

GENERAL CONDITIONS FOR THE ASSEMBLY OF THE ELECTRONIC BOARDS

1. TECHNICAL CAPABILITIES

- Panel dimensions for the SMD assembly: usual max. 460 x 400 mm, min. 50 x 40 mm (L x W)
possible 740 x 460mm (one p&p machine)
- Panel dimensions for double wave soldering (THT assembly): 410 x 380 mm
- PCB thickness: 0,5 – 4 mm
- SMD components' dimensions: min. 0201 (0,5 x 0,25 mm)
max. 55 x 55 x 15mm (L x W x H)
odd shape possible with special or gripper nozzle
- Minimum pitch for ICs, connectors: 0,3 mm

Note:

If your project doesn't fit into the above-mentioned specifications, please contact us in order to analyse it if we can provide you the requested operations.

2. TECHNOLOGIES

- SMT assembly on both layers (top and bottom), paste and glue possible to be applied
- SMT soldering by reflow (air or nitrogen)
- Automatic optical inspection (2,5D or 3D AOI)
- Wave soldering and selective (wave) soldering
- In-circuit testing (ICT) and functional testing (FT)

3. INFORMATION NEEDED FOR THE PRODUCTION:

- The quantity of products per production batch and per year.
- The materials list, containing at least the type, the value, the reference and the number of pieces. Additional information as tolerances, alternative material, order code, other specific information about the component (temperature coefficient, isolation type/quality ...) will be very helpful.
The materials used for the final assembly, packaging, or other materials used for the project, must be also mentioned in the list (as a separate part of it).
The materials free issued (sent by the customer) should be marked accordingly.
- The PCB files (Extended Gerber RS-274X format) and drilling/milling files, information about the PCB (number of boards per panel, colours of the solder mask and of the marking, thickness of the PCB and of the copper layer, finishing), information about the stack build for the multilayer PCBs; the Gerber files are mandatory for producing the SMT or adhesive stencils.
We are able to generate the Gerber files from the specialized PCB design files from ORCAD, Protel, Eagle, PADS.
The exported files need to be verified by the board designer because the exporting process can generate errors which we cannot check.
- The SMT assembly file (pick&place file) containing at least the reference, the value, x coordinate, y coordinate, turning angle and case;

- Other information regarding the production:
 - Photos of the assembled board (if available)
 - Specific for the board assembly (labels and their positions, zones which must be protected during the wave soldering, height or position of the components ...)
 - Firmware to be programmed into the microcontroller (or at least the dimension of the microcontroller's firmware), programmer type to be used;
 - Specific for the final assembly (drawings about how the boards must be assembled together or within the case, ...), testing information (specific files for the ICT testing, test procedure for the functional test)
 - Packaging instructions and materials to be used
 - Applicable standards, special technical requests, ...

- For the in-circuit testing (ICT) we need the file containing the complete information about the board, please see the picture nearby, or separate files including:

- Gerber files
- Test pads list and their coordinates
- Net list
- Bill of materials including information about the values' tolerances
- Electronic schematic, including the test points

Altium (Protel-Advanced-PCB, VERSION 5.00) Files (*.pcbdoc)
 Ariadne (PCB Version 7.6) Files (*.pca)
 Cadence Allegro Skill Output Files (*.cad)
 Visula Files (*.paf)
 Cadstar Files (*.paf)
 CADIF 7.0 Files (*.paf)
 CAMCAD Files (*.cc)
 CAMCAD 4.8 Files (*.cc)
 EN-HEF-DIF Files (*.dif)
 FABmaster Files (*.fab)
 Eagle (FABmaster ULP) Files (*.fab)
 GenCAD 1.4 Files (*.cad)
 CAMCAD 4.7 (GenCAD 1.4) Files (*.cad)
 Board Station Neutral File Files (*.*)
 PADS (POWERPCB-V2007) Files (*.asc)
 PADS (POWERPCB-V4.0) Files (*.asc)
 PADS (POWERPCB-V5.0) Files (*.asc)
 Parts & Nets Files (*.csv)
 Protel (PCB file 6 version 2.8) Files (*.pcb)
 SPECCTRA ASCII Files (*.txt)
 TOPCAD 8.01 (Windows Mode) Files (*.bxf)
 Integra Station 3.5 Files (*.bxf)
 TOPCAD 8.01 (DOS Mode) Files (*.bxf)
 Zuken Board Designer Version 2.0 Files (*.pcf)
 CT3xx Board Data Files (*.ctbrd)

Note:
 We prefer to receive the documentation in electronic format for an easier and secure handling and management.