PCB 1:
INTRODUCTION

PRINTED CIRCUIT BOARD (PCB)

Topics

- What is a PCB?
- Why are PCBs important?
- How does one design a PCB?
- How is a PCB manufactured?

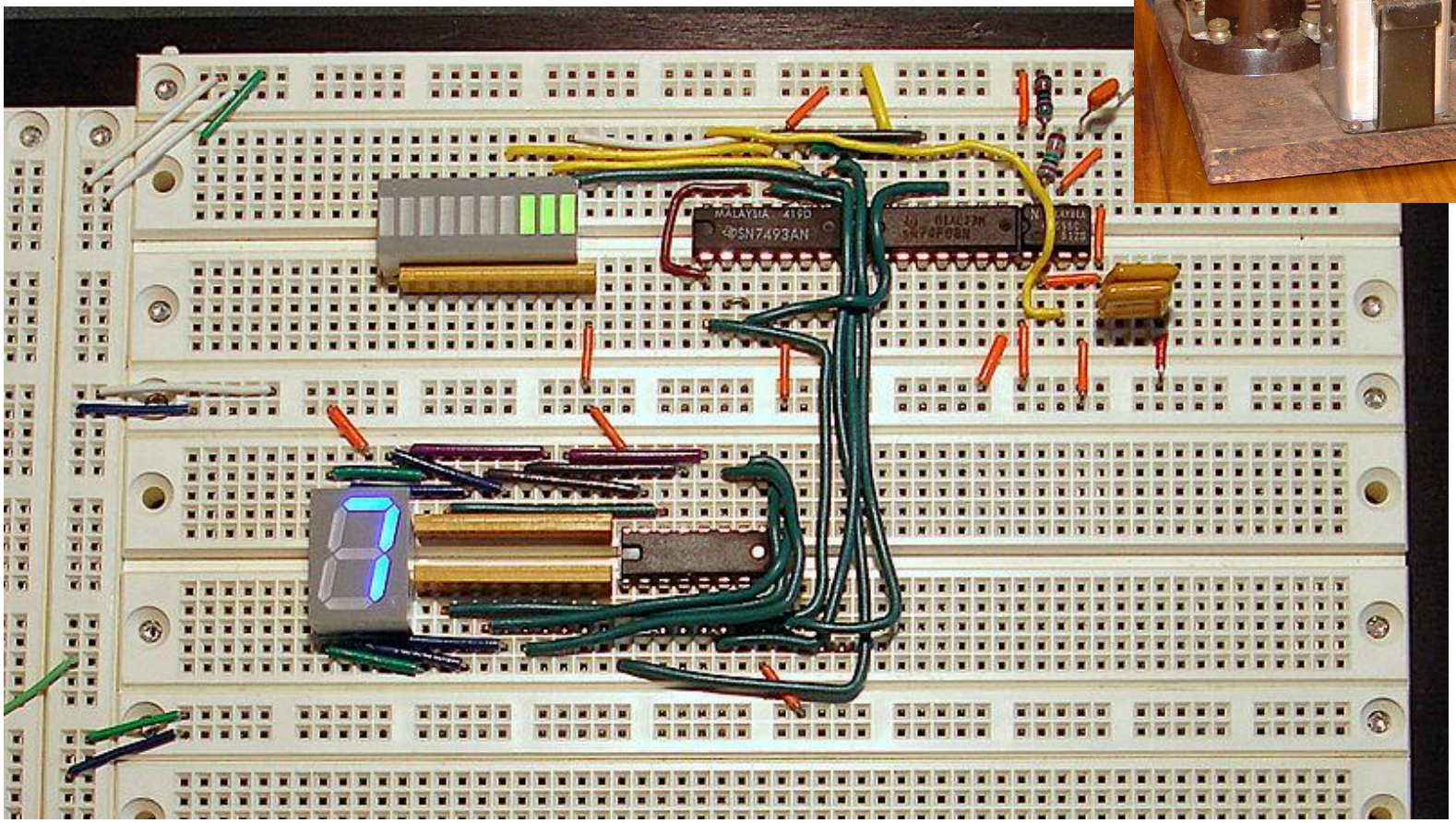
At the end of this section, you should be able to:

- Understand basic terms describing a PCB
- Explain the advantages and disadvantages of a PCB

MANY OPTIONS FOR CREATING ELECTRICAL CIRCUITS

- Breadboard (Plugboard)
- Wirewrap
- Perfboard/Solderboard
- Printed Circuit Board (PCB)
- Flex PCBs

BREADBOARDING ALLOWS FOR RAPID PROTOTYPING

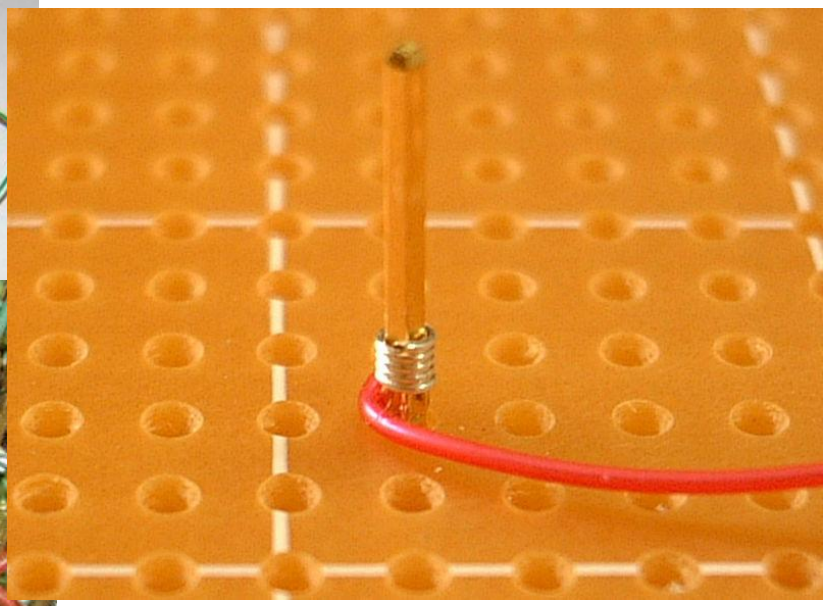


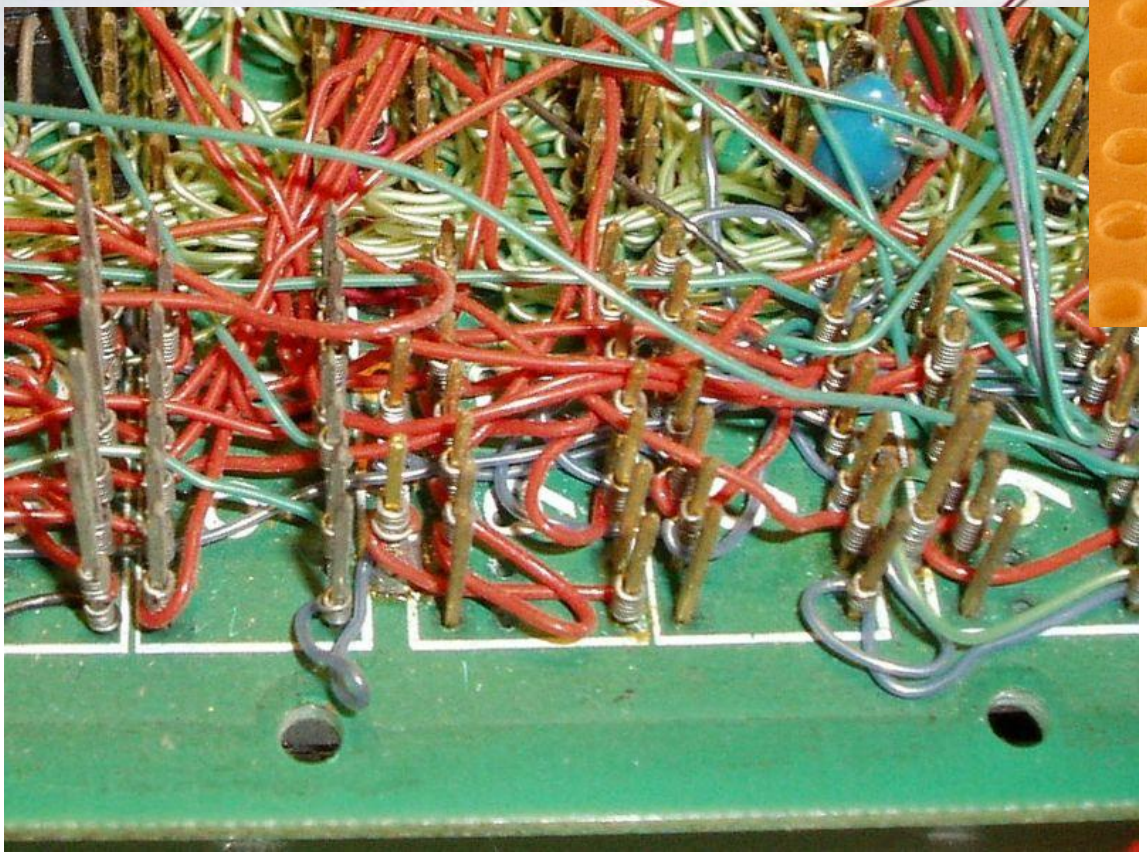
Images: https://en.wikipedia.org/wiki/Breadboard

BREADBOARDING HAS SOME DOWNSIDES

- Unstable behavior may result from
  - Bad connections on breadboard
  - Loose wires accidently touching wrong components
  - Wires coming loose during transportation
- Currents limited by inner connections and wire size
- Pin dimensions and configurations not always conducive to use on a breadboard

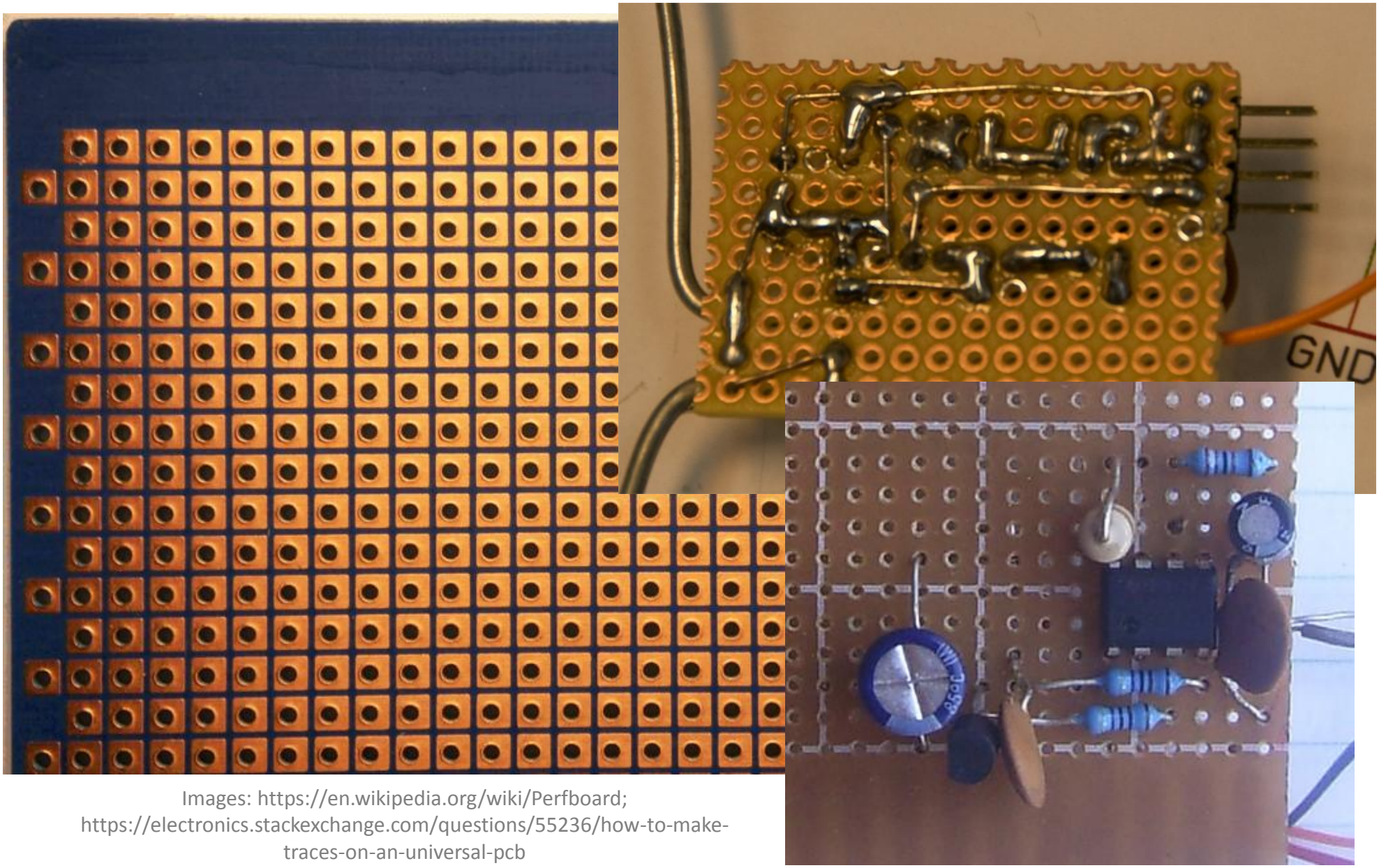
WIREWRAP IS (SLIGHTLY) MORE ROBUST THAN BREADBOARDING





Images: https://en.wikipedia.org/wiki/Wire_wrap

PERFBOARD ALLOWS CONNECTIONS TO BE SOLDERED IN PLACE



Images: https://en.wikipedia.org/wiki/Perfboard; https://electronics.stackexchange.com/questions/55236/how-to-make-traces-on-an-universal-pcb

PERFBOARDING HAS SOME DOWNSIDES

- Difficult to make changes, as solder must be removed
- Requires good soldering skills
- Chip sockets required for easy replacement of components

Image: http://www.newoldsounds.com/images/medium/chip_socket_2_MED.jpg

A PCB RELIABLY IMPLEMENTS A FIXED CIRCUIT DESIGN

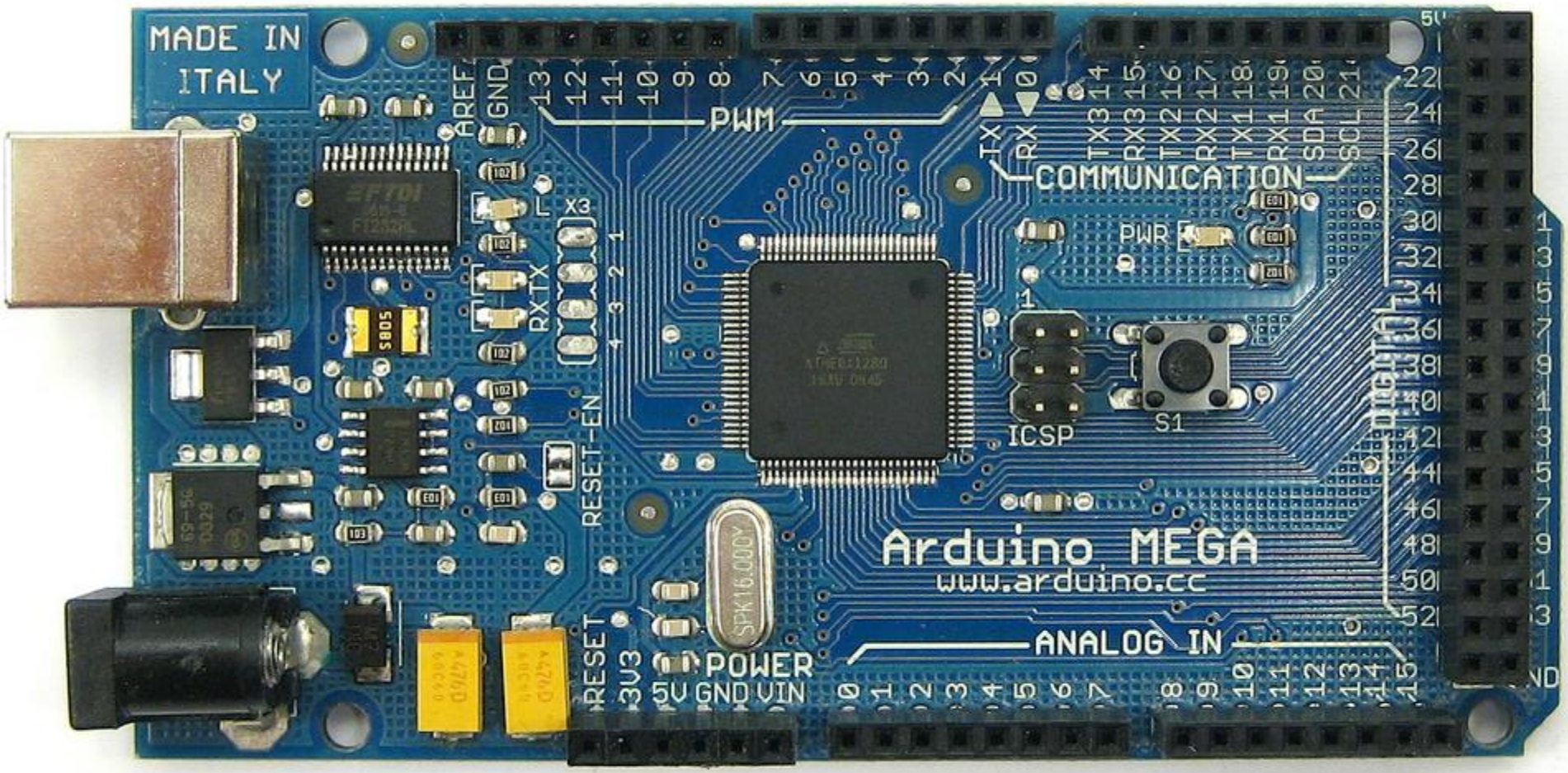
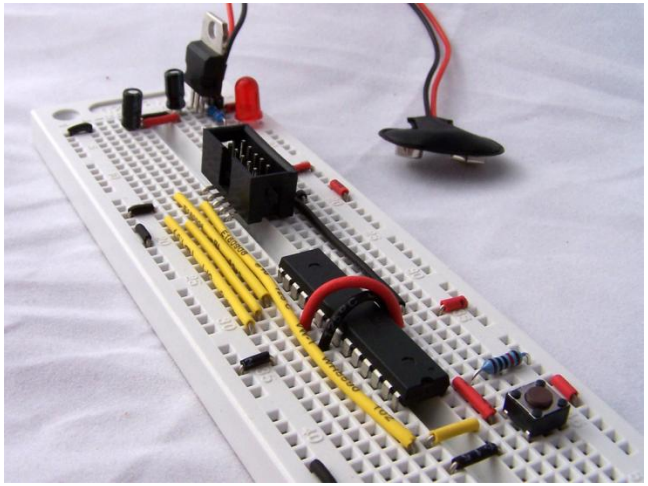


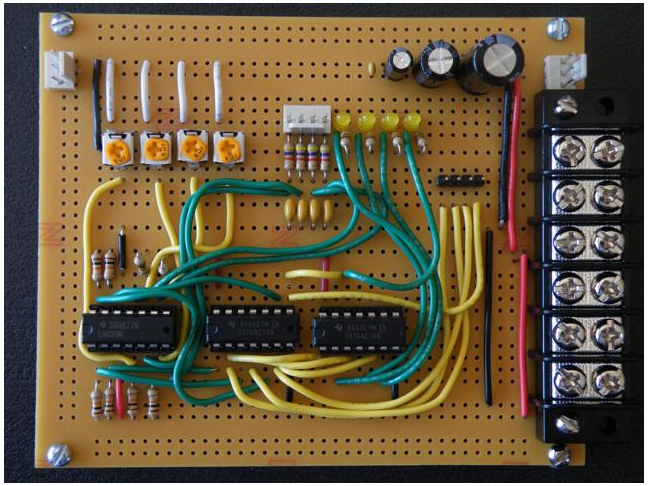
Image: https://www.flickr.com/photos/mellis/4784333051/ (by David Mellis)

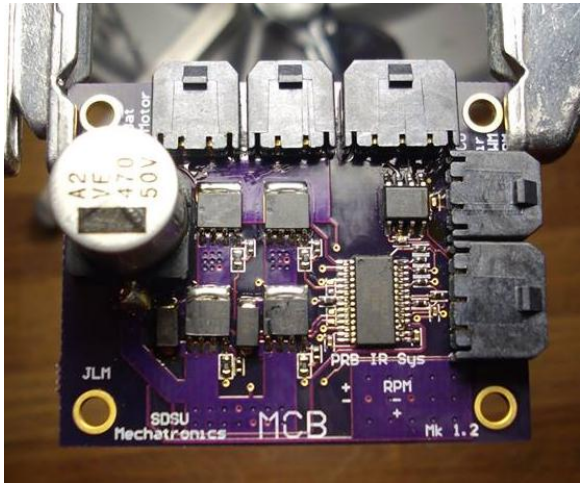
FLEXIBLE SUBSTRATES CAN ALSO BE USED

Image: https://en.wikipedia.org/wiki/File:Olympus_Stylus.jpg

SUGGESTED PROTOTYPE STAGES





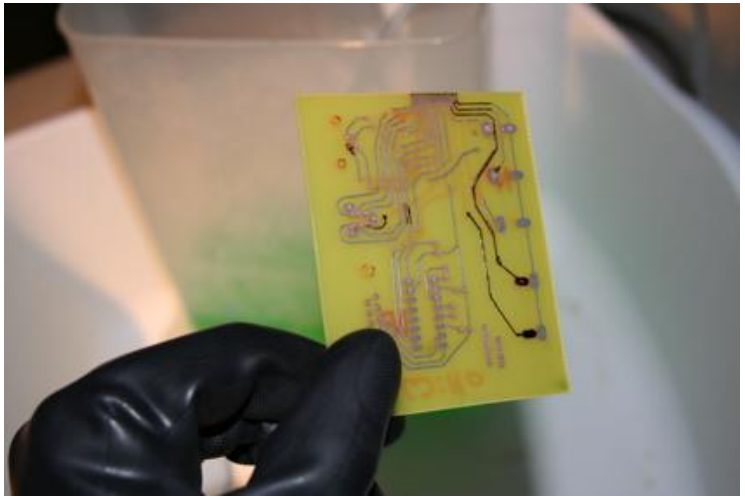


- Your goal should be to deliver a PCB
- Consider other implementations as intermediate

Images: http://d32zx1or0t1x0y.cloudfront.net/2009/07/atmega8_breadboard_1_lrg.jpg
http://mavweb.mnsu.edu/quistd/Classes/EE-336/Perfboard/perfboard-06.jpg
http://sdsurobosub.org/wp-content/uploads/2014/02/MCB-Mk-1.2-Solo.jpg

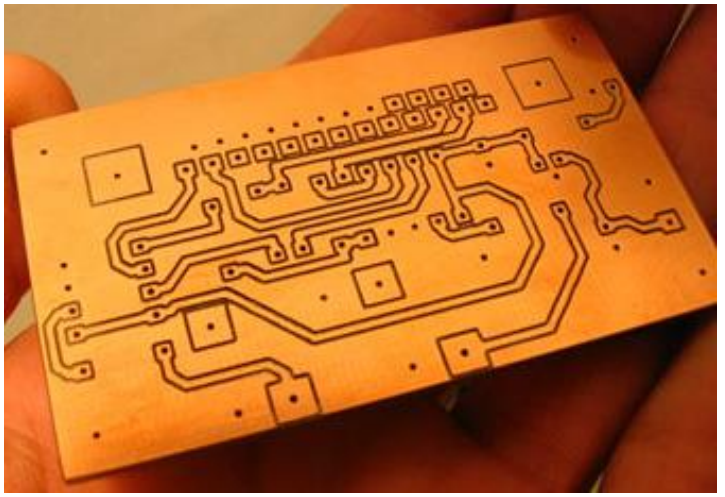
RIGID PCBs CAN BE MANUFACTURED IN SEVERAL WAYS

Image: www.engadget.com/2006/05/10/how-to-design-your-own-ipod-super-dock-part-4/

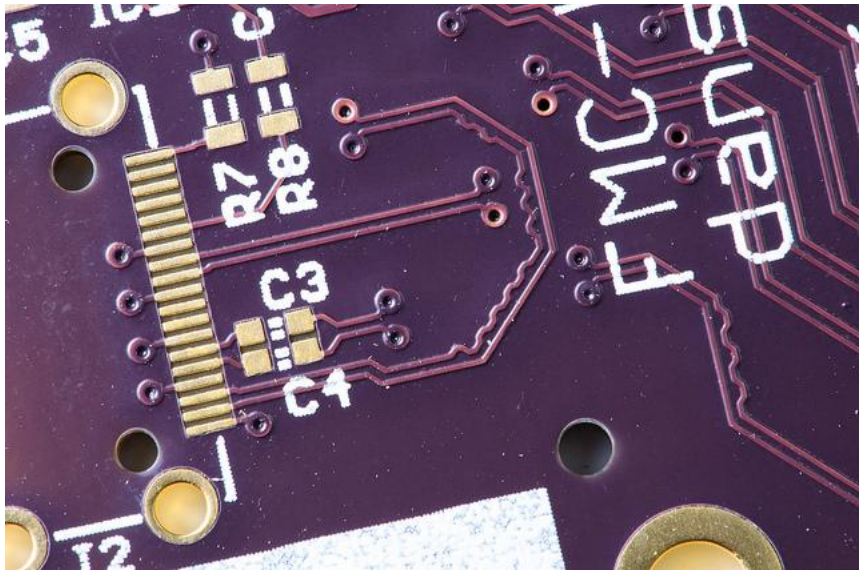


Wet Etching

Image: metalab.at/wiki/PCB_CNC



Milling



Silk-screen Printing

Image:
electronics.stackexchange.com/questions/58594/why-is-silkscreen-called-that-way

PCB PROS AND CONS

Pros

- Permanent connections
- Mechanically rugged
- Precise geometric control
- Less time spent in debug
- Easy to replicate

Cons

- Manufacture wait time
- Cost of manufacture
- Learn new software
- Develop layout skills
- Build component library

PCB NOMENCLATURE

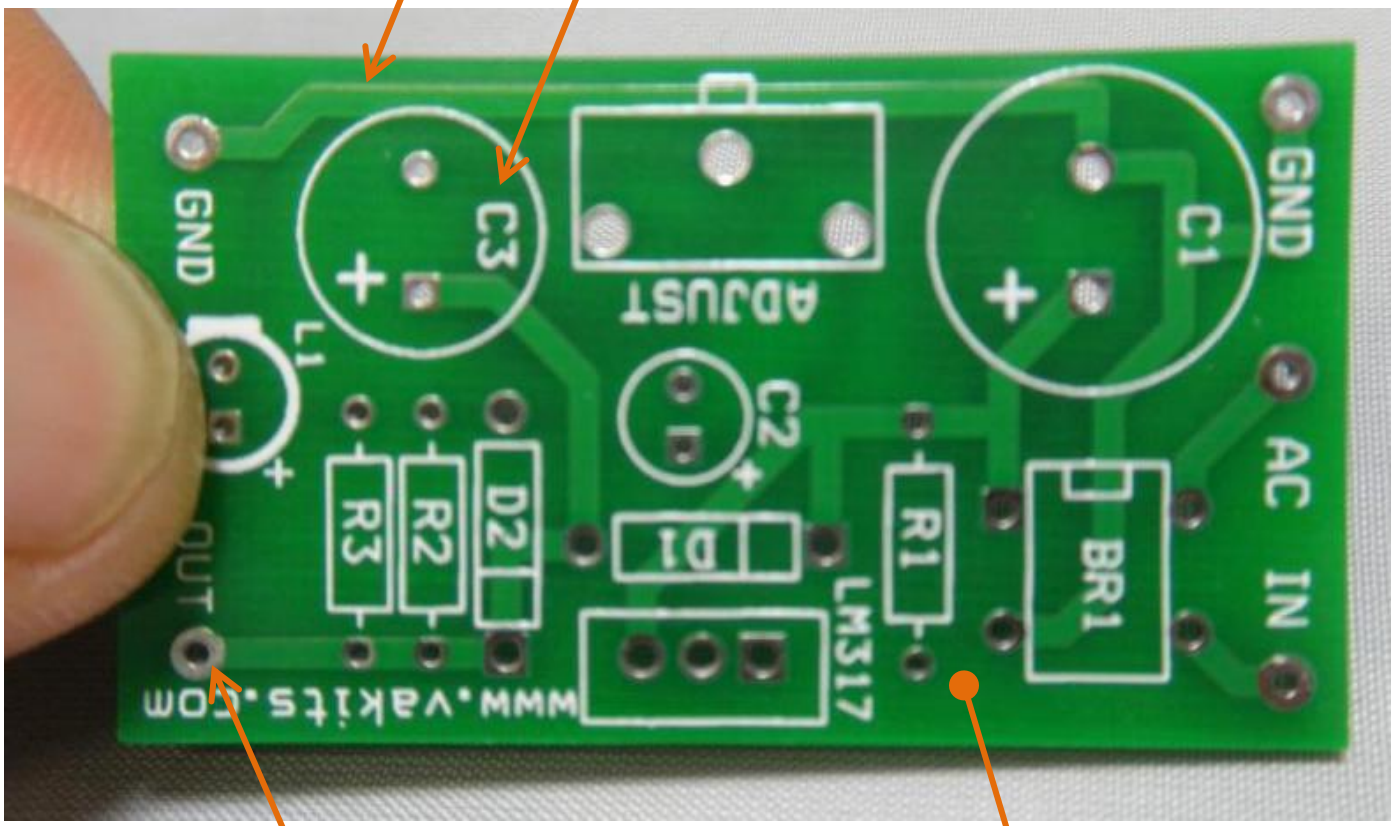


Image: http://vakits.com/sites/default/files/imagecache/product_full/APS3.JPG

A FIBERGLASS SUBSTRATE (CORE) PROVIDES MECHANICAL STRENGTH

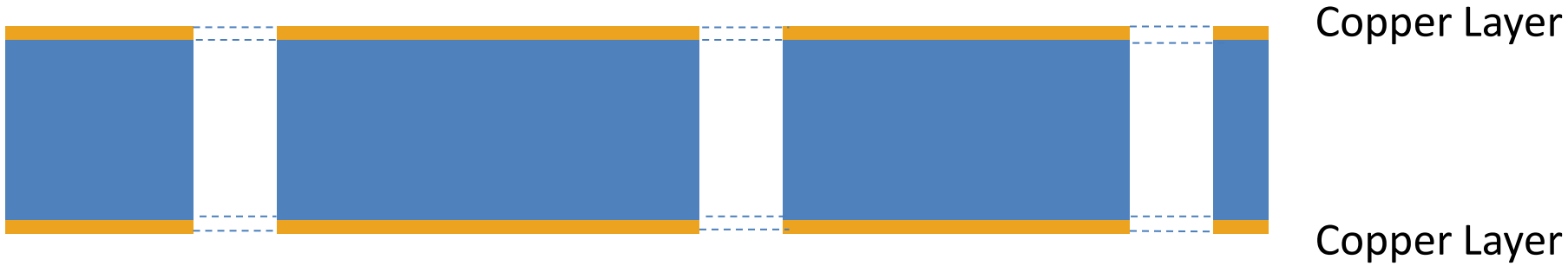
"FR4"

- Solid fiberglass core is at center of PCB
- Acts as insulator between electrical traces
- Commonly 0.6 to 2.0 mm thick
- Your board will have 1.6 mm thick core

A "TWO-SIDED" PCB HAS COPPER ON BOTH SIDES OF THE SUBSTRATE

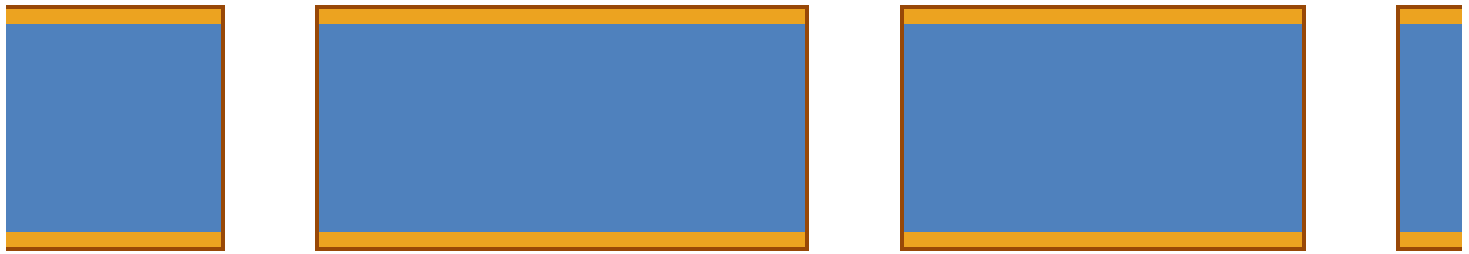
- PCB fabrication starts with copper-clad substrate
- Copper usage expressed in ounces (per square foot)
- Commonly 35 μm (1 oz) or 17 μm (1/2 oz) thick

HOLES ARE DRILLED THROUGH STACKS OF PANELS



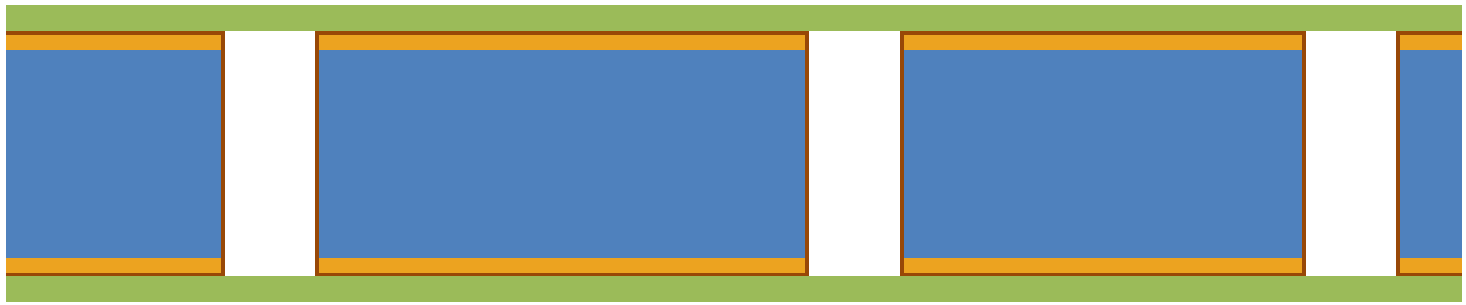
- Holes are made larger than desired, to account for plating processes that will reduce hole size
- Deburring removes raised edges surrounding holes

ELECTROLESS COPPER DEPOSITION COATS BOARD SURFACE



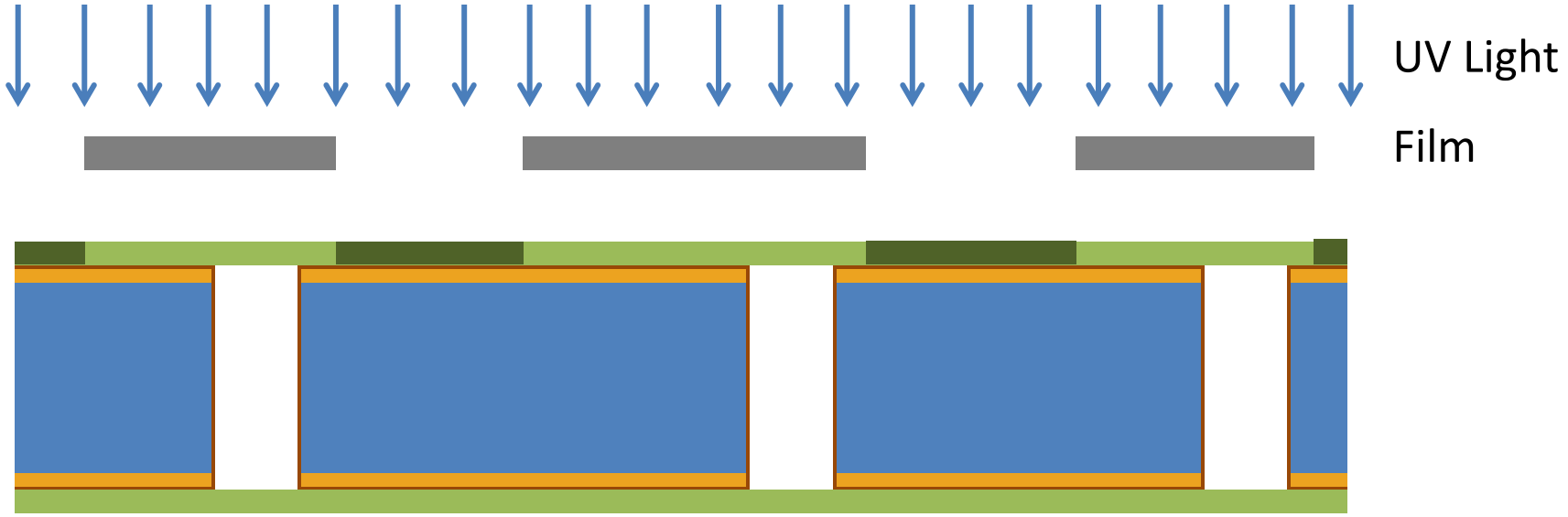
- Deposited copper has a thickness of 0.7 to 1.5 μm
- Accomplished by repeatedly submerging entire board in deposition bath

PHOTO RESIST MATERIAL IS APPLIED TO OUTER LAYERS



- Film covers entire surface, including holes
- Photo resist film hardens when exposed to ultraviolet (UV) light

UV LIGHT HARDENS AREAS WHERE COPPER IS TO BE REMOVED

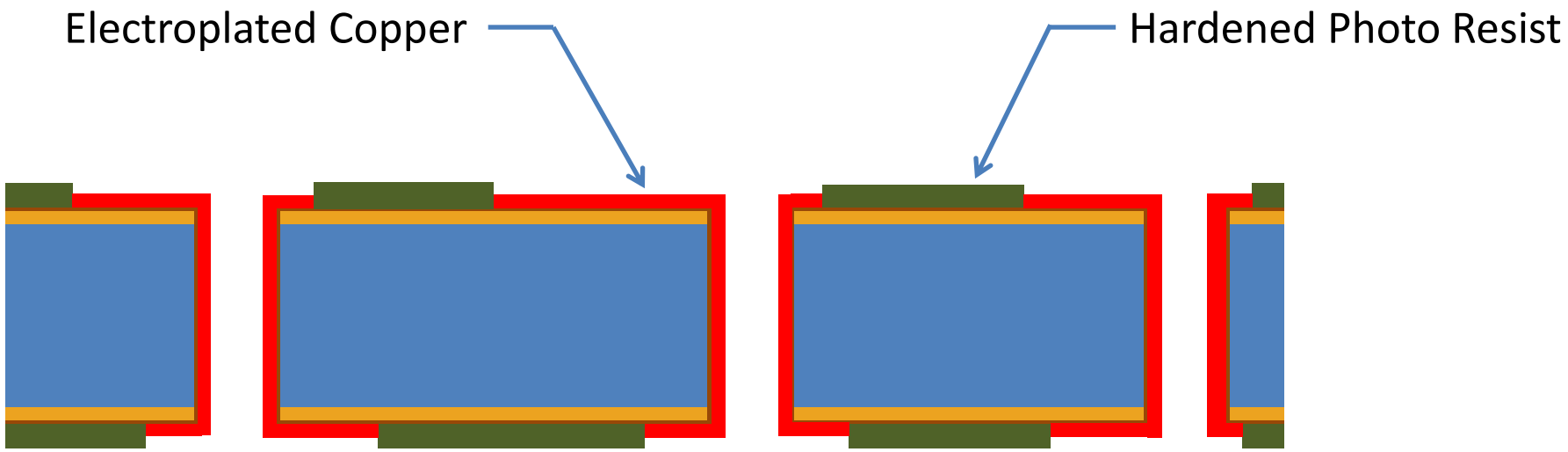


- High intensity UV light polymerizes photo resist
- Exposed regions will eventually be removed
- Both top and bottom layers treated in this manner

UNHARDENED PHOTO RESIST IS CHEMICALLY REMOVED

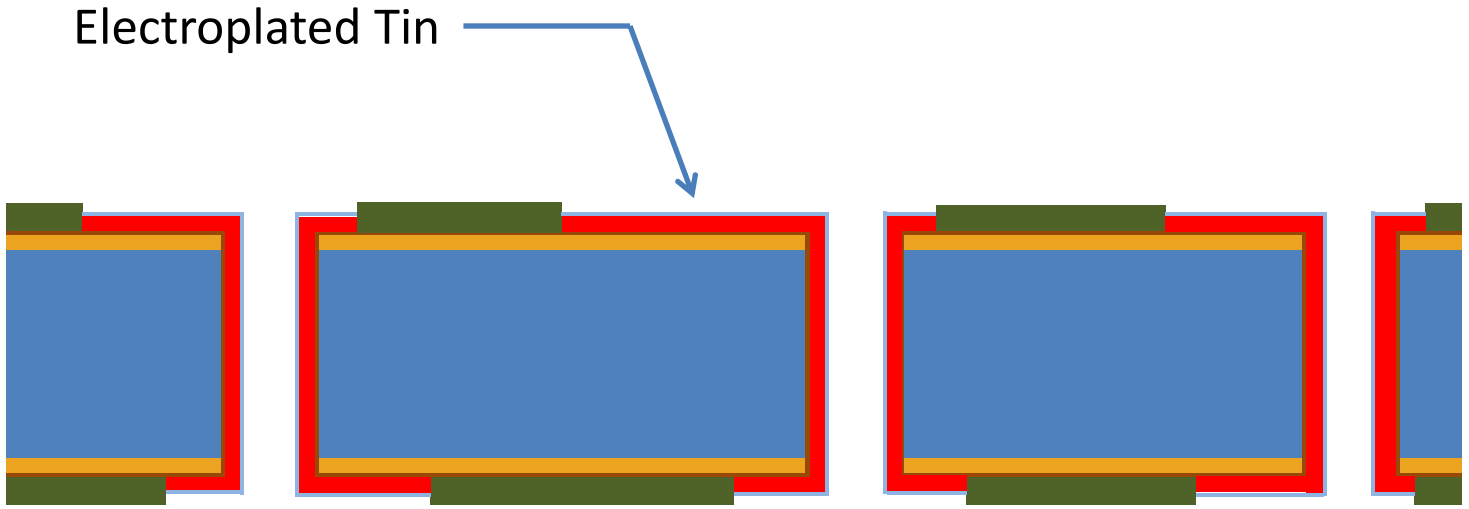
- Regions that will eventually become traces, pads, and other features are now exposed

ELECTROPLATING ADDS THICKNESS TO TRACES AND THROUGH HOLES



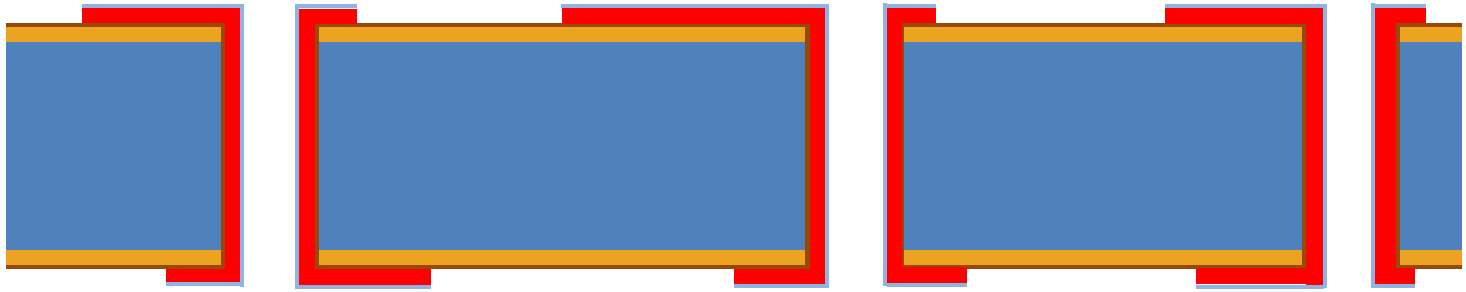
- Approximately 25 μm of additional copper is added to through holes, and 25-30 μm to outer surfaces
- Accomplished by repeated submersion in electroplating baths

TIN IS DEPOSITED IN A THIN LAYER ONTO EXPOSED COPPER



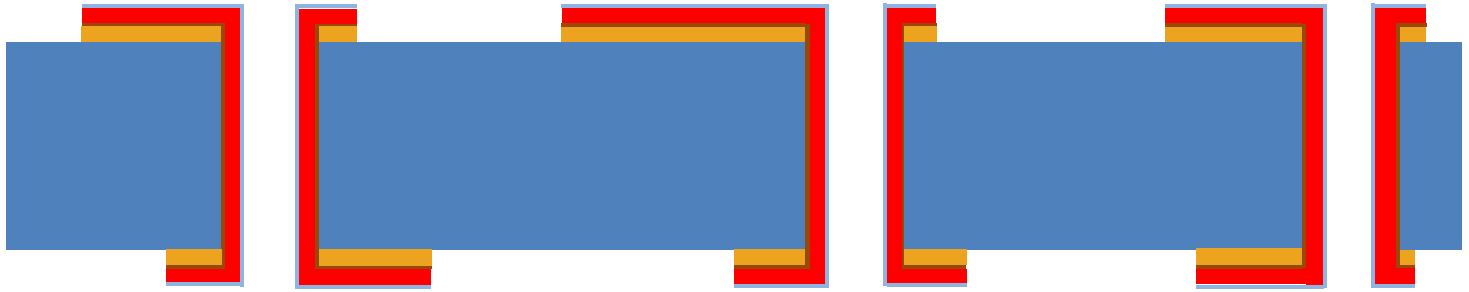
- Tin protects copper traces during etching process to follow

HARDENED PHOTO RESIST IS CHEMICALLY REMOVED



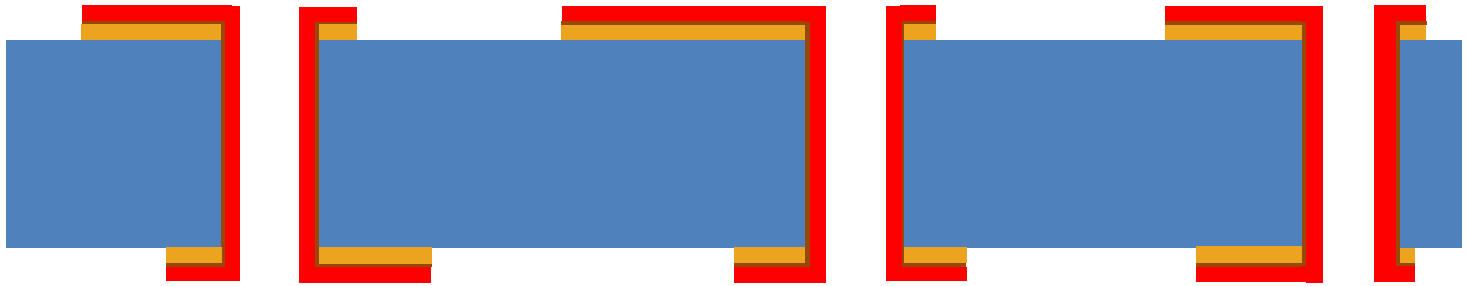
- Copper to be removed is not protected by tin layer

AN ALKALINE SOLUTION IS USED TO ETCH AWAY EXPOSED COPPER



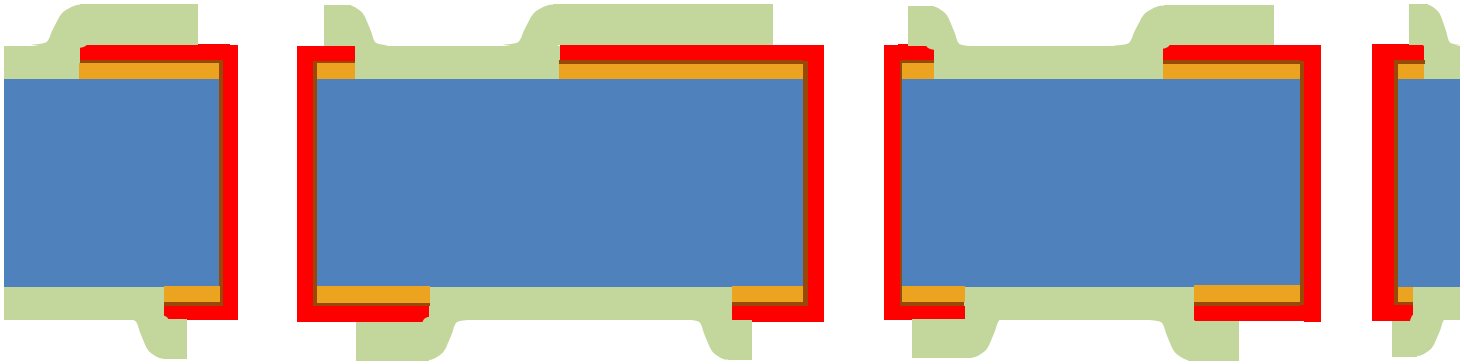
- Core material now exposed between traces

TIN LAYER IS CHEMICALLY REMOVED



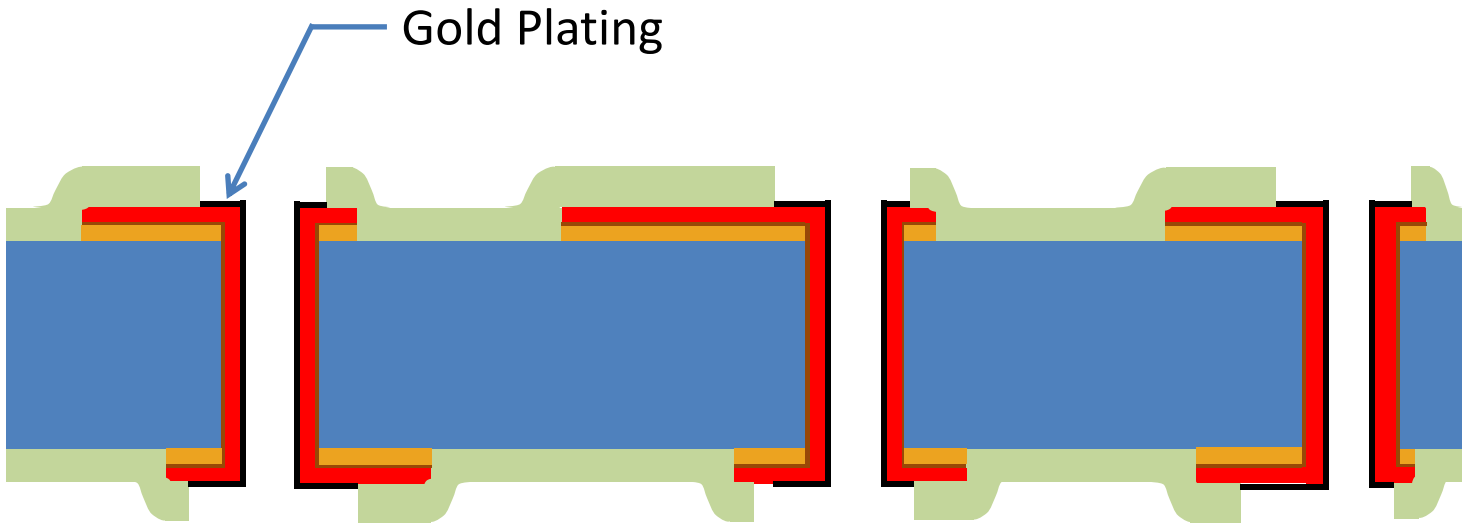
- All copper features in final board now exposed

A SOLDERMASK IS PRINTED ONTO EACH SIDE OF PCB



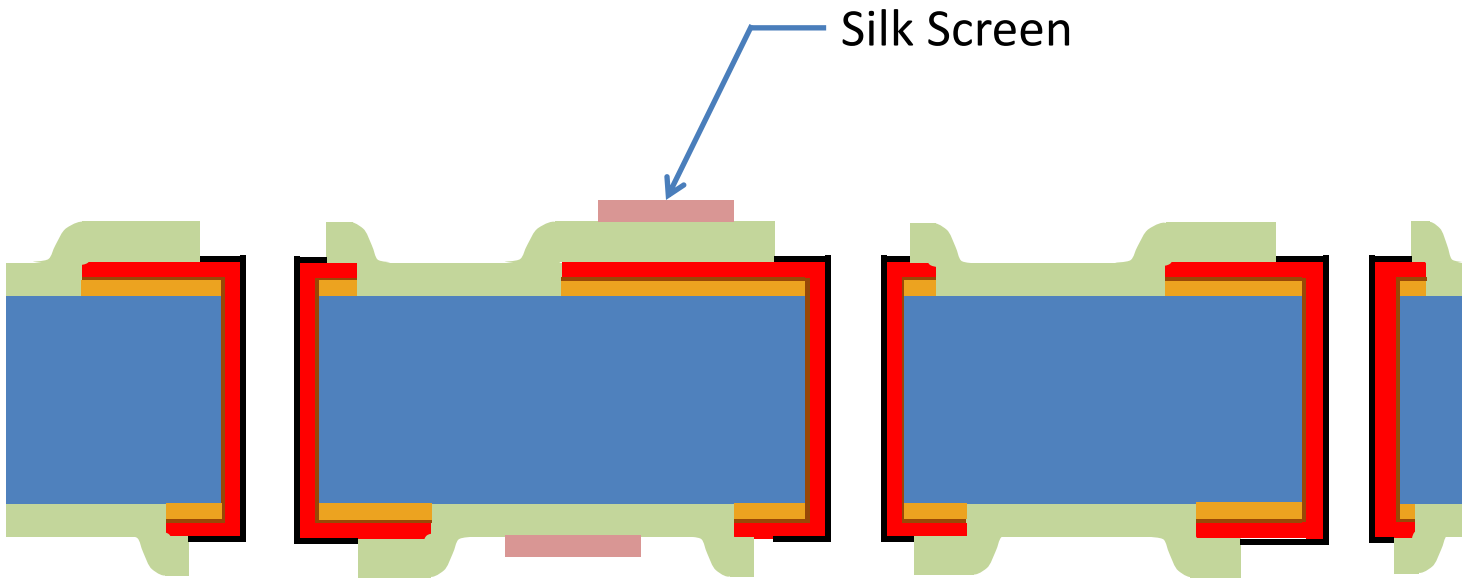
- Solder mask (or just “mask”) keeps copper traces from oxidizing
- Prevents solder from shorting traces or components
- Traditionally green, but other colors becoming popular

ELECTROLESS GOLD PLATING PROTECTS COPPER FROM OXIDATION



- Gold keeps exposed copper from oxidizing
- Improves reliability of soldered joints

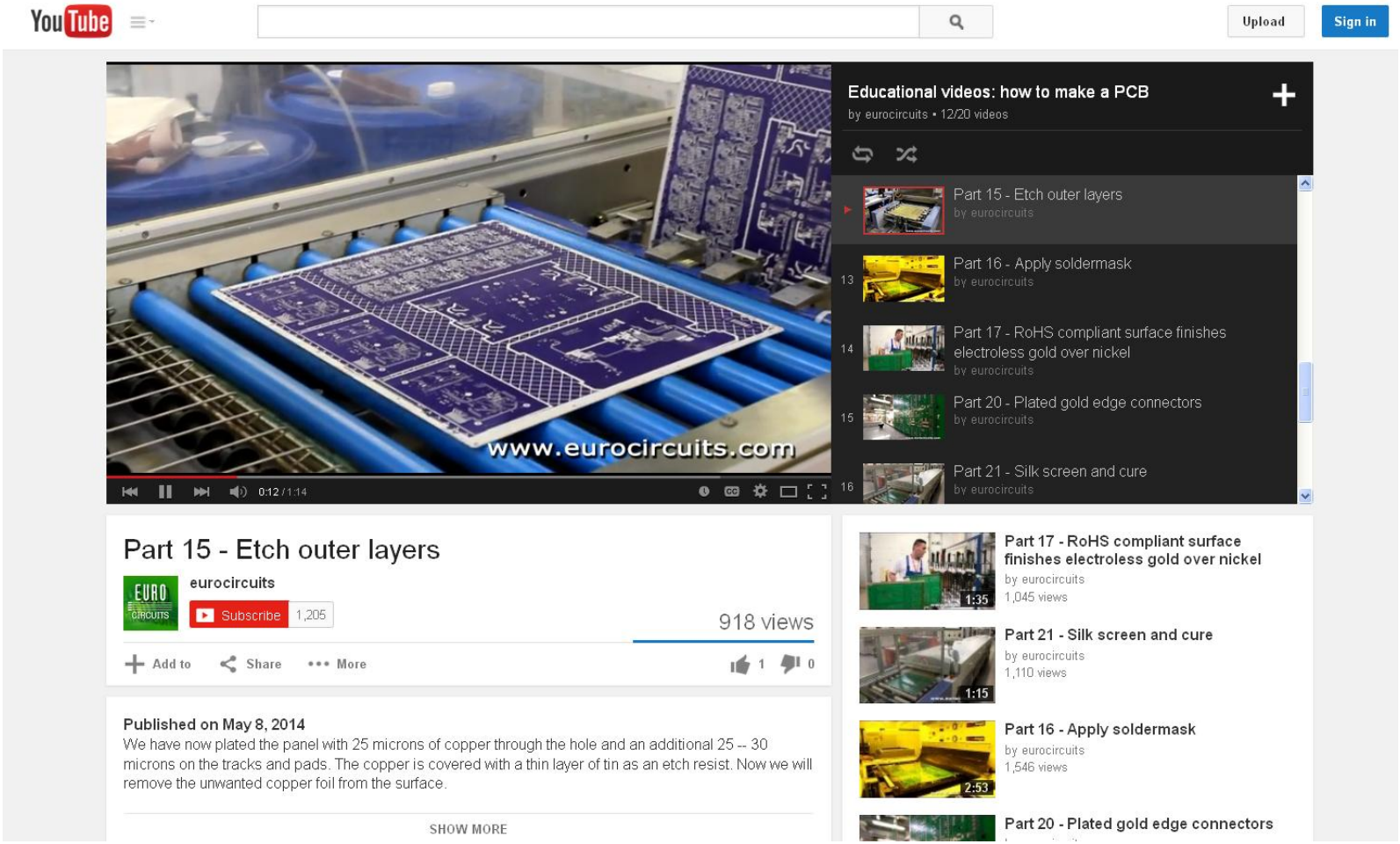
INK JET PRINTER DEPOSITS EPOXY INK TO PROVIDE BOARD MARKINGS



- Also known as “silk screen”
- Usually printed using white ink
- Markings aid in assembly and debug

GOOD VIDEO OVERVIEW

Eurocircuits: How to Make a PCB



CAD PACKAGES AID IN PCB LAYOUT

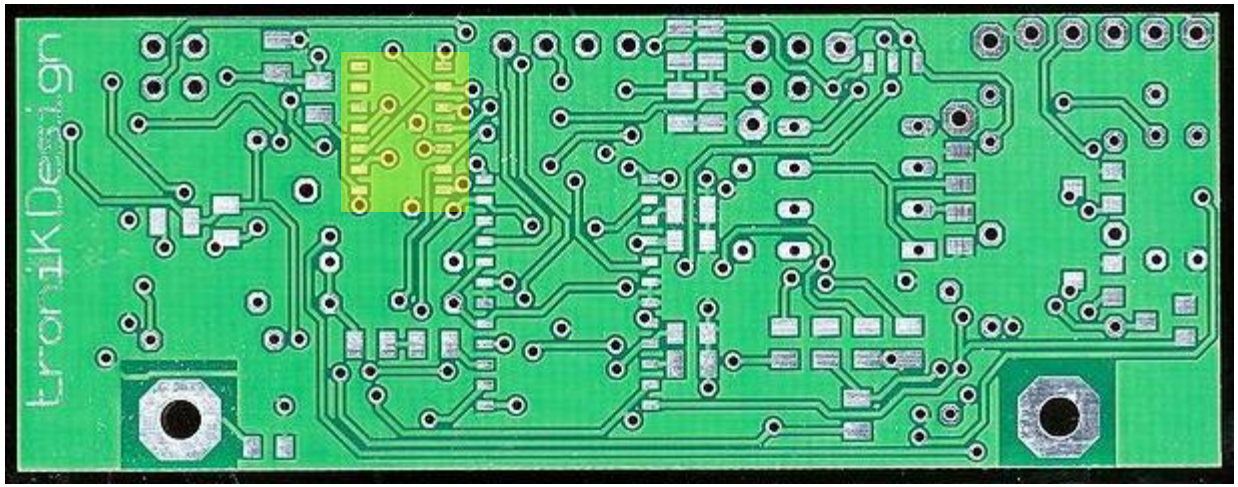
- Eagle
- KiCad
- Fritzing
- DipTrace
- Altium
- many others…

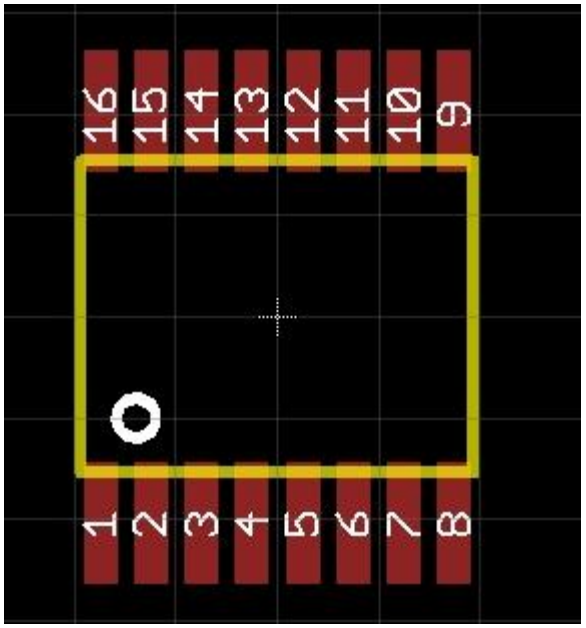
ExpressPCB, PROTEL, CADSTAR, ORCAD, CIRCUIT MAKER, P-CAD 2000, PCB ELEGANCE, EDWIN, VISUALPC, BPECS32, AUTOENGINEER, EXPERT PCB, CIRCAD, LAYOUT, CIRCUIT LAYOUT, MCCAD, DREAM CAD, E-CAD, POWERPCB, PCB ASSISTANT, PCB DESIGNER, QCAD, QUICK ROUTE, TARGET 3001, WIN CIRCUIT 98, BOARD EDITOR, PCB, VUTRAX, CIRCUIT CREATOR, PADSPCB, DESIGN WORKS, OSMOND PPC, LAY01, SCORE, GElectronic, PRO-Board, PRO-Net, CSIEDA, VISUALPCB, WINBOARD, ULTIBOARD, EASY PC, RANGER, PROTEUS, EPD - Electronics Packaging Designer, AutoTrax Eda, SprintLayout, CADINT, Merlin PCB Designer, FREE-PCB, TinyCAD, WINQCAD, Pulsonix

BOARD MANUFACTURERS NEED MULTIPLE FILES

- **Drill File**: Stores drill dimensions and locations
- **Gerber Files**: Stores 2D vector images of various board layers
  - Front and Back Copper (Cu)
  - Front and Back Silk Screen (SilkS)
  - Front and Back Solder Mask (Mask)
  - Board Outline (Edge.Cuts)

FOOTPRINT IS THE CIRCUIT BOARD PATTERN WHERE PARTS ATTACH





- CAD software will need to know footprint dimensions for each component
- Often provided in a component "library"

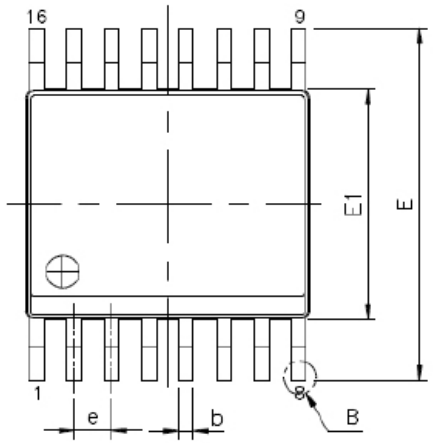


Image: https://learn.sparkfun.com/tutorials/designing-pcbs-smd-footprints

NETLIST DESCRIBES HOW COMPONENTS ARE CONNECTED

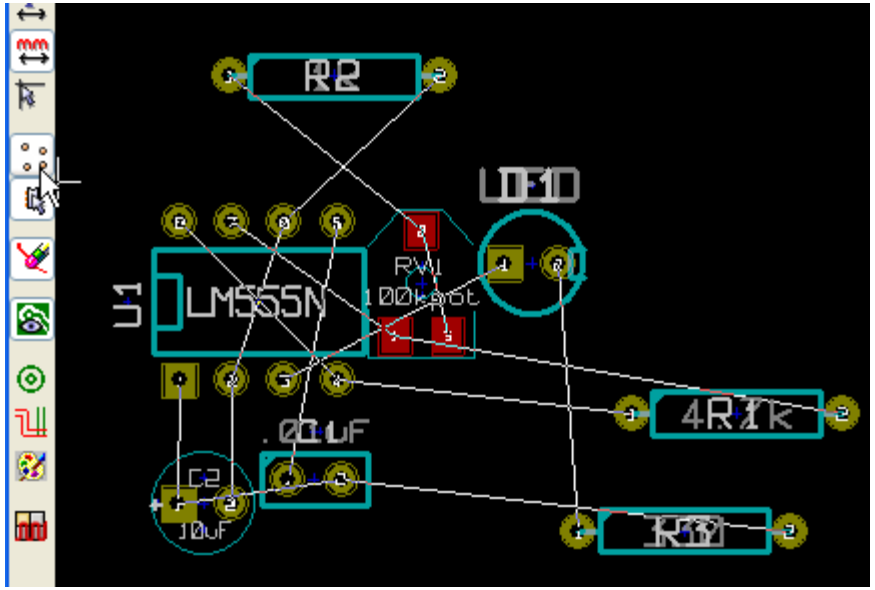
```
Sample Netlist File

Gnd J1-2 J2-3 U1-4 R7-2 C5-2 C6-2
Vgnd U1-3 U1-5 R6-2 R7-1 C5-1 VR1-3
Vcc J2-2 C6-1 U1-8 R6-1 R1-2
Shld J2-4
In J1-1 R1-1 C1-1
Cp1 C1-2 R2-1
Cp2 R2-2 R3-1 C2-1 U1-2
Cp3 R3-2 C2-2 VR1-1 U1-1
Cp4 VR1-2 R4-1
Cp5 R4-2 R5-1 C3-1 U1-6
Cp6 C3-2 R5-2 U1-7 C4-1
Out C4-2 J2-1
```

- Node names on the left can be arbitrarily chosen
- A node is where two or more part pins connect

Image: https://learn.sparkfun.com/tutorials/designing-pcbs-smd-footprints

RAT'S NEST USES STRAIGHT LINES TO SHOW CONNECTED PINS



- Sometimes "messy," but useful in layout

Image: https://learn.sparkfun.com/tutorials/designing-pcbs-smd-footprints

GERBER FILES DESCRIBE 2D SHAPES

- Used to describe copper layers, solder mask, legends, etc.

```
G04 Short version a file taken from the Example Job
1, created by Filip Vermeire, Ucamco*
%TF.FileFunction,Copper,Bot,L4*%
%TF.Part,Single*%
%FSLAX35Y35*%
%MOMM*%
%TA.AperFunction,Conductor,NotC*%
%ADD10C,0.15000*%
%TA.AperFunction,ViaPad*%
%ADD11C,0.75000*%
%TA.AperFunction,ComponentPad*%
%ADD12C,1.60000*%
%ADD13C,1.70000*%
%SRX1Y1I0.00000J0.00000*%
G01*
G75*
%LPD*%
D10*
X7664999Y3689998D02*
X8394995D01*
X8439999Y3734999D01*
X9369999D01*
D11*
X7664999Y3689998D03*
X8359999Y1874998D03*
X9882998Y3650498D03*
D12*
X4602988Y7841488D03*
D13*
X10729976Y2062988D03*
X10983976D03*
X11237976D03*
M02*
```

Sample Code: https://en.wikipedia.org/wiki/Gerber_format

NEXT TIME...

- Using KiCad to layout a PCB

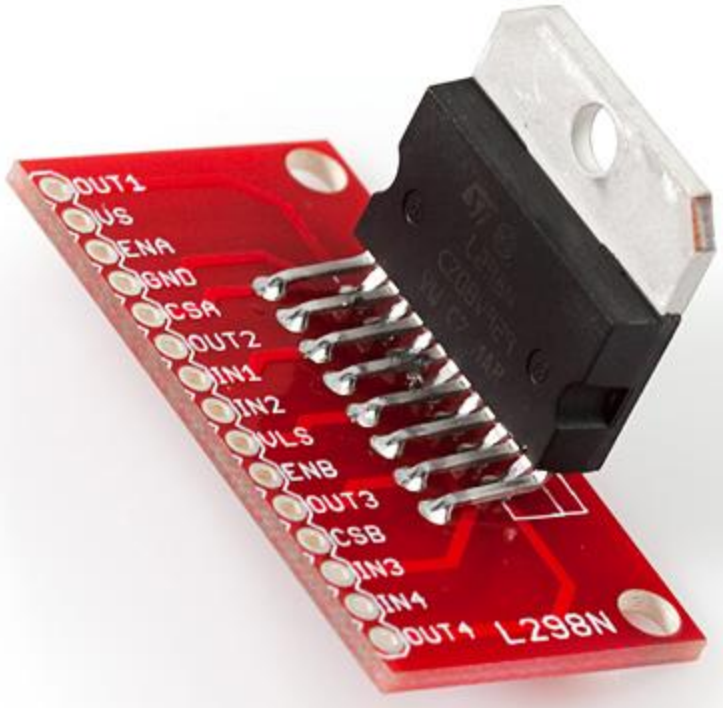


Image: https://www.sparkfun.com/products/9540