

Printed Circuit Boards (PCBs)

Part I

Outline

- Motivation
- Terminology
- Board Structure
- Development Process
- Getting Ready: What You'll Need
- Manufacturing Methods
 - Milled PCBs
 - Professionally Fabricated PCBs
 - In-house Etched PCBs
- Tolerances

Motivation

- The objective is to develop a prototype that is closer to a “final product”
- Printed Circuit Boards (PCBs) are used in pretty much every mass-produced electronic item

Terminology

Gerber: a file format that is typically used to communicate a board layout to a manufacturer. A full submission consists of a set of files – one for each layer of the board.

Excellon: a file format that is used to communicate the drill sizes and hole locations used on a board.

Terminology

RS-274X: a particular sub-format of the Gerber file format.

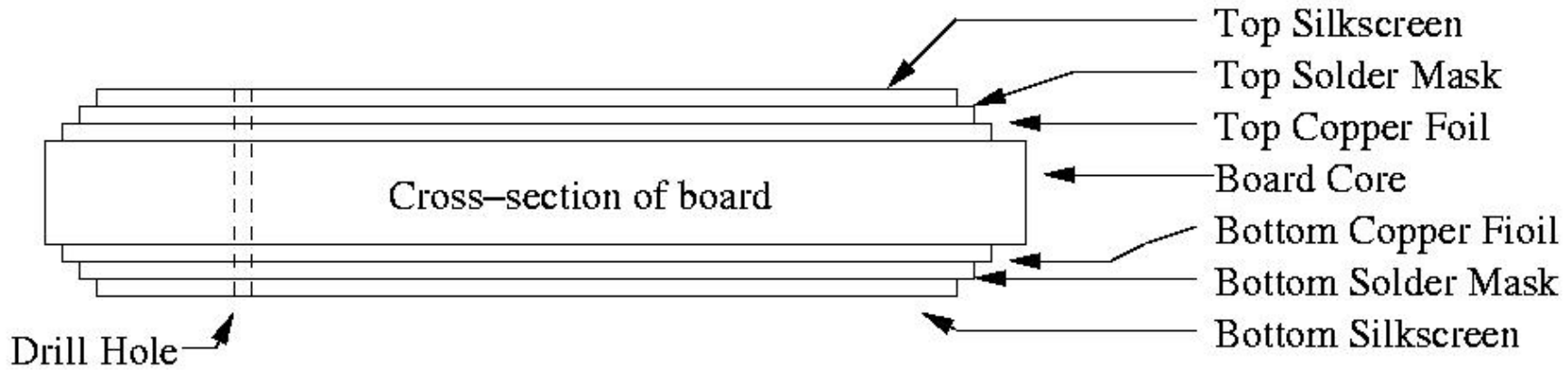
Laminate: the type of copper-clad board that is used to make the PCB. Usually specifies the thickness of the copper, the thickness of the (fibreglass) board itself, and other features.

Via: a connection between copper layers on a board. Via holes are used only for signal connection – not for mounting a component.

Plate-through Hole: a hole that is electroplated to make a connection between the copper layers.

Board Structure

Component Side



Solder Side

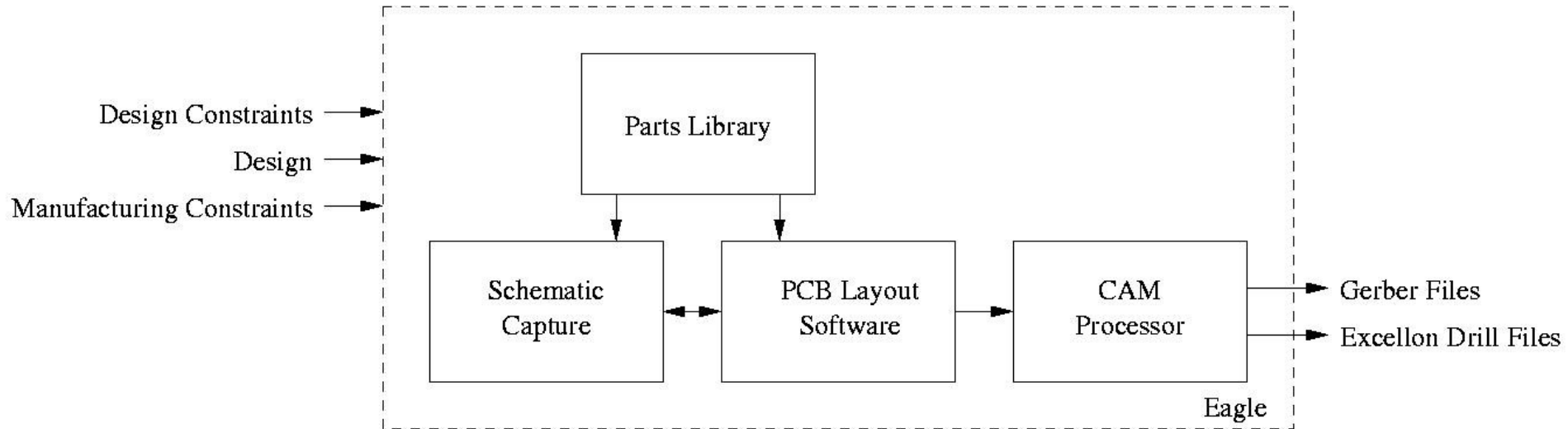
Board Structure

- Hopefully these layers look familiar: they are (roughly) the same as the layers used in Eagle
- Not all board manufacturing processes support the creation of all layers, as discussed later.
- Some manufacturing processes support more layers – additional copper layers in the board core, for instance.

Board Structure

- A “single sided board” has copper only on the “solder side”.
- A “double layer board” has copper on both the top and the bottom.

Development Process



Getting Ready: What You Will Need

To layout a board successfully, you need:

1. Knowledge about the manufacturing process that will be used: tolerances, supported layers, etc.
2. The parts on-hand (ideally).
3. Datasheets that describe the component footprints.
4. Callipers/micrometer.
5. A schematic!
6. PCB layout software.

Manufacturing Methods

- Regardless of the method of manufacture, the PCB ends up existing in the real world and needs to accept the components it is designed for.

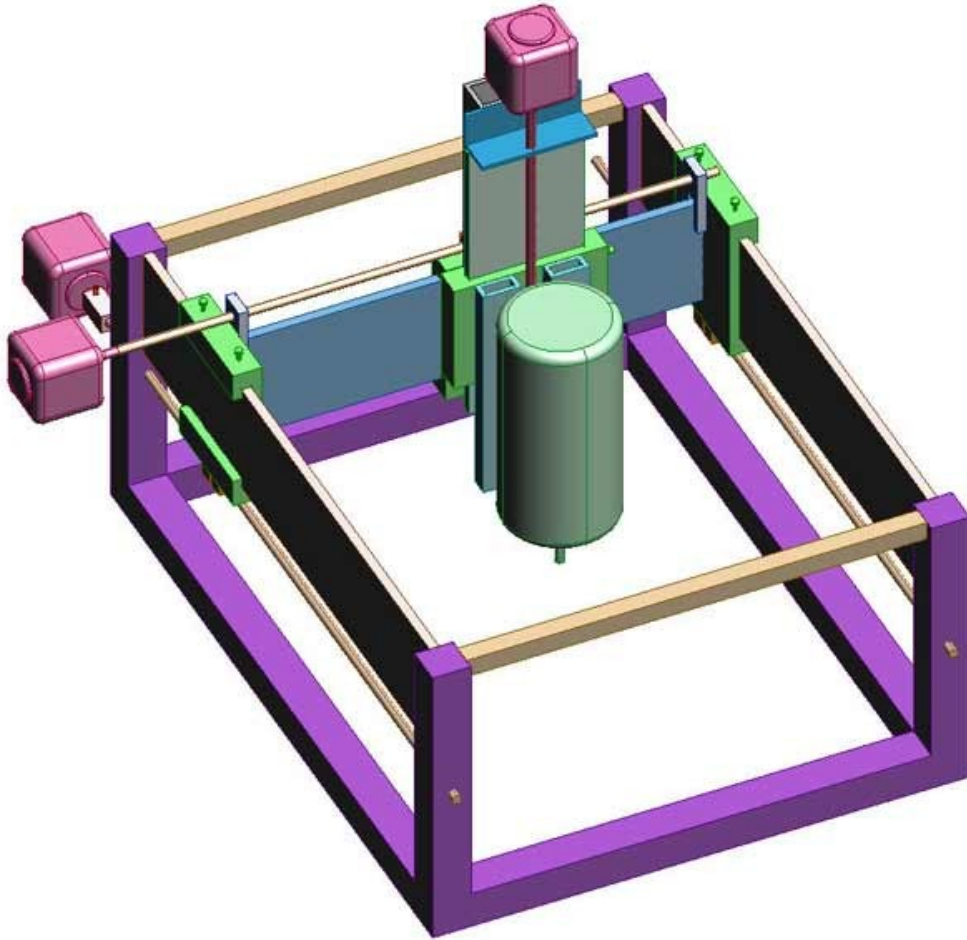
Check and double-check footprints, hole sizes and other physical features.

- Even the smallest of design errors will make a board difficult to use.

Manufacturing Methods

- There are many manufacturing methods (and hybrids) that can be used to create a board. We will take a quick look at three:
 1. Milled PCBs
 2. Professionally Manufactured PCBs
 3. In-house Etched PCBs

Milled PCBs



- Historically, this is the method that has been used the most in this course.
- An X-Y stage is used to guide a router bit along the contours of the PCB traces.

Milled PCBs

- The result is a copper-foil that has a minimum of copper removed in order to form the connections.
- The “backplane” that is left on the board can actually be useful for noise control if it is connected to ground appropriately.
- Advantages: chemical free, quick, in-house (cost)
- Disadvantages: relatively poor tolerances, no plate-through holes, usually single-sided (in this course), single drill size.

Professionally Manufactured PCBs

- “Board Houses” such as Alberta Printed Circuit (<http://www.apcircuits.com/>) offer prototyping services.
- These services vary, but most use a drill, electro-plate, photo-etching process.
- Advantages: good tolerance, multiple drill sizes, multi-layer, plate-through holes.
- Disadvantages: cost (compared to in-house production), conforming to drill sizes can be time-consuming, chemical-oriented.

In-house Etched

- Creating your own board is not too difficult! Many hobbyists choose this option.
- This will see chemical etching of your board in the chemical preparation lab.
- Advantages: potentially multi-layer, better tolerances than milling, support for (larger) surface mount packages.
- Disadvantages: generating artwork, drilling by hand (likely), tolerances subject to operation, chemical-based.

Tolerances

- You need to design your board according to the constraints imposed by the manufacturing process.
- To find the constraints (tolerances):
 - Check the course website under Resources|Manuals
 - Professionally Manufactured PCBs: see the manufacturer's websites.