

designation: LV6000000 Printed Circuit Boards (PCB)

document id.: 90.03300.404-101-10-A
state: released

Change history:

version	change description
08	update items 3.2.2, 3.2.4, 3.2.8, 3.7, 3.8, 4.1, 5
09	layout adjustment, new template
10	update items 3.1, 3.2, 3.2.2, 3.2.4, 3.8, 3.11, 4.1, 5

Contents

1.	Scope	2
2.	General.....	2
2.1.	Basis.....	2
2.2.	Specifications	2
3.	Specific properties.....	3
3.1.	UL requirements.....	3
3.2.	Material requirements	3
3.2.1.	Base material	3
3.2.2.	Conductive Patterns	4
3.2.3.	Plated Through Holes	4
3.2.4.	Solder resist	5
3.2.5.	Carbon coating	5
3.2.6.	Protective coatings	5
3.2.7.	Conductive coatings - HAL.....	5
3.2.8.	Immersion coatings	6
3.3.	Printed contacts.....	6
3.4.	Rework, repair and touch-ups	6
3.5.	Drillings.....	6
3.6.	Board edges	6
3.7.	Scoring	7
3.8.	Flatness.....	7
3.9.	Cleanliness.....	7
3.10.	Solderability.....	8
3.11.	Marking.....	8
4.	Quality assurance.....	9
4.1.	First Samples / Release requirements.....	9
4.2.	Certificate of conformity	9
4.3.	Visual inspection	9
4.4.	Electrical testing	9
4.5.	Microsection examination.....	9
4.6.	Cleanliness testing	10
4.7.	Quality assurance report	10
5.	Packaging, labeling	10
6.	Valid documents.....	10
7.	Appendix	11

1. Scope

These delivery specifications describe the features of single-, double-sided and multilayer printed circuit boards (PCB) with and without plated through holes. They shall serve as a binding agreement between the E.G.O.-group (EGO) and the supplier, and are part of the supply and purchase contract.

However, the requirements of these delivery specifications do not release the supplier from securing with his own, internally determined inspection parameters, systematic procedures and regular process supervision to deliver only qualitatively faultless products.

2. General

2.1. Basis

The quality criteria has to meet at least IPC-A-600H class 2 (exceptions are specially listed in this document; applicable standards are listed in chapter 6 "Valid documents". For all standard the most current version respectively is valid.

2.2. Specifications

Additionally to this delivery specification the following documents (specifications) are binding:


- EGO PCB drawing – governing document above all manufacturing documents
- EGO original templates for the creation of the films – generally available as files
- EGO List of forbidden substances (appendix)
- EGO packaging specification (appendix)

The documents are attached or available on request and refer to the finished PCB. All documents provided to the supplier are property of EGO and strictly confidential. The supplier is not allowed to make any change or amendment on his own. Claims concerning open issues, missing information or non-performance of the specifications should be made in writing immediately.

The compliance with the EGO documents has to be confirmed in writing. The supplier may describe the deviations of those items, which do not comply with the requirements and give reasons why these items can not be reached.

3. Specific properties

3.1. UL requirements

All PCBs defined with the  approval mark as per drawing have to comply with the following requirements:

1. ZPMV2 listing of the PCB
2. Temperature rating according to drawing
3. Flammability Classification 94 V-0
4. CTI (Comparative Tracking Index) according to drawing
5. Marking according to UL File and E.G.O. drawing. Additionally the UL File number should be marked on the PCB.
 - If column CTI is marked with "***", the CTI - Performance Level Category (Table 1) has to be marked on any individual board (Example of CTI marking on each PCB => CTI 3)
 - If column CTI is marked with the PLC level, the PLC has to reach the requirements according to the drawing and the CTI - Performance Level Category (Table 1) has to be marked also on any individual board (Example of CTI marking on each PCB => CTI 3)
 - If column CTI is marked with "-", the CTI is not UL approved and is not acceptable.

Table 1: CTI-PLC

CTI Range-tracking Index (TI in Volts)	Performance Level Category
600 and greater	0
400 and up to 600	1
250 and up to 400	2
175 and up to 250	3

The supplier is committed to guarantee the compliance by the UL file.
Any lack of clarity has to be discussed with EGO by mutual consent.

3.2. Material requirements

The required base material type, thickness of the finished PCB, copper thickness of the finished PCB (after electroplating) and the surface refining layers are defined in the drawing.

The finished and "final thickness of the PCB" is the defined in the drawing based on with the specified parameter "Thickness of Laminate".

The finished PCB material has to maintain the material testing, realized at E.G.O., according to:

- IEC 60695-10-2 Ball pressure test at 180°C
- IEC 60112 CTI (Comparative Tracking Index) according to drawing
- IEC 60695-2-12 GWT 650 if I < 0,2 A GWT 850/750 if I > 0,2 A
- IEC 60695-5-11 Needle-flame test burning time max 30 s
- Dielectric breakdown min 500 V each 0,025 mm

3.2.1. Base material

The appearance of the base material of the finished PCB has to meet the requirements according to IPC-A-600H item 2.2 and 2.3 class 2.

3.2.2. Conductive Patterns

- The appearances of the conductive patterns have to meet the requirements according to IPC-A-600H item 3.2 class 2.
- Table 2 lists the required tolerance of the copper thickness for Inner Layer -, Single Layer -, Double Sided - and Multilayer PCBs - acc. IPC 4562, class 1, 2.
The minimum finished copper thickness, the final copper thickness of the finished PCB after processing must be adhered to in accordance with the columns marked in bold (min. finished thickness after processing "Final Copper").

Table 2: Copper thickness tolerances

Double Sided and Multilayer

language use	nominal thickness base copper	min. copper (-10%)	+ min. plating (20µm)	+ min. plating (25µm)	max. process reduction	min. finished thickness after processing "Final Copper"	
			Class 1,2	Class 3		Class 1,2	Class 3
6 µm	5,10 µm	4,59 µm	24,59 µm	29,59 µm	1,50 µm	23,09 µm	28,09 µm
9 µm	8,50 µm	7,65 µm	27,65 µm	32,65 µm	1,50 µm	26,15 µm	31,15 µm
12 µm	12,00 µm	10,80 µm	30,80 µm	35,80 µm	1,50 µm	29,30 µm	34,30 µm
18 µm	17,10 µm	15,39 µm	35,39 µm	40,39 µm	2,00 µm	33,39 µm	38,39 µm
35 µm	34,30 µm	30,87 µm	50,87 µm	55,87 µm	3,00 µm	47,87 µm	52,87 µm
70 µm	68,60 µm	61,74 µm	81,74 µm	86,74 µm	3,00 µm	78,74 µm	83,74 µm
105 µm	102,90 µm	92,61 µm	112,61 µm	117,61 µm	4,00 µm	108,61 µm	113,61 µm
140 µm	137,20 µm	123,48 µm	143,48 µm	148,48 µm	4,00 µm	139,48 µm	144,48 µm

Single Layer / Inner Layer

language use	nominal thickness acc. IPC4562 (non processed material)	min. copper (-10%)	max. process reduction	min. finished thickness after processing "Final Copper"
6 µm	5,10 µm	4,59 µm	1,50 µm	3,09 µm
9 µm	8,50 µm	7,65 µm	1,50 µm	6,15 µm
12 µm	12,00 µm	10,80 µm	1,50 µm	9,30 µm
18 µm	17,10 µm	15,39 µm	2,00 µm	13,39 µm
35 µm	34,30 µm	30,87 µm	3,00 µm	27,87 µm
70 µm	68,60 µm	61,74 µm	3,00 µm	58,74 µm
105 µm	102,90 µm	92,61 µm	4,00 µm	86,61 µm
140 µm	137,20 µm	123,48 µm	4,00 µm	119,48 µm

3.2.3. Plated Through Holes

- Plated through holes have to meet the requirements according to IPC-A-600H; item 3.3 class 2.
- Table 3 lists the required tolerance of the metallization.

Table 3: Thickness and tolerance of metallization (plated through holes)

Description	Nominal size [µm]	Tolerance [µm]
metallization – inside plated through holes	25	+15
metallization – neck	≥ 15	

3.2.4. Solder resist

- The solder resist has to meet the requirements acc. to IPC-A-600H; item 2.9 class 2 and IPC SM-840.
- The solder resist type - matt or glossy - must meet the E.G.O. drawing.
If not specified in the E.G.O. drawing, only a matt solder resist is required.
- Bare copper is not allowed. All copper surfaces must be covered with solder resist.
- The PCB path necks have to be covered with a layer thickness of at least 4 μm .
- The PCB traces have to be covered with a layer thickness of minimum 10 μm .
- Solder resist covering the pads or the testing point is not acceptable.
- The PCB manufacturer has to be capable to produce solder resist lands with the minimum width of 150 μm or less.
- With few exceptions, the E.G.O. solder resist gerber data is equal with the associated copper (pad) or the associated (non plated) hole. This means, that area must be free from solder resist.
- The PCB manufacturer is required to open (widen) our Gerber solder resist data as much as it is needed to equalize his offset between the solder mask and the copper.
This circulating opening is allowed up to 0.1mm.
Exceptions for the previously described solder resist opening:
Between copper structures up to a minimum clearance of 0.2mm, there has to be a solder mask dam, if Gerber data does not show different.
Copper structures up to a minimum clearance of 0.15mm can sometimes also require a solder mask dam if Gerber data does not show different. These rare cases may be handled with a TQ-Report between E.G.O. and the PCB manufacturer if needed.
- Via may be without solder resist in the Gerber data. In these cases, it is accepted, that there will be solder resist in the via hole.
- If there is a test point placed on a via, only the test point is free from solder resist but also the opposite site of the via.

3.2.5. Carbon coating

- Carbon coatings have to meet the requirements according to IPC-2221 item 4.2.4.
- It has to be secured that this layer thickness is applied by one process step (print).

3.2.6. Protective coatings

- Protective coatings have to meet the requirements according to IPC-2221 item 4.5.3.
- The PCB manufacturer has to guarantee the removability, tear resistance and the temperature stability (machine-solderability).

3.2.7. Conductive coatings - HAL

- Conductive coatings have to meet the requirements according to IPC-2221 item 4.4.
- For HAL tin-plating SN100CL should be used.
- If another plating is used, the alloy has to be released by EGO.
- Layer thickness at least 2 μm inside the holes.
- Layer thickness at least 1 μm on the pads.
- Total layer thickness 1 - 40 μm with an absolute possible flatness (aim maximum unevenness of 15 μm).
- Appearance with a metallic shine.
- Readability of the fiducial marking has to be guaranteed.

3.2.8. Immersion coatings

- Immersion coatings have to meet the requirements according to IPC-4552.
- Uniform plating and complete coverage of surface is required.
- No evidence of plating removed acc. IPC-TM-650 TM 2.4.1.
- Solder durability must meet the J-STD-003 (cat. 3) with min shelf life of 6 month ex incoming at E.G.O.
- Evidence of plating thickness and capability must be demonstrated with a certificate for each lot.
- The layer thickness must meet the measures in the below table

Immersion type	min. thickness	max. thickness	Measure method
Tin	1,0 µm	1,2 µm	Mean of a measured pad size of 1,5mm x 1,5 mm
Gold	0,05 µm	0,12 µm	
Nickel	3,0 µm	7,0 µm	

3.3. Printed contacts

- The printed contacts have to met the requirements according to IPC-A-600H; item 2.7 class 2.
- Edge contacts have to be provided with a chamfer ($0.5 \pm 0.2 \times 45^\circ \pm 5^\circ$).

3.4. Rework, repair and touch-ups

- Conductor welding is not acceptable.
- Removal of shorts between conductors is acceptable.
- Removal of residual plating, include copper, is acceptable.
- It is not mandatory to remove metal orphans in areas free of traces, pads and vias.
- Rework/touch-up of the solder resist is acceptable, with maximum five touchups per panel and maximum one touch-up per inch².
- All provided reworks/touch-ups are not allowed to affect the requirements of the PCBs, as set in this document.
- All reworks/touch-ups have to be executed by trained, skilled and experienced personnel.

3.5. Drillings

- If not stated in the PCB drawing, tolerances correspond to DIN ISO 2768-m part 1/table 1; passing through drill holes ≤ 0.6 mm are excluded.
- An annular of min. 0.05 mm is required for all plated-through.
- Lose burrs or burrs > 0.1 mm can not be accepted.
- Annular rings according to IPC-A-600-H:
 - class 3 for THT
 - class 2 for vias

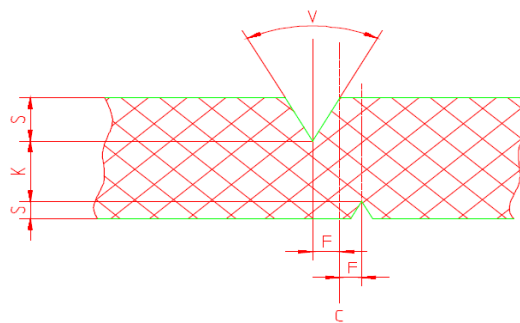
3.6. Board edges

- Imperfections such as burrs, nicks along the edge of the board are not acceptable.
- Chamfering is to be generated by scratching it will be indicated in the drawing.
- For metallic and nonmetallic burrs the edge condition has to be smooth and without any lifting (IPC-A-600H; item 2.1).

3.7. Scoring

- Web 0.8-0.2 mm for CEM and 0.45 mm for FR4
- Scoring depths min 0.1 mm
- Blade Offset 0.1 mm
- Cutter Angle 30° or 45°
- For the entire PCB length < 250 mm a max. deviation of the scoring of ±0.1 mm (for PCBs ≥ 250 mm max. ±0.2 mm) is admissible.

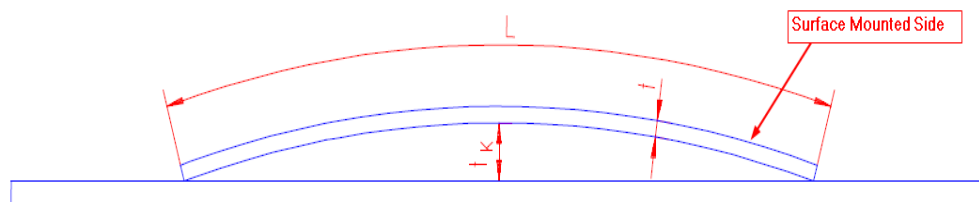
Lamination type	notch angle - V	core thickness - K (scoring depth - S min. 0.10 mm)	offset - F
CEM1 / CEM3	30 or 45 °	0.80 - 0.2 mm	±0.10 mm
FR4	30 or 45 °	0.45 ± 0.1 mm	±0.10 mm



3.8. Flatness

- Following mentioned values are valid for PCB's consisting of CEM-1, CEM-3 and FR-4 with nominal values (t) of laminates from 1,5 – 3,2 mm.
- Always valid, if no other statements have been made in the drawing are the values acc. to following table with the additional requirement that the maximum tolerable convexity and torsion tK ≤ 2 mm for PCB-copy has to be met.

	Single Sided PCB	Double Sided PCB	Multilayer PCB
K	≤ 1,0 %	≤ 0,7 %	≤ 0,75 %



Calculation formula
$$K = \frac{tK}{L} * 100\%$$

3.9. Cleanliness


- All PCBs have to be free of residues (flux-, chemical-, white- and salt residues, particulate matter, fingerprints, corrosion (oxide), oil or any other kind of foreign particles).
- The solvent resistivity shall be in accordance with IPC-6012.

3.10. Solderability



- All PCBs have to withstand a soldering process at $T = 270\text{ °C}$, $t = 10\text{ sec}$.
- During the soldering process no conductor parts may be lifted from the basic material. The surface must not show any bubbles and the basic material must not show and delamination. The solder resist must not be infiltrated and labeling must remain readable.
- The soldering has to be guaranteed for 6 months (IPC-A-600H; item 5.1 category 2).

3.11. Marking

The PCB manufacturer has to label every PCB with his manufacturer logo and the manufacturing date (preferably as WWYY). Therefore a "FA" – Free Area is defined in the solder mask. This marking ("FA" and frame) has to be replaced by the manufacturer with his own labeling. It must be good readable and the minimum distances to conductor parts have to be kept.

If the PCB is marked with the  or lead-free label according to drawing, the marking on the PCB has to be UL conform (see item 3.1) and lead-free respectively. The single PCBs of the panel have to be marked (N1, N2,...). The PCB drawing indicates the color of the component notation. Other labeling, such as stamps are not allowed (IPC-A-600H; item 2.8)

Summary of markings, which must be included necessarily on each single PCB:

- PCB supplier and manufacturer data (name / mark / symbol)
- PCB manufacturing date (WWYY)
-  symbol
- UL mark (94V-0).
- UL file number (Exxxxxx)
- CTI index (see item 3.1)
- Lead free symbol ()
- Single PCB mark according to specifications (N1, N2, ...)

4. Quality assurance

4.1. First Samples / Release requirements.

Before starting the series delivery the supplier has to carry out first samples.

The sample quantity is defined in the corresponding purchasing order.

Additionally a first sample inspection report is required. The template for the report is available on request.

First samples generally have to be manufactured with serial operating facilities and tools in a serial process.

EGO will issue the delivery release after a positive review of the first-sample inspection report (if required, under conditions). Without the delivery release the serial deliveries may not be started or re-started.

Generally, only those materials (laminates, varnishes, pastes, masks, coatings) may be used that are stated in the technical documents of EGO with a proven release from VDE and UL.

For the serial delivery the released materials have to be used.

The following listed documentation and evidences must be provided for each initial sampling:

- E.G.O. First Samples Test Report + Measurement Sheet + EGO drawings with numbered dimensions (E.G.O. First Samples Test Report Template is available on request).
- Datasheet - base material
- Datasheet - solder resist
- Datasheet - silk screen / component print
- Datasheet – HASL, chemical Sn, gold or other surface protections
- Confirmation and approval documents of UL / VDE for base material, solder resist, silk screen / component print and others (UL-file no., UL-Database - CTI release and other evidence).
- Certificate of conformity (raw materials and the RoHS compliance)
- Documented evidence of the performed layer thickness measurements by means of a micro section report (base material, copper layer thickness of the inner and / or outer layer, HASL / chemical Sn / gold or other surface protections, etc.)
- By the supplier created and by E.G.O. answered documentation of the "Technical Questions (TQ)" or "Engineering Questions (EQ)"
- Min. 5 pieces of test coupon 20 x 40 mm with free copper surface for CTI testing.

E.G.O. will start the sampling and verification only on base of a complete documentation.

Otherwise the sample will be rejected.

4.2. Certificate of conformity

The usage of the released raw materials, the RoHS compliance and the UL approval has to be confirmed with the delivery of each batch.

4.3. Visual inspection

100% visual inspection of the PCBs for the applicable dimensional and workmanship characteristics, as described herein and in the drawings, with the aid of a magnifying lens of at least 1.75x, higher magnification may be used as required to verify defects.

4.4. Electrical testing

100% electrical testing for open and short shall be performed on all PCBs delivered as "good".

4.5. Microsection examination

- Microsection examination shall be performed per IPC-TM-650 method 2.1.1 or 2.1.2.
- The test coupon has to include at least one plated through hole and one conductor of each layer.

4.6. Cleanliness testing

- The PCBs shall be tested for ionic contamination in accordance with IPC test method 2.3.25.
- The maximum admissible residuals are 1.5 µg/cm². (IPC-A-600H; item 5.0).

4.7. Quality assurance report

All results of item 4.2 to 4.5 should be summarized in a quality assurance report, which has to be provided in written form with each batch. The cleanliness report (item 4.6) has to be provided in written form with the first sample documents.

5. Packaging, labeling

- Generally EGO specification LV 6104000 is valid for packaging and labeling.
- Only complete PCBs copies without any bad PCB have to be delivered.
- Deliveries of x-out PCBs are generally not allowed. In case of a board qty. >10 panels, exceptions can be discussed with the QM- or purchasing department .
- Therefore the x-out PCBs (maximum 3 each panel) have to be marked with a black label at a declared position of the PCB. A detailed procedure of bad-board marking is available on request.
- It is not allowed to mix good PCBs and x-out PCBs in the same packaging and the packaging of the x-out PCBs has to be marked obviously.
- The single package units must be packed in a stable, vacuumed, full sealed ESD conform foil.
- Each single package must contain a moisture absorber, a moisture indicator is preferred.
- Underlines for surface protecting or stabilization are only allowed on request.
- All PCBs have to be packed in the right position.
- The single trading bin has to be large enough that it may be closed after withdrawing single parts.

6. Valid documents

Indent No.	Description
IEC 60695-2-10:2000	Fire hazard testing - Part 2-10: Glowing/hot-wire based test methods; Glow-wire apparatus and common test procedure
EN 60695-2-12:2009	Fire hazard testing - Part 2-12: Glowing/hot-wire based test methods – Glow-wire flammability index (GWFI) test method for materials
EN 60695-10-2:2003	Fire hazard testing - Part 10-2: Abnormal heat - Ball pressure test
IEC 60112:2003+ A1:2009	Method for the determination of the proof and the comparative tracking indices of solid insulating materials
ISO 2768-1:1989	General tolerances; tolerances for linear and angular dimensions without individual tolerance indications
IPC-A-600H	Acceptability of Printed Boards
IPC-2221	Generic Standard on Printed Board Design
IPC-SM-840D	Qualification and Performance of Permanent Solder Mask
IPC-TM-650 series	Test methods manual
IPC-4552	Specification for Electroless Nickel / Immersion Gold
IPC-4562	required tolerance of the copper thickness for outer- and inner layer
IPC-6012	Solvent resistivity
J-STD-003	Solderability durability

7. Appendix

Appendix	Declaration of consent
Appendix	Deviations to the delivery specification
Appendix	EGO List of forbidden substances (document apart)
Appendix	EGO Packaging specification Doc. No.: 90.35101.036 (document apart)

delivery specification

company: 0083 / E.G.O. Germany (E.G.O. Elektro-Gerätebau GmbH)
function: 16 / quality



Declaration of consent

Supplier/company:

Postcode: _____ Place: _____ Street: _____
 Telephone: _____ Fax: _____
 Contact person / department: _____ Telephone extension: _____

The supplier mentioned before has reviewed, taken notice of it and commits himself to meet the requirements of the:

E.G.O. delivery specification LV 6000000, version 10

Any deviations to this delivery specification have to be defined in the addendum below and approved by E.G.O.

In case of question, please contact the below-listed contact persons or the responsables of the plant ordering from the delivery schedule:

	Contact person	Telephone Telefax	Email
commercial	Mr. Rainer. Hoffmann Corporate Purchasing	(+49) 07045 45-67905	rainer.hoffmann@blanc-fischer.com
technical	Mr. Sven Gollasch QM purchased parts	(+49) 07045 45-67751	sven.gollasch@egoproducts.com

Date: _____ Authorized signature(s): _____

Deviations to the delivery specification

item	deviation

Date: _____ Authorized signature(s) E.G.O. : _____