Design Guidelines

For Manufacturing of Circuit Boards and Modules and for Layout Design

exceet electronics GesmbH

Wildbichler Straße 2e / A - 6341 Ebbs

Tel: +43 5373 43143-0

Fax: +43 5373 43143-888

info@exceet.at

www.exceet-electronics.at

Doku-Nr.: PE3086, Version: 10

Table of Contents

Table of Contents **2**

1 Intro **3**

1.1 Overview 3
Circuit Board 3
Surface Mount Technology 3
Through Hole Technology 3

2 PCB / Layout **4**

2.1 Min/Max PCB and Panel Dimensions 4
2.2 Clearance for transport bands 5
2.3 Base Material 5
2.4 Pad Surface 5
2.5 Layer Thickness of Circuit Board Surface 5
2.6 Solder Resist Mask 6
2.7 Silkscreen 6
2.8 Clear Space for Barcode Labels 6
2.9 Fiducials 6
2.9.1 Circuit Board Fiducial 6
2.9.2 X-out Fiducial 6
2.9.3 Recommended Fiducial 6
2.10 Panel design 7
2.11 Dataset ..... **Fehler! Textmarke nicht definiert.**

3 SMT **9**

3.1 Component packaging 9
3.2 Pad Geometry 9
3.3 Note on BGA's 9
3.4 Assembling Data (Pick & Place data) 10
3.4.1 Example 10
3.5 SMT Soldering 10
3.5.1 Double-sided 10
3.5.2 Soldering Profile Vapor Phase 11
3.5.3 Soldering Profile Reflow 11

4 THT **12**

4.1 THT Soldering 12
4.1.1 Selective Wave Soldering 12
4.1.2 Soldering Profile Wave 13

5 Design for Testability **14**

5.1 Constructive Designrules 14
5.1.1 Contacting 14
5.1.2 Center Holes 14
5.1.3 Test Points 14
5.2 Electrical Designrules 14
5.2.1 Functional tests with high current loads 14
5.2.2 Arrays / Serial connections of resistors 14
5.2.3 4-wire measuring 14

6 Design for Coatability **15**

7 References **15**

8 Contact Person **15**

9 Disclaimer **16**

1 Intro

This document contains the design guidelines respectively the requirements for manufacturing high-quality printed circuit boards at exceet electronics. If there are for some reasons deviations, the detailed technical conditions have to be discussed.

1.1 Overview

Circuit Board

This section describes all preconditions relevant for production-oriented circuit boards like dimensions, properties, fiducials, surface, panel design etc.

Surface Mount Technology

In this section some design rules for designing SMT layouts (pad geometries, component gaps etc.) have been summarized. Another topic is the optimal data format and the production-oriented component packing.

Through Hole Technology

This section describes the proper positioning of THT components in order to achieve an optimal wave soldering result. Further some recommendations regarding component gaps for mixed assemblies are given.

2 PCB / Layout

exceet electronics expects acceptance criteria for circuit boards according to IPC-A-600.

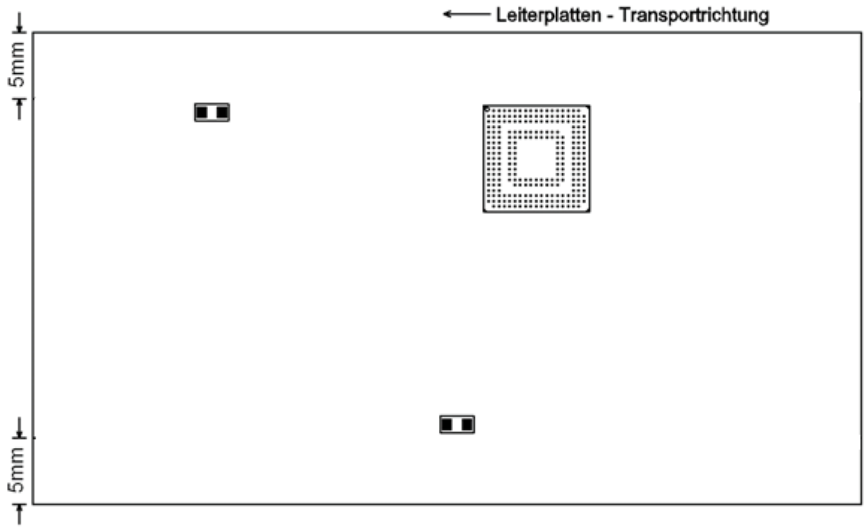
2.1 Min/Max PCB and Panel Dimensions

SMT Production Line		**PCB width**	**PCB length**	**Thickness**	**Component size**
Production Line 1/2	Min.	100 mm	100 mm	0.5mm	0201 components
	Max.	410 mm	460 mm	4.0mm	
Assembling automation		**PCB width**	**PCB length**	**Thickness**	**Component size**
Siemens 1 (D1 and D2)	Min.	50 mm	50 mm	0.3 mm	0201 components
	Max.	460 mm	460 mm	4.5 mm	
Siemens 2 (D1 and D2)	Min.	50 mm	50 mm	0.3 mm	0201 components
	Max.	460 mm	460 mm	4.5 mm	
AOI		**PCB width**	**PCB length**	**Thickness**	**Component size**
Omron VT-S730	Min.	50 mm	50 mm	0.4 mm	
	Max.	460 mm	510 mm	4.0 mm	component height: 40mm
VI 3K2	Min.	50 mm	50 mm	0.5 mm	
	Max.	410 mm	530 mm	5 mm	
Screen printer		**PCB width**	**PCB length**	**Thickness**	**Pads from edge**
Horizon 265	Min.	50 mm	40 mm	0.4 mm	0.3 mm
	Max.	510 mm	508 mm	6 mm	--------
Vapor phase / reflow		**PCB width**	**PCB length**	**Thickness**	
Asscon VP 53	Min.	--	--		
	Max.	500 mm	350 mm		
Asscon VP 56	Min.	--	--		
	Max.	600 mm	500 mm		
Reflow Seho	Min.	50 mm	50 mm		
	Max.	500 mm	650 mm		
Wave soldering		**PCB width**	**PCB length**	**Thickness**	**Soldering bottom side**
Seho MWS - 2340					
	Min.	--	--		---------
	Max.	500 mm	400 mm		6.00 mm
Vacuum packing		**PCB width**	**PCB length**	**Thickness**	
NT1/46N (Boss)	Min.	--	--		
	Max.	500 mm	350 mm		
Others		**PCB width**	**PCB length**	**Thickness**	
Depaneling Maestro 4M (CAB)	Min.	--	--		
	Max.	450 mm	350 mm		
Screen cleaner Sys 152 / 2000	Max.	650 mm	650 mm		Screen prints / PCB

2.2 Clearance for transport bands

For transporting and clamping of the PCB, the SMT components shall have a distance to the edge of at least 5 mm.

If the required 5 mm cannot be kept, then consultation with the NPI or CAD department of exceet electronics is required. Maybe a special panel design or an additional edge has to be provided.



2.3 Base Material

FR4 (Standard at exceet electronics)

If other base materials should be used / required, please consult the CAD department of exceet electronics.

2.4 Pad Surface

With respect to cost effectiveness and customer preference, the following procedure are preferred to obtain a solderable surface.

- Immersion tin (Sn) (on customer preference)
- Immersion silver (Ag) (on customer preference)
- Immersion nickel-gold (Ni-Au) **(Standard at exceet electronics)**
- Hot Air Leveling (HAL) (lead-free) (after consultation)
- Organic Cu solderability preservative (OSP) (after consultation)

2.5 Layer Thickness of Circuit Board Surface

Layer thickness of circuit board surface has to ensure sufficient wetting during repeated soldering.

The following layer thicknesses are required:

- Immersion tin (Sn) ≥1.0 µ
- Immersion silver (Ag) ≥0,15u
- Immersion nickel-gold (Ni-Au) 3u bis 6u / 0,05u bis 0,12u
- Hot Air Leveling (HAL) (lead-free) ≥1u ≤25u
- Electrolytic gold typical 1,0u

2.6 Solder Resist Mask

Since the manufacturer of the circuit boards adapts the data himself, solder resist data should have the same size as the pads.

Solder resist must also be also provided between fine pitch pads.

To prevent short-curcuits, vias should be covered with solder resist (only onesided) unless they are used as test points. Different colors (on demand) can be realized - standard is green.

2.7 Silkscreen

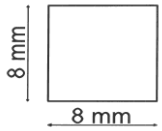
The print width and the gap to the pads is at least 0.2 mm. Pads must not be overprinted.

Different colors (on demand) can be realized - standard is white.

2.8 Clear Space for Barcode Labels

If possible, a clear space of 8 x 8 mm for 2D barcode labels should be provided on the circuit board.

The distance/clearance between the barcode label and SMT parts should exceed 5mm.



2.9 Fiducials

Basically there are two different kinds of fiducials on the circuit board.

2.9.1 Circuit Board Fiducial

In order to have a reference point on the circuit board, at least two fiducials should be on the board, preferably in diagonal direction and with maximum distance. Fiducials should be placed on Top-Side and BOT-Side. (Even at onesided designs)

Preferably use differents shapes for the two fiducials. This is especially important when providing symmetric designs.

Minimum distance of fiducial to circuit board edge (x/y): 5mm / 5mm

For boards in panels the panel should provide at least two additional fiducials. For positioning these marks the same guidelines as for the circuit boards should be applied.

2.9.2 X-out Fiducial

For PCBs in a panel there should be a separate x-out fiducial for every single board, placed on the edge of the panel, respectively close to the corresponding PCB.

In case of bad parts in the panel, the corresponding x-out fiducial will be pasted over by the PCB manufacturer.

Recommended fiducial geometry see 2.9.3

2.9.3 Recommended Fiducial

The surface of the mark should be smooth (even) and free of solder resist!

Following marks are preferred:

Circle (diameter 1.6 mm), circle (diameter 1.0mm), square, cross

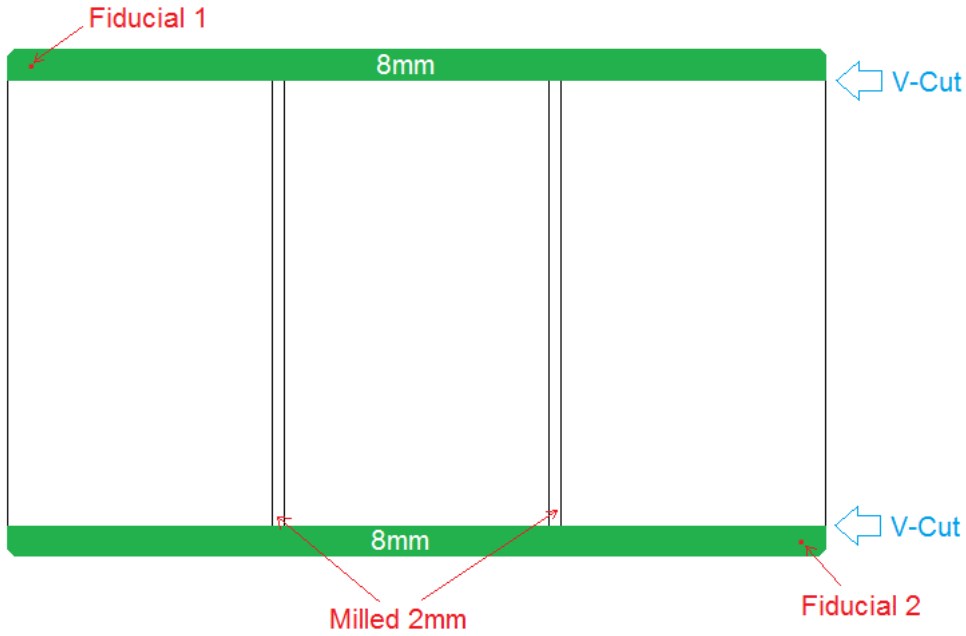
2.10 Panel design

For de-paneling a minimum distance of 0,5 mm between the nearest circuit track / copper surface and outline of the circuit board shall be provided (also for inner layers - power planes). For bigger panel designs please consider some additional space (example: addinional panel boarder in the middle) for the mechanical assistance system of the production line. Panel corners have to be chamfered.

For optimal panel design, please consult the NPI department of exceet electronics.

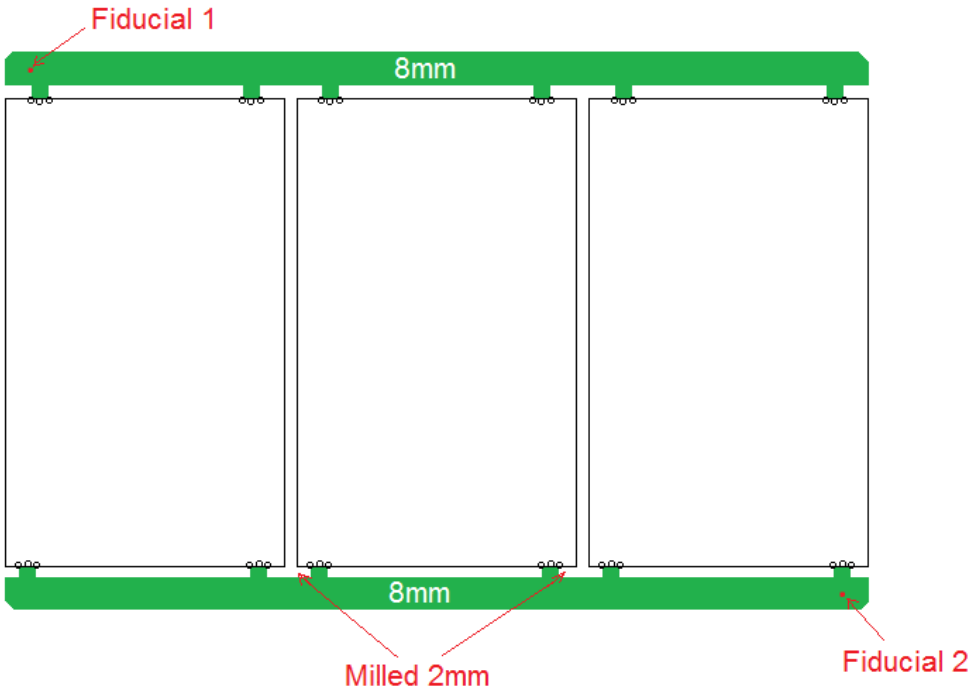
V-Cut panelization (preferred):

Keep a minimum distance of 2mm between assembled parts and outline (V-Cut).



Drilled panel tabs panelization:

If the outline is milled, the drills (panel tabs) should be placed into the circuit board
→ no rework required



2.11 PCB data

The circuit board data should contain the following information:

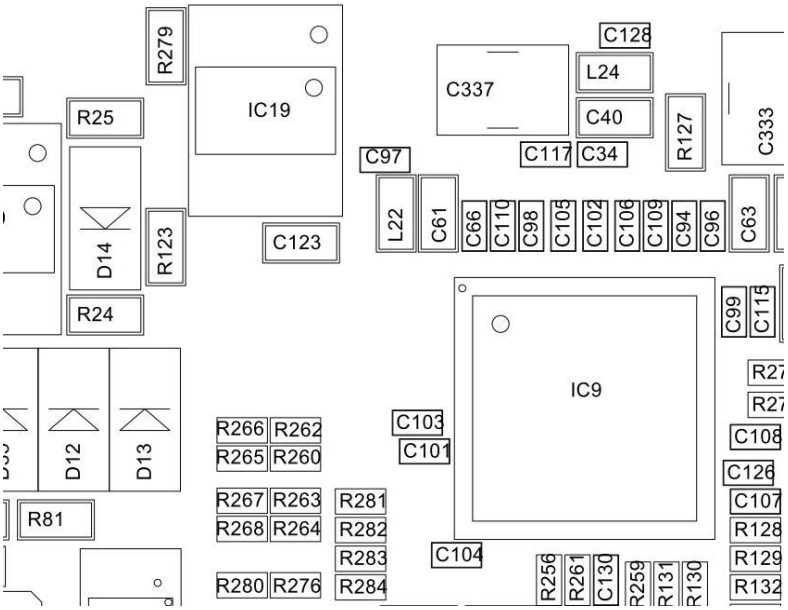
- Layer data in Extended Gerber Format
- Readme file with notes on:
  - Base material
  - Circuit board thickness
  - Copper thickness (base copper / finished copper)
  - Surface
  - Colour of solder mask
  - Assembly print *
  - Layer design / key to Gerber data names
  - Special remarks
- Drill data in Excellon format (seperate file for NPTH and PTH)
- Layer for Pastemask
- Design file for optical check and InCurcuit-Test (ODB++, Cadif, for Eagle Designs brd file or Fabmaster)

* Assembly Print Notes:

The Assembly Print (pdf, etc...) has to contain:

- All placeable Parts (not assembled parts too)
- Part-Outlines
- Part-Names (R1, C1, …)
- Direction of unipolar Parts

Example:



3 SMT

3.1 Component packaging

- Strip (Tape&Reel) (preferred)
- Tray (preferred)
- Tube (only for small batches)

The components should be capable for automated assembling. If special shapes or plugs are used, the components should be available with pick & place pads. If not, please contact the NPI department at exceet electronics; maybe pick tests can be made or special tools can be designed.

3.2 Pad Geometry

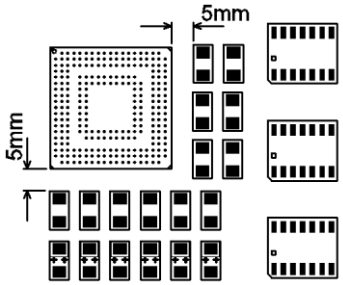
Basically, for each component there's a land pattern which represents the manufacturer's recommendation for the pad geometry – see datasheet.

Optimal pad geometries are described in detail in the IPC standard. To achieve the optimal geometry, a Land Pattern Generator can be used:

http://www.pcblibraries.com/

3.3 Note on BGA's

To enable optical checks or reworking (unsoldering) of a BGA, the following minimum distances to proximate components should be kept:



BGA environment on the circuit board

3.4 Assembling Data (Pick & Place data)

Attention: coordinate data for SMT assembling have to contain following information.

- Assembly position of all placeable parts (not assembled parts too)
- Component identification / design
- X coordinates from center of component (unit = mm)
- Y coordinates from center of component (unit = mm)
- Rotation

The coordinate file should also contain the positions of the Fiducials.
The origin of assembly data (xy) should be the lower left edge of the PCB.

Data have to be separated by a distinct separator mark.

3.4.1 Example

D5|0805LED_KPT2012MGC|38.418|29.210|270.0
D6|0805LED_KPT2012MGC|38.418|24.130|270.0
D7|0805LED_KPT2012SYC|59.055|66.358|90.0
F1|EIA481-2_SMD2920P075TS|43.180|26.670|270.0
IC1|TQFP44_T89C51CC01UA|35.560|56.515|0.0
IC2|SO08_82C251T|45.085|5.080|180.0
IC3|NB|31.750|26.670|0.0
FID1|Fiducial_1|10.000|5.000|0.0
FID2| Fiducial_2|150.000|95.000|0.0

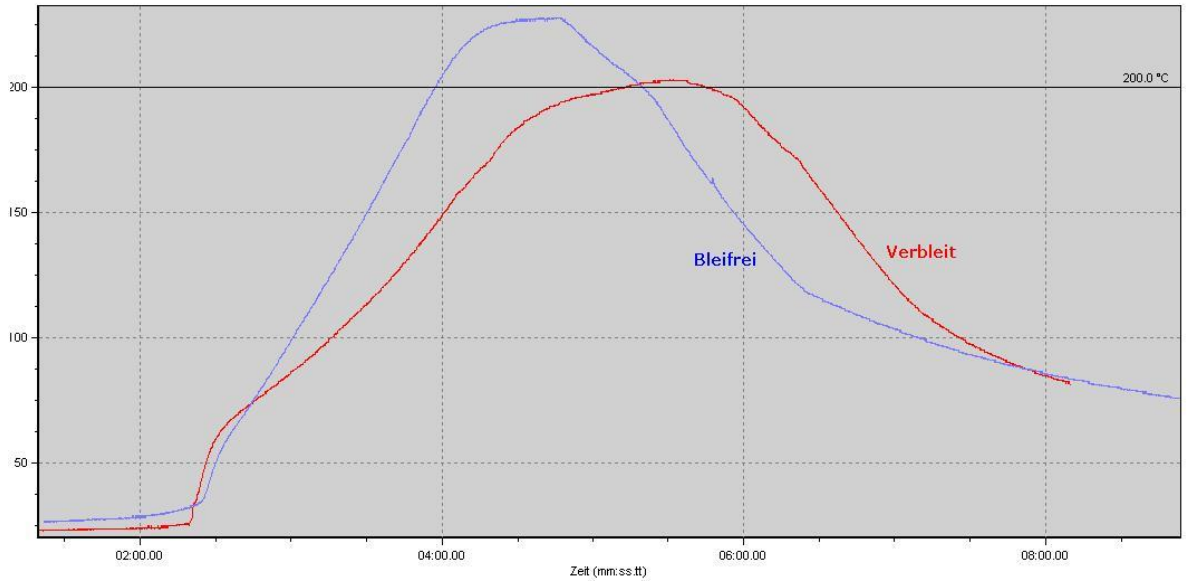
3.5 SMT Soldering

3.5.1 Double-sided

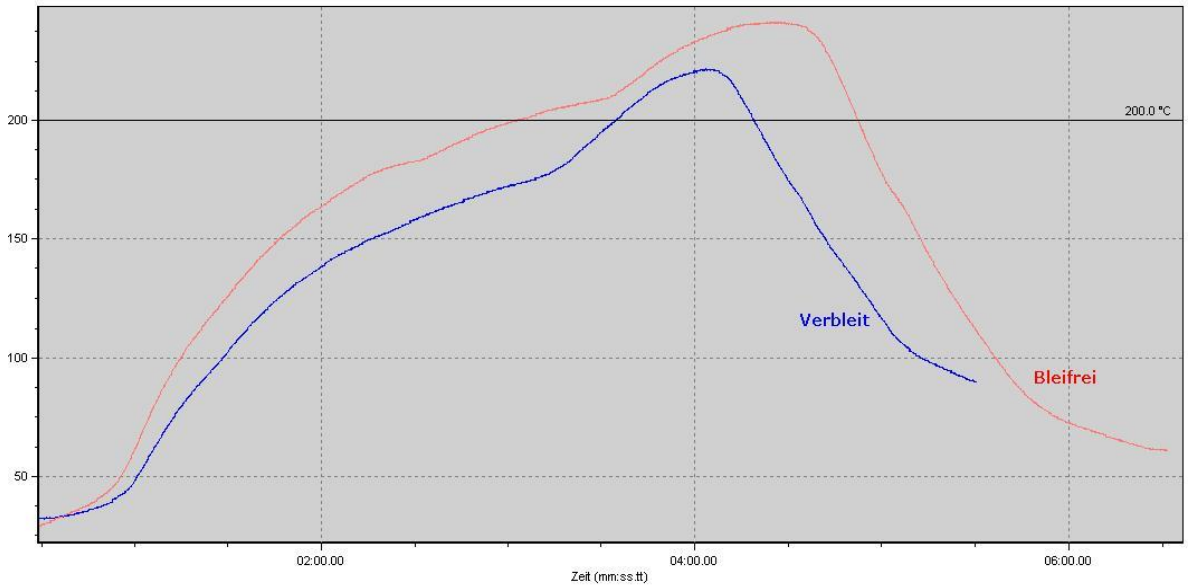
In order to avoid that components detach from the circuit board during the second soldering process, following notes should be considered:

- The circuit board and the components should endure the additional temperature stress without defects.
- The components should be arranged on the circuit board such that all light-weight parts are on one side and all heavy parts on the other side. This is a precondition that no component detaches from the board during the second soldering process.

3.5.2 Soldering Profile Vapor Phase



3.5.3 Soldering Profile Reflow



4 THT

4.1 THT Soldering

By means of optimized component arrangement the efficiency during manufacturing can be increased considerably.

4.1.1 Selective Wave Soldering

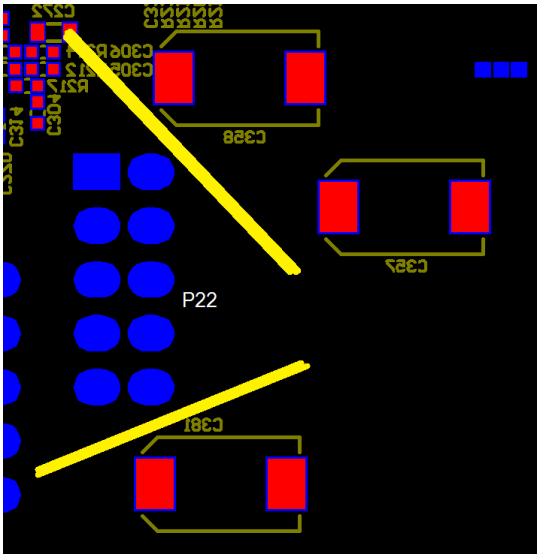
In case there are THT components on a double-side SMT circuit board, the layout has to permit machine wave soldering with a soldering mask.

Preconditions for the use of a soldering mask are:

- Clearance space between the THT pins to be soldered and SMT chip components (1206, 0603, 0402, …) of at least 2mm. In case of higher SMT components, the distance has to be increased.
- SMT component height on the soldering side max. 10mm

Examples:

Critical THT-part to be soldered: P22

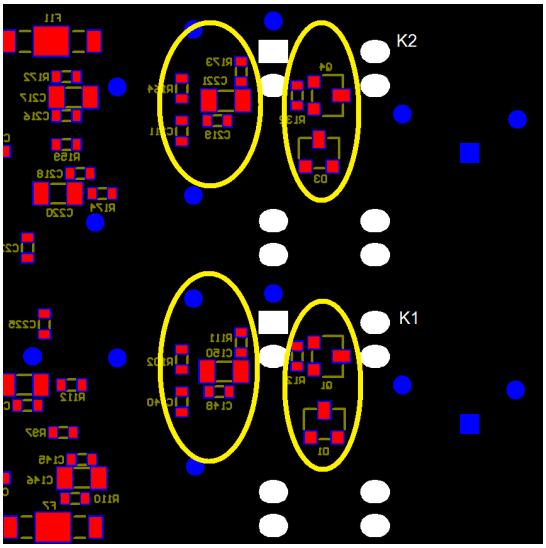


The distance between the SMT-parts C358/C381 and the THT-Pins (part P22) that have to be soldered is 2.1mm.

If C358/C381 would be chip size (0402, 0603, 0805, …) the distance would be enough.

But the height of C358/C381 exceeds chip-size, so the distance to P22 has to be increased.

Critical THT-parts to be soldered: K1, K2



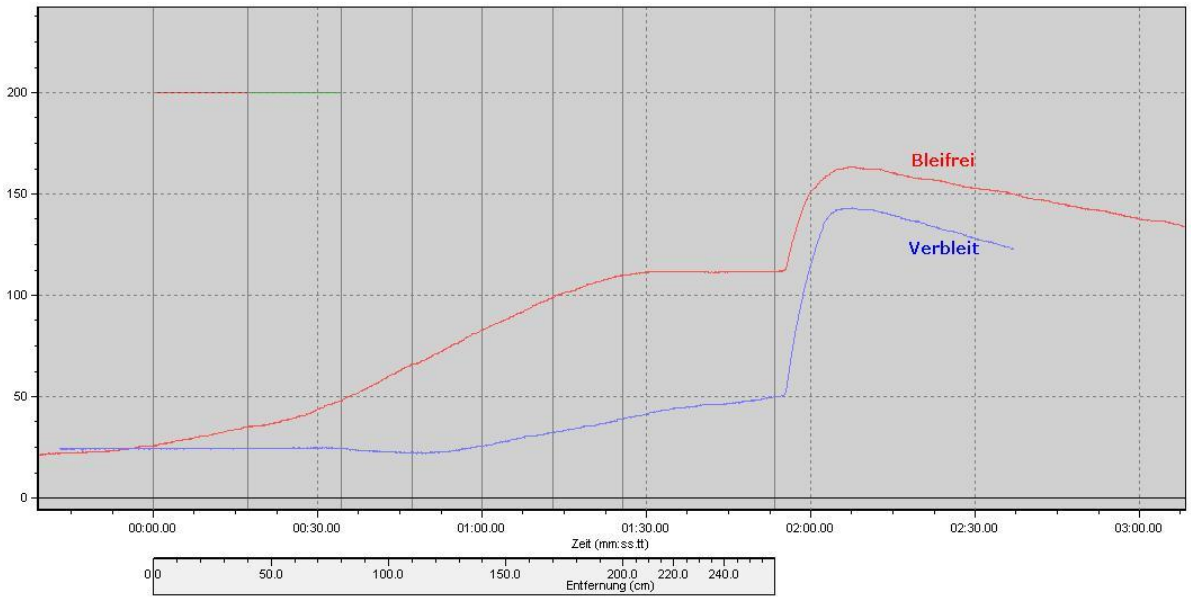
The distance between the SMT-parts and the THT Pins to be soldered is only 0.15mm.

Even if the SMT-parts would be chip-size, the distance would be too small.

To solder K1, K2 with selective wave soldering process, the SMT-parts have to be replaced by the CAD engineer.

White… THT-Pins (parts K1, K2) to be soldered
Red… SMT-Pads/Parts
Yellow… SMT-parts that have to be replaced

4.1.2 Soldering Profile Wave



5 Design for Testability

The design of the board has an significant impact on the achievable quality of testing.

5.1 Constructive Designrules

5.1.1 Contacting

To permit an InCurcuit-Test, all electrical nets of the board have to be contacted. It is possible to contact on test pads (priority 1 – preferred), THT-pins (priority 2) and vias (priority 3).
Single-side contacting (only TOP, only BOT) keeps adapter cost low.

5.1.2 Center Holes

To center the board in the test adapter, the design should contain additional (same drill-process than the other holes on the board) center holes. The diameter should be in the range of 2.00m – 3.5mm. To avoid an wrong insertion of the board, the center holes should be placed asymmetrical.

5.1.3 Test Points

Priority 1: 100mil Contact-Needle (default)

- Required diamter of test pads: >0.8mm
- Required distance from test pad to test pad (center <-> center): >2.05mm

Priority 2: 75mil Contact-Needle

- Required diamter of test pads: >0.4mm (preferred >0.5mm)
- Required distance from test pad to test pad (center <-> center): >1.65mm

Priority 3: 50mil Contact-Needle

- Required diamter of test pads: >0.2mm
- Required distance from test pad to test pad (center <-> center): >1.25mm

5.2 Electrical Designrules

5.2.1 Functional tests with high current loads

For currents above 0.5A, provide more than one contact points (test pads, THT-pins, vias)

5.2.2 Arrays / Serial connections of resistors

To measure resistance-arrays make sure that at least 1 of the internal resistors is contactable.
To measure resistors that are connected in series it's sufficient to provide contact points at both ends of the serial connection.

5.2.3 4-wire measuring

To measure resistors smaller 22Ω, 4 contact points are required. (-> drawing). If not possible, at least 3 contact points have to be provided (1, 2, 3 or 2, 3, 4)

Drawing: 4-wire measuring

6 Design for Coatability

The layout has to be suitable for all common coating methods (dipping, flooding, spraying, etc.). Components that have not to be coated (plugs, sockets, switches, buzzer, dip switches, LEDs, etc.) must be maskable.

Requirement:
The clearance around components and areas of the PCB that are not allowed to be coated must exceed 2mm.

7 References

Acceptance criteria for electronic assemblies - IPC-A-610

8 Contact Person

Ing. Andreas Treichl
Leitung NPI / NPI manager
mail: andreas.treichl@exceet.at
tel: +43/5373/43143-611

Ing. Christoph Rieser
New Product Introduction
mail: christoph.rieser@exceet.at
tel: +43/5373/43143-651

Johann Kitzbichler
New Product Introduction
mail: johann.kitzbichler@exceet.at
tel: +43/5373/43143-661

Dipl.Ing.(FH) Wolf Dietrich Metz
Entwicklung Layout / PCB Design
mail: wolf-dietrich.metz@exceet.at
tel: +43/5373/43143-542

Ing. Günther Unterrainer
Entwicklung Layout / PCB Design
mail: guenther.unterrainer@exceet.at
tel: +43/5373/43143-541

Andreas Unterberger
Prozesstechnik / process engineer
mail: andreas.unterberger@exceet.at
tel: +43/5373/43143-786

9 Disclaimer

The processes and suggestions for use described in this document do not in any way constitute a guarantee.

The issuer accepts no liability for the accuracy and completeness of this document and its utilization.

Exceet electronics reserves the right to make technical changes to the document at any time without liability to inform the user thereof.

The use of the know-how in this document does not imply a liability of the issuer for claims of any kind.