EASYLOGIX.DE

CAD/CAM for electronics

DFM/DRC

- *Solder Paste*
- *Solder Mask*
- *Signal*
- *Drill*
- *Component*
- *Component Pins*

Schindler & Schill GmbH
Bruderwöhrdstr. 15b
93055 Regensburg
Deutschland

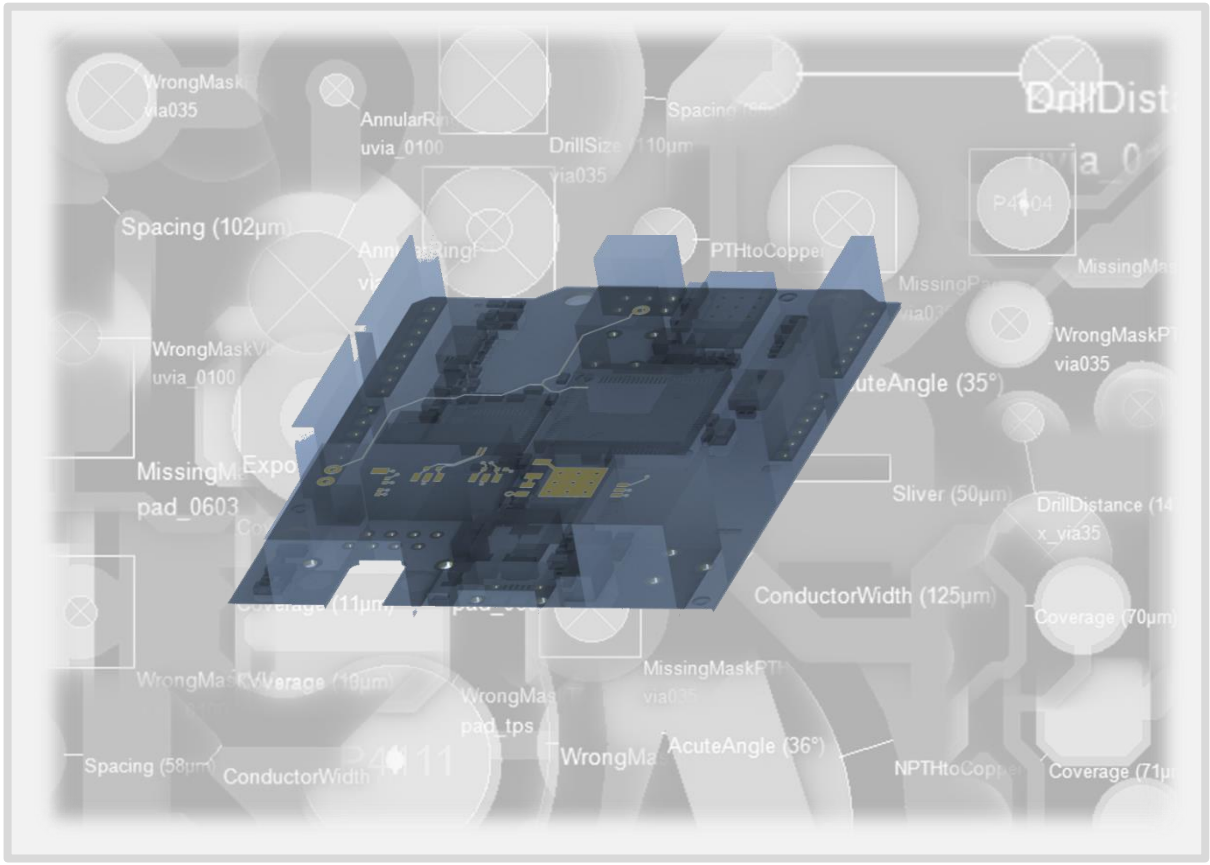
Tel: +49 941 604 889 719
Email: info@easyLogix.de
Web: www.easyLogix.de

11/2015

Design for Manufacturing

- DFM
- Data Formats
- Workflow
- Preparations
- Results
  - Copper
  - Mask
  - Drills
  - Components

DRC Rules to increase quality and reduce costs.
The rules apply to **PCB Manufacturing** and **Assembly**

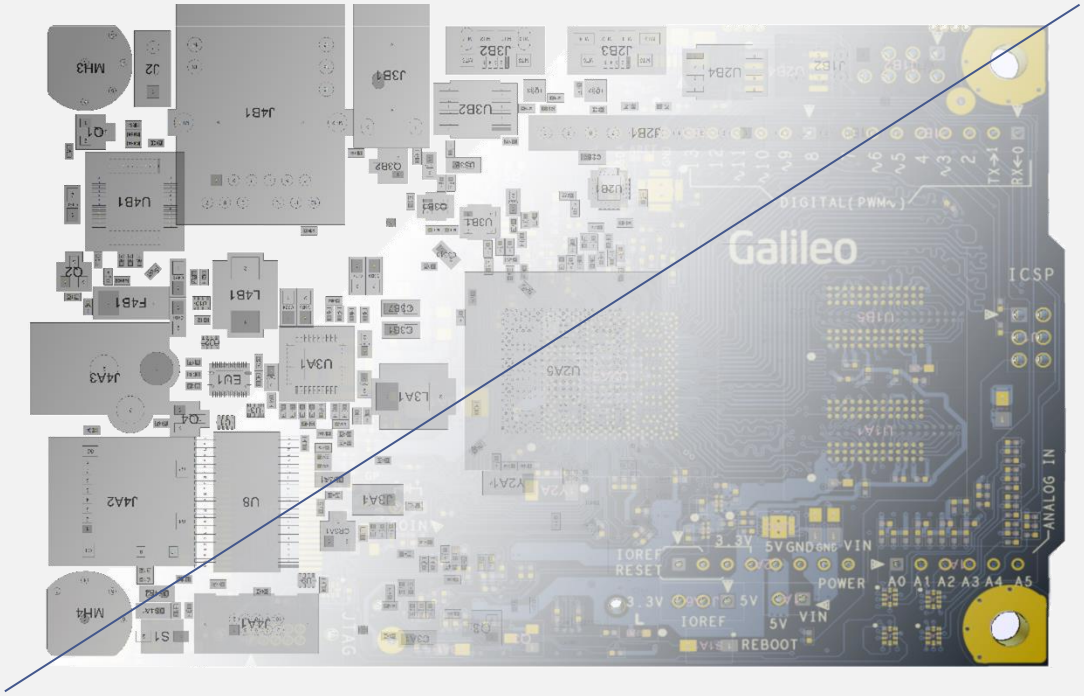


Design for Manufacturing

- DFM
- Data Formats
- Workflow
- Preparations
- Results
  - Copper
  - Mask
  - Drills
  - Components

There are many checks requiring the knowledge of both, component and Bare Board information.

PCB Assembly



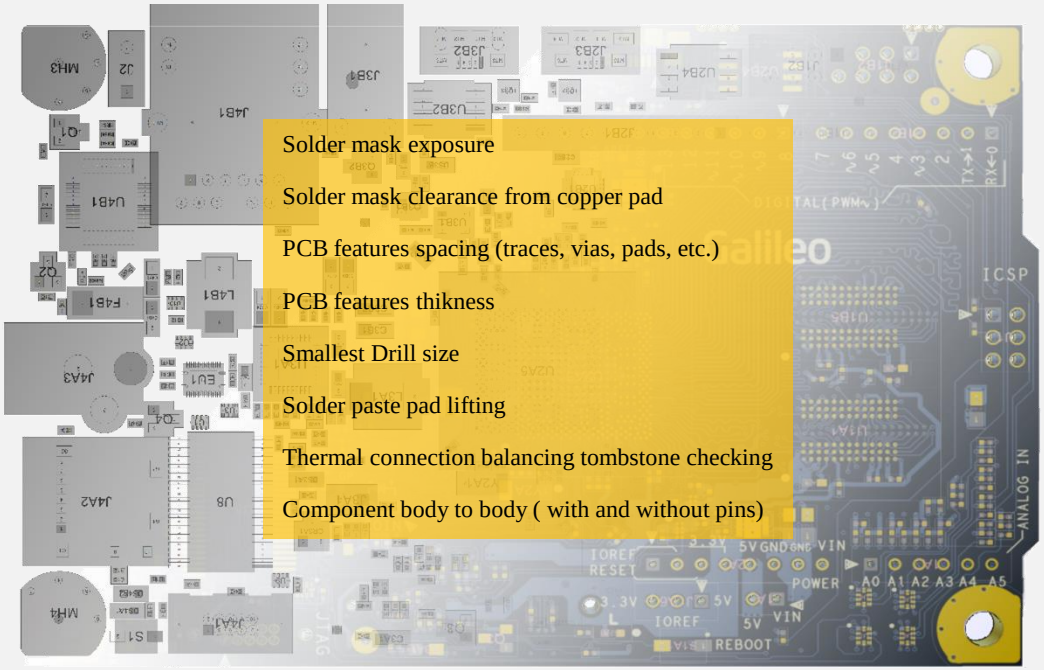
PCB Bare Board

Design for Manufacturing

- DFM
- Data Formats
- Workflow
- Preparations
- Results
  - Copper
  - Mask
  - Drills
  - Components

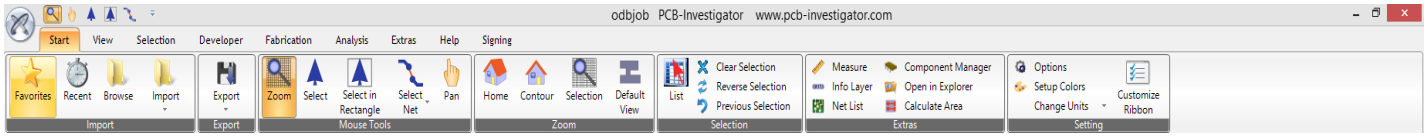
There are many checks requiring the knowledge of both, component and Bare Board information.

PCB Assembly

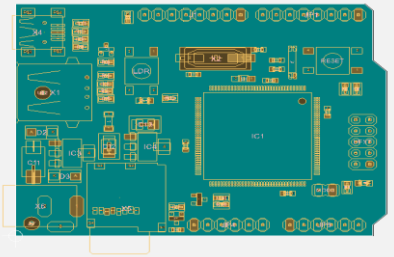


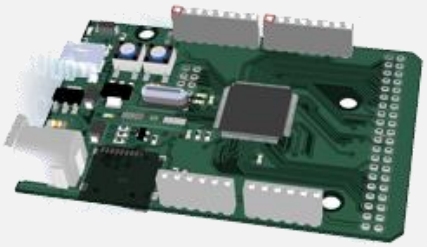
PCB Bare Board

DFM – Data Formats



Format 2D	Import	Export
ODB++	Yes	Yes
IPC-2581	Yes	Yes
GenCAD 1.4	Yes	Yes
EAGLE „ULP“	Yes	No
Gerber274x	Yes	Yes
Excellon	Yes	No
Sieb&Meyer	Yes	No
DXF	Yes	Yes
PDF	Yes	Yes

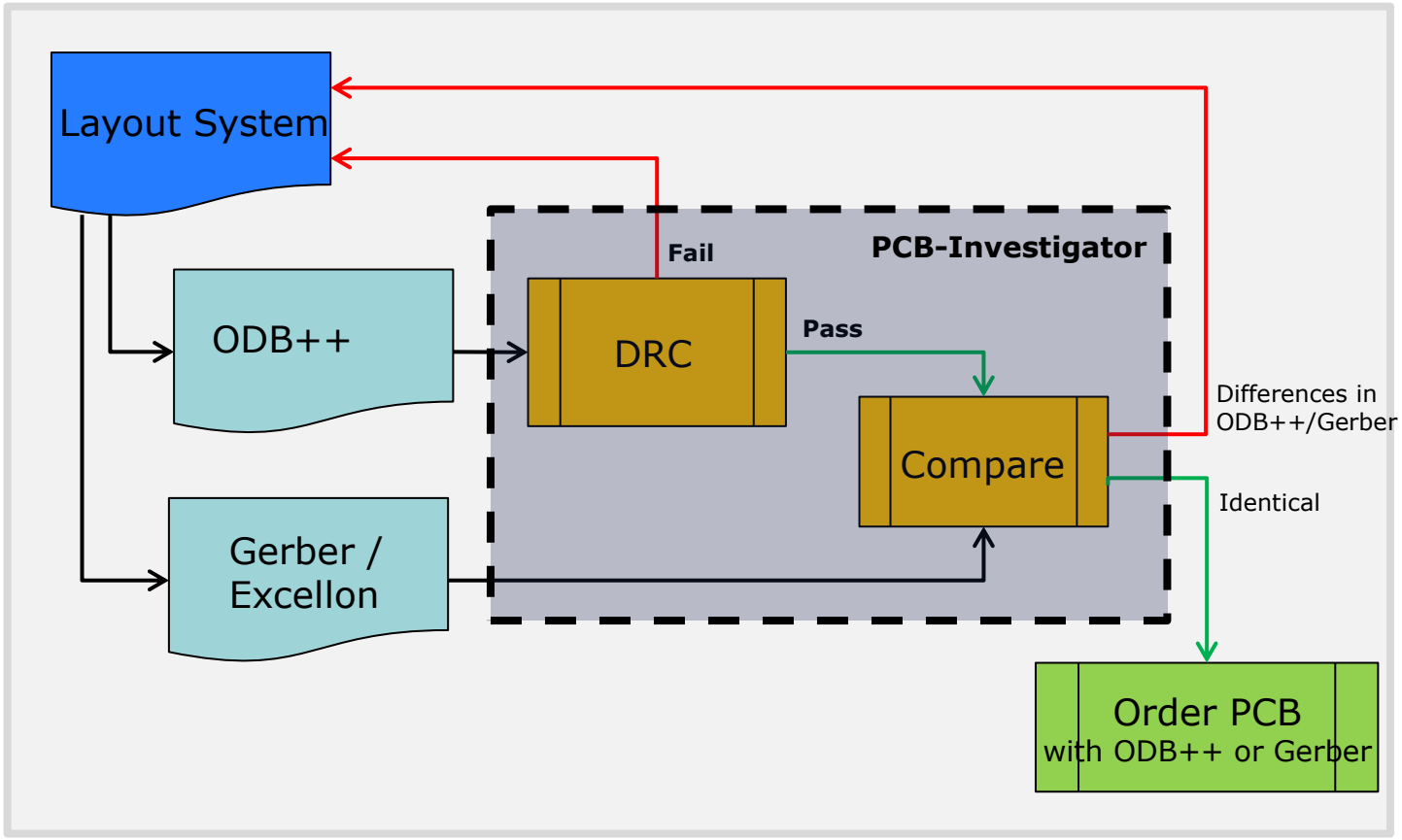




Format 3D	Import	Export
VRML	Yes	Yes
Obj	Yes	Yes
X3D	Yes	Yes
XAML	Yes	Yes
IDF 2.0/3.0	Yes	Yes

DFM – Workflow

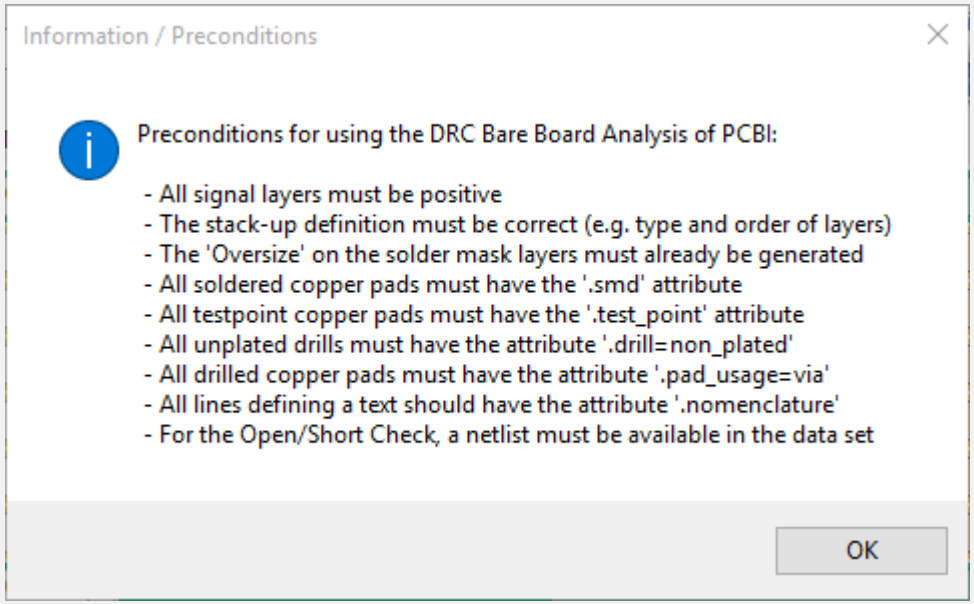
DRC check should be done on ODB++ to utilize all information integrated in the ODB++ structure



DFM – Preparations / Setup

Is data preparation necessary?

In case your ECAD System Library does not add the proper attributes to the design, it is possible to add the missing attributes with a small script to prepare your PCB-Data.

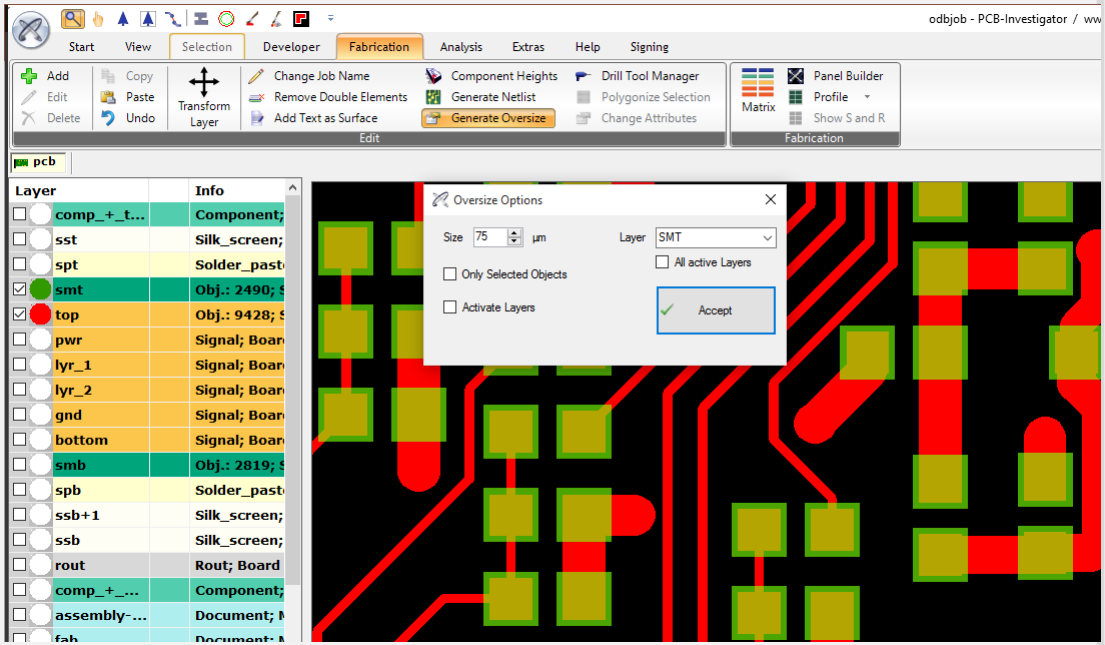


DFM – Preparations / Setup

Is data preparation necessary?

Depending of the used library, the Solder Mask Oversize is already applied in the data or must be generated with PCB-Investigator.

The Solder Mask oversize is used to make sure, that all soldering pads are free of lacquer, when taking the tolerances of manufacturing into account.

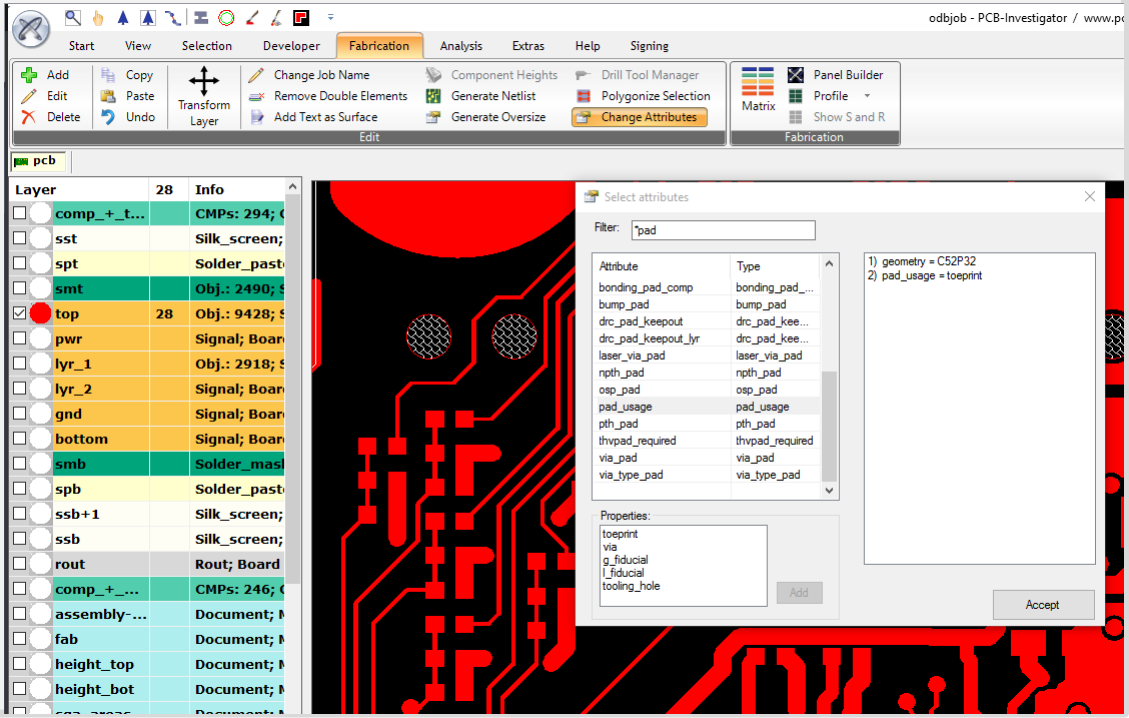


DFM – Preparations / Setup

Is data preparation necessary?

Preparing PCB fabrication data from Gerber, Excellon Sieb&Meyer

Gerber data does not include information about the type or purpose of the files.
This information must be added.

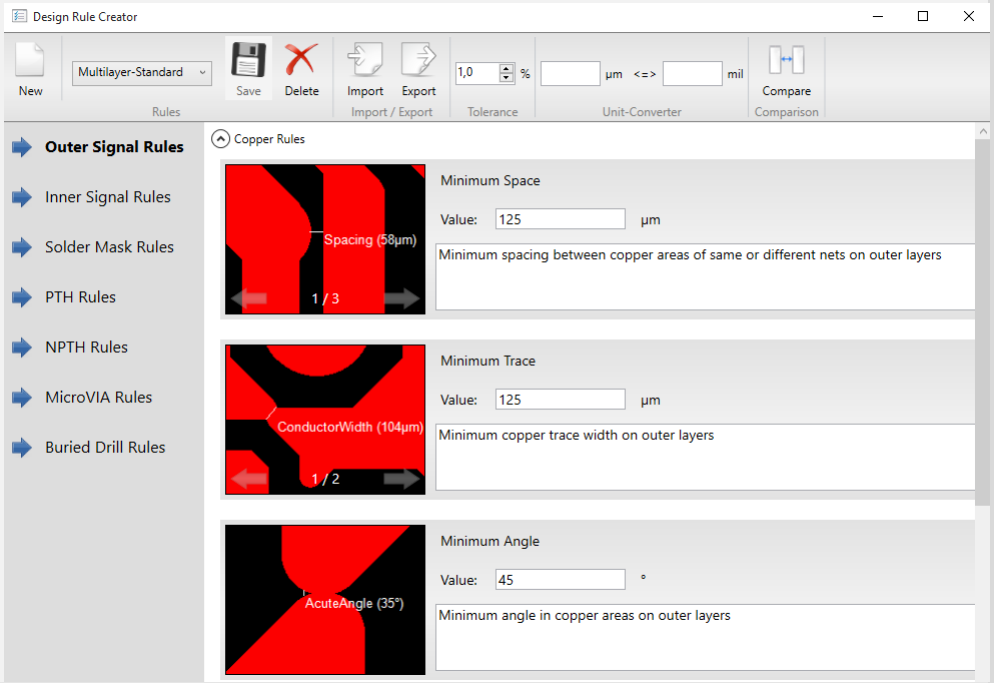


DFM – Preparations / Setup

Is data preparation necessary?

The DRC Rule File Manager let you set up different categories of PCB types, like standard, advanced, HDI, SBU or any other used technology

Creating rule files is supported by the rule file editor, which creates a XML file to prepare and exchange setups



DFM – Preparations / Setup

Is data preparation necessary?

The created rule sets can easily be compared to find all different check values and characteristics of a certain rule set.

All different values between two selected rule sets are highlighted

Compare Rule Sets

Multilayer-Standard | Multilayer-HighEnd

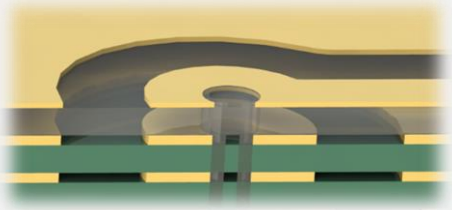
Rule	Multilayer-Standard	Multilayer-HighEnd	Description
Outer Signal Rules			
Minimum Space	125 µm	100 µm	Minimum spacing between copper areas of same or different nets on outer layers
Minimum Trace	125 µm	100 µm	Minimum copper trace width on outer layers
Minimum Angle	45 °	45 °	Minimum angle in copper areas on outer layers
Mask Clearance for SMD Pads	75 µm	50 µm	Minimum needed solder mask clearance arround SMD pads (.smd Attribute)
Mask Clearance for Testpoint Pads	75 µm	50 µm	Minimum needed solder mask clearance arround test point pads (.test_point Attribute)
Inner Signal Rules			
Minimum Space	125 µm	100 µm	Minimum spacing between copper areas of same or different nets on inner layers
Minimum Trace	125 µm	100 µm	Minimum copper trace width on inner layers
Minimum Angle	45 °	45 °	Minimum angle in copper areas on inner layers
Solder Mask Rules			
Distance to Copper	75 µm	50 µm	Minimum distance from the solder mask opening to surrounding copper
Minimum Space	70 µm	70 µm	Minimum spacing between solder mask clearances (=smallest solder resist fillet)
Minimum Trace	70 µm	70 µm	Minimum width of solder mask clearances
Minimum Angle	45 °	45 °	Minimum angle in solder mask clearances
PTH Rules			
Mask Clearance for PTHs	75 µm	50 µm	Minimum needed solder mask clearance for plated through holes
Mask Clearance for PTH-Pads	75 µm	50 µm	Minimum needed solder mask clearance for plated through hole copper pads
Accept masks smaller as PTH-Pad	on	on	If active, mask clearances which are smaller than the PTH-Pad are not reported
Annular Ring on Outer Layers	150 µm	100 µm	Minimum annular ring for the drill on outer signal layers
Annular Ring on Inner Layers	150 µm	100 µm	Minimum annular ring for the drill on inner signal layers
Accept missing Pads on Inner Layers	on	on	If active, missing copper pads on inner signal layers will not be reported
Distance to Copper	275 µm	275 µm	Minimum distance to surrounding copper on inner layers, if missing pads are accepted
Minimum Diameter	250 µm	200 µm	Minimim diameter of plated through holes
Minimum Drill Distance	300 µm	300 µm	Minimum Distance to any other Drill
NPTH Rules			
Mask Clearance for NPTHs	225 µm	200 µm	Minimum needed solder mask clearance for unplated through holes
Distance to Copper on Outer Layers	300 µm	300 µm	Minimum distance to surrounding copper on outer signal layers
Distance to Copper on Inner Layers	300 µm	300 µm	Minimum distance to surrounding copper on inner signal layers

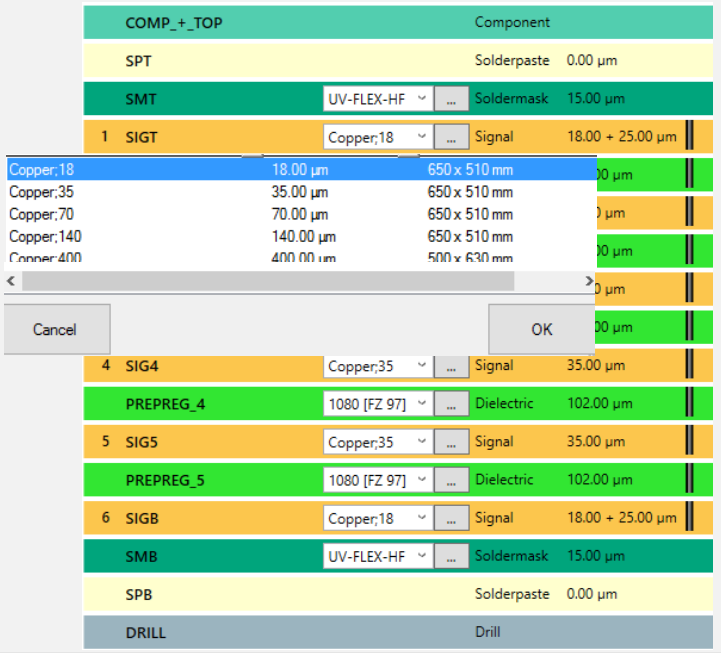
DFM – Preparations / Setup

PCB Stack-Up Management

Layer
.position
.type (Component, Solder_Mask,Signal,Drill,Rout)
.thickness

The stack-up definition can be done in the ECAD system or with PCB-Investigator. A library is used to share stack-ups. The Stack-Up can be imported / exported as XML, ODB++ and IPC2581



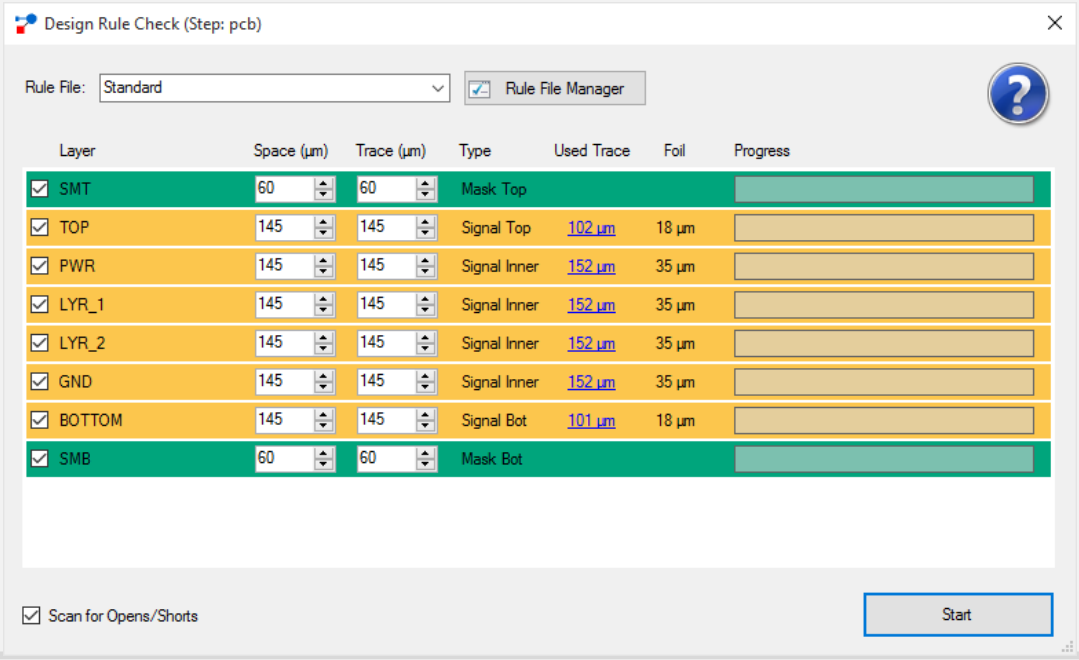


DFM – Preparations / Setup

DRC violations have a strong impact on the quality and price

Running the DRC helps to find violations and to categorize the design.

Before the DRC is started a pre-check reports the copper foil and smallest line used for each layer

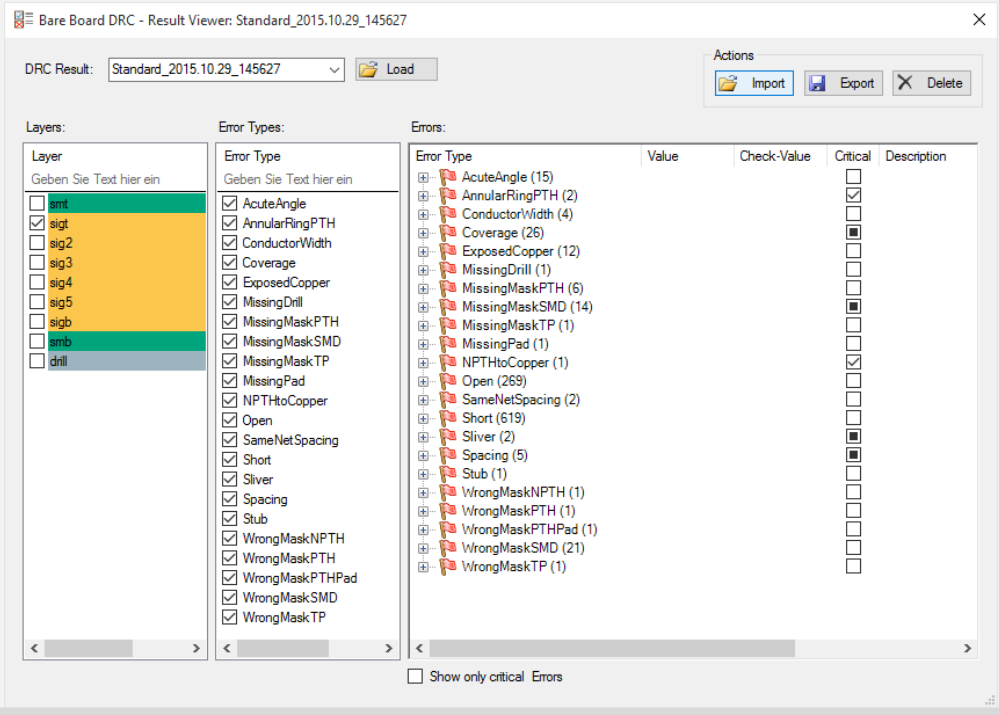


DFM – Results

DRC results for Bare Board and Assembly

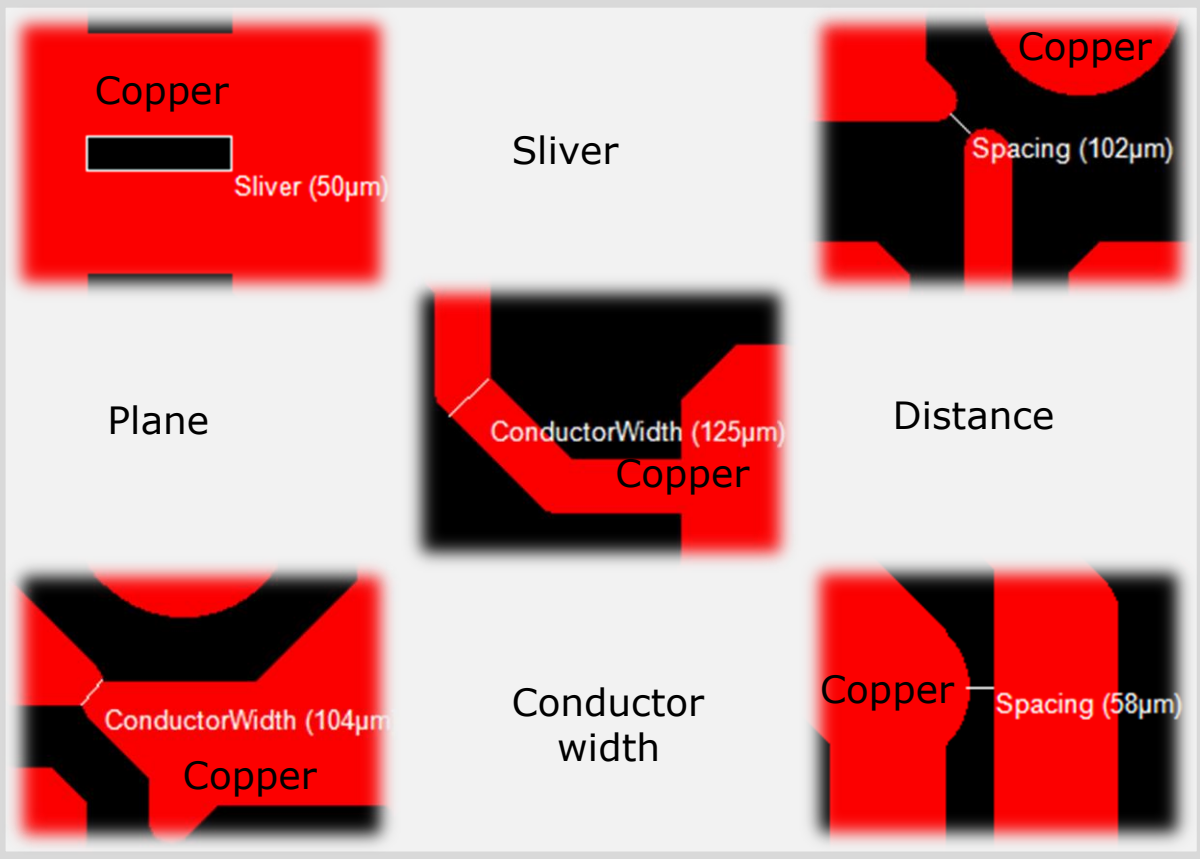
DRC for Solder Mask, Signals and Drills in conjunction to each other

All results are categorized and summed up per layer.



DFM – Copper Signal Check

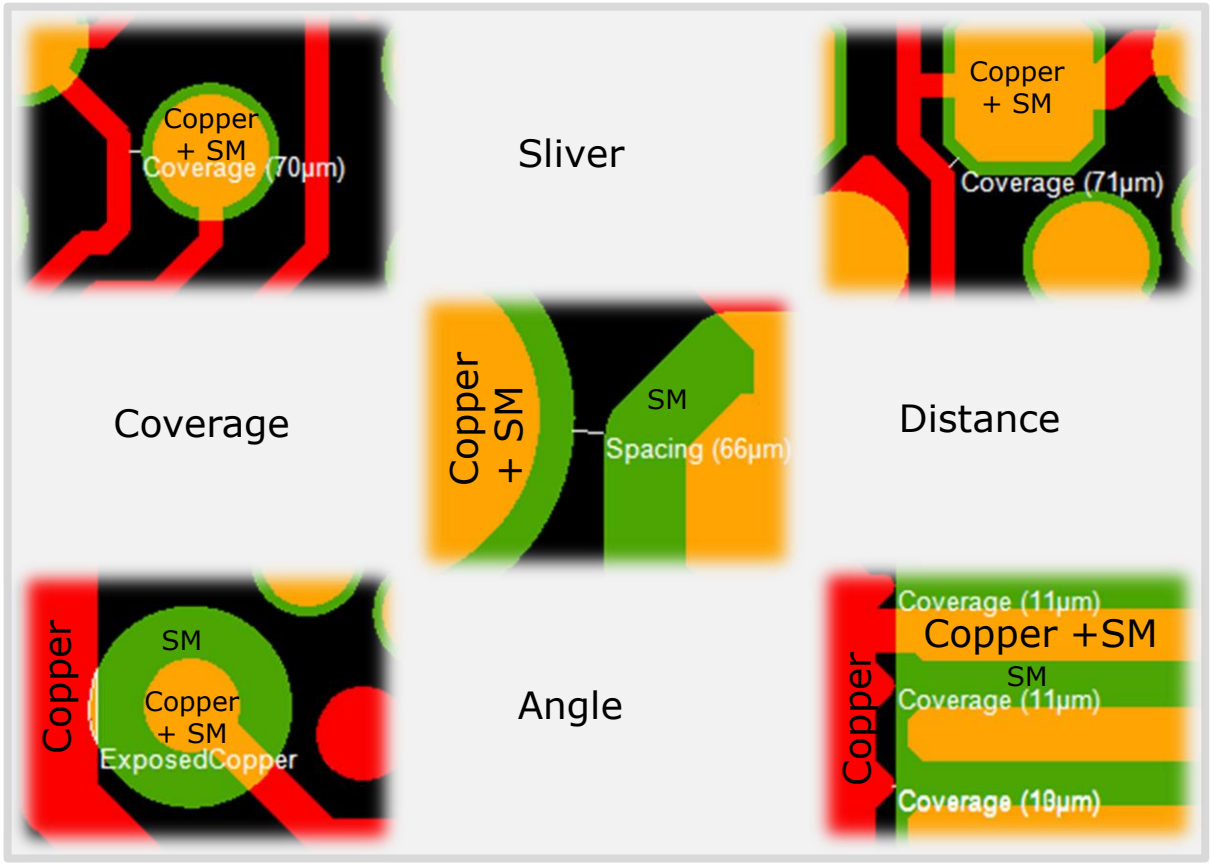
DRC Signal Analysis for outer and inner layers



DFM – Solder Mask Check

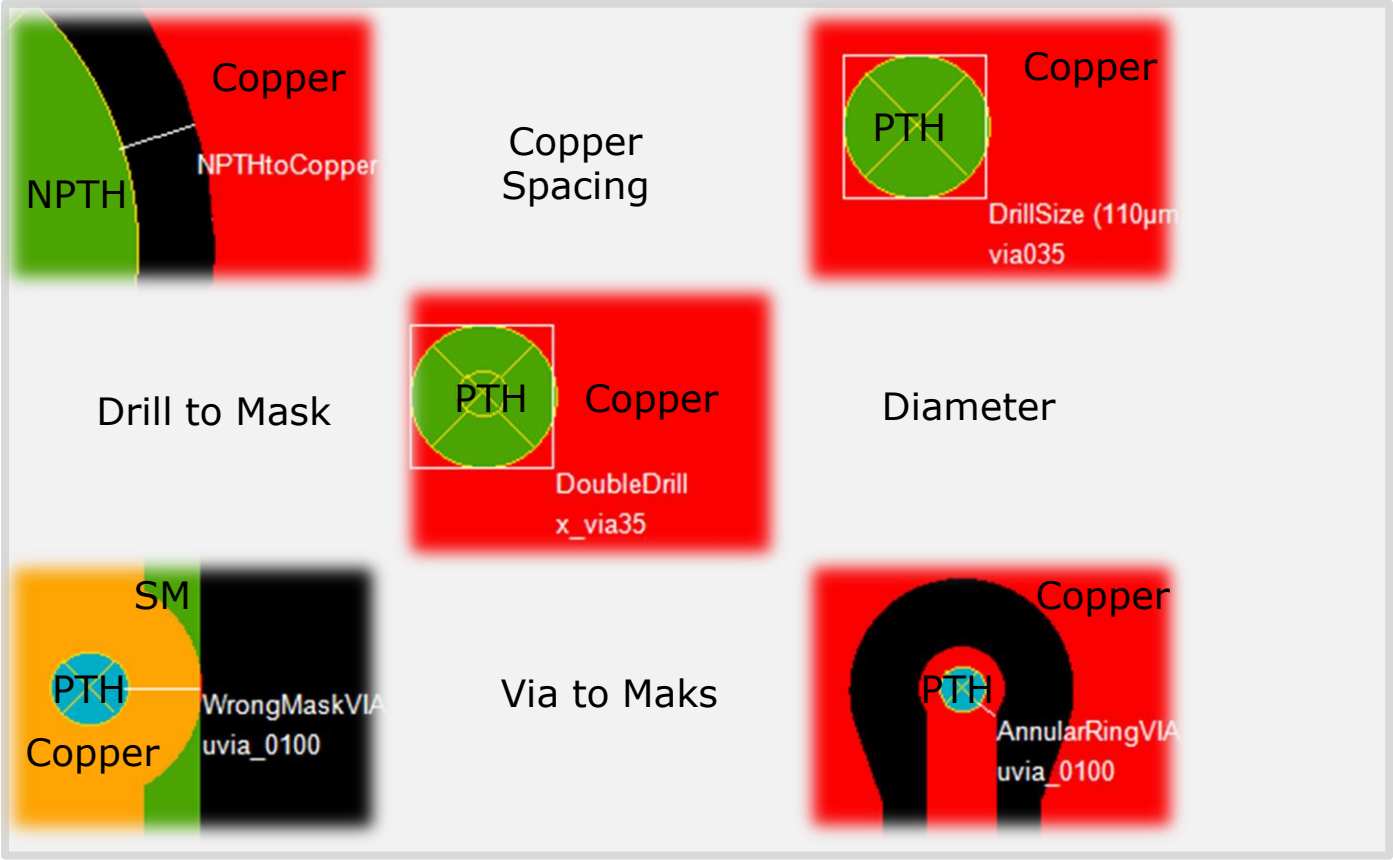
- DFM
- Data Formats
- Workflow
- Preparations
- Results
  - Copper
  - Mask
  - Drills
  - Components

DRC Solder Mask Analysis



DFM – Drill Check

DRC Drill Analysis for mechanical and laser drills



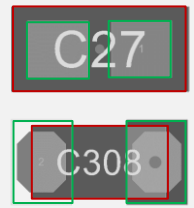
DFM – Component Check

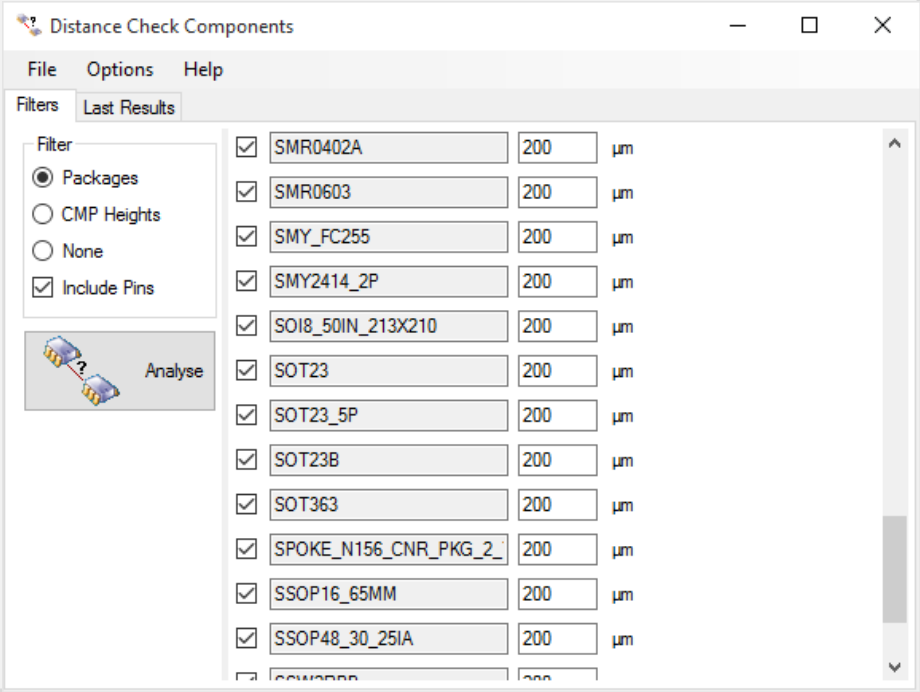
Spacing between components to test manufacturing constraints

Set-up the rules to find AOI, Pick & Place and Test Point issues

Depending on the Component design in the library, the values have to be adjusted.

Component Body
Component Pin



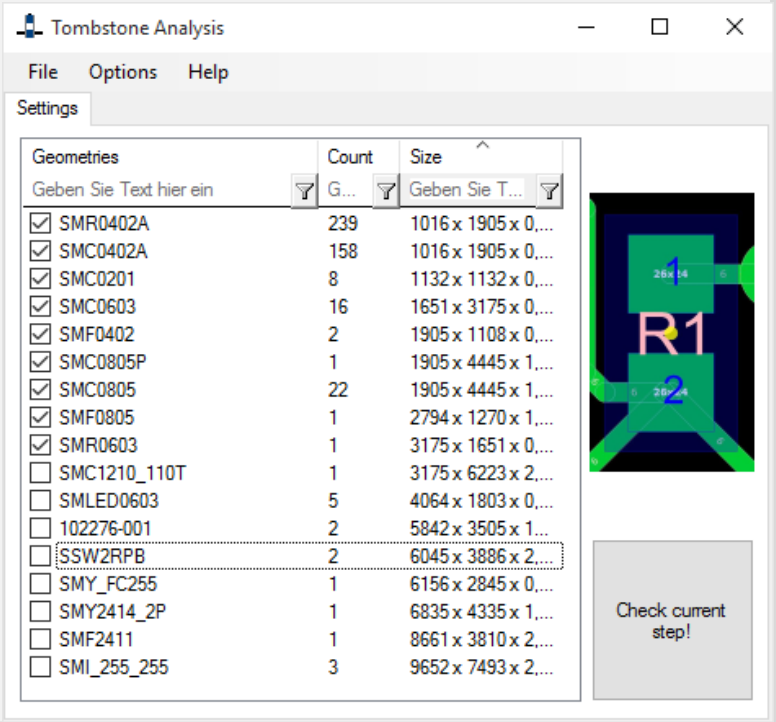


DFM – Component Check

Tombstone effect during soldering, caused by different pin connection

All 2 Pin components are checked for their signal connection

Possible hazard areas are small components with two pins. In cases where the difference from one pin connection to the other is too big, the component might be tipped upright during the soldering process.

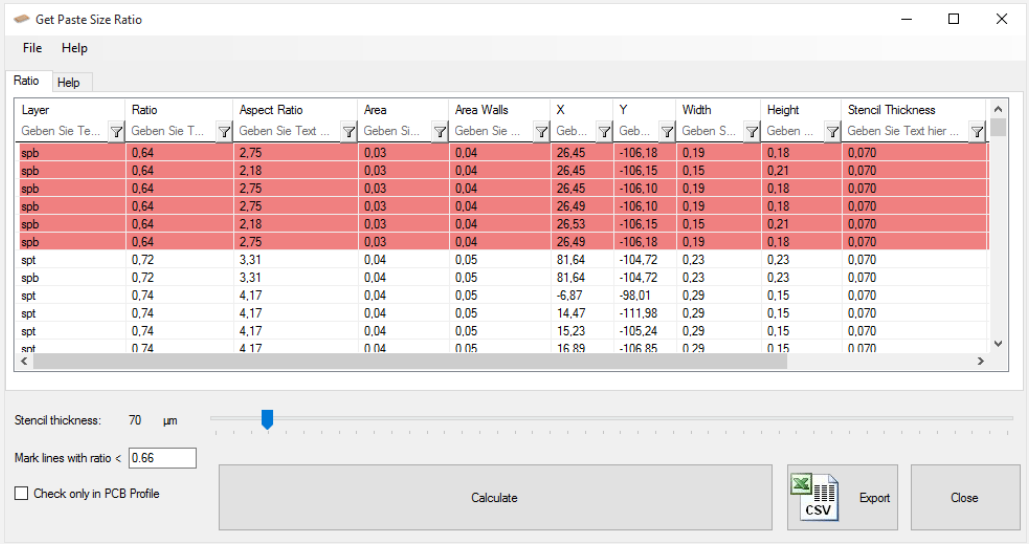


DFM – Component Check

Stencil optimization to reduce pad lifting regarding IPC7525

DRC for SMT soldering using a stencil

IPC Guidelines show how to define solder paste pads to ensure high quality during the process. To find Pads with a larger wall adhesive compared to the Pad surface, all Pads are calculated regarding the tested stencil thickness.



Layer	Ratio	Aspect Ratio	Area	Area Walls	X	Y	Width	Height	Stencil Thickness
spb	0,64	2,75	0,03	0,04	26,45	-106,18	0,19	0,18	0,070
spb	0,64	2,18	0,03	0,04	26,45	-106,15	0,15	0,21	0,070
spb	0,64	2,75	0,03	0,04	26,45	-106,10	0,19	0,18	0,070
spb	0,64	2,75	0,03	0,04	26,49	-106,10	0,19	0,18	0,070
spb	0,64	2,18	0,03	0,04	26,53	-106,15	0,15	0,21	0,070
spb	0,64	2,75	0,03	0,04	26,49	-106,18	0,19	0,18	0,070
spt	0,72	3,31	0,04	0,05	81,64	-104,72	0,23	0,23	0,070
spb	0,72	3,31	0,04	0,05	81,64	-104,72	0,23	0,23	0,070
spt	0,74	4,17	0,04	0,05	-6,87	-98,01	0,29	0,15	0,070
spt	0,74	4,17	0,04	0,05	14,47	-111,98	0,29	0,15	0,070
spt	0,74	4,17	0,04	0,05	15,23	-105,24	0,29	0,15	0,070
spt	0,74	4,17	0,04	0,05	16,89	-106,85	0,29	0,15	0,070

Software Portfolio

PCB-Investigator
www.pcb-investigator.com

Native Board Import (3D Interface to CATIA, SiemensNX, SolidWorks, SolidEdge)
www.sts-development.biz

GerberLogix
www.GerberLogix.com

Online Gerber Viewer
www.Gerber-Viewer.com

Software Development, CAD Converter, data connection
www.easyLogix.de

Get in touch,
info@easylogix.de
Guenther Schindler
Tel. +49 941 604 889 719