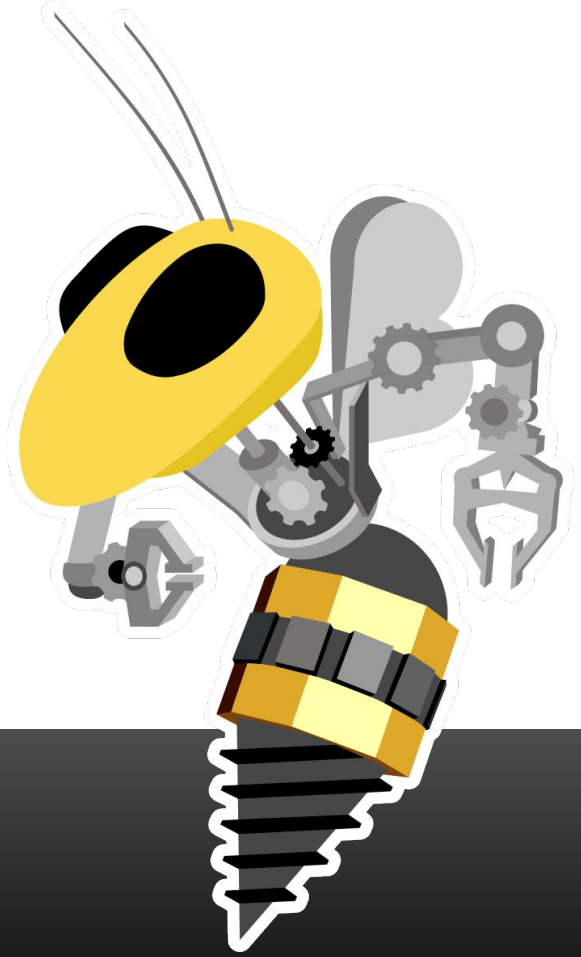
1. Recap
2. Board Layout
    a. Layers
    b. Arranging components
    c. Routing
    d. Polygons
    e. Lab!

Recap

- Previously studied Parts/Libraries and Schematics
- There are separate **symbol** and **footprint** libraries
- In schematics, we use **nets** to link **pins** on a **symbol** together to represent a device's function

Board Layout

Place and route



RoboJackets

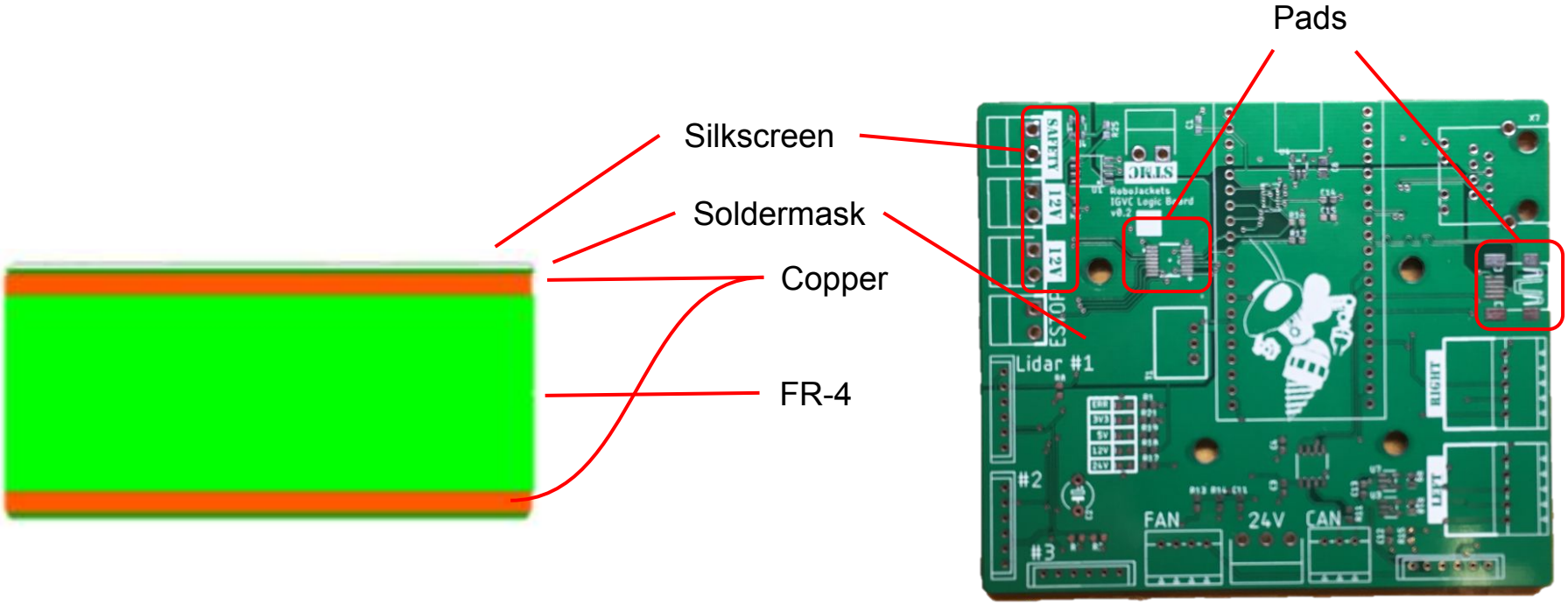
Competitive Robotics at Georgia Tech

www.robojackets.org

Board Layout

- Physical placement of components on the PCB
- Replace the abstract **nets** from schematic with physical **traces**
- Access this feature by pressing the “Open PCB in board editor” button at the top of your schematic window or opening the PCB editor from the Project Manager window

PCB Structure



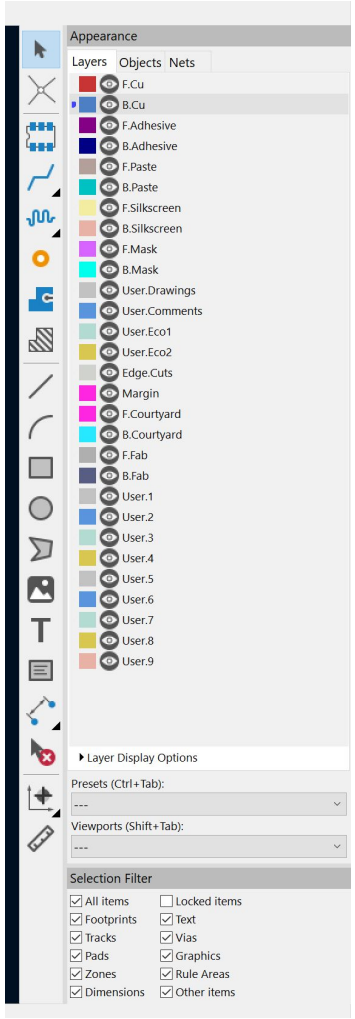
Color	Layer Name	Layer Purpose
	F.Cu	Top/front layer of copper
	B.Cu	Bottom/back layer of copper
	Edge.Cuts	Outline of the board
	F.Silkscreen	Silkscreen for top
	B.Silkscreen	Silkscreen for bottom
	F.Fab	Top/front documentation layer (just for reference)

NOTE: Pads (SMD and through-hole) are part of footprints and vias are not part of any layer

RoboJackets
Competitive Robotics at Georgia Tech

Changing Layers

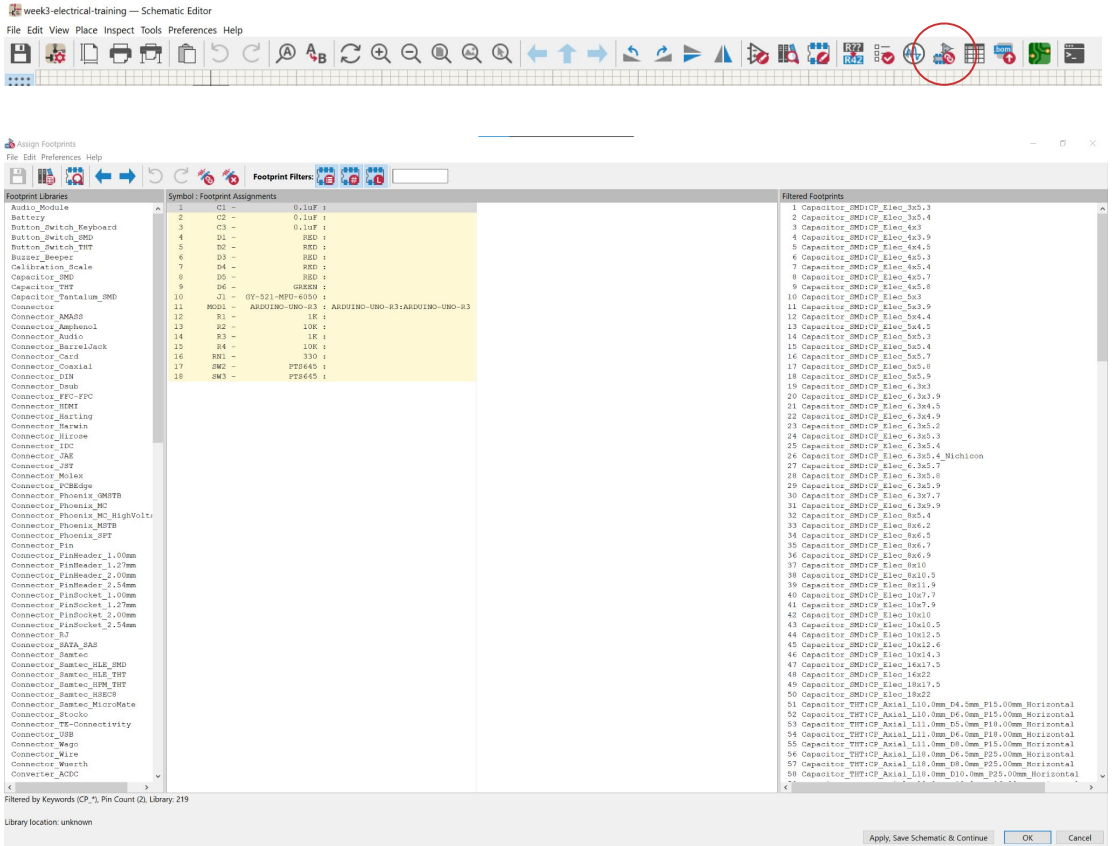
Select the layer you want to edit



ROBOJACKETS
COMPETITIVE ROBOTICS AT GEORGIA TECH

Assign Footprints to Symbols

This must be done from the Schematic Editor



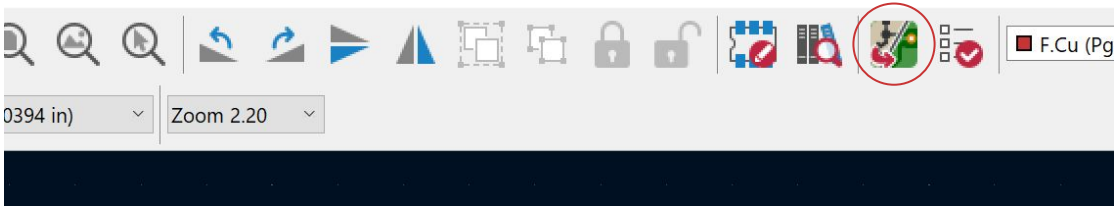
Assign Footprints to Symbols

1. At the top, click Tools -> Assign Footprints
2. Match the schematic symbol to the PCB footprint that you want
   - You can search the footprint you want in the search bar
   - KiCAD doesn't have the footprint for all parts, so you will have to download them from SnapEDA or make them yourself beforehand

RoboJackets
Competitive Robotics at Georgia Tech

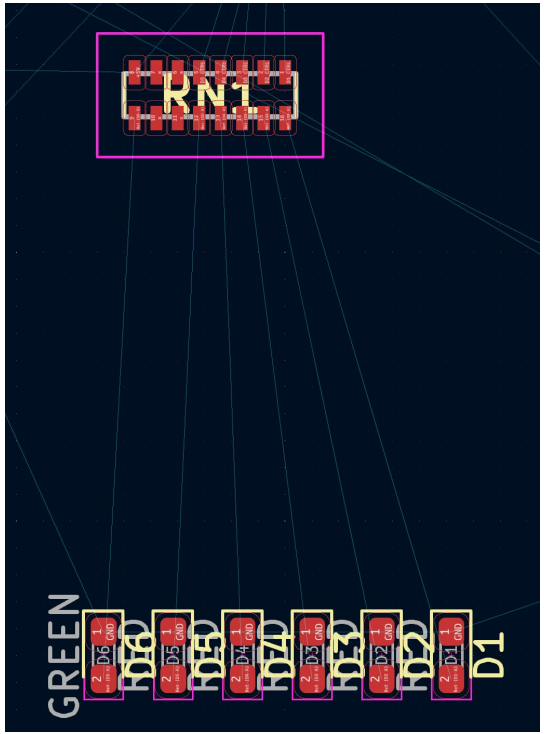
Board Generation

This is in the PCB Editor



Arranging Components

- Click on part and drag to move parts around
- Press ‘R’ to rotate
- Position components on the board area

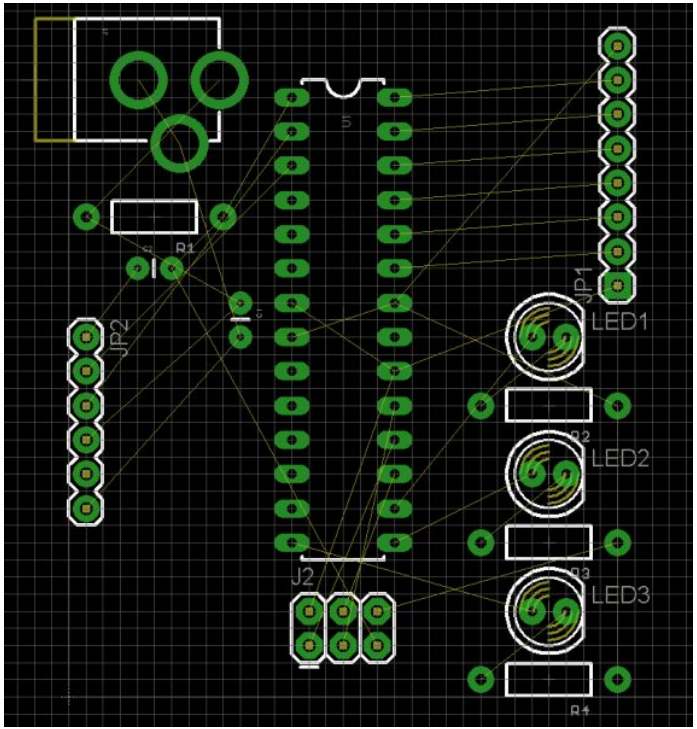


Arrangement Considerations

- Minimize airwires (the unrouted wires)
  - Collection of all airwires is ratsnest
- Maximum size of board
- Clearance between mounting holes and components
- Location of specific components
  - Connectors on board edge
  - Decoupling capacitors near decoupled pins
  - Communicating/related components near one another

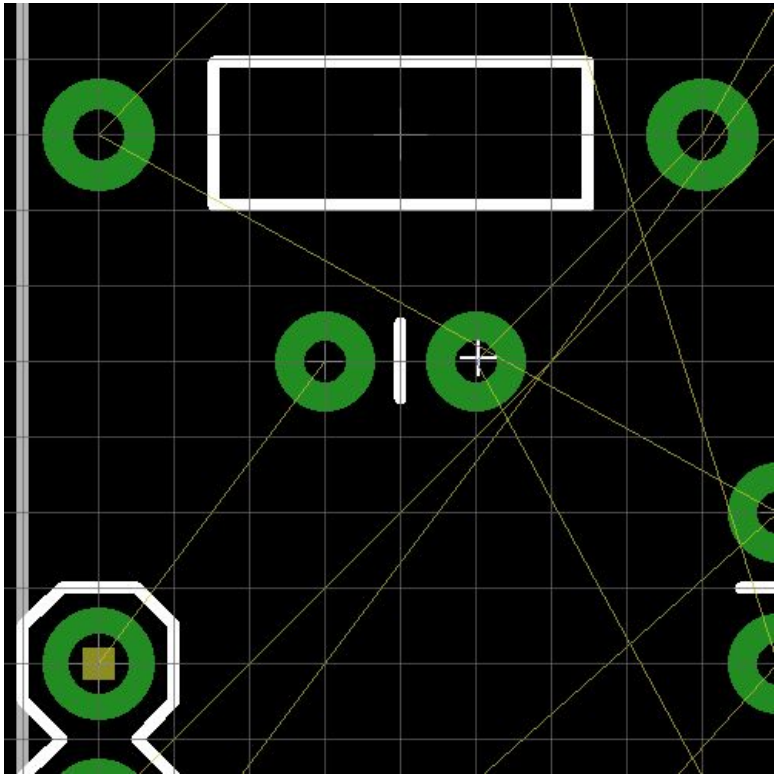
Arranging Components

- Leave some space
  - Room for traces
  - Room to solder
- Minimize the number of intersecting airwires
  - Easier to route traces



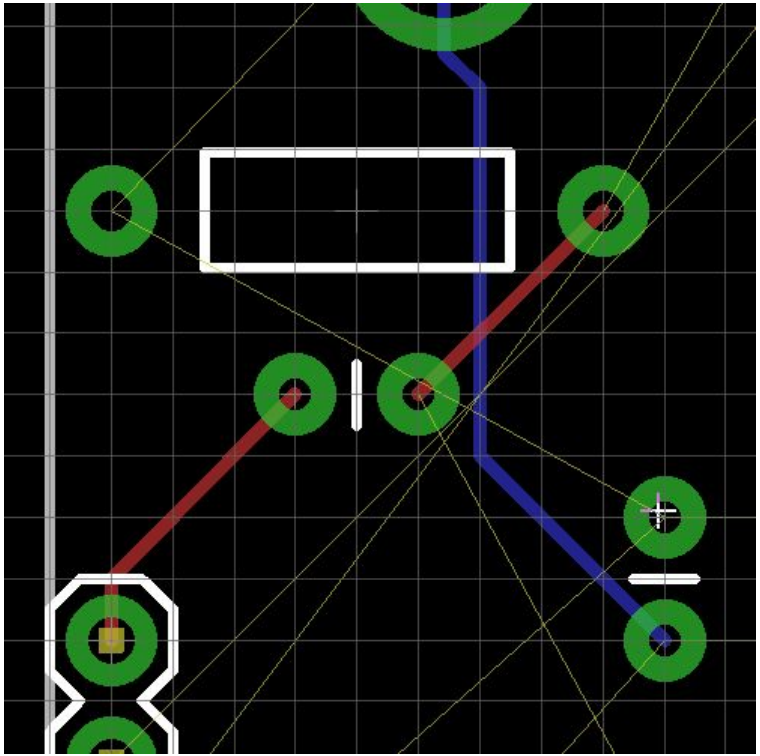
Drawing Traces

- Use the Route tracks command in the right bar
  - Left click on starting point and left click on end point
- Follow start and end of airwire
- Routing angle of 45 degrees



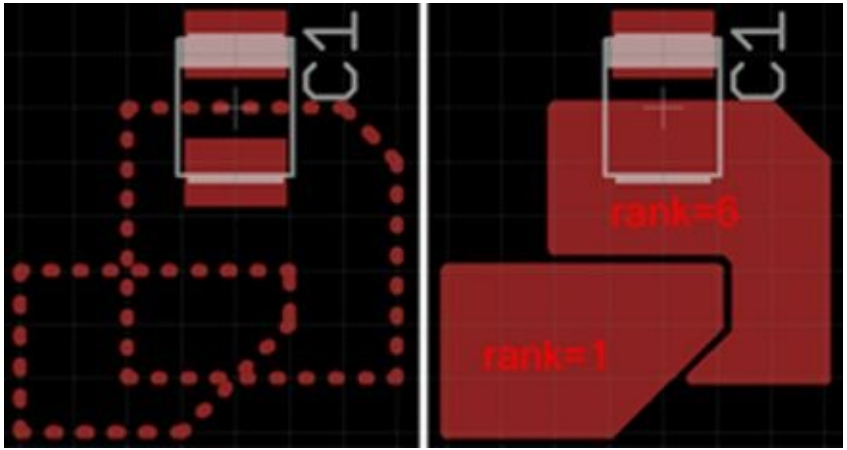
Changing Layers

- Press “v” to change the layer and put a via
  - You can also right click, then click on “Place a via”



Filled Zone

- Allows you to fill a drawn area with copper connected to a specific net
- Useful to make “ground planes”
- Click on “Add a filled zone”
  - Then draw a shape such as a rectangle or polygon
- Fill the polygon with Edit > Fill All Zones



Design Rules and DRC

Design Rules

Predefined rules that your board layout should follow to be correctly manufactured.

DRC (Design Rules Checker)

This tool will check if your board layout is attending to all of the design rules and notify you accordingly.

Note: Load Design Rules early on your design.

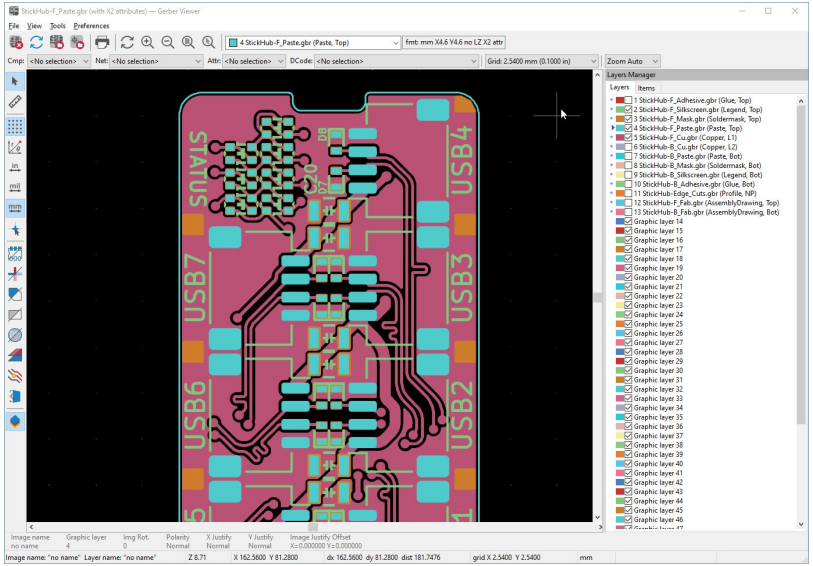
Gerber Viewer

Preview

- Make sure the silkscreen looks good (Both top and bottom side)

Board

- Shows board properties: height, width, area, layers, board thickness, etc



Lab Summary

- Start a new board layout
- Set up the dimensions
- Arrange the components
- Connect the traces
- Touch-up the silkscreen
- Check your design with DRC

For more information, access the LAB Document on GitHub.

Live Demonstration

- Watch me embarrass myself

Resources

KiCad Getting Started:
https://docs.kicad.org/7.0/en/getting_started_in_kicad/getting_started_in_kicad.html#tutorial_part_3_circuit_board

How to branch on GIT:
https://github.com/RoboJackets/robocup-firmware/blob/master/doc/Git.md

For next time...

We will be soldering! Please bring the following things:

- Yourself
- Computer/Laptop
- A Coke for Kyle

Location: The HIVE

When: TBD (sorry)

Feedback/Attendance