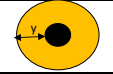
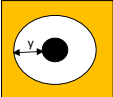


	Criteria	Standard	After CIRLY's validation	Out of Cirly' competence	Notes
Panel Dimensions	single or double Layer Panel format	575 mm x 435 mm	600 mm x 435 mm (Pas de test électrique)	-	
	Maximum Multilayer Format	320 mm x 265 mm	560 mm x 340 mm	-	
	Maximum Board Thickness	$0.4 \text{ mm} \leq y \leq 3.2 \text{ mm}$	0.2 mm	$y > 3.2 \text{ mm}$	
	Maximum number of layer	$y \leq 12$	$12 < y \leq 20$	$y > 20$	Board tickness will not over 3.2mm
Tracks Width	Track width / Iso mini cuivre fini $\leq 35\mu$	$120 \mu \leq y$	$80 \mu \leq y < 120 \mu$	$y < 80 \mu$	
	Track width / Spacing final copper Tickness= 50 μ	$150 \mu \leq y$	$120 \mu \leq y < 150 \mu$	$y < 120 \mu$	
	Track width/Spacing final copper Tickness= 70 μ	$210 \mu \leq y$	$150 \mu \leq y < 210 \mu$	$y < 150 \mu$	
	Track width / Spacing final copper Tickness= 105 μ	$310 \mu \leq y$	$210 \mu \leq y < 310 \mu$	$y < 210 \mu$	
	Track width / Spacing final copper Tickness= 400 μ	$1.2 \text{ mm} \leq y$		$y > 1.2 \text{ mm}$	
	Track width Inner Layer 70 μ	$250 \mu \leq y$	$170 \mu \leq y < 250 \mu$	$y > 0.25 \mu$	
Drilling Milling	Drilling hole diameter (Mechanical)	$200 \mu \leq y$	$100 \mu \leq y < 200 \mu$ ***	$y < 100 \mu$	*** under final board thickness : Drilling 200 μ in 1,0 mm max Drilling 150 μ in 0,8 mm max Drilling 100 μ in 0,4 mm max
	Ratio for Plated-Through-Hole : Board Thickness / hole Diameter	$8 \leq y$	$8 < y \leq 10$	$y > 10$	Exemple : Drilling of 0,2mm with board thickness of 1,6mm = 8 = standard
	Ratio for Non-covered-blind Vias : Thickness / hole Diameter	$y \leq 0.6$	$0.6 < y \leq 0.8$	$y > 0.8$	
	Number of Drilling sequence	$y = 3$ 2 sequences for BURIED Via + 1 sequence for PTH Vias	$4 \leq y \leq 6$	$6 < y$	
	Number of Pressing cycle	1	2	$2 < y$	
	Cut Plated-Through-Hole diameter	$y > 0,4 \text{ mm}$		$y < 0,4 \text{ mm}$	Trous bord de carte
	Milling Tolerance	+/- 200 μ	+/- 100 μ	$y < 100 \mu$	
V-Cutting	PCB Thickness	$0,8 \text{ mm} < y < 2,4 \text{ mm}$		$y < 0,8 \text{ mm}$ $y > 2,4 \text{ mm}$	
	PCB Tickness	$y \leq 105 \mu$		240 μ	
Copper-filled Vias	Drilling	Mechanical $100 \mu \leq y \leq 125 \mu$		Laser Drilling	
Plugged-and-Covered Vias(with resin)	Drilling	Mechanical $150 \mu \leq y \leq 500 \mu$		$y > 500 \mu$	Min. copper basis 17 μ Min. Milling = 125 μ
	PCB Thickness	$0,8 \text{ mm} \leq y$	$0,6 \text{ mm} \leq y < 0,8 \text{ mm}$	$y < 0,6 \text{ mm}$	
Design	Minimal copper pad	$75 \mu \leq y$	$50 \mu \leq y < 75 \mu$	$y < 50 \mu$	
	Min.Via Spacing (inner layer)	VIA : $170 \mu \leq y$ AUTRE : $250 \mu \leq y$	VIA : $150 \mu \leq y < 170 \mu$ AUTRE : $200 \mu \leq y < 250 \mu$	VIA : $y < 15 \mu$ AUTRE : $y < 20 \mu$	
Coating HAL étain	PCB Thickness	$0,8 \text{ mm} \leq y \leq 3,2 \text{ mm}$		$y < 0,8 \text{ mm}$	White soldermask is not suitable
	PCB Dimensions	420mm x 420mm		$y > 420 \text{ mm}$	
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 with Nickel Gold or HAL Sn is not appropriate
Flex (Kapton / Coverlay)	Spécifications	Dimension max 310 mm x 260 mm Tickness 125 μ cuivre 35 μ Coverlay 25 μ ou 50 μ + Adhésif 50 μ Coating : Chemical Silver		- Coating Nickel Gold - Plated-Through-Hole	