List of things to consider when ordering PCBs from Elektronikk- og Prototypelaben

General information

Currently, the lab can only manufacture single or double sided PCBs. For a lot of applications this is sufficient, at least for a first prototype.

Vias cannot be placed under surface mount components due to the type of via we can offer.

Substrate thickness	Substrate type	Copper thickness
0.5mm	Single sided	35um/18um
0.8mm	Double sided	35um/18um
1.0mm	Double sided	35um/18um
1.4-1.5mm	Double sided	35um/18um
1.6mm	Double sided	35um/18um

All tracks going to headers or hole-mounted components must connect from the opposite to the side where you place the component or header. This is because the holes for the components are not normally metallized, so there is no connection between top-and bottom side. If you still need to place tracks on the same side as the component, you must start routing from the bottom side, then place a dedicated via to be able to continue routing on the top layer.

Files to include when sending a job to ELPROLAB

Layers to include for a two-layer card when generating Gerber files:

- Top Layer - The copper layer that is visible on the top of the PCB.
- Bottom Layer - Corresponds to Top Layer. Often used to make a good ground return (when connected to GND net).
- NC Drill file - This file defines hole sizes and positions of the holes.
- Board Outline - Defines the board size. Often, the Keep-Out Layer is used to draw a board outline in Altium.

Please include an explanation as to which layer is which. The file names are not always self-explanatory.

Format: Please restrain from using a format different than

Holes and vias

Hole sizes: We have drills as small as 0.3 mm, but please avoid using smaller holes than 0.7 mm since smaller drills tend to break easily. We have drill tools in steps of 0.1 mm from 0.7 mm up to 2.0 mm (inclusive). Bigger holes will be cut using a contour router which can cut arbitrarily large holes.

Slotted holes: If you need slotted holes, the contour router will also be used. Slot holes must be wider than 1 mm.

Vias are made in one of two ways: One option is to drill holes of 0.7 mm and solder tinned copper wires to either side of the holes. The other option is to place a rivet type via through a hole and punch it down so it expands and creates a connection. The rivets come in certain sizes only. These rivet type vias require a larger hole than their inner diameter. To avoid unintentional short circuits, the outer diameter of the via in the PCB design must be equal to, or larger than, the outer diameter of the rivet. See table below for outer, inner and drill diameters

Vias size			
Inner diameter	**Drill/hole size**	**Outer(annulus) diameter**	**Typical used for:**
0.6mm	0.9mm	1.4mm	Signal routing
0.8mm	1.1mm	1.6mm	Small plated holes
1.0mm	1.5mm	2.4mm	Larger plated holes
1.2mm	1.7mm	2.7mm	Even larger plated holes

Tracks and distance

Track width: If the tracks on a PCB are very narrow, this may result in very little residual copper, which could mean tracks end up broken. Avoid making tracks narrower than 0.2 mm/8 mils, and preferably make them wider (more than 0.3 mm/12 mils). Side note: tracks should start out slightly narrower than the pad it is connecting to so the molten solder does not wick onto the track from the pad when components are soldered onto the PCB.

Technical data	**Sizes and distances**
Distance between tracks	0.02mm
Track width	0.3mm(0.2mm)
Minimum hole	0.7mm(0.3mm)
Working area	300mm x 230mm

Distance between tracks (clearance): absolute minimum 0.02 mm, however this should only be used in extreme cases. Judging from experience, a clearance of 0.1 mm should work great. If you have room, make the clearance even bigger!

All jobs must be sent by email! Send to elprolab@ies.ntnu.no and mark with your own name and supervisor. Also include soft and hard deadline.