

HARDWARE DESIGN FUNDAMENTALS

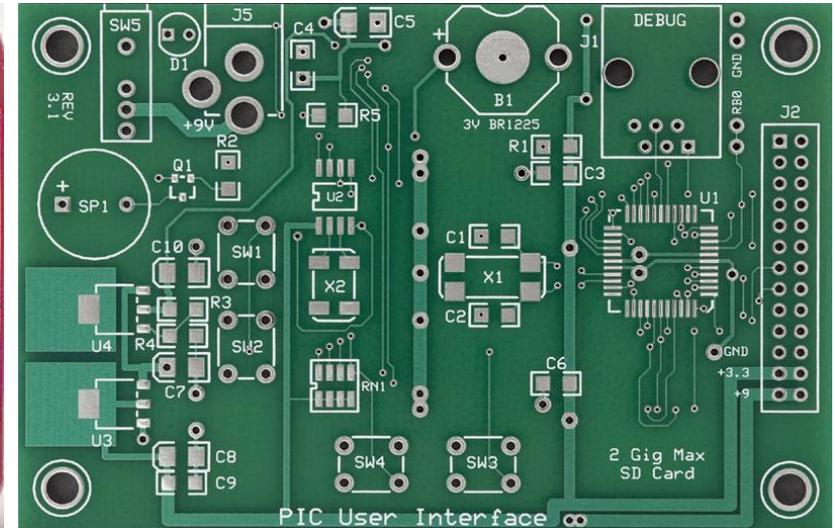
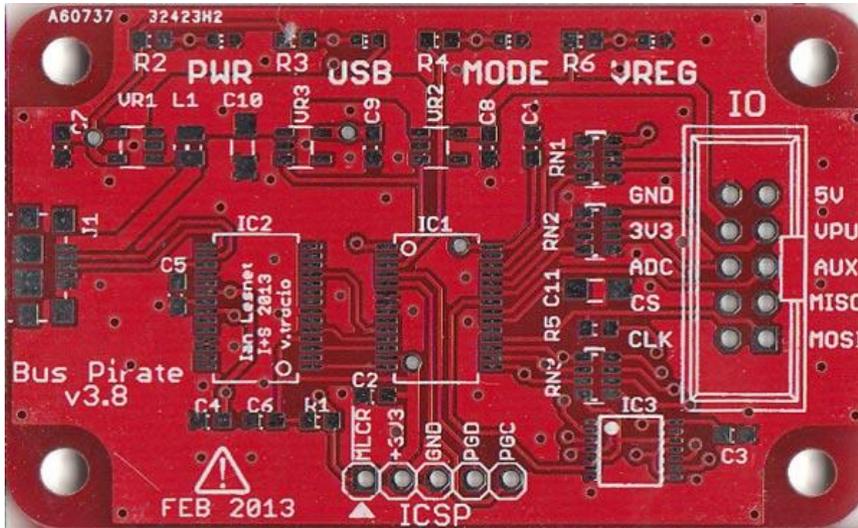


George Hadley ©2014, Images Property of their Respective Owners.

OUTLINE

- PCB Design Objective
- PCB Manufacturing Process
- Design Automation Tools
- The Design Process
 - Parts
 - Schematics
 - Layouts
 - DRC and Toolfile Generation
 - Ordering Circuit Boards
- Miscellaneous PCB Topics

OBJECTIVE



- Q: What's the “goal” of the hardware design process?
- A: Well-designed circuit boards that connect electrical components while meeting electrical requirements (signal integrity, power, etc.)

PCB MANUFACTURING PROCESS

Overview

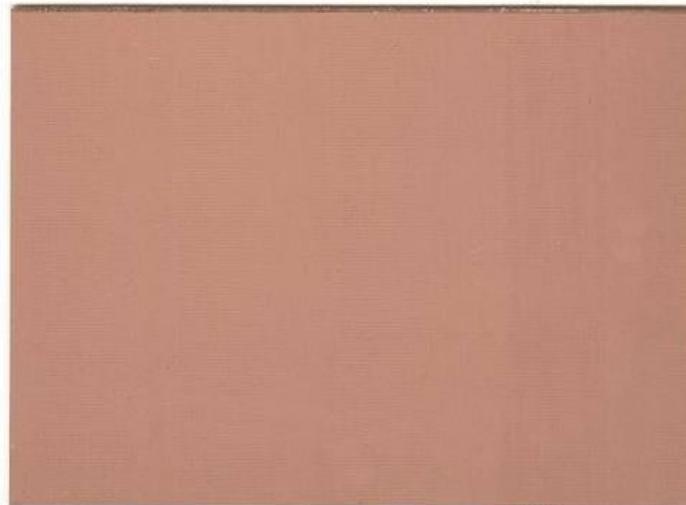
- The printed circuit board (PCB) manufacturing process contains a number of steps:
 1. Base Materials
 2. Etching
 3. Hole/via Drilling
 4. Through-hole plating
 5. Solder mask application
 6. Silkscreen application
 7. Electrical testing



PCB MANUFACTURING PROCESS

Base Materials

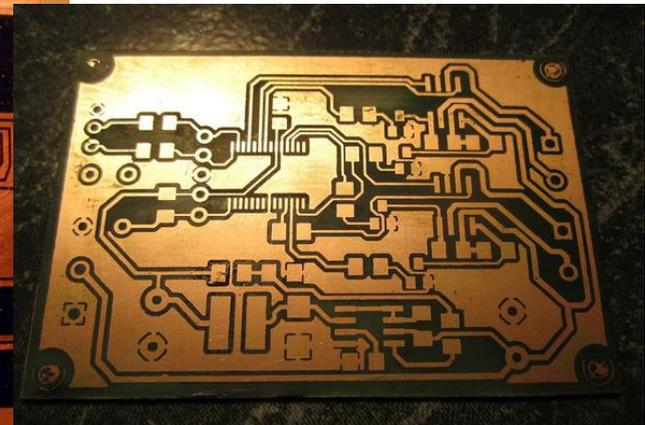
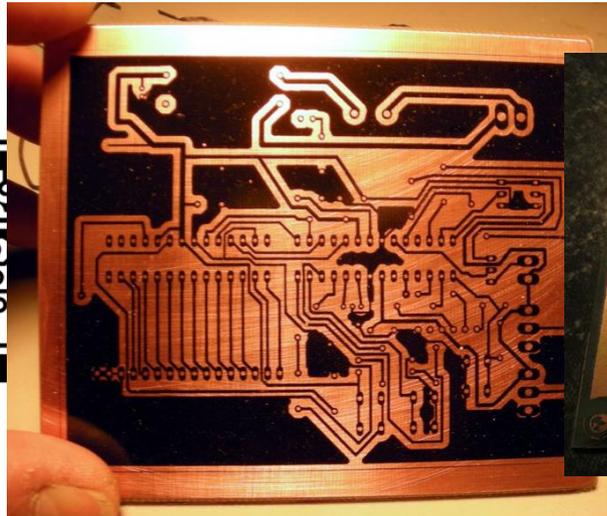
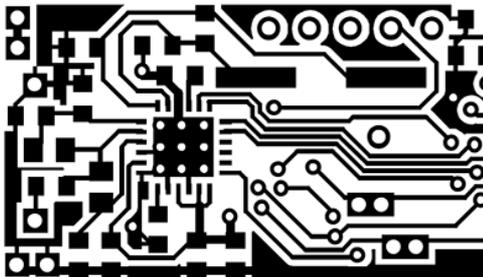
- To make a (simple) circuit board, require:
 - PCB Blank: Fiberglass board (substrate) with copper laminated on both sides
 - Resist: A material which protects the copper on the board from the etchant
 - Etchant: A chemical solution used to dissolve copper on the circuit board



PCB MANUFACTURING PROCESS

Etching

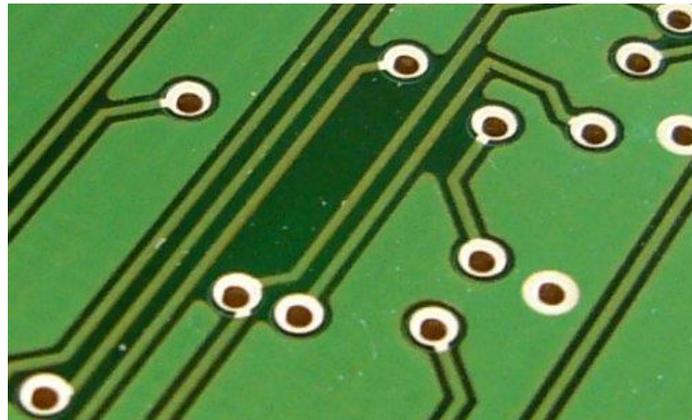
- Resist is applied to the PCB in form of circuit mask (black regions indicate electrical signals/copper)
- PCB is then immersed in etchant solution and allowed to sit. Etchant eats away areas of board not protected by resist.



PCB MANUFACTURING PROCESS

Drilling

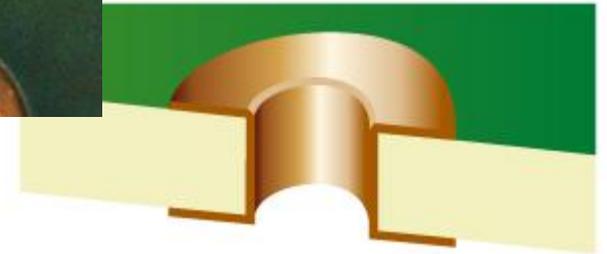
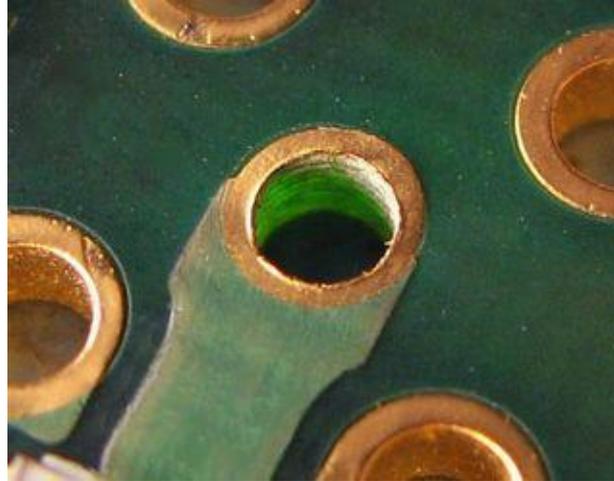
- Once etched, a drill with fine-diameter bits is used for drilling holes (used for mounting hardware, as part of component footprints, etc.)
- Holes used to connect signals between layers of the board are known as **vias**
- To ensure electrical connection is made (even if drill isn't precise), vias possess ring of copper known as an **annular ring**



PCB MANUFACTURING PROCESS

Drilling/Plating

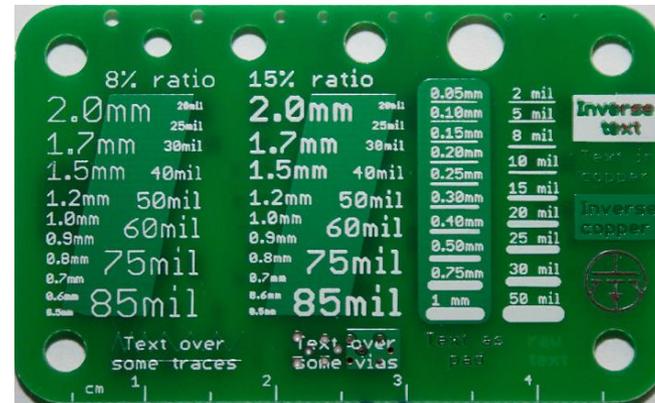
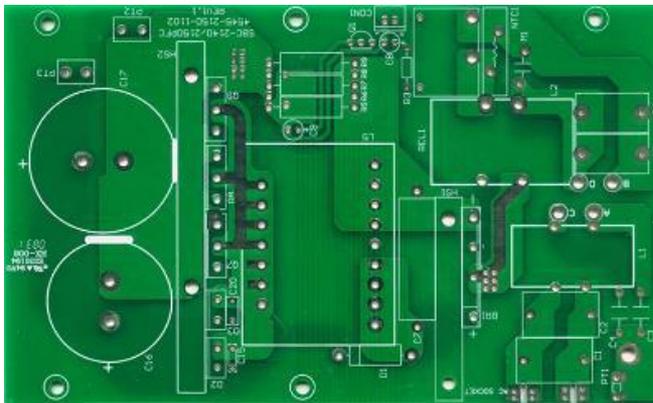
- A drill with fine-diameter bits is used for drilling holes
- Holes used to connect signals between layers of the board are known as **vias** (also used for heat transfer)
- To ensure electrical connection is made (even if drill isn't precise), vias possess ring of copper known as an **annular ring**
- Once drilled, vias are electroplated to ensure conductive path between board layers



PCB MANUFACTURING PROCESS

Soldermask/Silkscreen

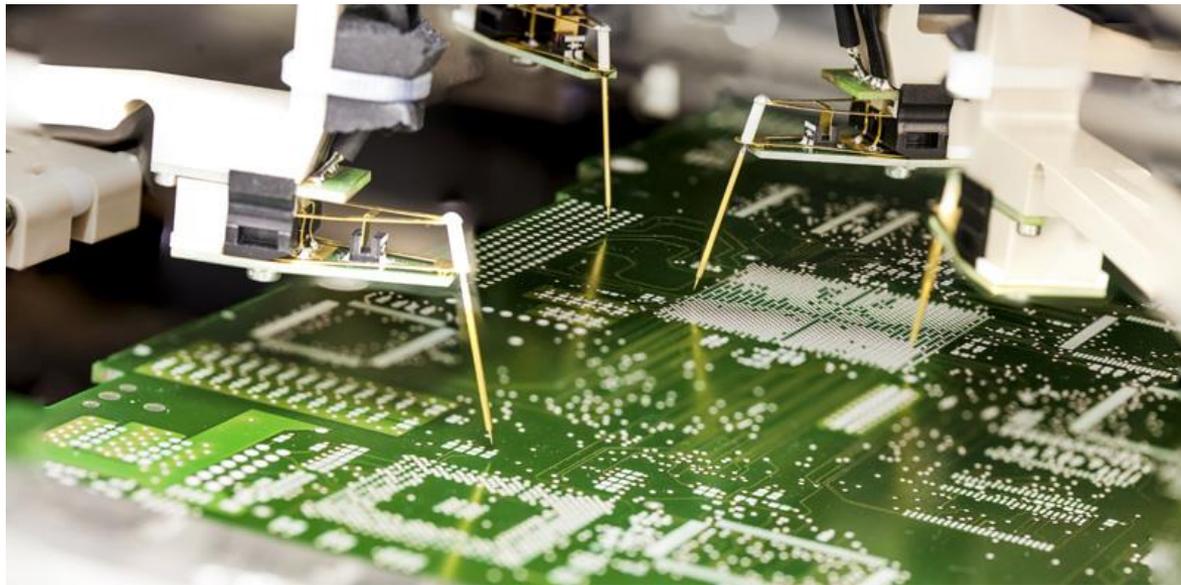
- Soldermask: Overlay material on PCB used to protect metal from corrosion, mitigate short circuits, and ease soldering (applied as a liquid, then cured with UV)
- Several soldermask colors available, though green is most common
- Silkscreen: Layer printed on UAV surface to assist with assembly, usage instructions, and other board information



PCB MANUFACTURING PROCESS

Electrical Testing

- Once finished, boards are subjected to electrical tests to ensure electrical connectivity across boards and that the boards were manufactured correctly
- In spite of such tests, students should carefully visually inspect boards upon reception to check for defects



DESIGN AUTOMATION TOOLS

List of EDA Tool Suites

- Many Electronic Design Automation (EDA) tool suites are available for circuit board design and simulation, such as:
 - Eagle: Easy-to-use, easy to learn, large hobbyist community <http://www.cadsoftusa.com/>
 - Altium: Simulation and advanced features, more professional, expensive <http://www.altium.com/>
 - KiCAD: Unrestricted free and open source software, used by CERN <http://www.kicad-pcb.org/>
 - OrCAD, Allegro, PADS, etc.: Other proprietary suites used in industry (particularly by older, larger companies)

THE DESIGN PROCESS

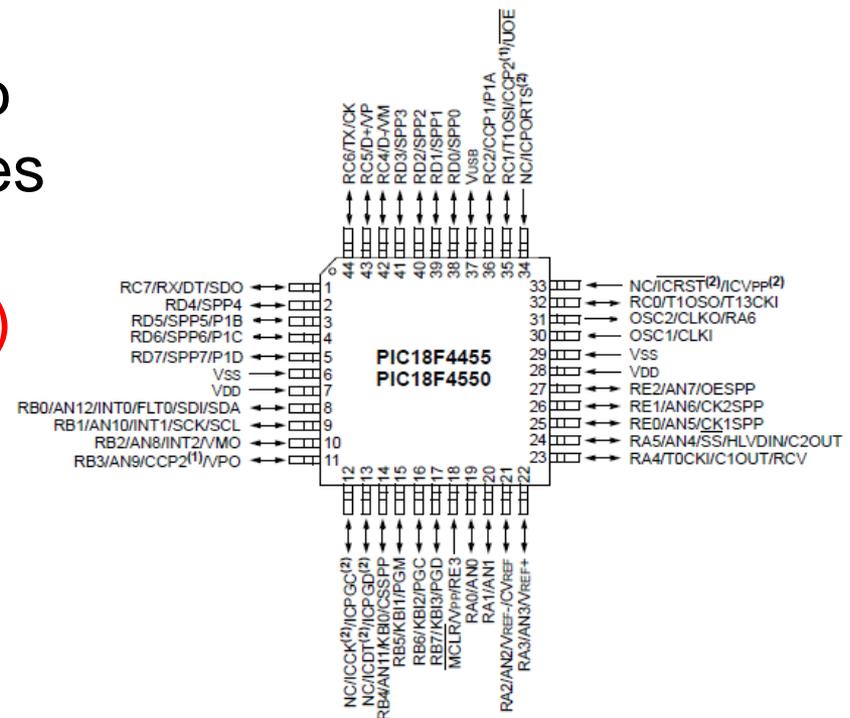
Parts

- In EDA suites such as Eagle, a part is an object which represents an electronic component used in a circuit board. Parts form associations between various views:
 - Symbol: A schematic representation of the component, featuring pins and pin names
 - Footprint: A PCB layout representation of the component, featuring dimensions on board, as well as various layers (copper, silkscreen, documentation, etc.) used in the final PCB
- Part libraries are available in Eagle and online, or users can create their own parts (see tutorial)

THE DESIGN PROCESS

Parts – Tips and Tricks

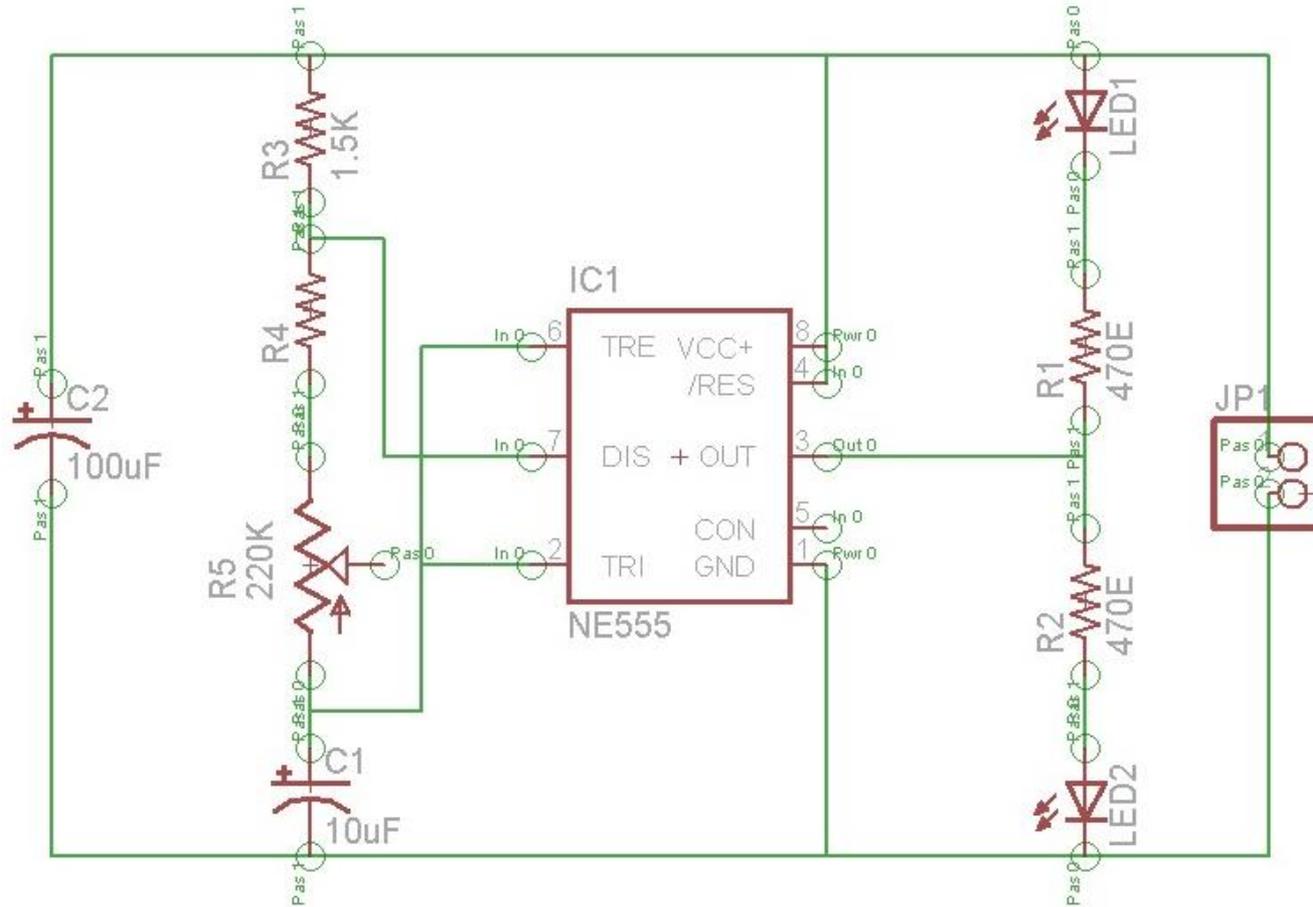
- Many ICs (especially microcontrollers) multiplex many functions onto each pin. Choose only those functions called for in your application to avoid schematic clutter
- Many ICs have multiple pins which have the same name (VCC, GND, NC, etc.). Use `<Name>@n` ($n=1, 2, 3\dots$) to have pins with unique names which display the same in schematic view (**Eagle only**)



THE DESIGN PROCESS

Schematics

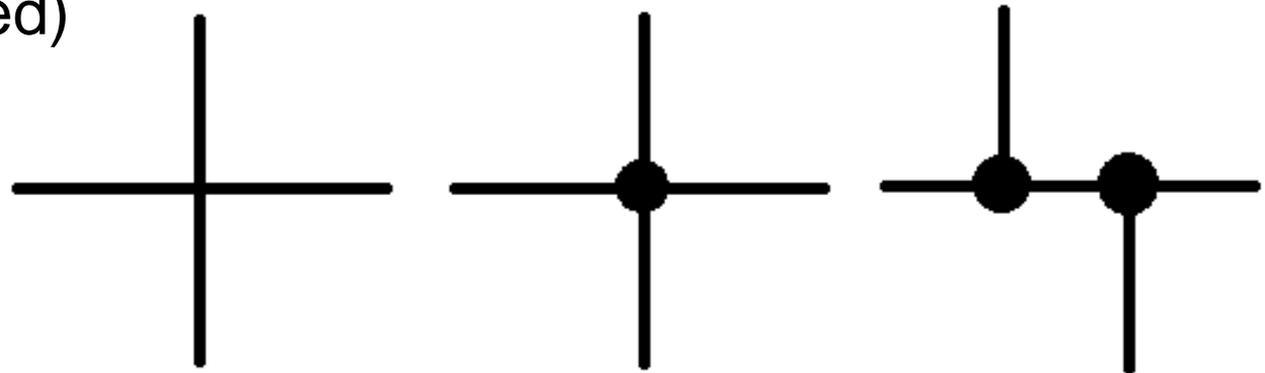
- Schematic: a symbolic representation of a circuit



THE DESIGN PROCESS

Schematics – Tips and Tricks

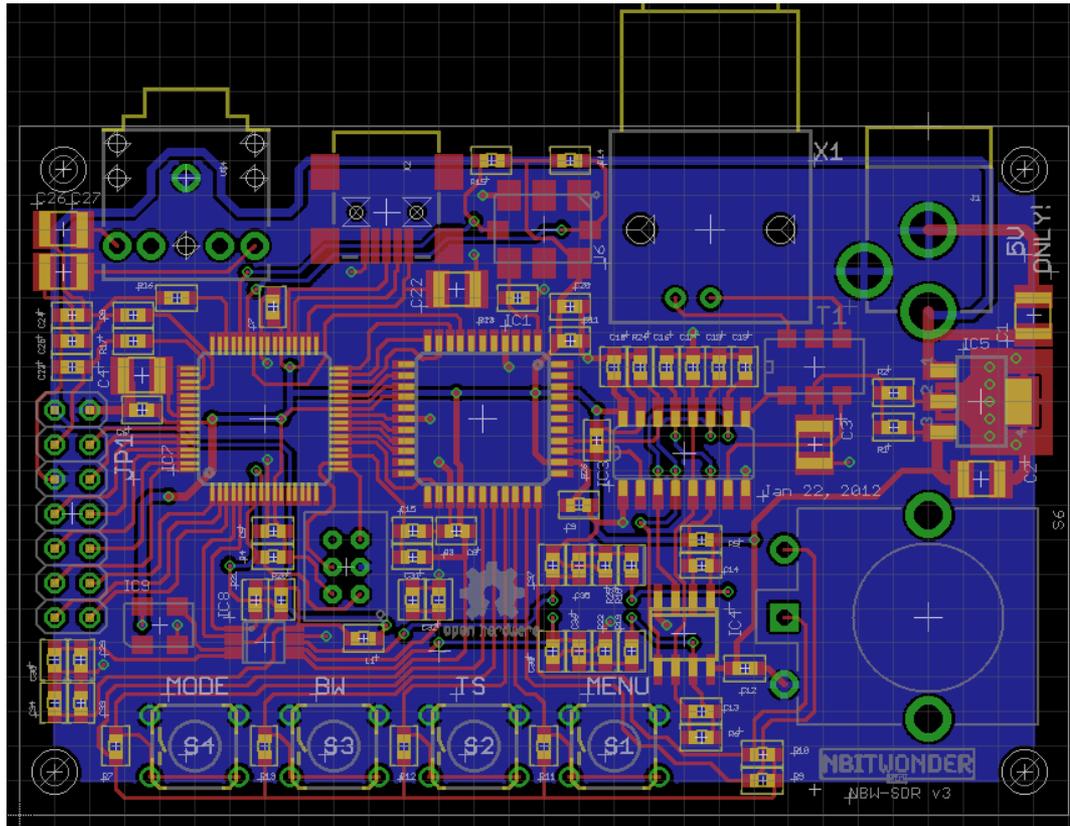
- Electrical Rule Check (ERC): Can be run to check for issues that might escape visual inspection (such as signals that appear to be connected but actually are not)
- Signals can be connected via net name rather than explicit wires to help clean up a schematic
- Junction dots can be used to explicitly define electrical connections (where multiple wires cross dots are strongly recommended)



THE DESIGN PROCESS

Layouts

- Layout: an electrical representation of a circuit board corresponding to the design of the finished board



THE DESIGN PROCESS

Layouts - Terminology

- Pin: A plated through hole used to connect the terminal of a part
- Pad: A flat conductive surface for connecting the terminal of a surface-mount part
- Via: A plated through hole used to route signals between layers of a circuit board
- Trace: A wire or 1-dimensional electrical connection
- Signal Plane: A 2-dimensional electrical connection (commonly used for signals such as power and ground)
- Mil (milli-inch): 1 mil = 0.001in.

THE DESIGN PROCESS

Layouts – Fabrication Tolerances

- Drills: 20 mil (min)
Tolerance: ± 5 mils diameter, ± 5 mil center
- Layer-to-layer alignment: ± 3 mils
- Etched feature size: ± 1 mil (min)
- Isolated trace size: 6 mil (min) (≥ 8 mil recommended)
- Solder mask size: ± 3 mil (min)
- Silkscreen size: ± 10 mil (min)

- Fabrication tolerances typically stored in Eagle design rule check (DRC) files

THE DESIGN PROCESS

Layouts – General Layout Guidelines

- Recommended trace/space: 10-16mil (general)
- Power and ground traces should be sized for current being passed (trace width current capacity charts available online)
- Follow all manufacturer PCB layout recommendations
- Decoupling capacitors should be placed as close to associated ICs as possible
- Provide space and mechanical support for connectors, heat sinks, and standoffs
- Incorporate headers or vias for verification and debugging

THE DESIGN PROCESS

Design Rule Check (DRC) and Tool File Generation

- Once a layout has been completed, it must be checked to ensure it can be manufactured by the board house. This is done by running a Design Rule Check (DRC)
- Once a design has been refined and passes DRC, a software tool (CAM processor, in the case of Eagle) must be run to generate the files used by the board house tools to assemble the boards.
- The industry standard for PCB tool files is the Gerber standard (RS-274-D). One file is produced for each layer of the board (top/bottom copper, top/bottom silkscreen, top/bottom soldermask, drills, etc.). Gerber files can be viewed using a Gerber viewer

THE DESIGN PROCESS

Ordering Circuit Boards

- Gerber files necessary to produce a board are compressed into a zip archive, and sent out to a PCB service. Some popular PCB services:
 - Advanced Circuits: Fast turnaround times, US-based, ITAR <http://www.4pcb.com/4pcb-monthly-specials.html>
 - OSH Park: PCB panelization service, 3 board copies per design submitted, US-based <https://oshpark.com/>
 - Seeed Studio: Low cost, China-based, longer lead times (~4 weeks), other services available (3D printing, laser cutting, stencils, etc.) <http://www.seeedstudio.com/>
 - Other Services: Silver Circuits, PCB Pool, Beta PCB, Express PCB, etc.

THE DESIGN PROCESS

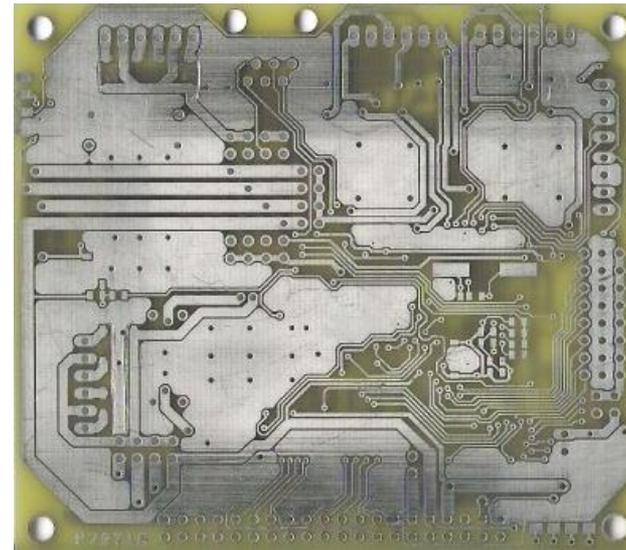
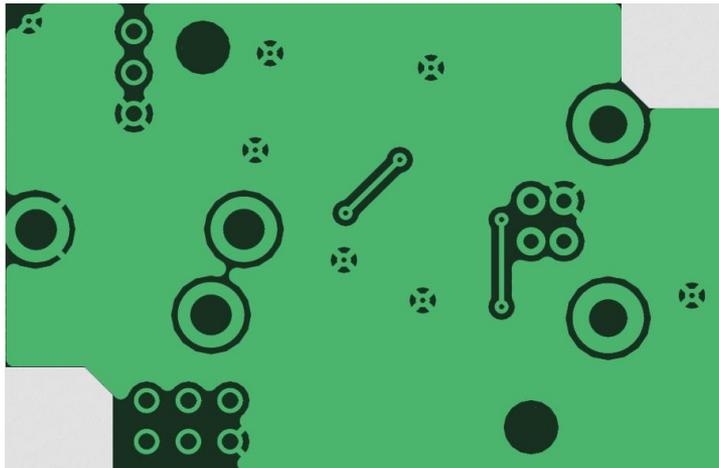
Ordering Circuit Boards 2

- Depending on timeframe and features, circuit boards can be fairly cheap or incredibly expensive. Consider:
 - Turn Time: Time needed to manufacture (“turn”) the board. 1 day turns can cost hundreds of dollars. 1-2 week turns will cost much, much less
 - Shipping Time: Time needed to ship the boards. Shorter shipping times bring higher costs.
 - Custom Tooling: Features such as custom cutouts, board shapes, scoring or other cutting can quickly increase the cost of a board
 - Quantity: Major cost in PCB production is tooling. Once that cost has been paid additional boards are quite cheap

MISCELLANEOUS PCB TOPICS

Signal Planes

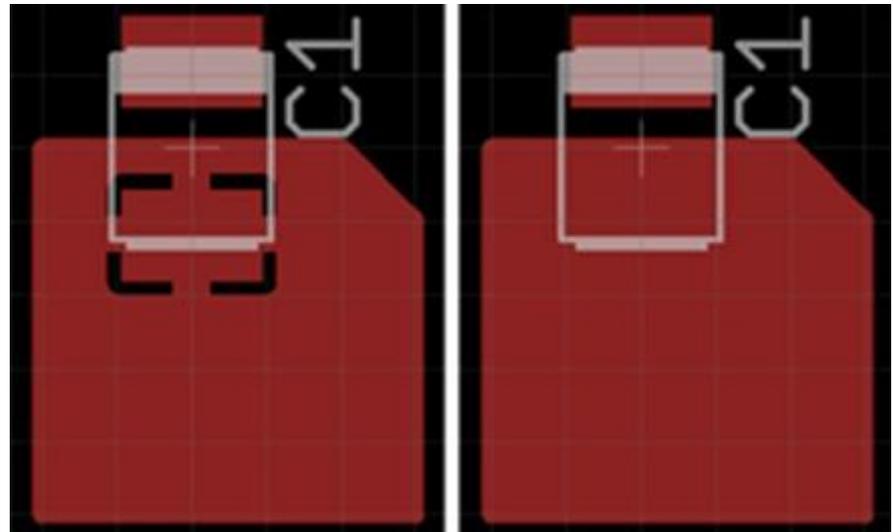
- Useful for creating large, low resistance signal paths
- Allow easy connection for common signals (e.g. GND)
- Saves copper (unused copper on a circuit board must be etched away)
- Can be used as PCB heatsinks (reduce or eliminate the need for external, dedicated heatsinks)



MISCELLANEOUS PCB TOPICS

Thermal Relief

- Electronic components must be soldered to boards; in order to solder (or desolder) a part, the copper of a component pin/pad must be raised to a sufficient temperature
- If a pin/pad is given a direct connection to a thick trace or large signal plane, the trace or plane may act as a heatsink, preventing the pin/pad from reaching soldering temperature
- Thermal Relief: Isolate a pin/pad from direct connection with large trace/plane (heatsink)



MISCELLANEOUS PCB TOPICS

Silkscreen Topics

- Every silkscreen layer should include:
 - Name of author(s) (ECE477 team members)
 - Name to identify the circuit board
 - Date last modified or revision number
- Placing component values in silkscreen may be unnecessary; consider instead giving components IDs (Rn, Cn, Ln, ICn, Dn, Sn, etc.) and using table of values
- Custom silkscreen images can enable addition of custom logos and image to circuit boards



MISCELLANEOUS PCB TOPICS

Other Miscellaneous Topics

- Tented Vias: Design the board to allow the solder mask to run over the top of a via (provides unbroken surface for more legible silkscreen, etc.)
- Polar Layouts: For use in circular designs

