Liudr's Blog

Arduinos and physics

| | GO |

HOME   ABOUT   BUY   DONATE   GADGETS   LIBRARIES   SHIELDS   POSTS   COMMENTS

ARDUINO   ANNOUNCEMENTS   GADGETS   CODING   PHYSICS   SENSORS   EAGLE   CIRCUIT BASICS   SOLDERING   UNCATEGORIZED

← Directional keypad for phi-panel   Happy New Year 2013! →

Ordering PCB from seeedstudio

DECEMBER 22, 2012   6 COMMENTS

This is a short tutorial on how to order your PCB from seeedstudio.com

I am not associated with seeedstudio but just a regular customer. They have competitive price and decent quality. They also recently automated design file submission process, although there is still some kinks.

Once you finish designing your PCB with EAGAL CAD, you should use seeedstudio's CAM file to export the design into several files. The CAM exports many files but only the following are useful and should be zipped in a .zip file, per their sales page:

- Top Layer: pcbname.GTL
- Bottom Layer: pcbname.GBL
- Solder Mask Top: pcbname.GTS
- Solder Mask Bottom: pcbname.GBS
- Silk Top: pcbname.GTO
- Silk Bottom: pcbname.GBO
- Drill Drawing: pcbname.TXT

Once you have a .zip file, direct your web browser to seeedstudio.com and you will see their store front:
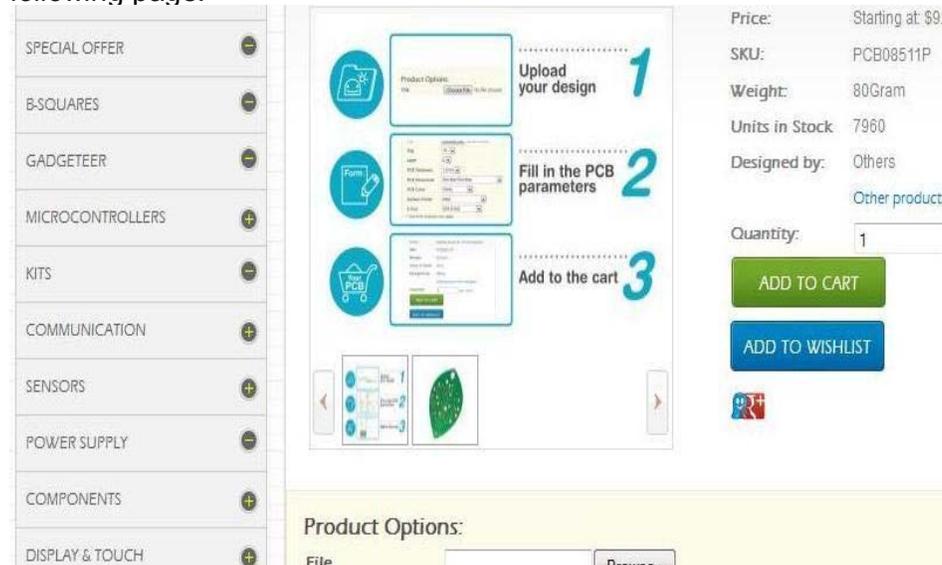
Support my arduino libraries
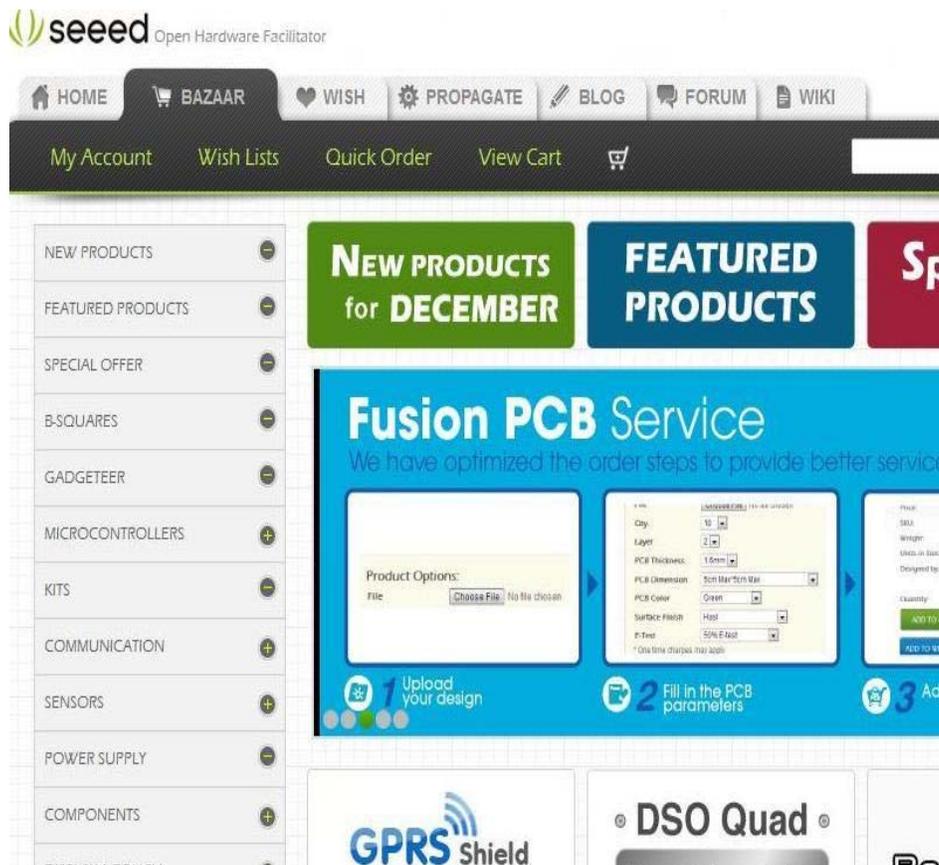
Once you are on their web store, look for "Services" on the navigation bar on the left and click it. It will bring up their services page, with three types of service.
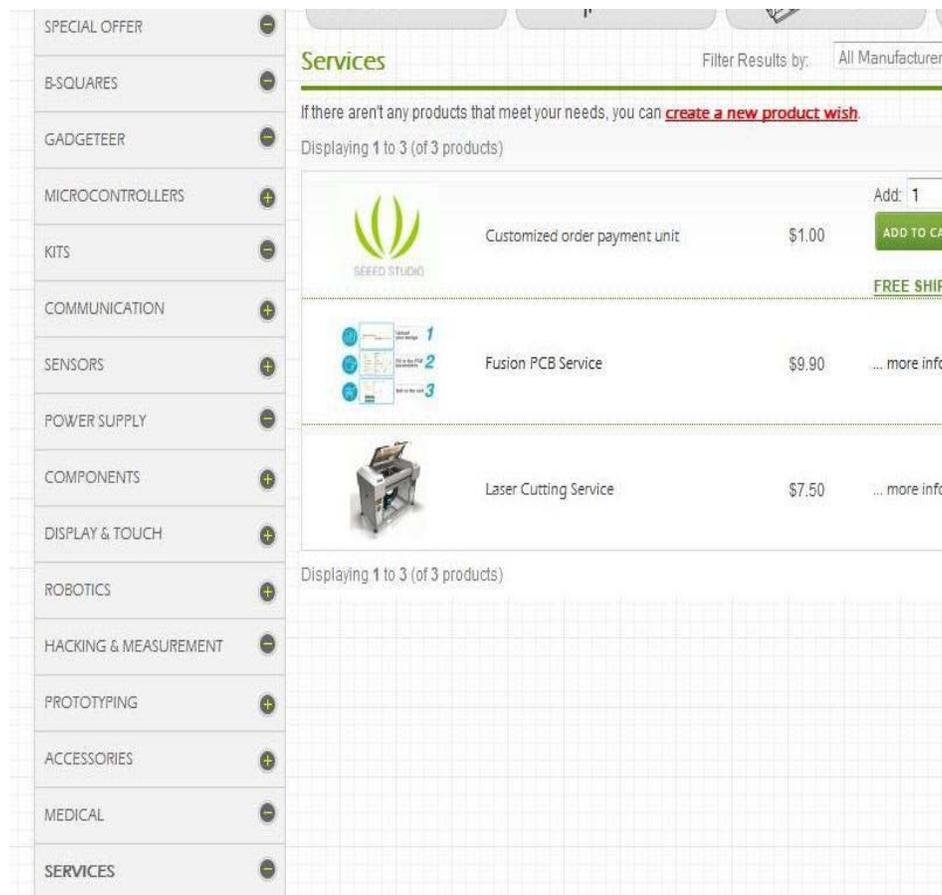
The second one "Fusion PCB Service" will take you to their online PCB ordering page. Click "Fusion PCB Service" and you will see the following page:

There are a lot of options on this page. I will list what they are below:

- File: Pick the .zip file that contains your design. I have two zip files, Shield_V1.0.5.zip, and Power_supply_V1.0.2.zip. I will select Power_supply_V1.0.2.zip
- Qty: Seeedstudio doesn't make 1 PCB, the least amount is 10 PCBs. Go ahead and pick 10, unless you want 50 or 100.
- Layer: You will pick 2 layers. This means you will have circuits both on top and bottom of your circuit boards. I don't see many single layer PCB any more but they were popular decades ago.
- PCB Thickness: Go ahead and pick 1.6mm. Most popular thickness is 1.6mm. If you want some added mechanical strength on your larger boards, you can go with 2.0mm.
- PCB Dimension: This is the maximal size of your PCB. Typical sizes they provide are 5cm*5cm (about 2″*2″) for $10 (10 boards), and 10cm*10cm (about 4″*4″) for $25 (10 boards).
- PCB Color: You can leave this as Green since other colors cost extra $10-$20. It's just the look. Your PCB will be painted with color paint.
- Surface Finish: Pick HASL (Hot Air Surface Leveling). It's a technique that plates holes with hot molten solder and the excess is removed by hot air. You can also pick leadless HASL if you want to go green. You may upgrade to ENIG, which is Electroless Nickel Immersion Gold. Holes will be covered by a thin layer of gold to prevent oxidation.
- E-Test: For somewhat sophisticated designs, E-testing is recommended. For something simple such as the power supply board, it's not very necessary.

Say I have two designs, Shield_V1.0.5.zip, and Power_supply_V1.0.2.zip. I will first select Shield_V1.0.5.zip in the file selector, select everything according to the list explained previously (Qty=10, Layer=2, Thickness=1.6mm, Dimension=10cm*10cmColor=Green, Finish=Hasl, E-test=100%), then push "Add to Cart".

Then I will push "continue to shop" and be brought back to the PCB ordering page. I will pick Power_supply_V1.0.2.zip and select 5cm*5cm. Similar settings to the other design (Qty=10, Layer=2, Thickness=1.6mm, Dimension=5cm*5cmColor=Green, Finish=Hasl, E-test=50%) and add to cart. I'm not testing these boards 100% since they are simple.
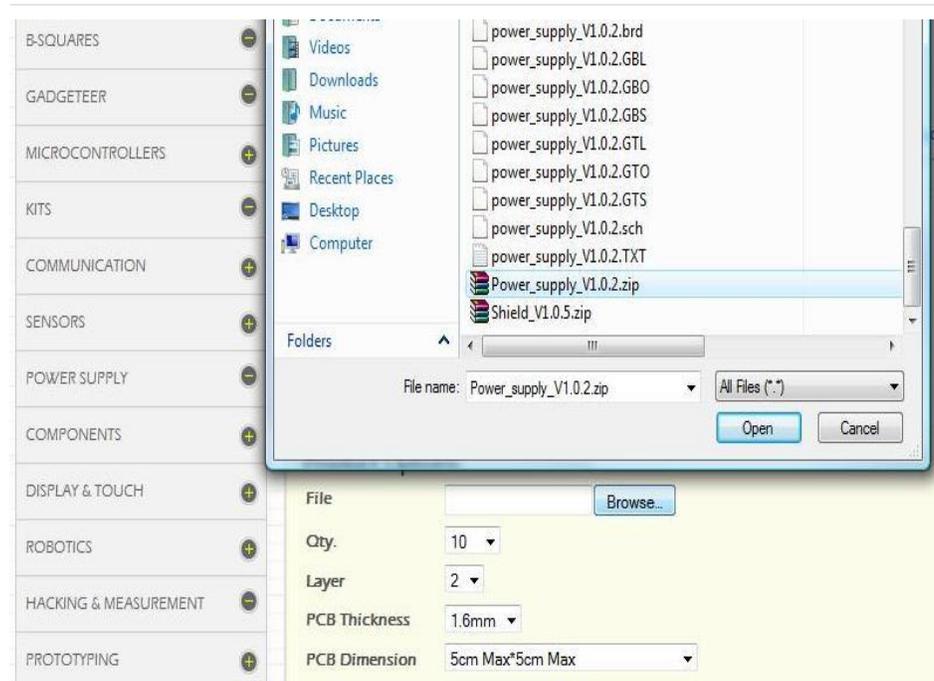
	PCB Color	Green ▼
ACCESSORIES ⊕	Surface Finish	Hasl ▼
MEDICAL ⊖	E-Test	50% E-test ▼
SERVICES ⊕	* One time charges may apply	
MISC ⊖		

After everything is added to cart, I would proceed to check out. You need to set up an account with seeedstudio and have a paypal account. Shipping is less than $10 but slow. I average about 3 weeks between design submission and delivery (to Mid-west, USA). You can also pay DHL and save maybe a week or more, although I never used it. I guess that's it. Well, one bug on their site though: if you select a design zip file and push add to cart, but later remove the item from the cart, you can't upload the same file anymore. I guess they buffer your file but don't remove them when you remove your item from cart. You will have to change the name of the zip file if you do encounter this problem. I'm sure they'll fix it soon.

Oh by the way, since you've read this far, I'll tell you a secret or two:

1. If you order $50+ at their store, they ship for free.
2. If you order 10, you will likely receive 11 or even 12 boards. If you order 50, you probably get 51. Sometimes it's cheaper to order 10 PCS the same design from time to time than ordering 50 pcs, since you get 10%-20% more boards and only like 2%-4% more boards with 50 PCS 🙂

Share this: ☐ ☐ ☐ ☐ ☐ ☐

Like this: ☐

Be the first to like this.

FILED UNDER ARDUINO, EAGLE   TAGGED WITH PCB

6 Responses to *Ordering PCB from seeedstudio*

John-Ove Larsen says:
January 27, 2013 at 3:27 pm

Hi, I have just received some PCB boards from Seeedstudio but none of them had drill holes.
Are the manufacturer supposed to drill the holes? Also the solder pad for the component on the top layer and the bottom layer(same point only there is one on the top and one on the bottom layer) does not seem to have contact between them. Do you know if it is suppose to be like that?

John-Ove

Reply

> **liudr says:**
> January 27, 2013 at 5:39 pm
>
> John-Ove,
>
> Did you send them the .TXT file needed for drilling? It contains the coordinates and sizes of every drill hole. Without these holes, the top and bottom solder pads are not connected and you have a bunch of coasters. They have a list of required files (Drill Drawing: pcbname.TXT). I have that on my post too. If you are just missing the file in your online ordering, I really thing that seeedstudio should have caught that. Anyway, if your CAM is not generating this file, replace the CAM file with the most recent seeedstudio CAM. I have one I downloaded over 3 years ago and it works just fine.
>
> **Reply**
>
> > **John-Ove Larsen says:**
> > January 28, 2013 at 2:46 am
> >
> > Thank you for your answer. Yes I did send in the txt file created from the CAM file. I have checked my gerbers in ViewPlot. The drill holes as far as I understand are correct.

Follow

I am however quite new to designing PCB and still learning.

Do I have to change sizes of the components drill holes or are the components in Eagle correct?

I have sent in the files that were suppose to be there.

The only holes that had been drilled was the stand off holes.

John-Ove

liudr says:
January 28, 2013 at 8:12 am

John-Ove,

That's odd. If you open just the .TXT file in viewplot and you can see all the drill holes, then contact seeedstudio. They could have messed it up.

John-Ove Larsen says:
January 28, 2013 at 1:33 pm

Yes, I can see all the drill holes in the .txt file.
I will contact Seeedstudio and ask what might have gone wrong.

Thank you for your help.

John-Ove

liudr says:
January 28, 2013 at 1:34 pm

They are decent business so once they realize what went wrong (probably on their side), they can make it right.

Leave a Reply

Follow