

## Instructions for providing the required files for PCB assembly

The purpose of this document is to describe the files that JOPACO needs to assemble your printed circuit boards and to describe what information should the files contain. Please send us all the information that your company allows. We treat all your information confidentially and it will never be handed over to a third party without your permission.

Putting together a comprehensive and precise package of the manufacturing files is the key to a successful result and fast turnaround time. In order for us to carry out our assembly services to you as smoothly and risk free as possible, we need the following data of your printed circuit board assembly.

- Bill of Materials
- Assembly Drawings
- Pick-and-Place file
- Gerber files
- Additional Product-specific instructions

## Bill of Materials

The purpose of the bill of materials is to describe every component that is to be assembled on the board precisely and to tell the position / reference designator (for example R13, C1002 etc.) of each component on the board. If you would like JOPACO to purchase all the parts for you, is the purchasing based on the bill of materials chart. The bill of materials should be in digital format, most preferably as a Excel chart that contains at least the following information:

- **Position (Reference designator)**
- **Quantity**
- **Description of the component (Value, material requirement, power rating, tolerance etc.)**
- **Precise manufacturer part number / numbers**
  - o Please define, for example, "Any similar", if not limited to a single manufacturer and part number
- **Package / Case type**
- **Manufacturer**

| #  | Ref. Des.  | QTY. | Description   | Value       | Part Number    | Case type | Manufacturer           | Notes                        |
|----|------------|------|---|-------------|----------------|-----------|------------------------|------------------------------|
| 1  | R1, R3, R7 | 3    | Chip Resistor 0.25W 1%                                      | 330kΩ       | Any            | 0805      | Any                    |                              |
| 2  | C1, C2     | 2    | Chip Capacitor X7R 50V 10%                                  | 100nF       | Any            | 1206      | Any                    |                              |
| 3  | C3, C4     | 2    | Low ESR Tantalum Capacitor 16V 20%                          | 2.2uF       | EEEF1C220AR    | CASE-D    | Panasonic              |                              |
| 4  | IC1        | 1    | 8-bit 128kB FLASH Microcontroller                           |             | ATMEGA128-16AU | TQFP64    | Atmel                  |                              |
| 5  | U1         | 1    | Switched Mode Voltage Regulator                             | LM3578AM    | LM3578AM       | SO8       | National Semiconductor | Alternative: LM3578AN        |
| 6  | XTAL1      | 1    | Crystal Oscillator 24.5935MHz                               | 24.5935MHz  | CFPT-9301-FX-A | HC49      | Rakon                  | Farnell Part Number: 1272089 |
| 7  | J1         | 1    | Through Hole Programming Header Connector 14-pin 2,54mm Tin | 16-675-191T | 16-675-191T    | NA        | Aries Electronics      |                              |
| 8  | U2         | 1    | Operational Amplifier                                       | LM741J      | LM741J         | CDIP8     | National Semiconductor |                              |
| 9  | R2, R4, R5 | 3    | NOT ASSEMBLED   | NA          | NA             | 1206      | NA                     |                              |
| 10 | C7         | 1    | NOT ASSEMBLED   | NA          | NA             | CASE-D    | NA                     |                              |

Image 1. Example Bill of Materials

Also the positions that are left unassembled should be included in the BOM. This makes the inspection of the assembly easier in various stages. In order to minimize the possibility of an error, the rows can be highlighted for example using red color.

As the purchasing of the materials and assembly program generation are based primarily on the bill of materials, it is important to be as accurate and careful as possible when filling the data into the excel file.

## Assembly Drawings

The purpose of the assembly drawings is to provide the assembly machine operator with enough information to check that the assembly rotation of each assembled component is correct and every component is on its correct position as specified by the customer. The file can be, for example, as.jpg or .pdf format.

The drawing should clearly indicate pin 1 position of each IC and the reference designator of each component. Other types of common polarity markings are positive terminal marking for tantalum capacitors and cathode marking for diodes.

The assembly drawing is typically made by exporting a image file of the silkscreen layer of the PCB (PADS Layout: File → CAM...). The file should include the options that are shown in image 2.

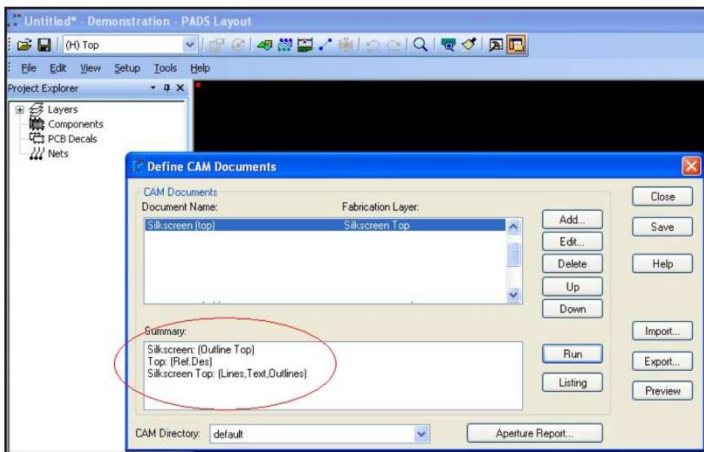


Image 2. Assembly drawing document options in Pads Layout

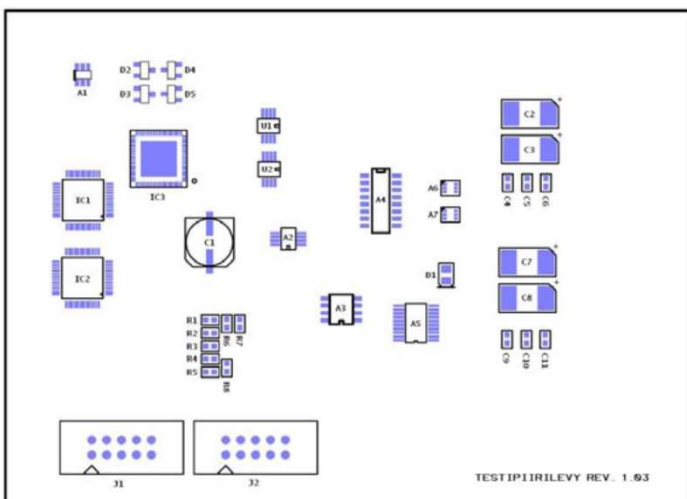
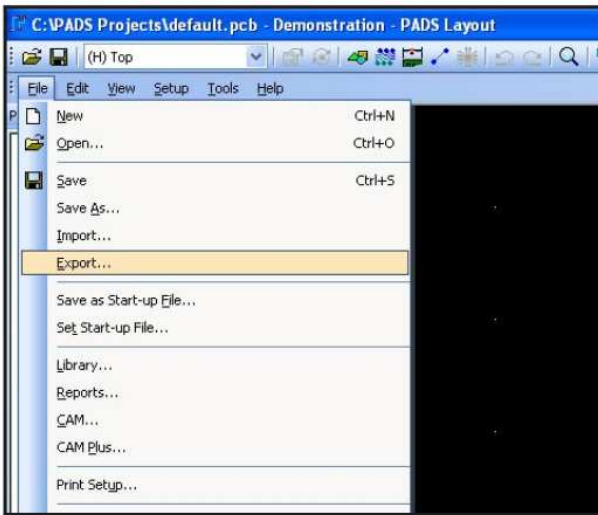


Image 3. Example Assembly Drawing

## Pick-and-Place data

The automatic assembly machine needs to know the exact position of each component that will be assembled on the PCB. This is why we need the center coordinates of each component on the PCB. The easiest way to provide us with this information is to send us the native CAD file exported from your design software. In this case we don't need separate centroid file (usually a tab separated text file) that contains the required coordinates.

Exporting native CAD file from PADS Layout is done by selecting File → Export... → ASCII-file (.asc)



Selecting "Export ASCII file" will result a new window called *ASCII output* to pop up. Please select all the sections to be included in the file.



You can find a more complete list of the native CAD formats that our software supports from the end of this document.

## Gerber files

Exporting the Gerber files from the design software is typically done by using the built in CAM-module. (PADS Layout: File → CAM...). Gerber files are like images of the different material layers of the PCB. Every PCB design software produces Gerber files because the manufacturers of the PCB and the stencil need these files.

The required files in Gerber format are as follows:

- Paste Mask layer(s) (Stencil)
- Copper layer(s) (Conductors)
- Silkscreen layer(s) (Markings on the PCB)
- Drill data (Vias and other holes)
- Solder Mask layer(s) (Solder resist)

All of the files mentioned above should include PCB outlines if possible.

## Additional Product-specific instructions

We also need additional instructions on how to meet your possible special requirements. Special requirements can be for example testing instructions, programming instructions, final assembly instructions, packing instructions, soldering material requirements, PCB specification etc.

## Sending the files

You can send the files described in this document by email to [jopaco@jopaco.com](mailto:jopaco@jopaco.com).

In case you would like additional assist on how to make the production documentation please don't hesitate to contact us at:

Tel. (+358 3) 752 7806 or Email: [jopaco@jopaco.com](mailto:jopaco@jopaco.com)

We are pleased to help you with all the possible questions that might arise in different stages from layout design to the final product. Making perfect files in the production start-up phase is typically difficult and communication plays a key role in achieving the best possible result.

The native CAD formats that our software supports

| Native CAD file   | Filename extension | Additional information                                  |
|---|--------------------|---|
| Accel EDA, Accel Tango, Accel PCAD                        | .pdf               |   |
| Cadence Allegro via Aegis Script                          | .cad               | Requires Aegis Script (Ask JOPACO)                      |
| DIF(Desing Interchange Format), PADS DFT Audit and C-Link | .pdf               |   |
| EE Designer ASCII file                                    | .ala               |   |
| GenCad v. 1.4   | .cad               | CadSoft Eagle<br>(requires gencad.ulp add-on)           |
| IPC-D-356   | .net or .356       |   |
| Mentor Graphics Neutral File                              | .neu               |   |
| OrCad   | .cad               |   |
| PADS PowerPCB, PADS Perform, PADS 2000-2007, PADS Work    | .asc               |   |
| Pantheon PDB file   | no extension       |   |
| P-CAD PDIF Design file                                    | .pcb               |   |
| SCI Cards Neutral file                                    | .cii               |   |
| Supermax CAD  | .ipl               |   |
| Ultiboard   | .cad               | Use the DDF2GenCAD converter<br>supplied with Ultiboard |
| Valor ODB++ CAD Project                                   | .tar or .tgz       | For example Altium Designer<br>(Protel)                 |
| Zuken Visula and Cadstar                                  | .paf               |   |