

EAGLE Cheat Sheet

Asha Bhandarkar, Varun Madabushi

Oct 2020

1 Schematic Window

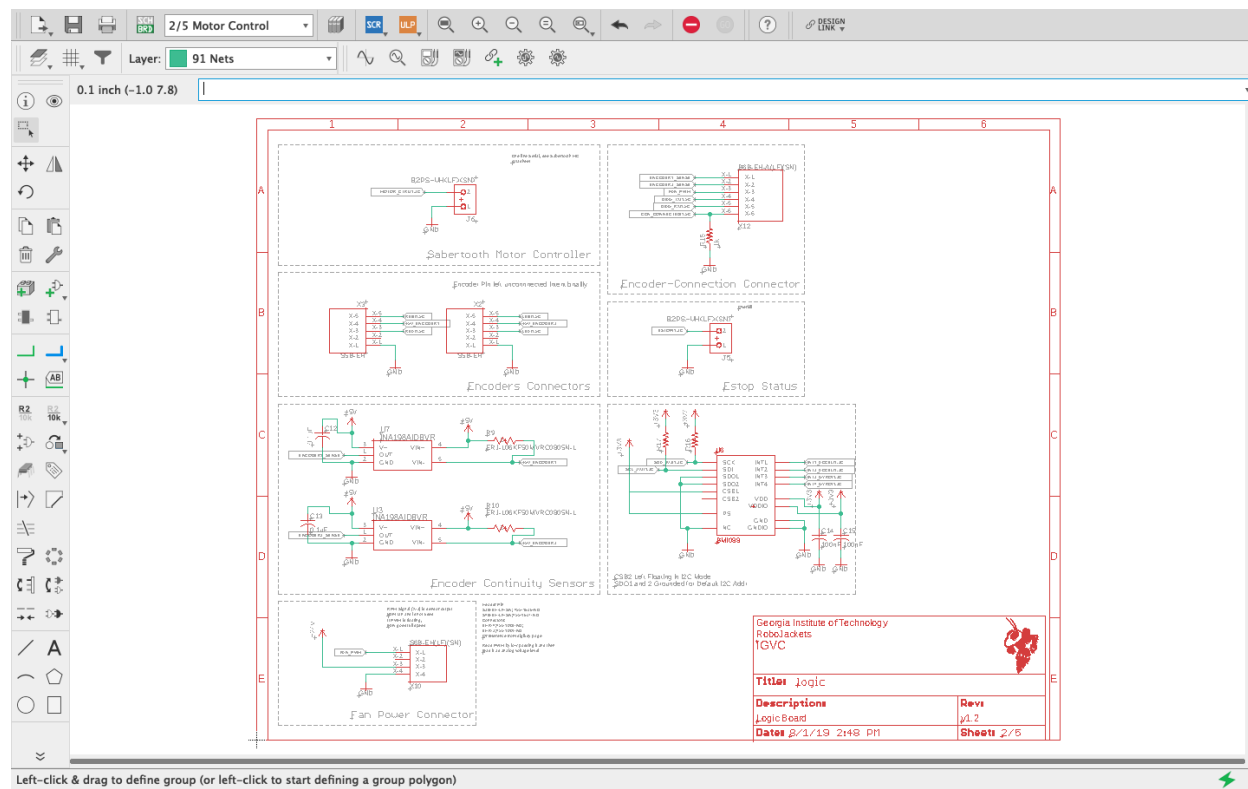



Figure 1: An example of what the schematic window looks like in EAGLE. This is the motor control sheet for IGVC's Logic Board v1.2.


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

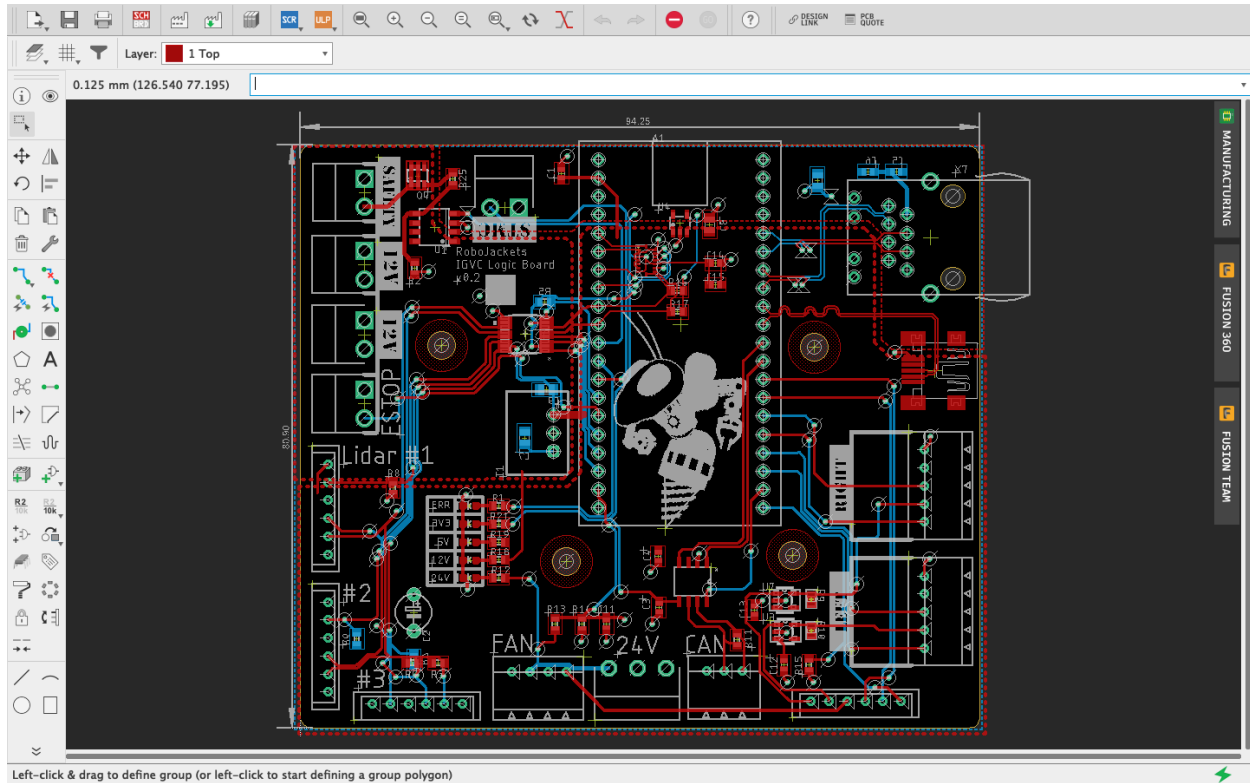



Figure 2: An example of what the board layout window looks like in EAGLE. This is IGVC’s Logic Board v1.2.

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

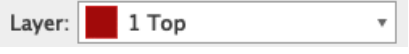

- Use the  icon or the `via` command to place connections between layers.
 - Top bar gives option to change drill size (Note that what DRC you use might limit the changes you can make).
 - Use the space bar to add vias while routing.

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.

3 Miscellaneous

- Inputting the command `set Interface.UseCtrlForPanning 1` into the EAGLE command line uses the CTRL key for panning instead of middle mouse wheel. Very useful when using a trackpad!
- Selecting the **Options > User Interface** menu in the top ribbon lets you select dark or light theme for the schematic and layout windows.

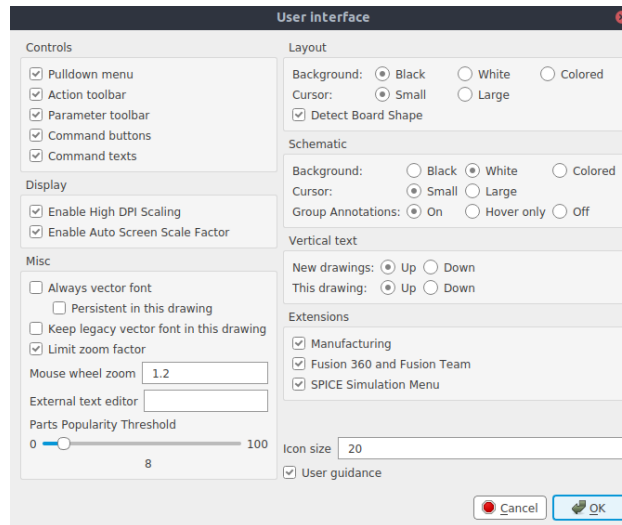


Figure 3: The User Interface Menu in EAGLE