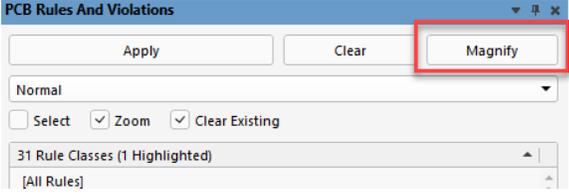# "DRC Revisited Part 1" Q & A

Note: Answers/Responses are based on Altium Designer 19.1.6

Comment/Question	Answer/Response
How is 'Height' rule different than "Component Clearance" rule in the height dimension?	The Height rule is a unary rule, meaning that the rule is specifically applied to the primitive itself. The default rule is 1000 mils (or 1 inch). If the 3D representation or the height provided to the component in its properties is over 1 inch, it will be flagged as an error. The Component Clearance rule is a binary rule, meaning the rule must perform a measurement between two primitives. In this case, the rule would be applied to two adjacent components. Note that the Component Clearance does not perform any copper checks. This is handled under the general clearance rule.
How about exporting and importing rules from different years?	Importing rules from prior versions of AD should generally not be an issue. It is rare for Altium to change the underlying structure of a rule. They may display it differently, but it should still import without issue. If Altium has added a rule to the latest version, importing from a prior rule set should not have any impact on the new rule. Conversely, there may compatibility issues if trying to import from the current version to a prior version if new rules are exported that were not available in the prior version. In this situation, it should be possible to successfully import as long as the new rule is not exported.

How would you recommend using the Component Clearance rules? I use a box surrounding the footprint on the Place Outline Layer. I want to be able to place these boxes overlapping for the tightest allowed component placement and set the rule to 0 mils (or even -5mils, say, for a 10 mil placement outline) for clearance. However, Altium seems to ignore component clearance rule errors when the place outlines completely overlap (rather than just touch), even with a positive clearance setting. Any recommendations? How do you set up a rule to keep two 3D bodies from causing DRC errors?	Though this will be further covered in the follow up webinar, the issue has to do with the fact that the component clearance rule is applied to the component depending on the existence of a 3D representation. If the component has a STEP, Parasolid or 3D body associated to it, the rule will evaluate the component based on the 3D representation only. If no 3D representation is added to the footprint, Altium will use a 'bounding rectangle'. This is defined as the smallest box Altium will need to draw around all the component's primitives on the copper, silk, and mechanical layers. The only exception to this bounding box is the designator on the silk layer. Please refer to the following feature request on Altium.com: #7465: Add "3D body only" option to Component Clearance Design Rule https://bugcrunch.live.altium.com/#Idea/7465
How do you recommend handling components that violate your rules? i.e. rules are 10 mil clearance and a component that has 8 mil space between pads?	Though this will be covered in the next webinar, the short answer: Create a rule specific to the component for the pad clearances. If the component is a BGA, a "regional rule" can be created to allow more board real estate for escape routing.
Some manufactures will update files and not give feedback to the designer. I've heard of times where the fabricator will claim what they did is proprietary to them. Is there anything you can do in that situation?	This happens ALL the time! They want you to be successful and that is why they will put the time and effort into cleaning up the design; but they do not want you to move to other fabricators with their artwork. Two solutions – 1) You and/or your organization start handling the cleanup of the manufacturing documentation and take responsibility for it with explicit request to design "as is." 2) Negotiate with fabricator. They may be willing to hand you the final artwork for an additional fee.

How do you control the level of zoom used when double clicking on an error? When you click on a violation in the list after running DRC, how can you set how close the view will zoom to find the exact error? Sometimes, I can't get it to zoom in enough to figure out precisely where is the error. How do you change the zoom level when you click on an error in the DRC report	There are several ways to zoom into a violation. In some cases, the level of zoom is handled in PCB Rules and Violations panel through the Magnify settings. The following is a list of features which zoom into DRC errors and whether the magnify feature is applicable: • PCB Rule and Violations panel. This feature is dependent upon the magnify settings within the panel • Design Rule Verification Report (generated after running the Tools » Design Rule Check). This feature is dependent upon the magnify settings in the PCB Rules and Violations panel. • Message panel (generated after running the Tools » Design Rule Check). This feature is dependent upon the magnify settings in the PCB Rules and Violations panel. • Jump feature in Violation Details dialog box. (Right clicking on the violation and selecting Violations in the menu). This feature is independent of the magnify settings in the PCB Rules and Violations panel.
For primitive unary and binary rules, would changing those override related rules such as net rules?	A rule that is applied to a primitive is based upon the query statement and the priority of the rule if there are two or more rules. This will be discussed and demonstrated further in the next webinar.

| Is there a record of all the waiver notes? What happens to the waiver if the violation is fixed, if the violation is waived, then fixed, then reintroduced will it still be waived? | When it comes to the DRC waivers summary, the Tools » Design Rule Check.. results (a.k.a. the Design Rule Verification Report) and the PCB Rules and Violation Panel will list the violations that have been waived within the editor. In the OutJobs under the validation outputs, the resulting report will list the waivers, if they exist.

If the violation is fixed due to a rule change, the waiver will be removed since the rule change remedied the issue.

If for some reason, the rule is changed (again) which reintroduces the violation, the prior waivers will not be restored. |
|---|---|
| Why do waived drc's come back after as errors after they are waived? | Waivers will not be changed to violations, unless the user deselects the waiver. If the primitive has multiple violations, the overriding color will be the violation color, not the waiver color. |
| From Tools -> DRC, can you explain exactly the difference between 'online' and 'batch'? | The online rules are those rules that will be reviewed as one goes about editing. For example, when routing, the rules are being checked to prevent the user from drawing traces that will result in a violation. The batch DRC is run to ensure that rules that may not be exercised during the editing of the layout are checked. Note that many of the rules are listed as both online and batch.

Running the batch when the layout is complete is highly advised. All rules that have been established are exercised against the board. This will also create a formalized violations list if errors are determined. This can happen if rules are changed as the design progressed. |
| Is it possible to import/export a single rule? For example, export/import a single clearance rule and not the whole clearance rules | The granularity of the export/import is at the rule category. If there are two or more rules associated to that category, all the rules will be exported. For example, if the Clearance rule is to be exported/imported, all rules established under the clearance category will be exported/imported. |