



Published on *Online Documentation for Altium Products* (<https://www.altium.com/documentation>)

[主页](#) > Silk To Solder Mask Clearance

Using Altium Documentation

Modified by by Admin on Apr 11, 2017

Rule category: [Manufacturing](#)

Rule classification: Binary

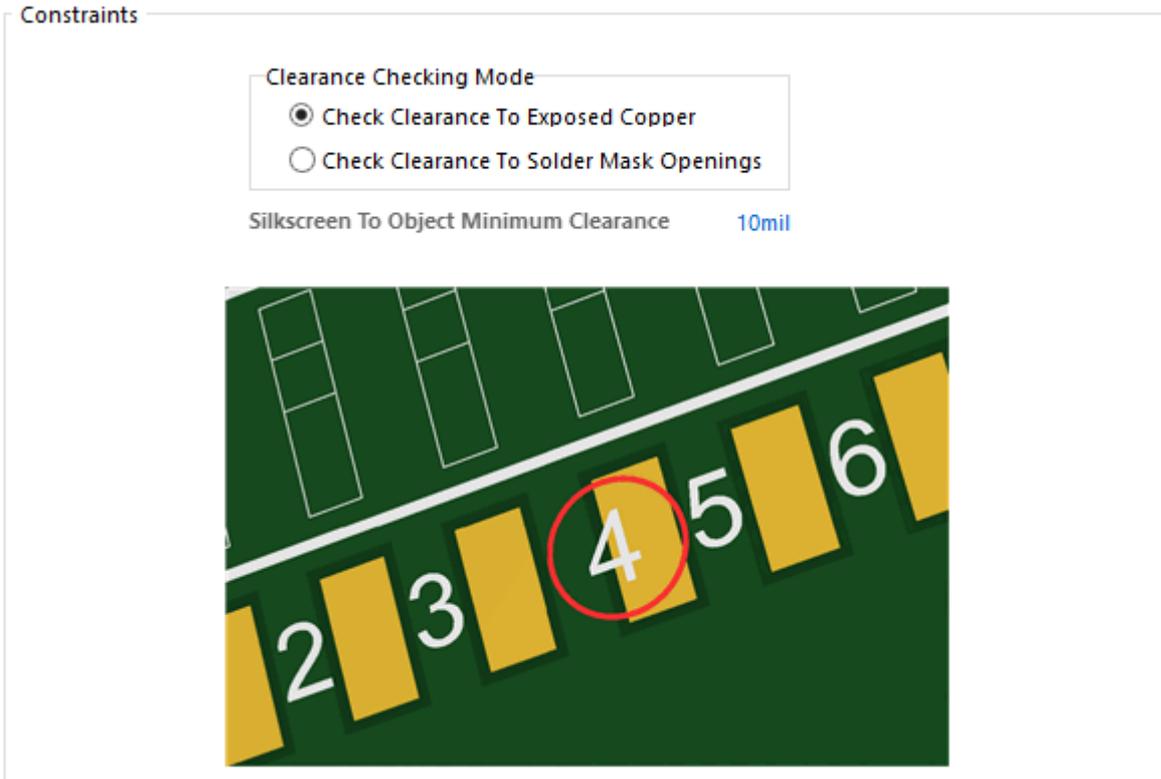
Summary

This rule checks the clearance between any silkscreen primitive and any solder mask primitive, or exposed copper-layer primitive (exposed through openings in the solder mask). The check ensures that the distance is equal to, or greater than, the value specified in the constraint.

Many manufacturers routinely strip (or 'clip') silkscreen to the mask opening and not just to the copper pad. However, doing so can render silkscreen text unreadable. Being able to catch such occurrences through DRC, allows you to manipulate offending silkscreen text prior to sending the board to manufacturing.

This design rule replaces the Silkscreen Over Component Pads rule found in previous releases of Altium Designer prior to Altium Designer 13.0. When loading a PCB document from such an earlier release, any defined Silkscreen Over Component Pads rules will automatically be converted to Silk To Solder Mask Clearance rules, with their scopes and constraints set to match legacy behavior. It is advised that you check your rule scopes and associated constraints to ensure accuracy in relation to design requirements.

Constraints



Default constraints for the Silk To Solder Mask Clearance rule.

- **Clearance Checking Mode** - choose a checking mode for the clearance:
 - **Check Clearance To Exposed Copper** - in this mode, clearance checking is between silkscreen (Top/Bottom Overlay layer) objects, and copper in component pads which is exposed through openings in the solder mask.
 - **Check Clearance To Solder Mask Openings** - in this mode, clearance checking is between silkscreen (Top/Bottom Overlay layer) objects, and solder mask openings created by objects that include a solder mask, such as pads, vias, or copper objects with the **Solder Mask Expansion** option enabled.
- **Silkscreen To Object Minimum Clearance** - specifies the minimum permissible clearance between a silkscreen object and either exposed copper, or solder mask openings, depending on the clearance checking mode chosen.

To match the legacy behavior of the old Silkscreen Over Component Pads rule, found in releases of the software prior to Altium Designer 13.0, the Silk To Solder Mask Clearance rule should have its **Clearance Checking Mode** set to **Check Clearance To Exposed Copper**, and the full query for one of its rule scopes set to **IsPad**. As mentioned previously, this is handled automatically when opening older designs.

How Duplicate Rule Contentions are Resolved

All rules are resolved by the priority setting. The system goes through the rules from highest to lowest priority and picks the first one whose scope expressions match the object(s) being checked.

Rule Application

Online DRC and Batch DRC.

Source URL:

[https://www.altium.com/documentation/cn/display/ADES/PCB_Dlg-SilkToSolderMaskClearance_Frame\(\(Silk+To+Solder+Mask+Clearance\)\)_AD](https://www.altium.com/documentation/cn/display/ADES/PCB_Dlg-SilkToSolderMaskClearance_Frame((Silk+To+Solder+Mask+Clearance))_AD)