Welcome!

Electrical Training
Week 4

Announcements

-

Last Week

- Communication Systems
- EAGLE Schematics

This Week

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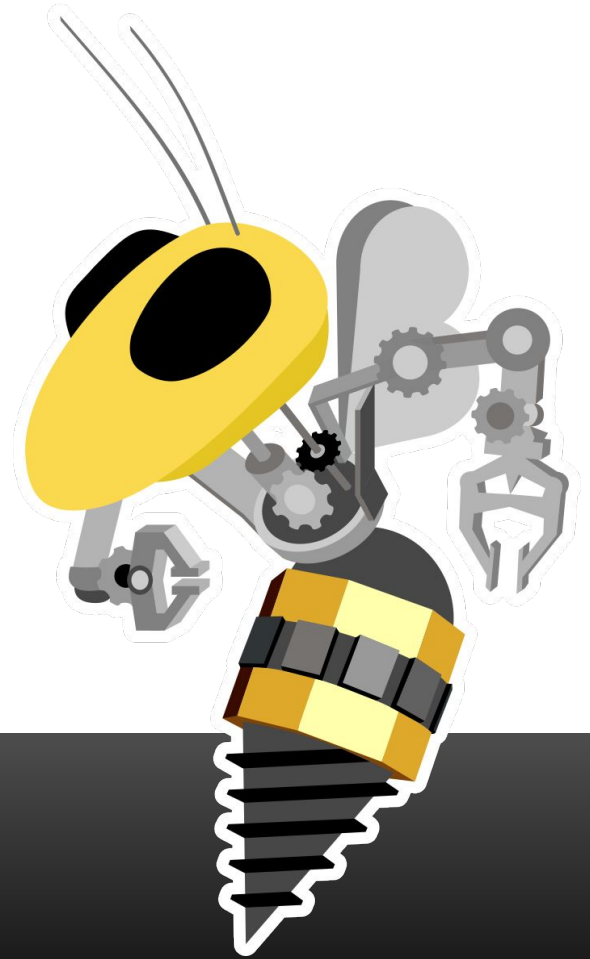   d. Polygons
   e. Commands
   f. Lab!

Recap

- Previously studied Parts/Libraries and Schematics

- Libraries contain a **device** which has a **symbol** and **footprint**

- In schematics, we use **nets** link **pins** on a **symbol** together to represent device function

Board Layout

Place and route

Board Layout

- Physical placement of components on the PCB
- Replace the abstract **nets** from schematic with physical **traces**
- Access this feature by pressing the SCH/BRD button

PCB Structure

Pads

Silkscreen

Soldermask

Copper

FR-4
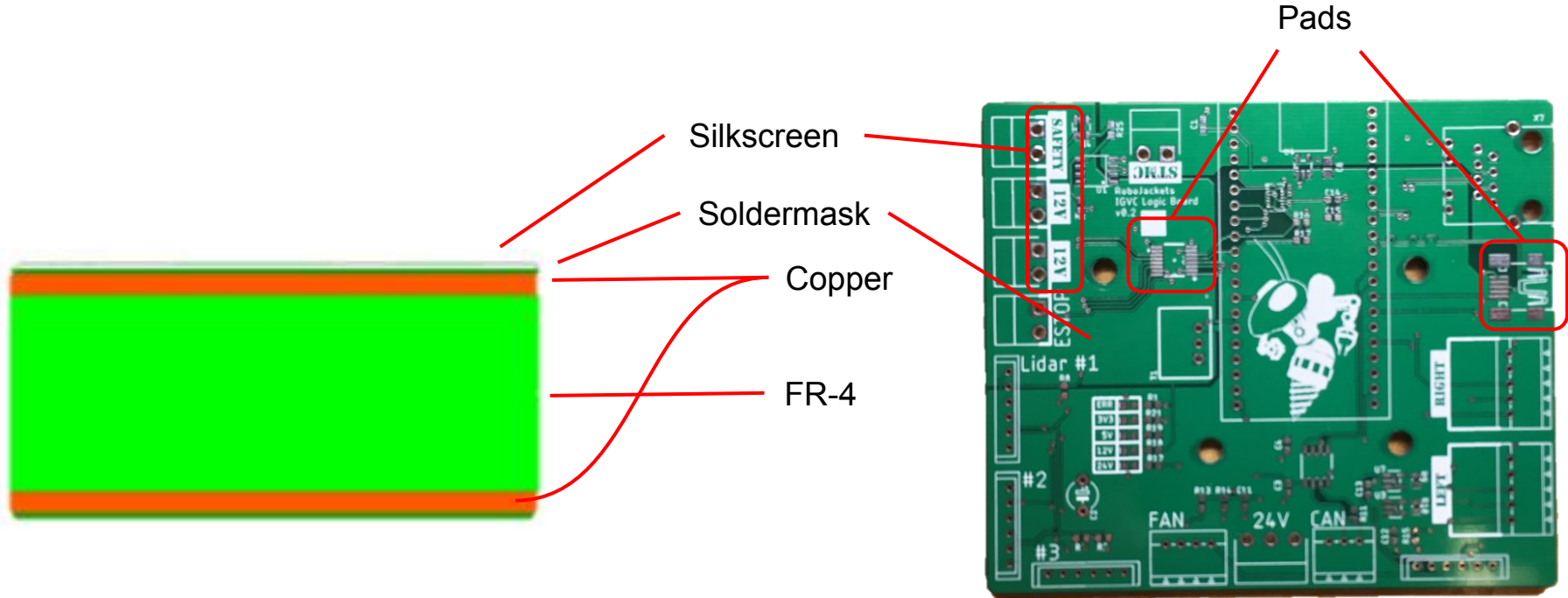
Color	Layer Name	Layer Num	Layer Purpose
	Top	1	Top layer of copper
	Bottom	16	Bottom layer of copper
	Pads	17	Through-hole pads (copper on top and bottom)
	Vias	18	Vias to route signal between layers (copper on top and bottom)
	Dimension	20	Outline of the board
	tPlace	21	Silkscreen for top
	bPlace	22	Silkscreen for bottom
	tDocu	51	Top documentation layer (just for reference)

RoboJackets
COMPETITIVE ROBOTICS AT GEORGIA TECH
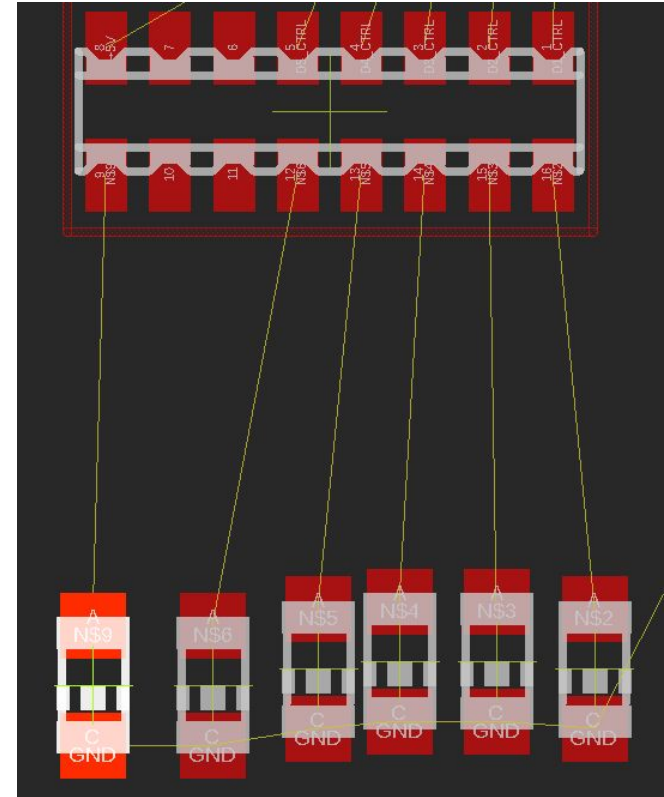
Changing Layers
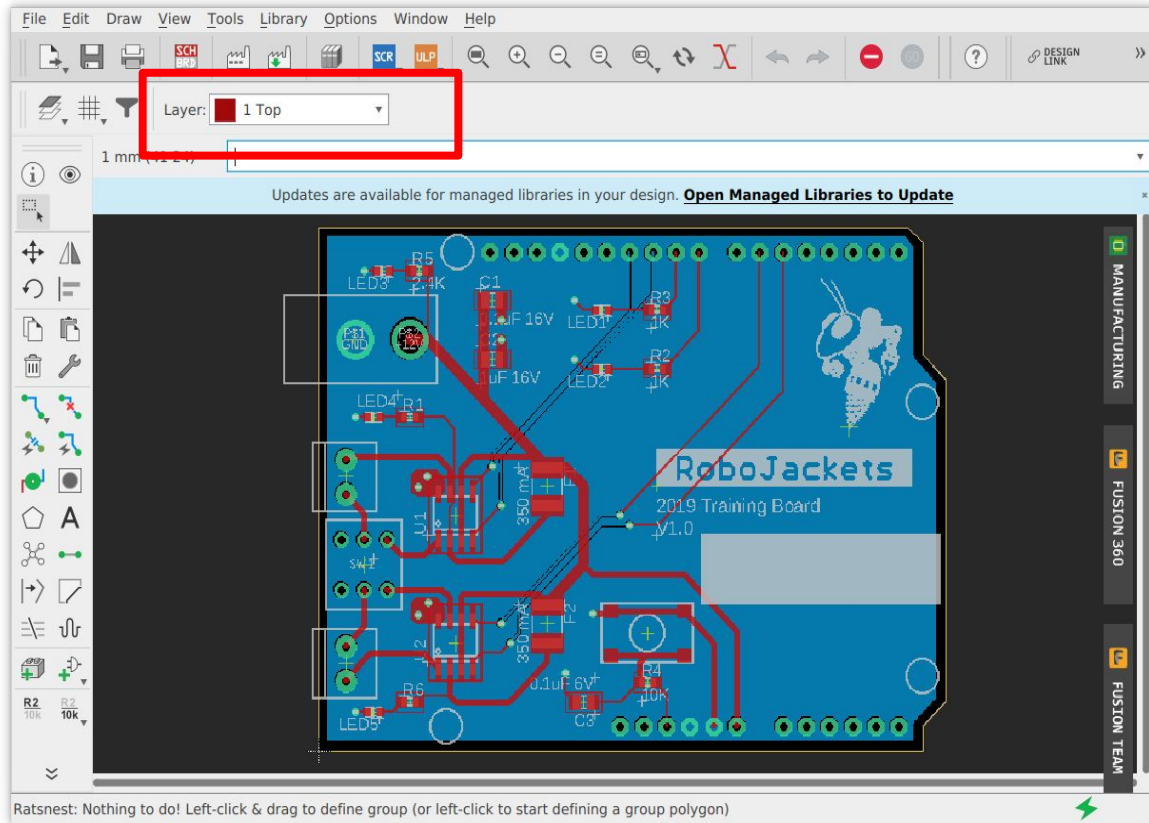
Board Generation
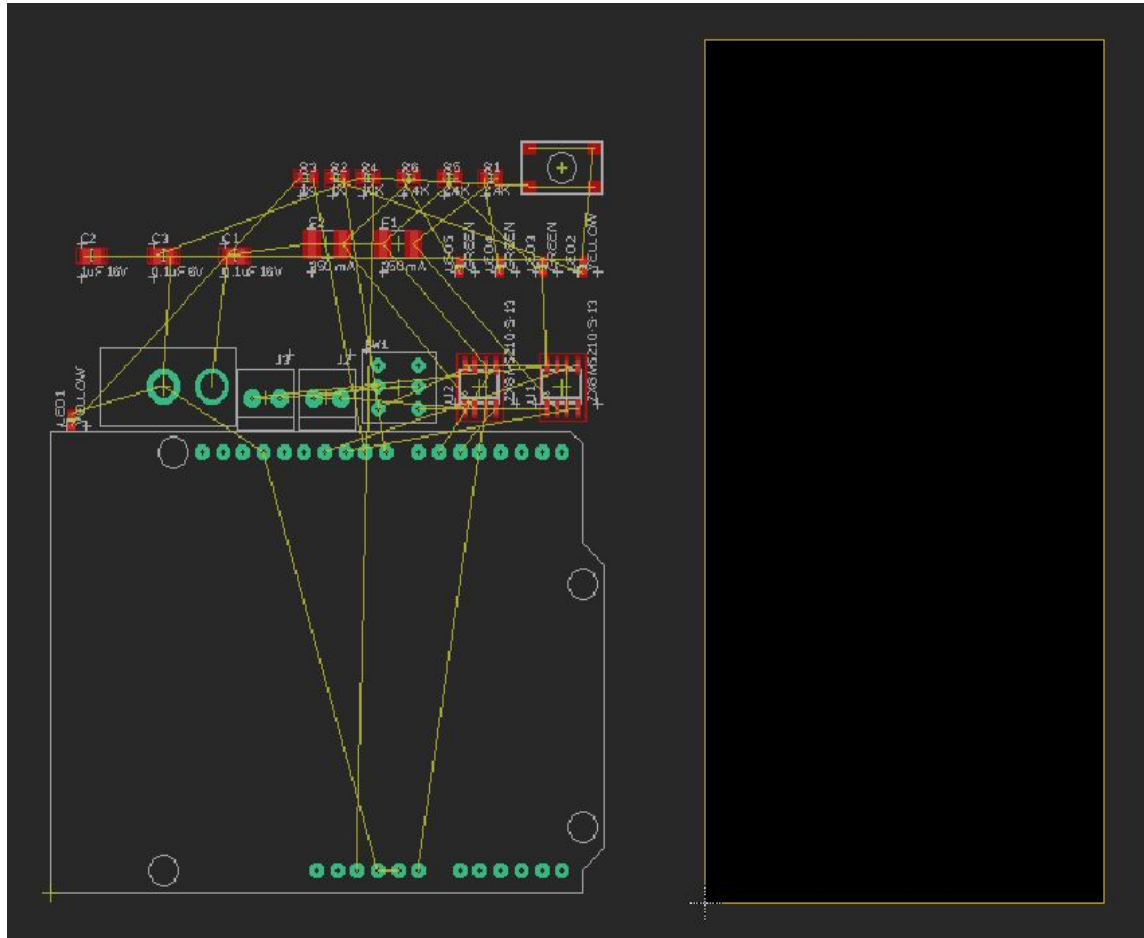
Arranging Components

- Click on crosshair and drag to move parts around (or use command `move`)
- Right click to rotate
- Position components on the board area

Arrangement Considerations

- Minimize airwires
- Maximum size of board
- Clearance between mounting holes and components
- Location of specific components
  - Connectors on board edge
  - Decoupling capacitors near decoupled pins
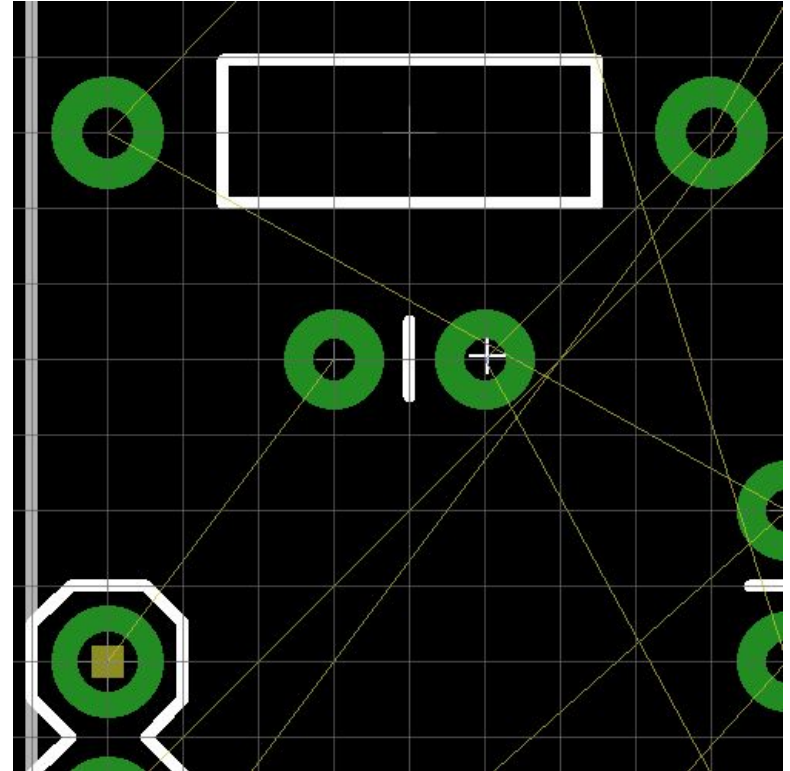 - Communicating/related components near one another

Arranging Components

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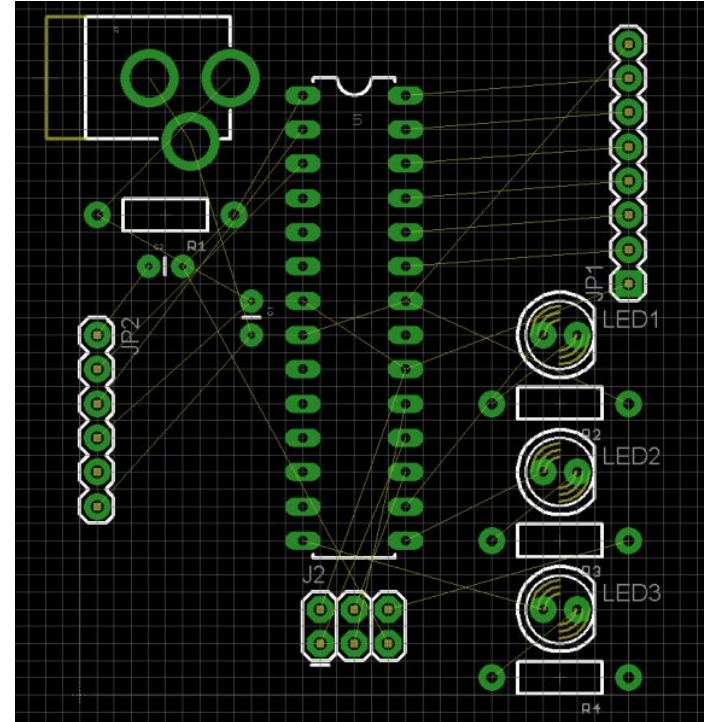
Drawing Traces

- Use the route command
  - Left click on starting point and left click on end point
- Follow start and end of airwire
- Culture of 45 degrees

Changing Layers

- Press "space" to change the layer and put a via
  - If your board has more than two copper layers you can press "space" again until finding the desired layer
  - You can also use the middle mouse button
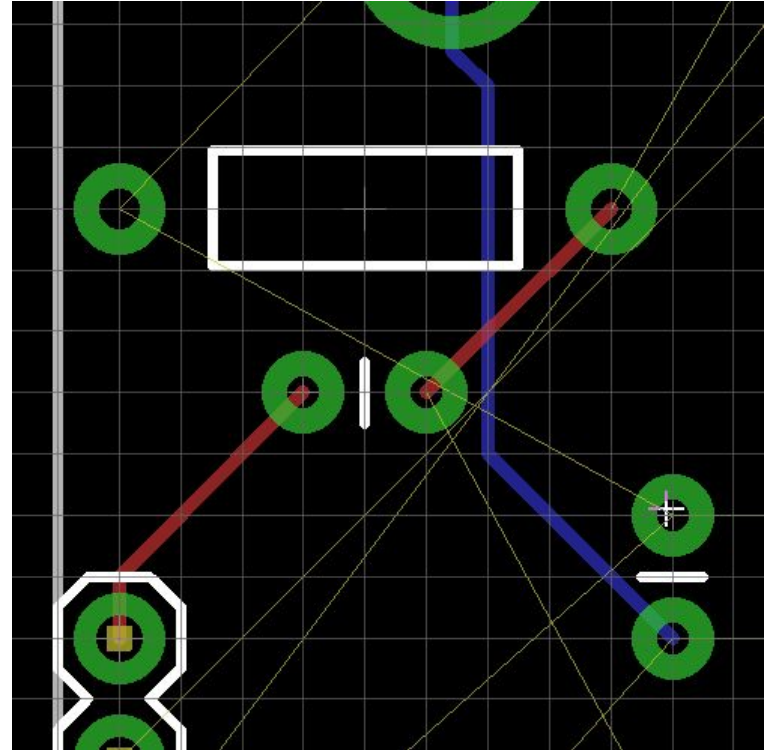
Polygons

- Allows you to fill a drawn area with copper connected to a specific net
- Useful to make "ground planes"
- Draw polygon around area with `polygon` command and name with desired net
- Running `ratsnest` will fill it
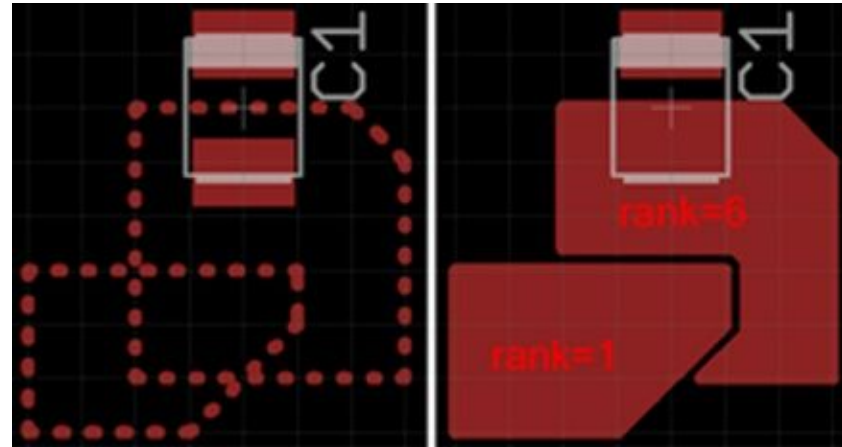
Design Rules and DRC

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.

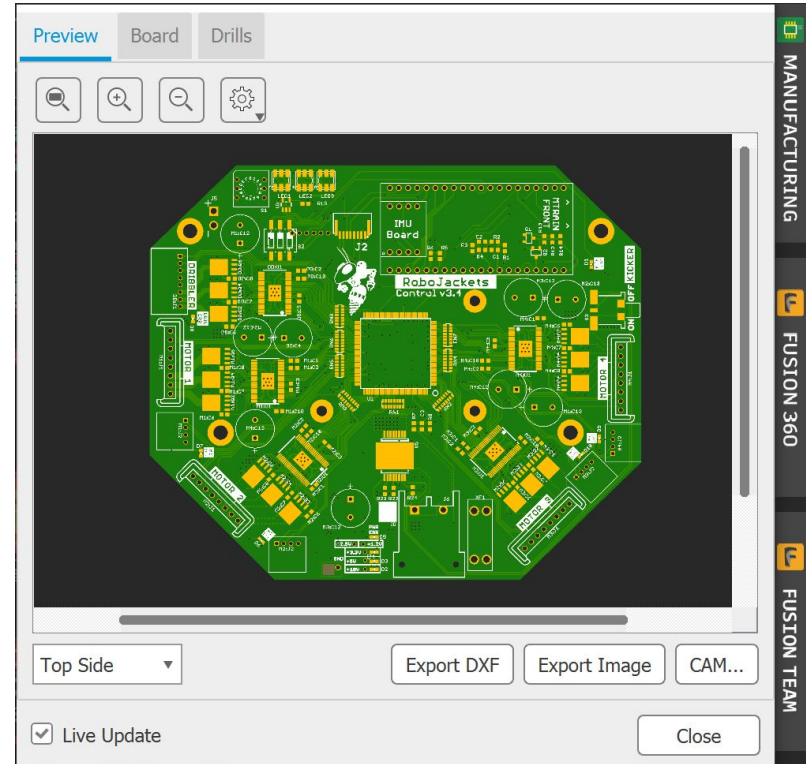Note: Load Design Rules early on your design.

Manufacturing View

Preview

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

Board

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

Relevant Commands

- ratsnest
  - Will update all the polygons and show them
  - Update airwires
- ripup @ ;
  - This will hide the polygons on the board view
- show @ <target_name> (**example:** show @ ENABLE)
  - Will highlight the objects with that target_name on your screen and draw an area around them
  - You can use * on the name for substituting every character (example: show @ EN* will highlight every object that begins with "EN")

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.

Resources

Eagle Training Guide:

https://github.com/RoboJackets/electrical-training/tree/master/references/eagle_training_guide

Eagle Style Guide:

https://wiki.robojackets.org/EAGLE_Style_Guide

How to branch on GIT:

https://github.com/RoboJackets/robocup-firmware/blob/master/doc/Git.md