

**BOARD CHARACTERISTICS**


Copper Layer Count:	4	Board Thickness:	62 mils
Board overall dimensions:	3550.00 mils x 3982.07 mils	Min hole diameter:	7.87 mils
Min track/spacing:	8.00 mils / 8.00 mils	Impedance Control:	No
Copper Finish:	ENIG	Plated Board Edge:	No
Castellated pads:	No		
Edge card connectors:	Yes, bevelled 30deg minimum.		

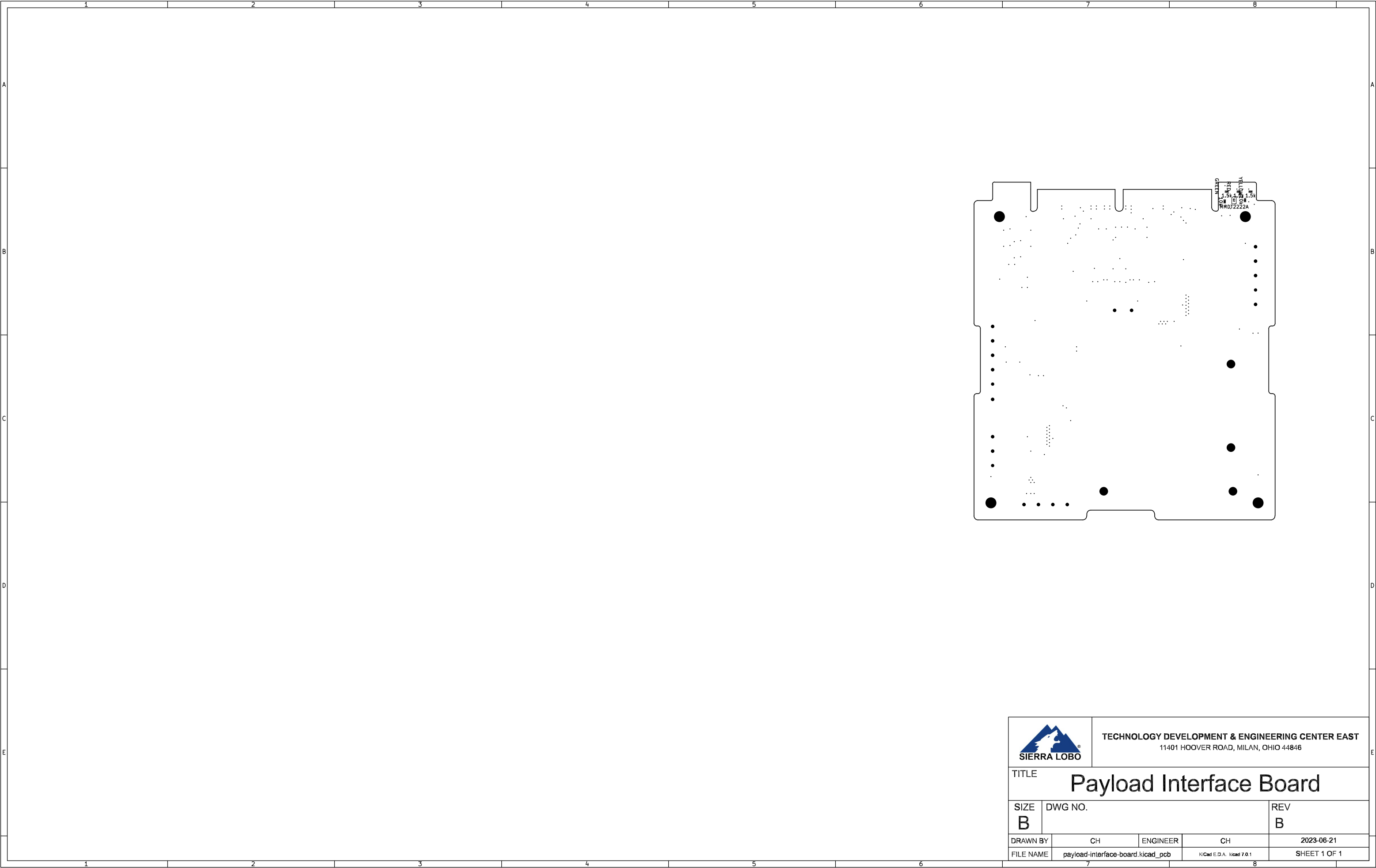
**FAB NOTES:**


1. IPC-6012E Class 2
2. Matte Green soldermask, White silkscreen.
3. Fabricate on SH260
4. 1oz outer, 1oz inner copper.
5. Bevel edge connector 30 degrees minimum.
6. Board target thickness shall be 0.062"
7. Immersion gold plating on finger thickness shall be 2U"

**VENDOR ASSEMBLY NOTES:**

1. J-STD-001G Class 2
2. BOM provided with submitted files shall be the controlling document for component information.
3. Do not apply solder to pads of DNP components
4. Assemble with leaded solder.

		TECHNOLOGY DEVELOPMENT & ENGINEERING CENTER EAST 11401 HOOVER ROAD, MILAN, OHIO 44846		
TITLE Payload Interface Board				
SIZE B	DWG NO.			REV B
DRAWN BY	CH	ENGINEER	CH	2023-06-21
FILE NAME	payload-interface-board.kicad_pcb		KiCad E.D.A. kicad 7.0.1	SHEET 1 OF 1



		TECHNOLOGY DEVELOPMENT & ENGINEERING CENTER EAST 11401 HOOVER ROAD, MILAN, OHIO 44846			
TITLE Payload Interface Board					
SIZE B	DWG NO.			REV B	
DRAWN BY	CH	ENGINEER	CH	2023-06-21	
FILE NAME	payload-interface-board.kicad_pcb		KICAD E.D.A. Kicad 7.0.1		SHEET 1 OF 1