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
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.)
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**: 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
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.
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

[Image of PCBs showing soldermask and silkscreen]
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.
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](https://www.autodesk.com/products/eagle/overview)

- **Altium**: Simulation and advanced features, more professional, expensive [http://www.altium.com/](http://www.altium.com/)

- **KiCAD**: Unrestricted free and open source software, used by CERN [http://www.kicad-pcb.org/](http://www.kicad-pcb.org/)

- **OrCAD, Allegro, PADS, etc.**: Other proprietary suites used in industry
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)
• 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).
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-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
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:
  - **OSH Park**: PCB panelization service, 3 board copies per design submitted, US-based [https://oshpark.com/](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/](http://www.seeedstudio.com/)
  - Various PCB services can be compared at [http://pcbshopper.com](http://pcbshopper.com)
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?