

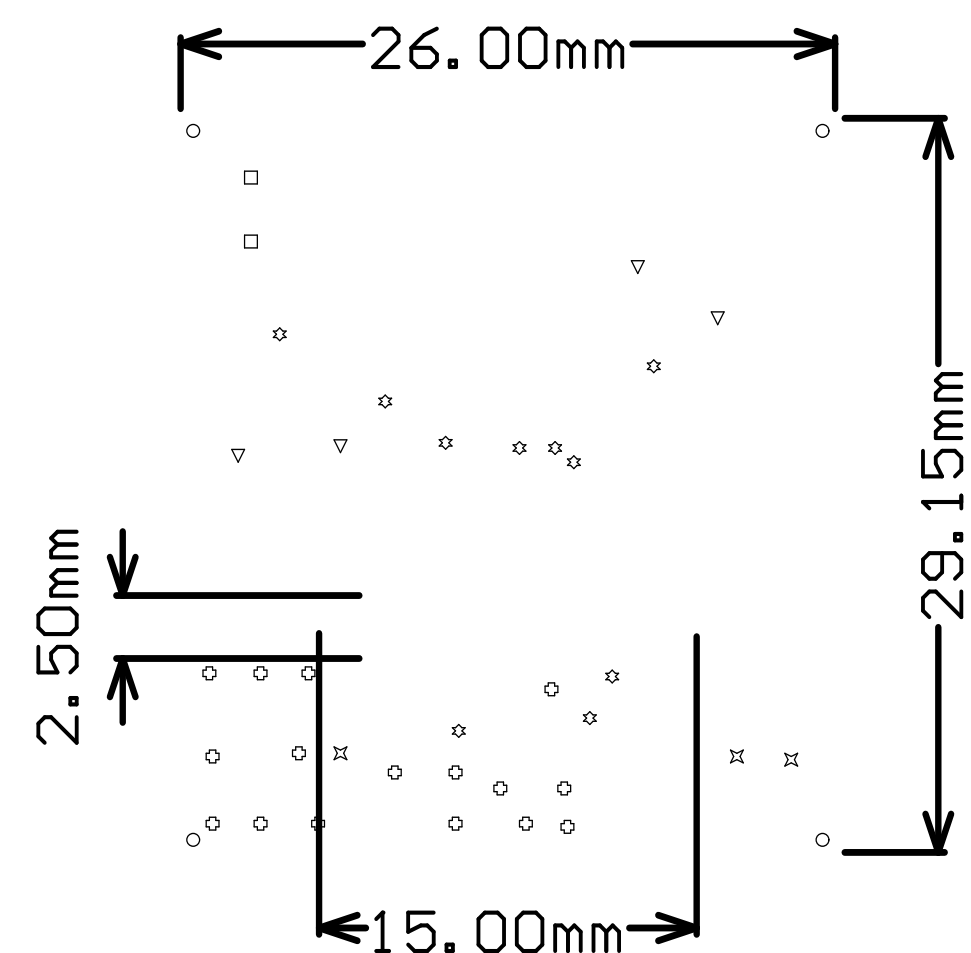



LAYERS: 2
MATERIAL: FR4, RoHS, 94v0
THICKNESS: 1.6mm (~62 mils)
FINISH: Gold (ENIG) RoHS
SILK SCREEN: White
SOLDER MASK: GREEN

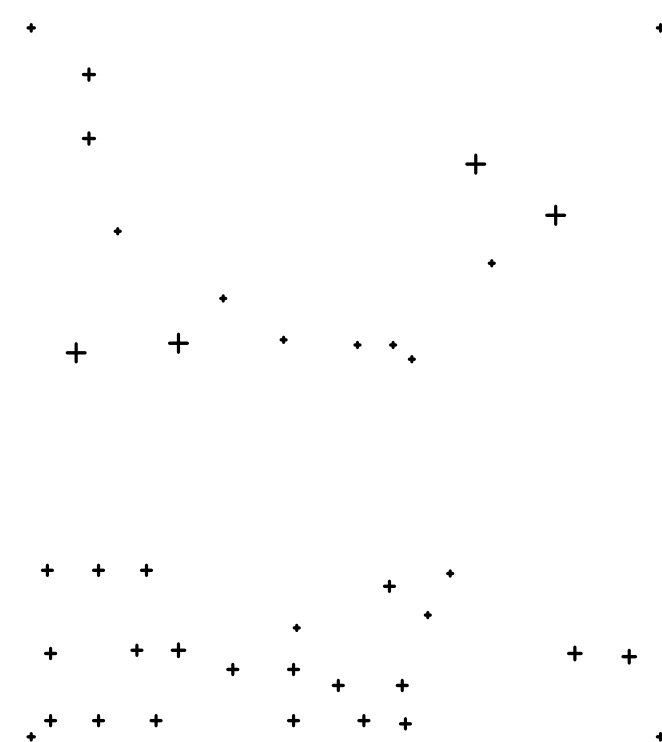
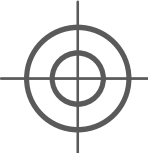
>> IMPORTANT FABRICATION NOTES <<


- 1) Many pads have intentional undersize solder mask openings/oversize pads for physical strength.
- 2) Make top and bottom solder masks exactly as shown. Do _NOT_ "open-up" the solder mask layers. This design uses "solder-mask-defined-pads".
- 3) Silkscreened Legend (white) must be clean and clear. Silkscreen legend must have good registration. Silkscreen is intentionally placed over bare copper. Do not clip silkscreen to solder mask layer.
- 4) Board must be exactly like: N/A

Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad	Pad Shape
☆	10	7.87mil (0.200mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded
○	4	10.00mil (0.254mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded
⊕	16	15.75mil (0.400mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded
⌘	3	19.69mil (0.500mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded
▽	4	28.00mil (0.711mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded
□	2	35.00mil (0.889mm)	PTH	Round	Top Layer - Bottom Layer	Pad	Rounded
	39 Total						



 Crystalfontz 12412 East Saltese Avenue Spokane Valley, WA 99216 USA Tel: (509) 892-1200 http://www.crystalfontz.com Copyright © 2021 Crystalfontz America, Inc.	PROJECT CFA-10107 VERSION h0v4	
	PCB 10107 REVISION h0v4 DATE 11/15/2021	
LAYER Drill Drawing/Notes	FILE NAME 10107 .GD1	



 Crystalfontz 12412 East Saltese Avenue Spokane Valley, WA 99216 USA Tel: (509) 892-1200 http://www.crystalfontz.com Copyright © 2021 Crystalfontz America, Inc.	PROJECT		CFA—10107		VERSION	h0v4	
	PCB		10107		REVISION		h0v4
		DATE		11/15/2021			
LAYER				FILE NAME			
Drill Guide				10107 .GG1			

