

designation: LV6000000 Printed Circuit Boards (PCB)

document id.: 90.03300.404-101-11-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
11	Update items: 2.2., 3.1., 3.2., 3.2.1., 3.2.2., 3.2.3., 3.2.4., 3.2.4.1., 3.2.4.2., 3.2.4.3., 3.2.5., 3.2.7., 3.7., 3.8., 3.10., 4.6., 6. New items: 3.2.9., 3.11.

Contents

1.	Scope	2
2.	General.....	2
2.1.	Basis.....	2
2.2.	Specifications	2
3.	Specific properties.....	2
3.1.	UL requirements.....	2
3.2.	Material requirements	3
3.2.1.	Base material	3
3.2.2.	Solder resist	3
3.2.3.	Conductive Patterns	4
3.2.4.	Vias, plated holes, test points and non plated holes.....	5
3.2.4.1.	Vias, plated holes / plated through holes	5
3.2.4.2.	Test points.....	5
3.2.4.3.	Non plated holes (drillings).....	5
3.2.5.	Electrically conductive carbon coating	5
3.2.6.	Protective coatings	5
3.2.7.	Conductive plating (HAL)	5
3.2.8.	Immersion plating.....	6
3.2.9.	Silkscreen markings / areas	6
3.3.	Printed contacts.....	6
3.4.	Rework, repair and touch-ups	6
3.5.	Board edges	6
3.6.	Scoring / V-Cut.....	7
3.7.	Flatness.....	7
3.8.	Cleanliness.....	7
3.9.	Solderability.....	7
3.10.	Marking.....	8
3.11.	Flex PCBs	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	9
4.7.	Quality assurance report	10
5.	Packaging, labeling	10
6.	Valid standards, requirements and documents.....	11
7.	Declaration of consent	12

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 (E.G.O.) as well as 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-600 class 2. Exceptions are specially listed in this document. Applicable standards are listed in chapter 6 " Valid standards, requirements and documents ". For all standards the most current version respectively is valid.

2.2. Specifications

The following E.G.O. documents are binding:

- E.G.O. Gerber Data (Governing and leading specifications)
- E.G.O. PCB drawings (In addition to gerber data)
- E.G.O. PCB delivery specification Doc. no: 90.03300.404 (In addition to gerber data and drawings)
- E.G.O. packaging specification Doc. no: 90.35101.036 (In addition to PCB delivery specification)
- E.G.O. Restricted Substances List Doc. no: 90.60035.942

The data (files) and documents are attached or available on request and refer to the finished PCB. All documents provided to the supplier are property of E.G.O. 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 E.G.O. 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 cannot be reached.

3. Specific properties

3.1. UL requirements

All PCBs must always comply with the following requirements and must be guaranteed by the supplier:

- UL approval, confirmation and certification according to ZPMV2 or ZPXK2
- Flammability classification 94 V-0
- UL marking according to ITEM 3.10.
- CTI (Comparative Tracking Index) confirmation acc. UL database, table1 and IEC 60112.

CTI - Comparative Tracking Index (TI in Volts) (E.G.O. PCB Specification -see E.G.O. PCB drawing)	PLC - Performance Level Category
600 and greater	0
400 and up to 599	1
250 and up to 399	2
175 and up to 249	3

Table 1: CTI (PLC) - PLC

PCBs which do not comply with the requirements mentioned above are not acceptable and will be rejected.

3.2. Material requirements

For series delivery only released material combinations (solder resist + base material) are acceptable.

The “finished and final thickness of the PCB” is defined in the drawing with the specified parameter “Thickness of Laminate” (Final thickness of PCB).

The finished PCB must meet the following material requirements:

- IEC / EN 60112: CTI (Comparative Tracking Index)
- IEC / EN 60695-10-2 Ball pressure test
- IEC / EN 60695-2 GWT (Glow wire test) - 650 °C if I < 0,2 A / 750/850 °C if I > 0,2 A
- IEC / EN 60695-11-5 Needle flame test - Burning time max. 30 s
- Dielectric breakdown min. 500 V each 0,025 mm
- Lead free
- RoHS compliance

The verification of the listed requirements is also a part of the qualification / release process by E.G.O.

3.2.1. Base material

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

3.2.2. Solder resist

- The solder resist has to meet the requirements acc. to IPC A-600 item 2.9 class 2 and IPC SM-840.
- If not specified differently in the E.G.O. drawing, only a “matt” solder resist is acceptable.
- Bare copper is not allowed. All copper surfaces / tracks 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 150µm 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 max. 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.

3.2.3. Conductive Patterns

- The appearances of the conductive patterns have to meet the requirements acc. to IPC A-600 item 3.2 class 2.
- Table 2 lists the required copper thicknesses with the allowed tolerances for Double Sided and Multilayer (outer layer) as well as for Single Sided and Multilayer (inner layer) PCBs acc. to IPC 6012 - class 1, 2.
- The final copper thickness of the finished PCB after processing must be adhered to in accordance with the columns marked in bold (see Table 2 - "Final Copper" - class 1, 2).
- The E.G.O. drawing specification "Thickness of copper: 35µm" regarding FR4-PCBs has the following meaning: The final thickness of copper after processing has to be minimum 33,40µm on outer layer and in case of a Multilayer PCB minimum 24,90µm on inner layer.

Double Side and Multilayer - outer layer

(µm)	nominal thickness acc. IPC 4562 (raw material / base copper) (µm)	min. copper (-10%) (µm)	+ min. plating (20µm) (µm)		max. process reduction (µm)	min. finished thickness after processing "Final Copper" (µm)	
			Class 1,2	Class 3		Class 1, 2	Class 3
5	5,10	4,60	24,60	29,60	1,50	23,10	28,10
9	8,50	7,70	27,70	32,70	1,50	26,20	31,20
12	12,00	10,80	30,80	35,80	1,50	29,30	34,30
18	17,10	15,40	35,40	40,40	2,00	33,40	38,40
35	34,30	30,90	50,90	55,90	3,00	47,90	52,90
70	68,60	61,70	81,70	86,70	3,00	78,70	83,70
105	102,90	92,60	112,60	117,60	4,00	108,60	113,60
140	137,20	123,50	143,50	148,50	4,00	139,50	144,50

Single Side and Multilayer - inner layer

(µm)	nominal thickness acc. IPC 4562 (raw material / base copper) (µm)	min. copper (-10%) (µm)	max. process reduction (µm)	min. finished thickness after processing "Final Copper" (µm)
5	5,10	4,60	1,50	3,10
9	8,50	7,70	1,50	6,20
12	12,00	10,80	1,50	9,30
18	17,10	15,40	4,00	11,40
35	34,30	30,90	6,00	24,90
70	68,60	61,70	6,00	55,70
105	102,90	92,60	6,00	86,60
140	137,20	123,50	6,00	117,50

Table 2: Copper thicknesses

3.2.4. Vias, plated holes, test points and non plated holes

3.2.4.1. Vias, plated holes / plated through holes

- Vias and other plated holes have to meet the requirements acc. to IPC-A-600 and IPC-2221.
- Vias and other plated holes must always comply with the E.G.O. gerber data as well as the drawings which define the tolerances etc.
The E.G.O. gerber data shows, which vias are covered or spared from solder resist (negative) as well as on which via a test point is placed.
- If vias in the E.G.O. gerber data are covered with solder resist (negative), there is no solder resist allowed on them without any exception!
- If vias in the E.G.O. gerber data are spared from solder resist (negative), solder resist is allowed to flow into the holes. Furthermore, our intention here is, to close the holes as much as possible.
- Exception: There are cases where test points are placed on vias without solder resist (negative) in the E.G.O. gerber data. This would result in vias, which are closed on one side and open on the other side. To avoid this situation, the PCB manufacturer is asked to open the solder resist on these vias on both sides.
- Item 3.2.2 has to be observed in respect of vias and other plated holes as well!
- Annular rings of min. 0.05 mm according to IPC-A-600 (class 3 for THT / class 2 for vias) and IPC-2221 are required for all plated holes.
- Table 3 lists the required tolerance of the metallization.

Description	Nominal size [µm]	Tolerance [µm]
metallization – inside plated through holes	20	+15
metallization – neck	≥ 15	

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

3.2.4.2. Test points

- Test points can be placed on vias, tracks or copper planes.

3.2.4.3. Non plated holes (drillings)

- If not stated in the PCB drawing, tolerances correspond to DIN ISO 2768m.
- Burrs > 0.1 mm cannot be accepted.

3.2.5. Electrically conductive carbon coating

- The electrically conductive carbon coating have to meet the requirements according to IPC 2221.
- The carbon layer has to be applied (printed) in one process step.

3.2.6. Protective coatings

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

3.2.7. Conductive plating (HAL)

- The conductive materials have to meet the requirements according to IPC 2221.
- For HAL tin-plating SN100CL should be used.
If a different plating should be used, this must be communicated to E.G.O. and released by E.G.O.
- 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 max. unevenness of 15µm).
- Appearance with a metallic shine.
- Readability of the fiducial marking has to be guaranteed.

3.2.8. Immersion coatings

- Immersion plating has 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 IPC J-STD-003 with min. shelf life of 6 month 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	
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	

3.2.9. Silkscreen markings / areas

- The silkscreen markings / areas must always comply with the E.G.O. gerber data and the E.G.O. PCB drawings.
- It is essential to ensure that the silkscreen markings / silkscreen areas are printed completely over the entire surface shown in data. Visible scratches, discoloration, defects, flaws and flaking on the edges are not acceptable and will be rejected.
- Silkscreen areas that extend flush to the edge in the Gerber data must also be designed in this way on the circuit board and not have any visible flaking. Deviations are unacceptable and will be rejected.
- The silkscreen colour must comply with the specifications of the E.G.O. drawings.
- If data shows silkscreen covering solder resist, the PCB manufacturer is asked to cut the silkscreen from the opened solder resist (acc. item 3.2.2 solder resist) with an offset of max. -0,15mm.

3.3. Printed contacts

- The printed contacts have to meet the requirements according to IPC A-600 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

- Reworked and repaired PCBs have to meet the requirements acc. to IPC A-600 and IPC-7711/21.
- 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 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.

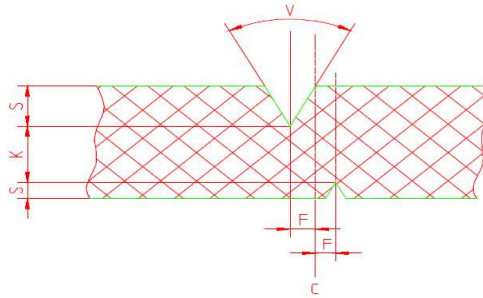
3.5. Board edges

- 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-600 item 2.1)

3.6. Scoring / V-Cut

- 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.

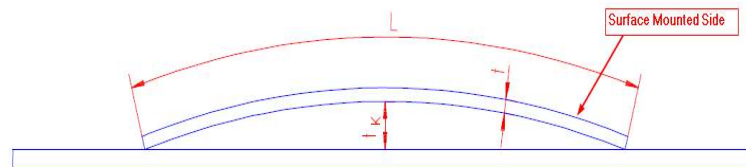
Lamination type	notch angle - V	core thickness - K (scoring depth - S min. 0.10 mm)	offset - F
CEM1 / CEM3	30 or 45 °	0.80 - 0.2 mm	± 0.10 mm
FR4	30 or 45 °	0.45 ± 0.1 mm	± 0.10 mm



3.7. 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.
- If no other statements have been made in the E.G.O. drawing, the values according to the following table are always valid with the additional requirement that the max. tolerable convexity and torsion $tK \leq 2$ mm for PCB has to be met.

	Single Sided PCB	Double Sided PCB	Multilayer PCB
K	$\leq 1,0 \%$	$\leq 0,7 \%$	$\leq 0,75 \%$



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

3.8. Cleanliness



- All PCBs have to be free of residues (flux-, chemical- 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.
- The admissible residuals / ionic contamination is max. $1.5 \mu\text{g NaCl} / \text{cm}^2 = 9,68 \mu\text{g NaCl} / \text{square inch}$.

3.9. Solderability

- All PCBs have to withstand a soldering process at $T = 270 \text{ }^\circ\text{C}$, $t = 10$ 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 any delamination.
The solder resist must not be infiltrated and labeling must remain readable.
- The soldering has to be guaranteed for 6 months acc. to IPC A-600 item 5.1 category 2.

3.10. Marking

The PCB supplier / manufacturer necessarily must be marked each single PCB with the following data:

- PCB supplier and manufacturer data (name / mark / symbol / logo)
- PCB manufacturing date (WWYY)
-  symbol
- UL mark (94V-0).
- UL file number (Exxxxxx)
- CTI 0, 1, 2 or 3 (E.G.O. PCB specification - see E.G.O. PCB drawing and ITEM 3.1 – table1)
- Lead free symbol ()
- Single PCB of the panel have to be marked according to specifications (N1, N2, ..., N...)

The data listed above should be placed in the defined "Free Area" - Resist B.

Please note that the frame and text are only placeholders and must be replaced by the above data.

If the given free area (gerber data) is not sufficient, the supplier/manufacturer must make a proposal to E.G.O. (for example in form of "Technical Questions (TQ)" or "Engineering Questions (EQ)").

The supplier / manufacturer must also ensure that the data is clearly identifiable / readable and the minimum distances to the conductor parts have be observed.

If the free area is located in a MESH Area, this space has to be filled up with full-surface copper.

Further markings (for example: E.G.O. Symbol, PCB type number, etc.) must be applied to each individual PCB according to E.G.O. gerber data.

Other labeling, such as stamps are not allowed (IPC-A-600; item 2.8)

3.11. Flex PCBs

General requirements

Flex PCB's has to meet the requirements acc. to IPC-2223, Design Standard for Flexible/Rigid-Flexible Printed Board.

The main requirements like the

- Layer stack up dimensions, released materials / suppliers (cover layer / PI / adhesive etc.).
- Insulation strength / test voltage / duration time
- Thermal requirements

are specified in the E.G.O. master drawings.

Specific test requirements in terms of function, reliability and storage conditions must be defined in a separate test specification upon request unless not specified in the drawing.

Metallic coatings like ENIG must meet the requirements specified in chapter 3.2.8 (immersion coatings).

Connector requirements

- The typically connection method for E.G.O. applications is a single row contact finger, ENIG coated.
- The contact finger must withstand min. 3 mating cycles.
- Any crack formation caused by production processes ((like handling, shape punching) or material properties are not allowed.
- The connector area must be free of any contamination or corrosion along the entire supply chain.
- The contact finger areas must be continuously inspected (visually / cross-sections) acc. the IPC 4552.

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, solder resist, silkscreen, HASL, chemical Sn, gold or other surface protections
- Confirmation and approval documents of UL / VDE for base material, solder resist, silkscreen and others (UL-file no., UL-Database - CTI release and other evidence).
- Certificate of conformity (raw materials and the Lead free / 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.)
- If available the 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 (min. 20 x 40 mm) with free copper surface for CTI testing. These are only necessary if a new material combination (solder resist, base material) which has not yet been tested and approved by E.G.O. is used.

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.

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 acc. to IPC A-600 – Item 5 and IPC TM-650 test method 2.3.25 / 2.3.26.
The admissible residuals / ionic contamination is max. $1.5 \mu\text{g NaCl} / \text{cm}^2 = 9,68 \mu\text{g NaCl} / \text{square inch}$.
- Printed circuit boards which do not meet these requirements are not acceptable for E.G.O. and will be rejected upon delivery.

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 the E.G.O. packaging delivery specification LV 6104000 - doc. no. 90.35101.036 - is valid for packaging and labeling.
- Only complete PCBs 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 standards, requirements and documents

Indent No.	Description
IPC A-600	Acceptability of Printed Boards
EN 60112 IEC 60112	Method for the determination of the proof and the comparative tracking indices of solid insulating materials - CTI (Comparative Tracking Index)
EN 60695-10-2 IEC 60695-10-2	Fire hazard testing - Part 10-2: Abnormal heat - Ball pressure test
EN 60695-2 IEC 60695-2	Fire hazard testing - Part 2-10: Glowing/hot-wire based test methods; Glow-wire apparatus and common test procedure Fire hazard testing Part 2-11: Glowing/hot-wire test methods - Glow-wire flammability test method for end-products (GWEPT) Fire hazard testing - Part 2-12: Glowing/hot-wire based test methods – Glow-wire flammability index (GWFI) test method for materials Fire hazard testing Part 2-13: Glowing/hot-wire based test methods - Glow-wire ignition temperature (GWIT) test method for materials
EN 60695-11-5 IEC 60695-11-5	Fire hazard testing - Part 11-5: Test flames - Needle flame test methods - Test set-up, confirmatory test arrangements and guidance
IPC-4562	Required copper thicknesses with the allowed tolerances for Double Sided and Multilayer (outer layer) as well as for Single Sided and Multilayer (inner layer) PCBs
IPC SM-840	Qualification and Performance Specification of Permanent Solder Mask and Flexible Cover Materials
IPC 2221	Basic guidelines for the design of printed circuit boards
DIN ISO 2768	General tolerances; tolerances for linear and angular dimensions without individual tolerance indications
IPC-4552	Performance Specification for Electroless Nickel/Immersion Gold (ENIG) Plating for Printed Boards
IPC-TM-650	Test methods manual
IPC J-STD-003	Solderability Tests for Printed Boards; with Amendments 1 & 2
IPC-6012	Qualification and Performance Specification for Rigid Printed Boards
IPC 7711/21	Reworked, repaired and Modification of Printed Boards + Electronic Assemblies
IPC-2223	Sectional design standard for flexible/rigid-flexible Printed Board
90.35101.036	E.G.O. packaging delivery specification - LV 6104000
90.60035.942	E.G.O. Restricted Substances List

For all standards the most current version respectively is valid.

7. 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. PCB delivery specification LV 6000000, version 11

Any deviations to this delivery specification have to be defined in the table 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
quality	Mr. Sven Gollasch QM purchased parts	(+49) 07045 45-67751	sven.gollasch@egoproducts.com
technical	Mr. Marc Blum Layout	(+49) 07045 45-67259	marc.blum@egoproducts.com

Date: _____ Authorized signature(s) - Supplier: _____

Deviations to the delivery specification - Supplier:

item	deviation

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