Fabrication limitations and design recommendations

Detail	Minimum	Recommended	Maximum
Panel/Board size	Not relevant	<160 x 160 mm	210 x 300 mm
Width (conductors):			
-Wire/trace	0.25 mm	≥0.4–0.5 mm	Not relevant
-Power wire	0.50 mm	≥1.0 mm	Not relevant
Clearance (conductive):			
-Generally	0.25 mm	≥0.5 mm	Not relevant
-Object - Polygon		1.0 mm	
Other dimensions:			
Annular ring VIA	0.25 mm	≥0.5 mm	Not relevant
Annular ring Component	0.35 mm	≥0.7 mm	Not relevant
Drill bit diameter	0.4 mm		5.0 mm
Silkscreen to soldermask	0 mm	Not used	
Solder mask sliver	0 mm	≥0.3 mm	Not relevant
Solder mask expansion	0.1 mm	~0.3 mm	Not relevant
Mill bit diameter	1.0 mm	1.5 mm	2.5 mm

- "Generally" means: Trace to trace, pad to pad, pad to trace, i.e., any conducting object that shouldn't risk being short circuited.
- Object means; trace, pad, any object not belonging to a polygon. Polygon (plane) refers to "polygon pour", fill metal usually connected to ground.
- Annular ring = (diameter of the pad - diameter of the hole) / 2, i.e., the ring radius. For components, it is rather the solder-ability and mechanical aspects than PCB fabrication aspects that determines the ring size. Also; it is a trade-off with clearance for, e.g., DIL/DIP ICs.
- Silkscreen to solder-mask (clearance); is not an issue since we don't print text on the PCB:s in the ITN PCB laboratory. Otherwise it is good to have 0.5 mm margins, and possibly margins against component keep-out.
- Recommendations are not digital; always consider the particular design. E.g., if a thicker metal is used it is harder to etch small details (Small clearances). Moreover, it is also about probability. One narrow gap that is short is often not as likely as if there are several narrow gaps.
- Some component pads require tight margins, i.e., the footprint must suit the component and exceptions to the above might then be necessary.
- Drill bits are available in 0.1 mm steps, and mill bits are available in 0.5 mm steps. For drill-bits round off upwards if a component has an odd size.
- Use "Teardrops" (chamfers) to make junctions smoother (Pad, wire junctions as well as wire crossings).
- Preferred to use "polygon pour", fill metal. On both top and bottom, connect both to ground. In the polygon settings, remove islands less than 5 mm^2.
- Use "via stitching" to add extra ground vias, for a more stable low impedance ground.