

Make hobbyist PCBs with professional CAD tools by modifying "Design Rules"

by **westfw** on May 4, 2006

Table of Contents

Make hobbyist PCBs with professional CAD tools by modifying "Design Rules"	1
Intro: Make hobbyist PCBs with professional CAD tools by modifying "Design Rules"	2
File Downloads	2
Step 1: Introduction, part 1 - my gripe	2
Step 2: Intro, part 2 - Cadsoft EAGLE	3
Step 3: Our sample circuit: Blink some LEDs.	3
Step 4: Placing the parts	4
Step 5: Autorouted using the defaults, and what's wrong with it... ..	5
Step 6: Let's fix the DESIGN RULES	5
Step 7: Modifying the CLEARANCE rules	6
Step 8: Modifying the SIZES rules	6
Step 9: Changing pad sizes with the RESTRING rules	7
Step 10: Optional: adjust pad SHAPES	7
Step 11: Save your chosen rules, and autoroute again	7
Step 12: But why stop there?	8
Step 13: Finalizing the PCB design	9
Step 14: But did it WORK?	9
Step 15: Summary	11
Related Instructables	11
Comments	12



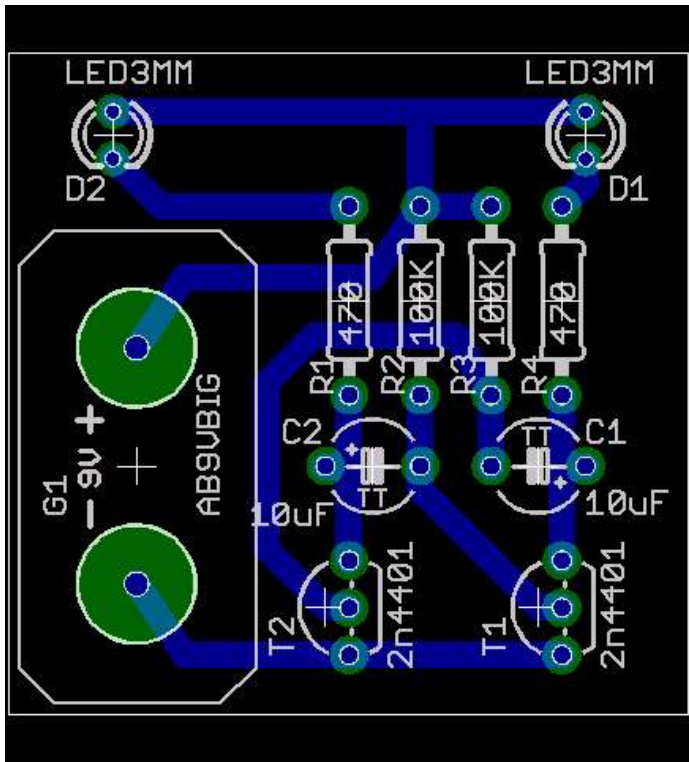
Author: westfw

Middle aged geek

username also works at yahoo.com, mac.com, comcast.net, wharton-10.arpa

Intro: Make hobbyist PCBs with professional CAD tools by modifying "Design Rules"

It's nice that there are some professional circuit board tools available to the hobbyists. Here are some tips for using them to design boards that don't need a professional fabricator to actually MAKE them...



File Downloads



hobby.dru (1 KB)

[NOTE: When saving, if you see .tmp as the file ext, rename it to 'hobby.dru']



blinkie-brd-sch.zip (23 KB)

[NOTE: When saving, if you see .tmp as the file ext, rename it to 'blinkie-brd-sch.zip']

Step 1: Introduction, part 1 - my gripe

There are numerous tutorials on the net about making your own printed circuit boards (PCBs.) Toner transfer, photo-sensitized PCBs, sharpies; all sorts of information...

Likewise, there are a number of Computer Aided Design packages (CAD) designed to help create PCB designs, possibly with accompanying schematics. Some of these have low-cost versions aimed at students and hobbyists.

But I see on various web pages PCBs created with these CAD packages, by hobbyists, that are not "friendly" to actually being fabricated by hobbyists using the methods described on the PCB pages. A lovely published PCB is not nearly so useful if it requires the \$50+ typical minimum price from a professional board maker.

I don't have any doubt that with the right equipment, and supplies, and some practice, you can get good enough at home PCB fabrication techniques (take your pick) to produce high quality board of significant complexity, with fine traces, small holes, and so on. But a lot of PCBs don't really need that complexity, and it would be nice if they were DESIGNED in such a way that you didn't NEED a lot of experience in PCB making to get a working PCB.

This document contains some hints on configuring a CAD package to create boards that are easier to manufacture in a hobbyist environment. It's based around Cadsoft's Eagle CAD package, but the principles are relatively general and should be applicable to other CAD packages as well.

Step 2: Intro, part 2 - Cadsoft EAGLE

Cadsoft EAGLE: <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."

This document is not a tutorial on how to use EAGLE, although it'll probably be somewhat useful in that role. It's more about how to configure and customize an Eagle installation to better suit the hobbyist.

See also:

Schematic Entry

Create PCB from schematic

Creating Library parts

Design rule modification

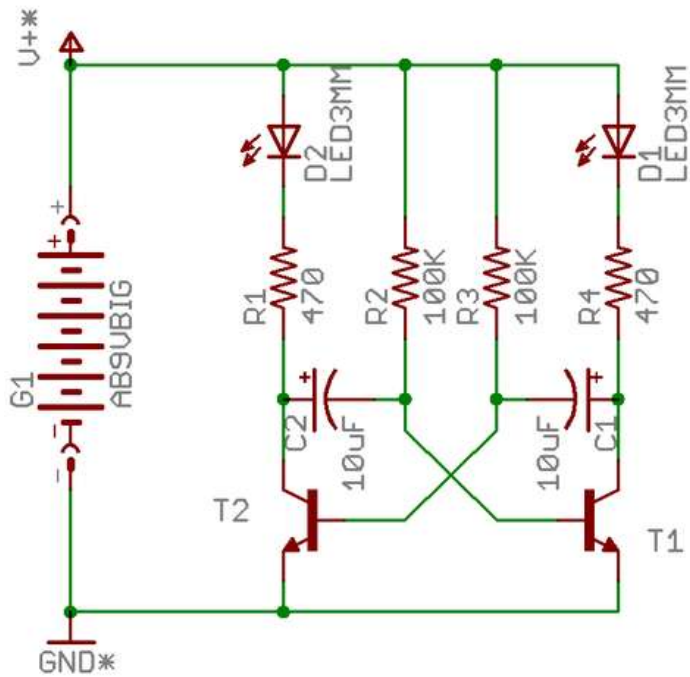
Send CAD Files to manufacturers



Step 3: Our sample circuit: Blink some LEDs.

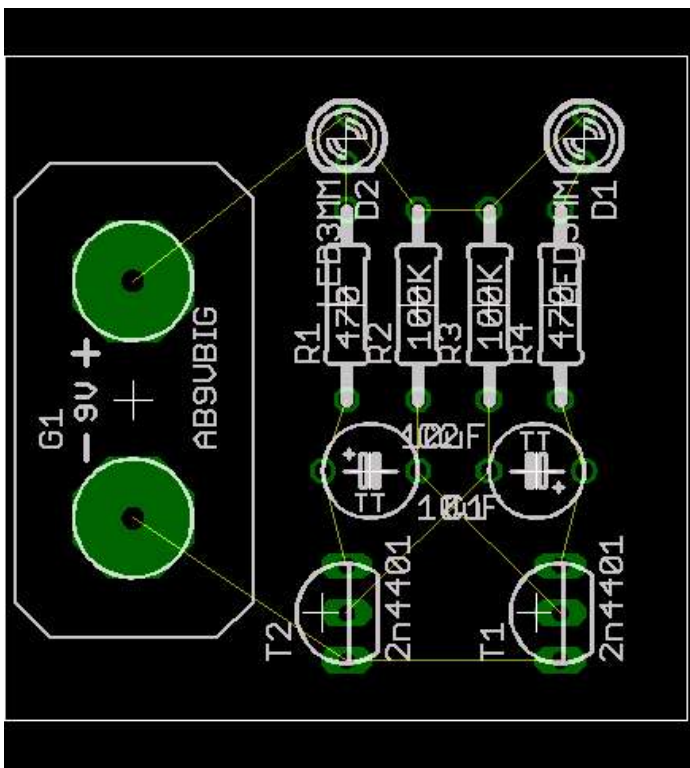
As an example, I'm going to use a simple and rather standard two-transistor, two-led "blinky" circuit. It looks like this.

(If you decide to actually build this, the transistors can be any general purpose silicon NPN types like 2n4401, 2n2222, 2n3904.) The ON time for each LED is about R*C (one second for the values here.) The battery can be 3V up to ... whatever, although you may need to adjust the current limiting resistors for higher voltages.) The caps should have a voltage rating a bit higher than the power source you intend to use. For a 9V battery, I used 16V caps. Resistors are 1/4 watt.)



Step 4: Placing the parts

It looks pretty simple, so we'll throw the components onto a board just about the way they look on the schematic:



Step 5: Autorouted using the defaults, and what's wrong with it...

Then we fiddle with the autorouter a bit, being careful to set the top layer direction to "N.A." to get a one-sided board (but using all the other default settings.) We get something that looks like this.

That actually looks pretty nice. So what's the problem? The problem is that if you try to make that board in your kitchen, you'll probably be in for a lot of frustration. There are two main issues:

1) Trace width. The default trace width is 10mil (a mil is 1/1000 of an inch) or about 0.2mm That's fine for most professional PCB fabricators; most can routinely and reliably make boards down to 6mils. But it's VERY fine to accomplish using something like toner transfer (recall that a fine-lead mechanical pencil is 0.5mm - nearly 3 times bigger!)

There's a similar problem with the amount of pad left around the holes; while it's fine for a fancy CNC-drilling machine, if you try to drill the holes with typical home equipment you'll probably end up removing the whole pad.

2) Clearance. This is the space left between tracks (or between tracks and pads.) Like the trace width, it defaults to a small number: 8 mils. that's just not a realistic value for a hobbyist...

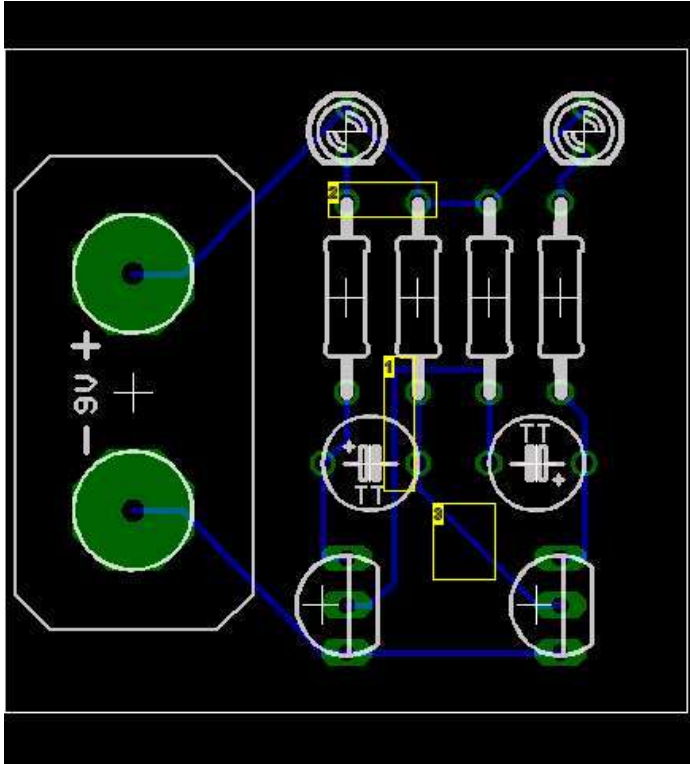


Image Notes

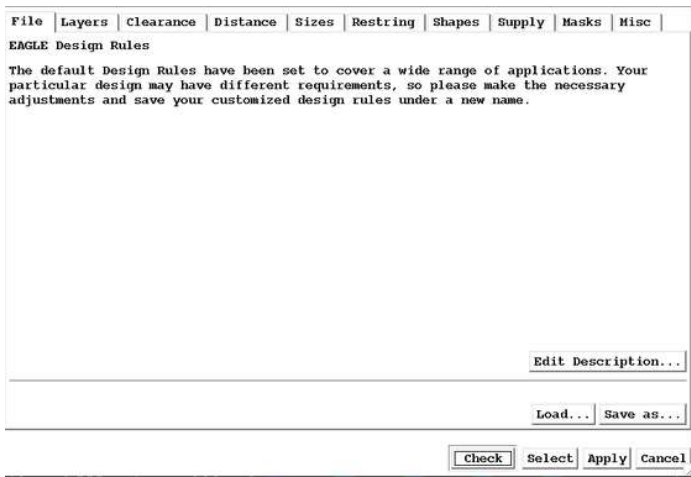
1. Tracks go too close to pads
2. Pads are too small
3. Tracks are too narrow

Step 6: Let's fix the DESIGN RULES

Collectively, these parameters (and many others) are called the "Design rules" for the board. Fortunately, they are designed to be changeable to meet the requirements for different PCB fabricators, and they can be changed to better match the needs of the hobbyist as well. You can get to the design rule check and options with the DRC command or button. It looks like this.

The DRC panel is usually used to do a design rule CHECK. After a board is laid out (usually with significant hand routing) you'd click the "CHECK" button and Eagle would go and make sure that what you've done conforms to the design rules you've specified. However, the autorouter also pays attention to the design rules you've set; it wouldn't be a very useful feature if the autorouter created boards that were "illegal."

As you can see, there are LOTS of parameters you can change. We're only interested in a few of them. (the individual parameters usually are illustrated with a nice picture showing the object you're actually changing. A nice help feature...)

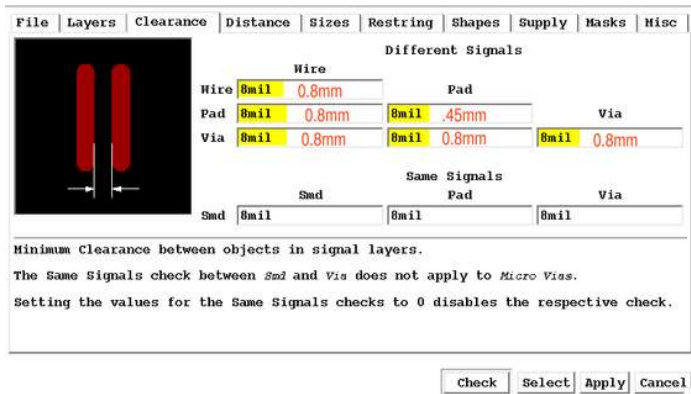


Step 7: Modifying the CLEARANCE rules

In the CLEARANCE panel, we can control the desired clearance between several different sorts of objects. The default clearance is 8mils for everything...

At some point you need to decide what you want the values to be. This is just an example, so I get to pick. I like 0.8mm, which is very close to 1/32 inch. So we can set a bunch of the clearance values to 0.8mm:

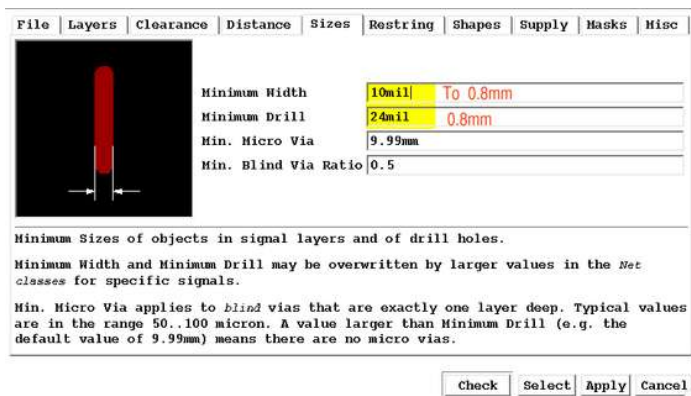
The "same signal" clearances can stay at small numbers; we don't care a lot about that. The PAD to PAD clearance has to be a significantly smaller 0.5mm; more about that later...



Step 8: Modifying the SIZES rules

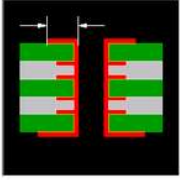
The SIZES panel has the next set of parameters to change.

We don't have to worry about micro or blind vias, cause they're not appropriate to hobbyists in the first place, and not supported by the freeware Eagle in the second place. We can set the minimum width and minimum drill to (again) 0.8mm (incidentally, .8mm is about a number 68 drill.)



Step 9: Changing pad sizes with the RESTRING rules

The RESTRING panel controls the size of pads. It'd be nice if we could make the ring be 0.8mm thick too, but by the time you have .8mm of hole and .8mm of ring on each side, you have 2.4mm diameter pads. Since many parts have the pads on 0.1inch (2.54mm) centers, that doesn't leave enough space BETWEEN pads. So I'll use 0.6mm here, and I'll still have to use the smaller clearance values between pads that I mentioned above. I'll still have problems with PADS that are much bigger than .8mm (it takes about a 1mm hole to hold a .025inch square post as found on many connectors.) You can trade off pad-pad clearance against pad diameters forced by the restring settings, depending on where you have more problems with whatever PCB technique you're using. One advantage of a large pad is it makes you less sensitive to the drill you actually use; even if the library is set up for a .6mm drill and you use a .8mm drill, you should have enough copper left so that you won't have a big problem. You don't need to set inner layer or micro-via values:

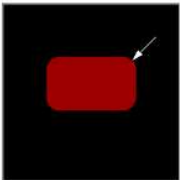
File	Layers	Clearance	Distance	Sizes	Restring	Shapes	Supply	Masks	Misc
				Min	%	Max	Diameter		
	Pads	Top	10mil	0.6mm	25	20mil	1mm		
		Inner	10mil		25	20mil			
		Bottom	10mil	0.6mm	25	20mil	1mm		
Vias	Outer	8mil	0.6mm	25	20mil	1mm			
	Inner	8mil		25	20mil				
Micro Vias	Outer	4mil		25	20mil				
	Inner	4mil		25	20mil				

Restrings for pads and vias are defined in percent of the drill diameter (limited by Min and Max). If the diameter of an actual pad or via would result in a larger restring, that value will be used in the outer layers.

If the Diameter option is checked the actual pad or via diameter will be taken into account in the inner layers, too.

Step 10: Optional: adjust pad SHAPES

In the SHAPES panel, I like to force the pad shape to ROUND, since I've already made the pads very large in the RESTRING panel. The oval pads get VERY large when you use big restring values... This is optional, though:

File	Layers	Clearance	Distance	Sizes	Restring	Shapes	Supply	Masks	Misc	
				Min	%	Max				
	Smds	Roundness	0mil		0	0mil				
	Pads	Top	As in library		ROUND					
		Bottom	As in library		ROUND					
First		Not special								
Elongation %				100		100				

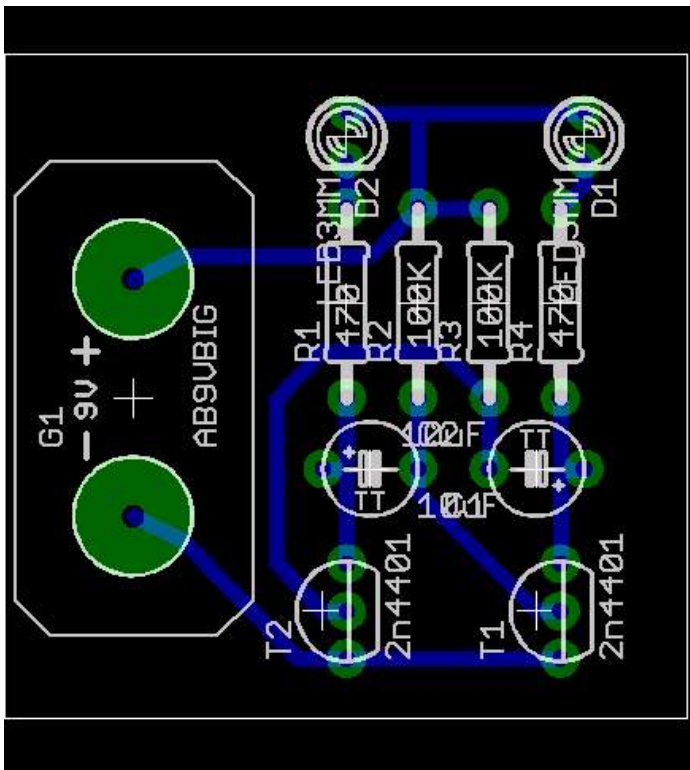
Shapes of pads and smds.

Step 11: Save your chosen rules, and autoroute again

Having changed all those parameters, we should APPLY them, and then we can go back to the FILE panel and save them somewhere appropriate:

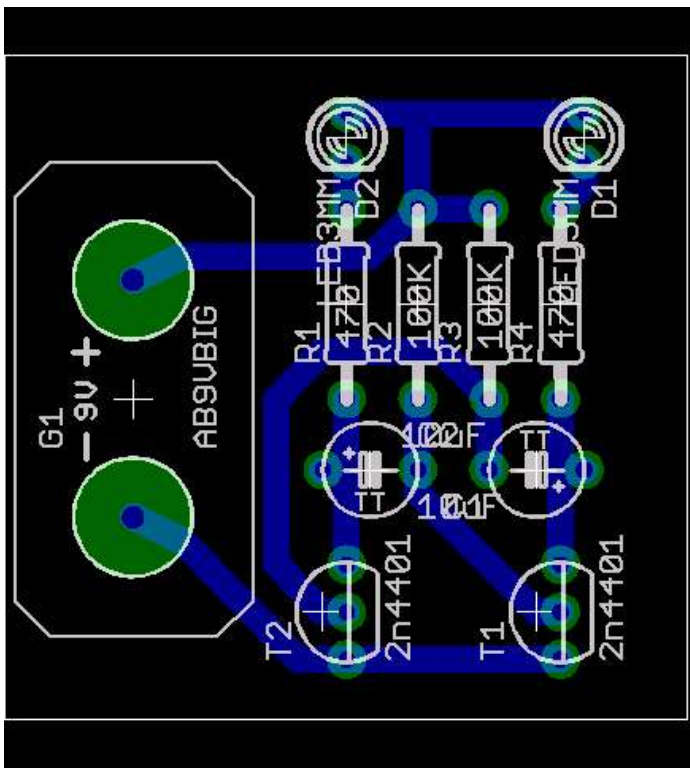
When creating future boards, you can use the FILE panel of the DRC window to read in the hobbyist-friendly parameters instead of having to retype them all. (Or just get the honny.dru file from the top page.) You can even suck them in you your init file.

Getting back to the circuit, if I run the autorouter NOW, I get a much more reasonable looking result...



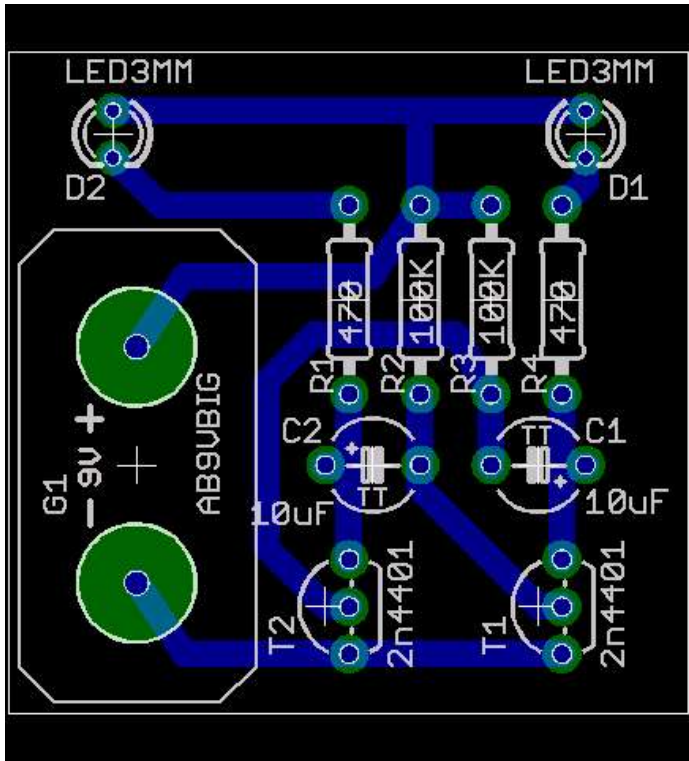
Step 12: But why stop there?

We could stop there, but we don't have to. The autorouter operates on a grid (defaults to 50mils), so what it's done is put tracks along the grid in places that don't violate the design rules. That probably means that there's significantly MORE room for even wider tracks or clearances. If we GROUP the entire board, we can "change width 1.0mm" or equiv, and use the DRC "check" option to see if we STILL pass our specs. Or we could have another DRC file with different parameters. In fact, this board can have it's trace width increased to 1.4mm without violating our clearance rules:



Step 13: Finalizing the PCB design

At this point, there are some traces that are reasonably close together, and it might make sense to manually move them apart a bit more, and clean up some of the stranger things that the autorouter has done. And I can decide that I want this to be one of those edge-of-stage warning lights that stands on its own by virtue of the 9V battery, which means I should reposition some of the components a bit. I can move around the silkscreen so that I can use toner transfer for that too. I end up with this:



Step 14: But did it WORK?

Let's see. I can be intentionally sloppy here, so as to better emulate someone without much experience, right? (Sure. That's a good excuse. I normally run off my boards on an LPKF PCB "plotter", so I genuinely suck at doing this the hard way.)

Scrap of board, magazine paper/toner transfer; looks not so wonderful at this point. Touch up with a sharpie.. etch, drill, clean... More toner transfer for the "silkscreen", add components and power it on...

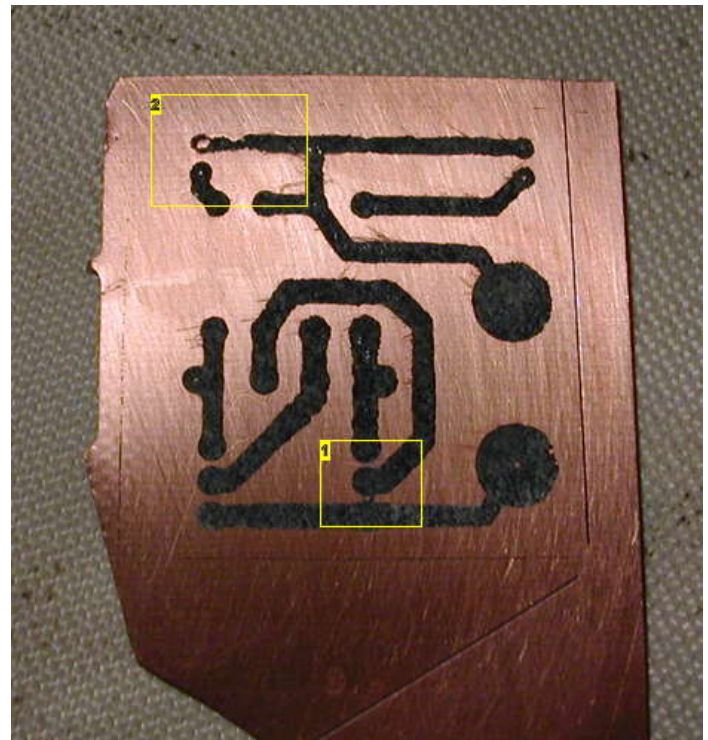


Image Notes

1. And over here there's a bit too much...

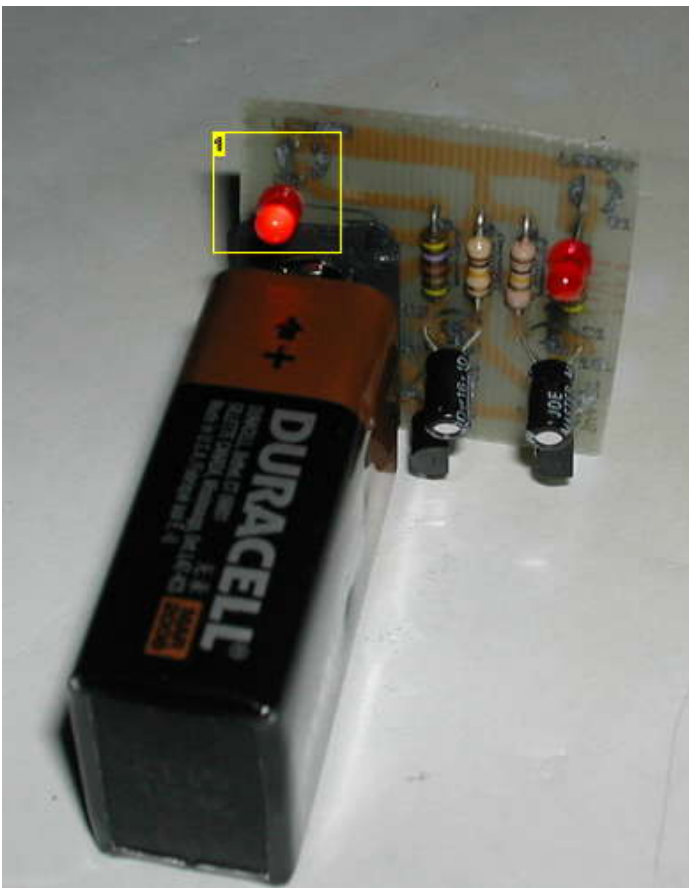
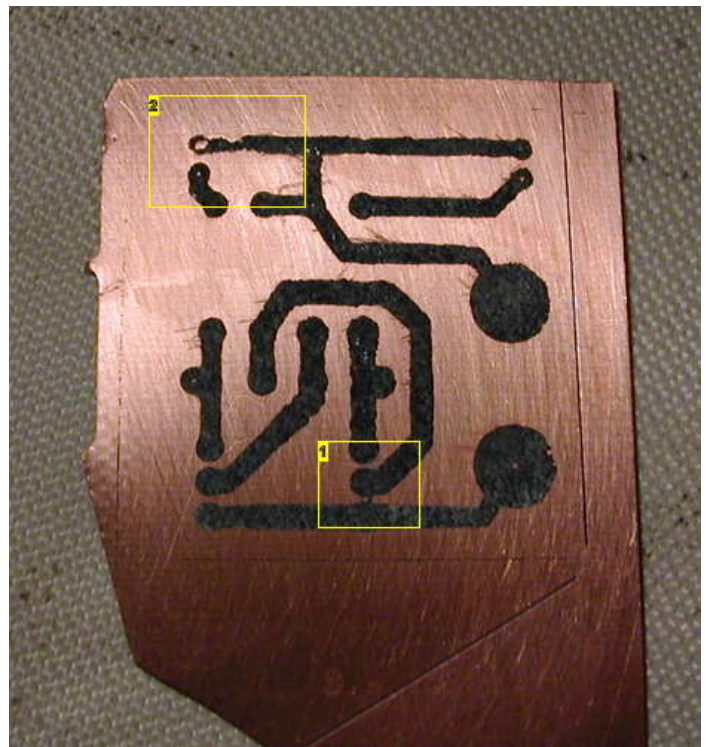


Image Notes

1. Blinking!



2. yuck. I could use some practice with toner transfer. Some didn't, here.

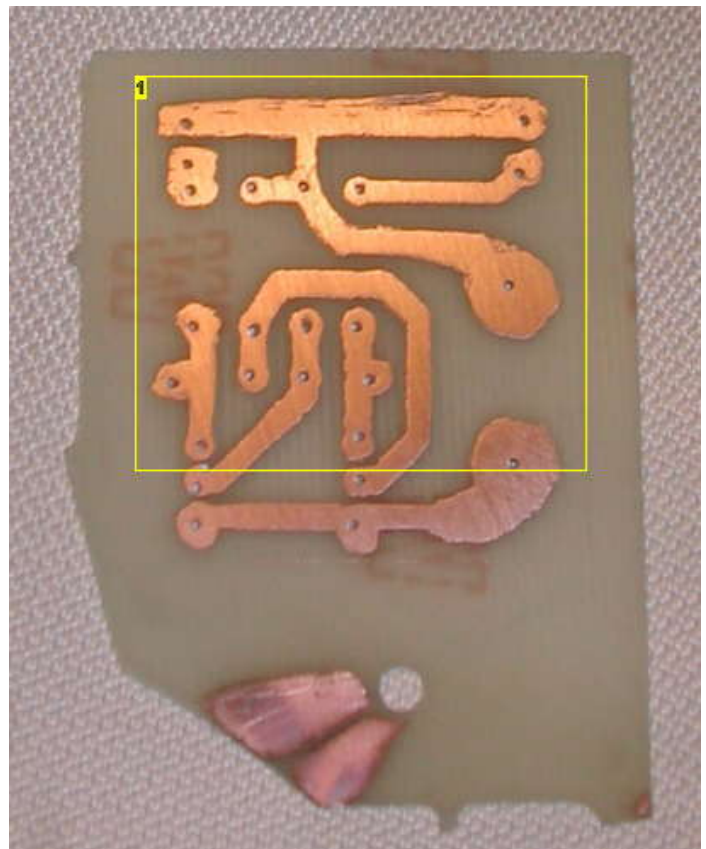
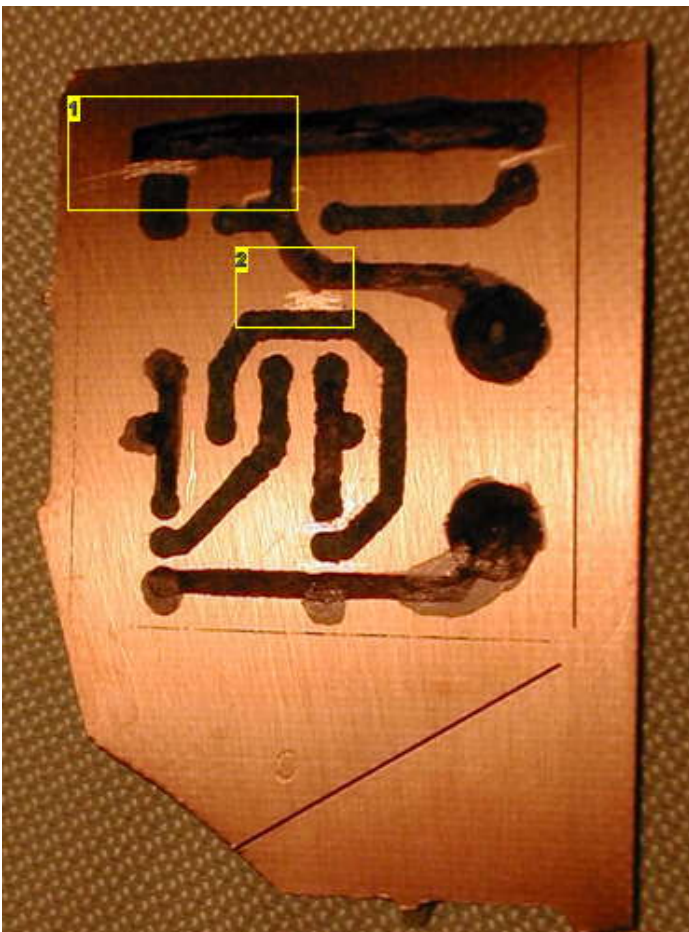


Image Notes

1. A piece of precision construction it's not. It doesn't HAVE to be. That's the point!

Image Notes

1. filled in with a sharpie marker.
2. Scraped away with x-acto. Isn't it nice to have ROOM!

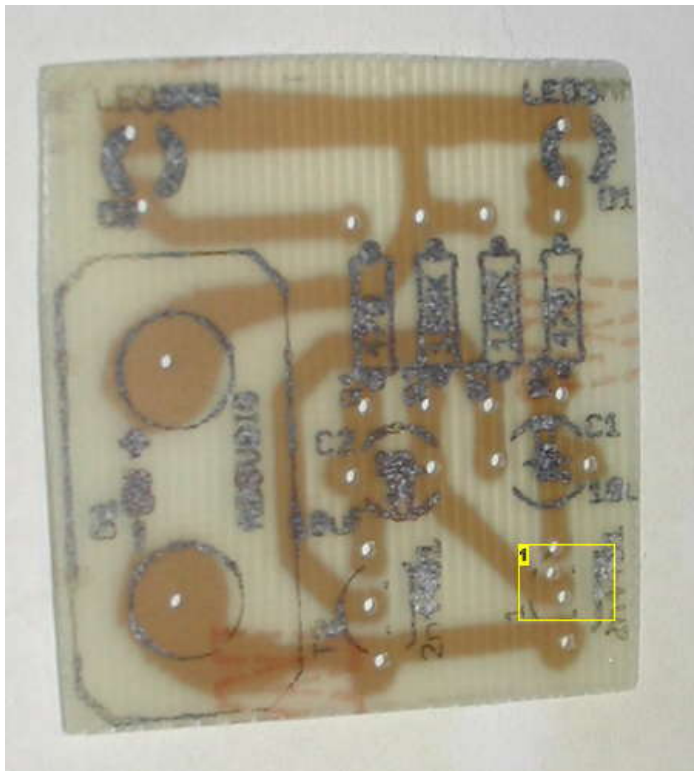


Image Notes

1. Pay no attention to the extra hole. I HATE drilling by hand!

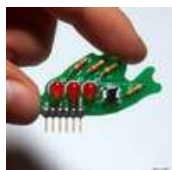
Step 15: Summary

This is just an example, based on some personal opinions. The key thought is that the wider your traces, and the more space between them, the easier your board will be to fabricate by hobbyists. And most PCB packages have settings that can be modified so that they'll do most of the work for you...

Related Instructables



Blinky the LED pet by ynze



Blinky Fish by marc.cryan



Render 3D images of your PCBs using Eagle3D and POV-Ray by ongissim



How To Make an Illuminated LED Eye Loupe by Gadre



Adding Custom Graphics to EAGLE PCB Layouts by iobridge



Eagle-ize Leevonk's PIC protoboard by westfw

Comments

50 comments

Add Comment

[view all 68 comments](#)



panic mode says:
nicely done, congratulation.

Dec 30, 2010. 10:25 PM [REPLY](#)

i was doing same thing until i got access to mill which makes prototyping simpler (you don't need to etch) but the soldering is a bit harder.

couple of months ago i started using KiCAD because I needed to make some larger boards but could not afford paying for software (KiCAD is free). it was quite easy to get familiar with too.



flyingpumpkin says:
Where, exactly, does one find the menu option to increase the trace width?

Feb 15, 2010. 6:28 PM [REPLY](#)



kdrummer says:
I just found that you can use the "change" tool (looks like a wrench) and select a new width from the dropdown. However, you have to click on each trace segment in order to change it. Can anyone help on how to automate this for changing many traces?

Mar 29, 2010. 1:36 AM [REPLY](#)



FazJaxton says:
Select the change tool, setting "Width" and the desired width. (The author is saying that you can do this by typing "change width 1.0mm", but you can select it with the mouse and the wrench tool as well). You can then select the "Group" tool and select everything to be changed. Then right click on any part of the selection and choose "Change: group". This will change all selected traces at once.

Sep 25, 2010. 7:02 AM [REPLY](#)



Sockles says:
Thanks so much for clearing this up for me! I always routed myself because I never could get the traces big enough for my cnc to route.

Sep 8, 2010. 11:29 AM [REPLY](#)



ookid says:
.8 mm or .8 cm?

Apr 10, 2009. 4:37 PM [REPLY](#)



Algag says:
i beleive .8cm or 8mm

Aug 13, 2010. 6:55 AM [REPLY](#)



raghavendraelectronic says:
Hi one and all., myself rbk., I want to design pcb., can any one please help me which software should i use for designing pcb. I'm new to design . So please guide me about softwares. Thanks to any replies

Jun 25, 2010. 4:58 AM [REPLY](#)



Algag says:
use eagle cad it is available for free with limitations for any nonprofit, prototyping, and home projects the instructables on how to use it are a dime a dozen

Aug 13, 2010. 6:46 AM [REPLY](#)



hgk says:
I am using eagle's free version to do a double sided board at home and would like to add some pad area to the top traces as the board will not have plated-thru holes. Is there a way to do this (or to edit out unwanted bottom pads if printing the top layer with pads?). Is there any other common way of connecting the top traces to components? Thanks for any help.

Dec 26, 2006. 5:28 PM [REPLY](#)



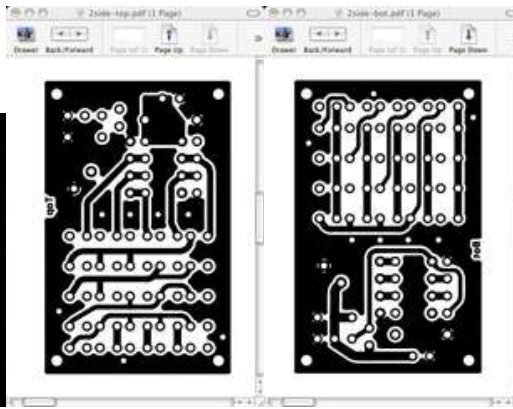
hgk says:
The hard way:
Open a **renamed** copy of finished project.
Goto board and select top, pads, and dimension layers.
Draw a wire x-hair somewhere on the board for future alignment reference.
Group, copy, and paste a copy of everything alongside (pads wont copy).
Add vias to traces wherever you want top layer solder pads. Via sizes can be adjusted to suit.
Goto **renamed** schematic then group and delete everything.
Goto board and group and move everything into the now blank dimension area. Use the wire x-hair as an alignment reference.
Now print the board to use for the top etch pattern.
If there's an easy way please let me know.

Dec 27, 2006. 11:13 PM [REPLY](#)



westfw says:
It doesn't do what you want if you select TOP, PADS, and VIAs layers, and then print as normal, perhaps with 'mirror' selected? here's sample "print to file" output from a 2-sided board I have:

Dec 28, 2006. 9:19 PM [REPLY](#)



Eonir says:

Jul 2, 2010. 12:01 PM [REPLY](#)

Oh. Looks like you've got yourself a ground loop. Just sayin :D Should a lightning crash nearby, a current would flow within your ground plane. That can usually cause damage. Of course, that doesn't really matter, the odds are close to zero, so don't worry. I'm just dropping random knowledge like a clumsy librarian.



hgk says:

Dec 29, 2006. 2:50 AM [REPLY](#)

At the time I felt that I didn't want all the pads etched on the top layer, just wanted the ones that were connected to top traces. But looking at your finished layout it looks just fine to have everything there. Thanks again.



westfw says:

Dec 29, 2006. 5:32 PM [REPLY](#)

Ah. Yes, it would be a challenge to only produce the top pads that HAD to be there; I don't know any way to do that.



westfw says:

Dec 26, 2006. 5:43 PM [REPLY](#)

I'm not sure what you mean. There are some "ticks" to making a double-sided board that won't have plated through holes (probably a good subject for another Instructable) that mostly consist of ensuring that the pads that pass a signal from top to bottom layers only occur on components where you can easily solder both the top and bottom sides (or use EXTRA vias, which is opposite the usual optimization for professionally manufactured PCBs.)



hgk says:

Dec 26, 2006. 6:17 PM [REPLY](#)

Sorry for the confusion. Normally one would etch the bottom layer and pads together and etch the top layer without pads. I would like to etch the top layer with just the pads associated with the top layer traces. Is there any way to something like that? Bottom line: how do I electrically connect the top traces to the components on a home made double sided board without including some pad area along with the top layer traces. Thanks once again for your help.



westfw says:

Dec 27, 2006. 3:00 AM [REPLY](#)

Normally both top and bottom copper layers include the "pads" EAGLE layer; you just include PADS in whatever output technique you're using. I'm working on an instructable about doing output from EAGLE, which is shaping up to be largely a discussion of LAYERS and what they really mean (and some of "why?")



janw says:

Jun 27, 2010. 10:15 AM [REPLY](#)

Hi, Really great tutorial and very clear to understand. But I must say that the standard settings for eagle suits me well. I must admit that I don't use the tonertransfer method but photosensitive PCB's. On the other hand, I do not have fancy equipment: I use a Philips facial tanner to expose the PCB's to UV light and drill press to drill the holes.



hgk says:

Jan 20, 2007. 9:24 PM [REPLY](#)

When making a board using the toner resist method, is there a way to set up multiple copies of the board lined up to print on a single page?



jeff-o says:

Jun 24, 2010. 11:19 PM [REPLY](#)

I wish! At best you can print two, three or four copies on the same page by running the same paper through, and aligning the design to different corners (possible in Eagle, not sure about other programs). The downside is that on each pass the paper darkens a bit from scraping toner off the rollers, so with this method you really can only do four boards max. Hmmm, maybe there's a way to "panelize" the designs, though. Does anyone know if Eagle can panelize?



westfw says:

Jun 25, 2010. 12:42 AM [REPLY](#)

Free Eagle will panelize up to the limits of the free version (80x100mm); there are some ULPs (eg panelize.ulp) that aid in the duplication of labels and such. And there are tricks you can do with postscript output, or gerbers, to panelize outside of EAGLE. Output tricks was supposed to be the subject of another instructable, but I got a bit bogged down.



jeff-o says:

Well there you go. I'll have a look for those ULPs. I've got the paid student version so I could panelize a slightly larger board.

Jun 25, 2010. 4:33 AM [REPLY](#)



mircerlancerous says:

I make home made PCBs all the time using the Toner Transfer method with photo paper. With regard to the software, while Eagle is a powerful software package and a free one at that, it is much more than you need for casual circuits. I prefer PCB123. It is easy, simple, and free. When you're done your design just choose print schematic, click black and white, select your layers and you're done. It's probably no good if you're planning on involving a board shop other than the PCB123 people but for homebrew it's great. All depends on how much learning you want to do.

Jun 24, 2010. 4:30 PM [REPLY](#)



westfw says:

I didn't mean this instructable to be entirely specific to EAGLE; other CAD packages probably have very similar features and even terms (like "design rules.") They're sort of industry-standard. The thing that attracted ME to EAGLE was the support for non-windows operating systems...

Jun 25, 2010. 12:48 AM [REPLY](#)



jeff-o says:

Excellent work! Wow, I just checked the published date. Nice to have your work suddenly recognized again, eh? Anyway, thanks for this. It's good to know that you can increase pad sizes using DRC. That often screws me up, with the drill pulling up the pad...

Jun 24, 2010. 11:21 PM [REPLY](#)



westfw says:

Yes; it's a bit odd to have Instructables "featured" and become "popular" when they predate the existence of those features... One wonder what other gems are back there.

Jun 25, 2010. 12:45 AM [REPLY](#)



renoir says:

Nice tutorial! It explains a few things that I was mis-understanding, like the word "check" = "modify" :-). I assumed it would just complain about pad sizes, etc, not change them for me. Nice tip about drill-aid.ulp too :-)

Jun 24, 2010. 10:38 PM [REPLY](#)



hondaman900 says:

Is there a way to start over with the PCB layout? I used autorouter and didn't like the routing, especially after I moved components. I ended up deleting tracks thinking autorouter would simply redo them, but now I'm stuck. Seems like there should be a re-route process/option, or at least the ability to remove a PCB design and start over from a schematic. Any suggestions? I can't find this via user manual, Eagle's help file or Google.

Mar 26, 2010. 11:22 AM [REPLY](#)



westfw says:

Next to the "route manually" button is a "ripup" button that converts tracks back into air-wires. To get rid of everything, click "ripup" and then click the traffic light that appears in the top toolbar. Or type "ripup ;" in the command-line window.

Mar 26, 2010. 9:41 PM [REPLY](#)



os_sanches says:

Hi this dru file, is optimized to milli a borad or just print???

Mar 1, 2010. 4:49 AM [REPLY](#)

Tks



os_sanches says:

Hi, I have a message of clearance in transistor tip122(package to220) in (my circuit) can i fix this problem??? I'm using you dru file.

Mar 4, 2010. 3:12 AM [REPLY](#)



westfw says:

Either? It's optimized to produce thicker tracks, further apart, and larger pads around drill holes. This gives you easier fabrication whether you are milling, using toner transfer, photoetching, or just drilling. The resulting board should hold up better to all forms of amateur etching, drilling, etc, as well as clumsier soldering by less experienced solderers.

Mar 1, 2010. 7:54 AM [REPLY](#)



wrangler says:

Well done! What I'd add, for those of us who do toner transfer, is the ulp (well, acknowledging that your topic is DRC, and not the add-on stuff) named drill-aid.ulp. I don't want the large pad holes on my board, I want max copper, which I get after the restrung change, and just enough hole to guide my Dremel tool bit to the center of the pad. Drill-aid.ulp closes down the hole to a size that you specify in mm. Then, when you print it out with the laser printer for toner transfer, just enough hole is there to help steer the bit.

Apr 26, 2007. 5:59 PM [REPLY](#)



flyingpumpkin says:
ulp?

Feb 15, 2010. 6:16 PM [REPLY](#)



wrangler says:

The ulp is "user language program", or some German phrase that means the same thing. A ulp is a script that automates some part of a process.

Feb 18, 2010. 2:39 PM [REPLY](#)



ldestefa says:

Thanks for the tips! Needed a bit of help and your explanations are clear and to the point. Thanks. I passed my PCB through a photopicture size laminator (\$10) several times and it does an even transfer using laser semigloss paper or cheap inkjet gloss in an older laser printer (some inkjet gloss paper can jam up the laser printer). Set laser printer to 100%contrast. A sandwichmaker with 2 pieces of flat 16 gauge steel over the PCB and paper is good too.

Aug 7, 2009. 12:15 AM [REPLY](#)



westfw says:

0.8mm. This is the width of the copper traces on the PCB, and the minimum hole size for component leads and such, both of which are typically quite tiny. A 0.8mm wire would be about 20g wire, which is quite thick for most electronics purposes (of course, it's a flat trace in this case, but...) 30g wire-wrap wire is about 0.25mm, and typical "standard" process for professional PCB manufacturing is about 0.2mm (8 mils - a "mil" is 0.001 inch) You can use other units if you prefer (0.8mm is about 32 mils or .032 inch), I just find mm to be the convenient unit for this sort of size range (alas, I'm not all-metric. Get up to BOARD sizes and I think in inches. Sigh.) The key point is to change the defaults, which assume professional manufacturing, to much larger values to give the amateur more room for "by hand" sort of tolerances. (Frankly, larger traces would improve many a "professionally made" board as well; easier to manufacture, less fragile, less prone to errors becoming "fatal", easier to apply rework...)

Apr 10, 2009. 6:14 PM [REPLY](#)



jomaro says:

Great 'ible, and so many usefull comments. i've got to start using eagle, but it been a very slow start... i'm gon'na come back here often..... Thanks guys!

Nov 27, 2008. 4:50 PM [REPLY](#)



Leroy says:

Thank you for taking time to explain EAGLE's design rules clearly and concisely. You have helped me enormously.

Jul 17, 2008. 7:05 AM [REPLY](#)



rebel09 says:

so i have a really stupid question, is there a way that i can make these with more easily attainable materials, because im a teenager and i dont have access to all these things, im trying to do the minty boost but would like it if i didnt have to buy the board? what can i do if anything?

Jun 4, 2006. 7:17 PM [REPLY](#)



factor grimm says:

Don't think that! In electronics, everything is available to you. There is no age check. Just find out where local suppliers are and go check it out. I wish I had gotten into electronics when I was a teenager instead of only recently. If I had realized how much stuff I had had access to, maybe I would have.

Feb 27, 2007. 6:56 PM [REPLY](#)



verbero says:

Since you recently got into electronics as well, i was wondering. Where did you start, because i've been looking around and i feel overwhelmed.

Apr 6, 2007. 11:46 PM [REPLY](#)



RedBinary says:

There is a very thorough theory and practice site at <http://www.electronics-tutorials.com/basics/basic-electronics.htm> that I would recommend. If you join the newsgroup there as well you will be connected to some high-end engineers and can not only get help you need as someone just starting out, but can also get an inkling of what kinds of things you can look forward to as your experience increases...

Jan 13, 2008. 7:39 AM [REPLY](#)



factor grimm says:

Well, I can't remember the exact page, but i just googled around for basic tutorials on electronics.. You have to start with a fundamental understanding of the basic kinds of components, resistors, capacitors, diodes, transistors.. then transformers, inductors, then zener diodes, variable resistors, etc... But I don't claim to know everything about those things.. I got a book called "the quintessential pic microcontroller" by sid katzen, and then followed that. All I need to know is that my PICs need 5 volts (or so) and some basic support components, and I can program the rest! Good luck with your studies!

Apr 7, 2007. 4:40 PM [REPLY](#)



Regax says:

First off, I understand that this thread is over one year old, but i figured I would reply and list a good beginner electronics site for those who read through these old threads as i just was. A very helpful site that taught me a lot about electronics is Kelsey Park School Electronics Club. Just google that or go to this site...

<http://www.kpsec.freeuk.com/>

Apr 27, 2008. 7:56 PM [REPLY](#)



francisew says:

I have ordered PCB's from professional places, but eventually started making them by hand because the turnaround time was much shorter making them myself. I have been able to do 64 pin LQFP surface mount microcontrollers pretty easily, and the boards work (I usually have 1 or two of the smallest traces that require jumpering using fine wire (24 guage) to get everything working 100%. My tutorial (which I will update in the next few months) is on my web site: [Esmonde-White.com](http://www.instructables.com/id/Make-hobbyist-PCBs-with-professional-CAD-tools-by-/)

Jul 14, 2007. 3:44 PM [REPLY](#)

I use eagle and then do a toner-transfer method using staples glossy paper. Then I iron the toner onto copper board, and finally I use an HCl/peroxide solution to dissolve the copper. This is quite fast (less than 30 minutes for the whole board) and gives really good resolution (LQFP size SMT components work out, and the through holes are aligned with sufficient resolution for jumpering through with wire). I now also use a liquid tin to tin the PCB, so that it will be easily solderable even if I don't solder it immediately (since otherwise the oxide makes it hard to solder).

<http://www.instructables.com/id/Make-hobbyist-PCBs-with-professional-CAD-tools-by-/>

I can print out multiple boards at once by duplicating the high resolution bitmaps and laying them out in Gimp so that I get mutiple printouts. I only ever etch things one at a time because the layer offset error is high if many boards are put onto a single large area. (I think it's because of contraction of the paper when it's hot).



josheeg says:

Jul 5, 2007. 7:46 AM [REPLY](#)

I want to use sparkfun's pcb design service I can get something really small for less than twenty and allows me to use *gasp* surface mount and 40 pin through hole. A instructable on that would be good.



westfw says:

Apr 26, 2007. 7:43 PM [REPLY](#)

Another instructable in how to do "output" from Eagle is coming and should be published "real soon now." It'll include info on drill-aid.ulp, as well as how to get similar output via the CAM processor to postscript.



rebel09 says:

Jun 4, 2006. 7:18 PM [REPLY](#)

o as an edit to this, i dont have a laser printer anywhere acessable except at school.

[view all 68 comments](#)