

ATWINC3400-XPRO

PCB Specification

Table of Contents

1	GENERAL INFORMATION.....	3
1.1	Board identification.....	3
1.2	Contact persons for PCB issues.	3
2	PCB SPECIFICATION	3
2.1	Manufacturing data	3
2.2	Layer stack-up	3
2.3	Gerber files.....	5
2.4	Special via considerations	5
2.5	Placement of fabrication ID mark	5
3	PANELIZING.....	5
4	QUALITY OF SILKSCREEN LAYERS	5

1 General information

1.1 Board identification

Name: ATWINC3400-XPRO

Board identification number: A08-2472 Rev3

1.2 Contact persons for PCB issues.

- Manager: Karthik Narayana, Karthik.Narayana@Microchip.com, +91 80 3090 4727
- Design Engineer: ChandanKumar Nanjegowda,
ChandanKumar.Nanjegowda@microchip.com, +91 80 3090 4630

2 PCB specification

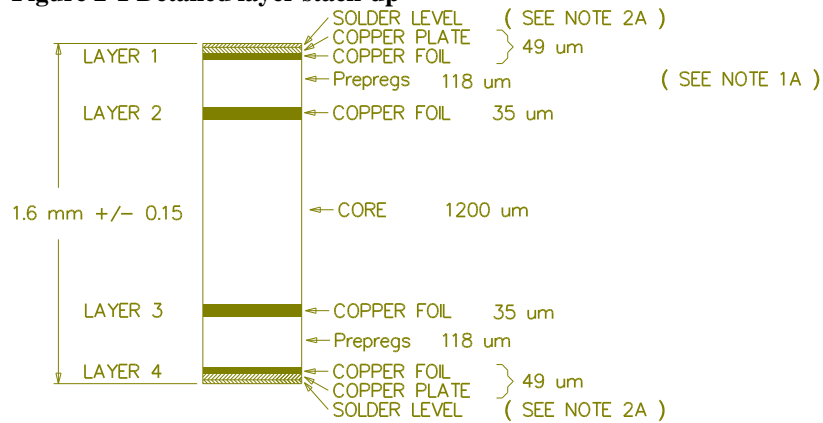
2.1 Manufacturing data

- Size: 56.525mm x 30mm
- PCB material: FR-4, 1.6mm thickness
- Layers: 4
- Finish: ENIG
- Minimum via hole size: 0.25 mm
- Minimum via pad size: 0.55 mm
- Minimum track width: 0.175 mm (6.9mil)
- Minimum spacing: 0.15 mm (5.9mil)
- Solder mask color: Dark BLUE
- Silk-screen color: White

2.2 Layer stack-up

Figure 2-1 shows the detailed layer stack-up for this PCB.

Figure 2-1 Detailed layer stack-up



NOTE 1A: DIELECTRIC FR4
2A: SURFACE PROTECTION: Chemical Gold
THE BOARD MUST BE RoHS COMPLIANT

DETAIL A (CROSS-SECTION)

SCALE = NONE

Note: Adjust core thickness to get board thickness of 1.6mm \pm 0.15mm

2.3 Gerber files

Table 2-1 Layer stack up corresponding Gerber files (listed from top to bottom)

File name	Description
ATWINC3400-XPRO_pcb_rev3.GTP	Gerber file for top paste-mask
ATWINC3400-XPRO_pcb_rev3.GTO	Gerber file for top overlay (silkscreen)
ATWINC3400-XPRO_pcb_rev3.GTS	Gerber file for top solder-mask
ATWINC3400-XPRO_pcb_rev3.GTL	Gerber file for top layer
ATWINC3400-XPRO_pcb_rev3.GP1	Gerber file for internal negative plane layer 1 (GND layer)
ATWINC3400-XPRO_pcb_rev3.G1	Gerber file for internal signal layer 1 (power layer)
ATWINC3400-XPRO_pcb_rev3.GBL	Gerber file for bottom signal layer
ATWINC3400-XPRO_pcb_rev3.GBS	Gerber file for bottom solder-mask
ATWINC3400-XPRO_pcb_rev3.GBO	Gerber file for bottom overlay (silkscreen)
ATWINC3400-XPRO_pcb_rev3.GM1	Gerber file for mechanical 1 layer (board outline)
ATWINC3400-XPRO_pcb_rev3.DRR	Drill file report
ATWINC3400-XPRO_pcb_rev3.TXT	Hole Drill File

2.4 Special via considerations

All plated through holes with solder mask covered area need to be filled/tented on both side of the PCB.

2.5 Placement of fabrication ID mark

The fabrication ID mark should be placed on the bottom side.

3 Panelizing

When making panels for this board the following issues should be considered.

- Fiducial marks should be placed on the panel.

4 Quality of silkscreen layers

The silkscreen layers for the PCB must be of high quality for several reasons:

- Very small text is used
- Text is close to pads and therefore the mask must be centered properly on the board
- The PCB is used for development boards and therefore the silkscreen is an essential part of the overall product quality.