

Designing a PCB using EagleCAD

Atlanta Hobby Robot Club
<http://www.botlanta.org>

Yes, you can design your own custom PCB and this tutorial will show you how. It is easier than ever to design your own PCB and having it manufactured is not as expensive as you may think. This tutorial will show you how to use CadSoft EagleCAD to design a schematic, layout a PCB, and generate Gerber files to send to a PCB manufacturer. As an example project, we will make a USB Night Light. Powered by a USB port, this circuit board has a light sensor and will turn on a couple LEDs when the ambient light drops below a set level.

Special thanks CadSoft for offering the free hobbyist version of EagleCAD and also to Sparkfun.com who has created some of the best Eagle CAD introductory tutorials on the internet.

WHY DESIGN A CUSTOM PCB?

Pros

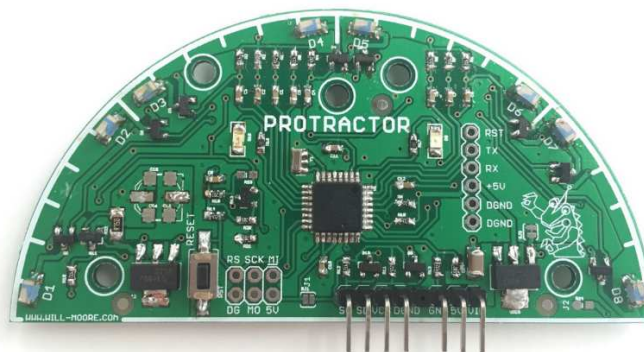
- Small Size and Weight
- Place components wherever you want
- Use any component you want
- Use PCB as robot structure
- Use Surface Mount parts
- Reliable connections
- Easier to replicate

Cons

- Takes time to design
- Hard to reconfigure / rework
- Not good for initial prototyping
- Need to learn electronics
- Need to learn soldering



The Pololu 3pi Robot demonstrates what can be done with a Custom PCB



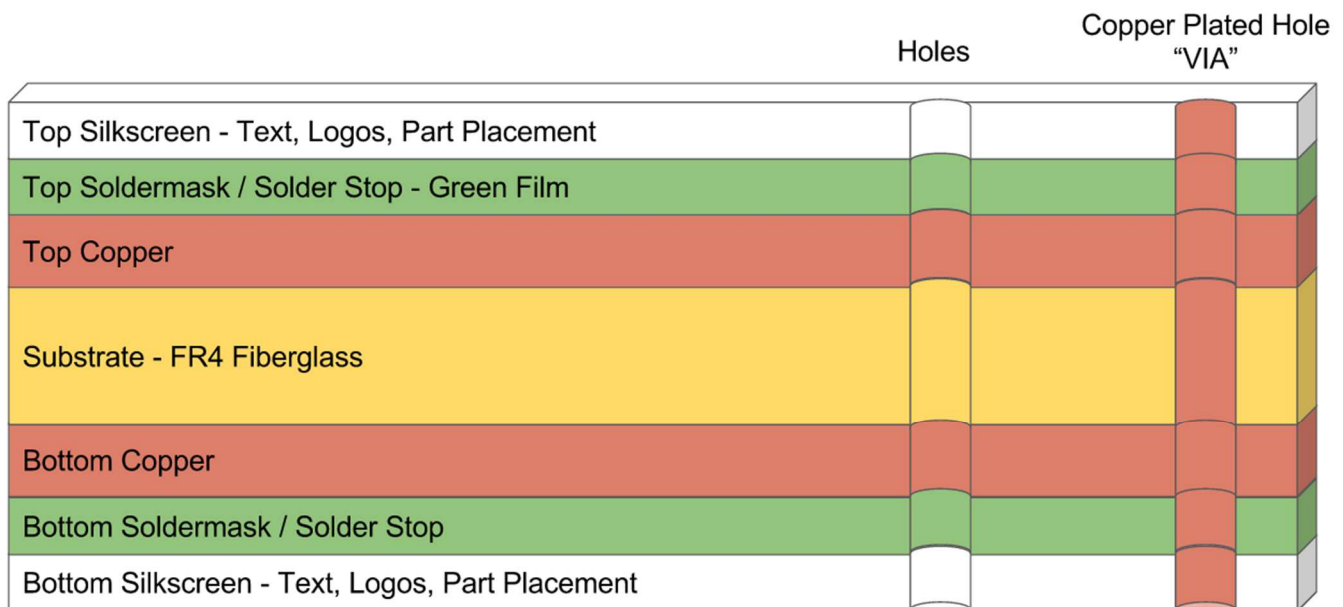
An IR Proximity Sensor designed by an AHRC member

PCB Layers

Before we design a circuit board, let's take a look at the layers that make up a PCB.

- Substrate – Typically made of Fiberglass, this layer provides mechanical strength to the board.
- Copper – This is where the wires are that connect all the parts. In a PCB, these wires are called Traces.
- Soldermask – A layer of insulating film that covers the Copper. Gaps in the Soldermask expose sections of copper so that the parts can be connected to the Traces with solder.
- Silkscreen – A layer of ink that is printed on top of all other layers, typically white or black. Reference designators, names, values, logos, or any other useful information can be put on the Silkscreen.
- Via – Copper-plated hole used to make connections between copper layers

The above layers are repeated on the bottom side of the board in addition to the top, creating a 2 layer circuit board. Most hobbyist projects work well with 2 layer boards, although it is possible to have boards with many internal layers.



The layers of a PCB

EagleCAD Layers

EagleCAD also has layers. In the Schematic view it's perfectly fine to use the default layers. Since a circuit board has a top, bottom, and layers in between, it will be useful to know what the layers are in the Board view so they can be turned on and off as needed.

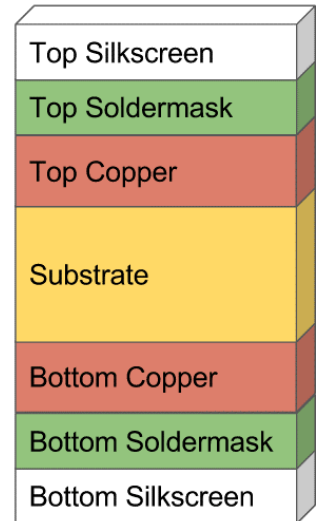
- Top, Bottom - Copper Traces, these are the wires connecting parts
- Pads - Where to solder connections are made between Parts and the Board
- Vias - Copper plated through-holes that connect traces on Top to Bottom
- Unrouted - Shows what connections need to be made
- Dimension - The outline of the PCB
- tPlace, bPlace - Shows outlines of the part locations
- tNames, bNames - Reference Designators of the parts, e.g. R5, C1, etc.
- tValues, bValues - Part Values, e.g. 10k, 0.1uf
- tOrigins, bOrigins - Small + signs you click on to move the parts around
- tStop, bStop - Where the Green film stops, exposing the copper
- Holes, Drills - Holes are not copper plated, Drills are for Vias and are plated
- tDocu, bDocu - Similar to tPlace/bPlace but more detailed
- tRestrict, bRestrict – Restricts where the copper can fill in a pour
- tKeepout, bKeepout – Indicates areas in which parts should not be placed

Gerber Files

Gerber Files are a common file format used to document a PCB design to send to a manufacturer. Each Gerber File represents a layer of the PCB. To generate each Gerber File, the appropriate layers in EagleCAD must be assigned to a layer of the PCB.

File Ext	PCB Layer	Eagle Layers
.GTO	Top Silkscreen	tPlace, tNames, tValues ¹ , Dimension ²
.GTS	Top Soldermask	tStop
.GTL	Top Copper	Top, Pads, Vias
.GBL	Bottom Copper	Bottom, Pads Vias
.GTS	Bottom Soldermask	bStop
.GTO	Bottom Silkscreen	bPlace, bNames, bValues ¹ , Dimension ²
.TXT	Drills File	Holes, Drills

1. Placing Values on the Silkscreen is optional
2. Sometimes Dimension will be placed in Copper layer instead of Silkscreen



What You Need

Before we get started, a number of items must be downloaded and installed. Sparkfun.com has a tutorial for installing and setting up EagleCAD which can be very helpful. Here's what we need:

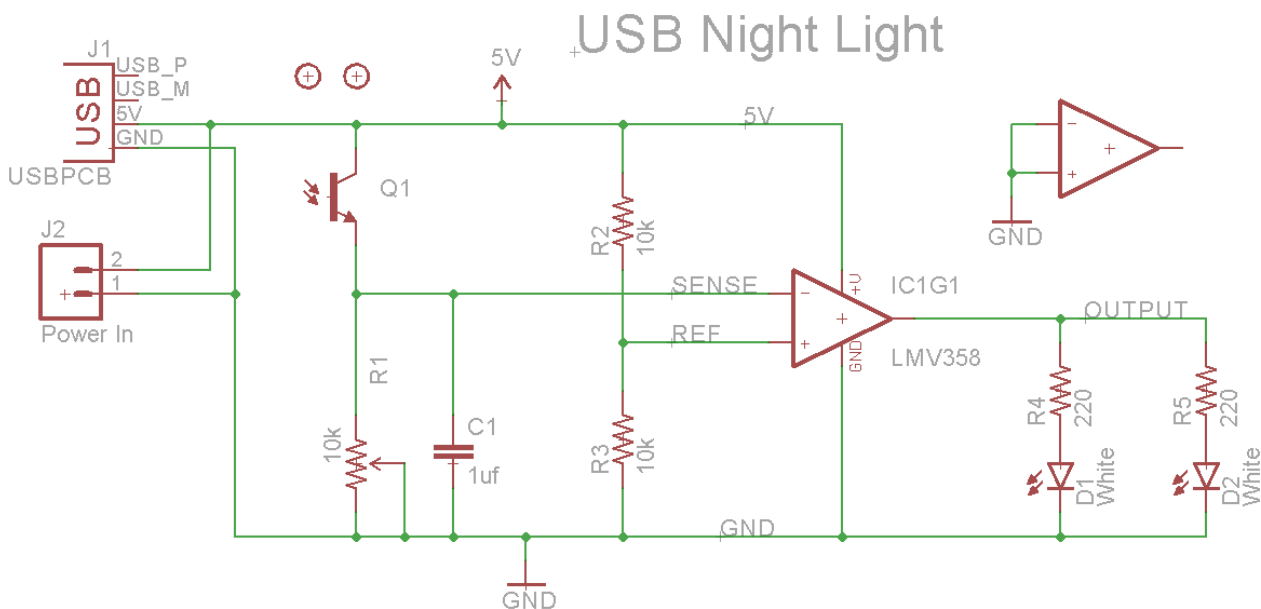
- Download and Install CadSoft EagleCAD: www.cadsoftusa.com/download-eagle/freeware
- Sparkfun Component Libraries: <https://github.com/sparkfun/SparkFun-Eagle-Libraries>
- Sparkfun Design Rule Check: <http://www.sparkfun.com/tutorial/Eagle-DFM/SparkFun.dru>
- Sparkfun CAM file: <http://www.sparkfun.com/tutorial/Eagle-DFM/sfe-gerb274x.cam>
- Viewplot Gerber Viewer: <http://www.viewplot.com>

How to Make a Schematic in Eagle

1. Open Eagle -> Control Panel window appears
2. Options - Directories, Add file path of Sparkfun Libraries as Directory if not already
3. File -> New -> Project -> USB Night Light
4. File -> New -> Schematic -> Save As "USB Night Light R1"
5. Control Panel -> Libraries -> Sparkfun...
 - a. Add all the Parts in the PARTLIST below to the Schematic
 - b. When adding the LMV358 Op Amp, click twice on the schematic window so that both gates appear
6. Use the Move, Rotate and Mirror commands to orient the symbols on the schematic as shown below
7. Use the Wire command to connect parts, see schematic below
 - a. Verify "NETS" layer is selected in the toolbar
8. Add Junction dots at all the net intersections
9. Use the Label command to display the names of some of the Nets -> N\$1, N\$2
10. Use the Name command to rename the Nets -> 5V, GND, SENSE, REF, etc.
11. Use the Value command to assign values to each component
12. Use the Text command to add a name and/or description of the circuit
 - a. In the Toolbar, change to the "INFO" layer, change the text size if desired
13. Use the ERC tool to perform an error check
14. Correct or Approve any Errors and Warnings found

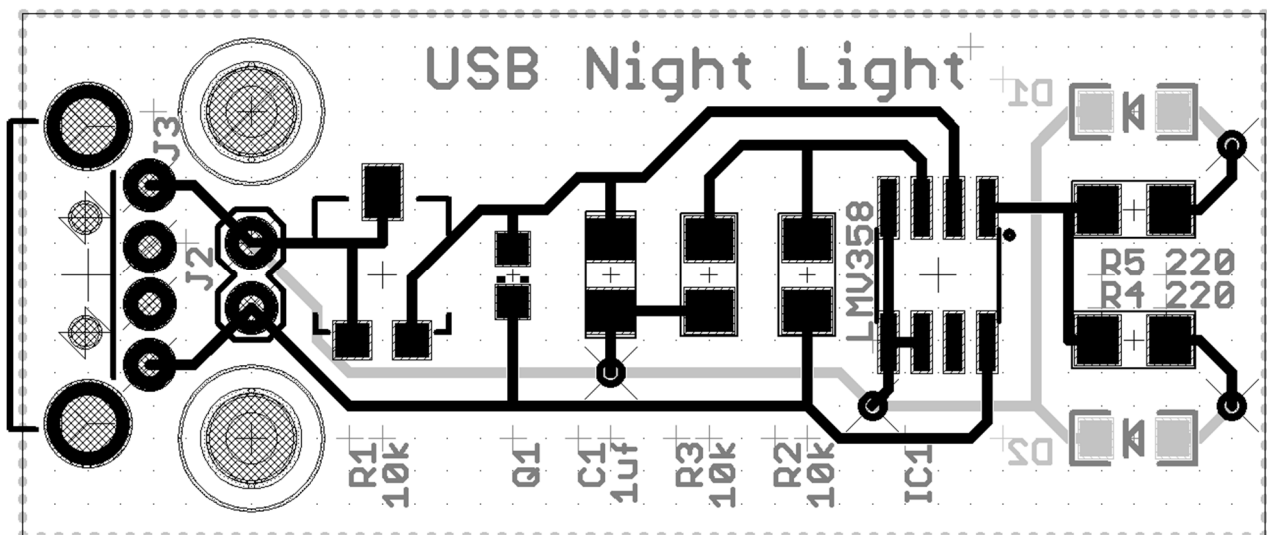
PARTLIST

PART	VALUE	DEVICE	PACKAGE	LIBRARY
C1	1uf	CAP1206	1206	SparkFun-Capacitors
D1		LED1206	LED-1206	SparkFun-LED
D2		LED1206	LED-1206	SparkFun-LED
IC1	LMV358	LMV358S0IC	S008	SparkFun-AnalogIC
J2		M02PTH	1X02	SparkFun-Connectors
J3	USB-A-H	USB-A-H	USB-A-H	SparkFun-Connectors
Q1		PHOTOTRANSISTOR	PT17-21C-L41-TR8	SparkFun-DiscreteSemi
R1	10k	TRIMPOT	TRIMPOT-5MM-V2	SparkFun-Electromechanical
R2	10k	RESISTOR1206	1206	SparkFun-Resistors
R3	10k	RESISTOR1206	1206	SparkFun-Resistors
R4	220	RESISTOR1206	1206	SparkFun-Resistors
R5	220	RESISTOR1206	1206	SparkFun-Resistors
		STAND-OFF		Sparkfun-Electromechanical
		5V		Sparkfun-Aesthetics
		GND		Sparkfun-Aesthetics



How to Make a PCB in Eagle

1. In the Schematic window, go to File -> Switch to Board. Opens a new window where the PCB is created.
2. Click on "Display". Verify the useful layers are visible:
 - a. Top, Bottom, Pads, Vias, Unrouted, Dimension, tPlace, bPlace, tOrigins, bOrigins, tNames, bNames,
 - b. tValues, bValues, tStop, bStop, tKeepout, bKeepout, tRestrict, bRestrict, Drills, Holes, tDocu, bDocu
3. Open the "Grid" and verify the Size is 0.05 inch. Turn on the Display if you like and choose Dots or Lines.
4. Click on "Move" and adjust the Dimension lines to a reasonable size
5. Place the parts inside of the Dimension outline
 - a. Start with placing any connectors and Mounting Holes
 - b. Hint: Hit the Ratsnest button to redraw airwires after moving parts
 - c. Move and Rotate other parts to minimize crossing of airwires
 - d. Use the Mirror command to flip the LEDs to the Bottom side of the PCB
6. Click the "Route" command, and choose the proper settings in the toolbar.
 - a. Set the Layer to Top, Wire Bend type as "_/" or "/-", trace Width to 0.016.
 - b. Set the Via style to Round, Drill Size to 0.023, and Diameter to auto.
7. Click on an airwire close to a pad and begin making connections.
 - a. Hint: Traces should be routed away from the pads, especially pads that are closely spaced.
8. Create a Via to jump from Top to Bottom layer
 - a. Click the airwire connecting the LED to the Resistor.
 - b. While routing the airwire, left-click in open space and then change the layer in the toolbar.
 - c. The trace changes color, and a Via is created once the connection is made.
9. Create Traces and Vias as necessary until all connections are made and there are no more airwires.
 - a. Hint: Visit <http://www.4PCB.com/trace-width-calculator.html> to calculate trace width vs current
10. Select the "Text" command, type "USB Night Light" and hit OK.
 - a. In the toolbar, change the text Layer to tPlace, Size to .07, Font to Vector, and Ratio to 15%.
11. If the Names and/or Values are going to be on the silkscreen, all text should be formatted
 - a. Smash each part. HINT: See Advanced EAGLE on next page to use the command line to smash and format.
 - b. Right click on each Name and Value, Set Size >= 0.032 mil, Font = Vector, Ratio = 15%
 - c. Move the Names and Values around to not overlap any parts. It may help to make the grid size smaller.
 - d. HINT: When moving items, hold down CTRL while left-clicking on an item to snap the origin to the grid.
12. Create a GND pour on the bottom side of the board.
 - a. Click the Polygon command.
 - b. In the toolbar, set the Layer to Bottom, Wire Bend type to "_|", Width = 0.012, Isolate = 0.012, Thermals On
 - c. Draw a box directly on top of the Dimension lines.
 - d. Use the "Name" command to set the Name of the dashed line to GND. Click Ratsnest to draw the GND pour.
 - e. Repeat on the Top layer.



13. Click the DRC button to check for errors.
 - a. Click the "Load" and select the SparkFun.dru file downloaded from Sparkfun.com then hit "Check"
 - b. Review any errors. Fix or Approve them as necessary.
14. Generate Gerber Files. Click on CAM to open the CAM processor window.
 - a. Go to File -> Open -> Job and select SFE-gerb274x.cam downloaded from Sparkfun.com
 - b. Each tab represents a Gerber file to be generated.
 - c. Notice which Eagle layers are mapped to which Gerber file layers in the scrolling list on the right.
 - d. The Top Paste tab is for making a Solder Paste stencil. It can be deleted if not needed.
 - e. In the Top Silkscreen tab, select tValues in addition to tNames and tPlace if you want the values shown
 - f. Click Process Job to generate the files. Close the CAM processor window. Don't save changes to SFE-gerb274x.cam if prompted. Save your changes as your own custom .cam file if you want.
15. Look at the current project folder within the Eagle Control Panel window. Note the new files GBL, GBO, GBS, etc.
16. Install and Open Viewplot V1.5 to check the Gerber Files
 - a. Go to File -> Load Files. Navigate to the current Eagle project folder "USB Night Light"
 - b. Hold down CTRL to select the Gerber Files GBL, GBO, GBS, GTL, GTO, GTS, and TXT, and click Open
 - c. Click OK on the Read Files window.
 - d. Select "Leading Zero Suppression" and "2 4" as the Drill Format on the Load Drill File Window, click OK
 - e. In the lower left, select which layer to be shown on top. Zoom in and inspect each layer.
 - f. If there are any issues, go back to EAGLE and fix the issue. Delete the original Gerbers, generate new ones and re-inspect.
17. Find a PCB manufacturer and place an order for their PCB fabrication service that matches your board
 - a. iTeed Studios <https://www.itead.cc/open-pcb/pcb-prototyping.html>
 - i. Extremely low cost for 10 boards, fair quality, choice of PCB thickness, color and finish
 - ii. Near zero customer service - if your files are bad, you'll never know until you get your boards
 - b. OSH Park <https://oshpark.com/pricing>
 - i. Reasonable cost, high quality, all boards are 0.063" thick, purple soldermask, and ENIG finish
 - ii. Reasonable customer service – if your files are bad, they're likely to contact you and let you fix it
 - c. Place the Gerber files in a .zip file and send to the PCB manufacturer per their ordering instructions.
 - d. If the manufacturer has their own .CAM or .DRU file for EAGLE, use their file to check your design and generate your gerbers instead of the Sparkfun files.

ADVANCED EAGLE

- EagleCAD has a command line just below the toolbar which can help eliminate tedious tasks.
 - To smash all parts at once, type in the following commands:
 - `display none tOrigins bOrigins;`
 - `group all;`
 - `smash (>0 0);`
 - `display last;`
 - To format all Names and Values at once, type in the following commands:
 - `display none tNames bNames tValues bValues;`
 - `group all;`
 - `change size 0.032 (>0 0);`
 - `change font vector (>0 0);`
 - `change ratio 15 (>0 0);`
 - `display last;`
- EagleCAD has User Language Programs (ULP) that do all kinds of fancy things.
 - In the Schematic window, go to File -> Run, BOM.ulp
 - In the Board window, go to File -> Run, statistic-brd.ulp
 - In the Board window, go to File -> Run, spiral-coil.ulp
 - Panelize.ulp makes it possible to panelize a board design, find a tutorial online for details.