EAGLE Cheat Sheet

Asha Bhandarkar

July 2019

1 Schematic Window

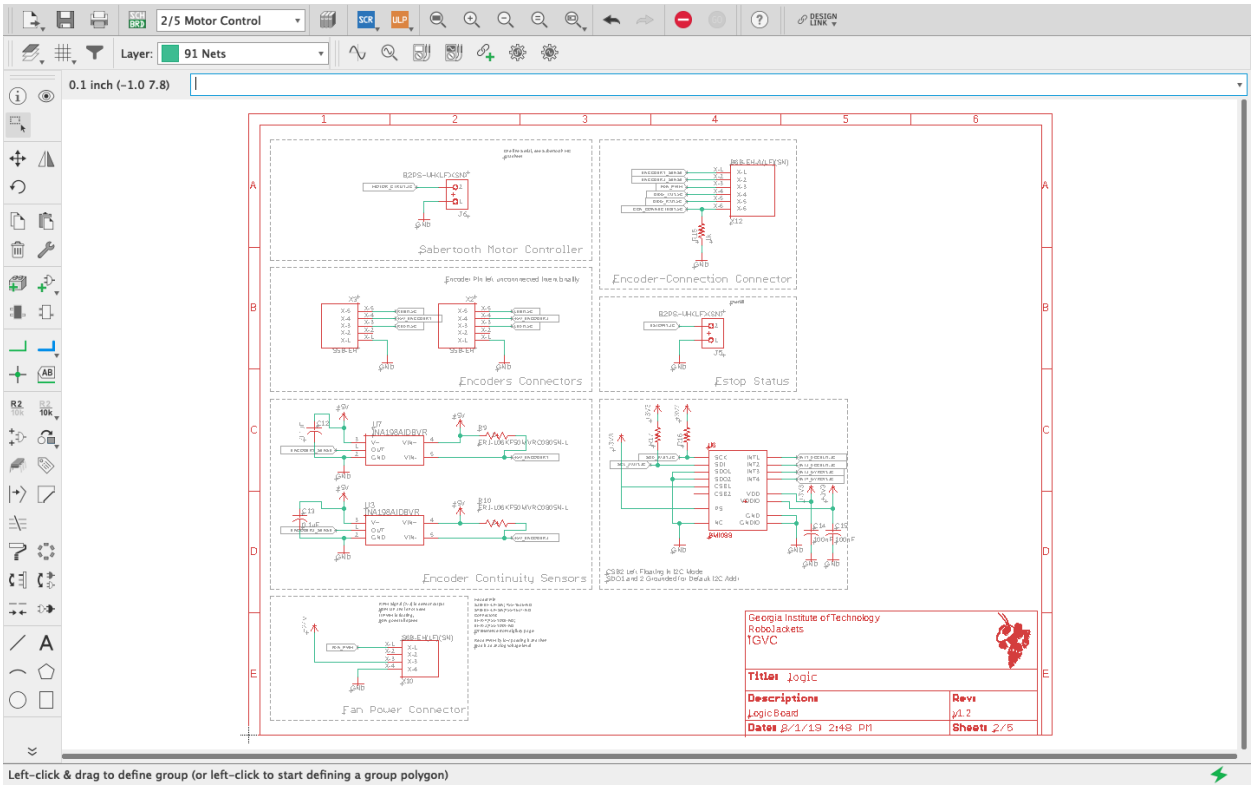

Figure 1: An example of what the schematic window looks like in EAGLE. This is the motor control sheet for IGVC's Logic Board v1.2.

1.1 Libraries

1.1.1 Adding Libraries

- Open the Library Manager using the  icon to add libraries that contain the parts you will need in your design.

1.2 Parts

1.2.1 Adding Parts

- Open the Add window using the  icon or using the `add` command to see available libraries to add parts from.

1.2.2 Viewing Properties

- Open using the  icon or using the `info` command.
  - You can change various properties such as Name, Position, and Angle.
  - You can see the Device, Footprint, and Library of the part.

1.2.3 Positioning

- Use the  icon or the `move` command to move parts.
- Use the  icon or the `mirror` command + left-click to mirror parts.
- Use the  icon or the `rotate` command + left-click to rotate parts.

Note: Many of these commands plus more can be accessed by right-clicking the part.

1.3 Connections

1.3.1 Drawing Nets

- Use the  icon or the `net` command to draw electrical connections between parts.

1.3.2 Adding Junctions

- Use the  icon or the `junction` command to indicate a shared electrical connection.

1.3.3 Adding Labels

- Use the  icon or the `label` command to create breakouts for more complex schematics. These labels display the Name property of the net.
  - Use the top bar to select the XREF label format.

1.4 Organization

1.4.1 Managing Sheets

- Use the 

dropdown to add, remove, and toggle between sheets in a schematic.

1.4.2 Layers

- Use the 

dropdown to toggle between layers in the schematic.

1.5 Review

1.5.1 Running ERC

- Use the  icon or the `erc` command to run error checking on your schematic.

1.5.2 Switching to Board Layout

- Use the  icon to open the board layout window from the schematic window.

2 Board Layout

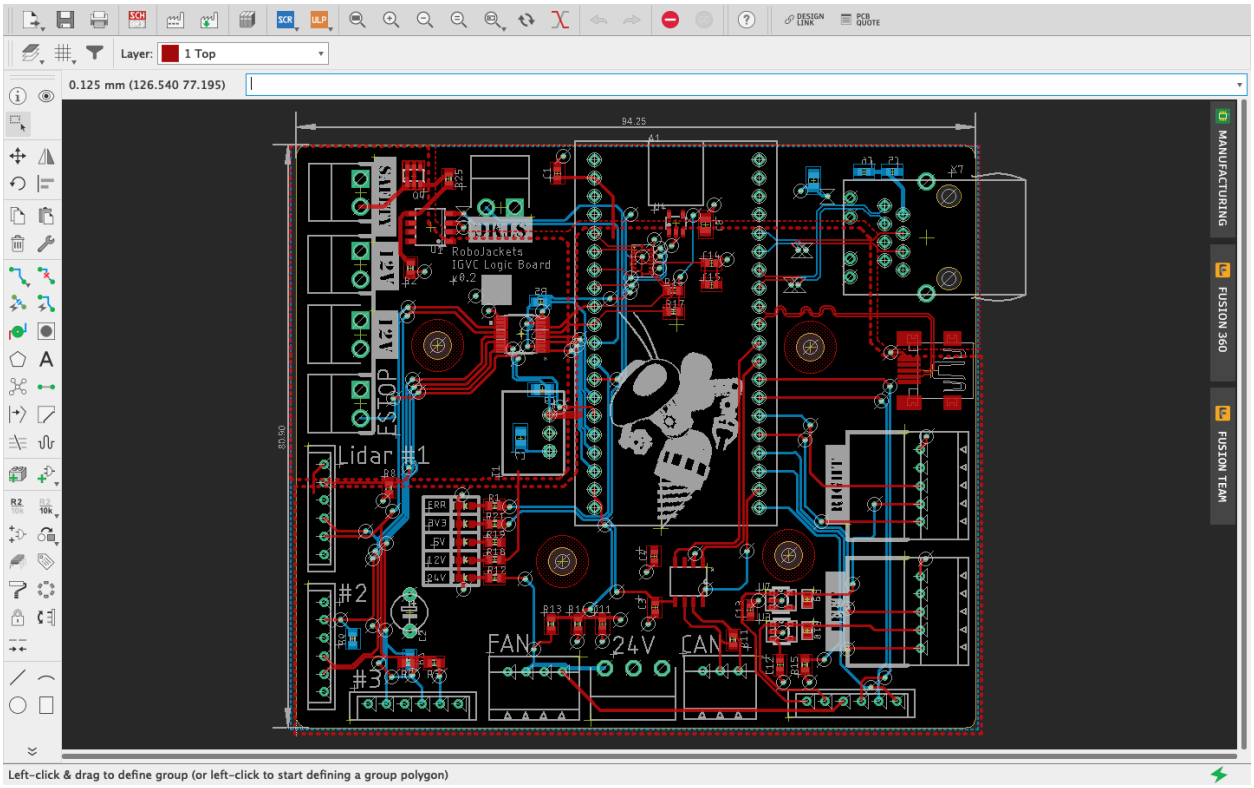

Figure 2: An example of what the board layout window looks like in EAGLE. This is IGVC's Logic Board v1.2.

2.1 Parts

2.1.1 Viewing Properties

- See section 1.2.2

2.1.2 Positioning

- Use the  icon or the `ratsnest` command to readjust airwires (lines that represent electrical connections on the PCB layout) and fill in polygons.
- See section 1.2.3

2.2 Connections

2.2.1 Adding Traces

- Use the  icon or the `route` command to connect airwires.
  - Dropdown on icon gives options to route multiple signals, a differential pair, etc.
  - Top bar gives options to change trace width, obstacle avoidance, trace angle (right-click shortcut), etc.
  - EAGLE does have an autorouter. You **should not** use this to route your boards.

2.2.2 Deleting Traces

- Use the  icon or the `ripup` command to undo traces.
  - Use the `ripup;` command to undo all traces.
  - Use the backspace key to delete segments while routing.
- Use the `ripup <POLYGON NAME>` command to delete polygons.
  - Use the `ripup @ < POLYGON NAME>` command to unfill polygons and `ripup @;` to unfill all polygons.

2.2.3 Adding Vias

- Use the  icon or the `via` command to place connections between layers.
  - Top bar gives option to change drill size (Note that what DRC you use might limit the changes you can make).
  - Use the space bar to add vias while routing.

2.2.4 Drawing Polygons

- Use the  icon or the `polygon` command to draw planes.
  - Thermal pads can be turned off (turned on by default) in the top bar.

2.3 Organization

2.3.1 Layers

- Use the 

dropdown to toggle between layers on the PCB.
  - Use the spacebar to toggle between layers to select the layer you want to route on.
  - Use  icon to access layer settings and create preset layers.

2.4 Review

2.4.1 Running DRC

- Use the 

or the `drc` command to run error checking on your board layout.
- In the DRC pop up window you can load specific DRCs to run.

2.4.2 Switching to Schematic Window

- Use the 

icon to open the schematic window from the board layout window.