Prototyping Printed Circuit Boards

From concept to prototype to production.

Chuck McManis (HBRC)

PCB Design and Fabrication

- Agenda
  - Introduction – Why PCBs?
  - Stage 1 – Understanding the “rules”
  - Stage 2 – Planning the board.
  - Stage 3 – Implementing the board
  - Stage 4 – Preparing for production
  - Stage 5 – Waiting for it to arrive in the mail…

Why Printed Circuit Boards?

- Printed Circuit Boards (PCBs) allow you to rapidly (and repeatedly) build a particular circuit.
- The are more compact than ad-hoc methods.
- They are more durable than ad-hoc methods.
- But …

Why **NOT** PCBs?

- They can be expensive in low volumes ($40 - $50 each)
- If they are wrong they either need rework or discarding (wasted $$$)
- When they are needed to do something different, or components change, they are harder to change.

What are all those terms?

- Gerber - Dead photoplotter language
- Aperture – The name for tools in the Gerber file
- NC Drill – Numerically controlled Drill file
- Tool File – Drill sizes for the NC Drill file
- FAB Drawing – Any special notes about the board.

Gerber – Dead plotter language

- Gerber – This is the format of data that was sent to a Gerber Photoplotter. No one uses Gerber machines anymore but the format lives on!
  - Text based format that consists of movement and "flash" or "stroke"
  - Aperture file relates the "outline" to the "action"
  - Example: X1000Y1000D5
    Move to 1.000,1.000 and flash "D5"

Aperture File

- The aperture file tells the photo plotter what “D10” means.
  - Example : D10 CIRCULAR 5 5 0
    This means tool D10 is a circle 5 mils x 5 mils and has no hole in the middle.
- These units are dropped as pixel patterns on to the film to expose your board.

NC Drill File

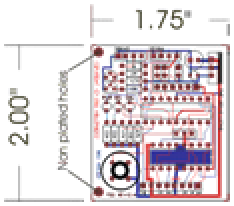
- Numerically controlled drill file.
  - This file tells the manufacturer where you wish to drill holes in the board. There are both binary and text versions.
  - The front of the file will identify the number of tools and then subsequent groups are each tool, where it should drill.

Tool File

- NC Tool file – This file associates tool numbers with actual drills.
  - Example: T01 42
    This means tool #1 is 42 mils in diameter.
- Usually Standard Sizes are free
  - APC Example : .028, .035, .042, .052, .0595, .0860, .125, and .152.

Fab Drawing

- Generally starts with a picture of the board as it will be and highlights any special relationships.
- Example on the right shows non-plated holes and finished dimensions.



Planning A Design

- There are a number of goals you must set up front:
  - Maximum number of layers to use?
  - How big can the board be?
  - Which components will I be using?
  - Will I make just 2 or 2,000? Will others assemble it?
  - How much can I spend?

Planning a Design (Cont'd)

- Layers, size, and components (through hole or SMT) will determine how difficult it is to route the board.
- Number to be made, and who will make them will determine if you need to really space out traces or can keep them close.

Design Rules

- General rules to keep in your mind when creating the board, and ones that you should enforce before sending off for a prototype.
- Rules come in two flavors:
  - Hard & Fast – You can't put large capacitor's leads into a .028 hole.
  - Fungible – Only one trace is allowed to go between two pads of an integrated circuit.

Design Rules : Holes

- Hole sizes:
  - 0.035" for "small" components.
  - 0.042" for "larger" components.
  - Pad must have at least 5 mils of additional space around the hole, so for .035, the minimum pad size is 45 and 60 is better.
  - For 0.42" holes a pad size of 52 mils is minimum and 70 is better.

Design Rules : Traces

- Trace rules come on two forms, size and separation.
  - Generally 10 mils separation is minimum for "easy" manufacturability, 5 mils separation is possible in all "pro" shops.
  - I stick to 12 mil traces for signals, 24 mils for power.
  - Consider two pads on an IC are 100 mils apart, if they are 60 mil pads, then there is 40 mils between them, with 40 mils – 12 mils for 1 trace is 14 mil separation, but two traces is 24 mils and that leaves only 6 mils to separate (hard not to make solder bridges!)

Design Rules : Traces (cont'd)

- Higher power (> 100mA) will need larger traces. So power lines need wider spaces.
- A long trace next to ground looks like a capacitor, high frequencies will become attenuated.
- Two traces that carry strong signals can cross couple their signals.
- For most circuits <= 20Mhz & <= 2 amps, these aren't much of a problem.

Design Rules: Component Layout

- Make sure all chips have pin 1 the same direction (prevents assembly errors).
- Nice to have all resistors and discretes placed orthogonally, easy for manufacture and enhances density.
- Axial lead components can be mounted vertically for more space, but watch for the "flying lead"
- Use silkscreens that show component diameters to insure you don't create impossibly close components.

Design Rules: Components (cont'd)

- Through hole components:
  - Easier to hand-assemble.
  - Lots of opportunities to route signals to other side of the board.
  - Easy to buy in small quantities.
- Surface Mount components:
  - Greater circuit density
  - Can have alternate signals route under their pads.
  - Look cool.

Putting Together a Circuit Board (1/3)

- Identify the “foot print” of all your components.
- Put related components together.
- “Think 3D”, are these components going to overlap? Is this chip going to heat up this other component?
- Identify replication if possible.

Putting Together a Circuit Board (2/3)

- “Stars” vs “Bars” approach to routing:
  - “Stars” find clusters of nodes that need to be connected and route to them.
  - “Bars” typically break the board into horizontal and vertical traces on each side.
- “Badness” to watch for:
  - Lots of vias means you’re components aren’t in the right places.
  - Ground loops, cross connects, overly long traces.

Putting Together a Circuit Board (3/3)

- Validating the schematic and board:
  - Print out test plots on overhead transparencies, one per layer with the bottom layer on opaque white paper.
  - Go through each pin of your schematic and verify it is connected correctly.
  - Look at each layer independently to see things that look “wrong.”

Preparing for Manufacture 1/2

- Prototype vs. Production
  - Prototype service typically does not do silkscreen and soldermasks.
  - Tolerances are often less than for production boards.
  - Per board cost can be changed by adjusting delivery times (called turns).
  - Determines which files you'll need to send to the vendor.

Preparing for Manufacture 2/2

- Generating the files:
  - Generate Gerber files from the CAD tool, you may need to add things to the Aperture file for this to complete.
  - Generate the drilling file and check to see that the correct drills are in the tool file.
  - If going to production you should really have a fab drawing to go with your files. It helps the vendors CAM department setup your job.

Getting It Produced

- Typically every PCB vendor can accept a zip file containing the following data:
  - File manifest – identifies the files in the archive
  - Gerber files for all layers – Each called out in the manifest
  - Aperture list used to generate the gerbers.
  - NC Drill file (binary and text versions if possible)
  - Fab drawing calling out any special requirements. Also provides a way to talk about your board over the phone.

Getting it Produced (Cont'd)

- Typical Manifest (for someone like APC)

```
In this file you will find:
  manifest – This file
  board.txt – Order for this board,
              qty, turn time, etc.
  board.GTL – Top Layer (Gerber)
  board.GBL – Bottom Layer (Gerber)
  board.DRL – NC Drill file (Text)
  board.TOL – NC Tool file
  board.APT – Gerber Aperture file
  board.pdf – Fab Drawing
```


Getting it Produced (Cont'd)

- Typical questions for vendors:
  - Turn around time vs. cost, "Can I get a break if I wait 15 days?"
  - ½ oz, vs 1 oz copper. The latter provides better trace definition and better current characteristics.
  - Quantity vs cost. "Is there an NRE charge?" "What's the price per board for qty 2, 10, 20"
  - How many fit on a panel. Certain sizes are easier to produce because they maximize the use of a panel.

Questions And Answers

- Demo of TraxxMaker 2