PCB-Investigator - Design Rule Check (DRC)

Check Result Explanations

12/2015

Schindler & Schill GmbH
Im Gewerbepark D33
93059 Regensburg
Deutschland

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

Introduction

- **Introduction**
- Copper
- Solder Mask
- Drills
- Components
- Conclusion

Content of this presentation

This presentation will explain, how to interpret the check results reported by PCB-Investigator 's **Design Rule Check (DRC)**.

The main focus is to illustrate the **technical background** and to give an understanding of the **unavoidable tolerances** during the PCB manufacturing process.

Although a lot of manufacturing limitations can be bypassed at prototype level, you will have to deal with them at **series production**.

At the end it's mostly a matter of **costs** and expected **failure rate**.

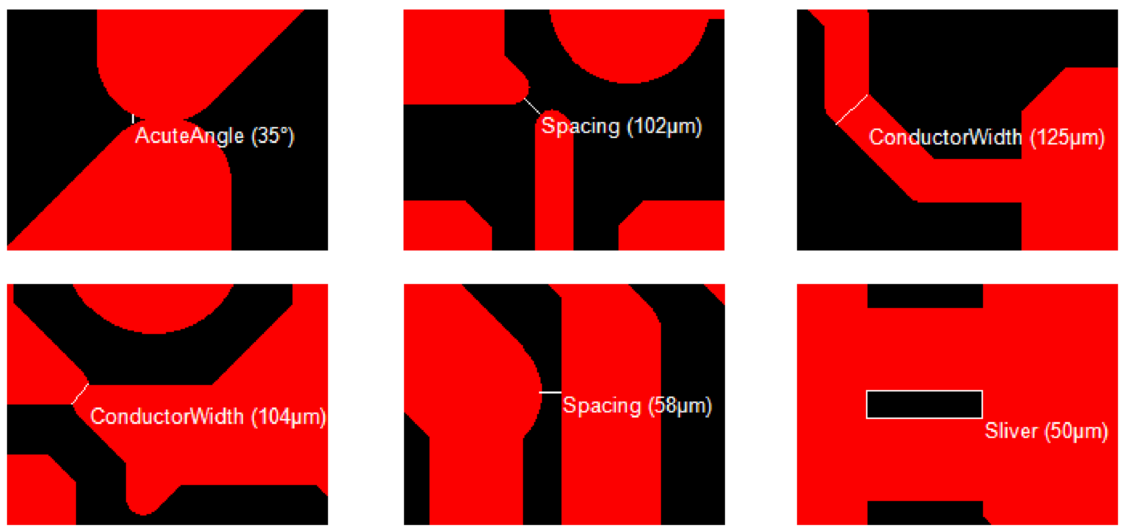
DRC – Copper Check

Bottlenecks and spacing

Small distances in copper, as well as thin copper areas might not be producible due to physical etching restrictions.

A chosen technology e.g. IPC Class II should be applied everywhere on the board, as already a single violation forces the PCB supplier to switch to finer production parameters (e.g. IPC Class I) for the whole board.

The finer the structures are, the more **expensive** the board will be.



DRC – Solder Mask Check (I)

- Introduction
- Copper
- **Solder Mask**
- Drills
- Components
- Conclusion

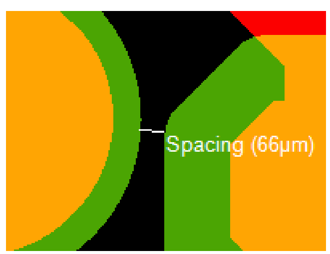
Solder resist webs and spacing

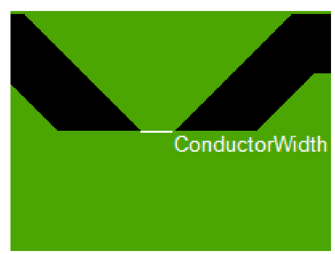
Solder resist webs with a width of less than app. 70µm are hardly producible with standard technology.

There is always the risk, that those small pieces will detach and adhere somewhere else, which can lead to **solder problems** and **failures**.

Smaller distances and webs might be only producible with an expensive special solder resists and less resist height, which influences the isolation quality.

To avoid unnecessary costs, PCB-Investigator reports all those violations.





DRC – Solder Mask Check (II)

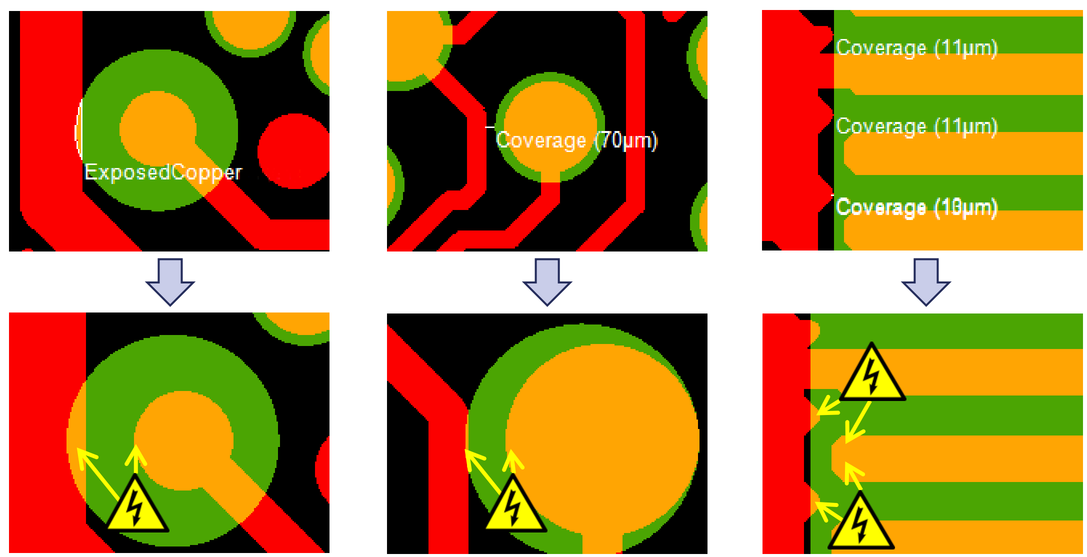
Coverages and exposures

As there will always be a slight displacement between the solder resist and the conductive pattern, surrounding copper should have a minimum distance of the allowed displacement from the solder resist opening. If not, there is the risk that the surrounding copper will be also exposed, which can lead to **electric shorts** by e.g. solder bridges.

The allowed displacement is app. 75µm, also 50µm is possible, but more expensive.

CAD data:

With allowed displacement:



DRC – Solder Mask Check (III)

- Introduction
- Copper
- **Solder Mask**
- Drills
- Components
- Conclusion

Soldering and testing

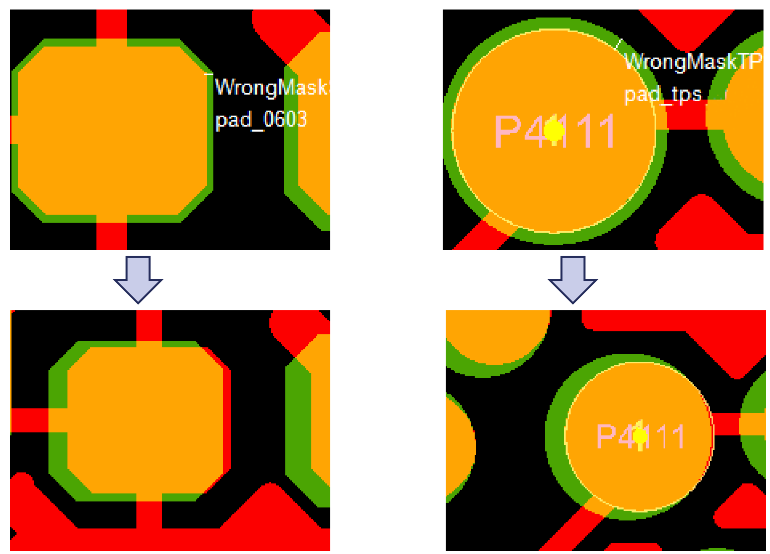
The displacement can also have negative impact on the **solderability** of SMD Pads or **testability** of test points.

To ensure the further processability, there should be a solder resist opening with an oversize of the maximum allowed displacement (e.g. 75µm) for all SMD pads and test points.

In this way the copper will always be completely solderable/testable.

CAD data:

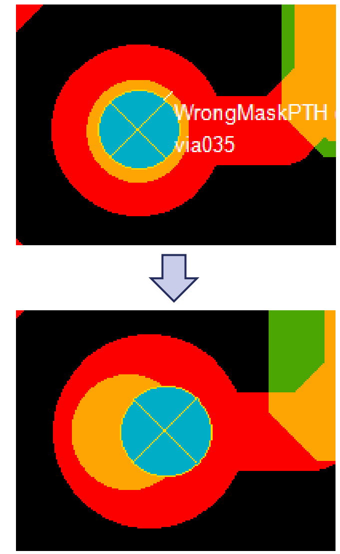
With allowed displacement:



DRC – Solder Mask Check (IV)

- Introduction
- Copper
- **Solder Mask**
- Drills
- Components
- Conclusion

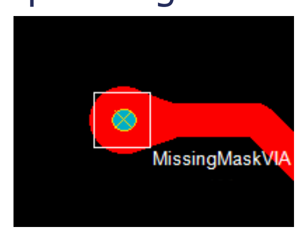
Drilling and cleanliness

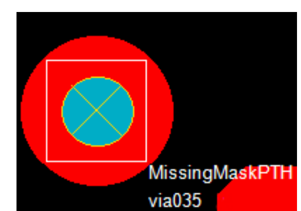


Each drill should have a solder resist opening which ensures, that the drill is free of solder resist despite the combined displacement of the solder resist and of the drill itself.

A partly covered drill, or solder resist in the drill sleeve, can detach and adhere somewhere else during the cleaning process. This can lead to **solder problems** and **failures**, and **contaminates** the chemical baths of the PCB supplier. It also affects the **EMC** behavior.

A completely covered drill without any solder resist opening on one side also cannot be cleaned and therefore contaminates the chemical baths. If covered on both sides, the enclosed air in the drill could break the solder resist cover when expanding due to heat. The result is unwanted **dirt** on the board.





DRC – Drill Check (I)

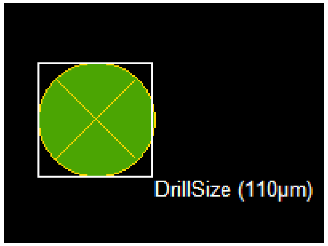
Drill size and distance

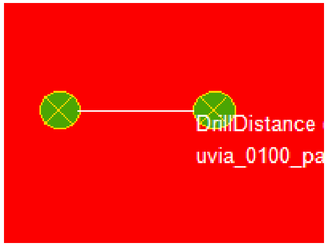
The drill diameter and the drill to drill distance are very important factors for the **price calculation**.

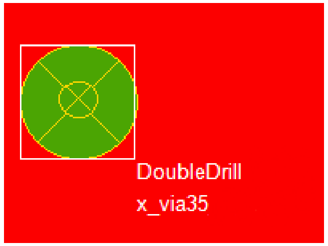
Very thin drills have a **short life period** and must be replaced often. They are also less long, which forces the PCB supplier to drill only 1-2 Panels at the same time instead of drilling e.g. 5 Panels in a package.

Small distances between two drills increase the risk, that the drill breaks or that the two holes are merged to one undefined shape.

Also two holes at the same location can lead to **broken drills** or an undefined hole shape.







DRC – Drill Check (II)

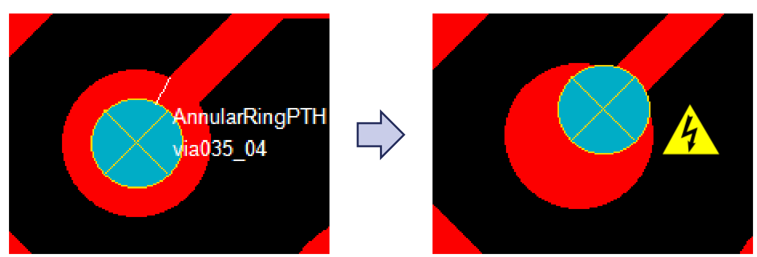
- Introduction
- Copper
- Solder Mask
- **Drills**
- Components
- Conclusion

Copper connection

To achieve a clearly defined connection between layers, the copper pads for the single drills must be large enough, so that a drill, displaced within the allowed tolerances, is still located completely within the pad.

If not, this can have a strong impact on the **EMC behavior** and can lead to **failures** due to a loose or broken contact.

Small annular rings and therefore narrow tolerances in the drilling process might still be producible, but force the PCB supplier to use high-end drilling machines and to drill only one Panel at the same time instead of e.g. 5 Panels in a package. This has a very strong influence on the **PCB costs**.



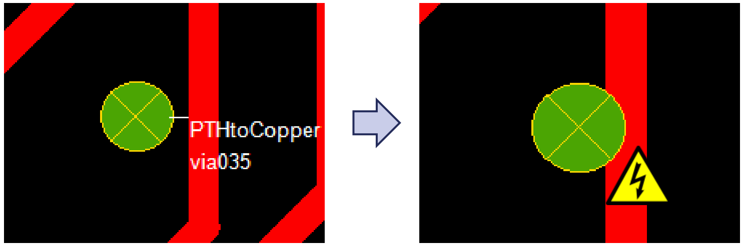
DRC – Drill Check (III)

- Introduction
- Copper
- Solder Mask
- **Drills**
- Components
- Conclusion

Distance to surrounding copper

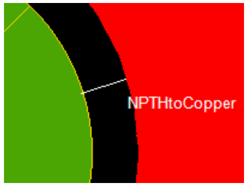
Here we must differ between plated and un-plated drills.

For plated drills, the distance to surrounding copper is important on inner layers, when the copper pad is omitted. Again due to the production tolerances, the displacement of the drill could lead to **broken connections** or **shorts**, if surrounding copper is too close.



For un-plated drills, the distance is needed for tenting the hole during the plating process to avoid copper in the hole. When the un-plated drill could not be securely tented, a second drill process after the plating process is needed instead.

This raises the costs enormously.



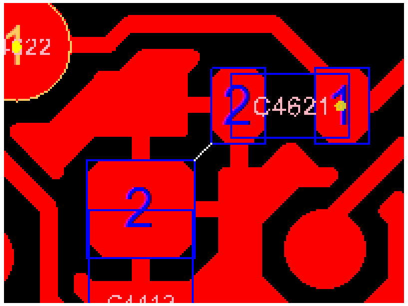
DRC – Component Check (I)

Distance between two components

For a reliable **pick and place** process, and to avoid **solder bridges**, a certain distance between two components must be maintained.

This distance depends on the height and size of the components, and partly also on the orientation of the components.

A closer distance might be possible, but has a negative influence on the First Pass Yield and the testability, especially in series production.



DRC – Component Check (II)

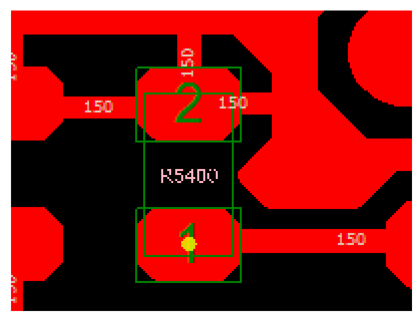
- Introduction
- Copper
- Solder Mask
- Drills
- **Components**
- Conclusion

Tombstone effect

When the copper connections of two pins of the same small component have an unequal strength (e.g. 150µm on one pin, and 450µm on the other pin), the heat after soldering is dissipating with a different speed.

As a result, the solder paste is cooled unevenly and there is a risk, that the component stands up due to the unbalanced forces. This is called "Tombstone effect".

To avoid this effect, the connections of both pins should be as similar as possible.



Conclusion

Summary

Performing the **Design Rule Check (DRC)** of PCB-Investigator is the first step to **avoid unneeded costs** and to **increase the reliability** of your Printed Circuit Board.

Although in some cases the standard rules must be violated to fulfill some requirements (e.g. Space requirements), there is always a **potential to save money** and increase the reliability with a few minor layout changes.

For more questions about our DRC, please contact us!

Software Portfolio

Useful Links:

PCB-Investigator
www.pcb-investigator.com

PCBi-Physics
www.PCBi-Physics.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