Categories

- Free Gift
- Arduino
- Dev Platform->
- Shields
- Breakout
- Wireless->
- RFID / NFC
- RepRap 3D Printer
- Robot->
- Sensors->
- Power Supply
- USB to 232
- Flowerpad Series
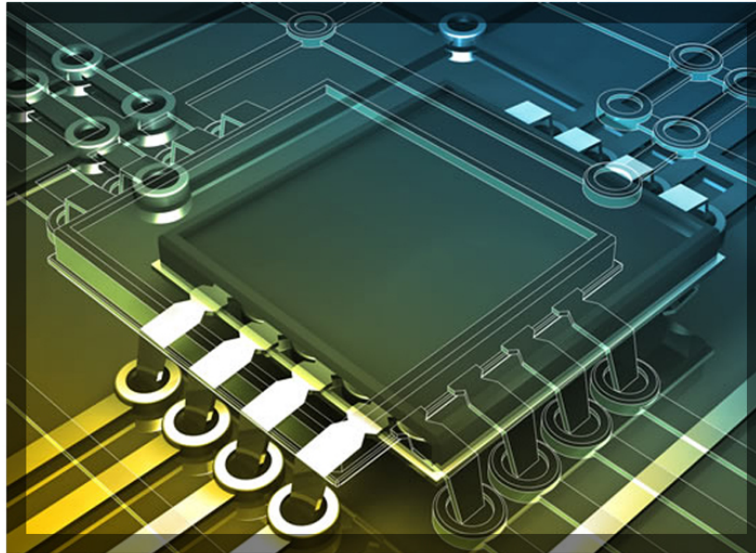- Octopus Brick
- Components->
- Prototyping->
- Kits
- LED->
- LCD->
- **PCB Service**
- Tools->

Specials …

2 Layer 10cm*10cm max -10pcs

larger image

Starting at: $9.90

- Model: PCB110816
- Shipping Weight: 100g
- 99826 Units in Stock

Contact Us   Add to WishList

Add to Cart:

Please Choose:

Qty.

New Products - more

Infrared Proximity
Sensor - Sharp
2Y0A710
$39.80

Infrared Distance
Sensor - Sharp
GP2D120
$22.50

Infrared Distance
Sensor - Sharp
GP2Y0A41SK0F
$17.00

10

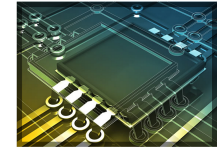|	
Layer	
PCB Thickness	
PCB Dimension	
PCB Color	
Surface Finish	
E-Test	100% E-test

2 Layer
10cm*10cm max -
10pcs

Description

You can chose 3 kinds of PCB dimension （5cm X 5cm Max,5cm X 10cm Max, 10cm X 10cm Max） in this service. We will provide you 10pcs PCBs. If this service can not meet your needs, you can chose '2 Layer 20cm*20cm max (5PCS or 10PCS optional)' or '2 Layer 20cm*20cm max-50pcs' service.The previous service provide more PCB dimensions and another one provide 50pcs PCBs. Please do check the price for different service and find the most economic one.

Fusion PCB Order Steps：

1. Purchase fusion PCB Service from online store and check out.
2. An order number will be provided in confirmation email from ElecFreaks services (**services@elecfreaks. com**)

3. Send the gerber file to pcb@elecfreaks.com. The gerber files must be packed into a zip or rar file and named Order (your order number)_(PCB size). For example: **Order100_10x10.rar.**

4. If the file and design meets requirements, they will be processed and shipped in 4-6 days. If the files have any issue **pcb@elecfreaks.com** will contact with you.

Gerber file requirement:(The following layers are needed)

Top layer:	pcbname.GTL
Bottom layer:	pcbname.GBL
Solder Stop Mask top:	pcbname.GTS
Solder Stop Mask Bottom:	pcbname.GBS
Silk Top:	pcbname.GTO
Silk Bottom:	pcbname.GBO
NC Drill:	pcbname.TXT

Note:

○ **Board outline must be included at least in one layer.**
○ **The Gerber file must be RS-274x format.**

PCB design tips：

○ SINGLE DESIGN, NO PANELIZING
○ Your Gerber file must include only one design
○ Drills line and long slot can't be accepted
○ Minimum slot 1mm*1mm
○ DRC compatible as rule file attached
○ The PCB size is smallest rectangle that can accommodate PC

Fusion PCB capabilities:

Multi Layers	1—2
PCB Material	FR-4
Available Color	Green, Red+$10, Yellow+10$, Blue+$10, White+$10
Silk Screen	White, Black (For White Solder Mask only)
Maximum Size	10cm X 10cm
Minimum Qty	10pcs
Board Thickness	0.8mm, 1.0mm, 1.2mm, 1.6mm, 2.0mm+$48
Thickness Tolerance (t≥0.8mm)	± 10%
Thickness Tolerance (t<0.8mm)	± 10%
Insulation Layer Thickness	0.075mm--5.00mm
Minimum Line	6mil (Recommend >8mil)
Minimum Space	6mil (Recommend >8mil)
Out Layer Copper Thickness	1oz(35um)
Inner Layer Copper Thickness	17um—100um
Drilling Hole (Mechanical)	0.3mm—6.35mm
Finish Hole (Mechanical)	0.3mm—6.30mm
Diameter Tolerance (Mechanical)	0.08mm
Registration (Mechanical)	0.09mm
Aspect Ratio	8:1
Solder Mask Type	Photosensitive ink
SMT min Solder Mask Width	0.1mm
Min Solder Mask Clearance	0.1mm
Plug Hole Diameter	0.25—0.60mm
Surface Finish	HASL, HASL (Lead Free) +$5, ENIG+$10.

E-TEST:

50% e-test, 100% e-test +$10

50% e-test (Tested PCB will have a mark on the PCB edge).
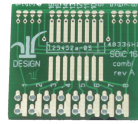
100% e-test optional for extra 10$

Down load file:

- Eagle Design Rule
- Eagle CAM file

Write a review ›

This product was added to our catalog on Tuesday 16 August, 2011.

Customers who bought this product also purchased...

Aplomb-boards SOIC8 adapters   Aplomb-boards SOIC16 adapters   Long Post Shunt for use with 2.54mm pitch (10pcs pack) - Yellow

125*40mm 5V125mA Solar Panels   2.0mm 40Pin Female Header (5 Pcs)   RFID/NFC Breakout Board