

How to design the perfect PCB

Eines de Disseny

Grau d'Enginyeria Electrònica

Universitat de Barcelona

Based on exceptional teaching work by:

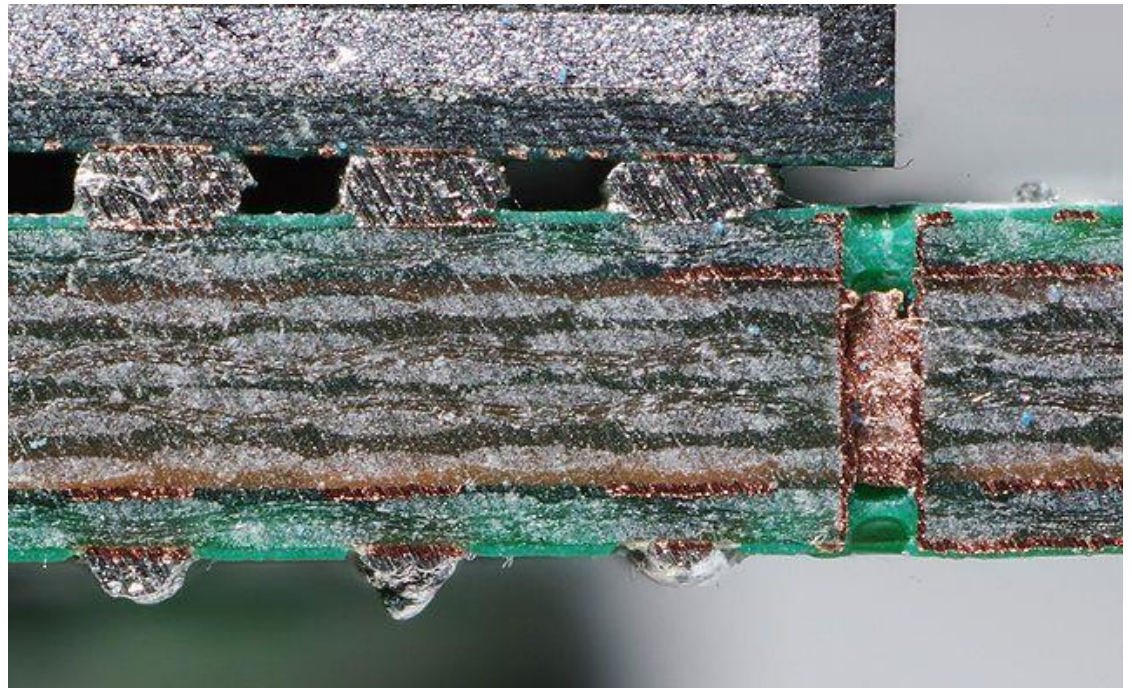
<http://michaelhleonard.com/>

Contents

- I. Anatomy of a PCB
- II. Designing your circuit
- III. Putting the design into software
- IV. Final preparations for PCB layout
- V. Get to work!
- VI. Manufacturing the board

I. Anatomy of a PCB

- Board materials
- Layers
- Copper traces
- Vias
- Other things



Board materials

- Base of PCB: solid, non-conducting material, generally glass-reinforced epoxy (FR-4)
- Lamination with copper sheet, which creates the conducting surface
- For PCB materials for high frequency circuits, [other materials](#) are required

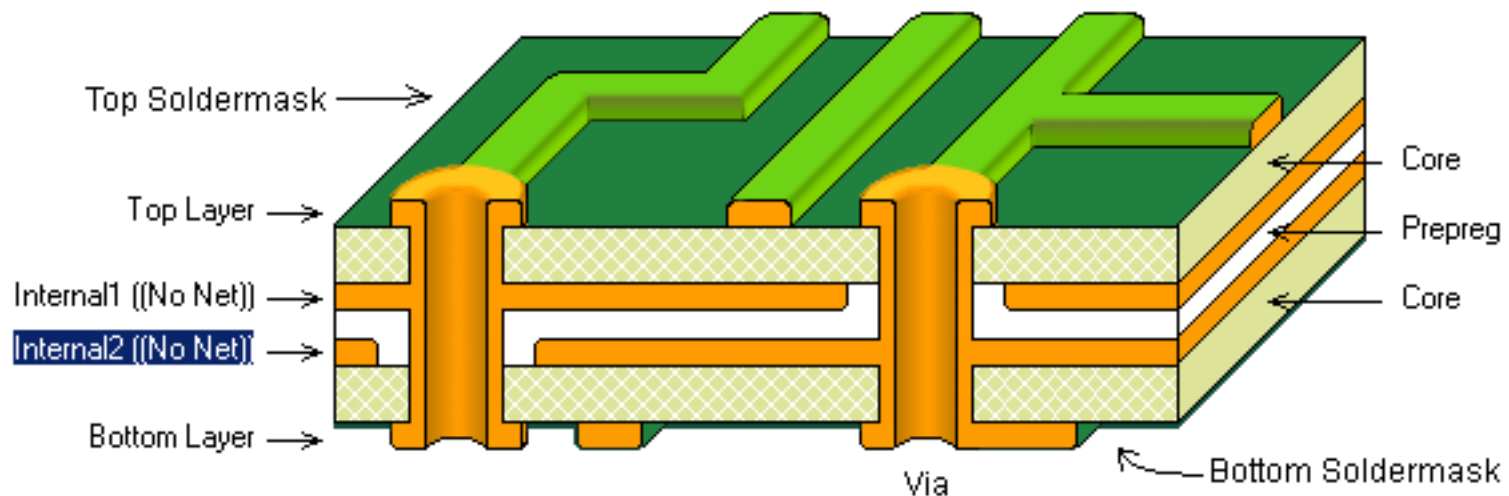
Layers

- Single-sided boards: base material plus single sheet of metal over the top
- Lamination with copper sheet, which creates the conducting surface
- Designing for double sided PCB is highly recommendable



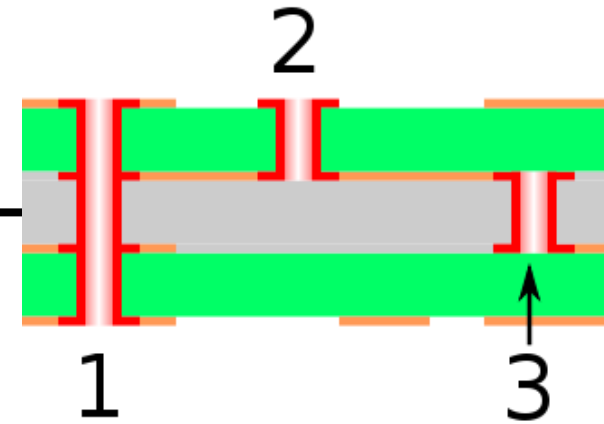
Copper traces

- Created by removing copper from the solid sheet that sits on top of the base material
- There exist some constraints to consider routing traces, namely size considerations



Vias

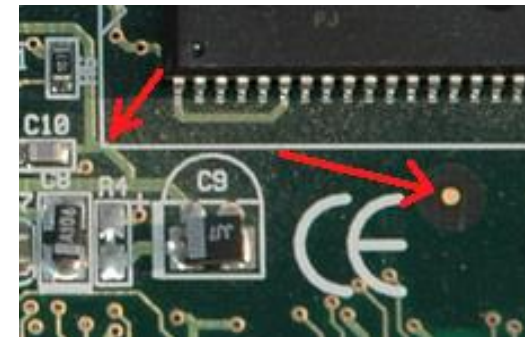
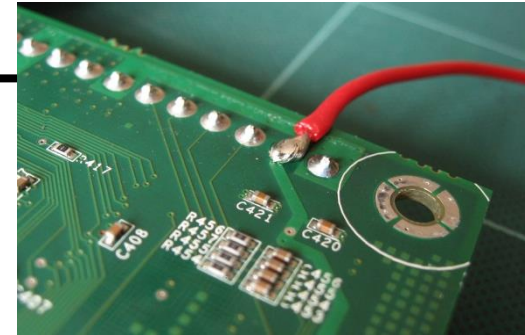
- Used in multi-layer boards to electrically connect one layer to another



- 1. Through hole** - a hole is drilled thorough the hole board and then electroplated so that is conductive
- 2. Blind** - used in designs with more than two layers to connect a surface layer to an internal layer without going all the way through
- 3. Buried** - similar to blind vias but only used to connect internal layers

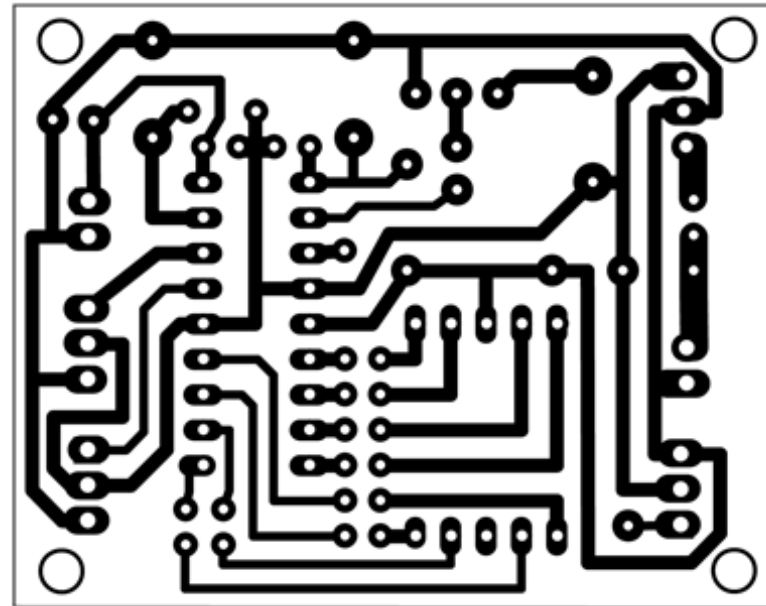
Other things

- **Soldermask** - layer applied after manufacturing, in order to keep solder paste from spreading where it shouldn't be
- **Fiducials** - special markings on the board that allow a pick-and-place automated assembly machine to calibrate itself. Fiducials are usually just a circle where the soldermask has not been applied with copper circle in the middle
- **Silkscreen** - layer added after fabrication, used to provide visual cues to the user, identify proper component placement, for branding, etc.
- **Copper fill** - using a ground/power plane is important for suppressing noise on the ground circuitry, dissipating heat from a particularly active device, etc.



II. Designing your circuit

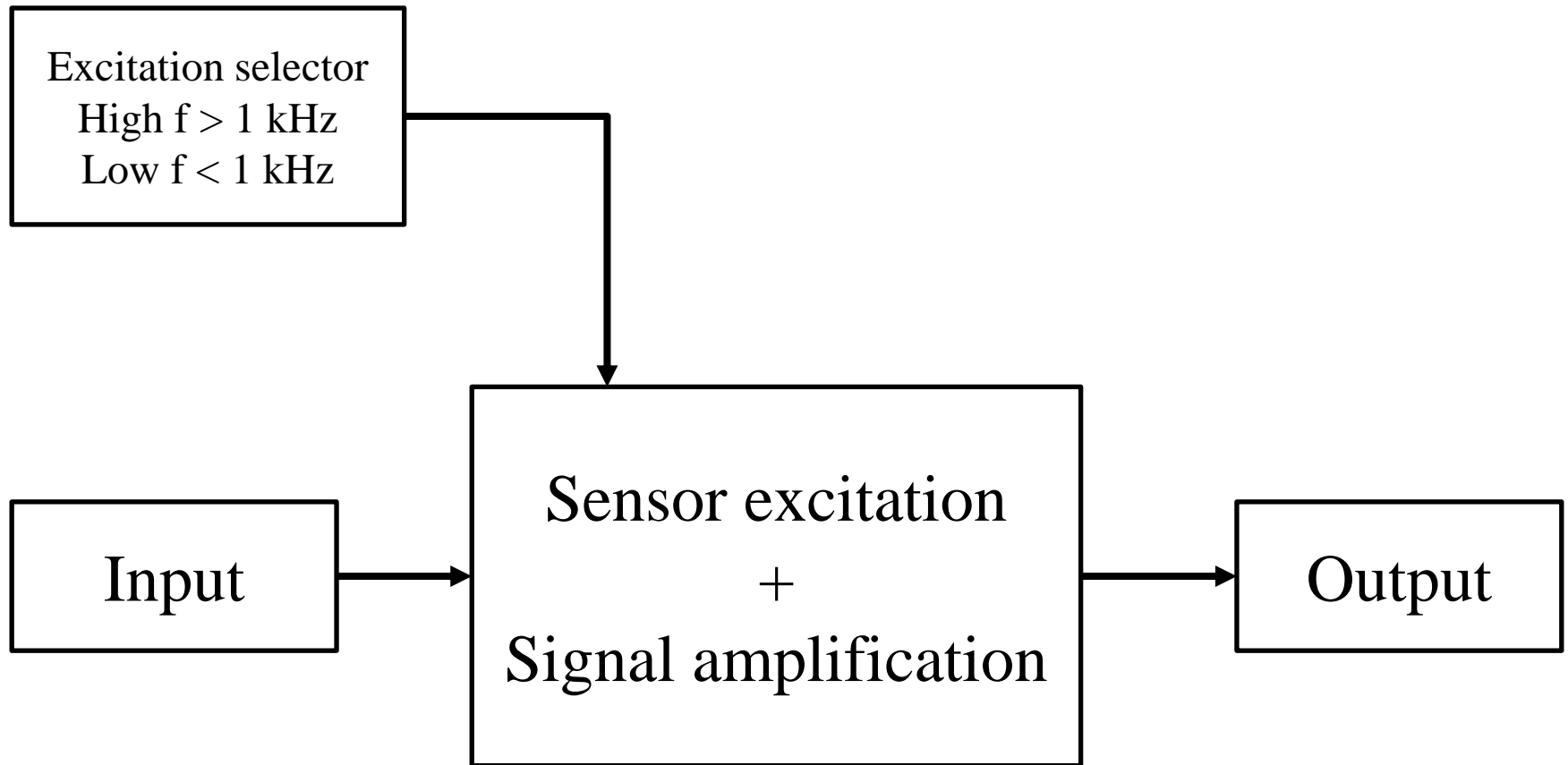
- Determining your goals
- Visualizing your design
- Choosing parts
- Sketching your connections



Determining your goals

1. We want to develop a circuitry capable of performing measurements of a particular device under test (DUT), namely a sensor
2. The measurement will consist in submitting the DUT to a determined potential (with a desired frequency) and acquiring the current intensity that runs through it
3. We need variable gain in the circuit to control the measurement amplification
4. We need to well establish some testing points on the board

Visualizing your design



Choosing parts

- Some places where parts can be found:

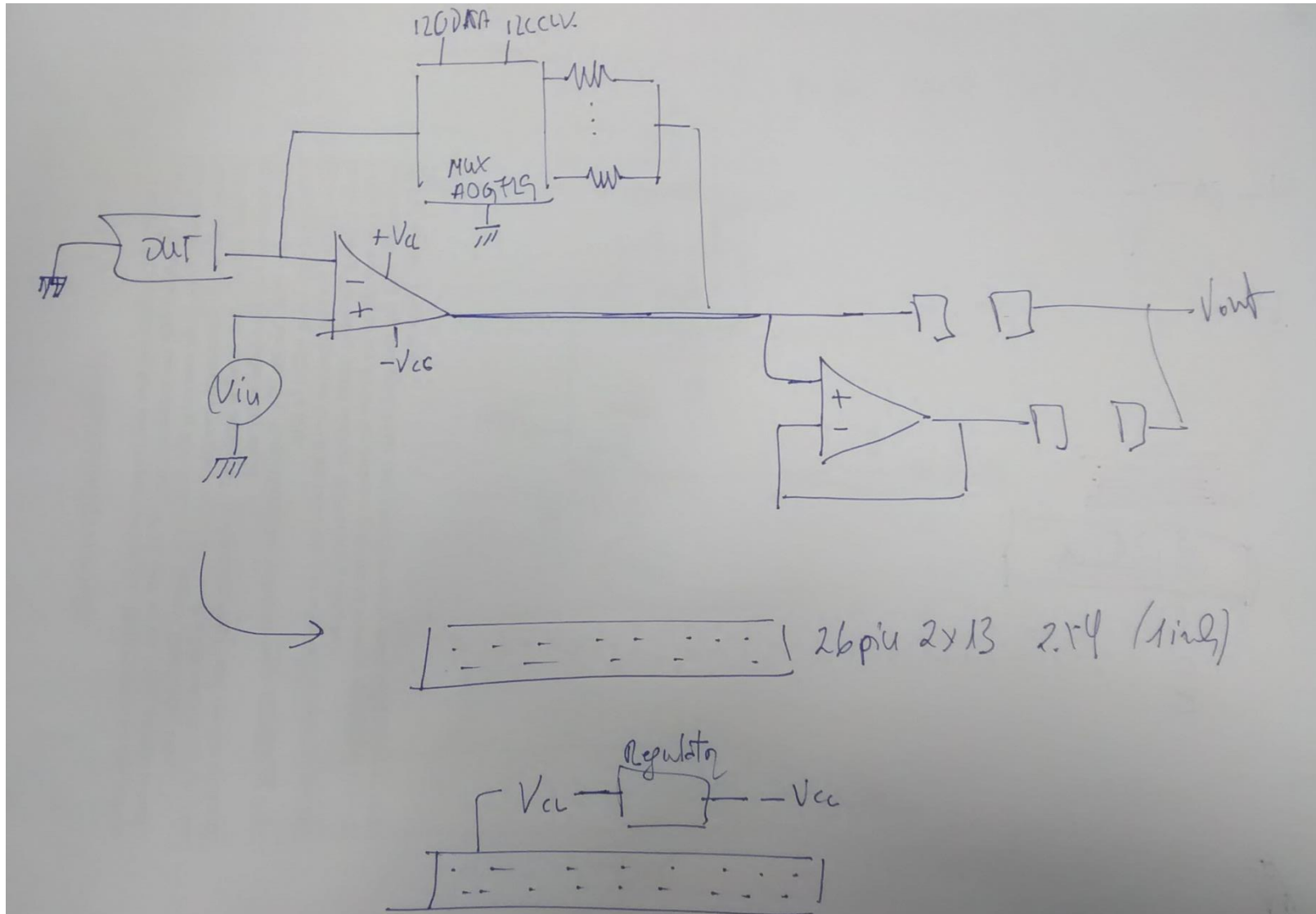
www.mouser.com

www.digikey.com

www.octopart.com

www.datasheets360.com

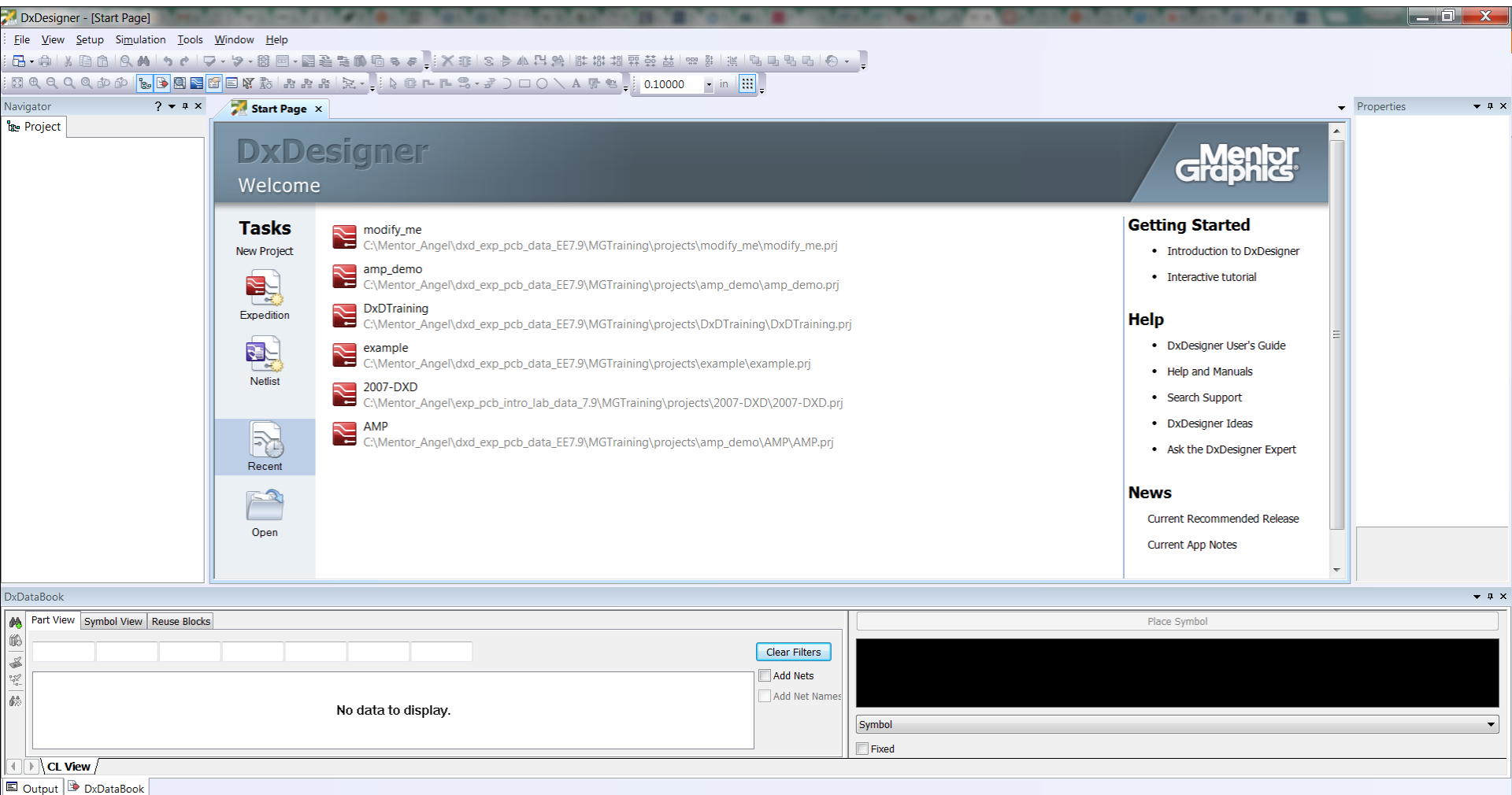
Sketching your connections



III. Putting the design into software

- Choosing a CAD package
- Best practices for electrical schematics
 - Labels and comments
 - The logical schematic
 - Using hierarchy the right way
 - Other tips

Choosing a CAD package



Best practices for electrical schematics

The difference between a good and a bad schematic is a matter of how easy it is to understand

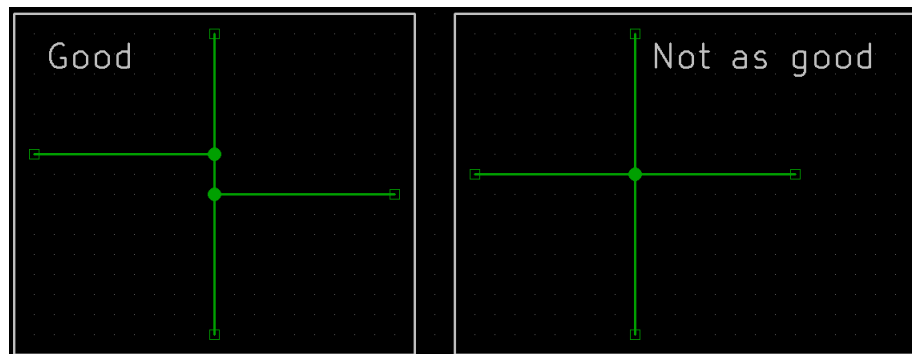
LABELS AND COMMENTS

- You should always label your components
- It is important to comment on non-standard parts or on important performance details
- Labels should appear in a logical position near the component, while non overlapping other labels or components
- It is important to label the most important nets
- Keep names as short as reasonable, use all caps, separate words with underscores

Best practices for electrical schematics

THE LOGICAL SCHEMATIC

- Inputs come from the left, outputs go to the right
- Power comes from the top, ground and negative voltages go to the bottom
- Make a very clear dot where two wires form an intersection
- Avoid 4-way connection points



Best practices for electrical schematics

USING HIERARCHY THE RIGHT WAY

- Inputs come from the left, outputs go to the right
- Power comes from the top, ground and negative voltages go to the bottom
- Make a very clear dot where two wires form an intersection
- Avoid 4-way connection points

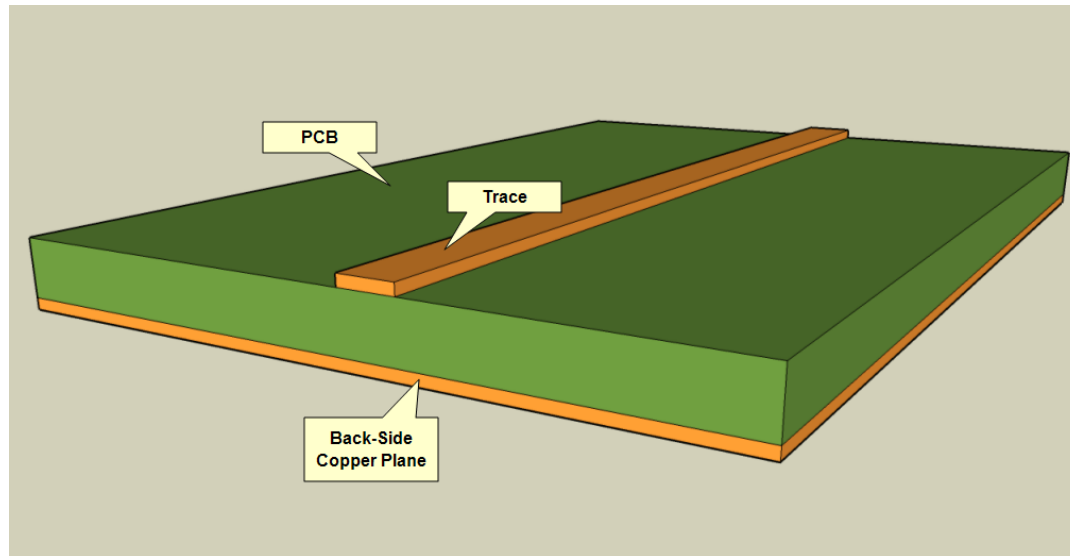
Best practices for electrical schematics

OTHER TIPS

- Show decoupling capacitors near the device they are protecting
- Design for easily printable schematics
- Make your schematics understandable even if they are printed in black and white
- Air wires

IV. Final preparations for PCB layout

- Choosing a manufacturer
- Defining your design rules
- Do I need a ground plane?
- Special considerations



Choosing a manufacturer

- A “mil” refers to a thousandth of an inch
- “Clearance” refers to the separation from the edge of one trace to the nearest edge of another
- Some specifications depend on board manufacturer
 - OSH Park
 - Advanced circuits
 - Express PCB



Defining your design rules

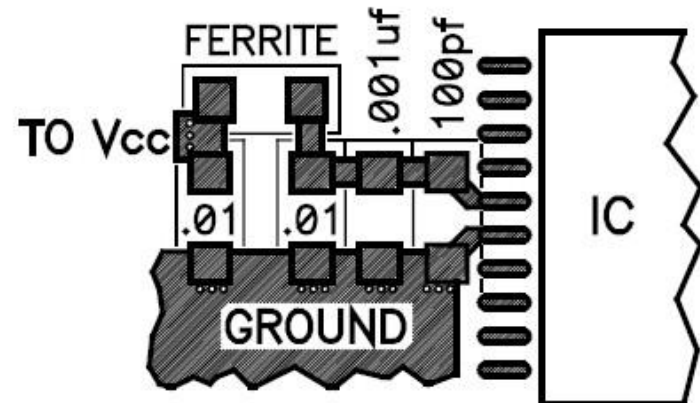
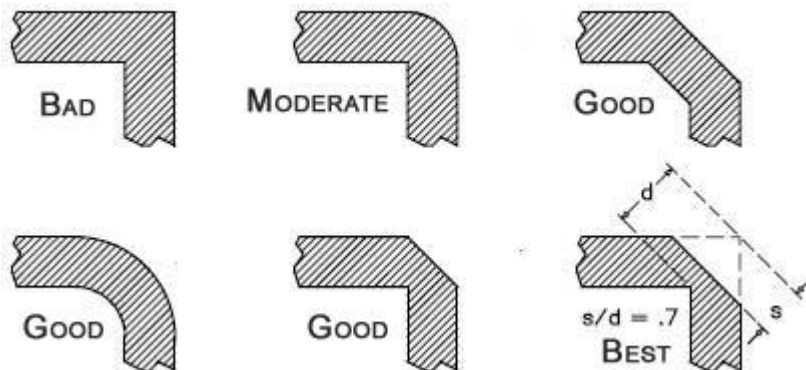
- Important! Manufacturer constraints, e.g. OSH Park:
 - 6 mil copper traces
 - 6 mil spacing between traces
 - 13 mil drill diameter
 - 7 mil annular rings – $(\text{diameter of pad} - \text{diameter of hole})/2$
- Using smaller features will likely result in broken traces, overlap of traces and copper fills, or busted vias
- It is good to define these rules in the CAD software

Do I need a ground plane?

- A ground plane is a copper layer on your PCB that acts as a common ground to many devices
- Benefits?
 - Electromagnetic shielding
 - Lower resistance of path to ground
 - Heat dissipation across the board
- Drawbacks?
 - Increase in parasitic capacitance, which makes the circuit less responsive than intended
- If designing for RF signals, always use ground plane
- If you have components that rely on fast changing input signals, you'd better not use ground plane

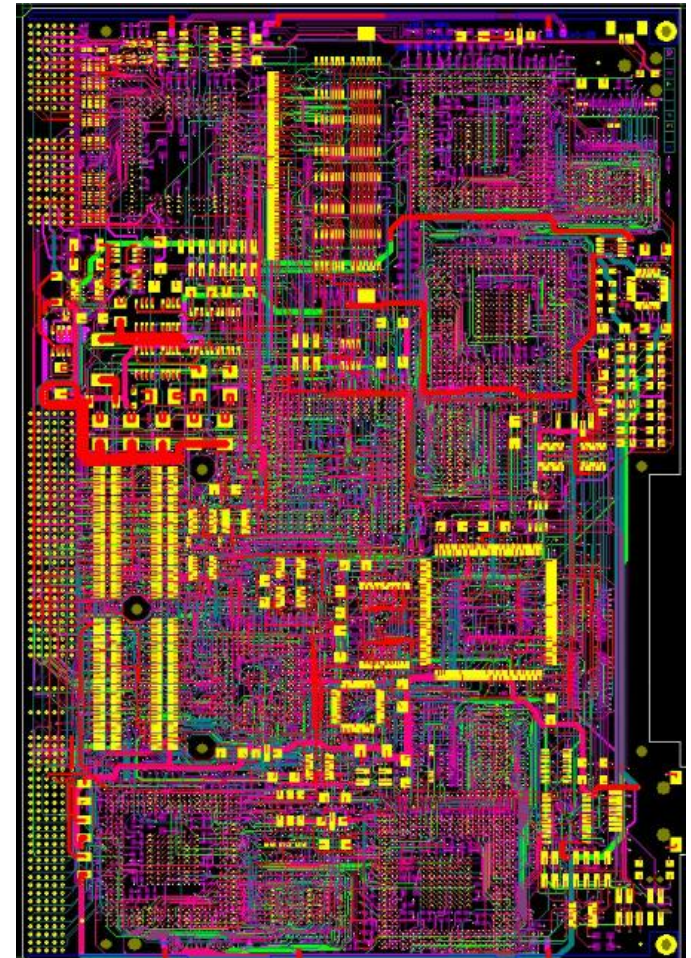
Special considerations

- **Designing for RF** – Visit [this site](#)
- **Mixed-signals designs** – if your PCB carries both analog and digital signals, their paths must be clearly separated
- **High voltage work** – these circuits require extra care when designing and testing



V. Get to work!

- To autoroute or not?
- Drawing the board outline
- Placing components
- Making connections
- Adding some style
- Final design checklist



To autoroute or not?

- Should you use the wonderful autorouting features of your CAD package or not?
- An autorouter automatically connects the traces in your board in a pattern the software deems is most efficient
- Some CAD packages even include an “Autoplacer” for automatically placing components
- In general you are probably better off avoiding both of these tools, depending on the complexity of your design

Drawing the board outline

- This layer tells your manufacturer where to cut to give you the right sized PCB
- ✓ Set your drawing grid at a reasonable spacing
- ✓ Consider using alternate axes
- ✓ Ensure that all the edges line up exactly. Otherwise, the manufacturer won't be able to fabricate the PCB
- ✓ This is a first step if there are predetermined requirements for the board
- ✓ Alternatively, you may want to wait until the end to draw your board edge

Placing components

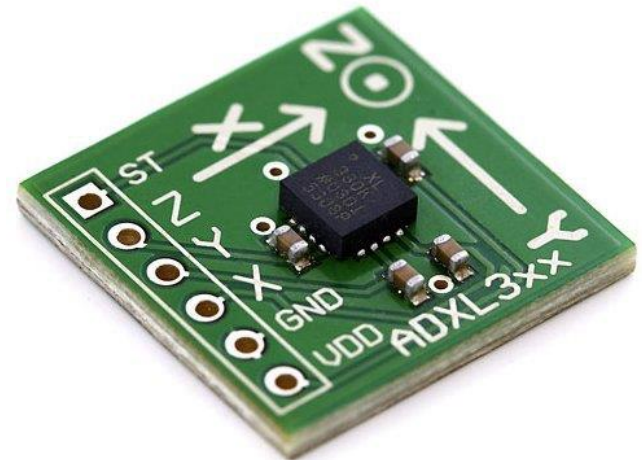
- You should start with components that have a set physical connection, e.g. connectors or sensors
- Continue by placing your integrated circuits (ICs), starting by the largest and then placing the smaller Ics as you go, trying to leave extra room around devices that have many pins
- Once all physical components and ICs are placed, you can place supporting devices such as resistors, capacitors, diodes, etc.
- Leave space for annotations and markings on the board

Making connections

- You can begin by routing your power traces first and then focusing on everything else (high frequency signals should be routed first)
- Make use of thick power supply traces, around 20 mil power traces
- [This tool](#) helps you calculate the correct width to use for your traces
- Avoid routing two or more high frequency signal traces in parallel with each other – remember Ampère's law!
- Try to group similar signals together

Adding some style

- Do you want your final design to indicate that “this” resistor is R11 and has a value of 4.7k or do you want to just mark it as R11? Maybe you don’t want to mark it at all? [Up to you](#)
- Generally you will want to label any LEDs, buttons, switches, connectors or otherwise important devices
- This is done in the silkscreen layer



VI. Manufacturing the board

- Run a Design Rules Check (DRC)
- Generate the Bill-Of-Materials (BOM)
- Export the board files: Gerber
- Final manufacturing checklist
- Send it to your manufacturer

The three basic DRC checks

