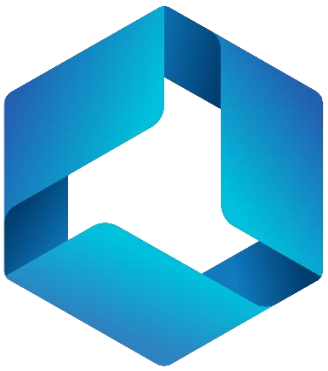

ALTIUM 365

# Altium Designer

## Essentials Course - Altium 365

### Module 20: PCB Design Rules Creation

# Table of Contents

# Module 20: PCB Design Rules Creation

## 1.1 Purpose

In this exercise, you will learn various methods to create PCB design rules. Rule priorities will also be defined, to ensure proper rule application on objects targeted by multiple rules.

Design rules are used to ensure the PCB layout meets the specified design requirements and highlights violations when items fall outside of their defined constraints. These rules cover various aspects of the design and collectively form an “instruction set” for the PCB editor to follow. Many of them can also be monitored in real-time by the online Design Rule Checker (DRC).

## 1.2 Shortcuts

Shortcuts when working with Module 20: PCB Design Rules Creation

| | |
|---|---|
| **D » I:** | PCB Import Changes from Project |
| **D » R:** | PCB Design Rules |
| **D » C:** | PCB Design Classes |
| **R:** | PCB Component Pushing mode |
| **D » U:** | SCH Update PCB Document |
| **Shift+F:** | Find Similar Objects |
| **Shift+C:** | Clear masking |
| **Ctrl+S:** | Save Document |
| **Ctrl+Q:** | Toggle Units |

# 1.3 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project `Module 20 PCB Design Rules Creation.`
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project `Module 20 PCB Design Rules Creation (Top\Projects\Altium Designer Essentials Training Course\...).`
5. Select **Open Project as Copy…** 

.
6. At the new dialog *Create Project Copy:*
   a) Add your name to the project: `Module 20 PCB Design Rules Creation – [Your Name].`
   b) Add a description: `Altium Essential Training – [Your Name].`
   c) Open the *Advanced* section.
   d) Select the Ellipsis Button  from the **Folder** configuration to open the *Choose Folder* Dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your Name]`
      ii) Select **OK.**
   e) Change the Local Storage path if needed.
   f) Select **OK** to create the copy.
7. Wait until Altium Designer created the copy of the project and opened the project for you at the *Projects* panel, this may take up to 1 minute.

For details how to Copy / Clone the predefined training project see Module 8 Making the Connection, Step 1.3 Preparation.

## 1.3.1 ECO

8. From the *Projects* panel, open the `Module 20 PCB Design Rules Creation.PcbDoc`.
9. Go to **Design » Import Changes From [Project Name].PrjPcb** and execute the ECO.

# 1.4 PCB Rules and Constraints Editor

## 1.4.1 Clearance Rule for 12V

10. Go to **Design » Rules…** (**D » R**), the *PCB Rules and Constraints Editor* dialog will open.
11. The tree on the left lists the rule categories. Expand the *Electrical* rule category. Notice that there are 6 rule types within this category. This can be seen in Figure 1.
12. Expand the *Clearance* rule type to display all clearance rules currently defined.
    a) Notice that there are two rules.
    b) The first is called `Schematic Width Constraint`, this rule was automatically created based on the Directives placed at the schematics.

During schematic creation - Module 12 Schematic Updating - we placed directives that created a `Power` NetClass, similar we prepared a CAN_Bus directive for the exercise.

    c) The second rule, the default rule, is called *Clearance*.
    d) Select the second rule to view the rule definition on the right of the window as shown in Figure 1.

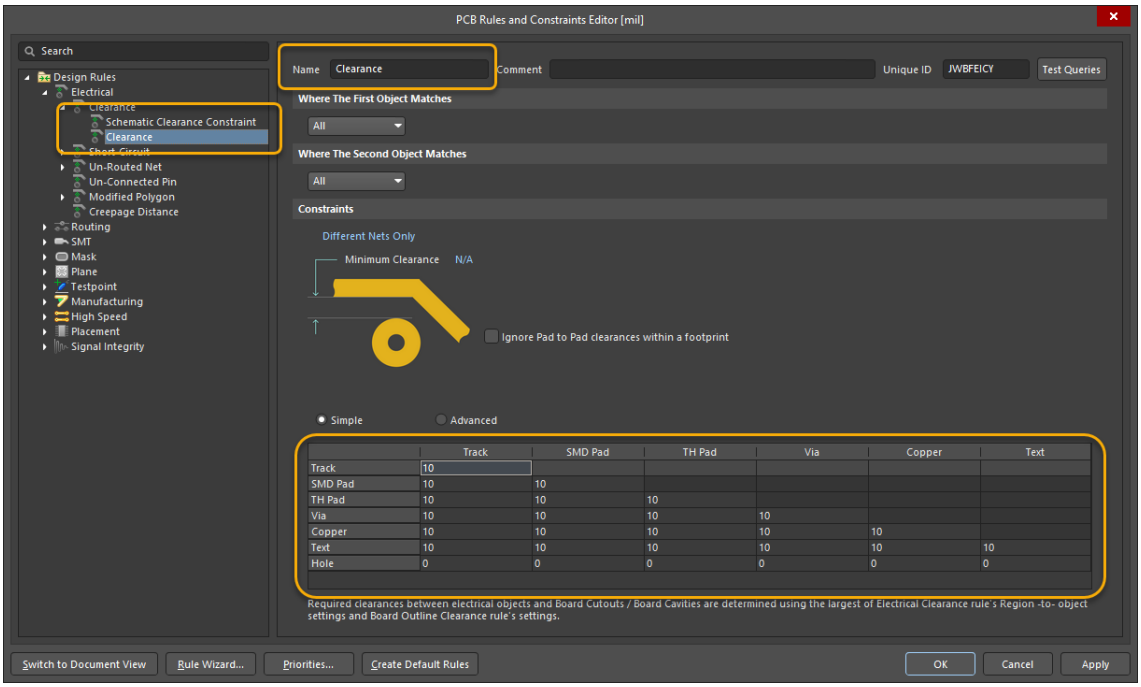


*Figure 1. PCB Rules and Constraints Editor*

13. First, we will rename a rule.
    a) Select the *Schematic Clearance Constraint* rule.
    b) Change the Name to `CAN_BUS`.

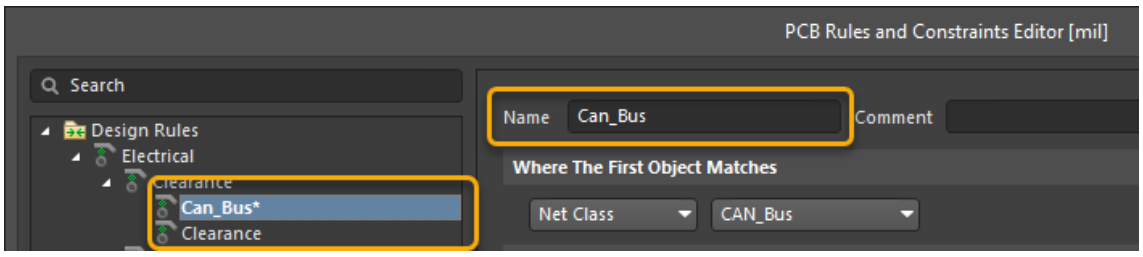


*Figure 2. CAN_BUS Clearance Rule*

14. We will now create a new rule which will target the 12V power net. Right-click on the *Clearance* rule type, just underneath *Electrical* and click **New Rule…,** Figure 3.

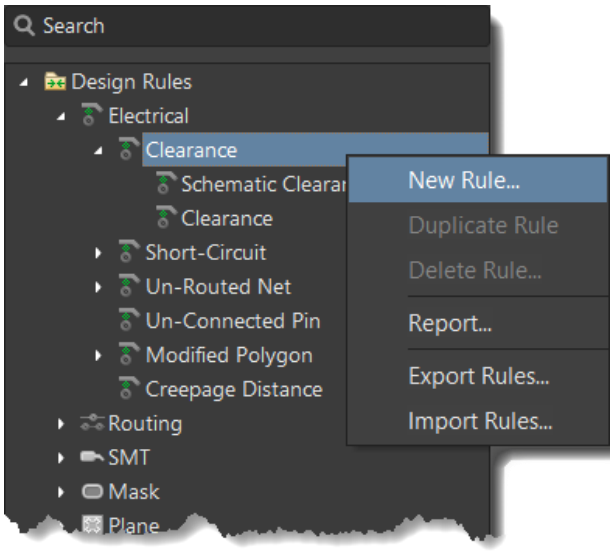


*Figure 3. Create a new rule*

15. You should notice a new rule added with a default name of `Clearance_1`.
16. Select the new rule `Clearance_1` on the left pane to open the rule.
17. Assign each field according to Figure 4 below.
    a) Set the *Name* field to `PowerNets12V`.
    b) Set the first query *Where the First Objects Matches* to `Net`, and then `12V`.
    c) Set the *Minimum Clearance* constraint to `15mil`.
    d) Select **Apply** to save the modification.

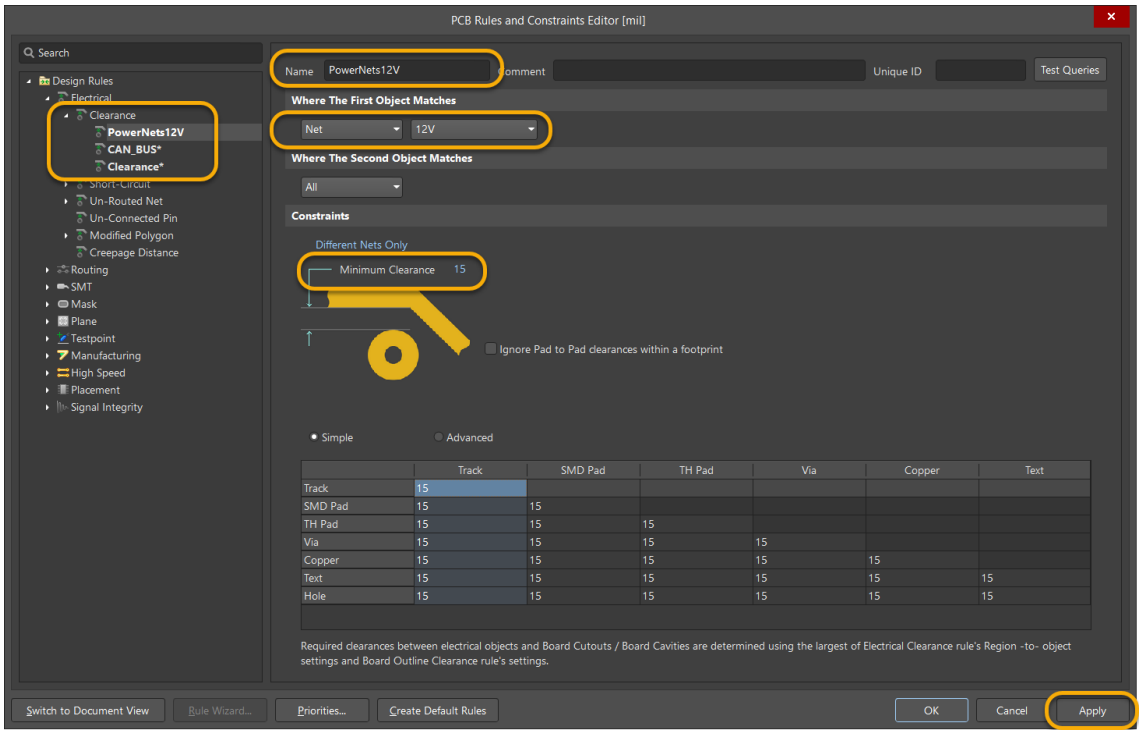


*Figure 4. PowerNets clearance rule definition*

## 1.4.2 **Rule Priorities**

18. Now that we have multiple clearance rules, let's ensure their priorities are set correctly. Click on the **Priorities…** button in the bottom left of the window to open the *Edit Rule Priorities* window.
19. Ensure that the newly added rule's priority is number **1** as shown in Figure 5 below. If it's not, select that rule and click **Increase Priority**.
20. Click **Close** and then **OK** to accept changes and close all dialogs.

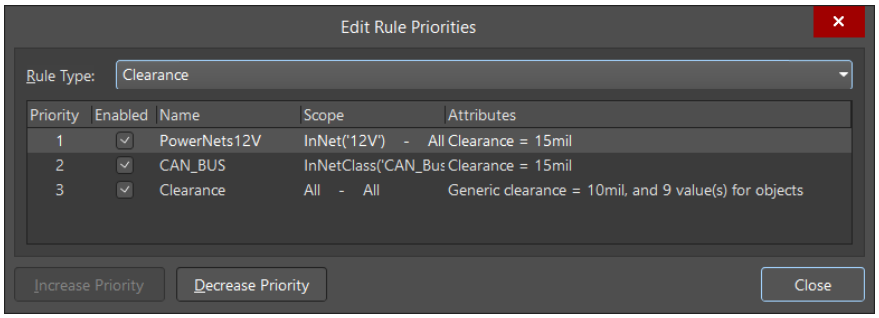


*Figure 5. Rule Priorities window*

To simplify the process of defining and managing rules, the idea is to first define the general rules that cover broad requirements for the PCB. Then, override these rules with specific rules in specific situations. Thus, it is a general practice to make the generic or "All-All" rule the least prioritised so that objects are applied with their specific rules.

## 1.4.3 **Design Rule Using Net Classes**

Instead of scoping objects of a single net in Design Rules, we can target objects from multiple nets by grouping them in Net Classes. We will now add a rule, scoping a Net Class, and assign a width to this Net Class so fore to all members from that class.

During schematic creation - Module 12 Schematic Updating - we placed directives that created a `Power` NetClass

21. Still at the *PCB Rules and Constraints Editor,* section *Electrical – Clearance*. Right-click on the *Clearance* rule type, just underneath *Electrical* and click **New Rule…**. You should notice a new rule added with a default name of `Clearance_1`.
22. Select the new rule `Clearance_1` on the left pane to open the rule.
23. Assign each field according to Figure 6 below.
    a) Set the *Name* field to `PowerNetsClass`.
    b) Set the first query *Where the First Objects Matches* to `Net Class`, and then `Power`.
    c) Set the *Minimum Clearance* constraint to `15mil`.
    d) Keep the second query as `All`.
24. Select **Apply** to save the modification.

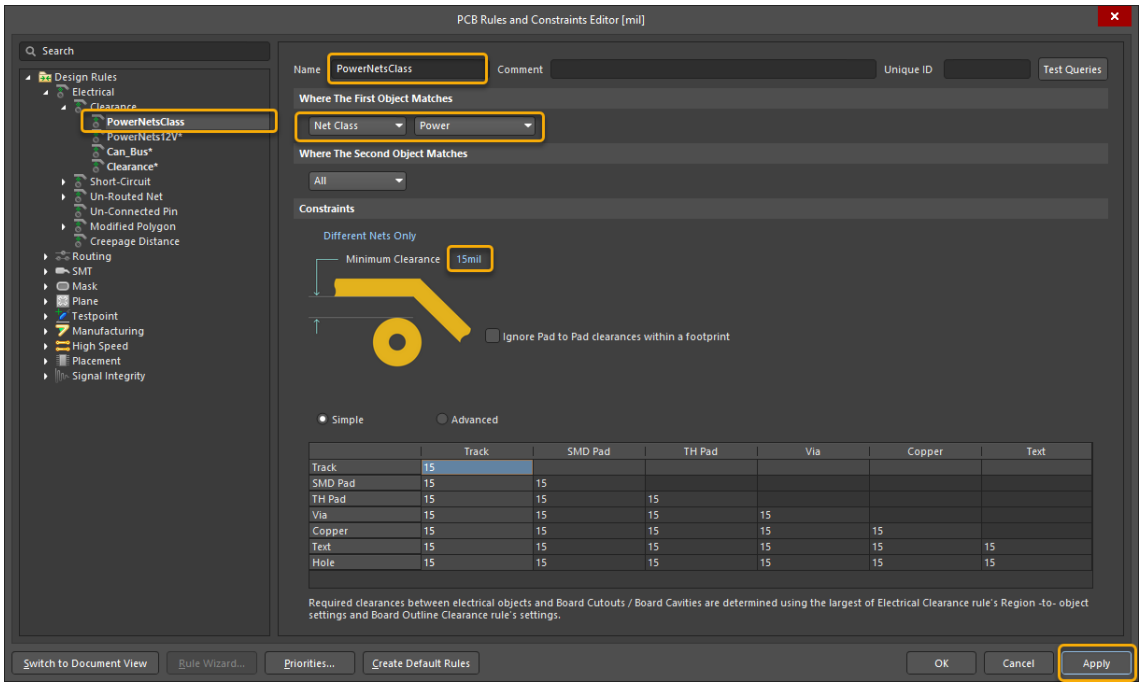


*Figure 6. Creating a PowerNets Net Class*

If you add a new net class directly from the PCB (**Design » Classes**), they will not be present on the schematic. When running the Engineering Change Order, you may be prompted of a possible removal of the net class. Simply disable the checkbox to prevent the removal within, or edit the **Project » Project Options » ECO** generation tab to prevent any removal of net classes.

## 1.4.4 **Schematic Driven Rule**

During schematic creation - Module 12 Schematic Updating - we placed directives that created a `Power` NetClass and a Width rule. We will now check the width rule for the *Power* Net Class.

25. Still in *PCB Rules and Constraints Editor,* navigate to the **Routing » Width** design rule.
26. Check the settings for the existing Schematic Width Constraint rules.
27. Rename the rules, based on the Nets / Classes the rules target, Figure 7.

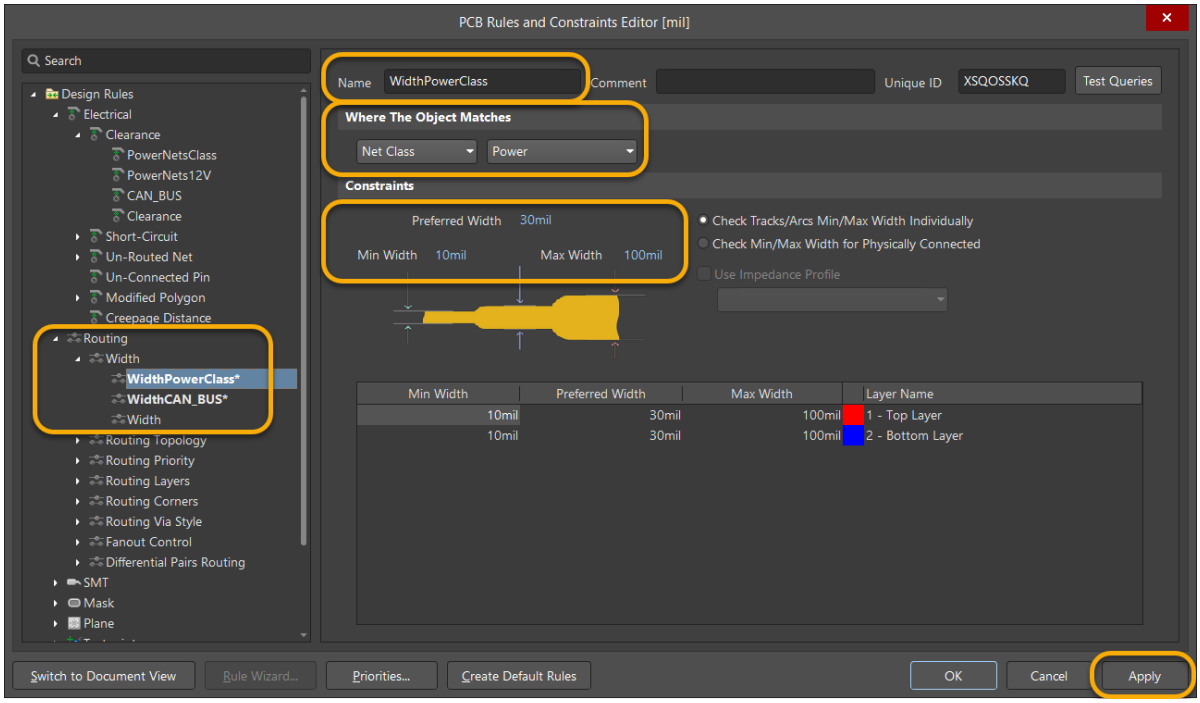


*Figure 7. PowerNets Width Rule definition*

28. Once again, ensure that the schematic driven *Schematic Width Constraint* rules have a higher priority at the default Width rule, as described at Section 1.4.2 Rule Priorities

## 1.4.5 **Vias**

### 1.4.5.1 Setting Via Sizes

To have multiple via sizes available during interactive routing, we'll modify the default *Routing Via Style* Design Rule.

29. Open the *RoutingVias* design rule in **Design » Rules**…, and head to the *Routing* **-** *Routing Via Style* rule category.
    a) Select the rule and modify the *Via Diameter* and *Via Hole Size* values as shown in Figure 8. Note that the Maximum values are in the middle.

| | | Diameter | Hole Size values |
|---|---|---|---|
| i) | Minimum : | 25mil | 10mil |
| ii) | Preferred : | 30mil | 15mil |
| iii) | Max: | 40 mil | 20mil |

    b) Click **OK** to accept these changes and close the dialog.

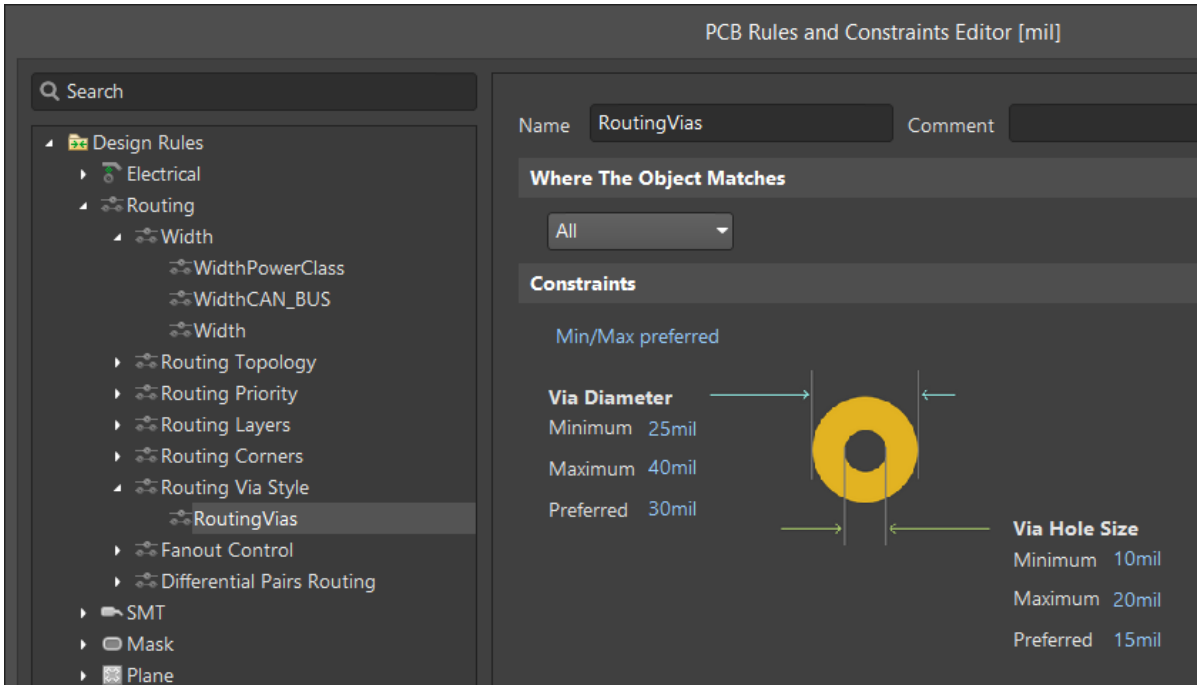


*Figure 8. Changing the Routing Via Style design rule*

If needed we could create a new RoutingVia rule specifically for Power Nets, similar to what we did for the Clearance rule.

30. Click **OK** to save and close *PCB Rules and Constraints Editor*.

# 1.5 Component Clearance Design Rule

In Altium Designer, the Component Clearance Rule specifies the minimum allowable distance between components on the printed circuit board (PCB). This rule helps ensure that there is sufficient space between components to prevent electrical and physical interference, avoiding short circuits or other potential issues during the manufacturing and assembly processes We will now create a design rule which targets components of a specific footprint. We can build a custom query for this purpose, using.

## 1.5.1 Creating a Component Clearance Rule

In this case we will create clearance between two relays.

31. Whilst still in the `Module 20 PCB Design Rules Creation.PcbDoc`, open the Design Constraints Editor, **Design » Rules…** or keyboard combination (**D » R**)
32. Navigate to the Placement section and add a new rule, change the rule properties as below, use F as reference:
    a) Name: `Relay_Clearance`
    b) From the: `Where The First Object Matches section` drop-down menu, click `Footprint` and select `Anderson_BLK_RHT`
    c) From the: Where The Second Object Matches section drop-down menu, click Footprint and select `Anderson_Red_RHT`
    d) Minimum Horizontal Clearance: `100mil`

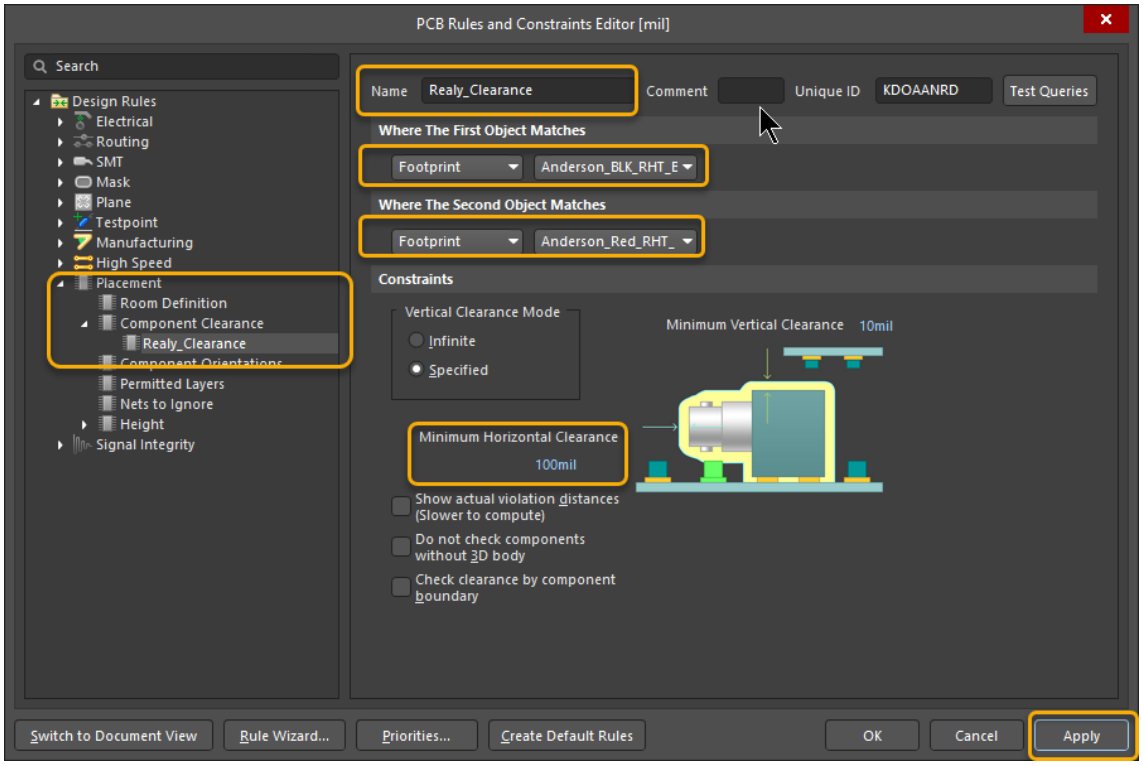


*Figure 9. Find Similar Objects dialog*

33. Select **Apply** to save the new design rule.
34. Click **OK** to close the DRC editor.

## 1.5.2 **Verifying Applied Design Rules in the PCB**

35. Go to *Preferences*.
    a) Under the *PCB Editor* branch, open the *General* page.
    b) Ensure that **Online DRC** is enabled, Online DRC
    c) Click **OK** to close the *Preferences*.
    d) Hit **Shift+C** to clear any masks or selection.
36. Moving the components (`RLY4`, `RLY3`, `PWRP1`, `PWRN1`, `RLY1`) will remove or add the green highlight depending on the distance between each other based on the new *Component Clearance* constraint rule. This would be easy to see in 3D mode.
37. Drag `RLY1, RLY2, RLY4` and `RLY3` on to the PCB, and place them close as possible. It will be clear to when the placement violations appear, the components should be highlighted in green, as seen in Figure 10.

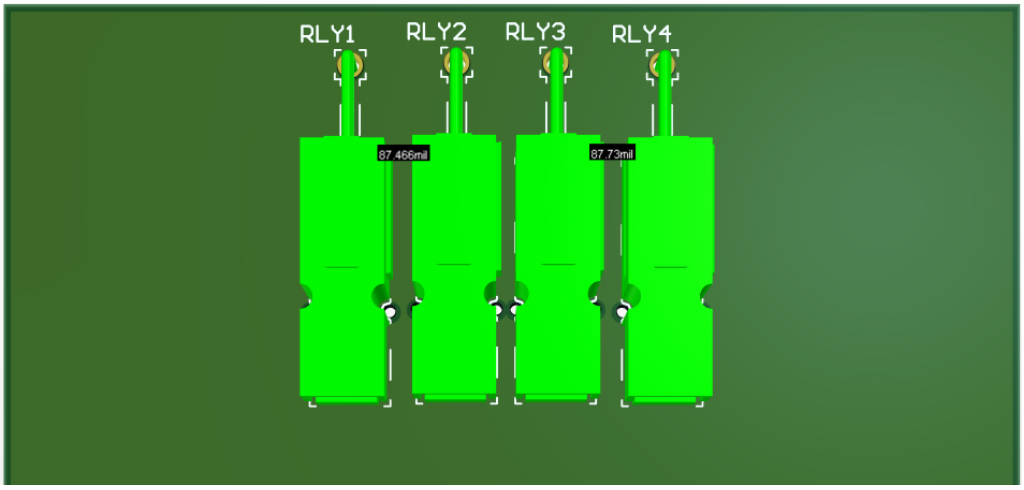


*Figure 10. Cmponent Clrance Rule*

38. Drag the relays away from one another, whilst dragging, hit **R** to change the component pushing mode to *Push:Avoid*, as indicated in the Heads-up Display, now try to place them close as possible, you will notice the components will not move any closer than the specified clearance design rule.

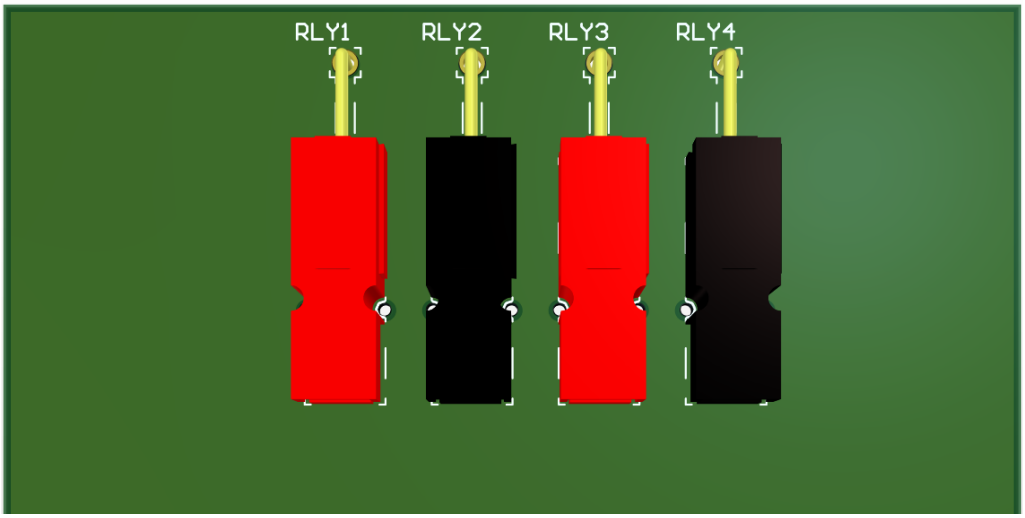


*Figure 11. Applied Anderson Component Clearance constraint*

# 1.6 Import / Export Rules

Design rules can be exported from and imported to the *PCB Rules and Constraints Editor* dialog. This allows you to save and load favorite rule definitions between different designs.

39. Open the *PCB Rules and Constraint* editor.
40. Right click to see the **Export/ Import Rules…** command.

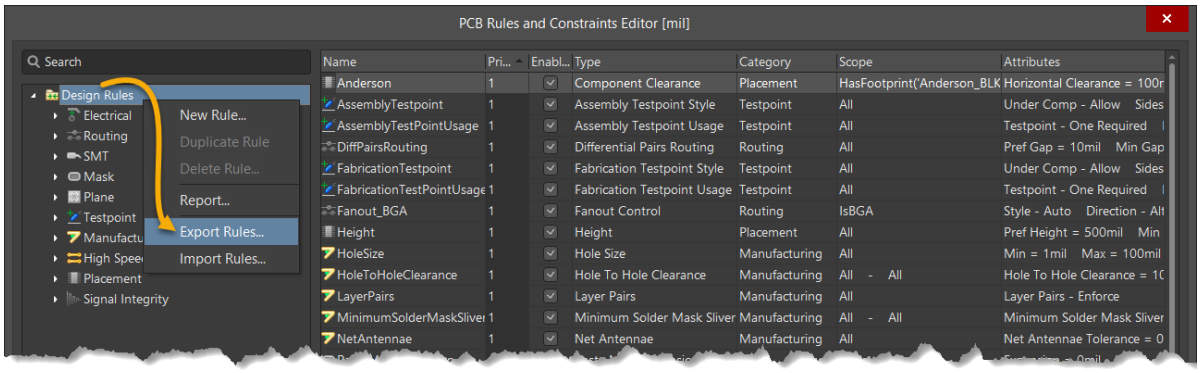


*Figure 12. Import / Export Rule*

41. Save all documents using **File » Save All**.
42. Save the modifications to the server:
    a) At the *Project* panel, next to the Project name you find the command **Save to Server** 
    
    .
    b) Select **Save to Server.**
    c) At the dialog *Save [Project Name],*
        i) Activate the checkboxes for the files that are not under version control.
        ii) Add the comment `Module 20: PCB Design Rules Creation - [Add Your Name]- Finished.`
        iii) Select **OK.**
43. **When ready, close the project and any open documents**.

**Congratulations on completing the Module!**

Module 20: PCB Design Rules Creation

**from the**

**Altium Designer Essential Course**
**with Altium 365**

**Thank you for choosing Altium Designer**