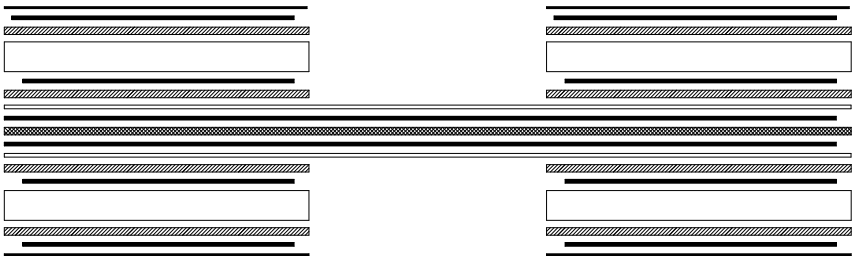


PCB STACKUP 6 layers : 2Ri-2F-2Ri



LAYER TYPE		THICKNESS	Dk	TRACE WIDTH	SPACING	IMPEDANCE
SOLDERMASK, COLOR = RED, PMS 200C	LPI					
LAYER 1	COPPER	0.035				
PREPREG	1x7628M	0.200				
CORE		0.510				
LAYER 2		0.035				
PREPREG	1x7628M	0.200				
NO FLOW PREPREG / PARTIAL COVER LAYERCOVER LAYER		0.035	3.40			
LAYER3	COPPER	0.035				
POLIYMIDE		0.100	3.50			
LAYER 4	COPPER	0.035				
NO FLOW PREPREG/ PARTIAL COVER LAYER		0.035	3.40			
PREPREG	1x7628M	0.200				
LAYER 5	COPPER	0.035				
CORE		0.510				
PREPREG	1x7628M	0.200				
LAYER 6	COPPER	0.035				
SOLDERMASK, COLOR = RED, PMS 200C	LPI					

1. GENERAL

- 1. Fabricate PCB in accordance with IPC-6012B, Class 2; per IPC-6011, and IPC-6013C, Class 2.
- 2. Bow and twist shall be 0.75% or less, measured in accordance with IPC-TM-650, method 2.4.22.
- 3. Artwork may be adjusted by the PCB manufacturer to compensate for manufacturing process tolerances or impedance requirements.
- 4. All dimensions in millimeters.

2. MATERIALS

- 1. Laminate and Prepreg (B-Stage) to be in accordance with IPC-4101C/128
- 2. Copper foil to be in accordance with IPC-4562. Unless otherwise specified, all copper weight for all layers to be 35um (1 oz).

3. HOLES

- 1. Hole sizes are finished hole sizes after plating.
- 2. No hole breakout allowed.

4. FINISH

- 1. Apply Liquid Photo Imageable solder mask, color RED, per IPC-SM-840, Class H, to both sides of the board over bare copper.
- 2. All exposed conductive pattern areas not covered with soldermask or other plating shall be plated with Electroless Nickel Immersion Gold, ENIG, per IPC-4552.
- 3. Solder mask images that are the same size as the component pads may be enlarged, but shall not be enlarged beyond 0.05mm per side. all other solder mask shall not be enlarged.
- 4. Soldermask overlap on smd lands is not allowed.

5. MARKING

- 1. UL logo, manufacturer's indentification and date code letter shall be rendered on the top side of the pcb in silkscreen.
- 2. Silkscreen on both sides of the pcb using non-conductive epoxy ink, color white. No ink shall be on exposed pads.

6. TEST REQUIREMENTS

- 1. 100% net list verification using the netlist information in the ODB++ data.

DGND2 and DGND3 are connected/shorted at one single point on layers 3,4,5 of the PCB at X,Y: -3.85,5.85
THIS IS AN INTENDED SHORT !!!!!

Min. Track/Gap {Outer} : 0.12/ 0.15
Min. Track/Gap {Inner} : 0.13 / 0.15

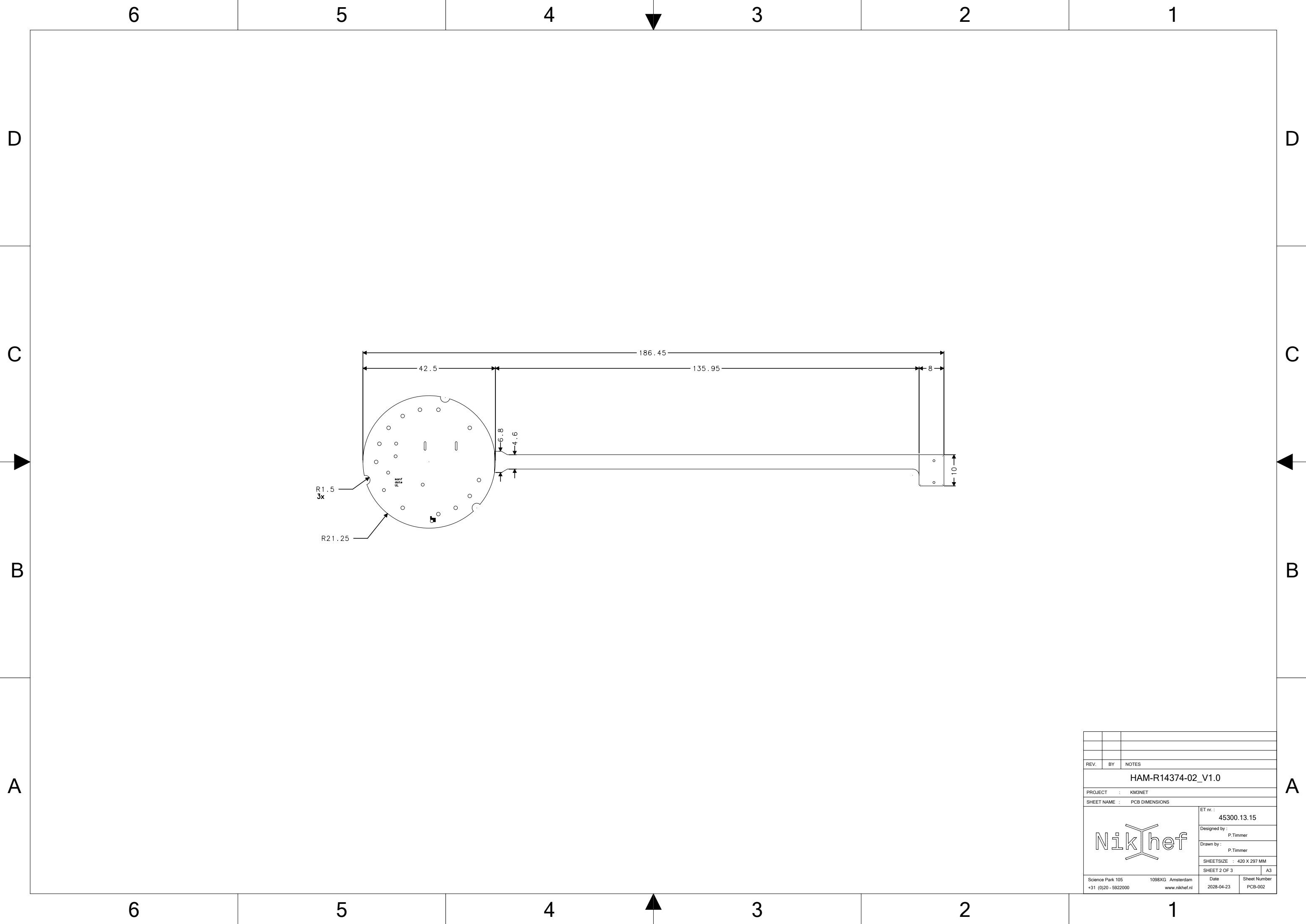
SMD Pads Top : 187
SMD Pads Bottom : 107
Minimum Pitch : 0.50mm
Minimum Annular Ring : 0.2mm

Total no. of holes : 78
Largest hole size : 1.2
Smallest hole size : 0.4
No. of hole sizes : 5

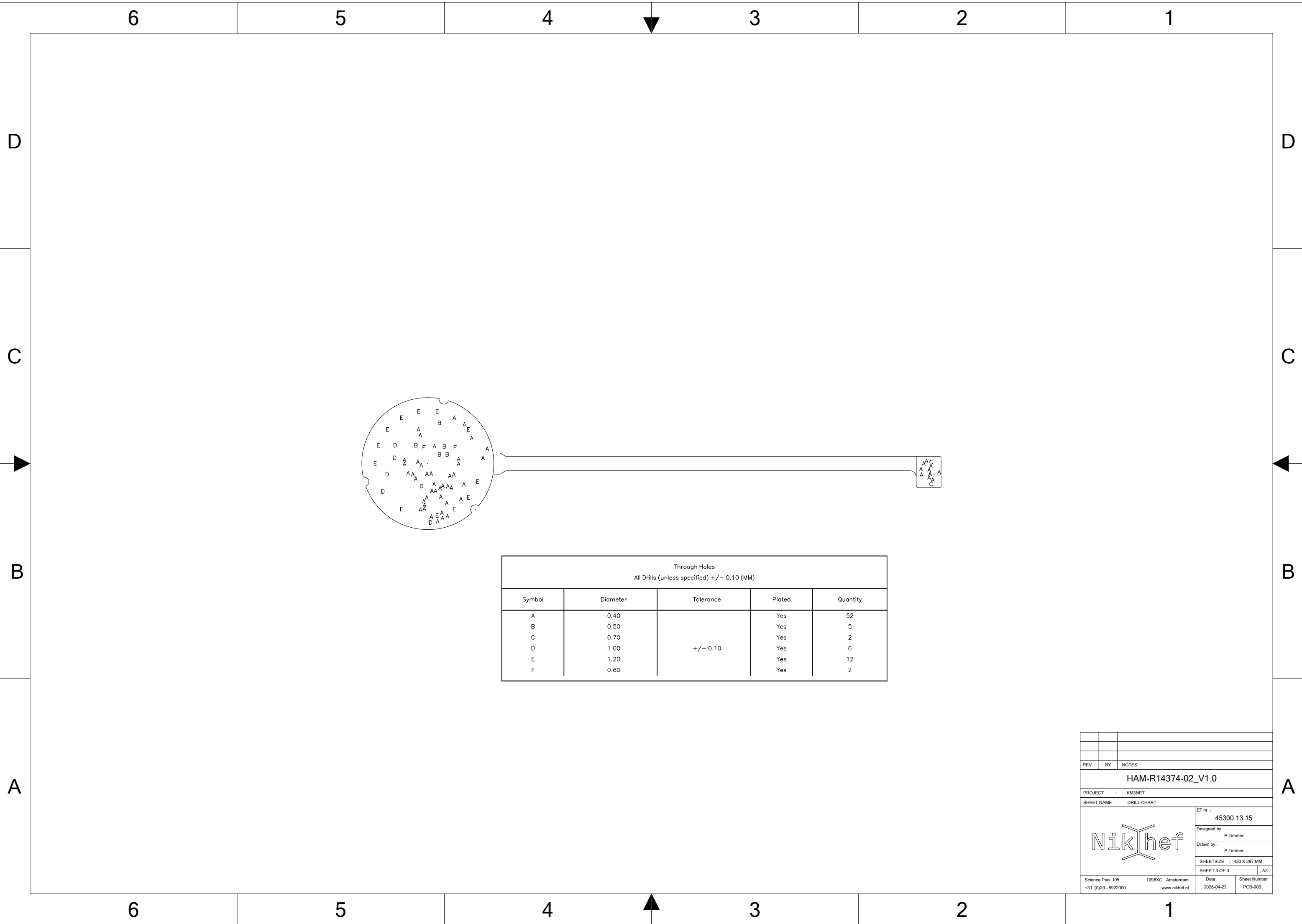
Connector Gold : No
Peel-Off mask : No
Scoring : No
Internal routing : No

Manufacturer's Logo : Yes, See drawing PCB-002
Date Marking : Yes, See drawing PCB-002
UL Marking : Yes, See drawing PCB-002

REV.			BY			NOTES		
HAM-R14374-02_V1.0								
PROJECT : KM3NET								
SHEET NAME : PCB SPECIFICATION								
						ET nr. : 45300.13.15		
						Designed by : P.Timmer		
						Drawn by : P.Timmer		
						SHEETSIZE : 420 X 297 MM		
SHEET 1 OF 3						A3		
Science Park 105 +31 (0)20 - 5922000			1098XG Amsterdam www.nikhef.nl			Date 2028-04-23		Sheet Number PCB-001



REV.	BY	NOTES
HAM-R14374-02_V1.0		
PROJECT : KM3NET		
SHEET NAME : PCB DIMENSIONS		
		ET nr. : 45300.13.15
		Designed by : P.Timmer
		Drawn by : P.Timmer
		SHEETSIZE : 420 X 297 MM
SHEET 2 OF 3		A3
Science Park 105 +31 (0)20 - 5922000	1098XG Amsterdam www.nikhef.nl	Date 2028-04-23
		Sheet Number PCB-002



REV.	BY	NOTES
HAM-R14374-02_V1.0		
PROJECT : KM3NET		
SHEET NAME : DRILL CHART		
		ET nr. : 45300.13.15
		Designed by : P.Timmer
		Drawn by : P.Timmer
		SHEETSIZE : 420 X 297 MM
SHEET 3 OF 3		A3
Science Park 105 +31 (0)20 - 5922000	1098XG Amsterdam www.nikhef.nl	Date 2028-04-23
		Sheet Number PCB-003