	Criteria	Standard	After CIRLY's validation	Out of Cirly' competence	Notes
Panel Dimensions	**single or double Layer Panel format**	575 mm x 435 mm	600 mm x 435 mm (Pas de test électrique)	-	
	Maximum Multilayer Format	320 mm x 265 mm	560 mm x 340 mm	-	
	Maximum Board Thickness	0.4 mm ≤ y ≤ 3.2 mm	0.2 mm	y > 3.2 mm	
	Maximum number of layer	y ≤ 12	12 < y ≤ 20	y > 20	Board tickness will not over 3.2mm
Tracks Width	**Track width / Iso mini cuivre fini ≤ 35µ**	120 µ ≤ y	80 µ ≤ y < 120 µ	y < 80 µ	
	Track width / Spacing final copper Tickness= 50µ	150 µ ≤ y	120 µ ≤ y < 150 µ	y < 120 µ	
	Track width/Spacing final copper Tickness= 70µ	210 µ ≤ y	150 µ ≤ y < 210 µ	y < 150 µ	
	Track width / Spacing final copper Tickness= 105µ	310 µ ≤ y	210 µ ≤ y < 310 µ	y < 210 µ	
	Track width / Spacing final copper Tickness= 400µ	1,2 mm ≤ y		y > 1,2 mm	
	Track width <u>Inner Layer 70µ</u>	250 µ ≤ y	170 µ ≤ y < 250 µ	y > 0,25 µ	
Drilling Milling	**Drilling hole diameter (Mechanical)**	200 µ ≤ y	100 µ ≤ y < 200 µ ***	y < 100 µ	***** under final board thickness** : Drilling 200µ in 1,0 mm max Drilling 150µ in 0,8 mm max Drilling 100µ in 0,4 mm max
	Ratio for Plated-Through-Hole : Board Thickness / hole Diameter	8 ≤ y	8 < y ≤ 10	y > 10	Exemple : Drilling of 0,2mm with board thickness of 1,6mm = 8 = standard
	Ratio for Non-covered-blind Vias : Thickness / hole Diameter	y ≤ 0.6	0.6 < y ≤ 0,8	y > 0,8	
	Number of Drilling sequence	y = 3 **2** sequences for BURIED Via **+ 1** sequence for PTH Vias	4 ≤ y ≤ 6	6 < y	
	Number of Pressing cycle	1	2	2 < y	
	Cut Plated-Through-Hole diameter	y > 0,4 mm		y < 0,4 mm	Trous bord de carte
	Milling Tolerance	+/- 200 µ	+/- 100 µ	y < 100 µ	
V-Cutting	**PCB Thickness**	0,8 mm < y < 2,4 mm		y < 0,8 mm y > 2,4 mm	
	PCB Tickness	y ≤ 105 µ		240 µ	
Copper-filled Vias	**Drilling**	Mechanical 100 µ ≤ y ≤ 125 µ		Laser Drilling	
Plugged-and-Covered Vias(with resin)	**Drilling**	Mechanical 150 µ ≤ y ≤ 500 µ		y > 500 µ	Min. copper basis 17 µ Min. Milling = 125 µ
	PCB Thickness	0,8 mm ≤ y	0,6 mm ≤ y < 0,8 mm	y < 0,6 mm	
Design	**Minimal copper pad**	75 µ ≤ y	50 µ ≤ y < 75 µ	y < 50 µ	y
	Min.Via Spacing (inner layer)	VIA : 170 µ ≤ y AUTRE : 250 µ ≤ y	VIA : 150 µ ≤ y < 170 µ AUTRE : 200 µ ≤ y < 250 µ	VIA : y < 15 µ AUTRE : y < 20 µ	y
Coating HAL étain	**PCB Thickness**	0,8 mm ≤ y ≤ 3,2 mm		y < 0,8 mm	White soldermask is not suitable
	PCB Dimensions	420mm x 420mm		y > 420 mm	
Silkscreen	**Silkscreen**	H mini > 0,6 mm L mini > 0,3 mm Minimum silkscreen width **> 100 µ**		H < 0,6 mm L < 0,3 mm Minimum silkscreen width **< 100 µ**	
IMS (with Aluminium plate)	**Drilling - Milling**	Milling 1 mm Drilling 1 mm			Coating wih **Nickel Gold or HAL Sn** is not appropriate
Flex (Kapton / Coverlay)	**Spécifications**	Diemension max 310 mm x 260 mm Tickness 125µ cuivre 35µ Coverlay 25µ ou 50µ + Adhésif 50µ Coating : Chemical Silver		- Coating Nickel Gold - Plated-Through-Hole	