

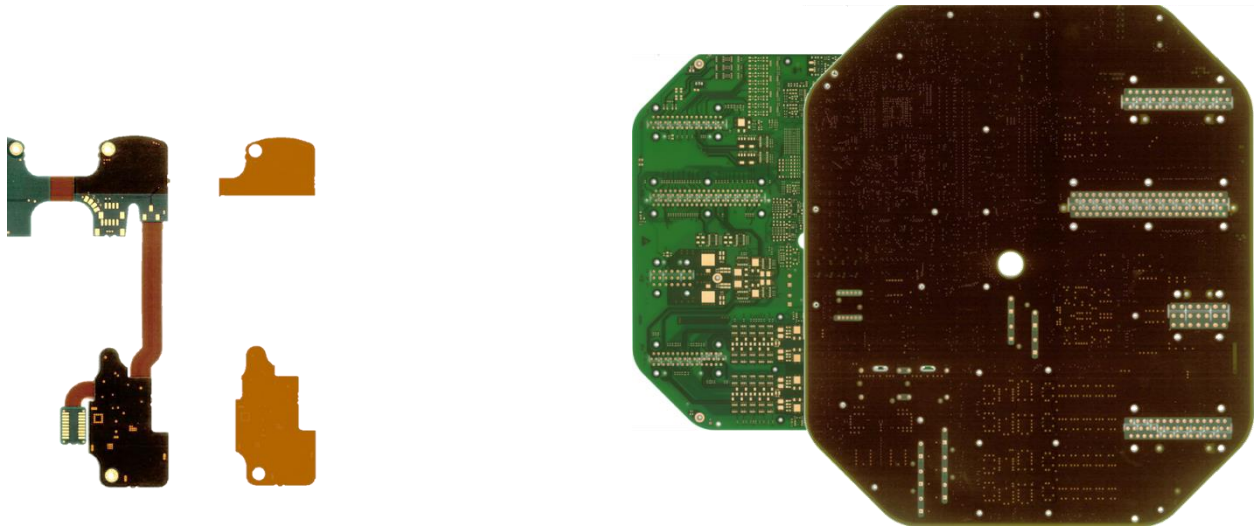
Design Rules

Insulation foil

These design rules apply to:

Insulation foils for insulating and protecting printed circuit boards from external impacts.

Examples:



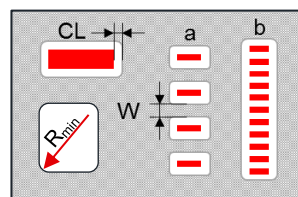
RIGID.flex with insulation foil on rigid areas

Motor control unit for screwing onto heatsink

Material specifications

Material	Standard	Spec. sheet	Descriptions	Applications
Coverlay	IPC-4203	1 / 2	Polyimide foil 25µm, Acrylic or Epoxy-adhesive	Insulation, protection

Clearances, radii and webs



SMD pads

- a single clearance
- b block clearance

Symbol	Description	Technical standard	Advanced requirements
	Minimum size insulation foil	15mm x 15mm	10mm x 15mm
CL	Coverlay clearance	500µm	
W	Minimum web width	500µm	
Rmin	Minimum radius	2mm	

Further specifications available on request, please contact us: flex@we-online.com

Design Rules

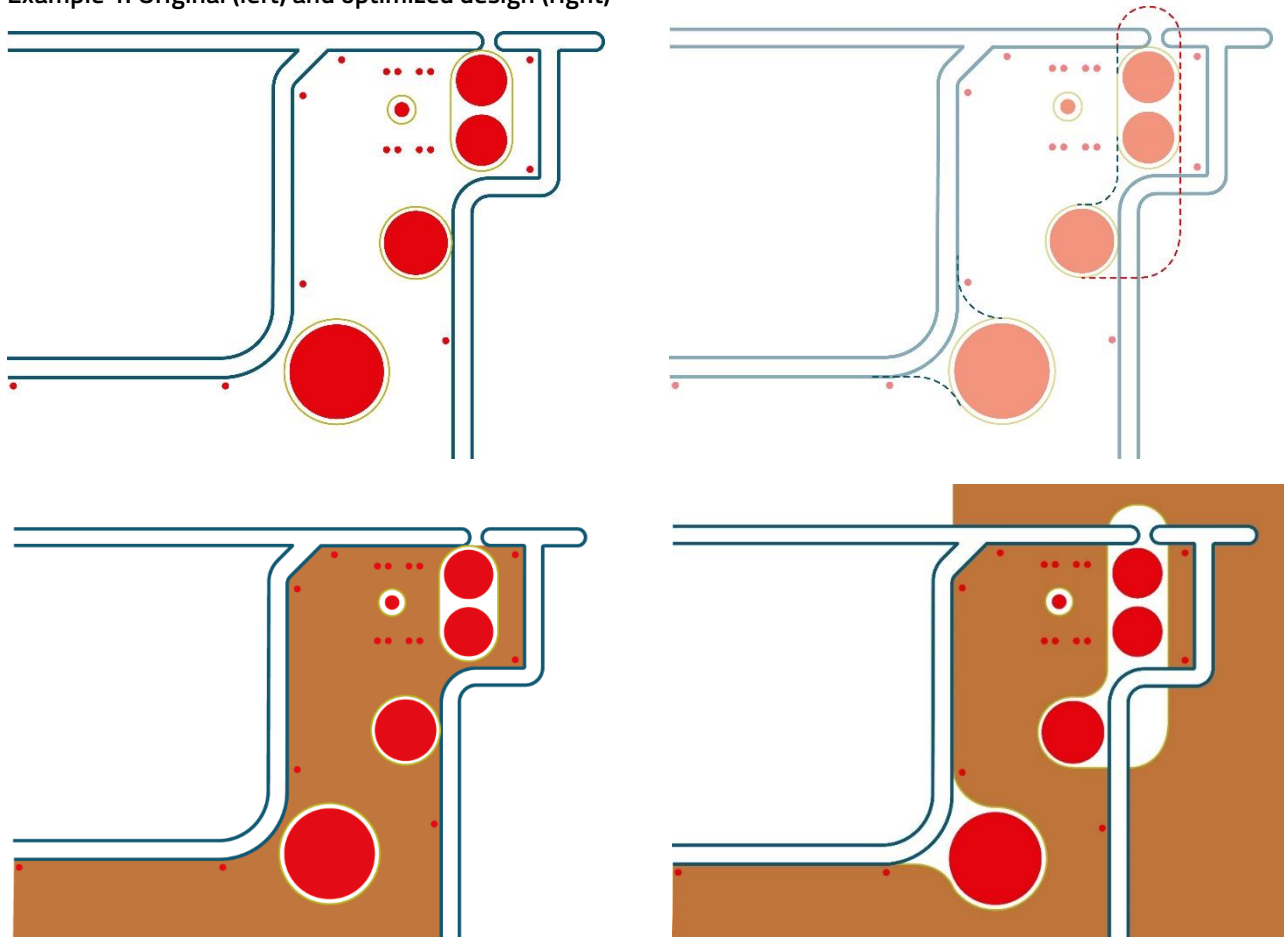
Insulation foil

Design specifications for good manufacturability

- Respect minimum web widths, see parameter "W".
- Avoid sharp edges, use radii.
- Round off outer and inner radii with at least R2.

Frequent problem: Web width W too small due to small distance between opening and outline.

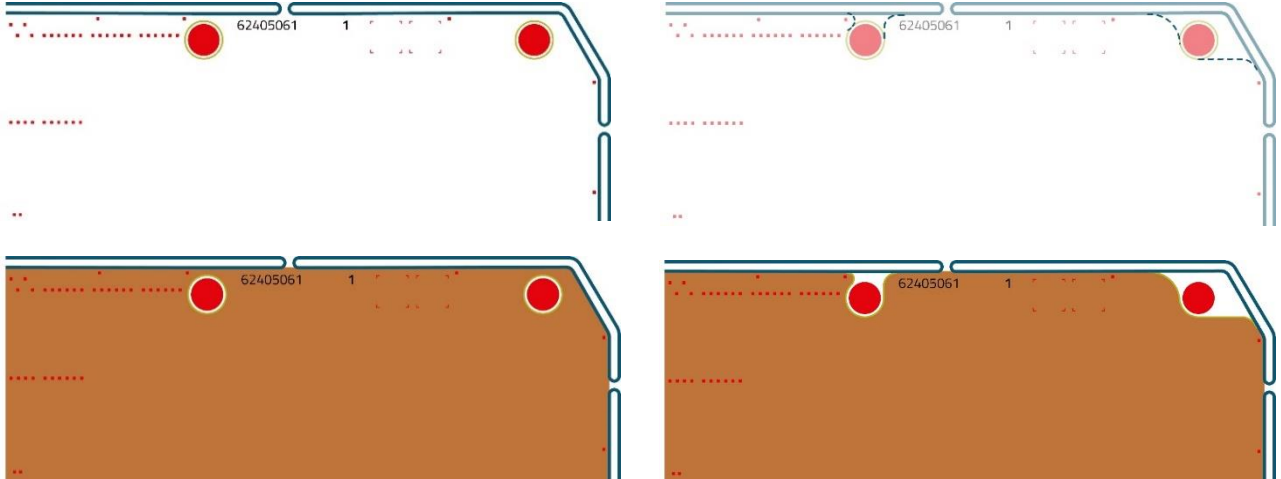
Example 1: Original (left) and optimized design (right)



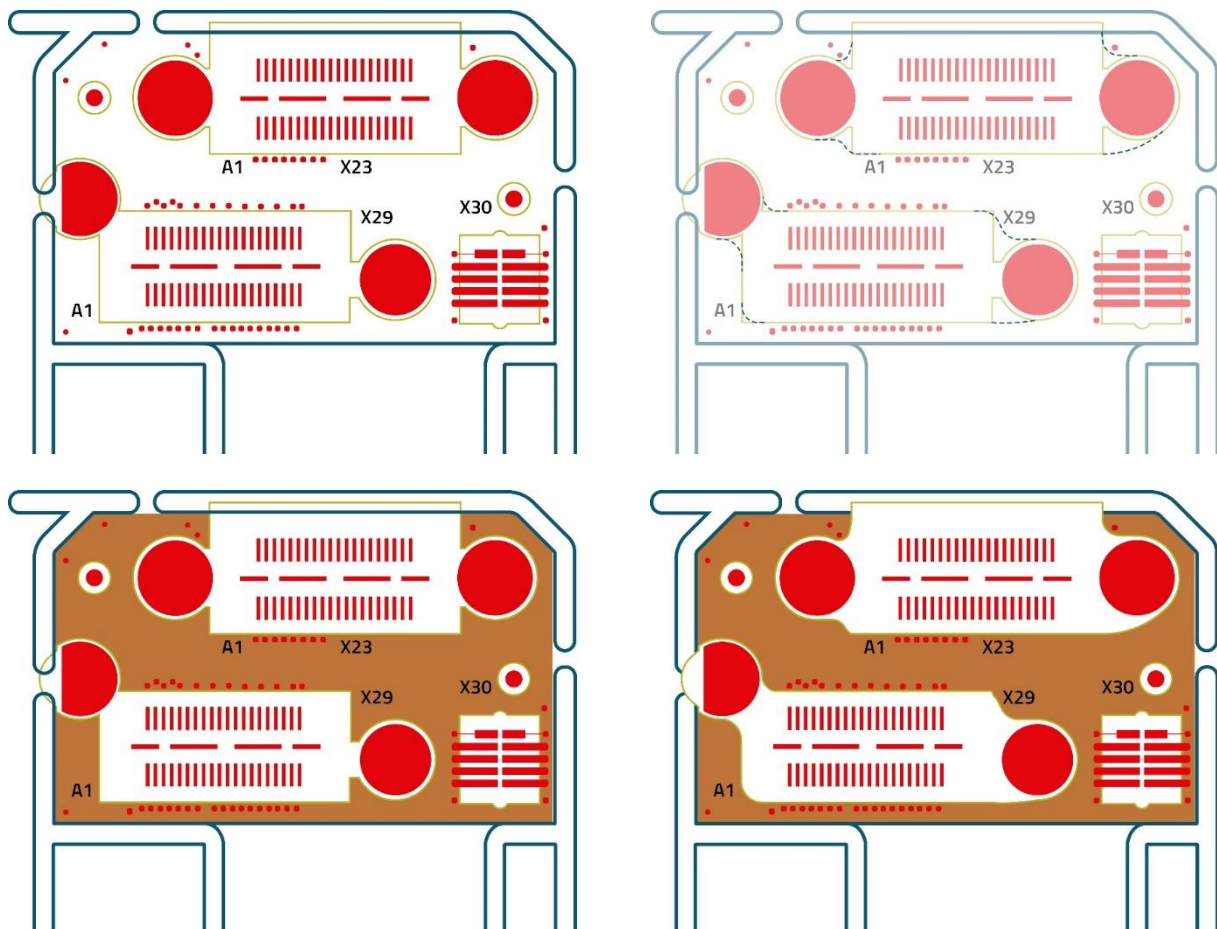
Design Rules

Insulation foil

Example 2: Original (left) and optimized design (right)



Example 3: Original (left) and optimized design (right)



Design Rules

Insulation foil

More information:

- The adhesive of the coverlay is only conditionally suitable for alkaline cleaning processes. The use depends on the cleaning process and the layout design, so must therefore be qualified by the user.
- Infiltration of the coverlay with cleaning media can be reduced or prevented if no conductor pattern structures (tracks, markings, plane splits) are contained in a 5 mm edge area of the cover-lay.
- In principle, coverlay can be applied to the final surface or to bare copper. If the coverlay is applied to bare copper, areas underneath and the edge area outside the coverlay (approx. 1 mm) are not or only insufficiently protected with the end surface. Moisture or contamination that is not removed can therefore lead to corrosion under certain conditions. Solderability in this edge area is limited. (See manufacturer information).
- Covering metallized holes or milled out areas on bare copper on one side leads to interference in the deposition of the final surface and thus reduces the reliability of the circuit. Therefore, holes and milled out areas must be exposed on both sides. This provides metallized holes with the final surface finish and protects them, as well as reducing the risk of the foil peeling off. Alternatively, holes and cutouts can be covered with coverlay on both sides.