Introduction

- PCB = Printed Circuit Board
Introduction

History

- PCB designed and laid out by hand, using adhesive tape
- Fluorescent light box used to remove unnecessary parts
Introduction

History

- Dissolving copper with caustic soda
Introduction

Today:
- CAD Software (e.g. Altium)
- Additional tools: capacitance calculators, etc.
Introduction

Design standards:

- Almost all aspects of PCB design are regulated by

  [IPC](http://www.ipc.org)

- Document IPC-2221 „Generic Standard on Printed Board Design”
The schematic

Rules & practices:

- Before you start the PCB layout, you must have the complete and accurate schematic diagram.
- Partially done schematic, non-accurate one – are not suitable.
- Have signals flowing from the inputs on the left to the outputs on the right.
- Make notes on the schematic with precautions („this pin requires guard-track to signal ground)
- If possible, use the same elements (e.g. for resistors when values are in range)
The grid

Rules & practices:

- Lay out the board on a fixed grid
  - Everything is snapped to a fixed points on the board
- It keeps your components neat and symmetrical
- Future edit is easier
The grid

Rules & practices:
- Choose fairly coarse grid size – according to sizes of elements
- Standard grid sizes:
  - 100 thou: basic through hole work
  - 50 thou: general tracking work
  - 25 thou: finer work
- Good practice:
  - start with 100 or 50 thou
  - go with progressively finer grid until design becomes tight in space
  - choose even division for each step: 50, 25, 10, 5 thou

1 thou = 1/1000th of inch
Grid types:

- **Snap grid** – the standard grid to which you can snap all objects
- **Electrical grid** – invisible, but makes your cursor snap onto the center of electrical objects (tracks, pads, etc.)
- **Component grid** – works just like a snap grid, but it’s for component movement only.
  - This can be considered as “secondary” grid
  - Allows to align components to the other grid than the rest
  - Remember to select the size of the component grid the multiple of the snap grid
Tracks

- Track sizes depend on:
  - Electrical requirements of the design
  - Routing space and clearance available
  - Personal preference

- All but basic non-critical circuits require mixture of track sizes

- Start with "the bigger track – the better"
  - Then include all other circumstances

- Bigger tracks:
  - Lower DC resistance
  - Lower inductance
  - Can be easier to and cheaper to produce
  - Easier to inspect and rework
Tracks

- **Lower limits:**
  - Depend upon the track/space resolution that your PCB manufacturer is capable of.
  - E.g. 10/8 track/space means:
    - Tracks can be no less than 10 thou wide
    - Spacing between tracks (or pads, or any part of the copper) can be no less than 8 thou

- No reason to push the limits – choose bigger size if possible
Tracks

- Recommended sizes:
  - 50 thou for ground and power tracks
  - 25 thou for signal tracks
  - 10-15 thou for going between IC and component pads

- Practically, width is dictated by:
  - Current flowing through a track – must be computed
    - Otherwise – risks of high temperature / physical damage
  - Maximum temperature rise a track can tolerate

- Remember that track has the resistance too
Choosing the size of a track:

- Get to know the thickness of the copper layer
  - Usually specified in ounces per square foot
  - Most commonly: 1oz / sqft
  - The thicker copper layers are useful for high current, high reliability designs
- Online calculator: http://www.ultracad.com/calc.htm
Tracks

- Online calculator: http://www.ultracad.com/ucadpcb.htm
- Rule of thumb: 10 Celsius degree rise – good start
Tracks

- Online calculator: [http://www.ultracad.com/calc.htm](http://www.ultracad.com/calc.htm)
- Rule of thumb: 10 Celsius degree rise – good start

<table>
<thead>
<tr>
<th>Current (Amps)</th>
<th>Width for 1oz</th>
<th>Width for 2 oz</th>
<th>milli Ohms/Inch</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10</td>
<td>5</td>
<td>52</td>
</tr>
<tr>
<td>2</td>
<td>30</td>
<td>15</td>
<td>17.2</td>
</tr>
<tr>
<td>3</td>
<td>50</td>
<td>25</td>
<td>10.3</td>
</tr>
<tr>
<td>4</td>
<td>80</td>
<td>40</td>
<td>6.4</td>
</tr>
<tr>
<td>5</td>
<td>110</td>
<td>55</td>
<td>4.7</td>
</tr>
<tr>
<td>6</td>
<td>150</td>
<td>75</td>
<td>3.4</td>
</tr>
<tr>
<td>7</td>
<td>180</td>
<td>90</td>
<td>2.9</td>
</tr>
<tr>
<td>8</td>
<td>220</td>
<td>110</td>
<td>2.3</td>
</tr>
<tr>
<td>9</td>
<td>260</td>
<td>130</td>
<td>2.0</td>
</tr>
<tr>
<td>10</td>
<td>300</td>
<td>150</td>
<td>1.7</td>
</tr>
</tbody>
</table>
Tracks

High frequency

- For bigger frequencies, not only resistance must be considered, but impedance
  - Resistance
  - Capacitance
  - Inductance

- Source and destination impedances must be considered too
Tracks

High frequency – few good practices

- Use as short tracks as possible
- Use as simple shape as possible
- Avoid tracks that have length of the multiplication of the wavelength of the current that is flowing through the track
  - If they will be the same, you will get an antenna
- Clock signals should be placed close to ground (ideally separated by layers)
Pads

- Pad sizes, shapes and dimensions depend on:
  - Component used
  - Manufacturing process
- Pad/hole ratio:
  - Relation of the pad size to the hole size
Pads

- Pad sizes, shapes and dimensions depend on:
  - Component used
  - Manufacturing process
- Pad/hole ratio:
  - Relation of the pad size to the hole size

- Pad should be at least 1.8 times the diameter of the hole, or 0.5mm larger
- Needs to be considered because of alignment tolerances,
- Pad/hole ratio gets more important the smaller PCB gets.
Common practices for pads:

- **Leaded components** – resistors, capacitors, diodes:
  - round shaped pads
  - around 70 thou diameter
- **Dual In Line (DIL) components** – ICs:
  - oval shaped pads
  - 60 by 90-100 thou
  - pin 1 of the chip should always be a different pad shape (usually rectangular) with the same dimensions as the other pins
Pads

Common practices for pads:

- Surface mount components:
  - Rectangular pads
  - Pin 1 = different shape

- Components that rely on pin numbering (connectors, SIP resistor packs, etc)
  - Follow the „rectangular pin 1” rule

- In general: use rectangular pads, unless you need other.
Vias connect the tracks from one side of the board to another through a hole in the board

- Usually made with electrically plated holes – Plated Through Holes (PTH)
Vias connect the tracks from one side of the board to another through a hole in the board

- Usually made with electrically plated holes – Plated Through Holes (PTH)
Vias

- Allow electrical connection between different layers of the board
- Via = pad (technically) but in terms of PCB design they should be treated differently
- Holes in vias are typically fair bit smaller than pads
- Using a via to connect two layers = **stitching**
Polygons

Polygon automatically fills in a desired area with copper, which „flows” around other pads and tracks.

- Useful for laying down ground planes
- Make sure you place polygons after you placed all of tacks and pads
Clearances

The distance between the copper around a hole and the copper in the surrounding polygon, or between any other copper item (line or arc) and a surrounding poly

- Too tight a clearance between tracks and pads may lead to hairline shorts and other problems during the manufacturing process
  - Usually very hard to find once board is assembled
- Don’t push the limit – stay above recommended minimum spacing
Clearances

Recommended clearance limits:

- At least 15 thou: basic through hole designs
- 10 thou or 8 thou: more dense surface mount layouts
- 315 thou: 240V mains voltage on PCB
  - At least 315 thou between 240V tracks and isolated signal tracks
  - Many other requirements
  - If you plan to have 240V on your PCB, you need to consult your manufacturer
  - Place bigger clearance if that’s possible
- For non-mains voltages: IPC provides standard tables
### Clearances

#### Recommended clearance limits:

<table>
<thead>
<tr>
<th>Voltage (DC or Peak AC)</th>
<th>Internal</th>
<th>External (&lt;3050m)</th>
<th>External (&gt;3050m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0-15V</td>
<td>0.05mm</td>
<td>0.1mm</td>
<td>0.1mm</td>
</tr>
<tr>
<td>16-30V</td>
<td>0.05mm</td>
<td>0.1mm</td>
<td>0.1mm</td>
</tr>
<tr>
<td>31-50V</td>
<td>0.1mm</td>
<td>0.6mm</td>
<td>0.6mm</td>
</tr>
<tr>
<td>51-100V</td>
<td>0.1mm</td>
<td>0.6mm</td>
<td>1.5mm</td>
</tr>
<tr>
<td>101-150V</td>
<td>0.2mm</td>
<td>0.6mm</td>
<td>3.2mm</td>
</tr>
<tr>
<td>151-170V</td>
<td>0.2mm</td>
<td>1.25mm</td>
<td>3.2mm</td>
</tr>
<tr>
<td>171-250V</td>
<td>0.2mm</td>
<td>1.25mm</td>
<td>6.4mm</td>
</tr>
<tr>
<td>251-300V</td>
<td>0.2mm</td>
<td>1.25mm</td>
<td>12.5mm</td>
</tr>
<tr>
<td>301-500V</td>
<td>0.25mm</td>
<td>2.5mm</td>
<td>12.5mm</td>
</tr>
</tbody>
</table>

- **Tracks on internal layer**
- **Tracks on the external surface**
- **Operational height of the board above sea level**
Component Placement & Design

Good start:

- PCB Design = 90% placement + 10% routing
- Each engineer will design different layout – there is no absolute right way to place your components
- However, some of basic steps can be defined, in order to:
  - Ease the design
  - Provide best electrical performance
  - Simplify large and complex designs
Component Placement & Design

Basic steps:

- Set your snap grid, visible grid and default track/pad sizes
- Throw down all the components on the board
- Divide and place your components into functional „building blocks” where possible
- Identify layout critical tracks on your circuit and route them first
- Place and route each building block separately, off the board
- Route the remaining signal and power connections between blocks
Basic steps:

- Do a general „tidy up” of the board
- Do a Design Rule check
- Get someone to check it.

- Consider floorplan according to the function
Component Placement & Design

Set your snap grid, visible grid and default track/pad sizes:

- Choose fairly coarse grid size – according to sizes of elements
- Standard grid sizes:
  - 100 thou: basic through hole work
  - 50 thou: general tracking work
  - 25 thou: finer work
- Good practice:
  - start with 100 or 50 thou
  - go with progressively finer grid until design becomes tight in space
  - choose even division for each step: 50, 25, 10, 5 thou
Component Placement & Design

Throw down all the components on the board:

- Do not place all the components planning “the best layout” at this point
  - Could work for small circuits
  - Will not work for bigger ones
  - By doing that, you will probably run out of routing space. Routing space – the space that is used by tracks
  - If you will place your components “just in the best layout”, then when doing the routing – you can end “in the corner”
  - If you will space the components too much – you end up with a large board with inefficient use of space
Component Placement & Design

Throw down all the components on the board:

- **Thing to avoid:**
  - Place all components evenly spaced out
  - Have thousands of tracks and vias crossing the board

- The components layout is important to be neat and clear, but not at the cost of having mess with the tracks

- If you have the schematic import option – use it and select components automatically
  - If not – place components manually

- Having all the components on the board it’s easy to determine, if all of them will fit the selected board size
  - Include tracks when judging the board size
Component Placement & Design

Divide and place your components into functional „building blocks” where possible:

- Example: you have a complex filter in your design, but it will have:
  - One input
  - One output
  - Complicated circuitry “inside”

- Combine functional blocks, and place them anywhere in your board
Divide and place your components into functional „building blocks” where possible:

- While building blocks, remember about separating non-compatible parts, e.g.
  - Analog and digital components
  - High frequency high current and low frequency low current
  - Thermal aspects.
  - Etc.
Component Placement & Design

Nice layout:

- Try to achieve the following:
  - Line up components
  - ICs in the same direction
  - Resistors in columns
  - Polarised capacitors arranged the same way
  - Connectors at the edge of the board
  - Symmetry

- Electrical parameters always take precedence over nicely lined up components. Don’t get good layout at the price of:
  - Electrically poor layout
  - Overly big board
Component Placement & Design

Bad layout:

Nice layout:
Component Placement & Design

Good? Bad?
Complete the last steps:

- Routing the different blocks separately
- Then arrange blocks into complete design
- Run Design Rule Check (DRC)
  - Essential step that ensures that the board is correct
  - Mainly checks for:
    - Correct connectivity of tracks
    - Correct widths
    - Correct clearances
  - No DRC available: do manually (compare layout with schematic)
- Get someone to check the board
  - Probably you overlooked something
Floor planning

Grounding:

- Common impedance
- Low potentials
- High impedance
Floor planning

Grounding:

- Contains currents
- Reduces inductance
Floor planning

Power distribution:

- Impedance coupled
Assembly techniques

Surface Mount Technique

Easier, but less performance
Floor planning

Heating:

- Determine which elements will be reaching high temperatures
- Reserve the space for heat sinks for your design (if necessary)
Floor planning

Tracks:

- Don’t place tracks close to the edges of the board
- Place power track as first
  - This track should have appropriate size, according to the current flowing through it
  - Should contain capacitors that eliminate interferences from the power supply
- Decoupling capacitors – place for VCC pins for each IC in the circuit
- Avoid tracks without an “end” – as it becomes antenna
- Power and ground tracks should be as wide as possible
Basic Routing (tracking)

**Routing** = the process of laying down tracks to connect components on your board

**Net** = electrical connection between two or more pads
Basic Routing (tracking) rules

- Tracks should only have angles of 45 degrees.
- Avoid right angle
- Never under any circumstances use angles > 90°
- If you don’t follow these rules – design looks terribly bad
- There also may be manufacturing problems
- Most packages have 45° enforcement turned on by default – don’t turn that option off
Basic Routing (tracking) rules

- If you don’t follow these rules – design looks terribly bad
Basic Routing (tracking) rules

- No reason to use nice rounded track corners
  - Harder and slower to place
  - No real advantage
  - 45 degree increments – are perfectly fine

- Snake tracks around the board
  - Don’t go directly ‘point-to-point’
Basic Routing (tracking) rules

- Enable Electrical grid
  - Sometimes referred as ‘snap to center’, ‘snap to nearest’
  - Then software find the centers of pads and ends track automatically
- Always take track to the center of the pad, not ‘just touch’
  - Software may consider such a thing as non-connected
Basic Routing (tracking) rules

- Use a single track, not multiple tracks tacked together end to end
  - No difference in the look
  - Difficult for the further edit

- Make sure tracks go right through the exact center of pads and components
  - If track doesn’t go through the center while you use snapping, then you are using the wrong snap grid

- Only take one track between 100 thou pads unless absolutely necessary
Basic Routing (tracking) rules

- For high currents, use multiple vias when going between layers
  - This will reduce track impedance
  - This will improve reliability
- Don’t drag tracks to angles other than 45 degrees
- If power and ground tracks are deemed to be critical, lay them down first
- Make your power tracks as BIG as possible
- Keep power and ground tracks running in close proximity to each other if possible
  - Don’t send them in opposite directions around the board
  - This lowers the loop inductance of power system
Basic Routing (tracking)

- Keep things symmetrical.
- Don’t leave any unconnected copper fills (“dead coppers”)
  - Ground them or remove them
- Don’t place vias under components
  - Once the component is soldered in place, you won’t be able to access the joint to solder a feed through
  - Solder joint can also interfere with the component
- Try and use through hole component legs to connect top tracks to bottom tracks
  - This minimizes the number of vias

- Each via adds two solder joints to the board
- The more solder joints, the less reliable the board becomes
- More solder joints = takes longer to assemble
Finishing touches

Should be done after all routing is finished

- For thin tracks (<25 thou) add a chamfer to any “T” junctions
- Check that you have any required mounting holes on the board
  - Keep them clear of any components and tracks
  - Allow room for screws
- Minimize the number of hole sizes
  - Extra holes = extra money
  - Manufacturing cost:
    - Number of holes
    - Additional charge for each size
Finishing touches

- Double check for correct hole sizes on all your components
  - Very common problem: receive your PCB and holes don’t fit
- Check that there is suitable physical distance between all your components
- Change your display to draft mode – it will display all your tracks and pads as outlines.
Design Rule Checking (DRC)

Automatically checks PCB design for connectivity, clearance and other manufacturing errors

- For bigger PCB designs, it is impossible to do such check manually

Examples of what DRC can check:

- **Circuit connectivity.** Checks that every track on board matches the connectivity of the schematic

- **Electrical clearance.** Clearance between tracks, pads, components.

- **Manufacturing tolerances** like min/max hole sizes, track widths, via widths, annulus sizes, short circuits.
Submitting design to manufacturer

What you can be asked, and you should prepare for:

- Desired manufacturing time
- Quantity of boards required
- Board thickness
- Type of board
- Number of layers
- Surface finish
- Color of solder mask
- Copper weight
- Do you want electrical testing
- Track/space clearance of the board
- Panelized or cut boards?
References and Sources

- Elektronika Praktyczna 2009
- PCB Design Tutorial, D. L. Jones