

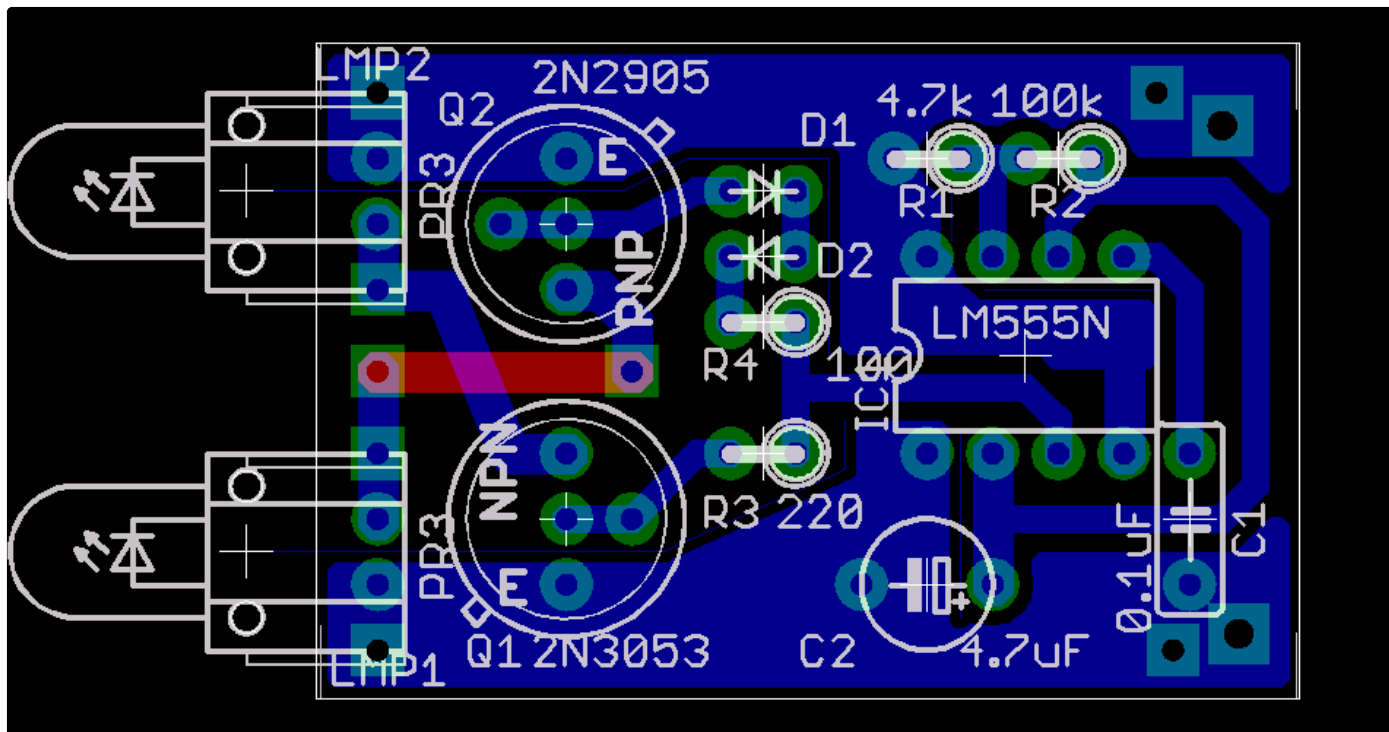
Turn Your EAGLE Schematic Into a PCB

By [westfw \(/member/westfw/\)](#) in [Circuits \(/circuits/\)](#) > [Tools \(/circuits/tools/projects/\)](#) 849,910 608 149 Featured



Download

Favorite



[\(/member/westfw/\)](#)
 By **westfw**
[\(/member/westfw/\)](#)

Follow

More by the author:



About: Middle aged geek username also works at yahoo.com, mac.com, comcast.net, wharton-10.arpa More About westfw » [\(/member/westfw/\)](#)

In a previous Instructable, I provided an intro to schematic entry using CadSoft's EAGLE editor. In this instructable, we'll make a printed circuit board from that schematic

I guess I should say that we'll make a PCB **DESIGN**; making the physical board is a different task, and there are a lot of tutorials on the net (and even some instructables) on making the board after you have the design.

Cadsoft EAGLE generic information:

Cadsoft EAGLE is available from <http://www.cadsoftusa.com/> (<http://www.cadsoftusa.com/>).

Cadsoft is a German company that is a veritable mecca of software distribution enlightenment. In addition to the reasonably-priced professional PCB design packages (\$1200), they have freeware, lite, non-profit, and other intermediate licenses. Their software runs under windows, linux, and MacOSX. It's slightly quirky, with a steep (but not too high) learning curve on the front end, but from most reports it is not any more so than other professional CAD packages. They have online support forums that are active from both the company and other users, the package is under current development and gets better with each release. A number of PCB fabricators will accept their CAD files directly. It's good stuff. Use it. Propagate it. Buy it when you "go pro."

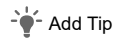
See also:

[Schematic Entry](https://www.instructables.com/id/Draw-Electronic-Schematics-with-CadSoft-EAGLE/) (<https://www.instructables.com/id/Draw-Electronic-Schematics-with-CadSoft-EAGLE/>).

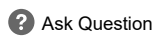
[Creating Library parts](https://www.instructables.com/id/ERHOQ180Y3EP286NOY/) (<https://www.instructables.com/id/ERHOQ180Y3EP286NOY/>).

[Design rule modification](https://www.instructables.com/id/EZVIGHUBGCEP287BJB/) (<https://www.instructables.com/id/EZVIGHUBGCEP287BJB/>).

[Send CAD Files to manufacturers](https://www.instructables.com/id/Professional-PCBs-almost-cheaper-than-making-them-/) (<https://www.instructables.com/id/Professional-PCBs-almost-cheaper-than-making-them-/>).



Add Tip



Ask Question



Comment

Download

Step 1: Starting From the Schematic...

So this is the schematic we have from the [Schematic Instructable](https://www.instructables.com/id/EXU9BO166NEOH08XFU/) (<https://www.instructables.com/id/EXU9BO166NEOH08XFU/>). Up in the file menu, there's a "Switch to board" selection. If we do that from a bare schematic, it will offer to create the board from the schematic for us (say "yes"), and then leave us sitting in the Board Editor.

 Add Tip

 Ask Question

 Comment

Download

Step 2: Menu Commands Used

<https://www.youtube.com/watch?v=9G1H9J5G0Z0>

The Board Editor looks a lot like the schematic editor, with some different commands. Here's a summary of the iconic commands that I use in this instructable, and some brief summaries:

INFO Shows information about an object (component, signal, trace, etc.)

MOVE Allows components to be moved (same as schematic.)

GROUP Groups a collection of objects into a "group" that can be manipulated simultaneously.

DELETE Delete an object. Items created in schematics need to be deleted there.

SMASH Separate the text labels of a part from the part itself, so they can be moved independently.

BREAK Add a corner to a line (or trace.)

ROUTE turn an airwire into a trace

LINE draw lines (usually in non-copper layers. ROUTE is for drawing copper.)

VIA create a hole and pad associated with some signal. (actually, we'll use a text command.)

HOLE a hole that isn't associated with a signal, ie for mounting.

RATSNEST recomputes airwires and polygons, eg after components have been moved.

CHANGE changes an object's properties.

RIPUP changes a routed trace back to an airwire. Sorta equivalent to "delete" for traces.

TEXT add text

POLYGON create a polygon (actually, we'll use a text command.)

AUTOROUTE invoke the autorouter.

DRC invoke the Design Rule Check and parameter setting.

I'll describe the remaining icons toward the end, and assign them "useful" or "useless."

Step 3: The Untouched PCB Design

<https://content.bentley.com/Products/Software/PCB/EAGLE/8.6.0/Getting%20Started/Getting%20Started%20-%20English.pdf>

This is what the newly created board design will look like. All your components will be in a clump over to the left of the origin, and there will be a frame that marks the allowed size of a board when using the freeware or "Lite" versions of EAGLE (80x100mm). All the component pads will have to be inside that outline when you move them around, although you can cheat a bit and have traces or board outlines that exceed the board size limit. This has the annoying side-effect that if you pick up a component from it's original localtion, you can't put it back down outside the outline (however, you can use ESC to abort the move, and the component will revert to its original location.)

Ok, a few defintions are in order

All the signals you created in the schematic are currently **AIR WIRES**; thin yellow lines that are drawn in the shortest possible way, crossing each other as needed. They stay connected to component pins even when you move the component around. The **RATSNEST** command recomputes and redraws these after you move things around (and, say, make two connected



 Add Tip



 Ask Question



 Comment

Download

Step 4: About Board "layers"

[/http://content.inetlib.org/bitstream/1854/4616/1/4272XCEFWB07E00EAE000429AE5E4A](http://content.inetlib.org/bitstream/1854/4616/1/4272XCEFWB07E00EAE000429AE5E4A)

The Eagle Board editor has MANY more layers than the schematic editor. A confusing multitude of layers. Most of the drawing commands have a layer-selection pulldown menu that you can use for specifying which layer you want to draw on (exceptions include objects like vias that span multiple layers.)

Here are some of the more important layers:



Add Tip



Ask Question



Comment

Download

Step 5: Move the Components Into the Legal Area

The first thing we want to do is move at least some components into the legal board area where we can work with them. If you have a particularly large board with many components, you might want to do this a section at a time. For this sample board, we have plenty of room and we can move them all at once, using the group-move feature.

Select the GROUP icon, then click and drag to make a rectangle that goes all the way around the components. Then select the MOVE icon and RIGHT click (right clicking selects the group instead of a single component) and drag the set into the board outline. Use the ZOOM button to tighten the view.



Add Tip



Ask Question



Comment

Download

Step 6: Shrink the Board Outline a Bit


<http://contributor.wikihow.com/How-to-Draw-a-Board-Game-Outline>

The full legal side of the board is bigger than we need. Shrink the outline by using the MOVE tool. Click on the center of the top horizontal line (which selects the whole line instead of an endpoint) and move it down, Then click on the center of the rightmost vertical line and move it leftward.

Clicking near the center of a line moves the whole line. Clicking near a vertice moves only the point.

It doesn't have to be perfect at this point; we're mostly looking for a better view for the next steps. (Oh yeah - click the zoom button to re-zoom the window on the smaller outline.)

 Add Tip

 Ask Question

 Comment

Download

Step 7: Start Placing the Components

In this case, I put the power output transistors near lamps that they're associated with, and I looked on the web for a 555 layout that would work well (for the longest time, I tried to do boards with the timing cap placed near the timing resistors, and I always needed a jumper. Sigh.) ("Let no one else's work evade your eyes.")

Step 8: Check Signals to See How They'll Route

One way to get hints on part placement is to look at some significant signals to see whether they have nice straight paths, or whether they zigzag all over the board. First use the RATSNEST icon/command to have EAGLE recompute the airwires. The way things are now, I have nice straight connections from the transistors to the lamps, but if I type "show gnd" in the command line, I see that this is at the expense of making the ground signal zigzag. So I swap the transistors because GND is more important to have straight. (IMHO, YMMV, etc.) (This ends up putting the transistors near the supplies that they switch, rather than near the lamps that they switch to, so that still makes sense from a circuit point of view too.)

After the rest of the components are placed in ok-looking relative locations, I can squeeze them together again (manually, moving them one at a time; no magic command for this!) and shrink the board outline some more.

[Add Tip](#)[Ask Question](#)[Comment](#)[Download](#)

Step 9: Load Design Rules

<https://content.instructables.com/88F0B9C4567A4320640045650B360E376A9D222405E9098510.png?>

Since we're hobbyists, we want to make our board with wide traces and big spaces (see

<https://www.instructables.com/id/EZVIGHUBGCEP287BJB/>

(<https://www.instructables.com/id/EZVIGHUBGCEP287BJB/>).

So we'll load up that set of hobbyist design rules before we start laying out tracks.

Click the Design Rule Check icon and use the LOAD button to load [hobby.dru](#)

(<https://static2.instructables.com/ID/FJ0/43CW/RTMEP27Y3QS/8B9C12B62C821029BC6B001143>

[E7E506.dru](#)) from my other instructable. Or you can modify values manually and individually, of course. Or leave them as is...



Add Tip



Ask Question



Comment

Download

Step 10: Fix Incorrect Package

You can see how the design rule change has already modified the board. Pads are bigger, and they're all round.

You'll also notice that one of the resistors is set as a non-vertical package, unlike the rest. This was probably an error in the schematic entry, and it didn't matter when all we had was the schematic. Now that we are making the board, we want to change the package as appropriate.

When you select the change->package tool and click on the part to change, you'll be shown a list of all the legal packages for that part (these should be the same ones that showed up in the schematic "add" dialog)

There are other ways to enter a "change" command in the text command entry area that you'll want to look into if you need to change a lot of devices to a particular package, so you can skip going through the list for each one. Something like "change package 'R-US/0207/2V'; and then just click on each component.

[Add Tip](#)[? Ask Question](#)[Comment](#)[Download](#)

Step 11: Try the Autorouter

<https://community.cadsoft.com/forums/thread/10464.aspx>

Now we'll see if the autorouter can do some of the work for us. The EAGLE autorouter isn't the best in the world, but even when it does a "bad" job, it will give us some general hints on how things need to look, or where the trouble spots are.

Click the AUTOROUTE icon, and a dialog box will pop up. The default parameters will produce a double sided board, and we want to at least TRY to make a single-sided board, so the first thing to do is set the preferred direction for the TOP layer to NA (Not Applicable.)

The other thing you may need to change is the routing grid. This defaults to the same default grid as the board layout editor in general: 0.05 inches (1.27mm, since I have my editor set up in metric.) Since this particular board has big parts, and we haven't moved any off the default grid, we're ok with that value. If you have SMT components or have moved things around on a finer grid, you might have pads that are not on the routing grid, which the autorouter doesn't like much ("unreachable pad", etc) You can make the grid very small, but it will take longer. IMO, it's better to start with a coarse grid and halve it each time it looks like routes fail because the grid is too large.

Also note that the autorouter obeys the board dimension lines, so if you haven't moved them close to your components, you might have traces travel all over the board. Or if you've moved the outline too close to the pads, you may have prevented traces from going places they need to go.



Add Tip



Ask Question



Comment

Download

Step 12: Route Remaining Tracks Manually

<https://content.muhimbi.com/Assets/65631614-0251-4020-BE5E-5D5D5D5D5D5D>

[3 More Images](#)

The autorouter did a pretty nice job here. There's only one trace left.

There are a couple of ways we could route this signal manually, including some snakey routes between transistor pins that the autorouter didn't use because of the design rules we specified. This is a relatively high current trace, and I decided that I won't manually violate the design rules either. Instead, I'll use a jumper wire on the component side, which I can model in EAGLE as a top-side trace.

Select the ROUTE tool and click on an endpoint of an unrouted (yellow) airwire, and you can position a trace pretty much anywhere you want, selecting width, layer, and type of bend from the menu bar as you go along. This is shown in the succession of pictures in this step.



Add Tip



Ask Question



Comment

[Download](#)

Step 13: Add Power Plane Polygons

<https://commitment.industrydocuments.ucsf.edu/docs/hd0918> | <https://commitment.industrydocuments.ucsf.edu/docs/hd0918>

In this case, the V+ signal is named n\$1, so we type "**poly n\$1**"

Step 15: Neaten Up: Smash Package Text

If we want the names of components to be legible on the top of the board (transferred via toner transfer), or just to look good on printouts, they names and values probably have to be moved from their default locations. In order to move the text separately from the device itself, we use the "SMASH" command. (Why is it called "smash" ? I dunno!)

Select the SMASH icon from the menu, then click each component whose text you want to move. If this is ALL of the components, there's a ULP that will smash everything (but ULPs are a subject for possible future instructables. Or the EAGLE manuals.)



Add Tip

Ask Question

Comment

Download

Step 16: Neaten Up; Move Traces

We can move some of the traces so they look neater, offer better clearance, etc.

Also, we shrink the board to its final size by squishing the components together some more.



?



Download

Step 17: Fixing an OOPS!

~~https://www.kicad.org/forum/viewtopic.php?id=14444~~

Remember back in the schematic I mentioned that there were a couple of things that had been left out? You should be noticing them about now...

POWER connections; there's no way to connect a battery or power supply to this circuit board. Oh sure, you can just tack some wires onto the supply polygons, but how elegant is *that*!

We could go back to the schematic and add some actual power connectors or battery holders, but those are a bit rigid for a circuit that's probably going to be connected to a battery pack with some wires anyway. Instead, let's add some Vias to act as connection points for the power wires.

When adding Vias like this, it is convenient to use the text command entry area so we can name the signal at the same time we add the via. Type "via 'gnd'" (yes, you need the quotes here, unlike for polygons.) You can adjust the drill size and via shape, and plunk the via down in the appropriate supply polygon. I like to use two vias as a sort of strain relief (one is made larger so you can feed wire + insulation through it, the other is sized for just the wire.) A click on the RATSNEST icon will make sure the vias are connected to the polygon. Then do the same for the V+ signal (named N\$1, you recall.)

Step 18: Neaten Up: Allow for Alternate Packages and Options

<https://content.instructables.org/IDG1G15F70F09156/4000070948660030/>

<https://content.kicad.org/images/486662162462954490465843064191573583301555701641102.png?>

We can drop some extra holes for mounting different packages. The transistors used in the published schematic that we entered apparently come in a sort of metal can package that has dropped in popularity. If we arrange for three in-line mounting holes, we can substitute a whole lot of different transistors whose package leads come that way (TO92 or TO220, to mention two popular modern packages.)

Use the info command to figure out the signal names, and then "via 'n\$X'" on the command line to create the via, followed by a manual route to the via if needed. In this case, one of the vias placed collides with a signal trace hidden by the GND polygon, so we have to remove that trace with the "ripup" command (the polygon will still connect to the pad.)

While we're at it, I'll add some text to the silkscreen to show where the emitter lead of the transistors should go. Use the "text" icon button, and change the layer to tPlace.

[Add Tip](#)[Ask Question](#)[Comment](#)[Download](#)

Step 19: Do Design Rule Check

/https://content-hub.s3.amazonaws.com/16120164004550133615176000022405008510.png?

We want to run a design rule check to make sure that none of the manual editing we've done violates the rules...



Add Tip



Ask Question



Comment

Download

Step 20: Output Using Exported Images


<https://contentassets.figma.com/61616/57X8M20P20N0E01B2DAA57X4M0W0D706502165DA.png?>

Save your work often. You've been doing that, right?

Now we're essentially done, and we should figure out how we're going to output our board for admiration on web pages, review by peers, transfer to physical PCB material, and so on.

One way to output the board is to "export" an image.

 Add Tip

 Ask Question

 Comment

Download

Step 21: Other Useful Menu Icons

amalik101 (/member/amalik101/) made it!

Electrical Technology (/member/Electrical+Technology/) made it!

Did you make this project? Share it with us!

I Made It!

Recommendations

(/DIY-REFLOW-OVEN/)

DIY REFLOW OVEN (/DIY-REFLOW-OVEN/) by Vulcaman (/member/Vulcaman/) in Electronics (/circuits/electronics/projects/)

♥ 172

👁 8.9K


(/3D-Printed-Mini-CNC-Machine/)


3D Printed Mini CNC Machine (/3D-Printed-Mini-CNC-Machine/) by SmartTronix (/member/SmartTronix/) in Arduino

♥ 112


👁 12K

(/contest/trashtreasure2021/) (/contest/microcontroller2021/)





Add Tip



Ask Question



Post Comment

We have a **be nice** policy.
Please be positive and constructive.

Add Images

Post

149 Comments

(/member/Frajol%25C3%25A3o/) Frajolão (/member/Frajol%25C3%25A3o/) 4 years ago

Reply

▲ Upvote

Thanks for taking the time and putting it together. Very, very helpful.

(/member/aladdin_2005/) aladdin_2005 (/member/aladdin_2005/) 4 years ago

Reply

▲ Upvote

Amazing,very helpful tutorial
thank you

(/member/bask185/) bask185 (/member/bask185/) 6 years ago on Introduction

Reply

▲ Upvote

eagle is so bad, there are no worse programs. I cannot delete traces I just placed. and the resolution is horrible. despite there is room for 4 or 5 lines, I can only draw 2 lines. I cannot draw lines between 2 pads cuz of this crappy resolution. and when I want to draw a line of a pad, it sometimes select totally different pads. There are so many yellow to be routed lines I can barely see what im doing, and when i disable them I cannot see nothing. I cannot believe how bad this program is made

3 replies ▼

(/member/Tanmay+Deuskar/) Tanmay Deuskar (/member/Tanmay+Deuskar/) 4 years ago

Reply

▲ Upvote

Hello,
This instructable is really helpful. Though I have a question - If you add polygons to the PCB design it makes it multilayered right? Then how to etch the design onto a copper board?

1 reply ▼

(/member/FeatherG/) FeatherG (/member/FeatherG/) 5 years ago

Reply

▲ Upvote

A great tutorial. If are interested in electronic circuit design, I'd like to share you with another useful tutorials <https://easyeda.com/Doc/Tutorial/> (<https://easyeda.com/Doc/Tutorial/>) it will be helpful to your study.

(/member/abraxas2/) abraxas2 (/member/abraxas2/) 10 years ago on Step 13

Reply

▲ Upvote

I can't thank you enough for this excellent tutorial. I thought I'd never get the hang of it and it's not a very intuitive program. Do consider turning these tuts into a book. Now hopefully one last question : after drawing the ground plane with the ratsnest command I got a nice big plane but also a bunch of

tiny little islands that serve no purpose but to make the board look dumb. I tried erasing these islands with the rip up but it just ripped up the closest vital wire instead. Is there away to zap these tiny little island ground planes ? Thanks MUCHO !!

3 replies ▾

(/member/jbste/) jbste (/member/jbste/) 5 years ago

Reply

▲ Upvote

great tutorial but how do you deal with tracks present in the schematic that don't appear on the pcb layout?

(/member/Electrical+Technology/) Electrical Technology (/member/Electrical+Technology/)

6 years ago on Introduction

Reply

▲ Upvote

(/member/J-Rob+McCoy/) J-Rob McCoy (/member/J-Rob+McCoy/) 6 years ago on Step 22

Reply

▲ Upvote

You have just taken me from dreading making the PCB from the schematic to having one done in a flash. Excellent instructable on taking the somewhat intimidating Eagle CAD and teaching the basic functions on the PCB aspect of it. Thank you.

(/member/gaurav157/) gaurav157 (/member/gaurav157/) 6 years ago on Introduction

Reply

▲ Upvote

hey plz help me
when i print pcb from printer the size of ic lage will be different in pdf and on paper
.How i set the sizes.

(/member/gaurav157/) gaurav157 (/member/gaurav157/) 6 years ago on Introduction

Reply

▲ Upvote

hey plz help me
when i print pcb from printer the size of ic lage will be different in pdf and on paper
.How i set the sizes.

(/member/gaurav157/) gaurav157 (/member/gaurav157/) 6 years ago on Introduction

Reply

▲ Upvote

hey plz help me
when i print pcb from printer the size of ic lage will be different in pdf and on paper
.How i set the sizes.

(/member/mjeon/) mjeon (/member/mjeon/) 7 years ago on Step 22

Reply

▲ Upvote

The most helpful tips for me ever saw!!
Thanks for writer!!!

[More Comments](#)[Post Comment](#)

Categories



Circuits
(/circuits/)



Living
(/living/)



Workshop
(/workshop/)



Outside
(/outside/)



Craft
(/craft/)



Teachers
(/teachers/)



Cooking
(/cooking/)

Find Us

About Us

Who We Are
(/about/)

Why Publish?
(/create/)

Jobs (/jobs/)

Resources

Sitemap (/sitemap/)

Help (/how-to-write-a-great-instructable/)

Contact (/contact/)

[\(https://www.instagram.com/instructables/\)](https://www.instagram.com/instructables/) [\(https://www.pinterest.com/instructables/\)](https://www.pinterest.com/instructables/) [\(https://www.facebook.com/instructables/\)](https://www.facebook.com/instructables/) [\(https://www.twitter.com/instructables/\)](https://www.twitter.com/instructables/)

© 2021 Autodesk, Inc. [Terms of Service \(http://usa.autodesk.com/adsk/servlet/item?siteID=123112&id=21959721\)](http://usa.autodesk.com/adsk/servlet/item?siteID=123112&id=21959721)
[Privacy Statement \(http://usa.autodesk.com/adsk/servlet/item?siteID=123112&id=21292079\)](http://usa.autodesk.com/adsk/servlet/item?siteID=123112&id=21292079)
[Privacy settings](#)
[Legal Notices & Trademarks \(http://usa.autodesk.com/legal-notices-trademarks/\)](http://usa.autodesk.com/legal-notices-trademarks/)
[Autodesk Jobs & Careers \(http://www.autodesk.com\)](http://www.autodesk.com)