

[Hackaday](#)

Fresh hacks every day

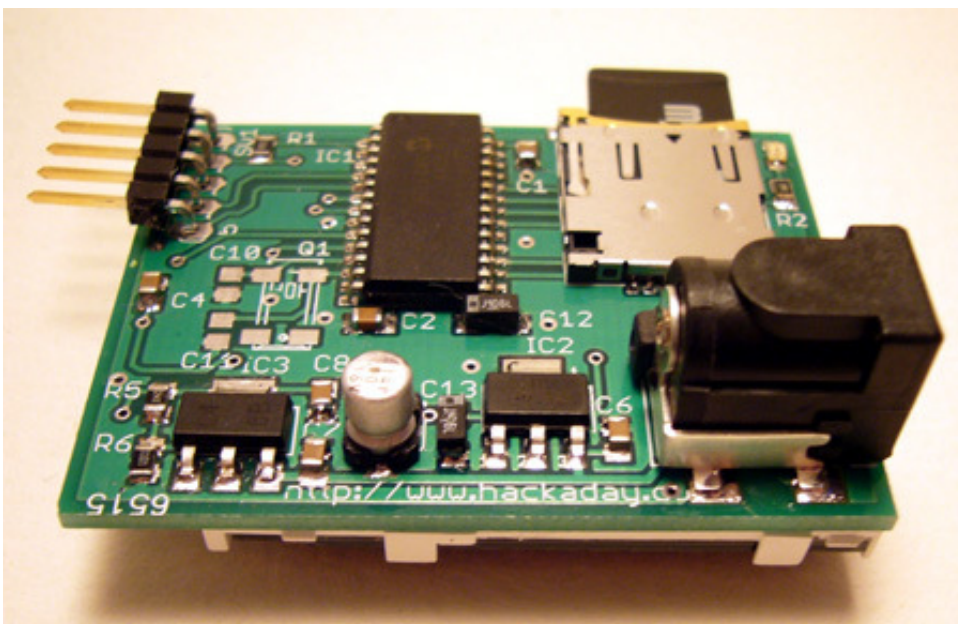
- [Home](#)
- [Hackaday Projects](#)
- [Store](#)
- [Submit a Tip](#)
- [Forums](#)
- [About](#)
- September 15, 2014



[See the Semifinalists](#) || [Learn about The Hackaday Prize](#)

## How-to: Prepare your Eagle designs for manufacture

January 15, 2009 By [Ian](#) [45 Comments](#)



[Cadsoft Eagle](#) is a multi-platform freeware circuit layout program. Lots of open source hardware is designed in Eagle, and it's become a hobbyist favorite. We use it for all of our [hardware designs](#).

There are several ways to turn an Eagle design into an actual [printed circuit board](#) (PCB). We'll show

you how to save Eagle designs as industry-standard gerber files that are accepted by any PCB manufacturer. You can use the gerbers to order a single prototype, or a full panel.

## Introduction

Toner transfer is the beginners' favorite way to make a PCB because the investment in materials is minimal. We've [covered toner transfer before](#). Most PCBs in our [how-tos](#) are made with the [photo-resist process](#). The photo process makes nice boards, but requires a bit of equipment; sensitized boards, developer, and an ultra-violet light source.

Some board manufacturers, like [Olimex](#), make PCBs directly from Eagle .brd files. Most require a minimum order of one [eurocard-sized](#) PCB (100mmx160mm). Good if you need a few boards, expensive for a single experimental prototype.

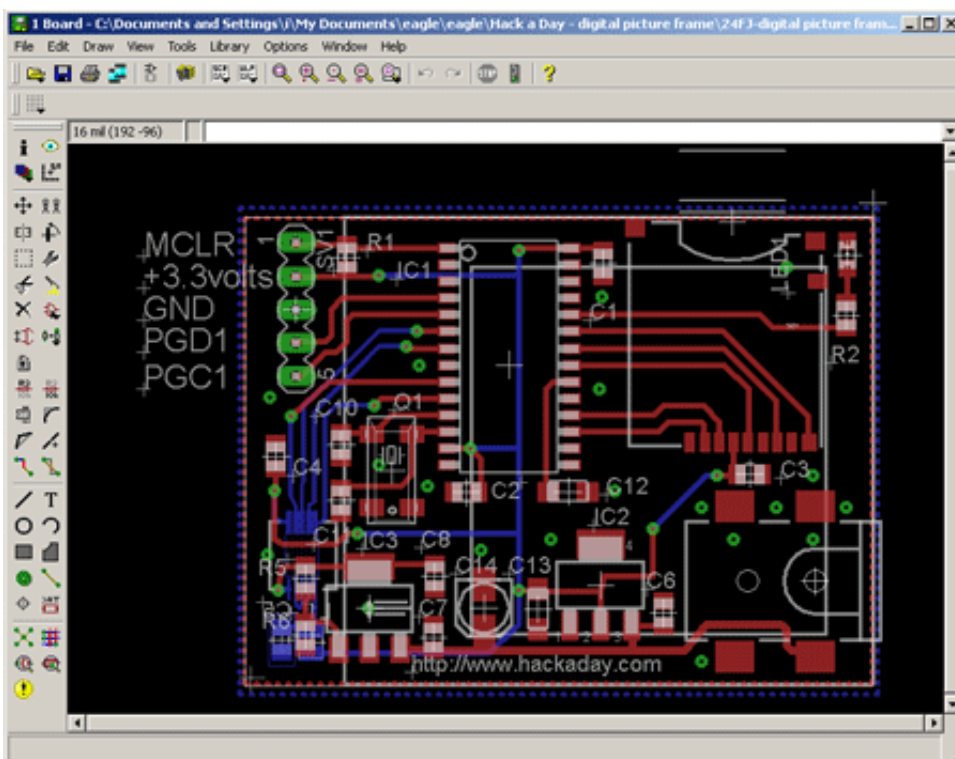
The cheapest option is to submit [gerber files](#) like the professionals. Any PCB manufacturer will accept gerber formatted design files. [Gold Pheonix](#) sells 155square inches of PCB panel for \$110. If you're looking for something smaller, services like [BatchPCB](#) and [PCB-Pool](#) combine small orders and submit them as a full panel. Either way, you'll submit gerber files to the board house. This is the process we describe.

## Process overview

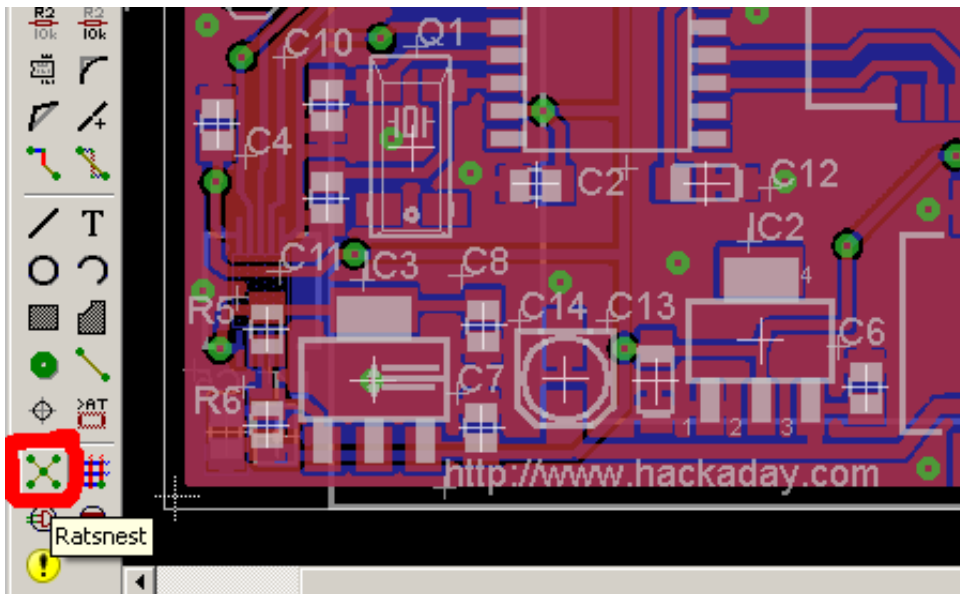
- Prepare the design.
- Create gerbers, generic files accepted by any PCB fab house.
- Verify that the gerbers are correct.
- Send the design for production.

## Prepare the design

We're going to walk you through the process of preparing our [digital picture frame](#) PCB for production. This design requires a double-sided board with fairly small traces.



Download the [project archive](#) (ZIP) from last week. Open the .brd file with the freeware version of [Cadsoft Eagle](#).

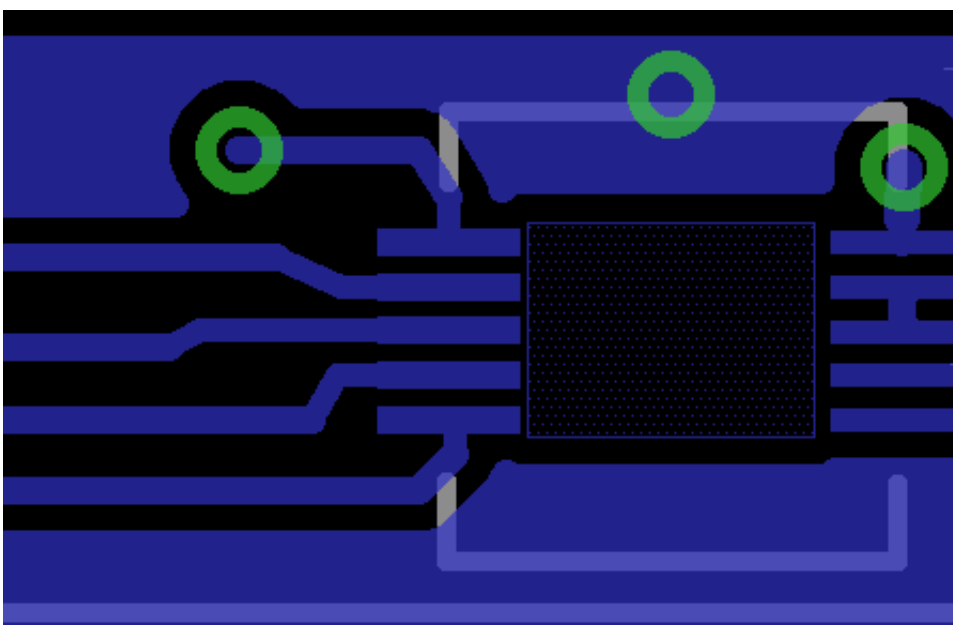


The ground fill is empty when the file opens. Press the ratsnest button (*or Tools->Ratsnest*) to fill in the empty polygons.

#### What are the PCB rules and limits?

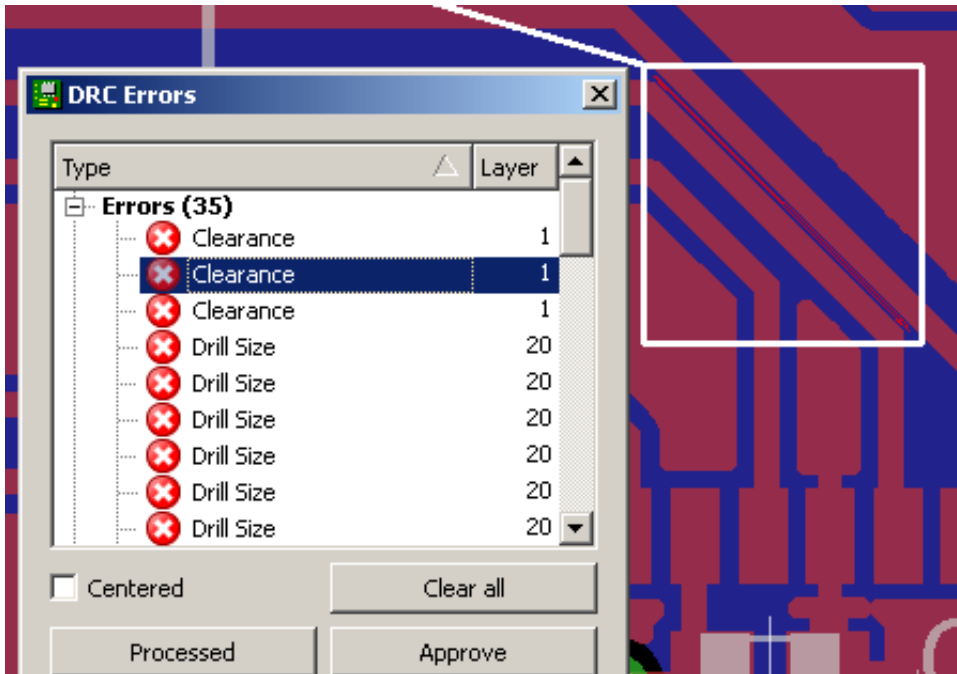
- 2 or 4 layer boards - 0.062" FR4 material
- 2 Layer sizing:
  - 8mil spacing minimum
  - 8mil traces minimum
  - 20mil minimum drill size
- 4 Layer sizing:
  - 6mil spacing minimum
  - 6mil traces minimum
  - 13mil minimum drill size
- 500mil maximum drill size
- No internal routes, no v-scoring

Board manufacturers publish specifications outlining their production capabilities, such as the smallest possible traces, spacing, and drill size. BatchPCB has 8mil [minimum traces and spacing](#), and 20mil minimum holes.



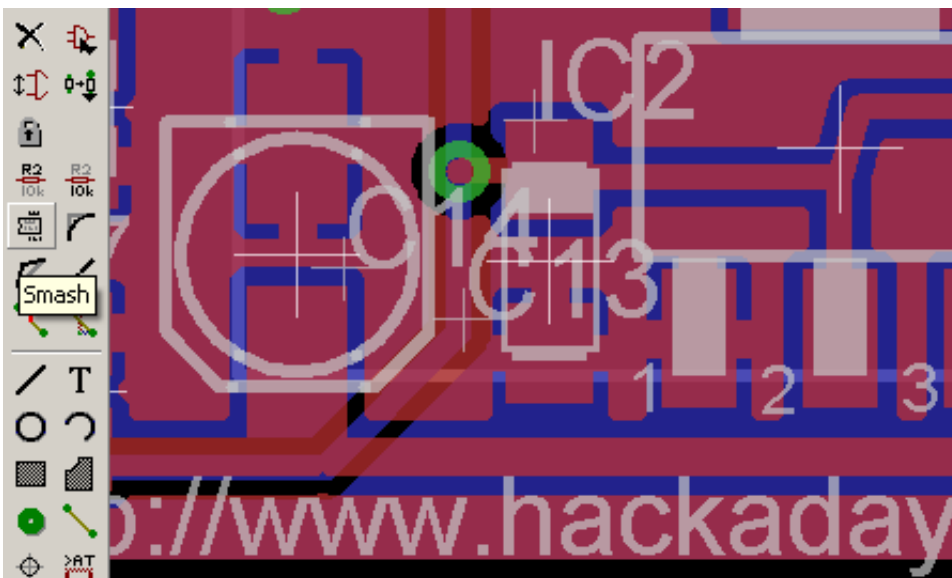
Don't torture the manufacturer. Just because they advertise 8mils, doesn't mean it's safe to make

every trace 8mils. Slightly larger-than-minimum tolerances will reduce [manufacturing errors](#). The digital picture frame has 8mil traces around the tiny [LCD connector](#), shown above. The traces are 8mils only until there's enough clearance to use 10mil traces.



Use Eagle's *design rule check* to make sure your board doesn't exceed the manufacturer's production abilities. Download the [SparkFun design rules](#) (DRU) for BatchPCB, or the Olimex [8mil](#) (DRU) or [10mil](#) (DRU) design rules. Click the DRC icon (or, *Tools->DRC*) and load the design rule file. Eagle analyzes the design and highlights any areas that violate the design rule parameters.

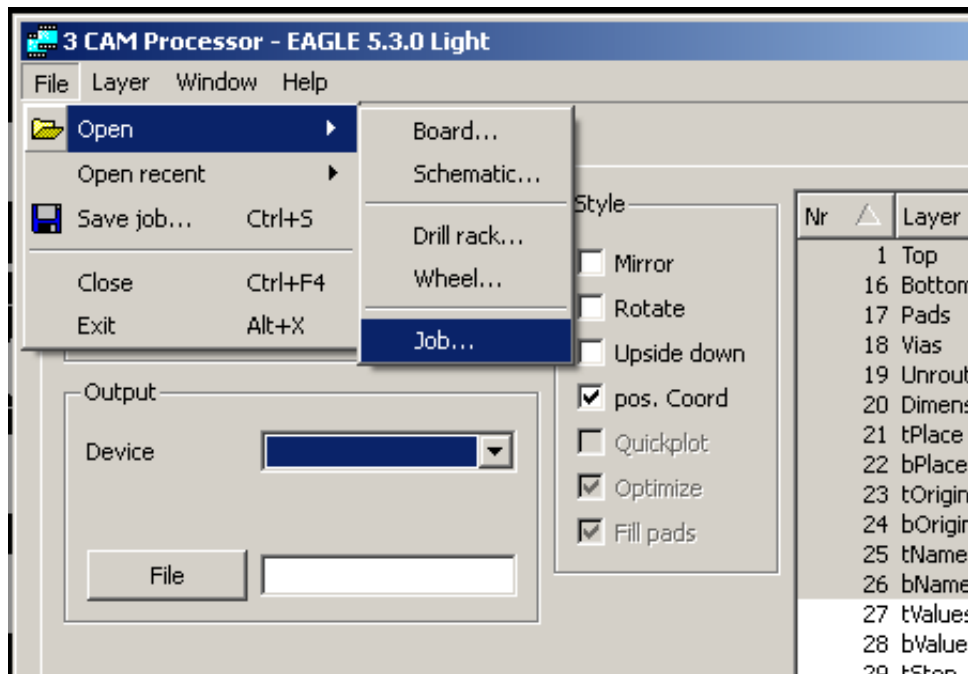
Correct any errors. Here, the spacing between traces is too close. Sometimes the spacing on a part footprint is too small to be manufactured. Sparkfun's default footprint for the Nokia LCD connector had pad spacing less than 8mils. We [edited the part library](#) to make the pads smaller, and the separation larger.



It's helpful to include part numbers on the printed silkscreen layer. BatchPCB prints a silkscreen on both sides. Be sure to see what your board house offers, some charge extra. Use the smash tool to unlink obscured labels, then move them to a better location.

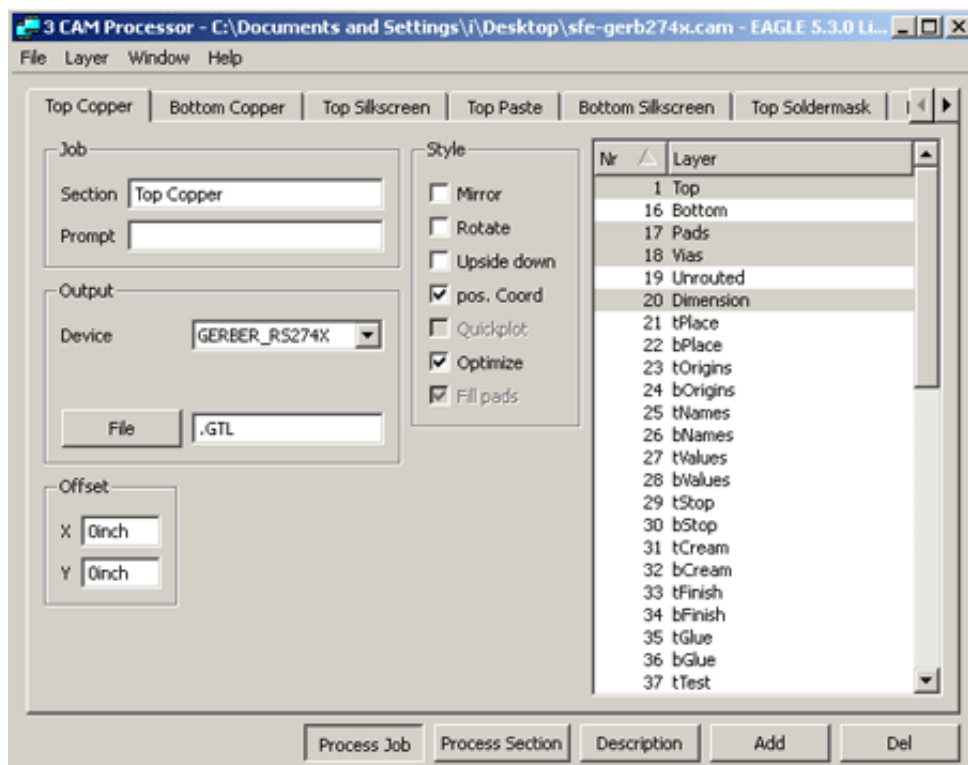
## Create gerber files

Gerber files are the [PDFs](#) of PCBs. Gerber files describe a PCB exactly as it should appear, agnostic of the display hardware. It's a final production format that isn't intended to be edited. We created our gerber files in Eagle using the procedure outlined in [SparkFun's Eagle tutorial](#).



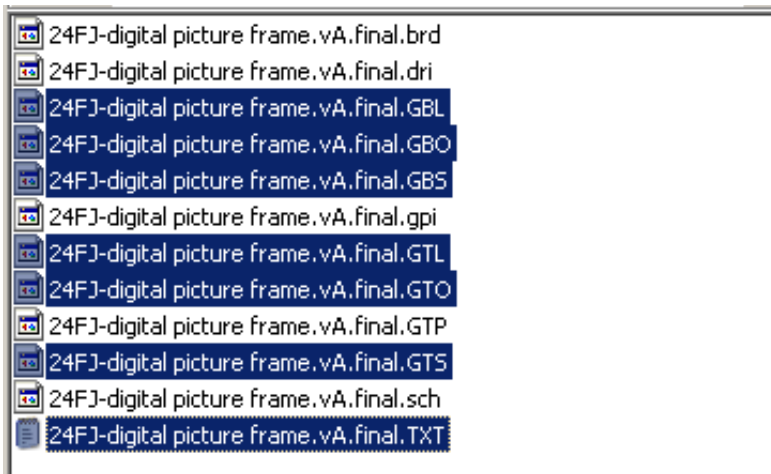
The Eagle CAM processor writes gerber files, open it from the menu under *File->CAM processor*.

SparkFun has a [script](#) (CAM) that configures the CAM processor to make gerber files. Load the CAM script using *File->Open->Job...*



By default, SparkFun's silkscreen configuration only includes the *place* layer. Our parts usually have labels on the *names* and *docu* layers, activate these layers on the top and bottom silkscreen tabs to add them to the output.

Click *Process Job* to create the gerber files.

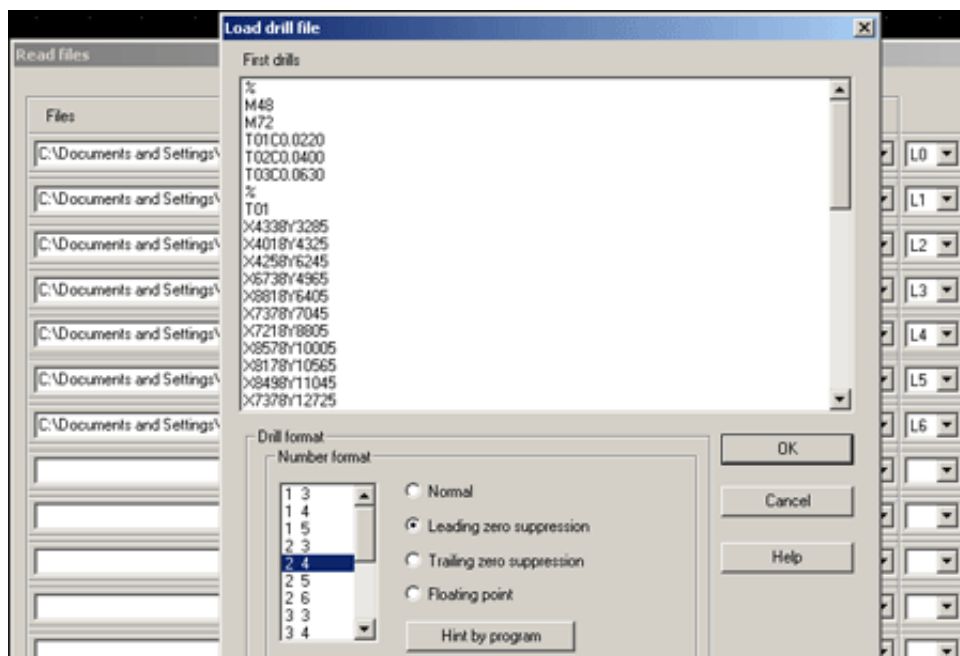


The CAM processor creates seven files that we need.

- Top and bottom copper (.GTL, .GBL)
- Top and bottom solder mask (.GTS, .GBS)
- Top and bottom silkscreen (.GTO, .GBO)
- Drill file, 2.4 leading (.TXT)

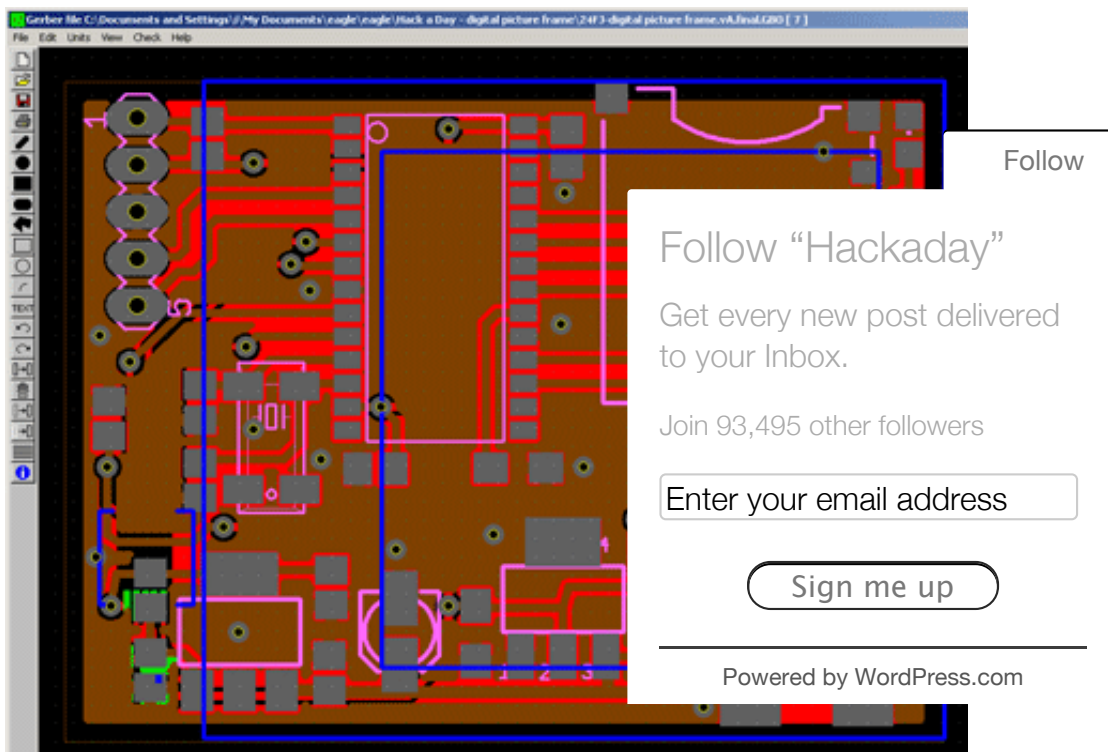
### Verify that the gerbers are correct

Verify the CAM output in a [gerber viewer](#) to make sure everything was positioned correctly. We followed SparkFun's suggestion and used [Viewplot](#).



Load the seven files with Viewplot. *Be sure to specify the drill file type as 2.4 leading.*





Check for errant vias, mirrored layers, and alignment. We've noticed that text added to the silkscreen layer is usually bigger than it was in Eagle. Correct any problems and run the CAM processor again.

When everything looks good, the board is ready for production.

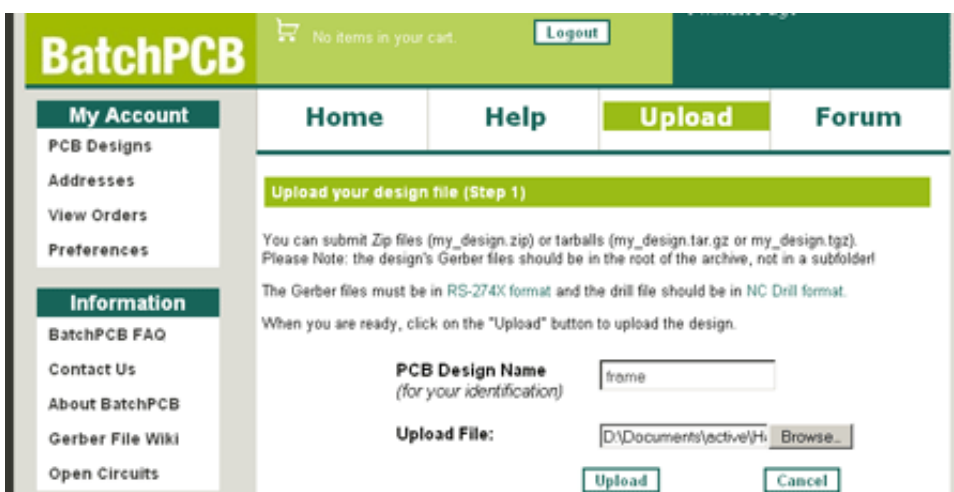
### Send the design for production

[Zip](#) the seven gerber files and submit them to the PCB fab house. *Remember to tell them that the drill file format is 2.4 leading.*

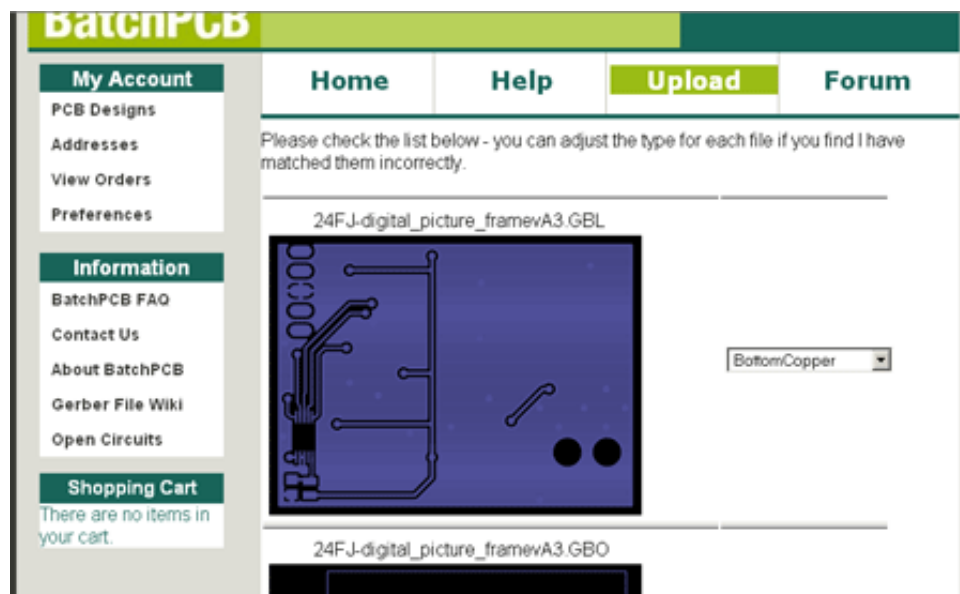
BatchPCB is a pooled panel service that sells space by the square inch. Other manufacturers and batch services require you to order *at least* a full eurocard. We use BatchPCB for prototyping because we never need the extra board space of a full eurocard, and we don't mind the average 20day wait.

At BatchPCB, \$2.50/square inch buys a PCB with silkscreen on both sides, unlimited vias, and a huge range of drill sizes; stuff that usually costs extra. BatchPCB's minimum traces, spacing, and drill are similar to other prototyping services. There's a \$10 *per order* setup fee, but an order can include multiple designs. Shipping, even internationally, isn't outrageous.

If you need a lot of the same board, look at Gold Phoenix. They manufacture boards for BatchPCB. A 100 square inch panel is \$100, a 155 square inch panel is \$110.



Create an account at [BatchPCB](#). Click upload to add a new design. Name the design and upload the zip archive containing the 7 gerber files.



Verify that the gerber layers were successfully detected.

Please check to make sure that the board statistics here are correct

Board Statistics				
<i>general</i>		<i>dimensions</i>		<i>price</i>
PCB Design Id:	5085	Weight (oz.):	0.13	Produ
Model:	frame	Size (Y Axis):	1.41	
Number of Layers:	2	Size (X Axis):	1.87	
Total Size (sq. in.):			2.64	

Verify that the correct size was detected.

## Thanks for using BatchPCB!

Your design has been sent to the DRC Bot for a design rule check. **receive an email with the results of the check in a few minutes**. For a complicated design this will take a little longer. Once the design passes, you can view it and order it from your home account page from the "Pcb Designs" section.

PCB Design Id:	5085
Model:	frame
Total Size (sq. in.)	2.637
Production Price:	7.50

[Upload Another](#)

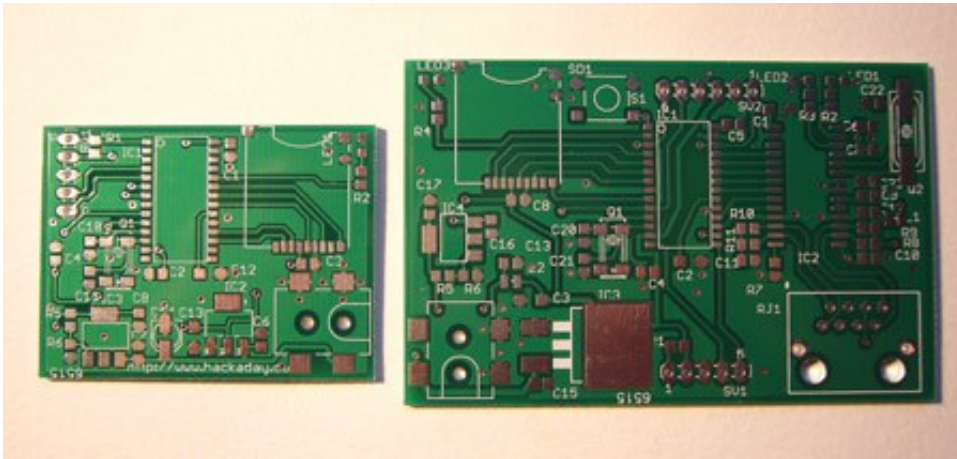
[Continue](#)

The BatchPCB rule check 'robot' will verify that your design meets production standards, and send an email with the results of the check in a few minutes.



e-mail in a few minutes. Since we ran our own rule check prior to sending the design, we can expect that everything will be fine. Click continue and you'll have the option to order the board. For more help, see the BatchPCB [help](#) and [support forum](#).

## Receive your boards



Boards arrive from BatchPCB in about 20 days. Check the boards for obvious errors before soldering. Some manufacturers test PCBs, BatchPCB doesn't. We've ordered PCBs from two of the popular hobbyist board houses, Olimex and BatchPCB, and all the boards have been satisfactory.

## Taking it further

It's easy to order professional PCBs using gerber files. Why not build that awesome DIY project you've been putting off?

What has been your experience with PCB fab houses?

UPDATE: the files have been moved! [find them here](#).

[Laser-Based PCB Printer](#) [Recreate a PCB with a Scanner and Inkscape](#)  
[DIY Circuit Boards Look Professional](#) [Make Any Shape Board in Eagle](#)  
[Designing And Printing A Custom Enclosure](#)



Filed Under: [how-to](#), [parts](#) Tagged With: [cadsoft eagle](#), [circuit boards](#), [digital picture frame](#), [diy pcb](#), [eagle](#), [electronics](#), [gerbers](#), [pcb](#)

## Comments

1. *Adam Ziegler* says:  
[January 15, 2009 at 4:43 pm](#)

Great article!

[Reply](#) [Report comment](#)

2. *jimmys* says:  
[January 15, 2009 at 4:46 pm](#)

Excellent write up. Even if you're not going to send it off to get 1000 boards printed, using CAD opens up great features like autowiring and checking for design errors.