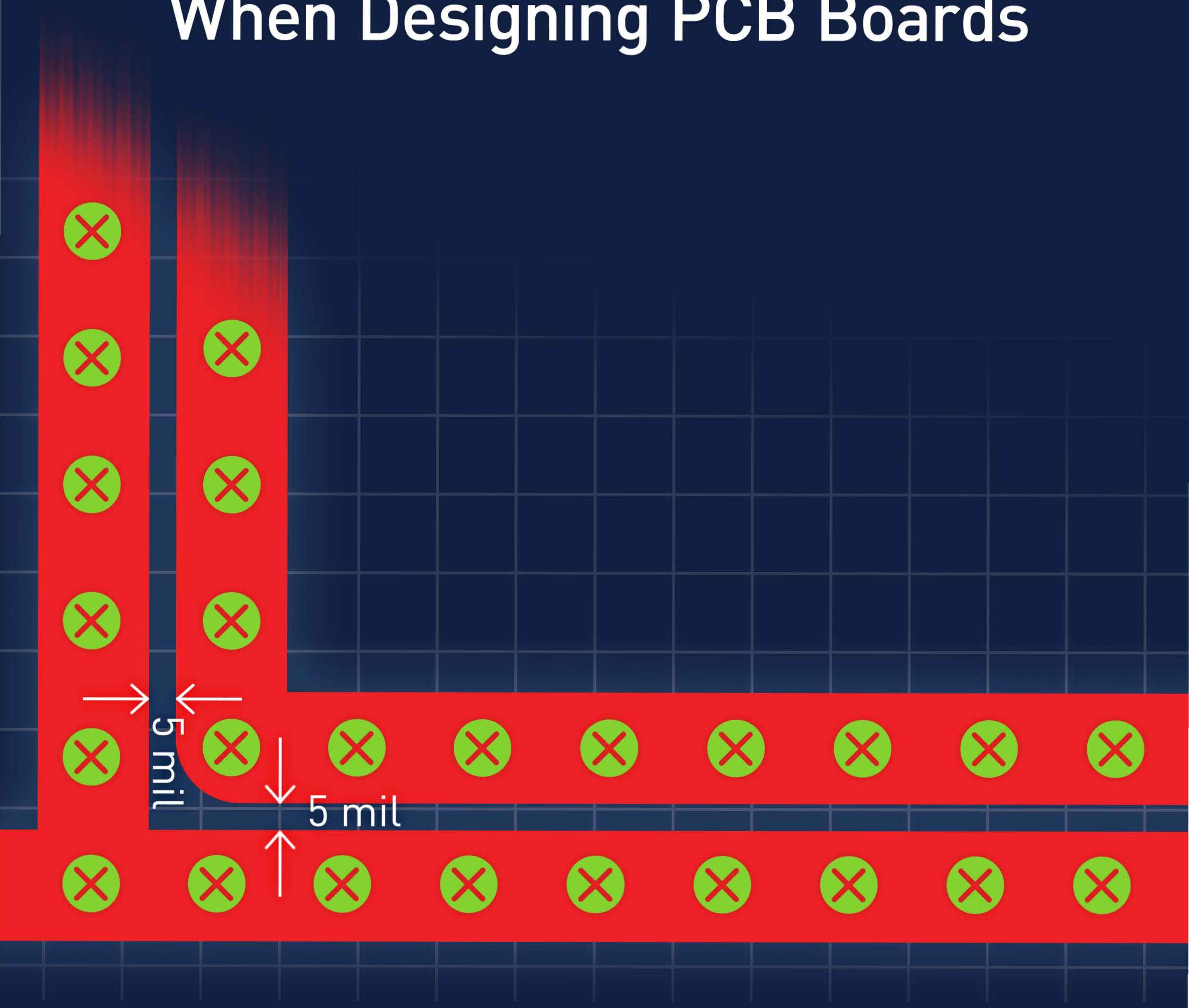
Implement Same Net Clearance Rules for Greater Flexibility When Designing PCB Boards



Steve Tran

Applications Engineer

Designers now have the ability to set specific parameters between same net objects when placing primitives on a board. Restricting space allows you to maintain clearance for primitives, and restricting primitives of the same net to a set clearance ultimately allows greater flexibility when forcing electrical primitives to interact with other electrical objects in a specific manner.

INTRODUCTION

Clearance rules set requirement constraints that define the minimum distance allowed between two objects; this is especially important for placing primitives on boards. The distance between the objects placed on the PCB is dictated by the clearance rules and, in most cases, are used to specify the distance between two differing nets to prevent short circuits. However, in some occurrences, a designer would want to be able to set a required clearance between primitives of the same net. While new functionality allows same net clearance checks between electrical objects, it excludes touching same net objects.

Setting the Distance

The ability to set the required clearance distance between two object primitives gives you the flexibility to force electrical primitives to interact with other electrical objects in a specific manner. Placement can be made in specific locations relative to the placement of other primitives.

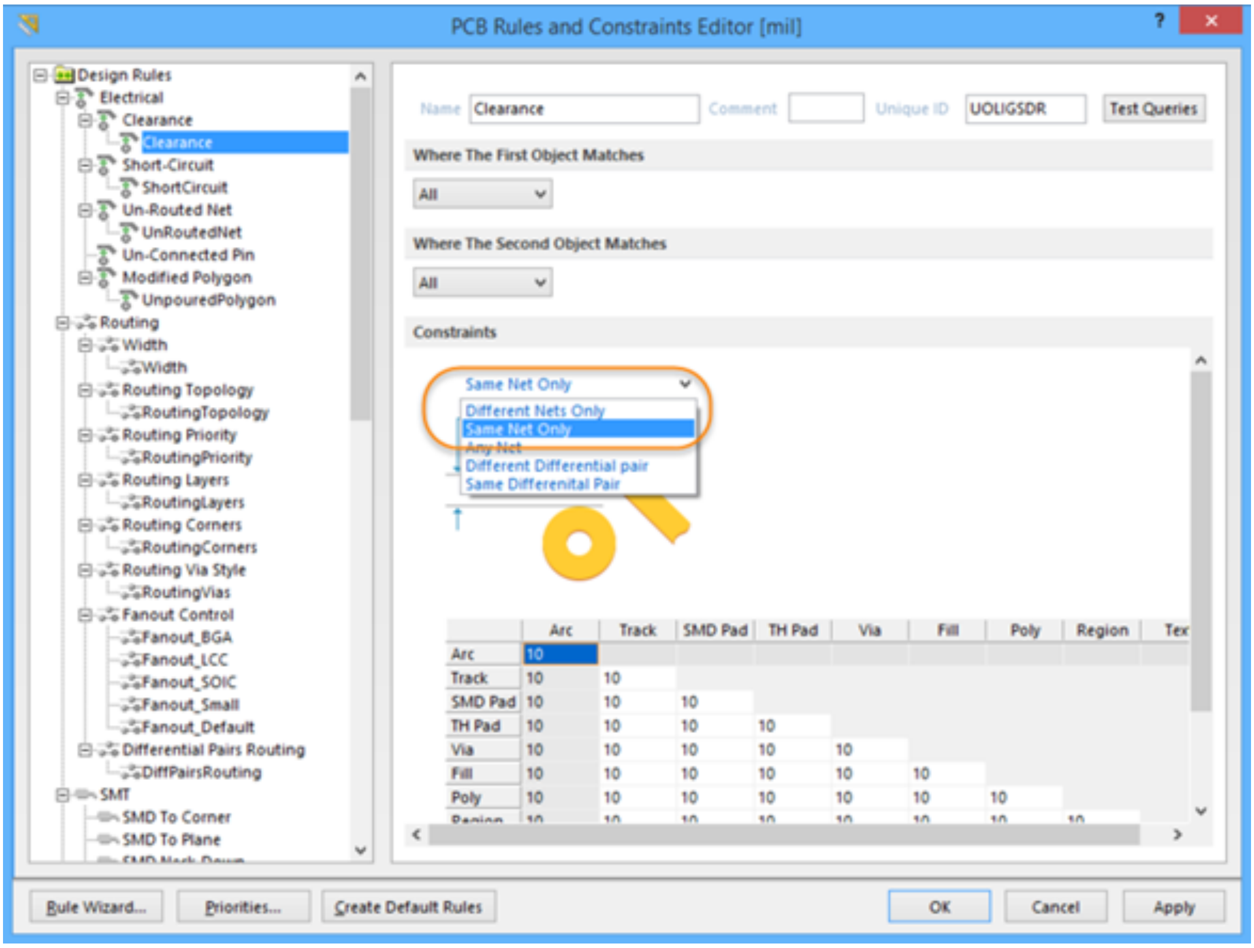


Figure 1: Setting clearance values in the PCB Rules and Constraints Editor.

These requirements are set through the design rules; By changing the clearance rules to only target objects that have the same net, you would be able to restrict primitive placement as well as check for any violations of objects being too close.

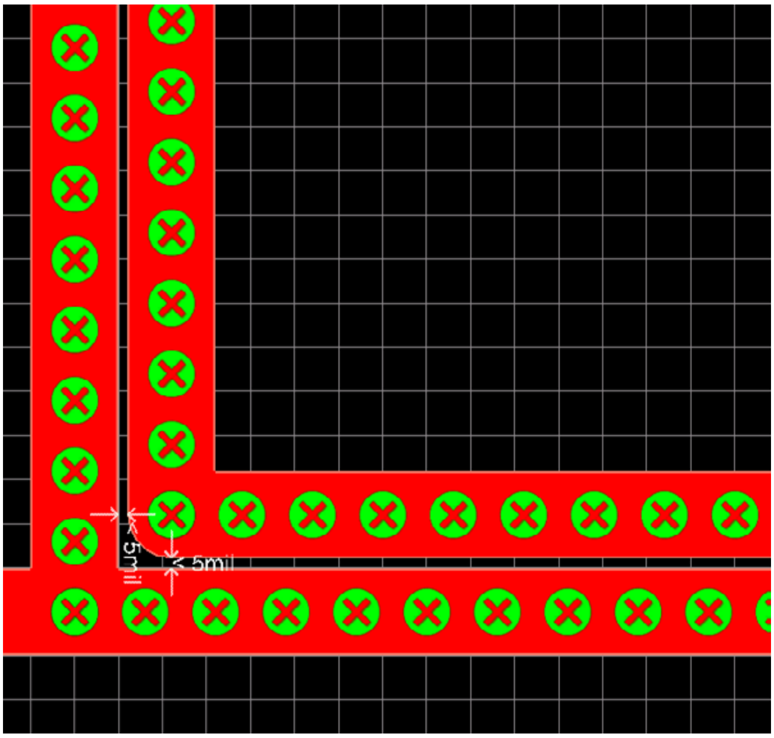


Figure 2: Checking for violations.

CONCLUSION

The ability to set a specific distance and enforce this requirement to the board is of particular importance for designers as it allows them to maintain clearance for primitives. Restricting primitives of the same net to a set clearance allows the designer to have the same rule applied to any object as compared to just different net objects; this allows for larger flexibility when creating a board.

REFERENCE LINKS

https://techdocs.altium.com/display/ADRR/PCB_Dlg-ClearanceRule_Frame((Clearance))_AD