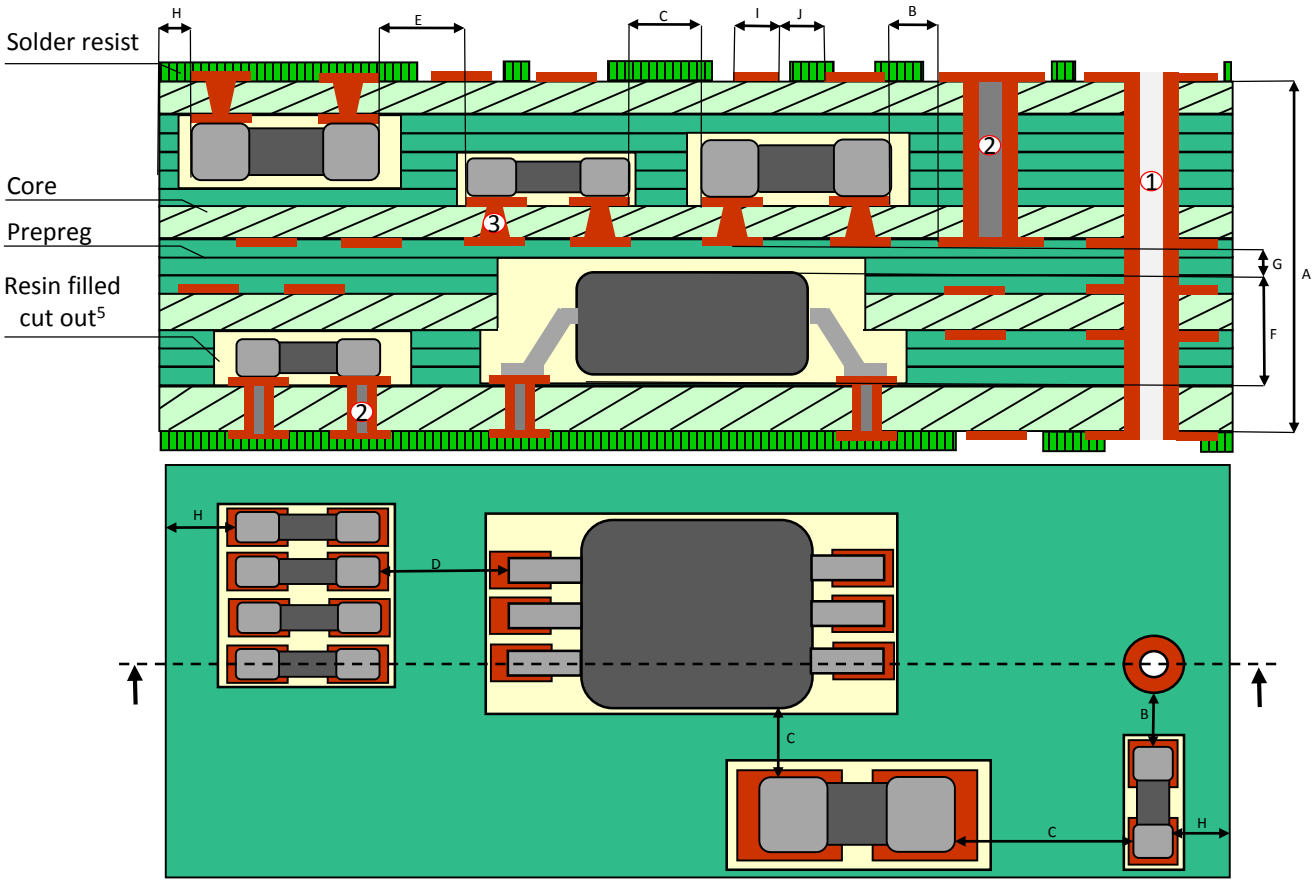
ILFA Designrules Embedding



ILFA PCBs General Design Rules		LEGEND	STANDARD	HIGH END (ON REQUEST)
Max. PCB dimensions			420x570mm	On request
Thickness multilayer	Depending on component heights	A	0.8 - 4.2mm	On request
Metallized holes (details refer to the diameter of the drilling tool)				
Through hole		1	Aspect ratio 1:8, minimum Ø 100 µm	Aspect ratio 1:10, minimum Ø 100 µm
Through hole, Buried Via, Blind Via plugged and capped [1]		2	Aspect ratio 1:8, minimum Ø 150 µm	Aspect ratio 1:10, minimum Ø 100 µm
Standard microvia	Copperfill on inner layer/ outer layer optional	3	Aspect ratio 1:1, minimum Ø 125 µm	Aspect ratio 1:1, minimum Ø 80 µm
Embedded components [2,3] (a BOM and a Pick&Place-List of all embedded components is nessesary)				
Distance component pad and via pad	Maximum component tolerance is always to be used	B	≥ 500 µm	On request
Distance component to component (not within group)		C	≥ 700 µm	On request
Distance component group [4] to component or group		D	≥ 1mm	≥ 700 µm
Distance components on different layers		E	≥ 2 mm	On request
Component height		F	≤ 1,6 mm	≤ 2,5 mm
Space component and next layer		G	≥ 250 µm	On request
Distance component and board outline		H	≥ 500 µm	On request
Conductive pattern				
Trace width on inner & outer layers (µm)	Depending on copper thickness	I	Without plugging ≥75 with plugging ≥100	Without plugging ≥50 with plugging ≥75
Conductor spacing on inner & outer layers (µm)	Depending on copper thickness	J	Without plugging ≥75 with plugging ≥100	Without plugging ≥50 with plugging ≥75

[1]Plugging 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.

[2]Components have be robust enough to sustain the embedding process.

[3]It is recommended to use IPC 7351B lmc (least material condition) = smallest possible pads or even better proportional landpattern design.

[4]A component group can be defined if the distance to the next components is < 700 µm. Each component of the group must face at least one outline of the cut out.

[5]Pregregs and cores will be cut out for the components. The clearance to the component is minimum 150 µm. The outline of the cut out will be defined by ILFA.

Furthermore, all ILFA design rules apply to multilayers.