

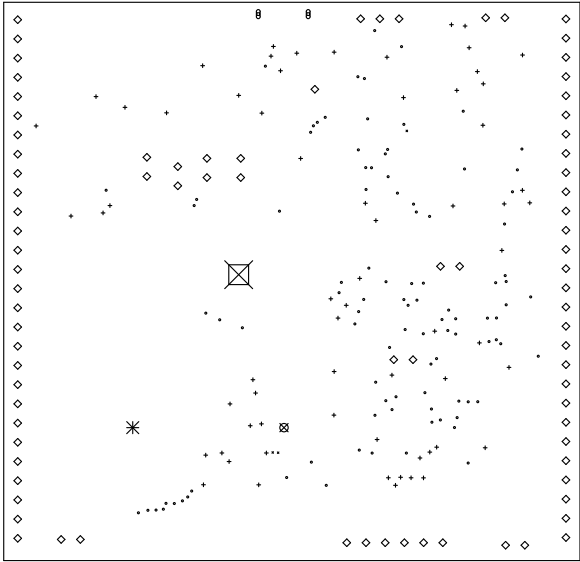
### BOARD CHARACTERISTICS

Copper Layer Count: 2 Board Thickness: 1.6000 mm  
Board overall dimensions: 76.2000 mm x 73.8320 mm  
Min track/spacing: 0.1524 mm / 0.1524 mm Min hole diameter: 0.2032 mm  
Copper Finish: None Impedance Control: No  
Castellated pads: No Plated Board Edge: No  
Edge card connectors: No

Layer Name	Type	Material	Thickness (mm)	Color	Epsilon R	Loss Tangent
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White	1	0
F.Paste	Top Solder Paste		0 mm		1	0
F.Mask	Top Solder Mask	Not specified	0.01 mm	Black	3.3	0
F.Cu	copper		0.035 mm		1	0
Dielectric 1	core	FR4	1.51 mm	Not specified	4.5	0.02
B.Cu	copper		0.035 mm		1	0
B.Mask	Bottom Solder Mask	Not specified	0.01 mm	Black	3.3	0
B.Paste	Bottom Solder Paste		0 mm		1	0
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	White	1	0

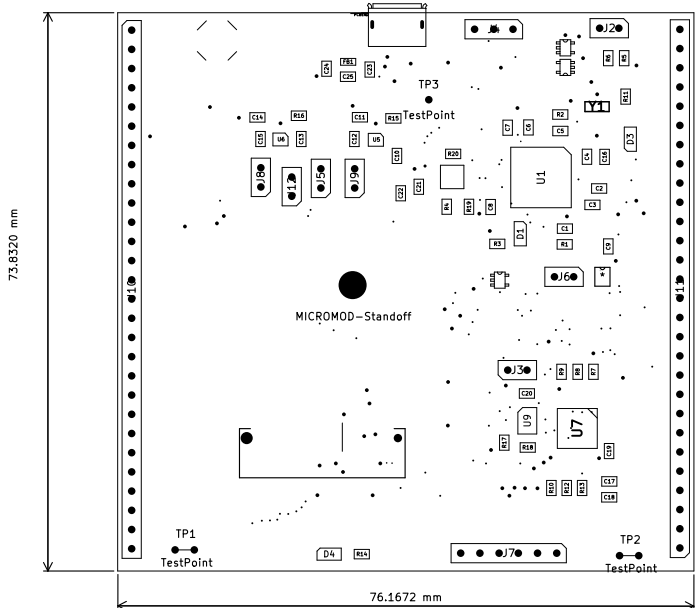
Notes:

1. Hole locations are indicated in separate .drl file, included with the gerber package. That file takes precedence over this drawing.  
2. Board outline is indicated in separate Edge.Cuts file, included with the gerber package. That file takes precedence over this drawing.



Drill Map:

- 0.203mm / 0.0080" (3 holes)
- 0.250mm / 0.0098" (98 holes)
- 0.500mm / 0.0197" (65 holes)
- 0.500mm / 0.0197" (0 holes + 2 slots)
- 1.000mm / 0.0394" (84 holes)
- ⊗ 1.100mm / 0.0433" (1 hole) (not plated)
- \* 1.600mm / 0.0630" (1 hole) (not plated)
- ⊗ 3.700mm / 0.1457" (1 hole) (not plated)



Efabless

Sheet:

File: Fab-Drawing-clear-dev-v1.kicad\_pcb

Title: Clear Chip Development Board

Size: A4

Date:

KiCad E.D.A. pcbnew 7.0.10-7.0.10-ubuntu22.04.1

Rev: REV 1A

Id: 1/1