

# pt\_X\_ic BV

## Technical Terms of Delivery

Revision 01 – 01.07.2004

### 1 Working days

1.1 The day, on which the order and the complete production data are made available not later than 6 p.m., does not count as a working day. If the order and/or the production data are made available after 6 p.m., the first working day is the day after next.

1.2 Technical preconditions: Productions data and orders which are faulty, incomplete, or vary from the details on which the offer is based lead to delays in delivery and/or incorrect execution of the order, etc.

### 2 Liability Disclaimer

2.1 We do not accept responsibility for any errors incurred when referring to offers or when enquiring.

### 3 Correct Side

3.1 In order to be able to recognise the correct side (mirrored or unmirrored image) legible type must be visible in an external position on the conductive pattern.

3.2 The order of layer construction must be given for multilayers. We are not liable for correct execution if we are not provided with this information.

### 4 Confusion

4.1 We cannot accept liability for any incorrect execution of PCBs if a file description is not provided.

### 5 Conversion

5.1 We are not liable for errors occurring as a result of converting Eagle-BRD files into gerber data.

### 6 Annular Ring

6.1 To guarantee an adequate annular ring encircling plated-through drills, the pad should have a surround of at least 300µ around the hole.

### 7 Pad

7.1 If non-plated holes are positioned on one pad, the pad must have a surround of at least 500µ larger than the hole. Otherwise some/all of the pads may be removed.

### 8 Solder Resists

8.1 Please refrain from oversizing when drafting solder resists, i.e. the solder stop spaces are 1:1 to the pads. This facilitates the incorporation and required stretching of the solder resists during the manufacturing process.

### 9 Silkscreen

9.1 In order to perfectly reproduce silkscreens, the type must be at least 1mm high and the stroke at least 200µ thick. Also, all surrounding solder areas must be clear of component print for at least 250µ or imperfect print and printed solder areas may result.

### 10 Non-plated Holes

10.1 This refers to plated through PCBs: If we do not have any information from you as to which holes are to be plated-through and which are not, we will determine this to the best of our knowledge.

## **11 Data Provided (Data Inconsistency)**

11.1 If drilling and measurement plans are included that do not match the drilling program or the contour according to the gerber-data, then the drilling program and the gerber-data will be binding in all cases.

## **12 PCB Contour**

12.1 If not otherwise stipulated, the PCB contour will be determined by the centre (=centre vector) of the contour lines provided in the order data. If Slots are represented by you by rectangular outlines, then we assume generally that the corner radius is contained.

## **13 Technical Specifications**

13.1 For further technical specifications of our manufacturing processes, please refer to our „Process & Capability Manual“ in the "Standard" category. These standard technical criteria also apply even if your manufacturing data and documents call for other features (special technical requirements are available in our Premium Jet Line).

## **14 UL Certification**

14.1 If the PCB is to be produced in line with our UL Certification, please follow the layout instructions.