

Fab Notes Unless Otherwise Specified:

1. Workmanship shall conform to IPC Class 2 Specifications.
2. PCB: 2 Layers (1.6mm) +/- standard tolerance
3. 1 Oz or 1/2 Oz Cu may be used.
4. Hole sizes after plating are defined in the Drill Guide: *.drl
5. Other Drill information can be found in the *.gd1,2,x and *.gg1,2,x Gerber Files
6. Minimums: Trace/space = 8mil / 7mil
7. **NO PCB Vendor Logo, date code, or other markings on PCB. On Rails ONLY!!**
8. **Boar Outline Defined by *.gm1**
9. Finish = ENIG
10. Silkscreen = WHITE
11. Solder mask to be BLACK and shall conform to the requirements specified.
Soldermask both sides.
12. Gerber File Extensions:

- .GTL = Top Layer
- .G1 = Internal Layer, L2(if present)
- .G2 = Internal Layer, L3(if present)
- .GBL = Bottom Layer
- .GTO = Top Silk
- .GTP = Top Pastemask
- .GTS = Top Soldermask
- .GBS = Bottom Soldermask
- .GBP = Bottom Pastemask
- .GBO = Bottom Silkscreen(if present)
- .GM1 = Board Outline/Dimensions
- .GM2 = Fab Notes/Dimensions
- .GM8 = Bottom Designators(if present)
- .GM9 = Top Designators(if present)
- .GM15 = Component Outline
- .GM16 = Border/Title
- .GD1 = Drill Drawing L1-L4
- .GG1 = Drill Guide L1-L4

V-Score Array Details:

25 piece v-scored array.

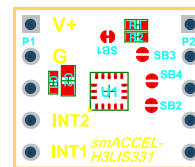
Individual PCB: 12.7 x 15.2mm

Add 7mm rails on sides of Array

Add Fiducials and Tooling Holes on Rails

as needed for proper PCB assembly.

Any Vendor IDs MUST be on Rails ONLY!!



Assembly Details:

NO PCB WASH

NO HIGH PRESSURE AIR-DRY

PCB contains an IC that has the DIE exposed that is easily

Damaged with exposure to water, high pressure air flow, or excessive vibration.

SensorMaestros www.sensormaestros.com	ENGINEER:	TITLE: smACCEL-H3LIS331	
	PCB DESIGNER:		
	DATE: 7/5/2022	PART NO.:	REV: 1.0.0
	FILE NAME: smACCEL-H3LIS331.PcbDoc	DWG NO:	SCALE:

