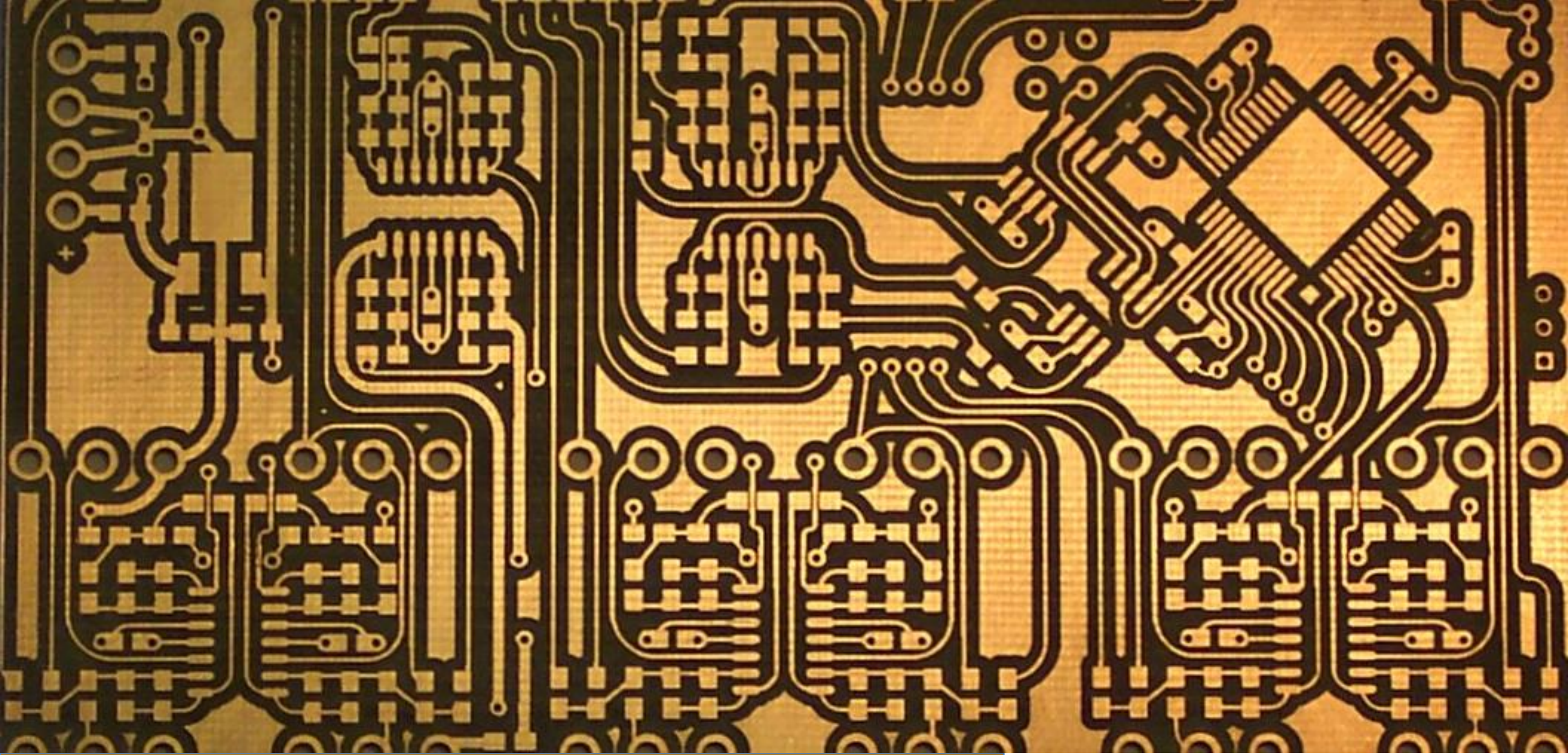


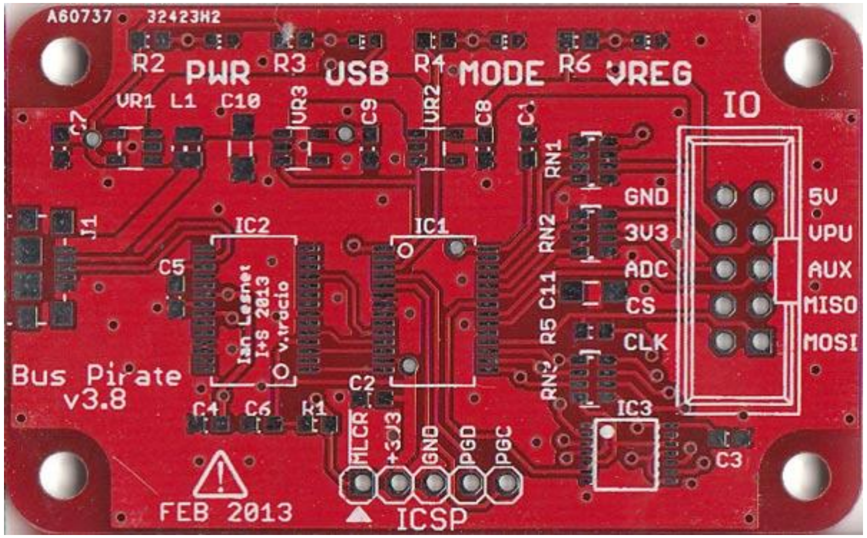
HARDWARE DESIGN FUNDAMENTALS

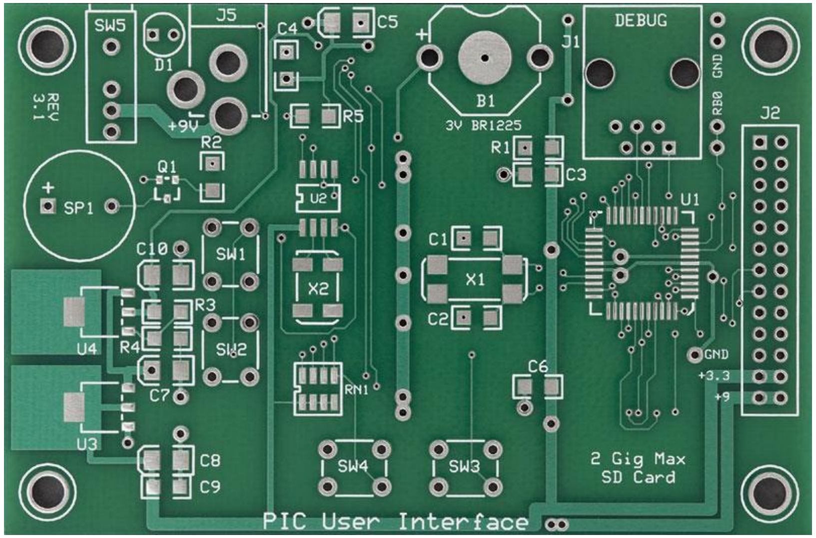
PURDUE UNIVERSITY

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

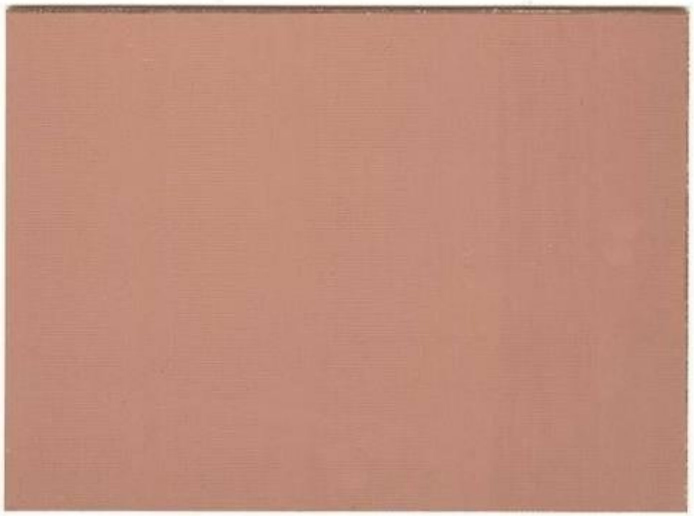
- The printed circuit board (PCB) manufacturing process contains a number of steps:
  1. Base Materials
  2. Etching
  3. Hole/via Drilling
  4. Through-hole plating
  5. Solder mask application
  6. Silkscreen application
  7. Electrical testing

PURDUE
UNIVERSITY

PCB MANUFACTURING PROCESS

Base Materials

- To make a (simple) circuit board, require:
  - PCB Blank: Substrate board (typically FR4) 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

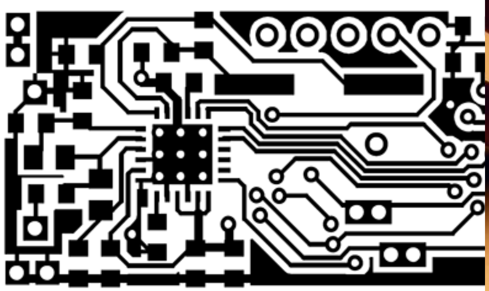


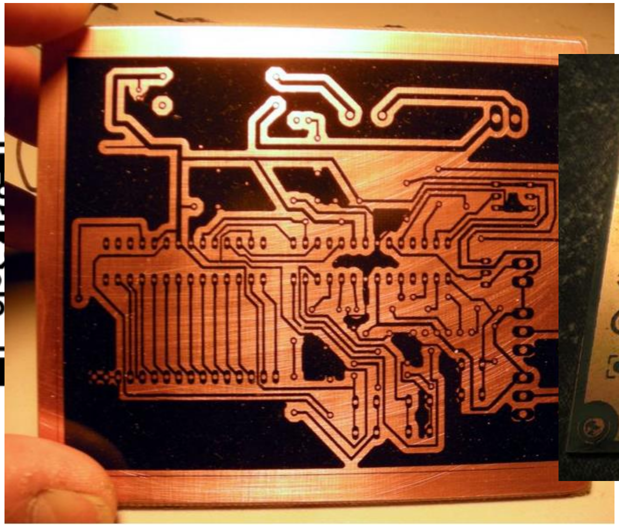
PURDUE
UNIVERSITY

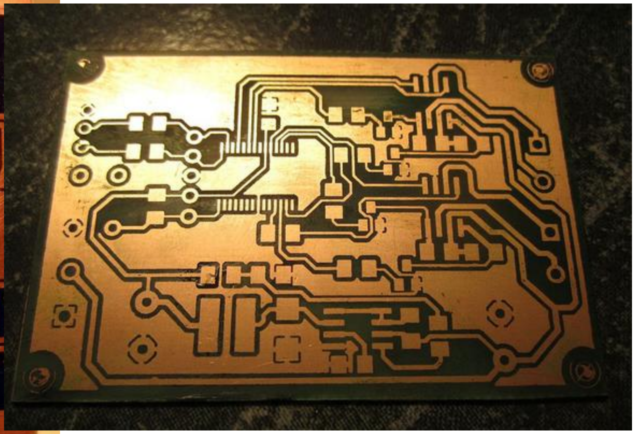
PCB MANUFACTURING PROCESS

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.



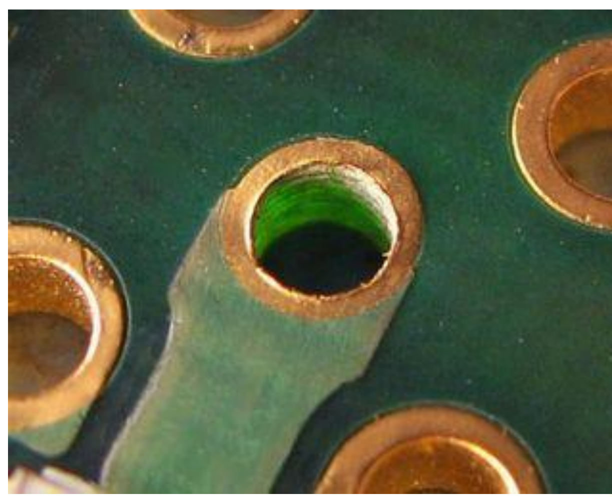




PCB MANUFACTURING PROCESS

Drilling/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

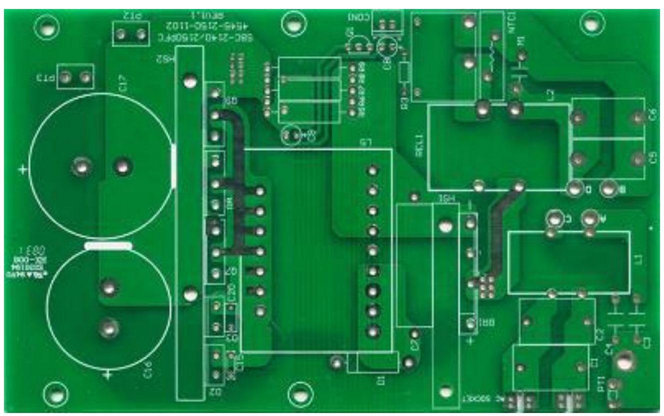


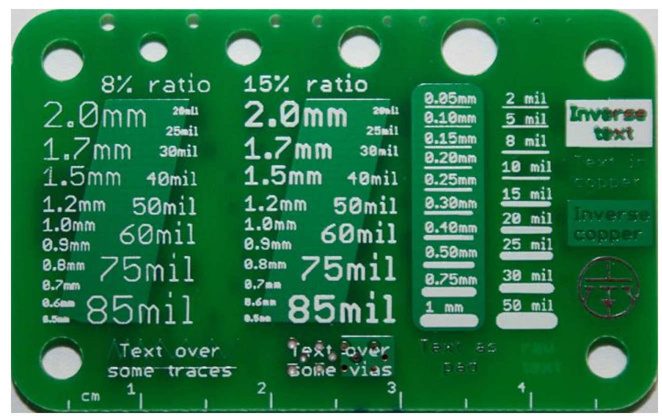
PURDUE
UNIVERSITY

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 generally most common
- Silkscreen: Layer printed on surface to assist with assembly, usage instructions, and other board information

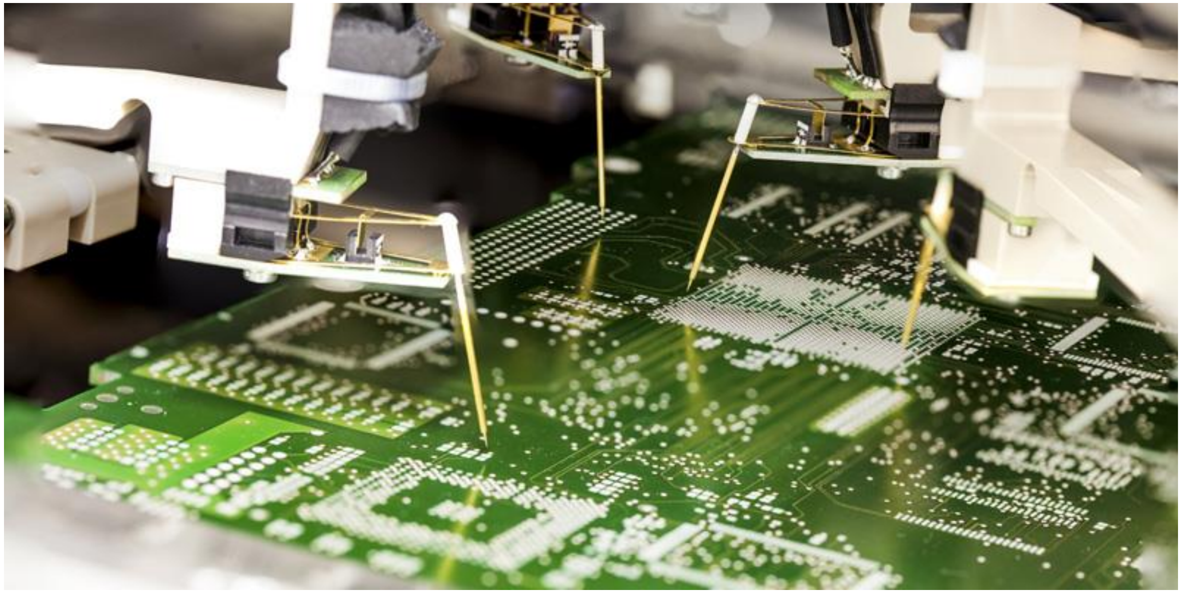




PCB MANUFACTURING PROCESS

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



PURDUE UNIVERSITY

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 https://www.autodesk.com/products/eagle/overview
  - 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

PURDUE UNIVERSITY

THE DESIGN PROCESS

Parts

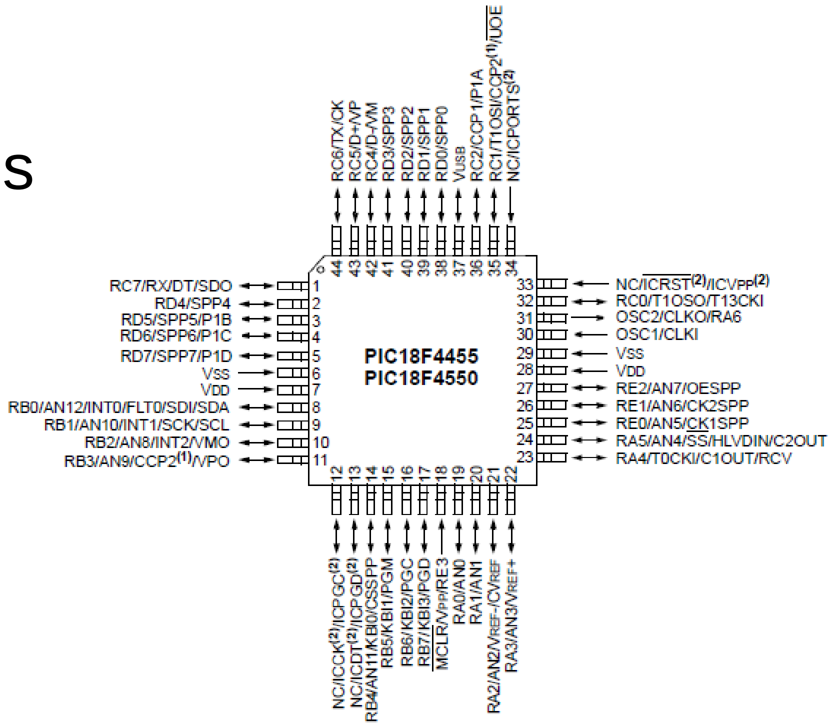
- In EDA suites such as Eagle, a part is an object which represents an electronic component used in a circuit board. Parts form associations between various views:
  - Symbol: A schematic representation of the component, featuring pins and pin names
  - Footprint: A PCB layout representation of the component, 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)

PURDUE UNIVERSITY

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…) to have pins with unique names which display the same in schematic view (Eagle only)

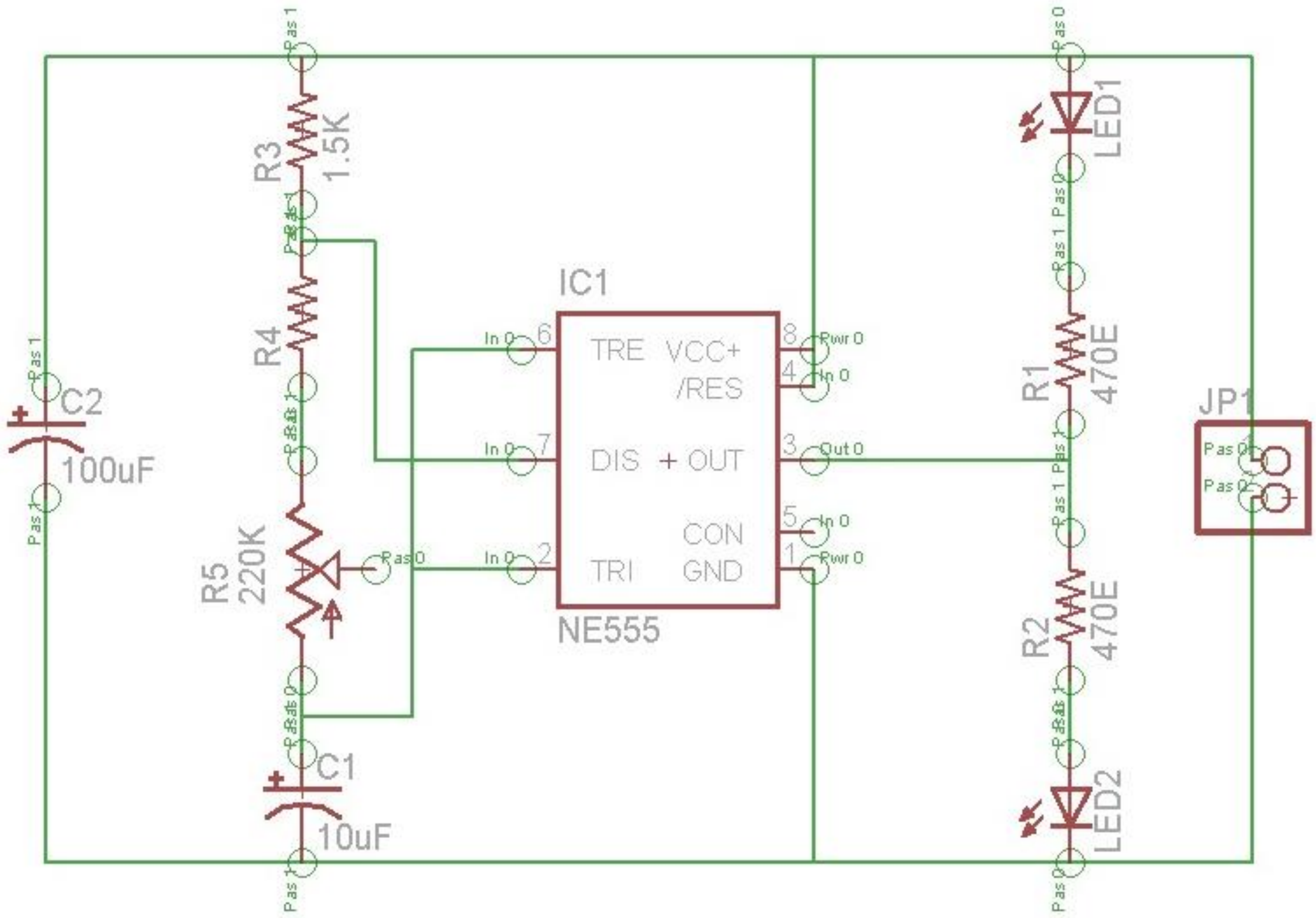


PURDUE
UNIVERSITY

THE DESIGN PROCESS

Schematics

- <u>Schematic:</u> a symbolic representation of a circuit

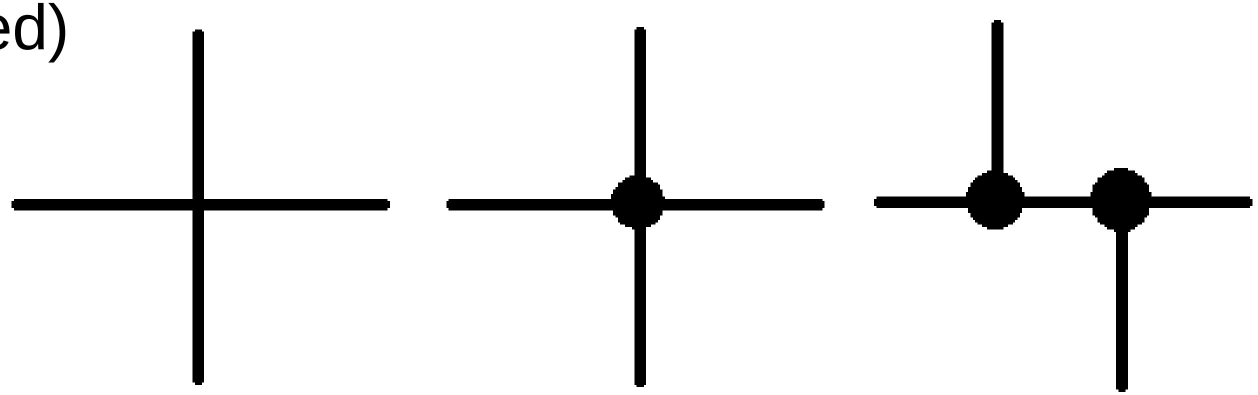


PURDUE
UNIVERSITY

THE DESIGN PROCESS

Schematics – Tips and Tricks

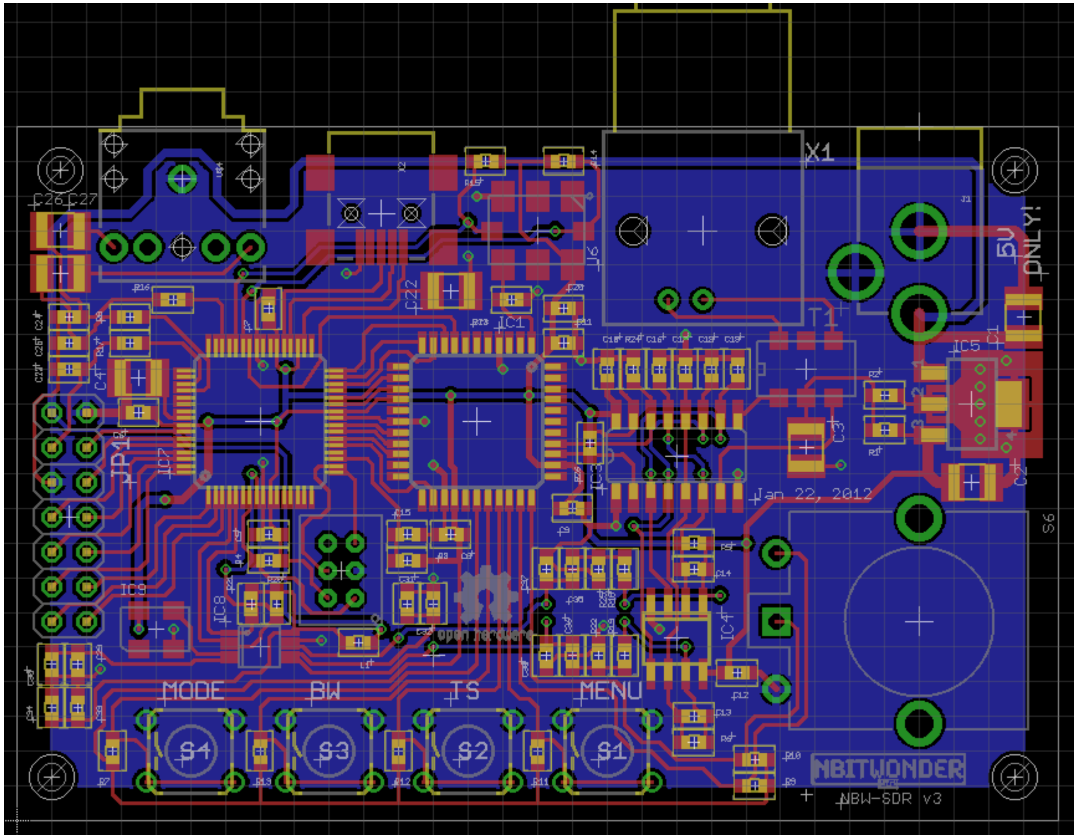
- Electrical Rule Check (ERC): Can be run to check for issues that might escape visual inspection (such as signals that appear to be connected but actually are not)
- Signals can be connected via net name rather than explicit wires to help clean up a schematic
- Junction dots can be used to explicitly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



THE DESIGN PROCESS

Layouts

- <u>Layout:</u> an electrical representation of a circuit board corresponding to the design of the finished board



PURDUE UNIVERSITY

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.

PURDUE
UNIVERSITY

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-X). 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

PURDUE
UNIVERSITY

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/
  - Various PCB services can be compared at http://pcbshopper.com

PURDUE
UNIVERSITY

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

Questions?