ILFA design rules for multilayer PCB's

**ILFA PCBs**

- **General design rules**
  - Max. PCB dimensions:
    - 420x570mm
  - Thickness multilayer:
    - On request
  - Metallized holes & milled cut-outs (details refer to the diameter of the drilling tool)
  - Drilling tool diameter:
  - Through hole:
  - Through hole, plugged and capped¹:
  - Blind via:
  - Blind via, plugged and capped¹:
  - Buried via, plugged or resin filled¹:
  - Buried via, plugged and capped¹:
  - Stacked via:
  - Edge metallizations:

**Conductive pattern / remnant annular rings**

- Trace width on inner & outer layers (µm):
- Conductor spacing on inner & outer layers (µm):
- Annular ring between end-Ø inner and outer layers (µm):
- Distance from hole to hole (µm):
- Distance bore to adjacent conductive pattern (µm):
- Overlap of edge metallization on outer layer (µm):
- Distance of conductive pattern to milling contour (µm):
- Distance from hole to milling contour (µm):

**Soldermask**

- Soldermask fillet width (µm):
- Soldermask clearance to copper (µm):
- Soldermask overlap solder mask defined pads (µm):
- Soldermask clearance of edge metallization (µm):
- Soldermask clearance via / component bore unplugged (µm):

**Drilling tool diameter**

- Possible deviation with press-fit technology

**Metallized holes & milled cut-outs**

- Specified end diameter + 100 µm
- On request

**Conductive pattern / remnant annular rings**

- Without plugging ≥775 with plugging ≥100
- Without plugging ≥50 with plugging ≥775
- Without plugging ≥50 with plugging ≥100

**Soldermask**

- Specified end diameter + 100 µm
- On request

---

¹Plugging is possible from a circuit board thickness of ≥0.3 mm excl. copper thickness. PCBs with external, flexible base materials, or materials without glass fabric cannot be plugged.

---

**Version:** ILFA DM 3