BUILD INSTRUCTION – PCB ASSEMBLY

Document ID:	4467_006
Revision:	1.0.0
Date:	2020-03-03
Author:	KJP
Classification:	CONFIDENTIAL

1. GENERAL

Item	Value			Comments
Nordic Product ID	PCA20054 - Direction Finding Antenna Board			
Vendor	InterPhone Service LLC	Site	Mielec	
Build	1.0.0			
Final test on site		yes	X no	

1.1. Build Codes

This instruction shall apply for the following product Build Codes:		
Code	Subcon Product ID	Description
1.0.0	PCA20054	Direction Finding Antenna Board

1.2. Service type

Volume production		Pilot series production /Pre production	X

1.3. Synopsis

Instructions for assembly of Printed Circuit Board PCA20054, build 1.0.0.

2. Document Control

2.1. Revision history

Rev.	Date	Description of change
1.0.0	2020-03-03	First version

2.2. Changes since last revision

Change #	Changes

2.3. Approval history

Revision	NOD responsible	Supplier approval date and resp	Comment
1.0.0	KJP		

3. PCB production and assembly files

The PCB production and assembly files are compressed into one single Zip archive file.

Zip archive file name	Revision
PCA20054_Build-Code_IPS_1_0_0.zip	1.0.0

Below is an overview of which files that can be found in which folders in the Zip archive file.

3.1. Bill Of Materials (BOM)

Folder path and file name	Revision
Assembly\PCA20054_BOM_IPS.xlsx	1.0.0

3.2. PCB laminate specification and PCB layer stackup

Folder path and file name	Revision
PCB \400503_PCB_Laminate_Specification.pdf	1.0.0

3.3. Gerber plot files

The Gerber file extension naming convention conforms to general industry practice:

.GTL	Top Layer
.GBL	Bottom Layer
.G1, G2	Mid-layer 1 and 2
.GTO	Top Overlay
.GBO	Bottom Overlay
.GTP	Top Paste Mask
.GBP	Bottom Paste Mask
.GTS	Top Solder Mask
.GBS	Bottom Solder Mask
.GM1	Mechanical 1 (Used as board outer boundary)

Copper layers	Folder path and file name	Revision
Top Layer	PCB\400503.GTL	1.0.0
Mid-layer 1	PCB\400503.G1	1.0.0
Mid-layer 2	PCB\400503.G2	1.0.0
Bottom Layer	PCB\400503.GBL	1.0.0

The Gerber files are generated with Embedded apertures according to the RS274X standard.

Gerber format:
Units: Inches
Format: 2:4 (the number of digits before and after the decimal point)
Leading zeros suppressed

3.4. NC Drill files

File type	Folder path and file name	Revision
Binary format drill file	PCB\400503.DRL	1.0.0
Drill report	PCB\400503.DRR	1.0.0
ASCII format drill file	PCB\400503.TXT	1.0.0

NC drill files format:
Units: Inches
Format: 2:4 (the number of digits before and after the decimal point)
Leading zeros suppressed

3.5. Pick and Place file

Folder path and file name	Revision
Assembly\400503_Pick_And_Place.csv	1.0.0

3.6. Assembly drawings/schematic/layout/instructions

Folder path and file name	Revision
Assembly\400503_Assembly_Drawing.pdf	1.0.0
Assembly\PCA20054_Schematic_And_PCB.pdf	1.0.0

3.7. Production Test Specification

Folder path and file name	Revision
N.A.	

4. Marking / Stick-on label specification

Label 1 shall have the following information:

PCA20054
Rev. 1.0.0
Prod. Year. Week
Serial number

Fields that begins with "#" is a dynamic field.
Prod. Year. Week shall contain the production year and week of the labeled kit. Example: *2010.01*
Serial number is a running number. It shall contain a unique serial number for the labeled kit. The first kit produced shall have serial number: *10000001* the second kit *10000002* and so on.
The font used on the label shall be Arial size 6.
The first line shall be in **bold.**
The three last lines shall be in regular.
Use the label as specified in the BOM file.

5. Assembly instructions

6. Packing

The PCA20054 module shall be packed in an ESD bag.