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
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
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

Jeff Shelton – 28 January 2015
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
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

SUGGESTED PROTOTYPE STAGES

- Your goal should be to deliver a PCB
- Consider other implementations as intermediate
RIGID PCBs CAN BE MANUFACTURED IN SEVERAL WAYS

- Wet Etching
- Milling
- Silk-screen Printing
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

- Signal Trace
- Silkscreen (Epoxy Ink)
- Plated Through Hole
- Soldermask (Green Layer)
A FIBERGLASS SUBSTRATE (CORE) PROVIDES MECHANICAL STRENGTH

- 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
UNHARDBENED 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 \( \mu m \) of additional copper is added to through holes, and 25-30 \( \mu m \) 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”
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
RAT’S NEST USES STRAIGHT LINES TO SHOW CONNECTED PINS

- Sometimes “messy,” but useful in layout
GERBER FILES DESCRIBE 2D SHAPES

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

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

- Using KiCad to layout a PCB

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