

# Printed Circuit Boards (PCBs)

## Part II

# Outline

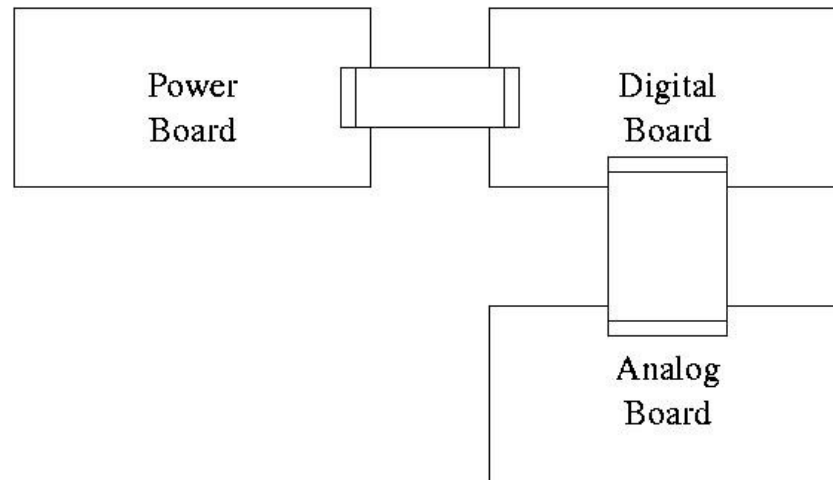
- Eagle Freeware Constraints
- Eagle Versions
- Manufacturing Tolerances and Restrictions
- Setting Up
  - Library Footprints
  - Schematics
- Designing the PCB
  - Component Placement
  - Routing Traces
  - Design Rule Checks
- PCB Artwork
- PCB Manufacture

# Eagle Freeware Constraints

- A recap of some constraints of the free version of Eagle:
  - Free only for non-commercial use
  - Single schematic page (and therefore a single board)
  - Boards are limited to two signal layers (top and bottom copper foil)
  - Board size is limited to 4" x 3.2"

# Eagle Freeware Constraints

- If the size constraints are an issue:
  - Logically partition your design into separate boards.
  - Design the individual PCBs with connectors that accept a cable for inter-board connection.
  - OR! Use the full version of Eagle 5.xx, a couple copies of which are available in the lab: on the same PCs as the Universal Programmers



# Eagle Versions

- Files created by older versions of Eagle can (usually) be loaded by a newer version, but *not* vice versa.
- In order to make a panel of boards, we like to use the professional version of Eagle 5.xx, but can do so at the gerber-level instead.
- If there is a chance that your design may end up migrating to professional manufacture, please use Eagle 5.xx only.

# Manufacturing Tolerances and Constraints

- As already mentioned, the PCB must be designed according to the constraints imposed by the manufacturing process.
- The next slide is a summary of the more important constraints.
- Recall that  $1 \text{ mil} = 0.001''$  (a thousandth of an inch).

## Milled PCBs:

- 40 mil trace width
- Single-sided board
- A single 40mil drill is used for the entire board, regardless of what sizes are specified. Keep in mind that the router bit passes along outline of the traces!

## Alberta Printed Circuits: (Basic Prototyping)

- 7 mil feature minimum size
- Double-sided boards
- 10 drill sizes (outlined below) (source APC web page – Basic prototyping)
- Min and max board sizes

## In-house etched:

- 15 mil feature minimum
- approximately 7" x 7" max board size
- Double-sided boards are possible, but more difficult to make
- Holes drilled by you

Drill Number Set	Drill Size	Finished Size	Approximate Use
#76	.020"	.017"	via holes
#70	.028"	.025"	via holes, fine lead devices such as trim pots etc.
#65	.035"	.032"	IC's, 1/4 watt resistors, small diodes, ripple caps etc.
#62	.038"	.035"	Square posted pins that measure .025" on the flat.
#58	.042"	.039"	TO-220 packages, IDC type square posted headers, 1/2 watt resistors, 1N9000 series diodes, IC chip carriers etc.
#55	.052"	.049"	larger connectors, transformer leads, etc.
#53	.060"	.057"	similar to .052" above
#44	.086"	.083"	TO-220 mounting holes, screw holes, general mounting
1/8 in.	.125"	.122"	mounting holes
#24	.152"	.149"	mounting holes

# Setting Up: Library Footprints

- Use calipers, data sheets, and other information to characterize the part packaging.
- Use this information to verify and/or draw the footprint in Eagle's library editor.
- To save a great deal of time, you can copy footprints from other libraries into your library:
  - by right-clicking on a footprint in the Control Panel Window; or
  - by opening the library, turning on all layers, selecting, and copying onto the clipboard

# Setting Up: Library Footprints

- Be sure to make the pads large enough to solder onto.
- Within reason, the larger the pads, the easier it is to solder.
- This footprint must be symbolically “connected” to the schematic symbol which also appears in the part library.
- Keep numbering of pins straight!

# Setting Up: Schematics

- Ensure that the schematic is using the parts in your library:
  - Use the library and/or update it via the menu. Sometimes it is necessary to reload the library.
  - Edit a part's “technology” or “package” to ensure that the correct footprint is selected – this applies for *all* parts, not just the ones you have created.
- Use the INVOKE command to place hidden power pins on the schematic, if necessary.
- Run the Error Rule Check (ERC) often. Each error/warning must be addressed!

# Designing the PCB

- From the schematic, select “Switch to PCB” to start a new layout.
- **Once a PCB is created, always ensure that both the schematic and the PCB are open at the same time.**
  1. Component Placement
  2. Routing Traces
  3. Design Rule Checks (DRCs)

# Designing the PCB: Component Placement

Good placement of parts is somewhat iterative and time-consuming, but makes subsequent stages simpler.

0. Connectors are typically located to the side of a board.
1. Components need to be separated to allow mounting.
2. The orientation of all polarized parts (including ICs) should be consistent.
3. Place the components on a coarse grid. This helps to line the parts up and assists subsequent routing.
4. Organize by subsystem where possible.
5. Separate analog and digital circuitry where possible.
6. Attempt to minimize connection length (refresh connections using the “Ratsnest” tool)
7. Place supply decoupling capacitors close to the ICs.

# Designing the PCB: Component Placement

- The smaller the board, the more cost-effective!
- The smaller the board, the more difficult it is to route the traces!
- Consider bringing unused MCU port pins to a connector on the edge of the board: allows for future expansion.

# Designing the PCB: Routing Traces

- *Try* to create a single-sided board: these are simpler to manufacture, and are therefore less expensive.
- When using a multi-layer board:
  - try to keep most of the traces on the solder side, particularly those around ICs: allows patches to be made
  - it helps to designate one layer as “horizontal” and the other “vertical”
- Start routing on a coarse grid and move finer as the need presents itself
- For single-sided boards:
  - make holes for jumpers by using a via
  - route only on the solder side (excepting jumpers)
- Vias are created by changing the signal layer while routing the trace: a small pad and hole are automatically created – configure the sizes according to your needs.

# Designing the PCB: Routing Traces

- Avoid 90° bends or T intersections – ensure that intersections are at 45°.
- Keep all trace angles quantized to 0, 45, or 90°.
- Try to limit the number of jumpers or vias in the board layout.
- Keep jumpers straight (0, 90°) and short.
- As with pad size, a larger trace is generally easier to deal with (in the manufacturing process and in board repairs, should they be necessary)
- Eagle has an auto-router. Results are generally less organized than laying traces out manually – but feel free to try it!
- Maze solving!

# Designing the PCB: Routing Traces

- For etched PCBs (in-house or professional), you are free to add text on the copper layers:
  - component designators
  - pin numbering
  - revision information
  - etc.
- These textual annotations help deal with the lack of a silkscreen layer.
- Note that the text must adhere to the minimum feature size constraints.
- Text on the signal layers of a milled PCB is deleted.

# Designing the PCB: Design Rule Checks (DRCs)

- Configure the DRC parameters to match the board manufacturing method.
- **Run the DRC often right from the very start** – dealing with errors sooner prevents having to redo work.

# PCB Artwork

- Besides the files that are used for manufacture of a PCB, other artwork is required.
- In particular, a component placement diagram is used when the board is populated: this shows the outlines of a part, part designators and part values, and is typically oriented by showing the PCB's pads.
- This artwork is typically placed on the silkscreen layer of the board.
- Keep this diagram organized!

# PCB Manufacture

- The Eagle installation and Cadsoft's website ([www.cadsoft.de](http://www.cadsoft.de)) have many User Language Programs (ULPs) that can help in various ways:
  1. Drill size reports
  2. Drill centering utilities (for hand-drilled boards)
  3. Reports about the footprints (packages) being used.
  4. The number of vias, holes, etc.

# PCB Manufacture

- It is up to you to ensure that the manufacturing constraints are met.

# PCB Manufacture

- For professionally manufactured PCBs:
  1. Ensure that only supported drill sizes are used.
  2. Place a copper trace outline around your board on both the top and bottom layers.
  3. Place text on the copper layer indicating which side is the “Solder Side” and which is the “Component Side”
  4. If you have not used Eagle 5 for your design, and the course is looking after manufacture, gerbers and drill files are to be submitted (in addition to the other files)

# PCB Manufacture

- For in-house etched PCBs:
  1. Place a copper trace outline around your board on both the top and bottom layers.
  2. Place text on the copper layer indicating which side is the “Solder Side” and which is the “Component Side”