

Autodesk Eagle CAD Guide

Evan Peterson

August 21, 2018

Contents

1	Introduction	2
1.1	What is a PCB	2
1.2	Eagle Intro	2
1.3	Eagle Installation	2
2	Libraries	2
2.1	Create Library	3
2.2	Library Part: Symbol	3
2.3	Library Part: Footprint	4
2.4	Library Part: Device	5
3	Schematic	6
3.1	Schematic: Adding Parts	6
3.2	Schematic: Wiring	7
3.3	Schematic: Error Checking	7
4	Board Layout	8
4.1	Board Layout: Layers	8
4.2	Board Layout: Arranging	8
4.3	Board Layout: Routing	9
4.4	Board Layout: Polygons	10
4.5	Board Layout: Silkscreen	11
4.6	Board Layout: Error Checking	11
4.7	Board Layout: Exporting	12

1 Introduction

1.1 What is a PCB

A printed circuit board (PCB) is used to mechanically support and electrically connect various electrical components. A PCB is designed and manufactured based on how you want to connect the components then the components are soldered to the PCB.

In its most basic form a PCB consists of a center substrate (typically fr4) with a layer of copper traces on the bottom and top. Pads are portions of the copper layer that are in specific shapes so a component can be soldered to it.

1.2 Eagle Intro

Eagle is a CAD tool used to design printed circuit boards (PCBs). The program is split up into three sections which build upon one another to result in files that are sent to a PCB manufacturer.

First all parts to go on the board are created in libraries which describe the pins on the part and the footprint of the part for when it is soldered to the board. Then these parts are used in the Schematic design which describes the electrical connections on the board. Then the board is laid out by following the parts and connections made in the schematic to place parts on the physical board then route where the physical connections will go between parts.

Tips & Tricks

- Everything in Eagle can be done through clicking buttons or by using its command line. Design can be done much quicker by learning to use the command line.
- In order to use some Eagle functions such as creating new files or adjusting certain settings you must switch back to Eagle's Control Panel window

1.3 Eagle Installation

1. Download at <https://www.autodesk.com/products/eagle/free-download>
2. Install Eagle using free or educational version
 - Free version: 2 Board Layers, 80cm² Board Area
 - Educational version: 16 Board Layers, Unlimited Board Area
 - <https://www.autodesk.com/education/free-software/eagle>

2 Libraries

Libraries are used to store information about a part which are used in the schematic and the board layout. Eagle has built-in libraries for many common parts but very often you have to make your own.

Libraries are split into three parts. Symbol, Footprint and Device. The Symbol is what is seen in the schematic and is how you describe the pins on your part which electrical connections go to. The Footprint is the exposed metal part used for board layout which you will solder the component to on the physical board. The Device brings the Symbol and Footprint together so what is created in the schematic can be translated into the board layout.

2.1 Create Library

1. (At Control Panel) ***File→New→Library***
2. Save to a directory where you will keep all Eagle libraries

Tips & Tricks

- Libraries are commonly split up into components types such as resistors or connectors
- Eagle has a default folder it searches for its libraries, but you can have it search additional folders for your libraries by (At Control Panel) ***File→Options→Directories*** then appending to the Libraries path a colon then the path to your directory

2.2 Library Part: Symbol

1. Create new symbol: ***Library→Symbol***
2. Name symbol with part number
3. Draw box using ***wire*** (Type into console or select GUI icon from left)
 - This aspect is purely visual so any size/shape is okay, but use standard symbols which are commonly represented in schematics helps everybody understands your work.
4. Add necessary amount of pins using ***pin***
 - These pins are what wires are connected to in the schematic
5. Name pins corresponding to datasheet using ***name*** then click on pin
6. Clean up Symbol: Adjust part outline so all pins fit and pin names can be seen
7. Add Name and Value using ***text***
 - >NAME and >VALUE are keywords for text that are automatically filled out with the name and value when inserted into a schematic.
8. Change Layer of Name and Value using ***info*** then clicking on the part and changing Layer to Name and Value respectively
9. Return to Library: ***Library→Table of Contents***

Tips & Tricks

- Always keep pins snapped to the 0.1 in grid (Hold option/alt and click components to snap them).
- Right click while placing or moving to rotate.
- To move a group use ***group***, select what you want to move then ***Right Click→Move:Group***.
- Objects are clickable and selectable from the grey + symbol when they have been inserted into schematics/layout.

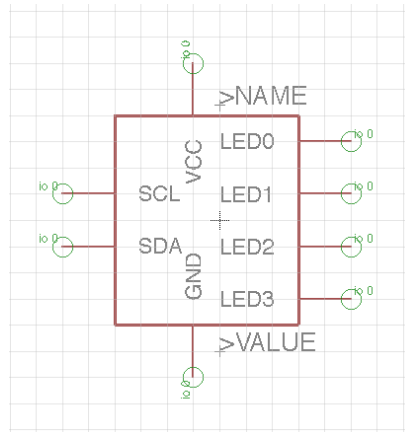


Figure 1: Example of completed symbol

2.3 Library Part: Footprint

This is typically the footprint for the component which is found in the datasheet for the part. In the datasheet you will find all measurements needed to create the footprint. This footprint will go on the PCB and is what the component will be soldered to. Before version 9.0, Eagle uses 'package' for 'footprint'.

1. Create new footprint: ***Library***→***Footprint***
2. Name footprint with package name
 - Common for multiple parts to have the same footprint.
3. Place first pad
 - Place surface mount using ***SMD***
 - Place through hole pad using ***pad***
4. Resize pad using ***info*** clicking on pad then changing SMD size to size given in datasheet
5. Use ***copy*** to copy correct amount of pads and place them in approximate positions
6. Use values given in datasheet to calculate position (to pad center) for each pad then move pads using ***info*** and change position values
7. Rename pads using ***rename***. Pad numbers given by datasheet
8. Add part outline using ***line*** then changing Layer dropdown to ***51 tDocu*** then draw basic part outline
 - The tDocu is just for reference to ensure no overlapping parts so it will not appear on the board.
9. Adjust part outline to the dimensions given on the datasheet. Adjust to exact values using ***info***
10. Add silkscreen
 - Silkscreen gets printed on the board and is used to help place components

- Draw lines on the *21 tPlace* layer as an outline that does not intersect with pads
- Add a dot to indicate where Pin 1 is on component to help when soldering component

11. Add Name and Value using *text*

- Use >NAME and >VALUE similar to in symbol
- Adjust Layer to Name and Value respectively

12. Return to Library: *Library*→*Table of Contents*

Tips & Tricks

- Change grid size using *grid*
 - Change grid size to whatever is helpful to you.
 - Use same units as given in datasheet. Values for position and size are in these units.
 - Create vias/pads that are bigger than physical component sizes will help you phenomenally.
 - Recommended: Size = 0.5mm, Alt = 0.25mm.
- Use positions such that (0,0) is at the center of the component

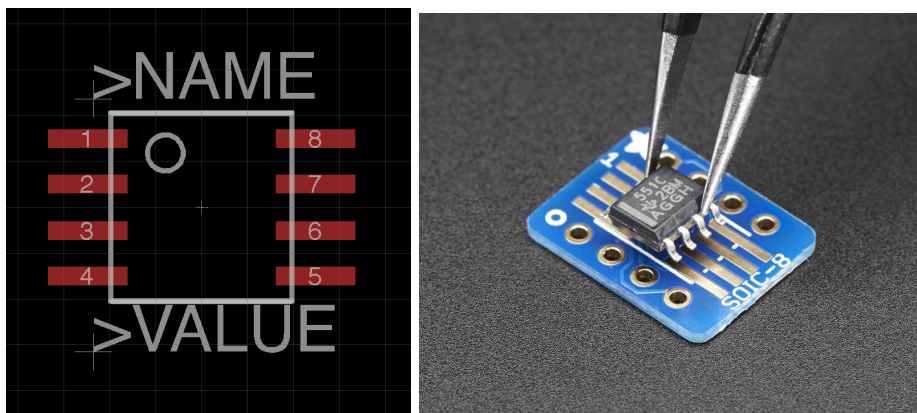


Figure 2: Example of completed footprint and how the physical component goes on a footprint that was created

2.4 Library Part: Device

This is where the Symbol and Footprint come together to create what will be added to the schematic.

1. Create new device: *Library*→*Device*

2. Name with general part number

- Not footprint specific because a single device can have multiple footprint

3. Add symbol using **add** then select previously created symbol
4. Place symbol so grey + is in center of component
5. Click new button in footprint window and select local footprint
6. Connect pins by clicking connect button in footprint area
7. Use datasheet to see what pin names correspond to what pad numbers then connect by selecting pin and pad then click connect. Repeat for each one
8. Click prefix button to add current prefix to match component type

Tips & Tricks

- A wildcard character ***** can be used in Device name to allow you have names for different versions of the part without creating a new device. Example: You have parts A45C and A46C that have the same pins and footprint, but have some minor internal difference. You can name device A4*C then specify two technologies: 5 and 6.
- You can add multiple footprints and set the variant name to specify the footprint. This variant name is appended to the end of the Device name

- Done with library part and ready to be added to schematic
- Many more parts can be added to this library. Parts that share footprints only need footprint to be created once so check if that footprint exists first so you don't have to remake it.

3 Schematic

Schematics are used to set up how the board is organized by adding all the parts used on the board and making the electrical connections between these parts.

1. (At Control Panel) **File**→**New**→**Schematic**
2. Save to a directory where you are storing this project

Tips & Tricks

- Always keep parts and wires snapped to 0.1 in grid

3.1 Schematic: Adding Parts

1. Prepare Eagle with necessary libraries
 - Libraries can be added by automatically searching a folder as shown in section 2.1
 - Libraries can also be temporarily added by **Library**→**Use** then select library you created
2. Add components using **add** then select the library containing the component then select the component

- Add a frame to keep schematic organized
 - Add all components needed for schematic
 - Resistors and Capacitors are found in rcl library. R-US and C-US are the parts commonly used and the SMD footprint 0603 is commonly used (parts named R-US_R0603 and C-USC0603 respectively)
3. Change value of parts using *value*
 - This value has no effect on the board, only used as reference

Tips & Tricks

- When adding components you can search for components but Eagle uses exact matching so you must add wild cards. Example: you want to search for part number containing 0603, search using *0603*

3.2 Schematic: Wiring

1. Organize components
 - Components need to be placed in a way so it is easy to read
2. Use the *net* tool draw connections between components
 - NOT the *wire* tool. The wire tool is only used for visuals, does not make any actual connections
3. Connect power and ground using parts from the supply library. All supply parts that are the same on a schematic are connected together
 - Good practice to have ground supply parts pointing down and power supply parts pointing up

Tips & Tricks

- Connections can be made without running nets between components. This is done by using *name* on a multiple nets and naming them the same thing. All nets with the same name are treated as being connected.
 - Pins cannot be named, add a short net connecting to the pin then name that net
 - Name these nets something useful so it is easier to follow
 - To see that nets are named the same and connected, you can use *label* on a net and place the label at the end of the net
- Connections can be checked with *show*

3.3 Schematic: Error Checking

- Use the Electrical Rule Check (ERC) to check for electrical errors and warnings
- The ERC is often useful to show connections you missed
- Many warnings can be ignored from the ERC because they often have to do with net names, etc.

4 Board Layout

The board file is how you layout the final physical board. All steps up to this point have been to assist you in the creation of this.

1. When done with schematic switch to board by **File→Use**
 - If no board file is found one will be generated
 - It is generated with parts from the schematic randomly placed and lines showing the connections to be made between parts called airwires

4.1 Board Layout: Layers

- What is actually printed:
 - Top Silkscreen (printed on top of board for reference, typically white)
 - Top Soldermask (applied over copper to protect from solder, typically green)
 - Top Copper (makes electrical connections and components are soldered to)
 - Substrate (FR4: used to support and separate copper layers)
 - Bottom Copper
 - Bottom Soldermask
 - Bottom Silkscreen
- Use **layer** to adjust what layers are currently viewed

Tips & Tricks

- Right click on Layer settings to create a group of currently viewable layers or quickly switch between created groups

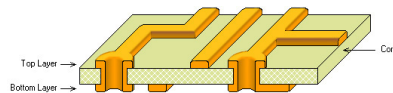


Figure 3: Sideview of layers that are created when PCB is manufactured

4.2 Board Layout: Arranging

- Use **move** and right click to rotate
- How you arrange your parts has a large impact on the routing difficulty in the next step
- First consider the requirements of your board
 - Consider the maximize size you want your board to be
 - Consider Location of specific parts
 - * Specific locations of connectors and other components that need to be accessible
 - * Decoupling capacitors very close to its IC

- * Maximum distances parts can be that must communicate
 - Consider mounting holes
 - Consider clearance with other boards or objects around it
- Then consider what is easiest placement for when you route the board
 - Leave space between parts so no parts collide and so you have enough space for routing
 - Group parts together based on function, referring back to schematic. Reduces required routing distances
 - Minimize intersecting airwires. It is much more difficult to route when airwires cross
 - Recommended keeping as many components as you can on the top layer so you can use the bottom layer for routing or ground plane
- Use *delete* to erase existing dimension then use *wire* with the layer set to **20 Dimension** then draw a new dimension which all your parts are within

4.3 Board Layout: Routing

- Routing is making all the connections shown by the airwires without overlapping anything
- Two layer board so you can place components and route on the top or the bottom
 - Vias are used to connect between the top and bottom layers
- Don't worry about ground connections yet, a ground plane will be added later
- Use *route* (not *wire*)
 - Use layer dropdown to select signal layer you are routing on
 - Optional selection for walkaround obstacles to assist you with routing to avoid overlapping based on DRC rules
 - Bend style is the angle of wires when routing (**good practice to use 45° angles**)
 - Width is how wide the copper trace is (recommended 0.3mm)
 - * Trace width must be considered for some applications such as power lines
 - * Signal wire width should be sizable compare to their footprint.
 - Via settings are for size and shape of via (recommended circle with drill of 0.35mm)
- Start from end of an air wire and route to the other end of that airwire
- Left click when routing to place segment and continue routing
- Ensure no overlap when routing
 - For pads and traces on the same layer there must be zero overlap
 - Traces on separate layers can overlap
 - Vias and through hole components (colored in green) are on both layers so traces cannot overlap with these on either layer

- Vias can be added by changing layers while routing (using middle click), or change layer (in top left drop-down list) or by adding manually with the via tool
 - Vias added manually do not have the net automatically set so must use ***name*** to change name of via to same as net you want to connect it to
- Different PCB manufacturers have specifications for the minimum distance they can produce between traces
 - You must ensure your traces are at least that far apart
- Traces and vias cannot be removed with ***delete***, you must use ***ripup***

Tips & Tricks

- Right click while routing to switch bend styles
- Use ***ratsnest*** to recalculate airwires to shortest length

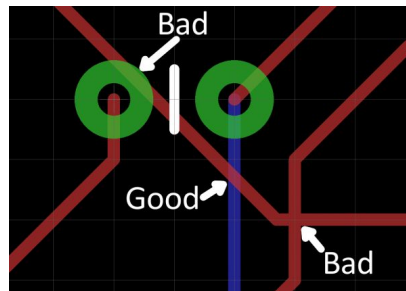


Figure 4: Example of allowable routing connections

4.4 Board Layout: Polygons

Polygons are how large sections of copper (like ground planes) are done. Using a ground plane allows you to connect many ground pins without routing as well as other benefits.

1. Use ***polygon*** to draw the shape around you want to fill
 - For ground plane you typically want to fill the whole board so draw the polygon along the dimension lines
2. Use ***name*** to connect this polygon to the net with the same name
3. Use ***ratsnest*** to fill in the polygons

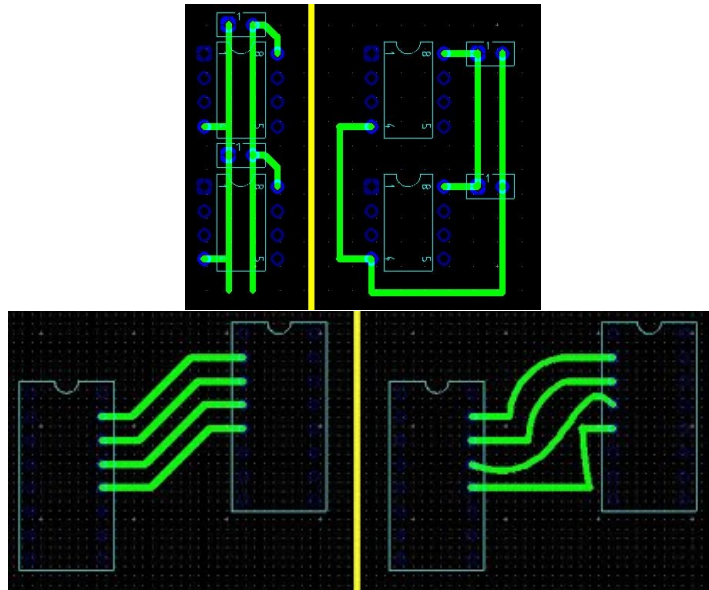


Figure 5: Example of Good vs Bad Routing Plans

Tips & Tricks

- Change the polygon's rank when you have intersecting polygons to give one higher priority
- Add polygon inside of another and change Polygon pour to cutout if there is a section of the board you don't want a polygon
- Check or uncheck thermals based on application (typically good idea to use when polygon overlaps pads so the component is easier to solder on)

4.5 Board Layout: Silkscreen

- Silkscreen is printed on the top and bottom of the board and has no effect on the function of the board. Rather it is used to identify parts on the board and to help with component soldering.
- Most of the silkscreen comes from individual parts on Names layer identifying part names
- Often these names for parts are misaligned. Use *smash* on the component to allow you to move these labels
- Add other markings to silkscreen on layers tPlace or bPlace for the top and bottom silkscreen respectively

4.6 Board Layout: Error Checking

- Use *ratsnest* then check the notification at the bottom left for how many airwires are left to see if there is stuff left to route. Everything is routed if "Nothing to do"
- Use the Design Rule Check (DRC) to check for clearance, overlap, etc

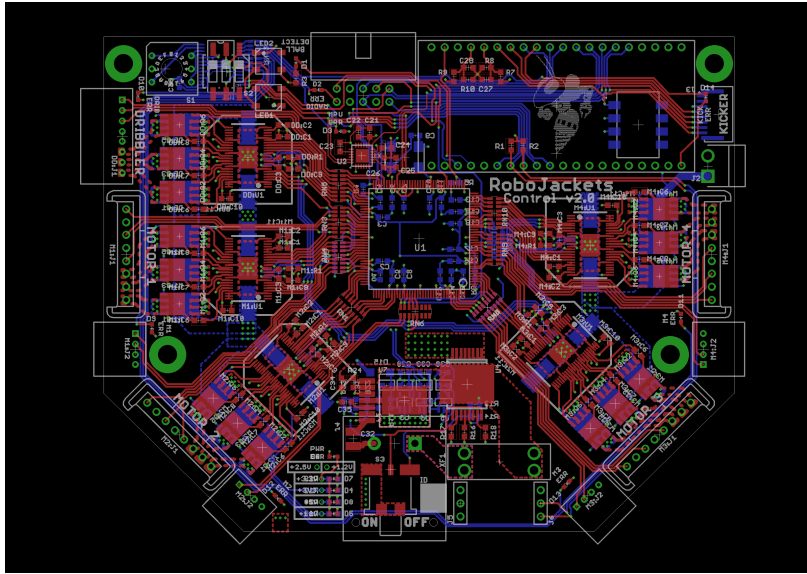


Figure 6: Example of fully completed board

- Settings of DRC can be changed to fit the requirements of where your board is being printed
- When "Ratsnest: Nothing to do" and "DRC: No errors" you are done

4.7 Board Layout: Exporting

You must export the board into Gerber files which are sent to the manufacturer to print. This is a set up multiple files, one for each layer

1. Use **File**→**CAM Processor** then **File**→**Open**→**Job** and select gerb274.cam
 - There is also a 4 layer version of this CAM
2. This exports a file for each layer with the exception of of bottom silkscreen. This can be added if desired by clicking add and selecting Dimension, bPlace, and bNames
3. Repeat CAM Processor with excellon.cam job to generate drill file