

# ATZB-RF-233-1-C ZigBit **PCB** Specification

## Table of Contents

<b>1</b>	<b>GENERAL INFORMATION.....</b>	<b>3</b>
<b>1.1</b>	<b>Board identification.....</b>	<b>3</b>
<b>2</b>	<b>PCB SPECIFICATION .....</b>	<b>3</b>
<b>2.1</b>	<b>Manufacturing data .....</b>	<b>3</b>
<b>2.2</b>	<b>Layer stackup .....</b>	<b>4</b>
<b>2.3</b>	<b>Gerber files.....</b>	<b>4</b>
<b>2.4</b>	<b>Via – Tenting.....</b>	<b>5</b>
<b>2.5</b>	<b>Placement of fabrication ID mark .....</b>	<b>5</b>
<b>3</b>	<b>PANELIZING.....</b>	<b>5</b>
<b>4</b>	<b>IMPEDANCE CHECK .....</b>	<b>6</b>

## 1 General information

### 1.1 Board identification

Name: ATZB-RF-233-1-C PCB

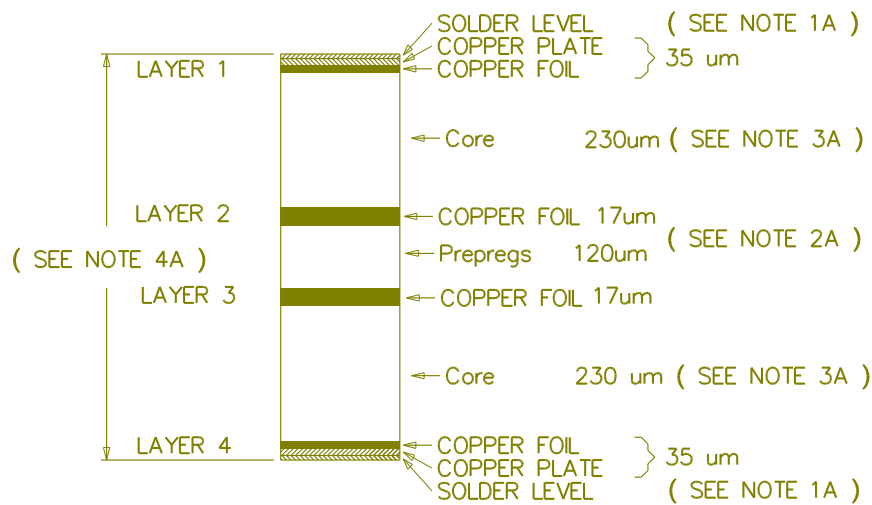
Board identification number: A08-1566 Rev 2

## 2 PCB specification

### 2.1 Manufacturing data

- Size: 25 mm x 20 mm
- PCB material: FR-4(S1000 and S1000B) –Refer Detailed Stack Up
- Layers: 4
- Finish: ENIG
- Minimum hole size: 0.2 mm
- Minimum pad size: 0.6 mm
- Minimum track width: 0.152mm
- Minimum spacing: 0.15 mm
- Solder mask color: Black
- Silk-screen color: White
- Impedance controlled PCB

## 2.2 Layer stackup



### NOTE :

- 1A: SURFACE PROTECTION: Chemical Gold
- 2A: DIELECTRIC FR4 (S1000B)
- 3A: DIELECTRIC FR4 (S1000)
- 4A: TOTAL THICKNESS TO BE DEFINED BY MANUFACTURER WITH RESPECT TO STANDARD FINISHING

THE BOARD MUST BE RoHS COMPLIANT

## DETAIL A ( CROSS-SECTION )

SCALE = NONE

## 2.3 Gerber files

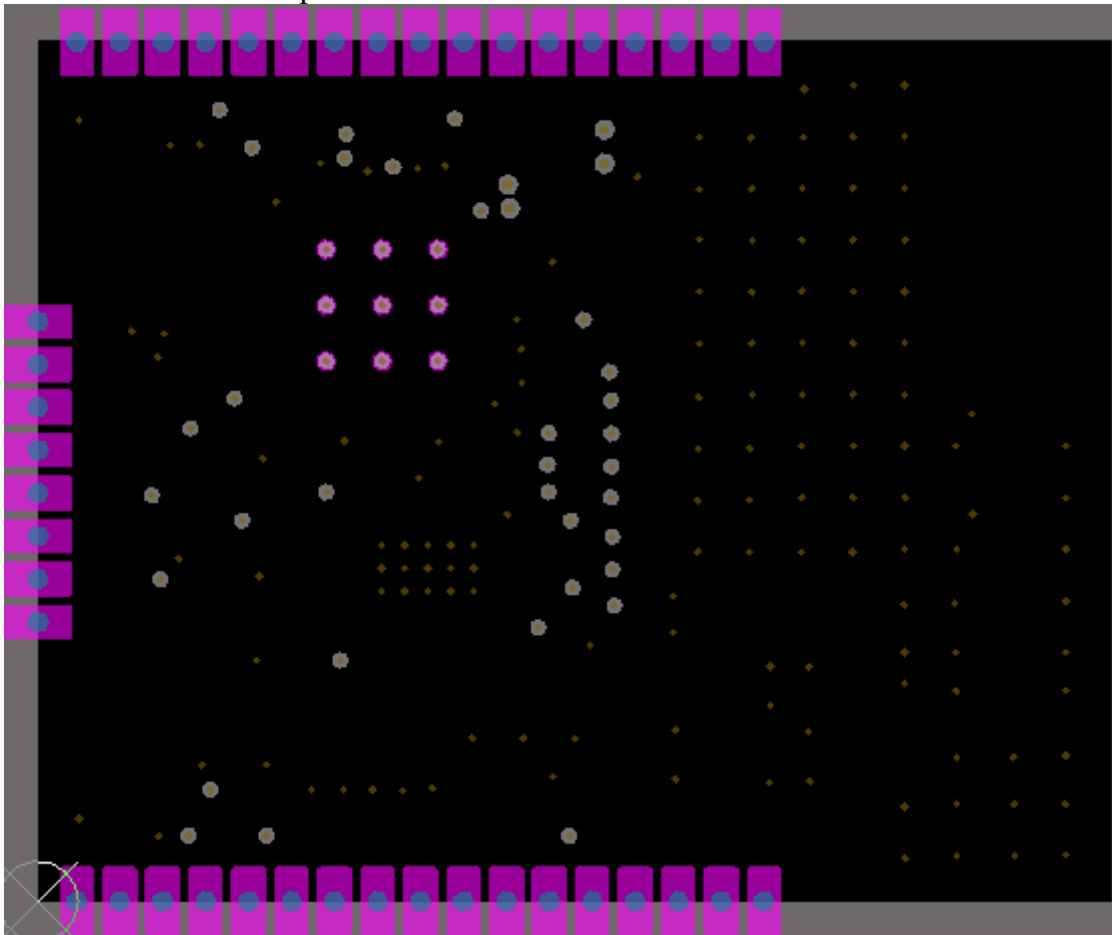
### Gerber File Description

File name	Description
A08-1566_Rev2.GTL	Gerber file for Top layer –L1
A08-1566_Rev2.G1	Gerber file for Gnd layer –L2
A08-1566_Rev2.G2	Gerber file for Power layer –L3
A08-1566_Rev2.GBL	Gerber file for Bottom signal layer –L4
A08-1566_Rev2.GTS	Gerber file for Top Solder Mask
A08-1566_Rev2.GBS	Gerber file for Bottom Solder Mask
A08-1566_Rev2.GTO	Gerber file for Top Over Lay(Silk Screen)
A08-1566_Rev2.GBO	Gerber file for Bottom Overlay (Silk Screen)
A08-1566_Rev2.GTP	Gerber file for Top Paste
A08-1566_Rev2.GM1	Gerber file for Board outline
A08-1566_Rev2.DRR	Drill file report
A08-1566_Rev2.TXT	Drill file
A08-1566_Rev2.DRL	Drill file
A08-1566_Rev2.GG1	Gerber file for drill guide
A08-1566_Rev2.GD1	Geber file for drill layer

## 2.4 Via – Tenting

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

Note: Solder Mask is opened for 9 Vias in Bottom solder mask .



## 2.5 Placement of fabrication ID mark

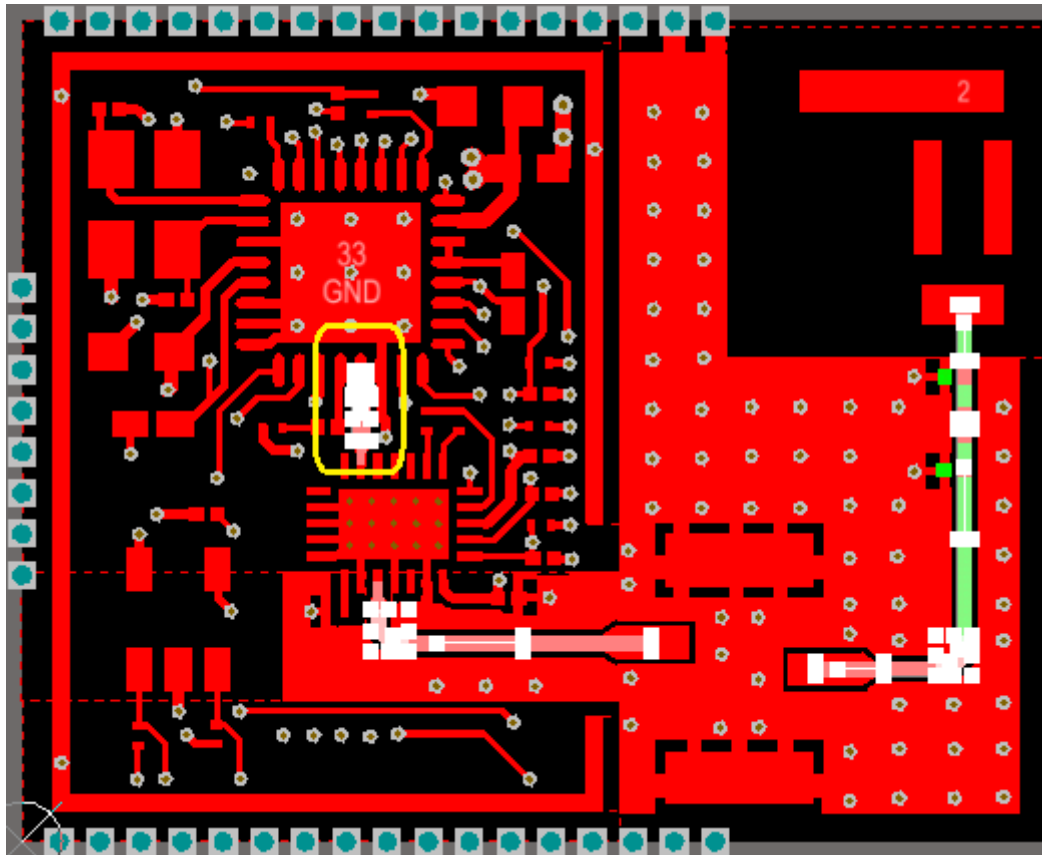
The fabrication ID mark should be placed on the bottom side.

## 3 Panelizing

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

- Fiducial marks should be placed on the panel.

## 4 Impedance Check



Highlighted Track (not inside yellow box) needs to be 50 Ohm impedance at 2.4GHz. Track highlighted inside yellow box too is 50ohm but of different type of waveguide. Priority is on the highlighted track (not inside yellow box).