



JonJu Tech Ltd.

(Read the JonJu’s ‘Disclaimer’ (end of document) before using/reading this document)

WP type: Business

Date: 5th February 2021

Title: The Minimum Information Required to Order an Assembled PCB, i.e., PCBA

Contents

What it is all about.....	2
Quick Learn	2
Rigour.....	3
Further Investigation	4
Disclaimer.....	4
Definitions.....	4

Without Prejudice



JonJu Tech Ltd.

(Read the JonJu's 'Disclaimer' (end of document) before using/reading this document)

What it is all about

Ordering the manufacture of PCBAs can be a painful experience because many customers, quite understandably, do not know the information that the manufacturer requires and where and how to get it.

There are times when a customer just needs to know that enough information is in his/her possession to transfer manufacture or design. Having the information described in this document covers this.

It is an uncomfortable reality that manufacturing relationships sometimes falter for some reason, and a pre-requisite for a customer is that it is possible to transfer production of the product to another supplier.

There are arcane details that can be specified in a manufacturing specifications, but for the average application the following will suffice.

Quick Learn

The minimum information required is:

- A Bill of Materials (BoM) – design authority has this.
- PCB manufacturing files (Gerber, drilling) – design authority will have this.
- Component assembly files, e.g. pick and place data (XY) – design authority
 - The number of PCBAs required.
 - The Cu weight – design authority
 - The type of finish, e.g., silver immersion- design authority.
- PCB board specification (layers, thickness, material, e.g., FR4,) – design authority
 - Firmware (software to be programmed into chips) – design authority.

Key Actions/Advice

- It may be possible to manufacture without all the information above, but a discussion with the manufacturer will be necessary if that is the case.
- If specialist help is required to recover from a situation in which only some of the above data is available contact JonJu Tech Ltd. (www.jonjutech.com).



JonJu Tech Ltd.

(Read the JonJu's 'Disclaimer' (end of document) before using/reading this document)

Rigour

The data that's necessary to manufacture a PCBA is generated by CAD software, eg. Altium, PADS, Easy PC, etc. This section doesn't cover the design of electronics so the details of that process aren't considered. In order to purchase a PCBA from a manufacturer it is necessary to provide the following data:

- **A BoM (bill of materials).** This is a list of all the components that are to be attached (usually soldered) to the PCB; their designators (how the components are identified on the board with the silk screen – R1, C1, D1, U3, etc.); part numbers, both for the supplier and the manufacturer; identification of both supplier and manufacturer; quantities of each unique component; and comments such as DNF – do not fit – if it is relevant. This data is usually presented in a spreadsheet, but the CAD software can usually auto-generate it.
- **PCBA Manufacturing Files.** It is necessary to send details of what copper to etch and what not to, i.e. the copper tracking and plane definition for each copper layer; where to drill holes for vias, etc.; and x,y data so that the pick and place machine can place the components in order to make the footprints match up with the pin configurations of the components. This data is usually auto-generated by the CAD software and will be in the form of either an ODB++ folder or Gerber, NC drilling and X-Y data files. Unless the designer has special requirements it is best just to allow the default settings to apply, and generating the above files then becomes simply one or two button pushes on the computer.
- **PCB board specification.** The mechanical details that are not usually defined by the Manufacturing files must be stated separately, usually in a PCB specification text file. The following illustrates the minimum extra detail that is required:
 - **The Cu weight** (this is in ounces/ft²). 1oz Cu is the normal, but up to 4oz Cu may be specified in cases in which a PCB may be required to carry significant current.
 - **The surface finish.** Bare PCBs with simply open copper or tin on their pads will oxidise quite quickly, and this is a problem because the oxidised copper pads will not solder properly. For this reason it is normal to put a protective layer over the pads that are to have a soldering process carried out on them. Either a very thin silver or gold layer is applied, and this known as **silver immersion** or **gold immersion** respectively. There is an alternative process called HASL (hot air solder level), which is a process by which the PCB is immersed in a lead/tin alloy and excess alloy is removed by high pressure air. There are other treatments, but the above are the most common. It is simply necessary to state that either Au or Ag immersion is required.
 - **Conformal coating.** PCBAs are sometimes coated in a high dielectric strength, water proof layer. This is known as conformal coating. This layer should only be applied if it is required for some engineering reason – moisture protection, added barrier against ESD, etc. It will be necessary to generate a document specifying which, if any, components are to be masked so that they are not covered in the coating material, eg. a connector. It also should be noted that PCBAs that have had a conformal coating are almost impossible to re-work and re-solder should modifications be necessary – don't conformally coat a prototype PCBA.
- **Firmware specification.** Many PCBAs have chips on them that need to be programmed. Unless it is the customer's intention to program the PCBAs himself, it is necessary to provide object code (hex file) and possibly programming instructions.
- **Special manufacturing instructions.** If there are any extra requirements, for example a small amount of mechanical assembly, instructions must be written.



JonJu Tech Ltd.

(Read the JonJu's 'Disclaimer' (end of document) before using/reading this document)

All the other detail should be available via the information in the manufacturing files and BoM.

The final thing to note is that all CAD software has DRCs (design rule checks) and these allow the user to set limits such that manufacturing limits are not exceeded, e.g. minimum track gap and width. It is necessary to make sure that the CAD has been configured such that it reflects the manufacturing performance of the manufacturer.

Further Investigation

Clyde Coombs, Happy Holden. *Printed Circuits Handbook*, McGraw Hill

Disclaimer

The information in this WP is provided on an 'as is' basis and no responsibility whatsoever is accepted by the JJ for the accuracy and/or validity of the information herein.

Any reliance placed on the content of this WP is entirely at the reader's risk.

No promise or warranty in respect of any use of the content of this WP, including mistakes or omissions, is offered to its reader by JJ. No legal responsibility is accepted by JJ for any loss, consequential or otherwise, damage, monetary burden, or reputational decline, that might be suffered using the WP's contents.

Definitions

BLE: Bluetooth Low Energy

ESD: electrostatic discharge

FMEA: failure mode and effect analysis

JJ: JonJu Tech Ltd

PCB: printed circuit board without components assembled on it

PCBA: printed circuit board with components assembled on it

Production Release (PR): date at which production is launched without any involvement or supervision from a design authority.

SLEEP: a low current condition that a device can be put into to reduce its requirement for current, and hence longevity if powered by a battery.

WP: white paper