EverMAX s.r.o. Oderská 333 196 00 Prague 9
Bank: KB a.s., Prague 1 CZK: 35-6405170267/0100 EUR: 35-6405190217/0100
*Reg.#:*27401006 *Tax#:* CZ27401006
phone.: +420 266 090 100 *Fax:* +420 266 090 101 evermax@evermax.cz http://www.evermax.cz

Recommendations and Requirements for Production Data

(version 3.0, issued 9.4.2010, by Ing. Martin Máša)

1. Introduction

This document was written as description of our technological options, of the format for production documents and technical recommendations adherence to which will facilitate production of your equipment and significantly cut down the risk of appearance of faults and possible problems. Some information is general, other pieces of information are specific for the equipment used by us. This, however, shall not mean that we will not produce the equipment for you, if you are unable to adhere to some of our recommendations for any reason. Do not hesitate to address us any time, we shall always try to find a suitable solution to your needs and requirements. We manage to address even off-standard situations flexibly and we have got rather extensive experience in this field. We would be happy to advise you also in the case that you will not find answers to your queries in this document or they will not be understandable enough. **We are looking forward to cooperation with you!**

2. Printed Circuit Boards

2.1. Dimensions of PCB and Panels

Regarding technology, we are able to process PCB (printed circuit boards) or panels from size 51 x 51 mm to 457 x 508 mm (including possible technological edges) in the automated line (Fig. 7). Mostly it is advisable to **panelize** PCB (particularly with smaller dimensions) for automated production. The optimum size of panel depends on many features, thus consult it with us, please, particularly in case a larger production series. In case of smaller production series we mostly recommend that the longer side of the general motive on the panel should not exceed approximately 230 mm so that the common width of stencil - 300 mm – could be used (more in details in Chapter 5.1 Uniprint). If the shorter side of PCB panel is up to 150 mm, in most cases two motives can be placed on a single stencil (e.g. for double sided fitted PCBs) while satisfying our conditions for the stencil and this way costs for production of another stencil can be saved.

PCB or panels must have **equal dimensions** and **their external sides must be parallel**. The ideal case is a milled circuit or a V-groove produced with a groove machine. Cutting with scissors is not always precise adequately (is highly depends on care of the PCB manufacturer). PCB must not bend and it must be stiff enough (this means adequately deep grooving or sound milling in the panelling). If you provide for PCB production by yourselves, we recommend you to warn the producer that the PCBs will be used for production in automated machines and that they must meet the above quality requirements.

2.2. Methods to Separate Panels

V-grooves or so called “bridge” milling are the most frequent methods to separate PCBs in panelling or from technological environment. The cheapest method (in PCB production and in subsequent separation of PCBs) uses to be V-grooving (Fig. 1). In some cases rather rough surface of PCB edge after separation and rather worse accuracy in dimensions (usually with tolerance approximately ±0.3 mm) are felt as disadvantage, compared with milling, where the tolerance uses to be ±0.2 mm. The “bridge” milling is of two types. The more frequented method is using an outside bridge (Fig. 2) which does not interfere in the PCB area.

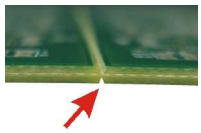

Fig. 1
V-grooving

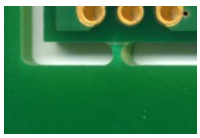

Fig. 2
Outside bridge

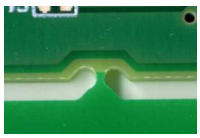

Fig. 3
Internal bridge

But being cut off, there is a residue on the PCB edge – outstanding burr, which is usually ground flat and this process rises the price for PCB separation. When using an internal bridge (Fig. 3), the mill runs at the bridge partially into the PCB area, and this must be considered in PCB design (particularly using suitable distance of components, connection and copper areas from the PCB edge near the bridge). But we have the advantage that we can cut off the bridge in such a way that the outstanding burr does not protrude outside PCB contour and thus it does not need removing.

When panelling PCBs that have components protruding outside the contour (e.g. connectors), V-grooves cannot be used for PCB separation, because then the PCBs could not be separated with a knife cutter. Even in case of milled panels with separating bridges, the bridges should not be placed near components protruding over the PCB edge (e.g. connectors). For easy use of a pneumatic cutter, the milled groove should be long 19 mm or more from one side (Fig. 4, dimension "A") and from another 6 mm or more (dimension "B") on each outside bridge, and no components should be in this area on the bottom side of PCB (red area "C") in distance 2 mm (dimension "D") from the PCB edge.

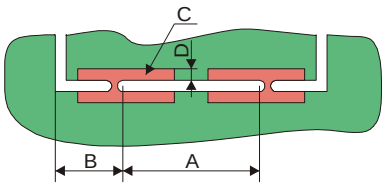

Fig. 4
Specification of dimensions for using pneumatic cutter

2.3. Recommended PCB Design – Distance of Components

We recommend to adhere to the minimum distance of neighbouring components or their pads 0.5 mm, or more, even in the case when they have adjacent pads connected. It is advisable to keep the minimum distance of components 1 mm from the PCB edge.

The minimum distance of the pad edge or of the body of SMD component from the PCB edge in direction of passing through a mounting line (usually the longer side of PCB or of panel) is **4 mm**, and **6 mm** at the bottom side of double-side fitted PCBs. If the components are nearer to the edge, PCB must be equipped with an **technological margins** (usually in range 5 to 10 mm).

2.4. Surface Preparation of PCB

PCB surface must be well solderable (it must not be oxidised or with badly prepared surface). We strongly recommend to use **chemical gold** or chemical tin, at least, for surface preparation, instead of HAL surface, particularly when mounting fine SMD components. Even well prepared HAL uses to have small bumps which markedly increase the risk of wrong mounting (the component can slip) or wrong soldering (the component can be raised and thus not soldered in some pins).

Chemical tin uses to be cheaper than chemical gold, but it is advised to use it just for PCBs with single-side fitted SMD components because it can significantly worsen solderability in repeated pass through the soldering process (e.g. through reflow oven).

2.5. Fiducial Marks for Mounting Machine

A mounting machine requires **fiducial marks** on PCB, e.g. circles with diameter 1 mm (Fig. 5). No connection or print shall be at the de-masked area near the marks (at least 0.5 to 1 mm from mark perimeter). And no similar shape, like de-masked metal pieces of similar diameter, should appear near to the mark.

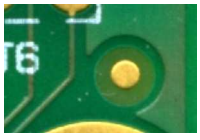

Fig. 5
Fiducial mark

A pair of marks is necessary on the PCB surface, best if diagonal and as far as possible one from another. It is advisable to have them not fully symmetric, this will prevent turn of PCB by 180°. A third mark can be in another PCB corner for the same reason. Distance of edge of any fiducial mark from the PCB edge (when panelling from panel edge) in the direction of pass through the mounting line (usually longer side of PCB or of panel) should not be less than 5 mm.

When panelling more PCB pieces, it is advisable that **each separate sub-board** should have an **own pair** (or triplet) of guiding marks.

When mounting components with terminal spacing below 1 mm or BGA, in case of larger PCBs it is advisable to place another pair of fiducial marks (so called "local marks") diagonally near the corners of each such component or group of components. Generally applies, that more guiding marks on PCB do not matter, but if they are lacking, it matters ...

3. Components

We offer you the opportunity to use common small components (particularly resistors, capacitors, transistors and diodes) **directly from our store**. You yourselves can supply other components, or we can to arrange for their delivery with our suppliers.

If you yourselves supply the components, it is advisable to adhere to the rules below, otherwise we have to charge you additional costs for demanding preparation of production:

- components must be supplied in tapes, tubes, or trays
- all components must have the same direction in one pack, polar components the same orientation
- in case of components in tubes and trays, it is advisable to supply some spare parts, within possible limits (e.g. because of price) which we will mostly return (the component can be damaged mechanically, in exceptional cases it can be damaged or lost when mounting and the like)
- tapes with components should be in a single piece (best whole reel), they should have an entry part without components, at least 5 cm and a free cover tape, at least 45 cm, they must have adequate margin in relation to the total volume to be fitted, but not less than 5 to 10 pieces (small components are sometimes lost when being fitted, e.g. because of wrong catch with spindle), they must not be unstuck or broken (e.g. rolled on too small diameter)
- components should not be too old, because oxidised pins are difficult to solder and wrongly soldered connection need not be visible.

4. Data and Documents for Fitting

4.1. Data for Mounting Machine

So that we could teach our mounting machine fit your PCB, we need the data below concerning positions of components and fiducial marks on PCB. Most current CAD systems for PCB design allow to create these data in some form. The best data are an **Excel table**, but even a common text file with fixed format is enough (the data are aligned with blanks so that the equal items are one under another) or better with a suitable spacers between items (like tabulator or semicolon).

The data file should contain a separate line with following data for each component:

- **references of components** – unambiguous labelling of each component (like R1, R2, C1, C2)
- **coordinates of component centres** – all coordinates must be given in the same coordinate system related to one point (best, but not necessarily, PCB corner), they can be either in inches or in the metric system (we prefer the latter)
- **angles of components turn** – it does not matter how precisely is the component turned in its initial position (i.e. with zero angle), this depends on the actual library of components in your design system, but it is important that the components of the same type (i.e. of the same case) should have defined it in the same manner and the angle of placing of the component on PCB related to this initial position should be given in the data file

- **types of component package** (like 0805, 1206, tantalum A, B, C, D, SOIC 8, PLCC 44, QFP 80, and the like)
- **values of components** (use the same marking for one type of components, different components must not be mismatched because of equal marking, like 1k5, 22p, 4u7/16V, BC857, GAL16V8)

The data file must also contain **coordinates of all fiducial marks** given in the same coordinate system like the components. These coordinates can be supplied also as a separate file.

When placing several PCBs on a single panel, it is advisable to present their precise spacing (or coordinates of corners of the PCBs on the panel and the like). If just global fiducial marks for the whole panel are given on the panel, supply of the precise position or spacing of the boards is necessary.

For double-sided PCBs fitted with SMD components, the data can be supplied in a separate file for each side or in a single file marking the side for each component in a dedicated column.

If there are THT components in a data file together with SMD components, it is advisable to distinguish them from SMD components, e.g. by placing in a separate block or description in a dedicated column.

4.2. Data File Example

Reference	Part label	TopCell	X	Y	Rot	Side
C31	22p	C0603	32,75	59,25	90	TOP
C32	10n	C0603	84,5	59,5	0	TOP
C33	10u/16V	CapTant A	87,75	59,25	270	TOP
C38	22p	C0603	32,75	55,75	270	TOP
C39	100n	C0603	23,75	43,75	180	TOP
C4	100n	C0603	25,75	102,25	180	TOP
D1	STPS1L30U	SMB	81,5	104,75	270	TOP
D2	LED_G	LED 0805	51,75	17,25	0	TOP
D3	LED_R	LED 0805	51,75	14,75	0	TOP
Q1	IRF7220	SO8	74,25	100,5	270	TOP
Q2	BC846	SOT23	71,25	94,5	0	TOP
R1	1R-1%	R1206	75,75	96	0	TOP
R10	1k	R0603	29	73,25	90	BOTTOM
R11	10k	R0603	14,75	68,25	90	BOTTOM
R18	1k	R0603	46	68,25	270	TOP
R19	680R-1%	R0603	49,25	69,5	0	TOP
R2	1R-1%	R1206	75,75	93,25	0	TOP
R20	680R-1%	R0603	52,25	68,5	90	TOP
R21	100R	R0603	75,5	67	180	TOP
R59	33R	R0603	69,75	37,75	90	BOTTOM
U1	LM1117MP-3.3	SOT223	57,75	99,75	90	TOP
U10	DS26LV32ATM	SO16	95	56,5	270	TOP
U11	EPM3032ATI44	TQFP44	25,5	34,75	0	TOP
U16	SN74LVC157AD	SO16	65,5	29,5	270	TOP
U6	AD8601ART	SOT23-5	76,25	86	90	TOP
U7	LM60CIM3	SOT23	48	73,25	0	TOP
U8	ATmega8L-8AU	TQFP32	40,25	59,25	270	TOP
Y1	7.3728 MHz	HC49/S-SMD	27,3	59,25	90	TOP
FDM1	Mark 1,0 2,0	Mark	16,75	31	0	TOP
FDM2	Mark 1,0 2,0	Mark	51,25	45,5	0	TOP
FDM3	Mark 1,0 2,0	Mark	113	107,75	0	BOTTOM
FDM4	Mark 1,0 2,0	Mark	68,75	79	0	BOTTOM

4.3. Other Documentation for Production

For sake of quick orientation, it is advisable to supply also a **list of components** (so called BOM) with quantity of component types on the board.

The supplied data must also contain a **drawing of the mounting** with easily visible orientation of polar components and integrated circuits and best if with legible and clear references, possibly also with values. The optimum option is to supply it in a suitable digital version, like is PDF or figure.

It is also advisable to present also a **list of comments and special wishes** concerning way of assembly or mounting (like "place power THT resistors 2 mm above the board, connector K2 is on the bottom side, special care for component XY, as it is expensive/fragile/sensitive to electrostatics" and the like).

The design should use (if possible) the least number of component values (e.g. resistors in digital technology can be usually selected in a vast range, so a value which already appears somewhere on the board can be used, and the like) and minimize number of different types for any used value, which differ in package, tolerance or voltage and the like. It is also possible to supply allowed ranges of values to replace with currently available better types, particularly when we provide the component for you, as the component documentation.

4.4. Recommendation for Labelling of Components

For labelling of components, we recommend to adhere to the conventions below, this minimizes risk of confusion or wrongly fitted component:

- do not use the number only for value of resistors and capacitors, but add a letter marking the order, like 100R for 100Ω resistor
- mark values of resistors with prefixes R (ohm), k (kilo), M (mega) placed best in the point for the decimal point (100R, 2k2, 4M7)
- mark values of capacitors with prefixes p (pico), n (nano), u (micro) (22p, 100n, 4u7/6.3V), do not use outdated prefixes k and M related to the basic unit pF, there is a risk of possible confusion with resistors
- particularly with tantalum, electrolyte and ceramic capacitors of higher capacity, it is advisable to mark the minimum required voltage
- voltage 6.3V is expected with capacitors, unless given otherwise
- tolerance 5% is expected with resistors, unless given otherwise
- material of ceramic capacitor Y5V is expected, unless given otherwise
- we mostly perform automatic replacement with higher voltage, less tolerance or better material (like X5R or X7R instead of Y5V), if so
- use of different letters for reference of component shall make the board easier to understand, recommended ones are: R for resistors, C for capacitors, L for inductors, K or J for connectors, U or Q for integrated circuits, D for diodes and LED, X or Y for crystals, F for fuses and polyswitches.

5. Stencil for Soldering Paste

Before mounting SMD components, it is necessary to put soldering paste on PCB on soldered pads. The most common way is to print soldering paste through a stencil on a screen printer. You can provide the stencil produced according to our requirements and recommendation by yourselves or let us provide for its production. If you have a stencil e.g. from previous production, consult its possible use with us.

We prefer and strongly recommend **laser cut stainless stencils**. Among etched ones, we prefer material ALPAKA.

For most PCBs, the suitable thickness of stencil is 0.15 mm, for fine motives (with spacing of component pins 0.65 mm and less) 0.13 or 0.12 mm.

5.1. Uniprint type Stencil

In most production, a template sheet with attachment type "Uniprint" is suitable. It consists of two lines of holes (perforations) along shorter sides of the sheet (see Fig. 6). Place please the motive on the stencil in such a way that the longer side of PCB (or of panel) together with technological margin is precisely 90 mm from the perforation centre. In the other direction, place the PCB contour together with technological margin symmetrically on the stencil centre (green dash and dot line on Fig. 6), and the lines of perforation holes are placed symmetrically to it. Each line has odd number of holes with the centre hole precisely in the centre of the stencil.

Motives of two different boards (or two sides of a single board) can be placed on a single stencil, if the distance between the motives is 70 mm or more. This is usually possible if the shorter panel side is not wider than 150 mm. The PCB edge of the other motive will be again precisely 90 mm from the perforation centre.

The width of the stencil should be by 50 mm, at least, wider than is the length of the blade which puts the paste on the template. And the motive of holes for paste (pads) should be then by 20 mm, at least, narrower, than the length of the blade is. We use standard blades 170 mm, 250 mm and 300 mm long, and a blade 400 mm long can be used for the maximum possible width of the stencil. A stencil 300 mm wide and blades 250 mm long are used most frequently.

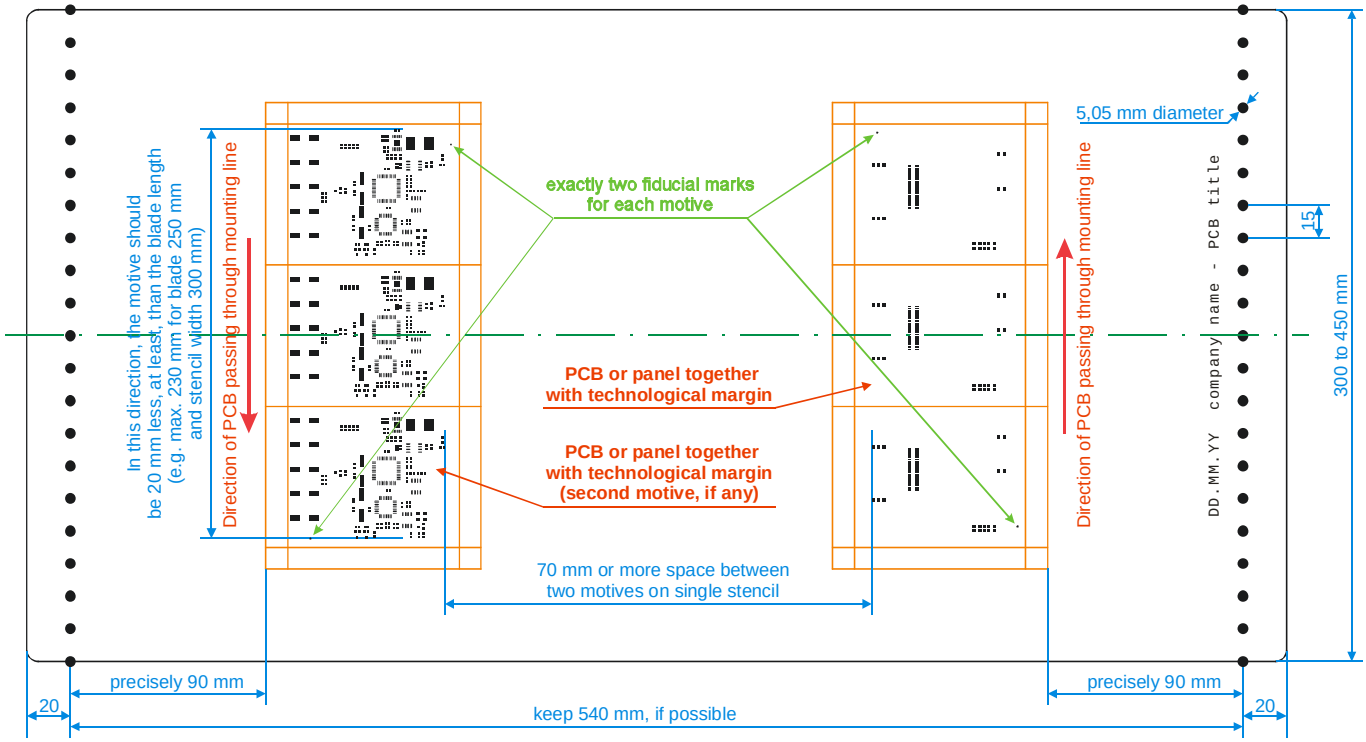

Fig. 6 Stencil - dimensions, fixing holes and positions of motives

5.2. Other Stencil Types

For large production series, frequently repeated production or because of size in case of large PCBs, we can use other stencil types. One option is a stencil with grips for fixing system VectorGuard by DEK, we have got a frame with size 23 x 29 inch, and a frame with size 29 x 29 inch exists for the largest PCBs.

Another option is a stencil hard fixed to the frame, if you need more detailed information on suitable configuration, contact us please. We have also broader blades 440 mm and 510 mm long available for these larger stencils.

5.3. Fiducial Marks for Stencil

It is necessary to have fiducial marks on the stencil that are precisely in the same spots as on PCB. Fiducial marks must be partially etched and blackened or partially burnt with laser (not through!!!) from the side that shall lay on PCB when depositing the paste ("bottom" side). **Just two guiding marks**, diagonally across the motive, as far as possible one from another, are necessary for a single PCB or one panel of a multiple PCB motive (do not place other "reserve" marks on the stencil, they rather complicate setting of automatic optical control of soldering paste putting). It is advisable also to etch or burn partially the PCB name and possibly also name of your company and date for easy identification of the stencil.

5.4. Data for Stencil Production

The most suitable way for production of a stencil are data in format **Gerber RS274-X**. The data should show pads for SMD components with dimensions equal to the size of soldering spots on PCB (this suits in most cases), and possibly a bit smaller (0.1 – 0.2 mm according to pads size). We recommend not to reduce the pads that are smaller in one dimension than 0.8 mm. The data must also **contain fiducial marks** on the same spots as PCBs are and also the **PCB or panel contour** must be supplied (together with used technological margin), for precise placement of the motive on the template. If the fiducial marks are generated in the same layer as pads for components, it is advisable to supply also a text description like "Fiducial marks are circles coded D14 with diameter 1 mm" and the like, together with the data. In case of panelised PCBs it is necessary to supply data for the motive of the whole panel or for one PCB and precise positions or spacings and numbers of multiple motives.

If you use BGA circuits with spacing of balls 1 mm, we recommend size of holes in the stencil with diameter 0.45 mm, and holes with diameter 0.4 mm for spacing 0.8 mm. For FinePitch QFP and similar with spacing of pads 0.5 mm we recommend to select hole width in the stencil as a half of the spacing of pads. Particularly for QFN packages, and possibly for packages with centre cooling pad we recommend **adhering to recommendations of manufacturers** of these parts concerning size and placement of soldering pads on PCB and holes in stencils consistently.

Fig. 7 Our automated mounting line