

Oak Sensor Dist

V1.4.001

PCB Production Instructions

Originally authored by Toradex AG. This work is now available under the terms and conditions of the Creative Commons License 'Attribution CC BY'

Details of which can be found here: <http://creativecommons.org/licenses/by/3.0/>



1. Overview, Statistics

1.1. PCB

PCB type	rigid
Insulator material	FR4
Board Size Extents	71 X 13 mm (final size)
Layer Count	2
Board thickness	1.6mm
Copper thickness	35µm
Hole count	49
Solder mask	on both sides
Silkscreen	no
Surface	HAL Pb-free

1.2. Statistics

Through Holes	49
Minimum line width	0.15mm
Smallest drill diameter	0.3mm
Smallest annular ring	0.15mm
Minimum spacing	0.15mm

1.3. Assembly

Assembly sides:	SMD components on top side
SMD component count	21
THT component count	1



1.4. Notes on handling

IC2 / Sharp-GP2Y0A60SZ0F is not RoHS compatible, e.g. its plastic casing will be destroyed by the temperature in the reflow oven.

Notes

- Please don't do washing. Washing may deteriorate the characteristics of optical system and so on. Please confirm resistance to chemicals under the actual usage since this product has not been designed against washing.
- Please do not do soldering by reflow and flow-soldering.
- Hence the SMD part needs a manual soldering step after reflow.
 - Please solder under the condition below within 2 times.
 - below 260°C at the tip of soldering tool
 - less than 5 seconds
 - Please take care not to let any external force exert on lead pins when soldering.



2. Layer Stackup

2.1. Sequence of the Gerber Files

Please stack the layers in the order given below:

File name	Description	Format
SoldermaskTop.gdo	<i>Solder mask (component side)</i>	RS274-X
EtchLayer1Top.gdo	top layer / component side	
EtchLayer2Bottom.gdo	bottom layer / solder side	RS274-X
SoldermaskBottom.gdo	<i>Solder mask (solder side)</i>	

2.2. Additional Files

SolderPasteTop.gdo	Solder Paste (component side)	RS274-X
Mechanics.gdo	Mechanical dimensions	RS274-X
ThruHoleNonPlated.ncd	Drill coordinates (plated)	NC drill
ThruHolePlated.ncd	Drill coordinates (non-plated)	NC drill
ContourPlated.ncd	Milling coordinates (non-plated)	NC drill

2.3. Layer thickness

The total thickness of the board must be 1.6mm±0.1 mm

Proposed thicknesses:

	Description	Material	Thickness
	Top Solder Mask		
1	Top Layer	Cu	35u
		Core	1530u
4	Bottom Layer	Cu	35u
	Bottom Solder Mask		
Overall thickness			1.6mm



Revision History

Date	File Name	Initial	Changes
2012-09-23	Oak_Dist_V1_4_Manufacturing_Instructions	dus	Initial release
2011-12-12	050514_Oak_Dist_V1_4_001_PCB_Production_Instructions_2012-01-12	ub	Rename File, correct "Layer thickness"
2012-29-02	050514_Oak_Dist_V1_4_001_PCB_Production_Instructions_2012-02-29	ub	New instruction for manual soldering (see Chapter 2.4)

Disclaimer:

Copyright © Toradex AG. All rights reserved. All data is for information purposes only and not guaranteed for legal purposes. Information has been carefully checked and is believed to be accurate; however, no responsibility is assumed for inaccuracies.

Brand and product names are trademarks or registered trademarks of their respective owners.

Specifications are subject to change without notice.