

## Introduction

The ever increasing requirements for small volume envelopes contribute to integrating electronics to the mechanical design. However, there are numerous solutions; this sheet provides the JPE standard in designing PCB's.

## PCB material & properties

Material	$\rho$	$E$	$\lambda$	$\alpha$
FR4*	1850 kg/m <sup>3</sup>	20 GPa	0.25 W/mK	11 $\mu$ m/mK

\* Flame-Retardant category 4

## Using PCBs in vacuum

FR4 is suitable for use in a vacuum environment. However, to address outgassing issues consider the following:

- Use a hatching pattern instead of (large) copper planes. Use a trace width of 0.2 mm and a trace to trace distance of 0.6 mm to create the hatch.
- Do not use a solder mask layer. Solder mask normally isolates copper traces, so be aware that traces on outer layers will be exposed!

## Component package selection

To be able to solder components by hand (for one-of PCBs, prototyping boards) make sure to use SMD components that are not smaller than a 0805-package (2.0 mm x 1.25 mm). Equally do not use IC's smaller than an LQFP-package (0.4 mm pitch between pins), or use an IC-socket (for example for Ball Grid Array (BGA) packages).

## Component placement

Try to keep all (large) through-hole components on one side and all (small) SMD components on the other (if necessary).

Do not place SMD components too close to holes (through-hole components, via's) as solder can flow into the hole. Also, if possible do not place SMD components too close to each other if manual soldering is required.

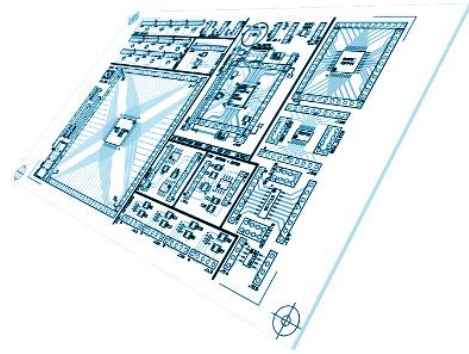
## Connectors on PCB

When selecting connectors and their positions on the PCB, consider the final integration of the PCB so that connectors are always easy to reach.

If (single) wires need to be soldered to pads on the PCB, use a pad dimension of: min 3.0 mm x 1.0 mm (for wires  $\leq$  0.3 mm<sup>2</sup>). Consider implementing a strain relief.

## Electrical layout considerations

- When drawing traces, avoid sharp corners ( $> 90^\circ$ ).
- Use only standard via's (from top to bottom), do not use *blind-* or *buried via's* (expensive to manufacture).
- Do not place via's in pads for the pins of an SMD component (note: thermal pads may contain via's)
- Check the pin reference of each component to make sure the electrical layout corresponds with the component pins.



## Mounting considerations

- Add a couple of mounting holes on the PCB suitable for M3 or M4 (M2 only for small PCBs)
- If the PCB is to be placed in between guide rails, make sure to keep components and pads away from those edges ( $\geq 2.00$  mm)

## Text and visual marks

The Silkscreen layer can be used to display reference text and/or visual marks. Place PCB name, part number, revision and [month/year] reference on the silkscreen for PCB identification. Anything on the silkscreen layer will clip against solder mask, copper or holes (in that order).

Text parameter	Minimum	Recommended
Line width	0.17 mm	$\geq 0.2$ mm
Character height	1.0 mm	$\geq 1.4$ mm

Make sure to mirror text on silkscreen bottom layer! This also applies to component reference text on the bottom layer.

## Export files for manufacturing

All major PCB manufacturers are able to use the *Gerber RS-274X* format. Export all PCB layers in separate files, common layers are:

- Board Outline
- Copper layers (top, bottom, inner layers)
- Drill
- Solder Mask (top and bottom)
- Silkscreen (top and bottom)

Do a net list / DRC check and connectivity check before exporting the Gerber data!

## PCB assembly by machine

If the PCB is to be assembled by machine (pick-and-place components and reflow soldering for example), the following additional actions need to be done:

- Fiducial marks need to be added to the PCB. A fiducial mark is a copper pad ( $\varnothing \sim 1.5$  mm), normally 3 pieces placed in a triangular pattern (each near a corner)
- Export of additional files: Paste Mask (top and bottom) in Gerber RS-274X and Parts Centroids (plain text file or CSV)