

About Me

- Graduated from NUS Computer Science in 2015



- Engineer at med-tech startup Algo Access

What is a PCB?

- **Printed Circuit Board**

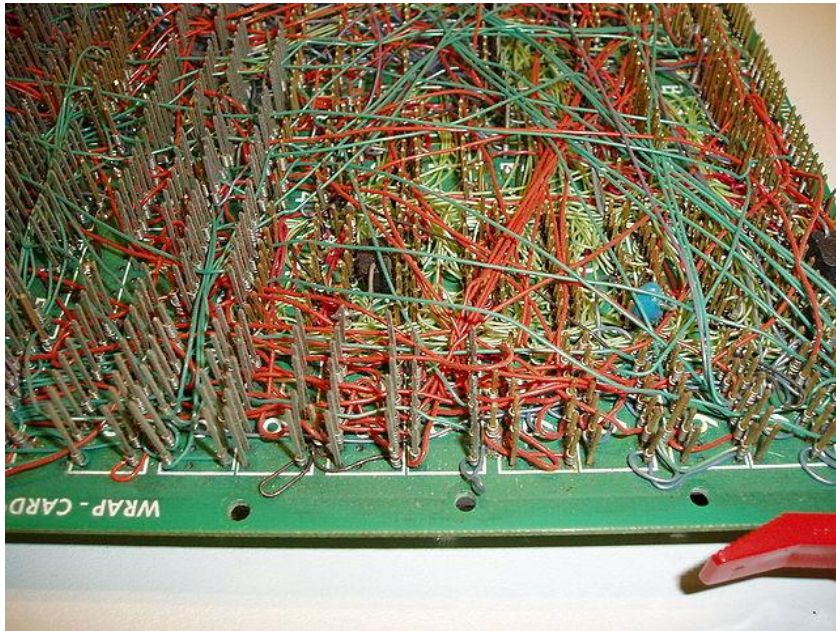
- A printed circuit board (PCB) mechanically supports and electrically connects electronic components using conductive tracks, pads and other features etched from copper sheets laminated onto a non-conductive substrate.

- Wikipedia

- **Layman definition:**

- A board that lets you attach components to make a circuit.

Before the PCB



Point to point wire wrapping

vs



Modern PCB of Novena laptop

Source: <https://cdn.sparkfun.com/r/700-700/assets/1/3/b/5/8/50cba0dcce395fb716000000.jpg>

Source: http://bunniefoo.com/novena/pvt2_release/novena_pvt2_top_sm.jpg

Motivation for makers?

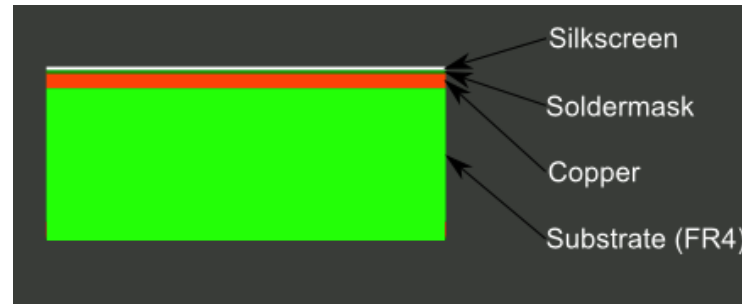
- Space and weight constraints
- Making multiple copies of a design
 - Reduce mistakes in manual wiring/soldering
- Aesthetics
- Durability

Objective of lesson

Fundamental knowledge to

- Understand PCB concepts
- Design a PCB
- Send for manufacturing
- Properly share your design with others
- Efficient SMD assembling techniques

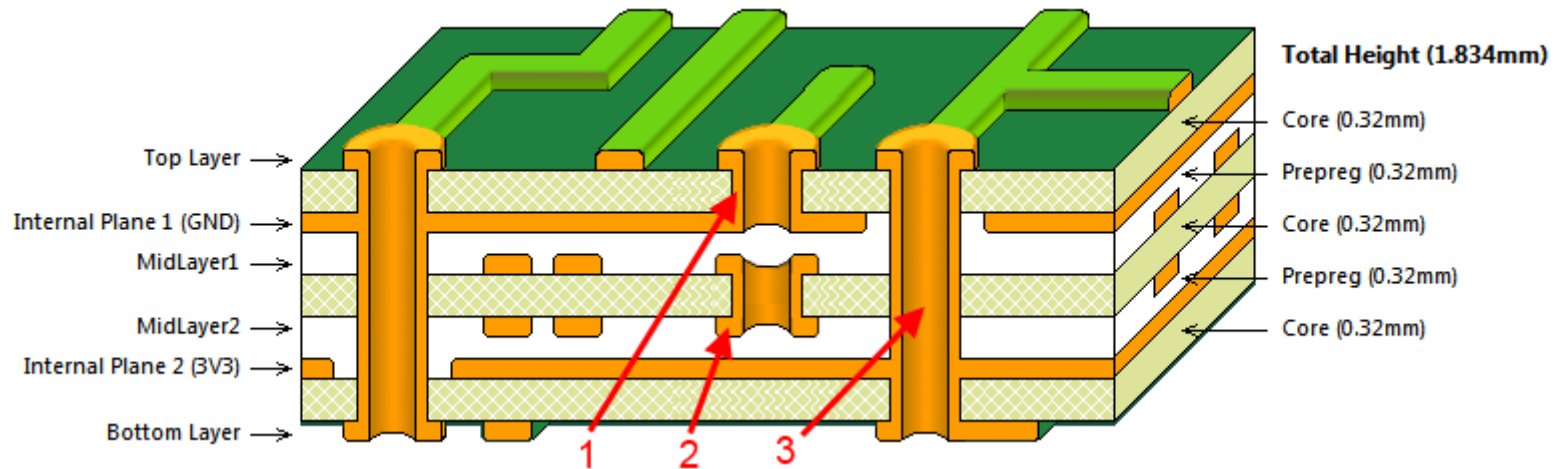
Structure of a 1-layer PCB board



Modified from: <https://cdn.sparkfun.com/assets/3/f/c/b/c/50d0c95bce395fd321000000.png>

- Usually 1.6mm thick – applies to 2-4 layers too
- Silkscreen (usually white):
 - Ink text. Usually used to mark out components
- Soldermask (usually green):
 - Prevent solder (paste) from flowing out of designated areas forming *solder bridges*
 - Made of lacquer polymer that is “solderphobic”
 - Protects copper layer from oxidation
- Copper:
 - Contains the traces (wires) between components
 - Also contains a ground/power plane (to be explained later)
- Substrate
 - Gives the PCB mechanical strength and rigidity
 - Usually made of FR-4, composite of fibreglass and epoxy resin

Structure of a (≥ 2) multi-layer PCB board



Source: [http://techdocs.altium.com/display/ADRR/PCB_Obj-Via\(\(Via\)\)_AD](http://techdocs.altium.com/display/ADRR/PCB_Obj-Via((Via))_AD)

- 2-layer board = 2 copper layers, 4-layer board = 4 copper layers, etc
- 2-layer boards good enough for most use-cases
- Silkscreen and soldermask on top and bottom only
- Vias
 - Hole in PCB that electrically connects points between copper layers
 - 1. Blind (only for >2 layer board)
 - 2. Buried (only for >2 layer board)
 - 3. Through-hole

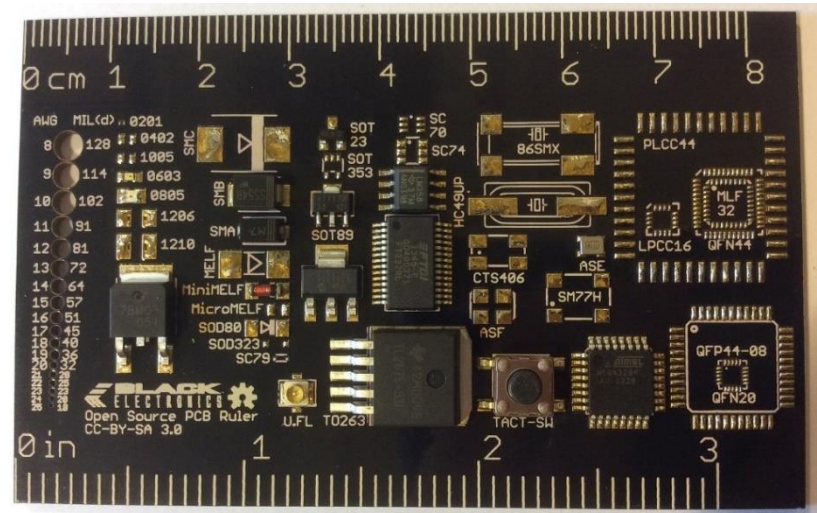
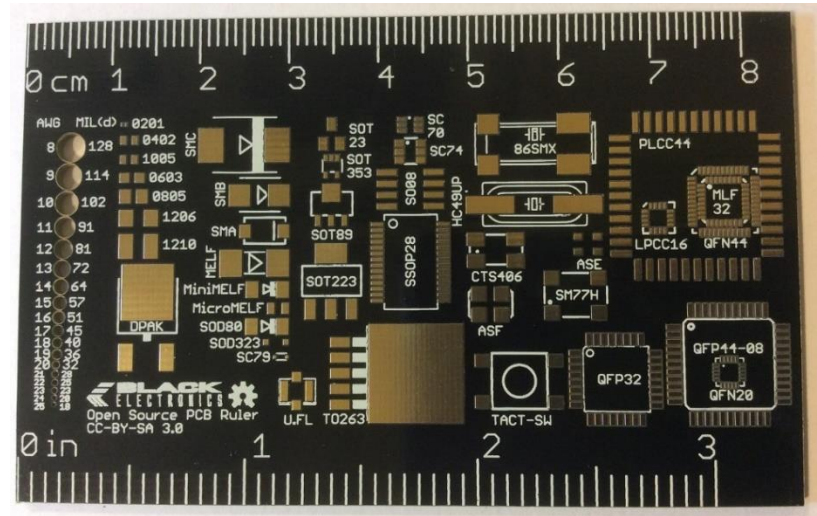
Electronic Design Automation (EDA) Software

- Software that helps you design PCBs
- Eagle
 - Commonly used by makers + Adafruit + Sparkfun
 - Extensive library support
 - User-interface so-so
 - “Lite” version supports max 2 copper layers, 100 x 80 mm board size
 - Not the best for high-speed designs
 - Multi-platform
 - Saved as XML (ASCII) files, great for version-control
- Altium Designer
 - Industry standard
 - De-facto option for high-speed designs and complex PCBs
 - Circuit simulation
 - User-interface is unrivaled IMO
 - File saved as binary format, terrible for version-control
 - Windows-only
 - Expensive: US\$10K/license
- KiCad
 - Recently becoming more popular
 - Free & open-source with no limitations
 - Libraries not as common
 - Saves in ASCII files, great for version-control
 - Multi-platform

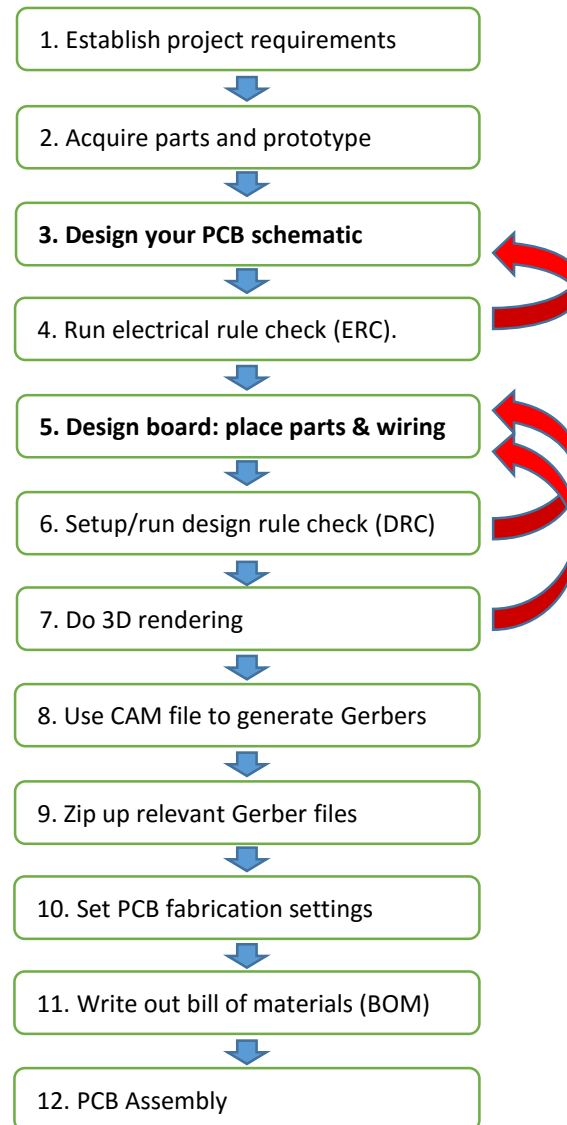


Component packages

- 2D physical dimension of a component
- Determines number of solder pads/sizes
- **Common surface-mount packages**
- 2-pin passives (don't rely on external power)
 - 0603 (0.063 inch × 0.031 inch), 0805 (0.079 in × 0.049 in), 1206 (0.126 inch × 0.063 inch), ...
 - SOD-123 (Small outline diode)
 - MELF (Metal electrode leadless face) cylindrical SMT component
- ≥ 3 pins Active Components (rely on external power)
 - SOT23, SOT-223-4 (Small Outline Transistor: 2.2mm pitch, 3 pins front, 4 pins total), ...
- Integrated circuits (ICs), usually dual-inline or square
 - SOIC-16 (Small Outline Integrated Circuit 16 pins)
 - SSOP-28 (Shrink small outline package 28 pins)
 - TQFP-32 (Thin Quad Flat Package 32 pins)
 - QFN-48 (Quad Flat No leads 48 pins)
- Others
 - Ports like USB, audio,
 - Board to board connectors like DF40C(2.0)-70DS-0.4V
 - Pin headers
 - uFL antenna connector

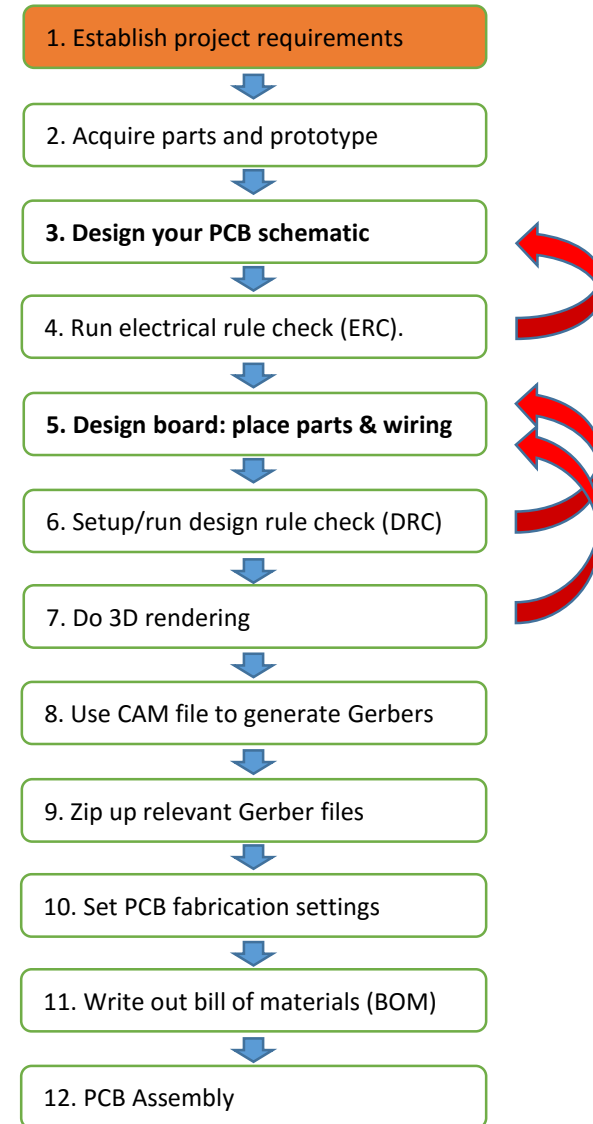


PCB Design/Manufacturing Workflow



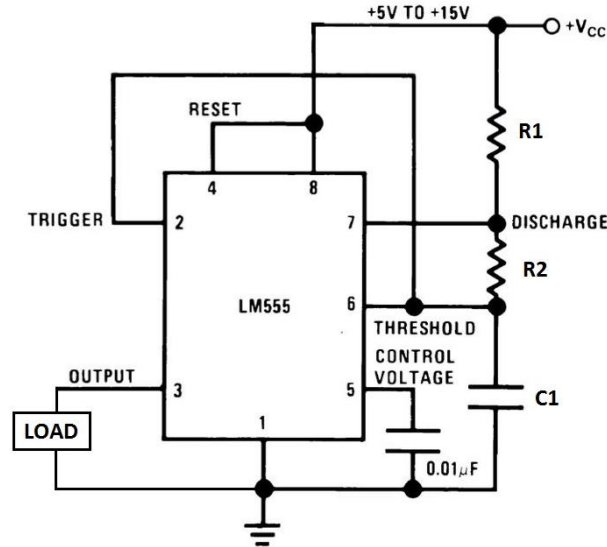
1. Establish project requirements

- Circuit that blinks an LED
- Customisable blink period
- Customisable blink cycle
- Run on CR2032 coin cells
- Can run off USB power
- Power switch

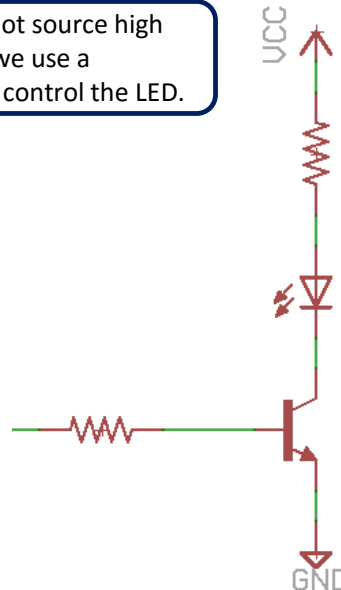


2. Acquire parts and prototype

- **Consult data sheets to understand part selection choices**
- **Logic Components**
 - Texas Instruments TLC555 timer
 - Switch (C1 selection)
 - C1: 33uF/100uF capacitor
 - R1: 4.7kohm resistor
 - R2: 10k potentiometer
 - 2x 0.1uF decouple capacitors
 - Required according to TLC555 data sheet
- **Output components**
 - 5mm 20mA Red LED
 - 100ohm current limiting resistor for LED
 - ON Semiconductor P2N2222A NPN Transistor
 - 1k ohm base resistor
- **Power components**
 - 2x CR2032 battery holders
 - Micro-USB female connector
 - 3.3V low-dropout TC1262 regulator
 - Diodes for battery protection
 - 1uF output decouple capacitor
 - Required according to TC1262 datasheet
 - Switch to ON/OFF circuit



TLC555 cannot source high currents so we use a transistor to control the LED.



1. Establish project requirements
2. Acquire parts and prototype
3. Design your PCB schematic
4. Run electrical rule check (ERC).
5. Design board: place parts & wiring
6. Setup/run design rule check (DRC)
7. Do 3D rendering
8. Use CAM file to generate Gerbers
9. Zip up relevant Gerber files
10. Set PCB fabrication settings
11. Write out bill of materials (BOM)
12. PCB Assembly

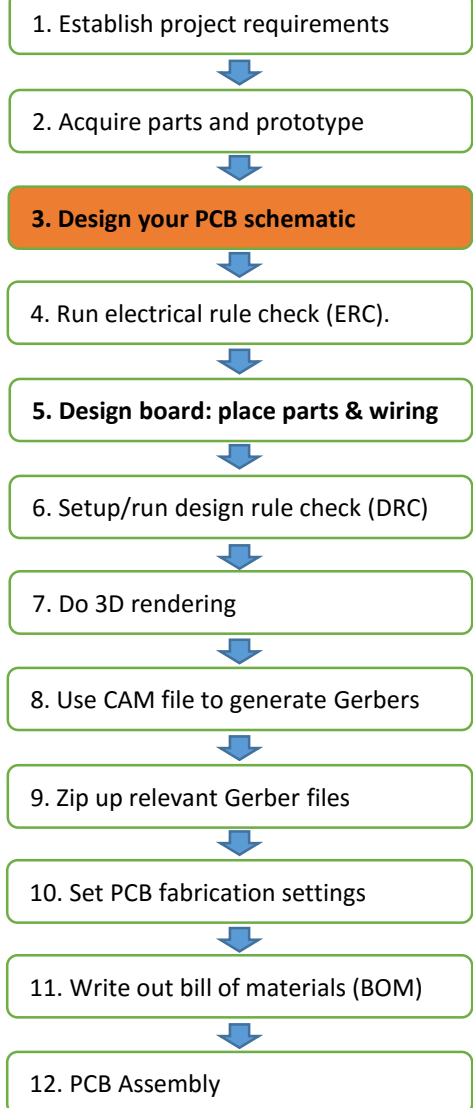
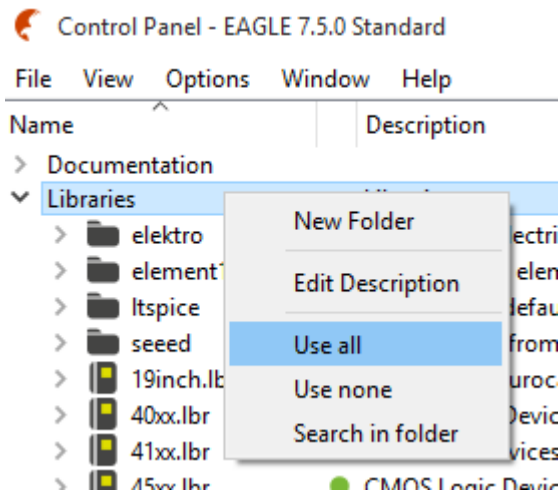
3a. Design your PCB Schematic

- Schematic defines connections between components

1. Open Eagle and add external libraries

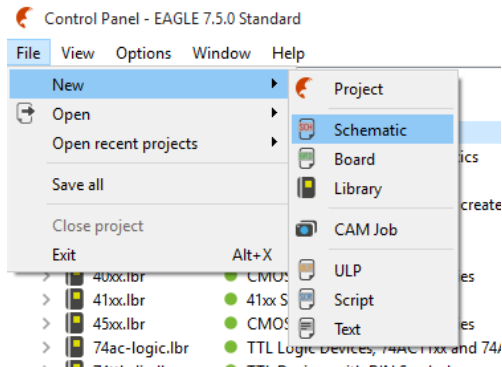
2. Import libraries.

- Locate the libraries folder I have provided
- Drag-and-drop the *.lbr files **one by one** into Eagle Control Panel Libraries folder
- Right-Click on Libraries Folder -> Use all

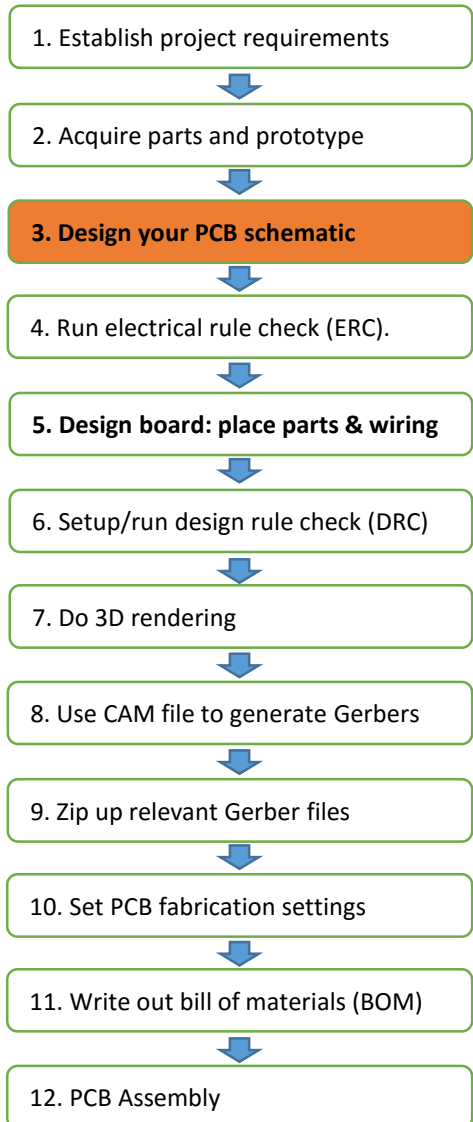
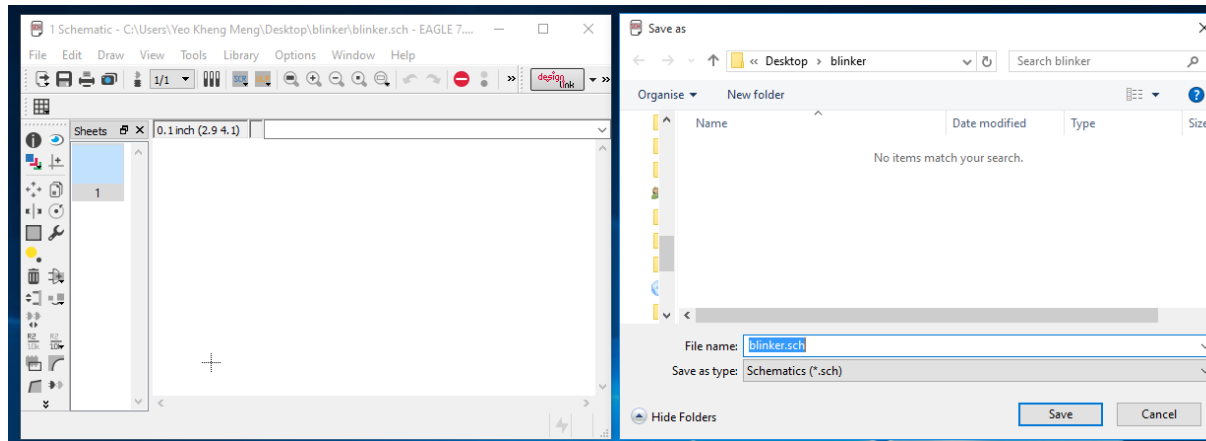


3b. Design your PCB Schematic

- Create and save schematic file (part and connection layout)
- Create a folder on desktop “blinker”
- Create new schematic *.sch: File -> New -> Schematic

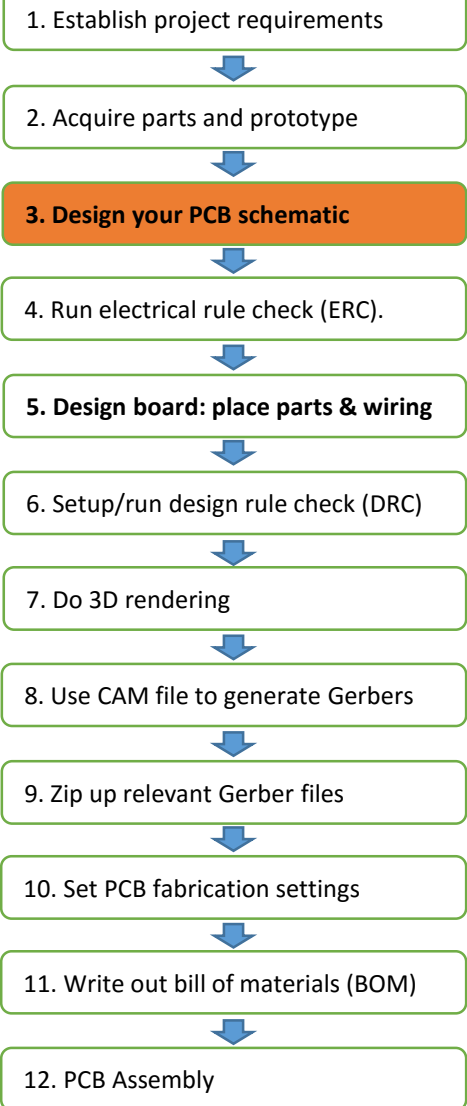
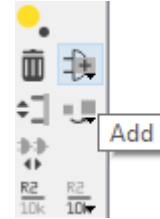


- Save schematic



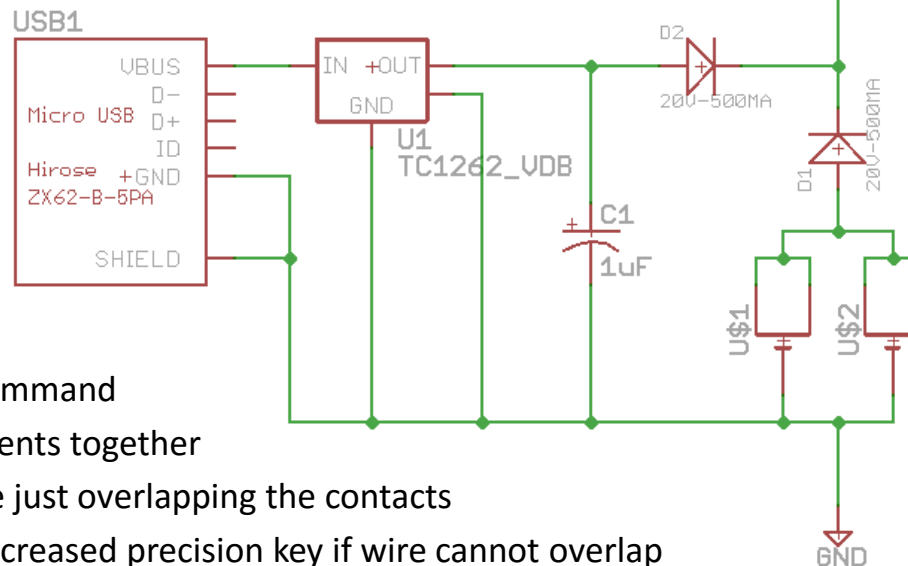
3c. Design your PCB Schematic

- Add all Power components first
- Micro-USB female connector
 - Seeed-OPL-Connector -> MICRO-USB-SMD(ZX62-B-5PA)
 - Or search 320010005 obtained via Seeedstudio OPL website
 - http://www.seeedstudio.com/depot/OPLopen-parts-library-catalog-c-136_138/
- 3.3V LDO
 - bt_regulator -> TC1262 -> SOT-223-3
- 1uF decoupling resistor
 - rcl- > CPOL-US -> CPOL-USUD-4X5,8
- 2x Diodes
 - Seeed-OPL-Diode -> SMD-DIODE-SCHOTTKY-20V-500MA(SOD-123)
 - Or search 304020028
- 2x CR2032 battery holders
 - cr2032-large-side -> CR2032SMT
- Through-hole switch
 - Sparkfun-Electromechanical -> SWITCH_SPDT -> SWITCH-SPDTPTH
- VCC and GND wiring net
 - Just search for them...
 - There are 2 functionally identical supply1 and supply2 options for VCC and GND, just pick any one you prefer.

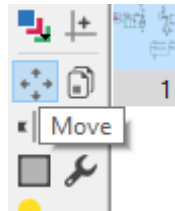


3d. Design your PCB Schematic

- Maneuvering and layout
- Zoom with scroll-wheel
- Click-drag middle mouse button to move sheet
- Rotate components
 - Choose Move command
 - Click component “+” sign then right-click
- Define component value if needed
 - Right-click component -> Value
- Delete object with trash icon
- Wiring



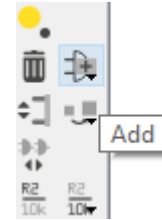
- Use “Net” command
- Link components together
- Hover mouse just overlapping the contacts
- Press “Alt” increased precision key if wire cannot overlap
- Left-click to set and also to change direction



1. Establish project requirements
2. Acquire parts and prototype
- 3. Design your PCB schematic**
4. Run electrical rule check (ERC).
5. Design board: place parts & wiring
6. Setup/run design rule check (DRC)
7. Do 3D rendering
8. Use CAM file to generate Gerbers
9. Zip up relevant Gerber files
10. Set PCB fabrication settings
11. Write out bill of materials (BOM)
12. PCB Assembly



3e. Design your PCB Schematic



1. Establish project requirements



2. Acquire parts and prototype



3. Design your PCB schematic



4. Run electrical rule check (ERC).



5. Design board: place parts & wiring



6. Setup/run design rule check (DRC)



7. Do 3D rendering



8. Use CAM file to generate Gerbers



9. Zip up relevant Gerber files



10. Set PCB fabrication settings



11. Write out bill of materials (BOM)



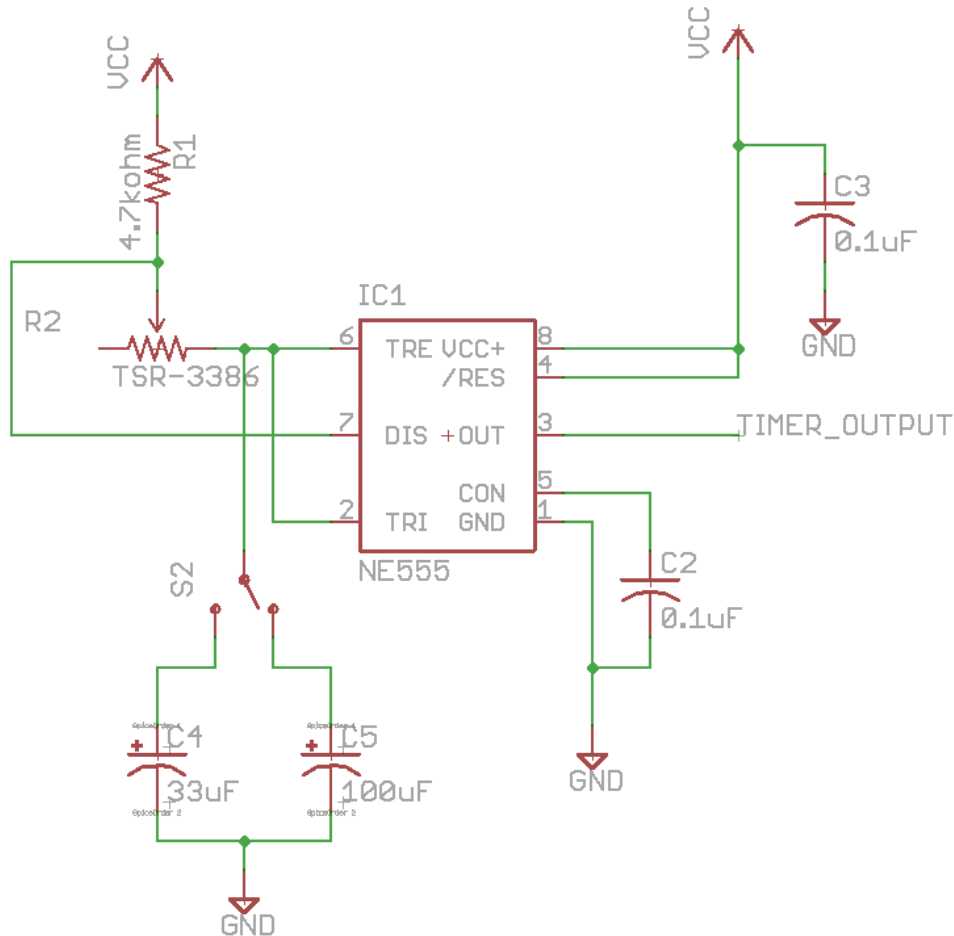
12. PCB Assembly



- Add and wire Logic Components
- Ignore TIMER_OUTPUT
- TLC555 timer (dual in-line package)
 - st-microelectronics -> NE555
 - Not exact component but good enough
- 4.7kohm axial resistor
 - resistor -> R-US_ -> R-US_0207/10
 - 2mm diameter, 7mm length, 10mm lead length
- 10kohm potentiometer
 - Sparkfun-Electromechanical -> TRIMPOT -> TRIMPOT-PTH-KNOB
- Through-hole switch
 - Sparkfun-Electromechanical -> SWITCH_SPDT -> SWITCH-SPDTPTH
- 33uF and 100uF aluminum electrolytic "can" capacitors
 - resistor -> CPOL-US -> CPOL-USE2-5
 - 2mm lead spacing, 5mm diameter, polarised
- 2x 0.1uF ceramic decoupling capacitors
 - resistor -> C-US -> C-US025-025X050
 - 2.5mm x 5mm area
- VCC and GND
 - Just search for them...

3f. Design your PCB Schematic

- Expected result of Logic circuitry
- Ignore “TIMER_OUTPUT” for now

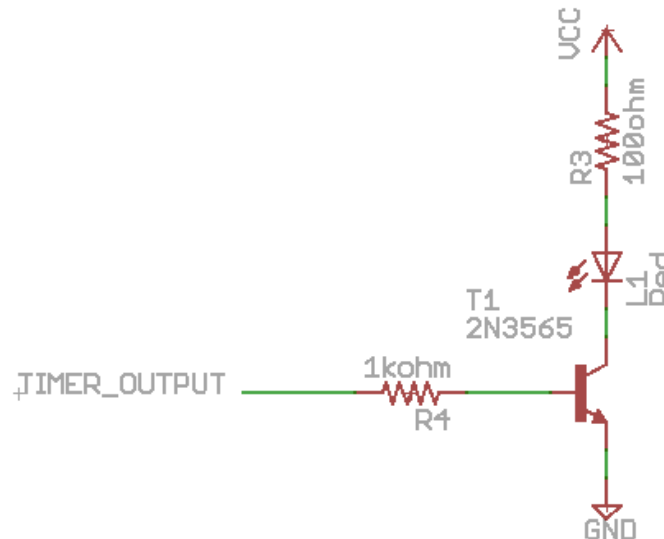


1. Establish project requirements
2. Acquire parts and prototype
- 3. Design your PCB schematic**
4. Run electrical rule check (ERC).
5. Design board: place parts & wiring
6. Setup/run design rule check (DRC)
7. Do 3D rendering
8. Use CAM file to generate Gerbers
9. Zip up relevant Gerber files
10. Set PCB fabrication settings
11. Write out bill of materials (BOM)
12. PCB Assembly



3g. Design your PCB Schematic

- Add and wire Output Components
- Ignore “TIMER_OUTPUT” for now
- 100ohm and 1kohm axial resistor
 - resistor -> R-US_ -> R-US_0207/10
 - 2mm diameter, 7mm length, 10mm lead length
- 5mm Red LED
 - led -> LED -> LED5MM
- ON Semiconductor P2N2222A NPN Transistor (TO92 package)
 - transistor -> 2N3565
 - Not exact component but good enough
- VCC and GND
 - Just search for them...

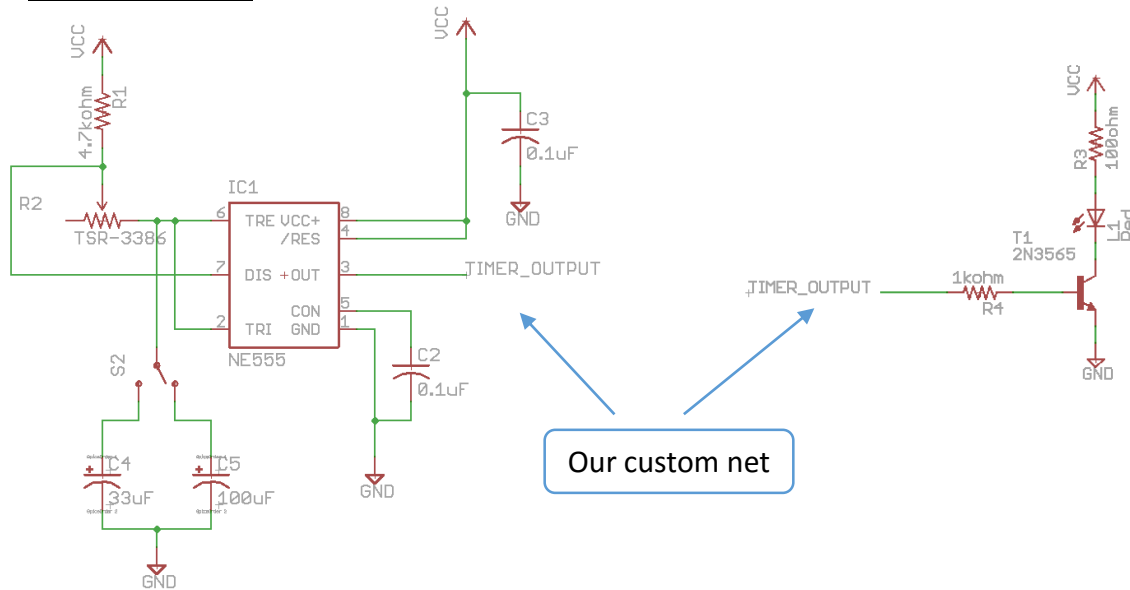


1. Establish project requirements
2. Acquire parts and prototype
- 3. Design your PCB schematic**
4. Run electrical rule check (ERC).
5. Design board: place parts & wiring
6. Setup/run design rule check (DRC)
7. Do 3D rendering
8. Use CAM file to generate Gerbers
9. Zip up relevant Gerber files
10. Set PCB fabrication settings
11. Write out bill of materials (BOM)
12. PCB Assembly



3h. Design your PCB Schematic

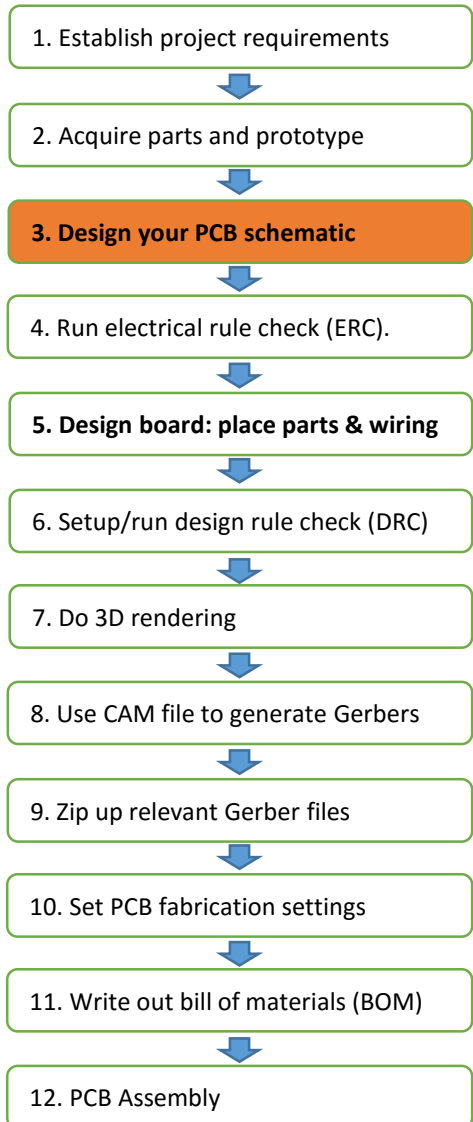
- Wiring nets



- Connects between two “regions”
- VCC and GND are Eagle’s provided wiring nets
- We now connect logic circuit to the output circuit via TIMER_OUTPUT

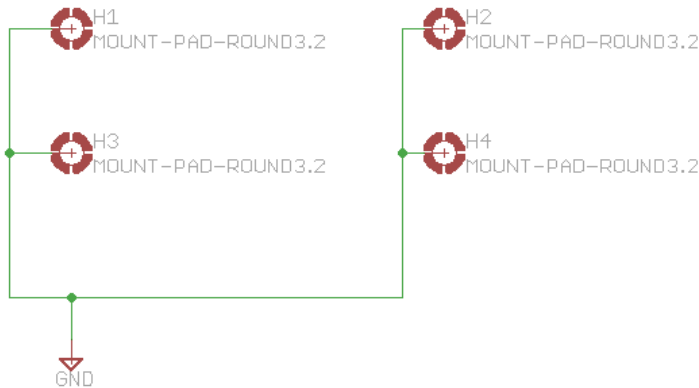
- Steps

1. Extend a short dummy wire on one side
2. Right click, choose “Label”
3. Position “N\$” label just beside wire
4. Rename to something easier like “TIMER_OUTPUT”
5. Repeat for the other end

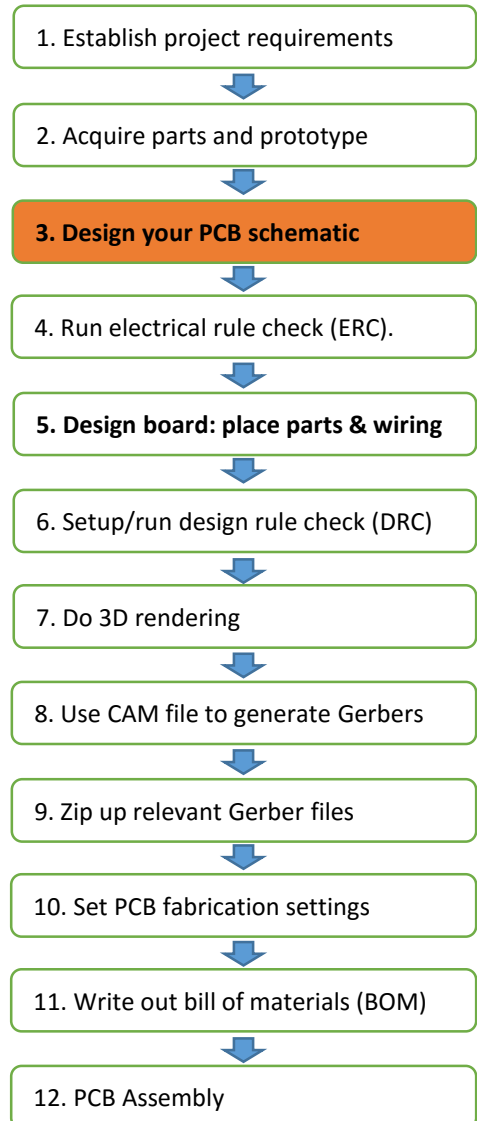


3i. Design your PCB Schematic

- (Grounded) Mounting holes

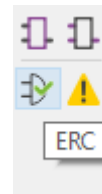
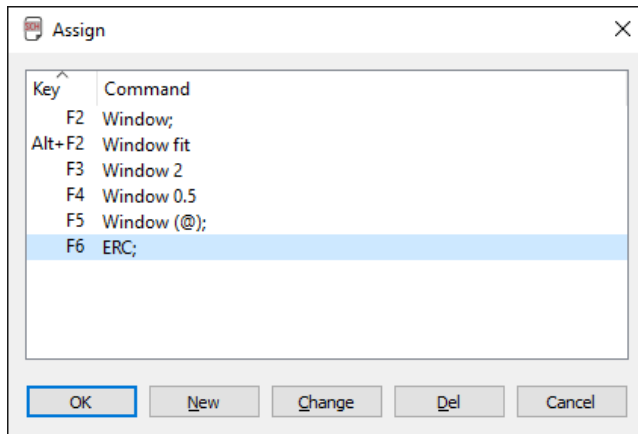


- Allows your PCB to be secured to external case
 - Grounded by screws.
 - Can also use normal ungrounded mounting holes
- Appears in schematic as it is an electrical connection
- Mounting hole
 - Grounded:
 - holes -> MOUNT-PAD-ROUND -> MOUNT-PAD-ROUND3.2
 - Ungrounded:
 - holes -> MOUNT-HOLE -> MOUNT-HOLE3.2
 - 3.2mm hole fits M3 screws with slight allowance

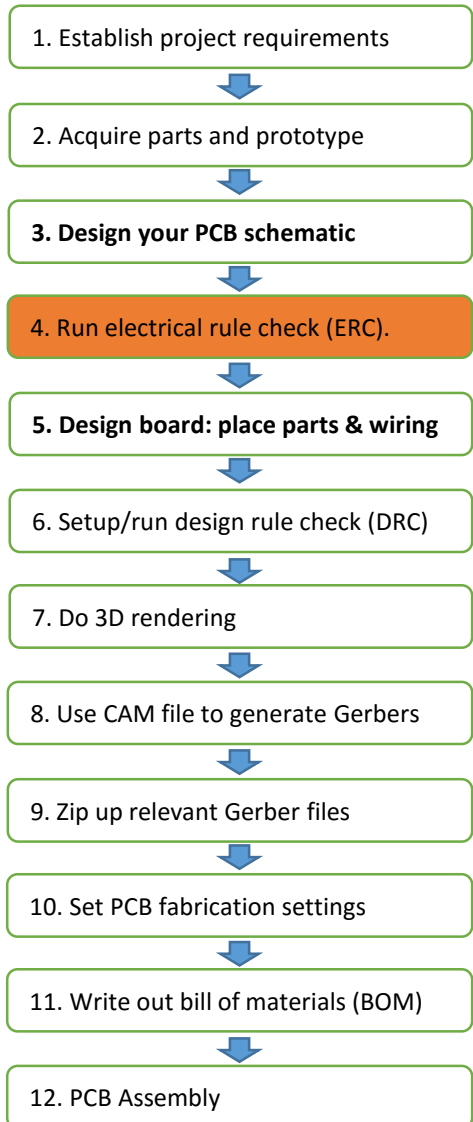
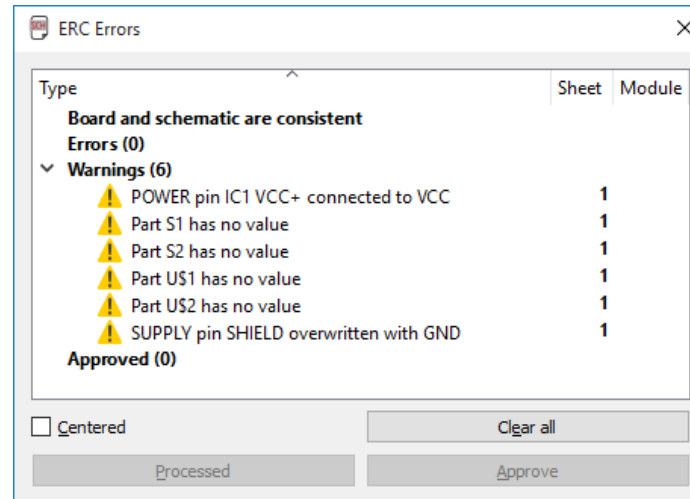


4. Run electrical check (ERC)

- Tests if your wiring has been done properly
- Not foolproof but usually good enough to detect weird or missing connections
- Run ERC regularly, assign to shortcut key. Options -> Assign



- Correct ALL errors
- Inspect all warnings
 - Correct if needed
 - Ignore if you are sure
 - I discourage approving



5a. Design board: place parts & wiring

- Create and save board file (board layout)

- File -> Switch to board

- Initial board configuration

1. Adjust grid settings to mm as shown.

- View -> Grid

2. Create keyboard shortcuts for commonly used functions.

- Options -> Assign
- Not case-sensitive
- End with “;”

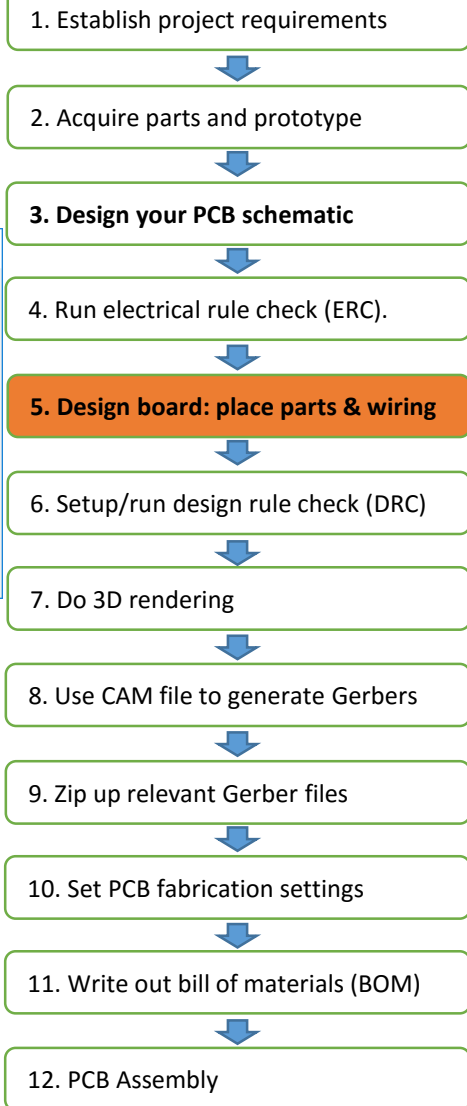
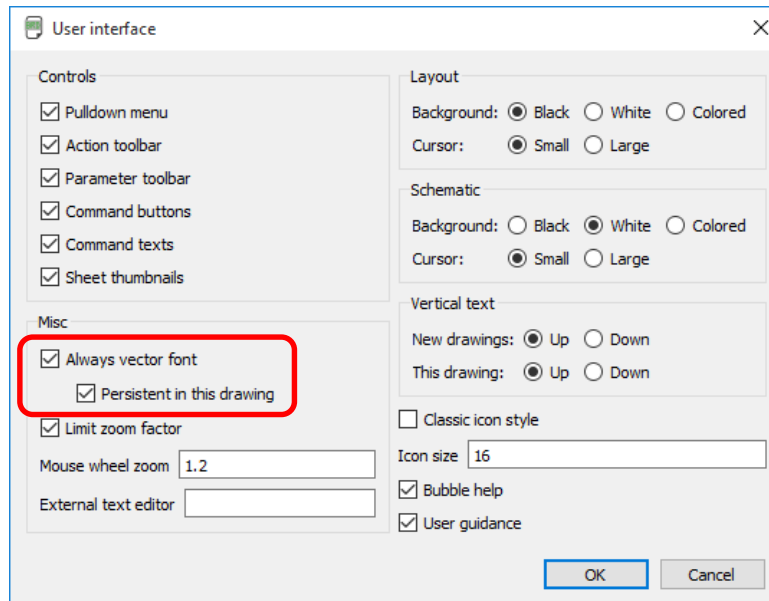
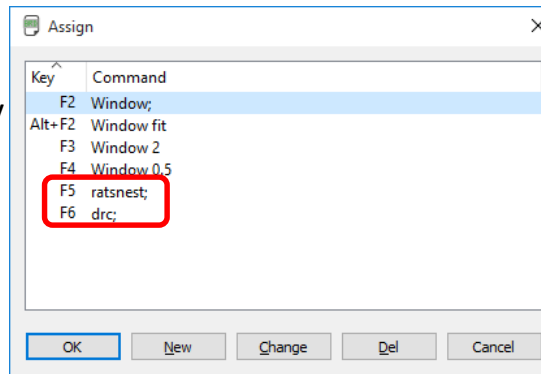
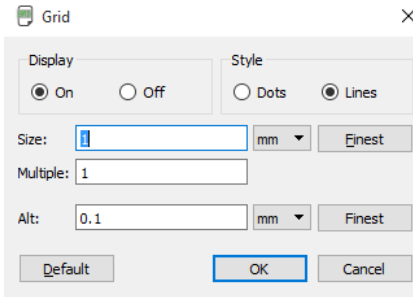
3. Enable Vector font

1. Options -> User Interface
2. Tick Always vector font
3. Tick Persistent

CAM Processor can only convert vector fonts into Gerbers properly.

See:

<http://www.cadsoftusa.com/training-service/faq/#c95>



6a. Setup/run design rule check (DRC)

- Design rules provided by fabricator to check if your PCB adheres to requirements
- Download PCB fabricator DRU, in this case Elecrow
- <http://www.elecrow.com/10pcs-2-layer-pcb-p-1175.html>
- http://www.elecrow.com/download/Elecrow_PCB_eagle_rule.zip

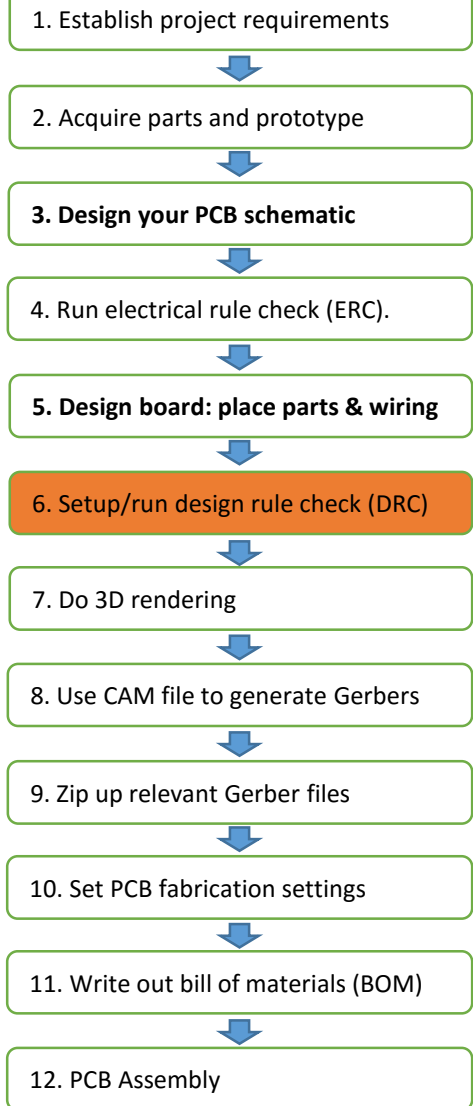


The screenshot shows a web browser window with the URL www.elecrow.com/10pcs-2-layer-pcb-p-1175.html. The page title is "10pcs- 2 layer PCB [SPPO] x". The main content is a table titled "2. Specifications" listing various PCB parameters and their values. Below the table is a section titled "3. Wiki & External links" with four links: "Eagle Design Rule", "Eagle CAM file", "Elecrow service Wiki", and "Q&A for Elecrow PCB service".

2. Specifications	
Multi Layers	1-2
PCB Material	FR-4
Available Color	Green, Red, Yellow, Blue, White, Black
Silk Screen	White, Black (For White Solder Mask only)
Maximum Size	45cm X 120cm
Minimum Qty	5pcs
Board Thickness	0.6mm, 0.8mm, 1.0mm, 1.2mm, 1.6mm
Thickness Tolerance ($\geq 1.0\text{mm}$)	$\pm 10\%$
Thickness Tolerance ($< 1.0\text{mm}$)	$\pm 0.1\text{mm}$
Insulation Layer Thickness	0.075mm—5.00mm
Minimum PCB track	6mil (Recommend $> 8\text{mil}$)
Minimum Track/Vias Space	6mil (Recommend $> 8\text{mil}$)
Minimum pads Space	8mil
Minimum silkscreen text size	32mil
Out Layer Copper Thickness	1oz(35um)
Inner Layer Copper Thickness	1oz(35um)
Drilling Hole (Mechanical)	0.3mm—6.35mm
Finish Hole (Mechanical)	0.8mm—6.35mm
Drill Diameter Tolerance (unplated)	0.05mm
Drill Diameter Tolerance (plated)	0.1mm
Outline Tolerance (Mechanical)	$\pm 0.20\text{mm}$
Aspect Ratio	8:1
Solder Mask Type	Photosensitive ink
SMT min Solder Mask Width	0.2mm
Min Solder Mask Clearance	0.2mm
Solder Mask Thickness	15um
Surface Finish	HASL, HASL (Lead Free), ENIG+\$14.9.

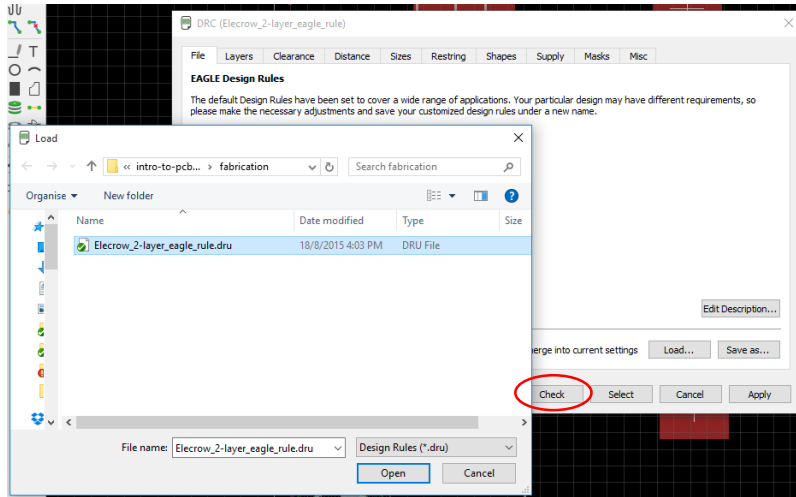
3. Wiki & External links

- [Eagle Design Rule](#)
- [Eagle CAM file](#)
- [Elecrow service Wiki](#)
- [Q&A for Elecrow PCB service](#)

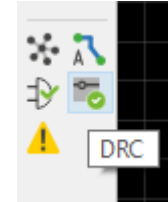
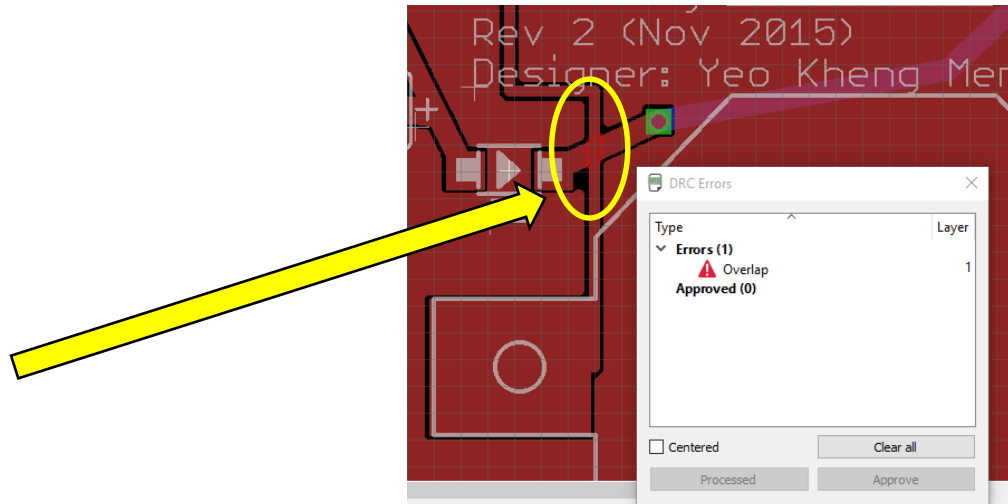


6b. Setup/run design rule check (DRC)

- Load DRU file just once
- Press “check” to check your board or use keyboard shortcut



- Rectify all errors



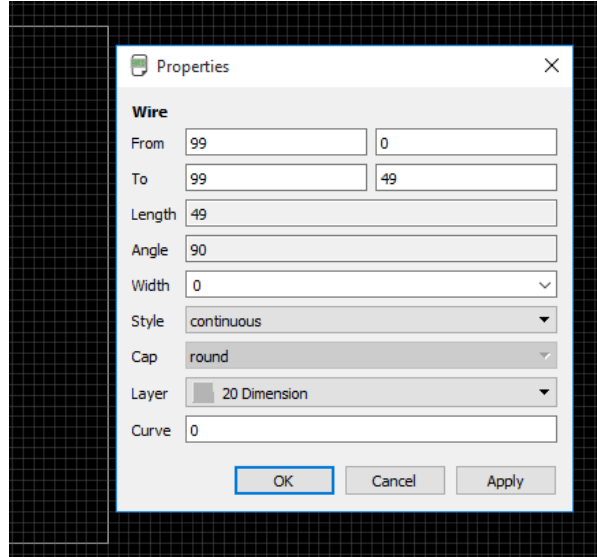
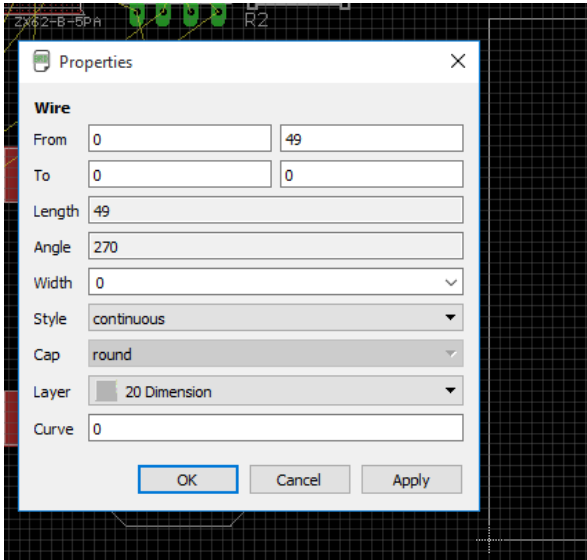
1. Establish project requirements
2. Acquire parts and prototype
3. Design your PCB schematic
4. Run electrical rule check (ERC).
5. Design board: place parts & wiring
6. Setup/run design rule check (DRC)
7. Do 3D rendering
8. Use CAM file to generate Gerbers
9. Zip up relevant Gerber files
10. Set PCB fabrication settings
11. Write out bill of materials (BOM)
12. PCB Assembly



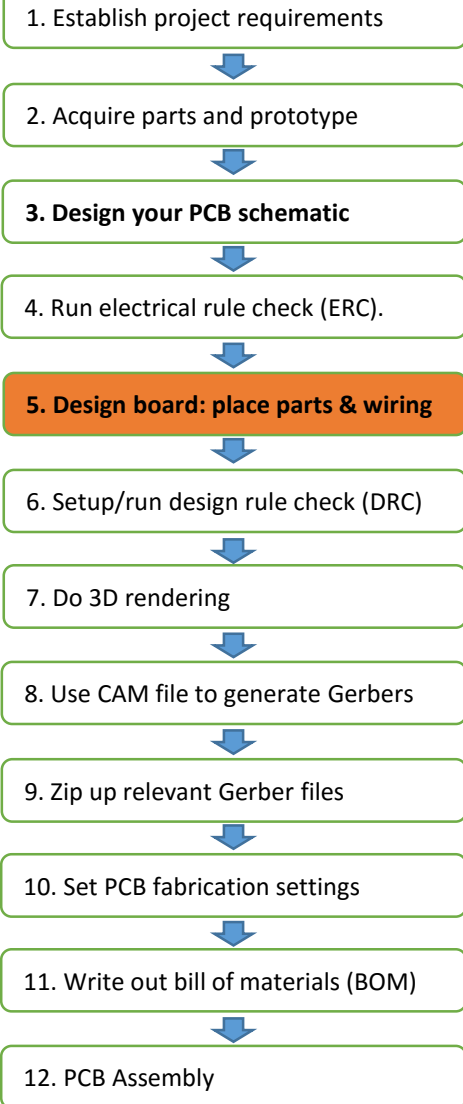
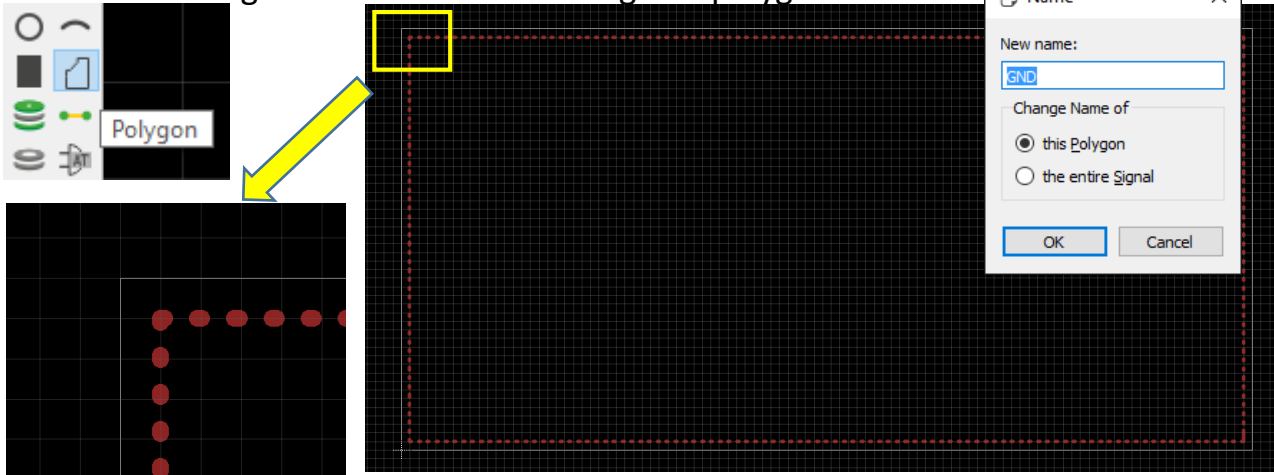
5b. Design board: place parts & wiring

General board steps

1. Adjust board size to your requirements eg: 99mm x 49mm (check fabricator price point)



2. Place ground/power plane on Top layer using polygon function. 1mm away from edges to ease future resizing. Set polygon name.



5c. Design board: place parts & wiring

- General board steps

3. Component placement order

1. Place mounting holes
2. User-facing components, like USB connectors, switches and LEDs
3. Large components
4. Everything else

4. Wiring order (Trace width usually the size of the solder pads)

1. Wire high-speed connections, avoid using vias
2. Wire power connections with wider traces
3. Wire everything else

5. Execute DRC check and rectify if needed

6. Silkscreen placement order (don't overlap with any holes/vias/parts)

1. Smash component & adjust part designators if required to minimise confusion
2. Add descriptive text to certain areas to silkscreen
3. Add PCB description/revision

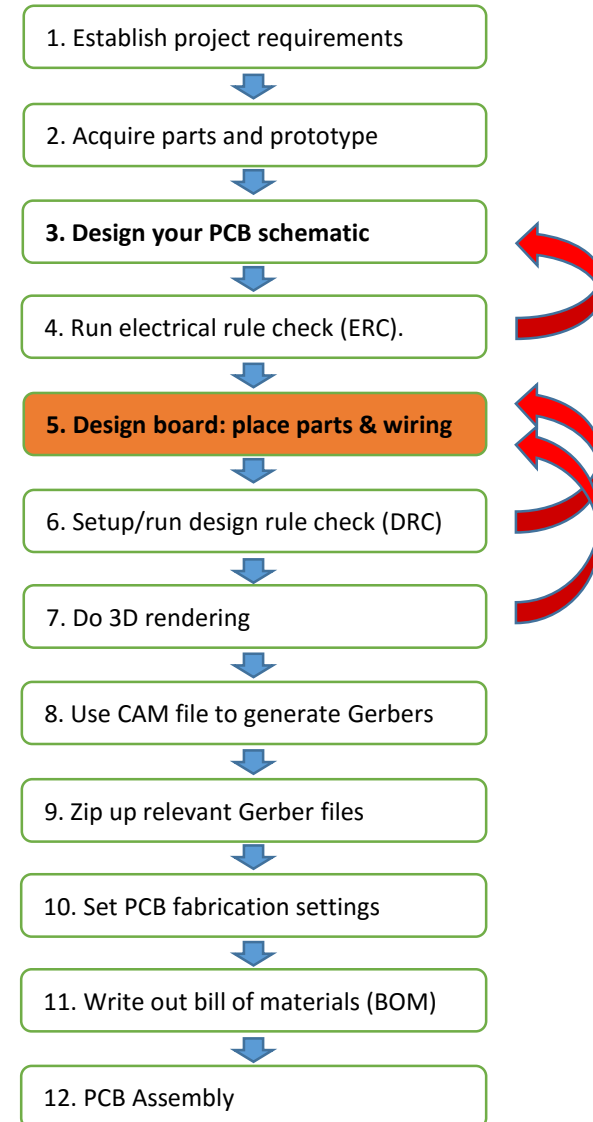
7. Miter corners

8. Make sure Ratsnest gives "Nothing to do"

```
Intel Edison Pin Breakout+  
Rev 1 (June 2015)  
Algo Access Pte Ltd  
Designer: Yeo Kheng Meng  
MIT License
```

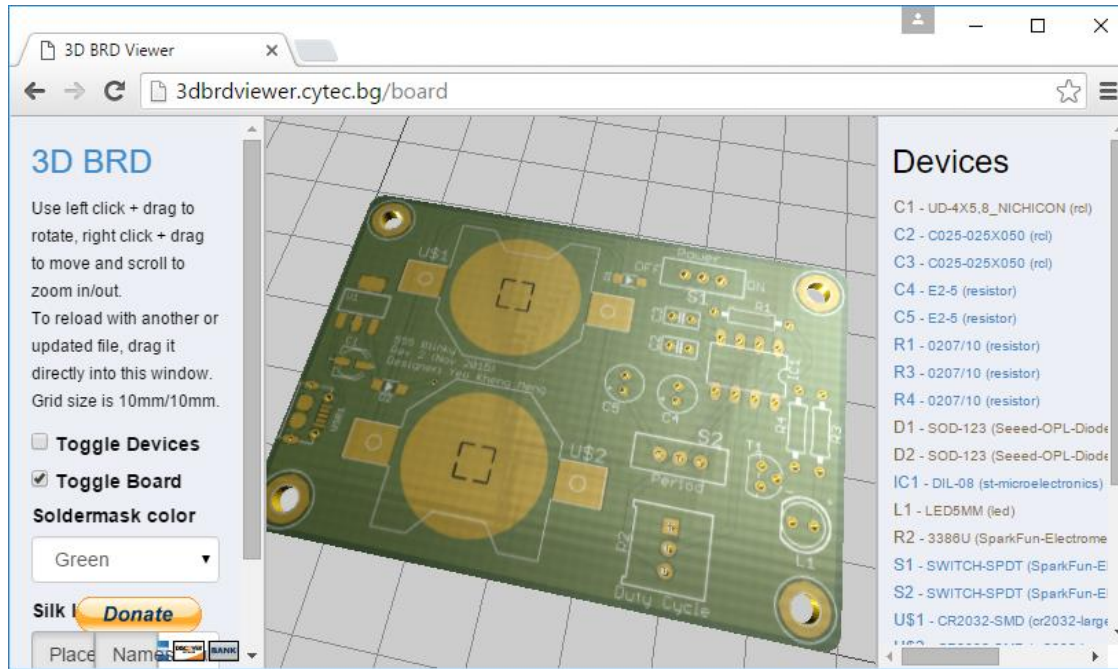
- Tips

- Execute Ratsnest to recalculate remaining airwires' position as much as possible after every operation
- Run DRC after you make every change to catch problems earlier
- Wire at 45 degree angles to reduce signal reflection

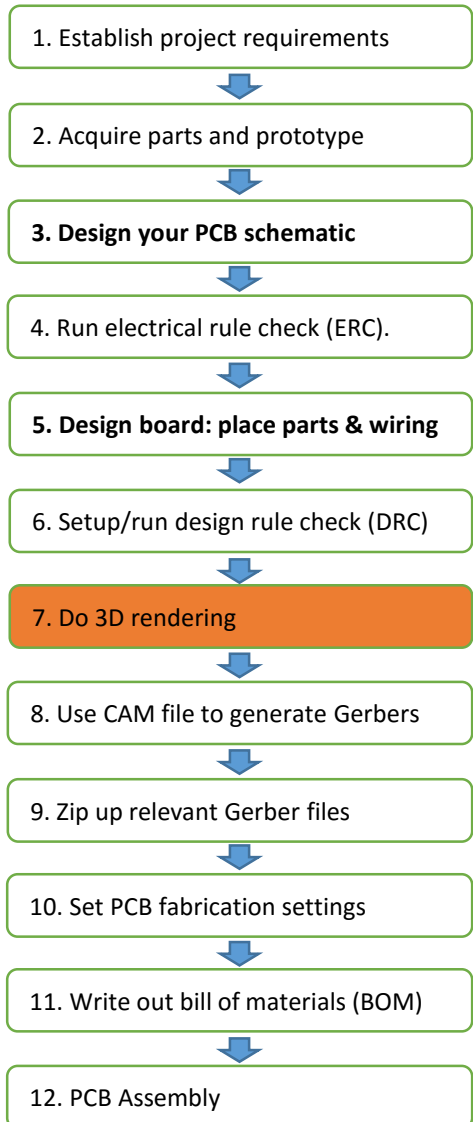


7. Do 3D rendering

- Use this website <http://3dbrdviewer.cytec.bg/>

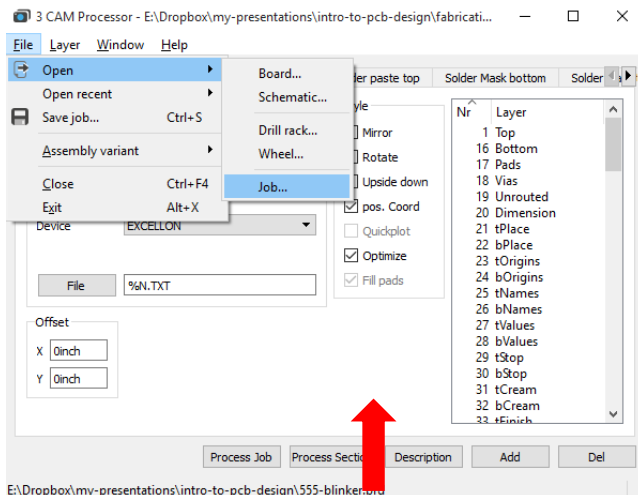


- Check for any visible errors, especially silkscreen overlap
- Check for components' "closeness".
- Go back to Step 5 if not satisfied

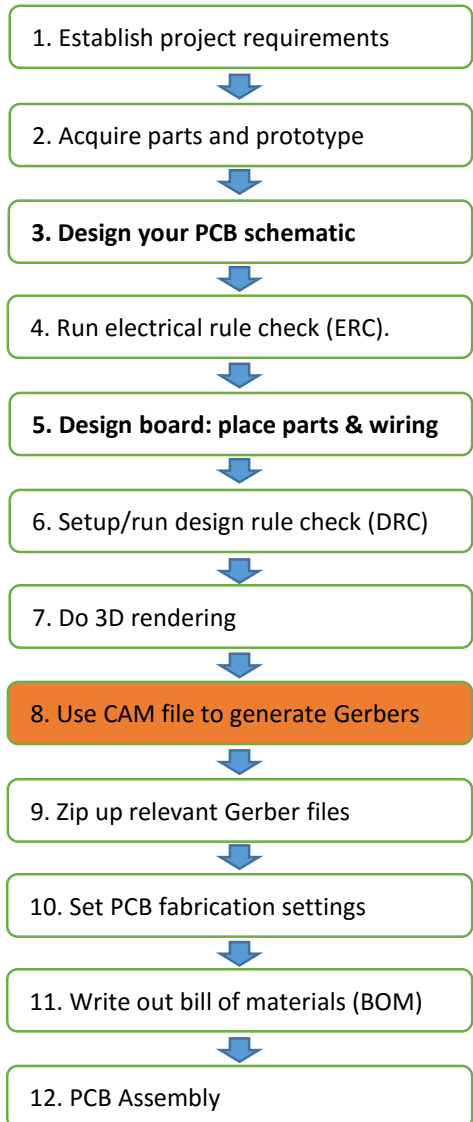


8. Use CAM file to generate Gerbers

- Gerber files
 - 2D vector format used by PCB fabricator to make PCBs for you
- CAM file provided by fabricator to help you generate Gerber layer files
- Download PCB fabricator's CAM processor, in this case Elecrow
- <http://www.elecrow.com/10pcs-2-layer-pcb-p-1175.html>
- [http://www.elecrow.com/download/Elecrow Gerber Generator Drill Align.zip](http://www.elecrow.com/download/Elecrow_Gerber_Generator_Drill_Align.zip)
- Use the correct CAM file according to number of layers like 2 layers
- Open CAM Processor: File -> CAM Processor
- Once in CAM Processor, load the CAM job as shown below



- Once you are done, click “Process Job”



9. Zip up relevant Gerber files

- See what files are required by the PCB fabricator
- Elecrow's requirements:

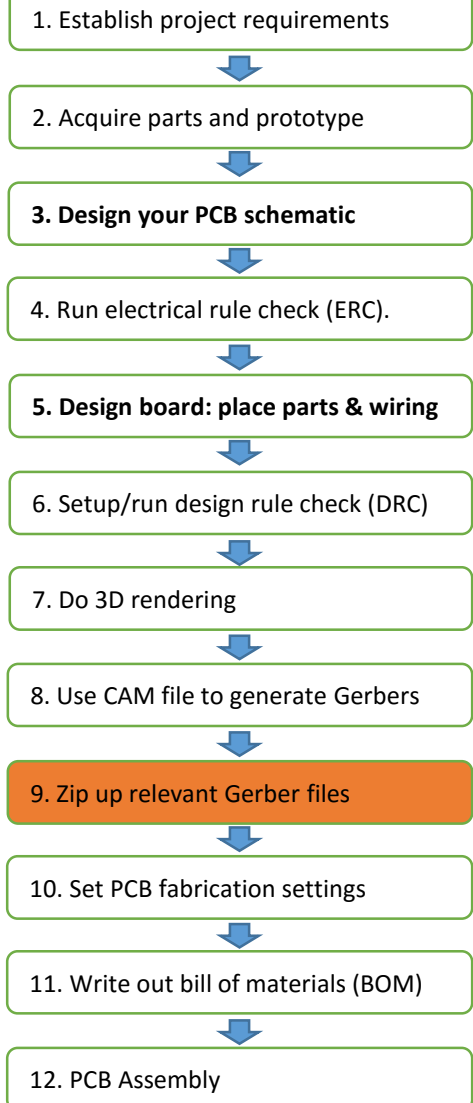
1. Gerber Files Required:

The following files would be needed for the 2-layers PCB production:

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
Mechanical layer :	pcbname.GML

Note: The Gerber file must be RS-274x format and board outline must be included at least in one layer; All the holes will be plated by default, if you need some holes not plated, please name the non-plated holes as pcb.name-NPTH.txt and leave a comment; For one layer PCB, If you need the solder mask on the top and bottom side, please order it as 2 layer PCB and leave a comment '1 layer PCB with solder mask both side'.

- Go to saved folder and zip up the required files
- Rightfully you should inspect your Gerber files for potential mistakes using software/websites like:
 - GR-Prevue: <http://www.graphiccode.com/GC-Prevue> Gerber Viewer
 - <http://circuitpeople.com/>
 - <http://mayhewlabs.com/webGerber/>
 - But so far I don't have any problems so I usually skip the Gerber file check step



10a. Set PCB fabrication settings

- Go to order page <http://www.elecrow.com/10pcs-2-layer-pcb-p-1175.html>

10pcs- 2 layer PCB

Your price: \$61.80

20 reward points

Model: SPP01010PP
Shipping Weight: 1211.00g
Designer: Others

Please Choose:

Layer: 2

PCB Thickness: 1.6mm

Copper Weight: 1oz 35um

The real dimension must be equal or smaller than the option.

PCB Size: 5cm Max * 10cm Max (+\$3.00) (+100g)

PCB Color: Black

Surface Finish: ENIG(Lead Free) (+\$14.90)

PCB Stencil: 30cm X 40cm with frame (+\$18.00) (+1001g)

Lead time: Rush 48h, Shipped in 3-4 days (+\$16.00)

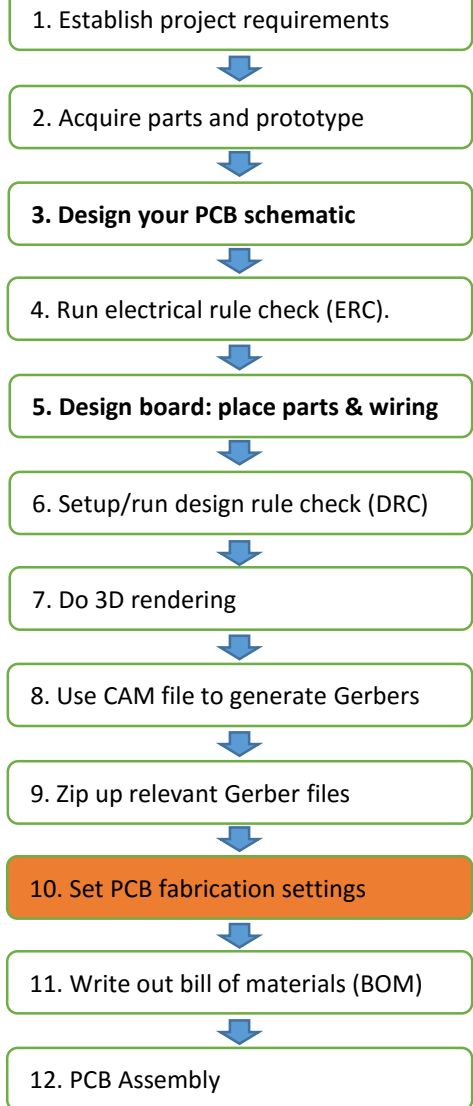
Panelizing: Single PCB with milling

***Upload the Gerber file must be packed into RAR orZIP format.**

File: Choose file No file chosen

Qty: 1 Unit(s) **ADD TO CART**

Add to Wish List
Submit a wholesale Inquiry



- Basic options like PCB size, color and lead time self explanatory

10b. Set PCB fabrication settings

Layer

PCB Thickness

Copper Weight

The real dimension must be equal or smaller than the option.

PCB Size

PCB Color

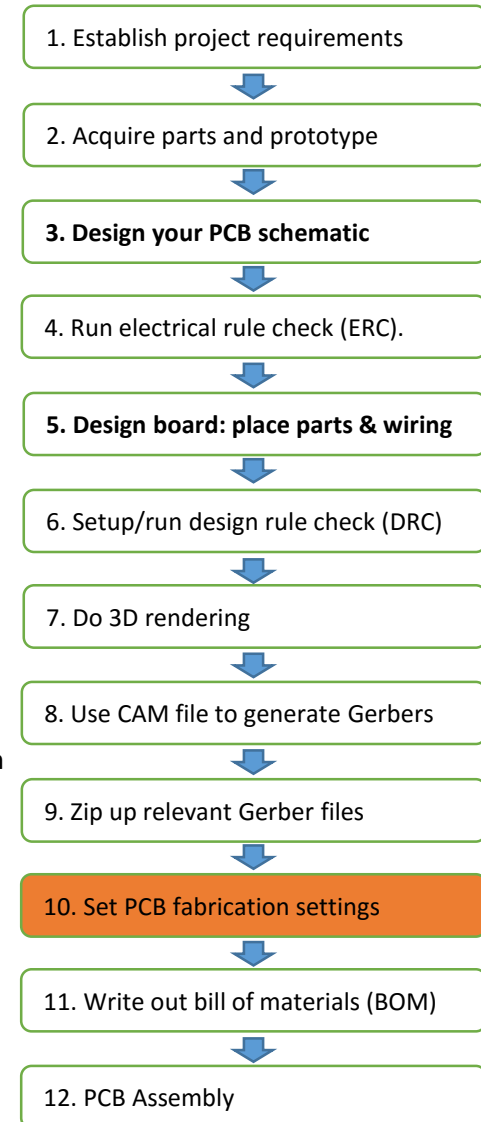
Surface Finish

PCB Stencil

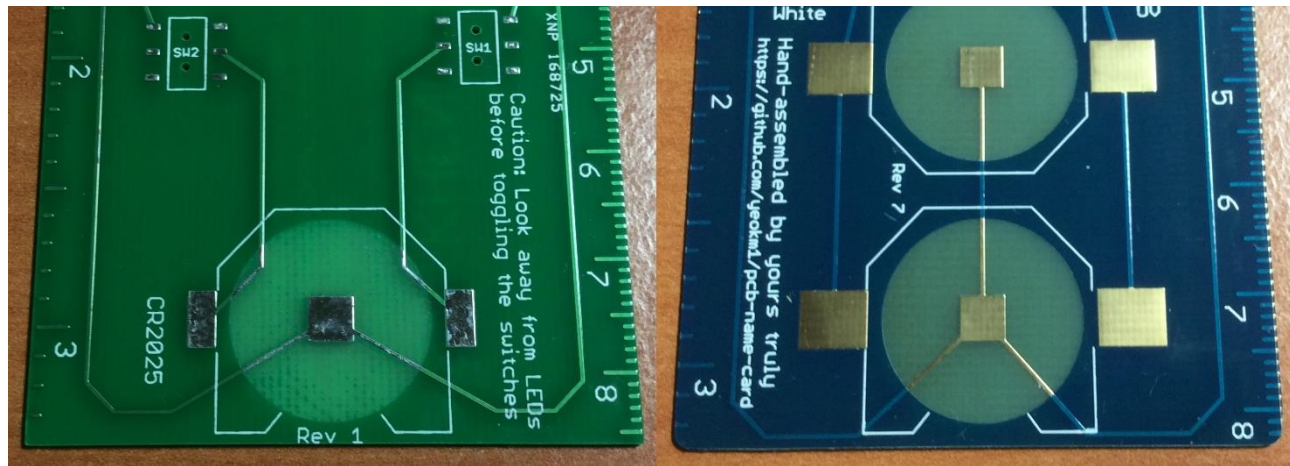
Lead time

Panelizing

- Layer
 - 1-layer with mask both side: 1 copper layer only but you want both sides soldermask/silkscreen
 - 1-layer with mask one side: 1 copper layer only and you want one side soldermask/silkscreen
 - 2-layer: normal two copper layer boards
- Copper weight:
 - Copper layer thickness and amount, 1oz is usually enough
- PCB Thickness:
 - Typical is 1.6mm
 - Elecrow can go as thin as 0.6mm for normal FR4 PCBs. Flex PCBs can go even thinner like 0.1mm
- Surface finish
 - HASL vs ENIG next slide
- PCB Stencil
 - Help you solder SMT components if you have solder paste and reflow oven/hotplate
 - Frame or no frame provided. I normally go with the frame as it just costs slightly more.
- Panelizing
 - Squeeze many small PCBs into one piece to save space but you have to cut after receiving.
 - I normally pick "Single PCB with milling" as I have no need to panelize my PCB



10c. Set PCB fabrication settings



HASL

vs

ENIG

	HASL (lead-free)	ENIG
Acronym for	Hot Air Solder Levelling	Electroless nickel immersion gold
How it is made?	Copper surfaces covered with solder by dipping PCB in solder then blowing away	Copper surfaces covered with nickel with gold coating via immersion
Coating evenness	Uneven, don't use for fine-pitch SMD components	Very even coat, must use if you have fine SMD components.
Durability	Reasonable, but don't expose too long to air	Better than HASL, can last for months
Solderability (should not factor in your decision)	Great	So-so
Cost	Low	High

1. Establish project requirements



2. Acquire parts and prototype



3. Design your PCB schematic



4. Run electrical rule check (ERC).



5. Design board: place parts & wiring



6. Setup/run design rule check (DRC)



7. Do 3D rendering



8. Use CAM file to generate Gerbers



9. Zip up relevant Gerber files



10. Set PCB fabrication settings



11. Write out bill of materials (BOM)



12. PCB Assembly

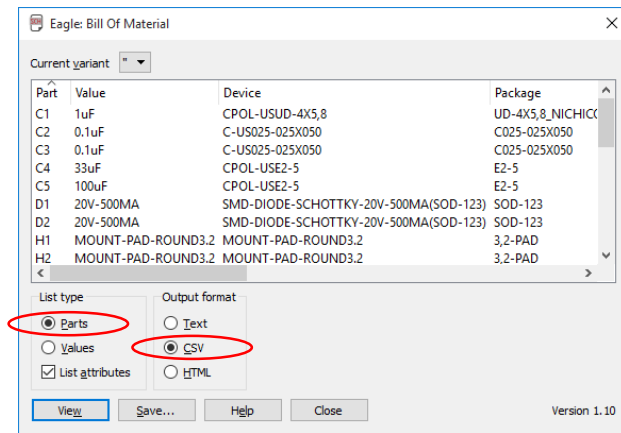


• Others:

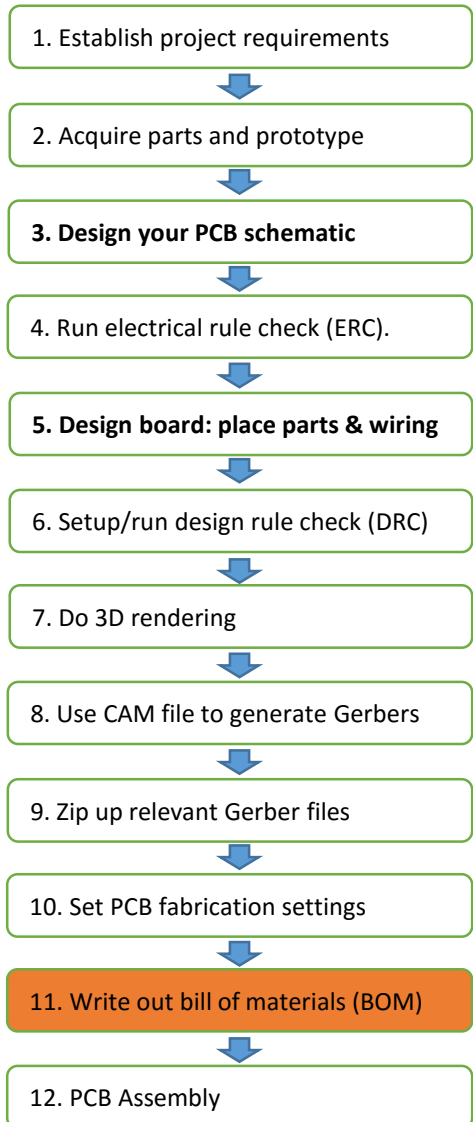
- Electrolytic Gold-plating, Immersion Silver, Immersion Tin, Organic Surface Protectant

11. Write out bill of materials (BOM)

- Export BOM and modify.
- So others can replicate your PCB
- Go to Schematic view, File -> Export -> BOM
- Select Parts type and CSV format as shown



- Use Libreoffice to open CSV file and save in its native ODS format
 - CSV not enough features, does not store column width
 - Some Excel versions have difficulty opening CSV file generated by Eagle
 - Open-source format
- Modifying the generated BOM
 1. Remove mounting holes
 2. Collapse duplicate devices and combine their part designators
 3. Remove: Description, OC_FARNELL, OC_NEWARK, PROD_ID, last VALUE column
 4. Add: Quantity, Unit Cost, Total Unit cost (=Quantity * Unit Cost), Source webpage
 5. Modify the generic Eagle BOM with the actual BOM data (also add PCB Fab cost)
 6. Can have final Total Cost at bottom if you wish



12a. PCB Assembly

- Manual Soldering
 - Soldering station
 - Through-hole and SMT components
- Specialised SMD Soldering techniques (with stencil and paste)



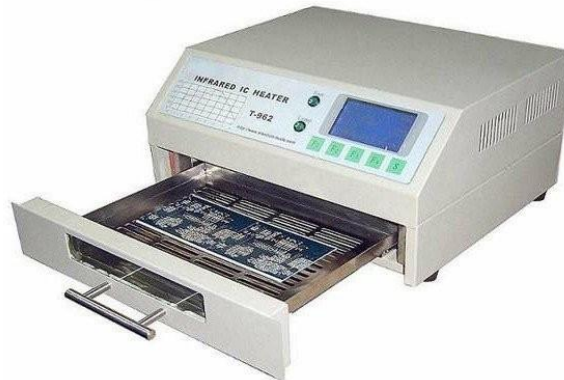
- Hot air blower



- Hot plate



- Reflow oven



1. Establish project requirements
2. Acquire parts and prototype
3. Design your PCB schematic
4. Run electrical rule check (ERC).
5. Design board: place parts & wiring
6. Setup/run design rule check (DRC)
7. Do 3D rendering
8. Use CAM file to generate Gerbers
9. Zip up relevant Gerber files
10. Set PCB fabrication settings
11. Write out bill of materials (BOM)
12. PCB Assembly

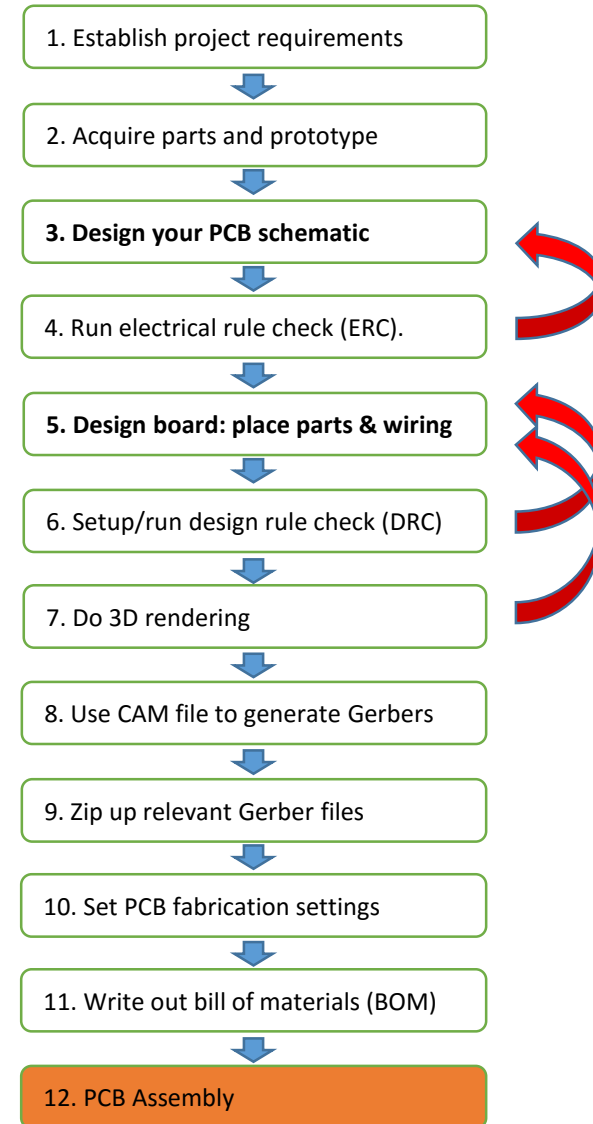


12b. PCB Assembly

- Reflow Soldering only for surface-mount components
- Requires stencil and solder paste
- **FIRST** stencil/reflow the SMD components **THEN** solder the through-hole components by hand.



- A typical thermal profile, consult data sheet to determine optimum profile
- Thermal profile based on:
 - Lead vs lead-free solder paste
 - Lead vs lead-free components
 - Lead vs lead-free use depends on RoHS compliance
- Profile Steps
 1. Preheat: Flux activation
 2. Soak: Ensure all board and components are at the same temperature
 3. Reflow: “Melts” solder paste and let it settle properly on the solder pads
 4. Cooling: Brings temperature back down at controlled rate to prevent thermal stress



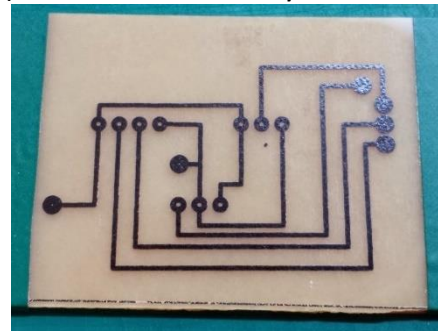
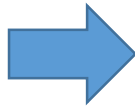
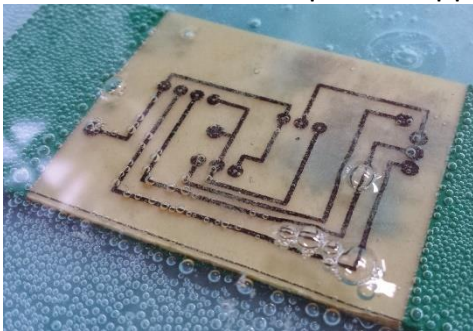
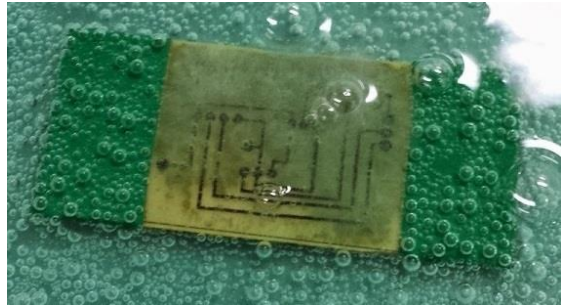
DIY PCB Fabrication (Toner transfer)

- Materials

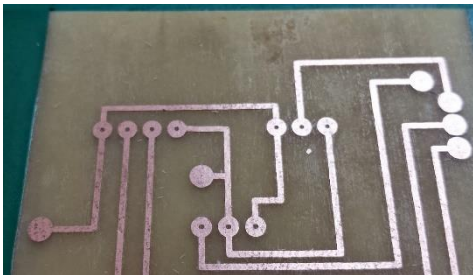
- Liquid Mixture: Vinegar, Table Salt, Hydrogen Peroxide (H_2O_2)
- Transfer paper
- FR-4 board

- Steps

1. Print circuit diagram on transfer paper using laser printer
2. Iron/Heat press transfer paper into FR-4 board
 - <https://www.youtube.com/watch?v=dvLRrXjktbE>
3. Mix liquid mixture of vinegar, H_2O_2 and salt
4. Submerge FR-4 board in liquid mixture and agitate
5. Continue until all exposed copper has been “eaten away”. (Toner covers areas you want to leave behind)



6. Use a solvent/paint thinner to wipe off the toner



7. Drill holes to create vias

Source, Leon Lim:

<http://gylim78.blogspot.sg/2014/07/the-diy-pcb-making-guide.html>

<http://gylim78.blogspot.sg/2015/05/pcb-board-gallery.html>

Export DIY PCB images from Eagle:

<http://yeokhengmeng.com/2016/03/eagle-file-to-diy-pcb-etching/>

Time to design your own board!

Final Words

- This is just the beginning
- Practice makes perfect, the more PCBs you design/fabricate, the better you will get
- Unlikely to get right on the first fabrication attempt
 - 2-3 times or more per design