designwest

center of the engineering universe

PCB Design in 45 Minutes

Matt Liberty
Jetperch LLC

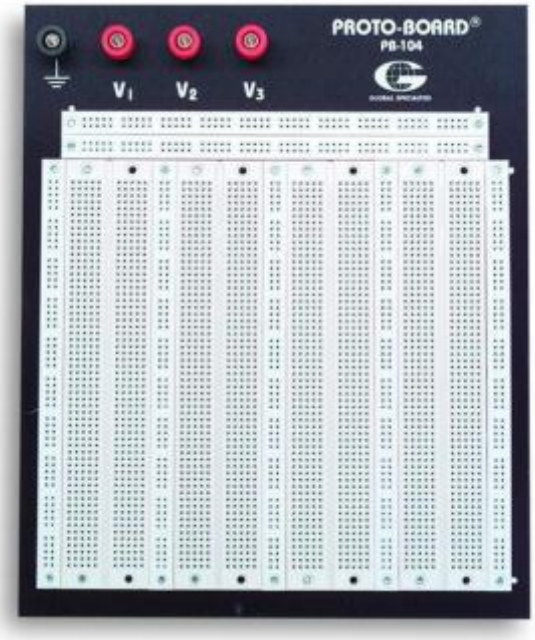
April 22-25, 2013
McEnery Convention Center
San Jose, CA
www.ubmdesign.com

UBM Tech

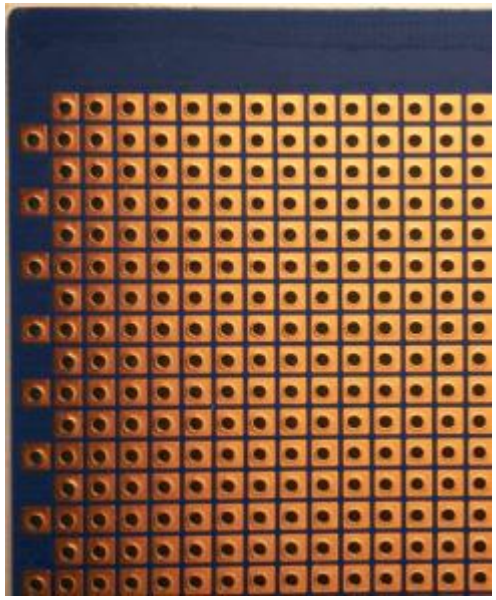
Motivation

- Time to market and the development window are decreasing
  - Want reliable prototype hardware early in the development cycle
  - Want to replicate the development platform across a team
  - Asked to do more with less
- Common solution
  - Combine development boards and reference platforms to emulate final product
  - But how? How to do better?

Methods

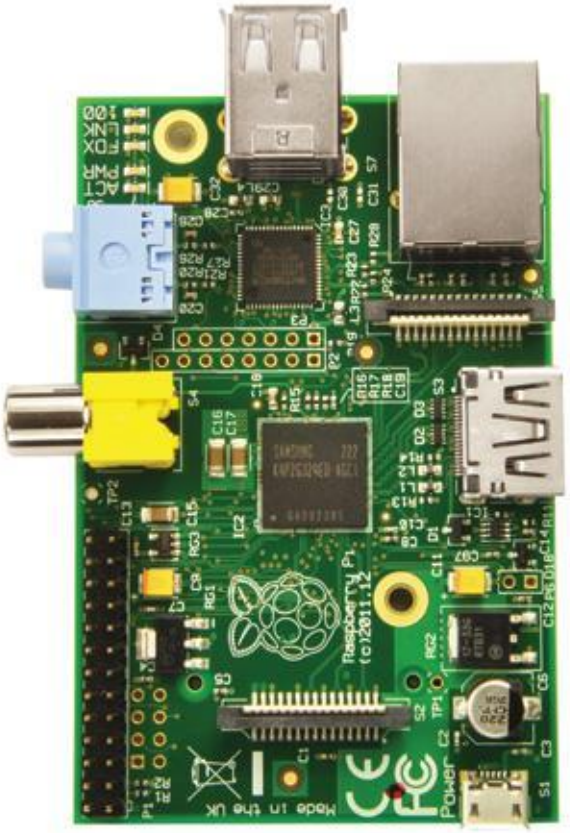


Breadboard



Perfboard

http://commons.wikimedia.org/wiki/File:CopperCladPerfboard_1.png



Printed Circuit Board

What is a PCB?

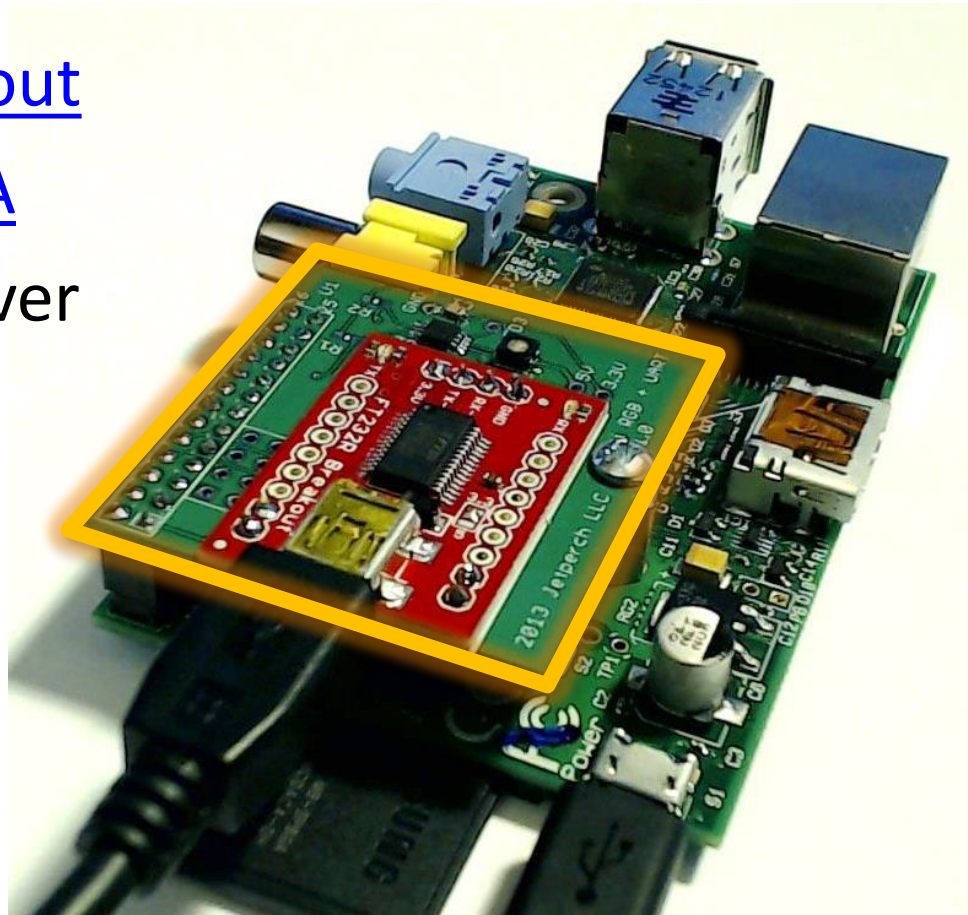
- Provides mechanical stability to form a single physical unit
  - Hold components together
  - Mounting holes & features integrate into the larger product
- Provides electrical connectivity between components
  - Wires between components
  - Multiple layers to allow wires to cross
  - Vias (holes) to connect wires across layers

PCB Design Steps

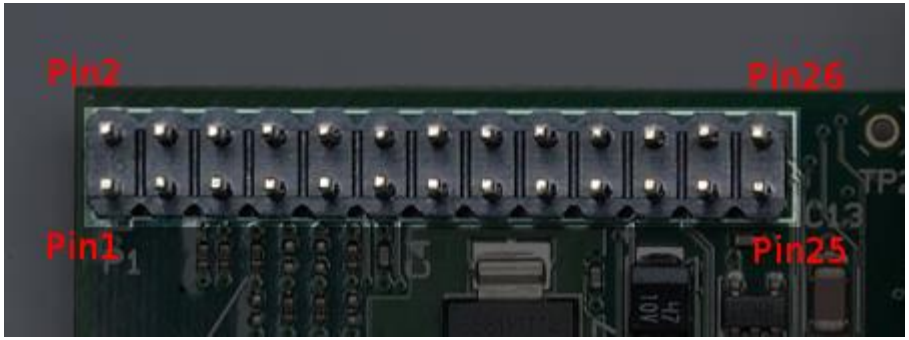
- Features & requirements
- Architecture & component selection
- Schematics: A complete, definitive graphical description for how all components are logically connected
- Layout: A complete, definitive graphical description for how all components are physically connected
- Fabricate Printed Circuit Board (PCB)
- Assemble Printed Circuit Assembly (PCA)
- Test

Example

- Objective: add USB/UART and RGB LED to Raspberry Pi
- Raspberry Pi 2x13 0.1” interface connector
- SparkFun FT232RL Breakout
- RGB LED: Cree CLVBA-FKA
- I²C RGB PCA9632 LED driver



Raspberry Pi

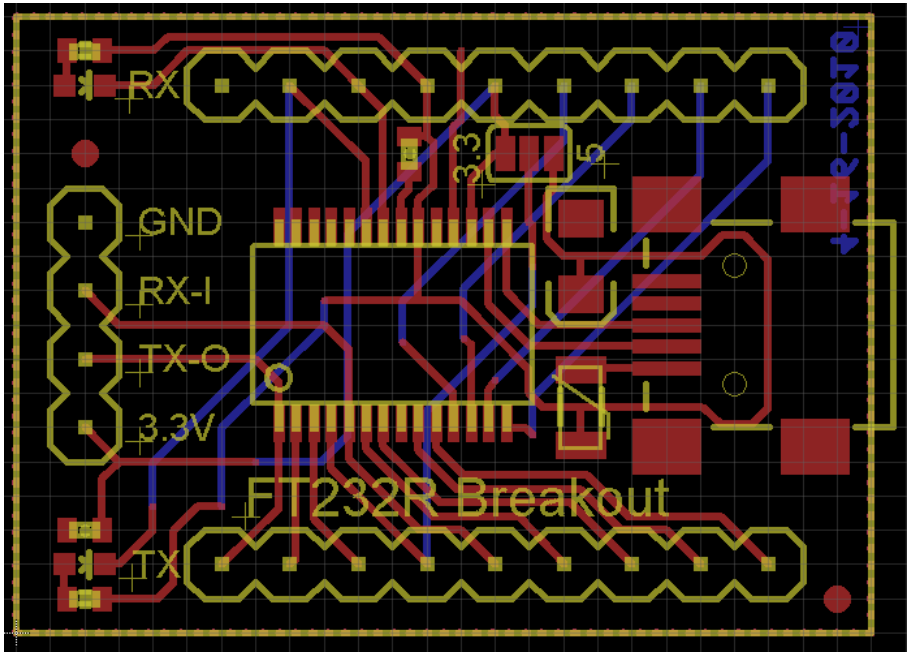


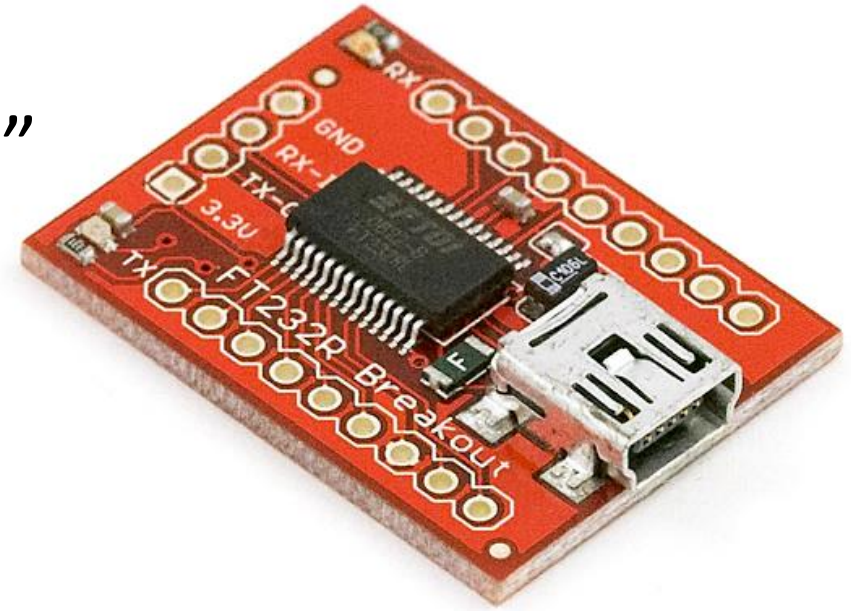
Source: Embedded Linux Wiki
http://elinux.org/RPi_Low-level_peripherals

Symbol & Footprint:
http://www.raspberrypi.org/phpBB3/viewtopic.php?f=9&t=4457

FT232RL Board

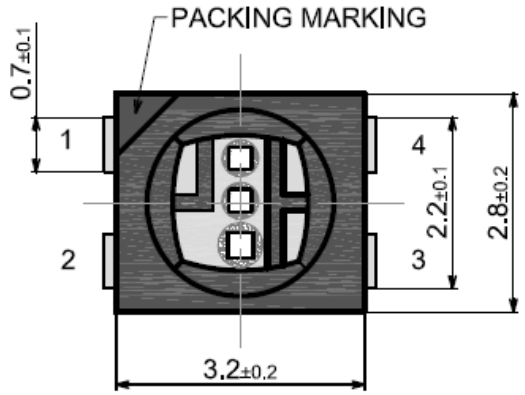
- 1.25” x 0.9”
- Connector at 0.1” x 0.45”

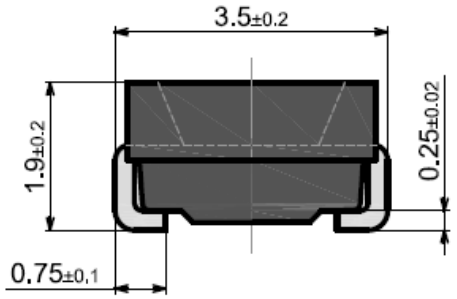


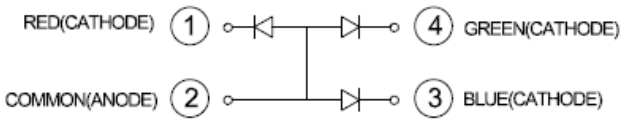


Source: SparkFun
https://www.sparkfun.com/products/718

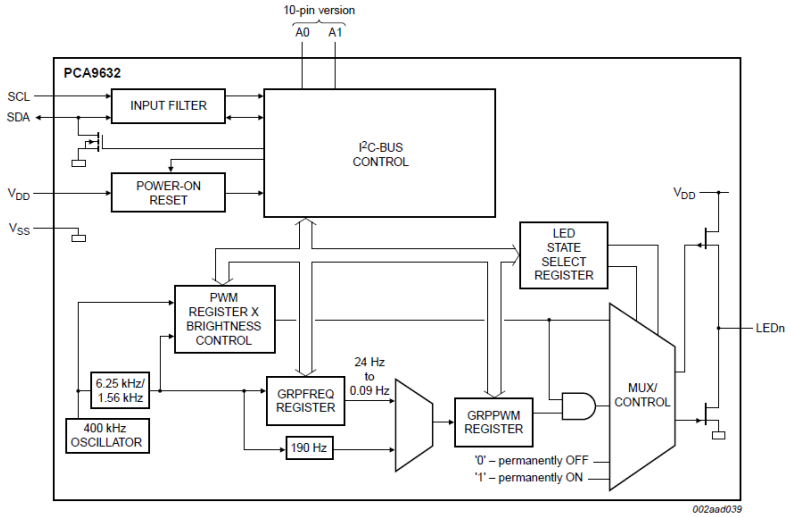
RGB LED and Controller

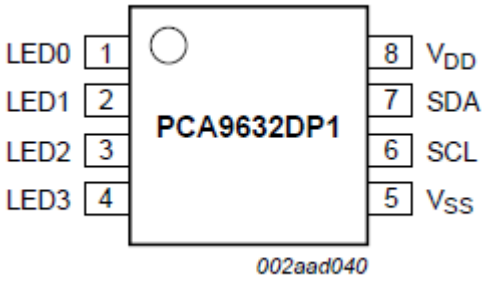


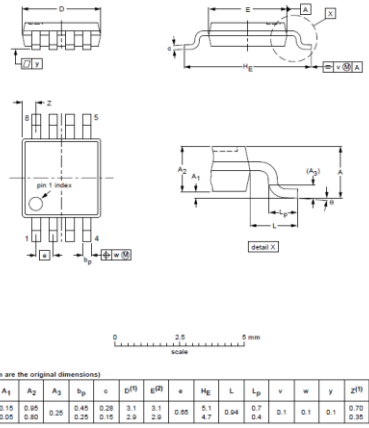




Source: [Cree CLVBA-FKA Datasheet]





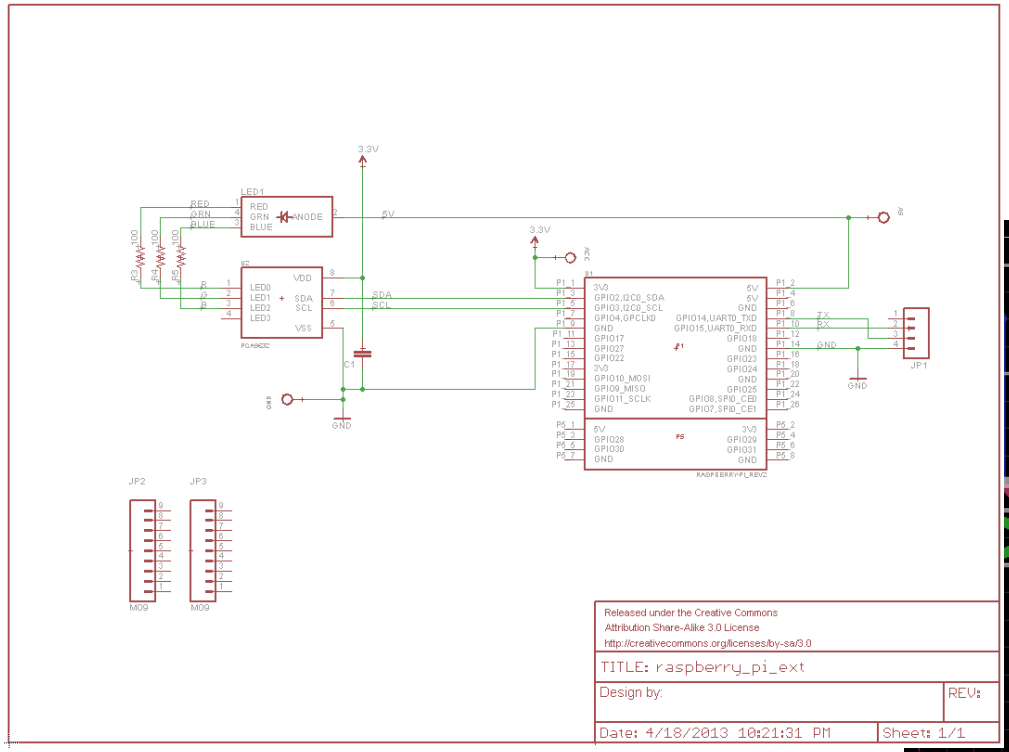


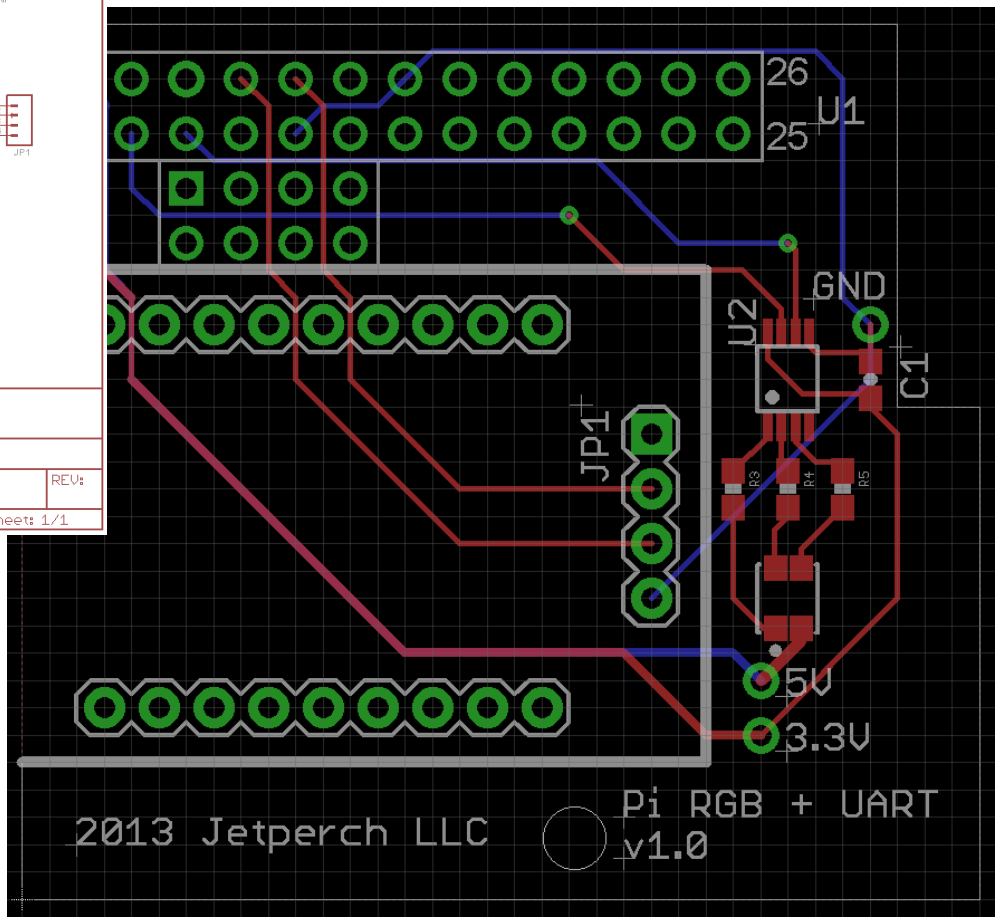
Source: [NXP PCA9632 Datasheet]

Tools

- Eagle PCB
  - Schematic capture
  - PCB Layout
  - CAM Generator
  - Sparkfun Eagle library
- ViewPlot (Gerber Viewer)
- Online DRC
  - BatchPCB, Advanced Circuits

Demonstration





Key Tips

- Be aware of grid (ALT key) and layers
- Provide options: solder jumpers & vias
- Double check footprints for all components
- Have someone else review the schematics and layout whenever possible
- Check your Gerbers with a separate gerber viewer (Drill file is 2-4)
- When soldering, keep tip clean and use plenty of flux

Suitable Projects

- Slow edge rates on interconnects
  - Low-speed signals (< 25 MHz)
  - I^2C, SPI, UART, GPIO, full-speed USB
- Few components and low signal density
  - Size is not critical
- No analog or < 10 bit resolution
- Simple components only
  - No BGAs
  - No fine-pitch parts

Next Steps

- Download this presentation from jetperch.com
- Explore reference resources
- Download & install tools
- Design your own PCB
  - Allocate 15 to 30 hours for first PCB
  - Tools & supplies: $200
  - PCB: $100 - $200 (1-week) or $20 (4-week)
  - Components: $50 but varies by design

Alternative: Discuss options of using rapid, low-cost PCB prototyping with your hardware engineers

designwest

center of the engineering universe

PCB Design in 45 Minutes

Matt Liberty
Jetperch LLC

April 22-25, 2013
McEnery Convention Center
San Jose, CA
www.ubmdesign.com

UBM Tech

Abstract

- Creating a printed circuit board (PCB) has become faster, cheaper and easier over the last several years. Both software and hardware engineers can now add PCB design to their toolbox to create more reliable and repeatable development hardware. This session walks through the full design process for a 2-layer interconnect PCB using EaglePCB, a schematic capture and layout tool free for non-commercial use. This session includes additional self-study references to demystify the tools, terminology and processes behind PCB design and assembly.
- Key Takeaways: Attendees walk through the PCB design process starting from requirements through schematics, layout, fabrication, assembly and test. Attendees also gain familiarity with an example schematic capture and layout tool.

Development Options

Option	Advantages	Disadvantages
Wires (soldered or with clips)	• Fastest to create	• Mechanically and electrically unreliable • Cannot easily add components
Breadboard	• Fast to create	• Reliability depends upon craftsmanship • Time-consuming to replicate
Perfboard	• Fast to create	• Reliability depends upon craftsmanship • Time-consuming to replicate
Printed Circuit Board (PCB)	• Easy to replicate • Mechanically & electrically reliable • Easy to add resistors, LEDs & components	• Slower to create • Requires more planning • Requires process and tools knowledge

PCB Layout

- Fabrication (fab) drawing
- Board outline & routing
  - PCB layer stack-up
  - Manufacturing process and options
- Layers
  - Metal (top, bottom & inner layers) [**GTL**, **GBL**]
  - Solder mask (top, bottom) [**GTS**, **GBS**]
  - Solder paste (top, bottom) [GTP, GBP] Used for printed circuit assembly
  - Silk screen (top, bottom) [**GTO**, GBO]
- Drill
  - Drawing
  - Drill size file [**TXT**]
- Panelization
  - Group PCBs into assembly panels (roughly 12” x 12”)
  - Group assembly panels into fabrication panels (roughly 2’ x 3’)
- Assembly drawing

Only items in red are required for low-volume prototyping

PCB Tutorials

- ITP Physical Computing Eagle PCB Tutorial
- PCB Design Tutorial (PDF)
- Eagle PCB Guided Tour - Cadsoft
- Eagle Schematic Tutorial - Sparkfun
- Eagle Layout Tutorial - Sparkfun
- Villanova Electronics Inventors Club PCB Tutorial
- How to Build a PCB – Advanced Circuits

Fabricate & Assemble

- Prototype PCB Fab Houses
  - Advanced PCB – Quickturn prototype PCBs
  - BatchPCB – Slow, inexpensive prototype PCBs
- Prototype Assembly Houses
  - Advanced Assembly
  - Screaming Circuits
- Self-Assembly
  - Soldering Iron: Weller WESD51 ($129)
  - Supplies ($50)

Soldering Tutorials

- How to Solder Video – Curious Inventor
- Soldering Tutorial Guide – Principia Labs
- Soldering Basics – Sparkfun
- Surface Mount Soldering – Sparkfun
- Soldering 101 – Indium

Recommendation: Learn using tin/lead solder, not lead-free.

Soldering Supplies List

Item	Manufacturer	Part #
Soldering iron	Weller	WESD51
Soldering iron fine tip	Weller	578-ETS
Solder (0.032” 22 gauge)	MG Chemicals	4890-18G
Solder (0.02”)	Chip Quik	SMDSW.020 2oz
Soldering iron tip cleaning wire sponge	CML Supply	Amazon
Flux pen	MG Chemicals	835-P
Flux solvent	N/A	Isopropyl alcohol
Desoldering wick	MG Chemicals	425-NS
Calipers	Neiko	01407A

Component Suppliers

- Small volume, prototyping & hobbyist
  - Digikey – The leader for professional prototyping & small volume
  - Mouser – Easier to navigate than Digikey, but less selection
  - Samtec (connectors)
  - iFixIt – do it yourself electronics repair parts
  - Sparkfun – Hobby market
  - SmallParts
  - Radio Shack – Yes, they still sell electronic components
  - Jameco
- Mass production
  - Avnet – The global leader in electronic component distribution
  - Arrow Electronics
  - Future Electronics
  - Newark
  - McMaster-Carr – Mechanical parts, fast

PCB Examples

- Sparkfun
  - Almost all designs have accompanying Eagle schematic and layout
  - You could design this UART to USB converter
- Cadsoft: User Projects

Available PCB Tools

- Eagle PCB
  - $49 2 signal layers 100x80 mm area
  - $747 6 signal layers 160x100 mm area
  - $1404 16 signal layers 1600x1600 mm area
- Altium Designer ($5000)
- KiCad (open source & free)
- Mentor Graphics (PADS)
- Cadence
  - Allegro
  - OrCAD
- gEDA (open source & free)
- upverter.com (Free for open source projects, $7 / mo)

See the open source list at http://www.opencircuits.com/Software_tool

Other Resources

- CircuitHub – Universal, free part library
- BoardForge – Product in development that fabs and assembles 2-layer PCBs at your desk
- Microbuilder – a good EaglePCB library

Using this board: sh

```
sudo apt-get install python-smbus
sudo -s
i2cget -y 1 0x62 0
i2cset -y 1 0x62 0 0x01
i2cset -y 1 0x62 8 0xaa
# All on: RGB
i2cset -y 1 0x62 0xa2 0xff 0xff 0xff I
# All off: RGB
i2cset -y 1 0x62 0xa2 0 0 0 I
exit
```

Using this board: Python

```
# RGB demo
import smbus
import time
_ADDR = 0x62

class RGB(object):
    def __init__(self, i2c_bus_number, i2c_addr):
        self._i2c_bus = i2c_bus_number
        self._addr = i2c_addr
        self._bus = smbus.SMBus(self._i2c_bus)
        self._bus.write_byte_data(self._addr, 0, 0x01)
        self._bus.write_byte_data(self._addr, 0, 0xaa)
        self._color = [0.0, 0.0, 0.0]

    @property
    def color(self):
        return tuple(self._color)

    @color.setter
    def color(self, *args):
        if len(args[0]) != 3:
            raise ValueError('Color must be RGB from 0.0 to 1.0')
        self._color = list(args[0])
        x = [int(y*255.0) for y in self._color]
        self._bus.write_i2c_block_data(self._addr, 0xa2, x)

    def fade(self, color, duration):
        step = 0.01
        iter = int(duration / step)
        d = [color[0] - self._color[0], color[1] - self._color[1], color[2] - self._color[2]]
        d = [x / iter for x in d]
        for i in range(iter):
            self.color = [self._color[0] + d[0], self._color[1] + d[1], self._color[2] + d[2]]
            time.sleep(step)
        self.color = color


rgb = RGB(1, _ADDR)
for i in range(10):
    rgb.fade((1.0, 0.0, 0.0), 0.5)
    rgb.fade((0.0, 1.0, 0.0), 0.5)
    rgb.fade((0.0, 0.0, 1.0), 0.5)
rgb.fade((0.0, 0.0, 0.0), 0.5)
```