Altium Designer

Essentials Training with Altium 365

Module 20B: PCB Design with Constraint Manager

Altium® TRAINING

Altium® TRAINING

Software, documentation and related materials:

Table of Contents

Altium
TRACKING

Module 20B: PCB Design with Constraint Manager

1 Purpose

In this exercise we will use the Constraint Manager for Project Design Rule creation. You will learn various methods to create PCB design 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).

> Caution: The Constraint Manager is only available if you are using an Altium Designer Pro License. Note that the Constraint Manager is not supported with the Altium Designer Standard Subscription. For details, please see our Online Documentation.
>
> This module and the exercises are an introduction to the Constraint Manager for Pro License Users. All other Modules for the Essential Training will not use the Constraint Manager.
>
> The Constraint Manager is by default active if you create a new Project. If you open a project created with AD23 or earlier versions, the "classic" PCB Rules and Constraint Manager, as described in Module 20 PCB Design Rules Creation, will be available. If you open the PCB without a Project, the rules are shown with the "classic" PCB Rules and Constraint Manager.

2 Shortcuts
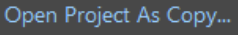
Shortcuts used when working with Module 20B: PCB Design with Constraint Manager

D » I	PCB Import Changes from Project
D » R	Constraint Manager (PCB)
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	Change Unit (Mil ← → mm)

ALTIUM
TRAINING

3 Preparation

1. Close all existing projects and documents.
2. Next, create a Copy / Clone of the Training Project `Module 20B PCB Design Rules Creation with Constraint Manager`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button [icon].
5. Navigate to the predefined Training Project `Module 20B PCB Design Rules Creation with Constraint Manager (Top\Projects\Altium Designer Essentials Training Course\...)`.
6. Select **Open Project as Copy...** `Open Project As Copy...`.
7. At the new dialog *Create Project Copy:*
   a) Add your name to the project: `Module 20B PCB Design Rules Creation with Constraint Manager – [Your Name].`
   b) Add a description: `Module 20B PCB Constraint Manager – [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.
8. 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.

> Hint: For details how to Copy / Clone the predefined training project see Module 9 Making the Connection, Step 3 Preparation.

3.1 ECO

9. From the *Projects* panel, open the `Module 20B PCB Design Rules Creation With Constraint Manager.PcbDoc`.
10. Go to **Design » Import Changes From [Project Name].PrjPcb** to open the ECO Dialog.
    a) Select **Execute Changes** to execute the ECO.
    b) After the ECO has finished, select **Close** to close the ECO dialog.

> Caution: Using the Constraint Manager replaces the functionality of defining Rules and Classes with Directives at the schematic.
> During schematic creation - Module 12 Schematic Updating - we placed directives that created a `Power` NetClass, similar we prepared a CAN_Bus directive with rules for the exercise.
>
> If you execute the ECO, the Class Definition for the Bus `Relay` could be seen at the ECO, but the other Classes and Rule definitions from the predefined directives are ignored.
> Instead of placing Directives for PCB Rules you would open the Constraint Manager from the Schematic to add Classes and Rules for the PCB.

Altium TRAINING

4 PCB Rules and Constraints Editor

4.1 Clearance Rule for 12V

11. Go to **Design » Constraint Manager...** (**D » R**), a new document with the Name *Module 20B PCB Rule Creation with Constraint Manager.PCB [Constraints]* will open.

> Hint: Based on your screen resolution it may be suggested to use the option **Multiline Document Bar** from the Preferences, Section System > View, Area *Document Bar*.

12. At the top of the document, there are four different constraint types; Clearance, Physical, Electrical and the All Rule area. This can be seen in Figure 1.
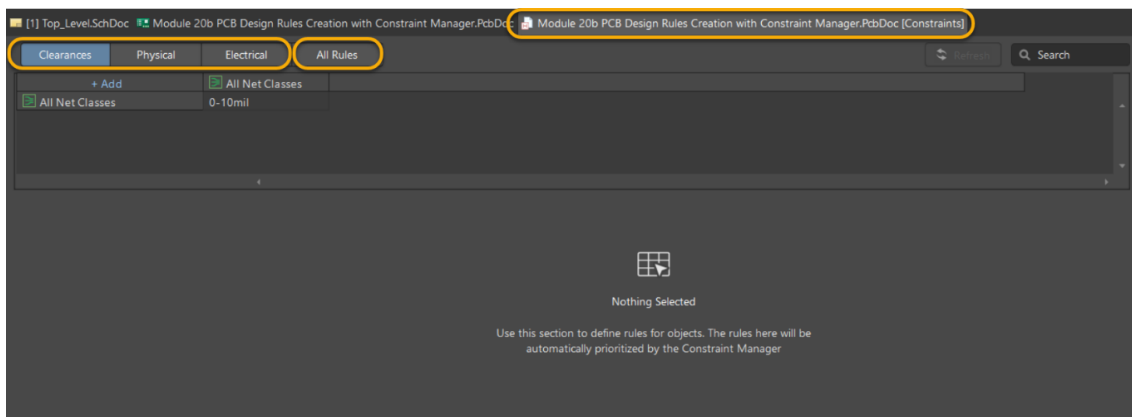

Figure 1. Constraint Manager

13. Select the four different constraint types on top to see the different configuration areas from the Constraint Manager, as seen at Figure 2.
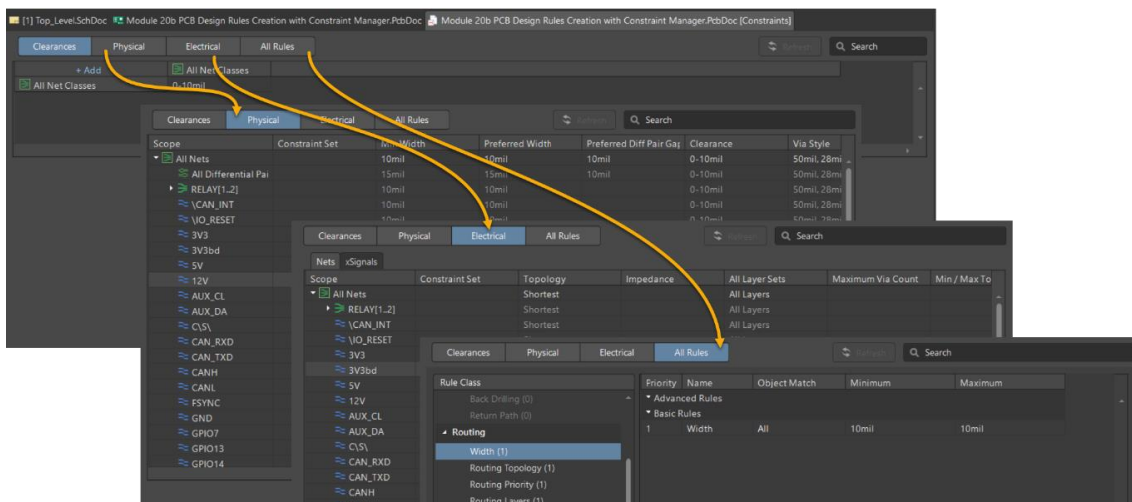
Figure 2. The four different constraint types from the Constraint Manager

14. We will now create a new rule which will target the 12V power net.

 a) Select the constraint types **Physical.**

 b) If needed scroll down, select the net `12V` and the cell for *Clearance*.

 Left-click on the *Clearance* cell and enter the value `15mil`, Figure 3.

 c) As alternative you could go to the graphic or the matrix area and change the values there.

Figure 3. Change the rule Value for 12V net

4.2 Design Rule Using Net Classes

Instead of scoping objects of a single net in Design Rules, we can target multiple nets by grouping them in a net class. We will now add a rule, scoping a net class, and assign a width to this net class, so for to all members from that class.
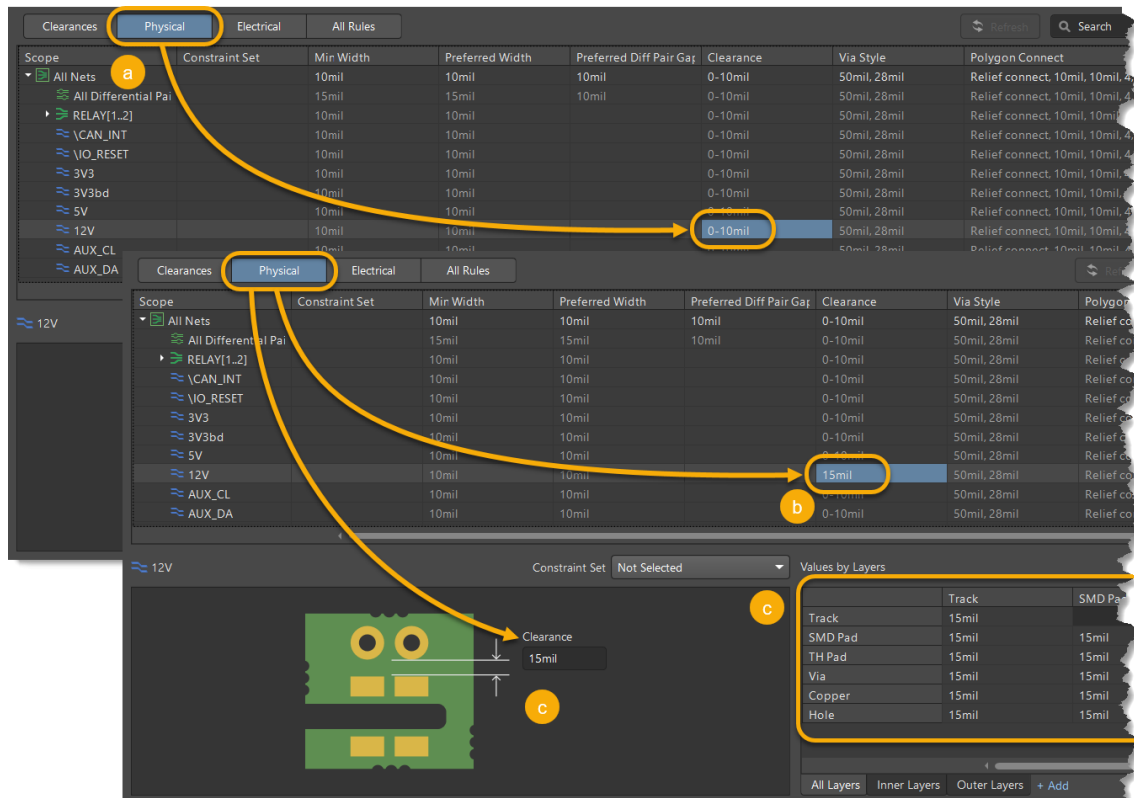
15. Still in the *Constraint Manager*, from the top choose constraint type *Clearance* Clearances . The constraint type *Clearances* is a matrix, that allows defining electrical clearances between classes of nets and/or differential pairs.

16. First, we will create a net-class `Power Net Class`, after that we define a rule for that net class.

17. Do a right-click, select the command **Classes » Add Class**. The *Add Classes* dialog will appear, Figure 4.

Figure 4. Add New Class
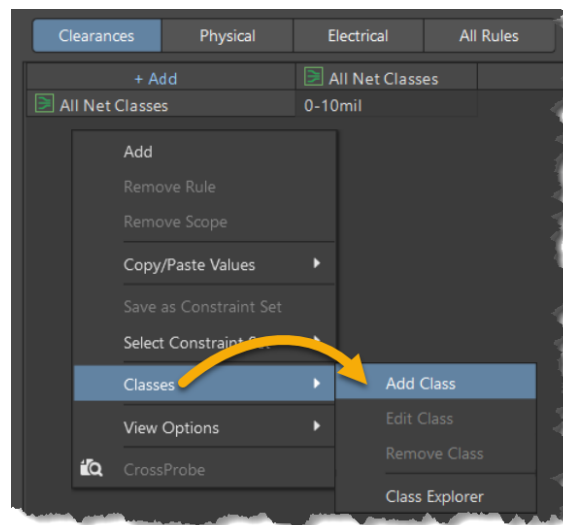
18. At the dialog *Add Class:*

   a) *Add the name* `Power Net Class.`

   b) *Add the net* `GND, 12V, 5V, 3V3,` *and* `3V3bd` *to the Class. Select* **OK** *to close the dialog, as seen at* Figure 5*.*
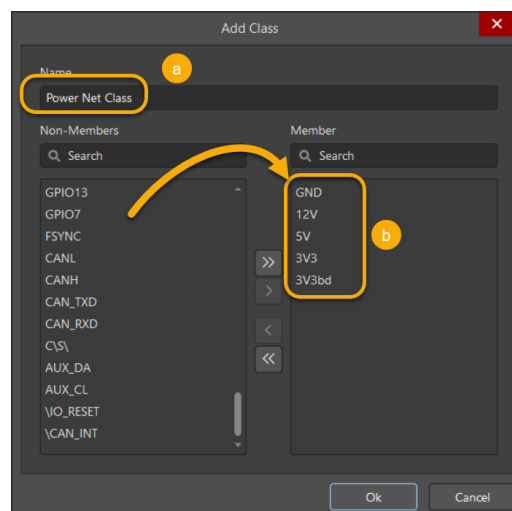
Figure 5. Define Power Class

7

19. Next:

    a) Select **+Add** or do a right-click to open the *Add Net Class* dialog.

    b) Choose the class `Power Net Class`.

    c) Select **Add**, the *Add Net Class* dialog automatically closes, you see the new net class added to the *Constraint Manager*.

Figure 6. Add the New Class to the Constraint Manager

20. From the matrix, select the cell `Power Net Class` in the *All Net Classes* column and change the value to `15mil`, as seen at Figure 7.

Figure 7. Clearance for the Power Net Class

21. Save the modifications you have done so far in the *Constraints Manager*

Altium
TRAINING

4.3 Rule Priorities

22. Now that we have multiple clearance rules and Values, let's ensure that their priorities are set correctly.
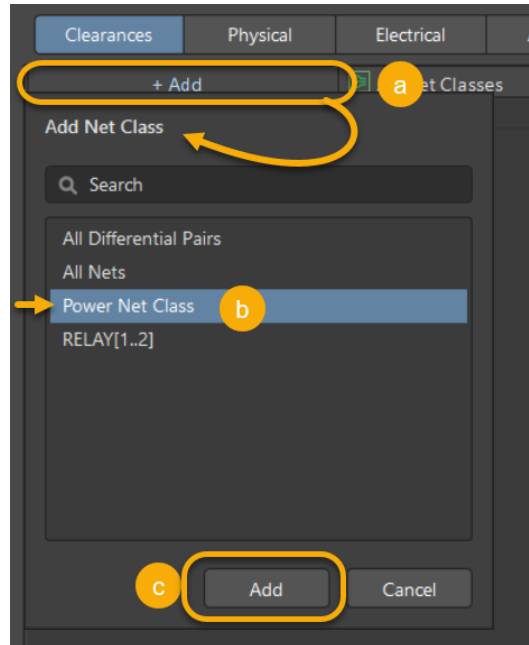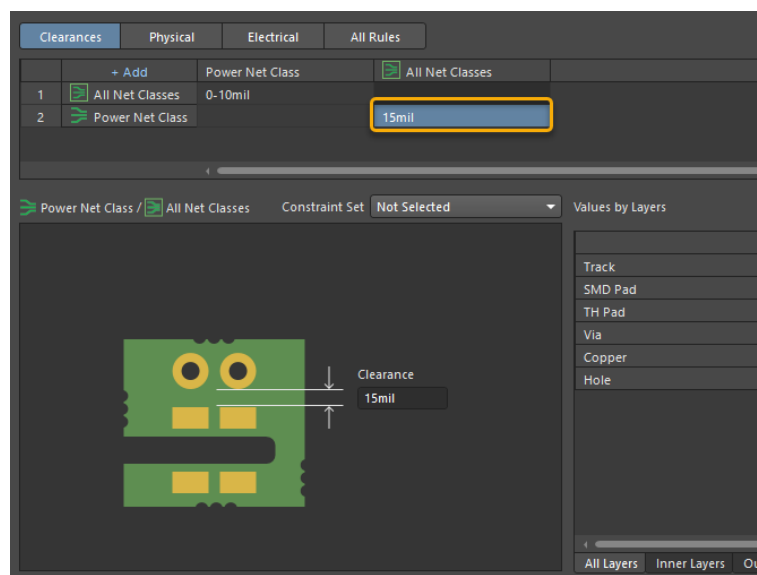
> Hint: Rule priority is automatic based on the natural hierarchy of design objects. The Constraint Manager defines the priorities of the rules in these views automatically: The priority is ordered from All (lowest) to object class to object (highest).

4.4 Vias

4.4.1 Setting Via Sizes

To enable multiple via sizes during interactive routing, we'll adjust the default Routing Via Style design rule.

We can modify the Via information within the Physical constraint types, just as we did for the 12V Net / Power Net class, to specify a single Via size for an individual net or a Net Class.

Now, we'll choose the constraint types *All Rules*, which are only accessible if you start the Constraint Manager from the PCB. From there, we'll define a new rule for the via size.

23. Select the constraint types **All Rules** All Rules .

24. Follow the steps shown in Figure 8 to create a new rule for the via style.
    a) Select from the section *Rule Class* the *Routing Via Style* configuration.
    b) Inside the configuration area do a right-click, select **Add Custom Rule** to add a new rule.
    c) Change the default rule name to `Custom Via Style.`
    d) Update the Values
        i) Diameter: Min 25mil   Max 40mil   Preferred 30mil
        ii) Hole Size: Min 10mil   Max 20mil   Preferred 15mil
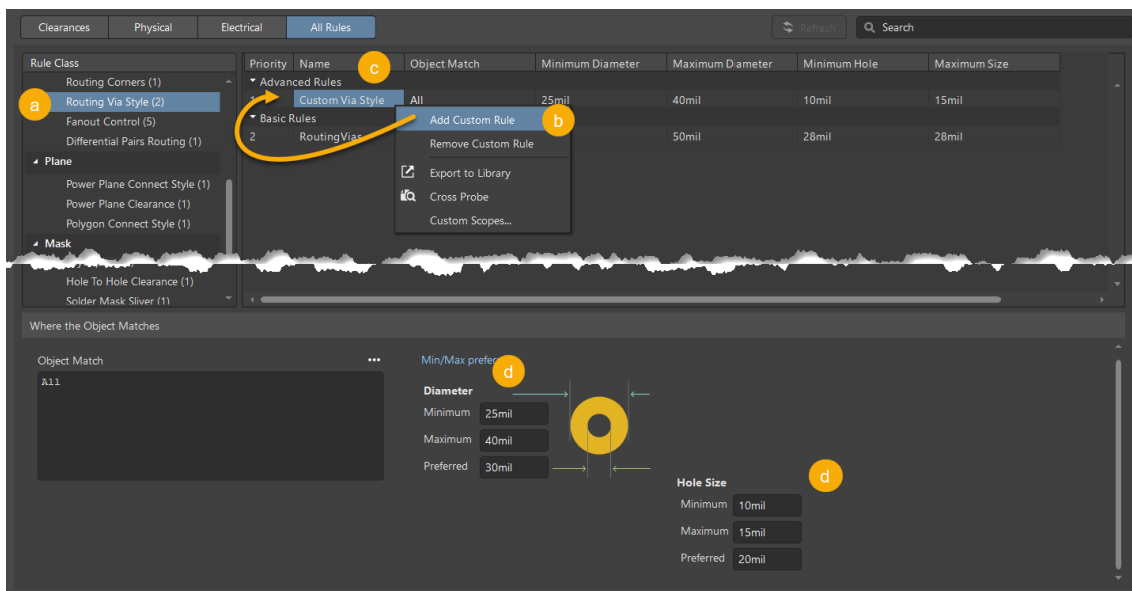   iii) Note that the Maximum values are in the middle.

Figure 8. New Via Rule

25. Save the modifications you have done so far in the *Constraints Manager* and close the *Constraints Manager*.

9

5 Design Rule Using Custom Queries

5.1 Create Queries Using Find Similar Objects

We will now create a design rule which targets components of a specific footprint. We can build a custom query for this purpose, using **Find Similar Objects**.

26. While in the `Module 20B PCB Design Rules Creation with Constraint Manager.PcbDoc`, right-click on component `RLY2` (**J»C**) and click on **Find Similar Objects**.

27. Ensure that *Object Kind* and *Footprint* are set to **Same**, and the **Create Expression** checkbox is enabled as shown in Figure 9. Then, click **OK**.

Figure 9. Find Similar Objects dialog
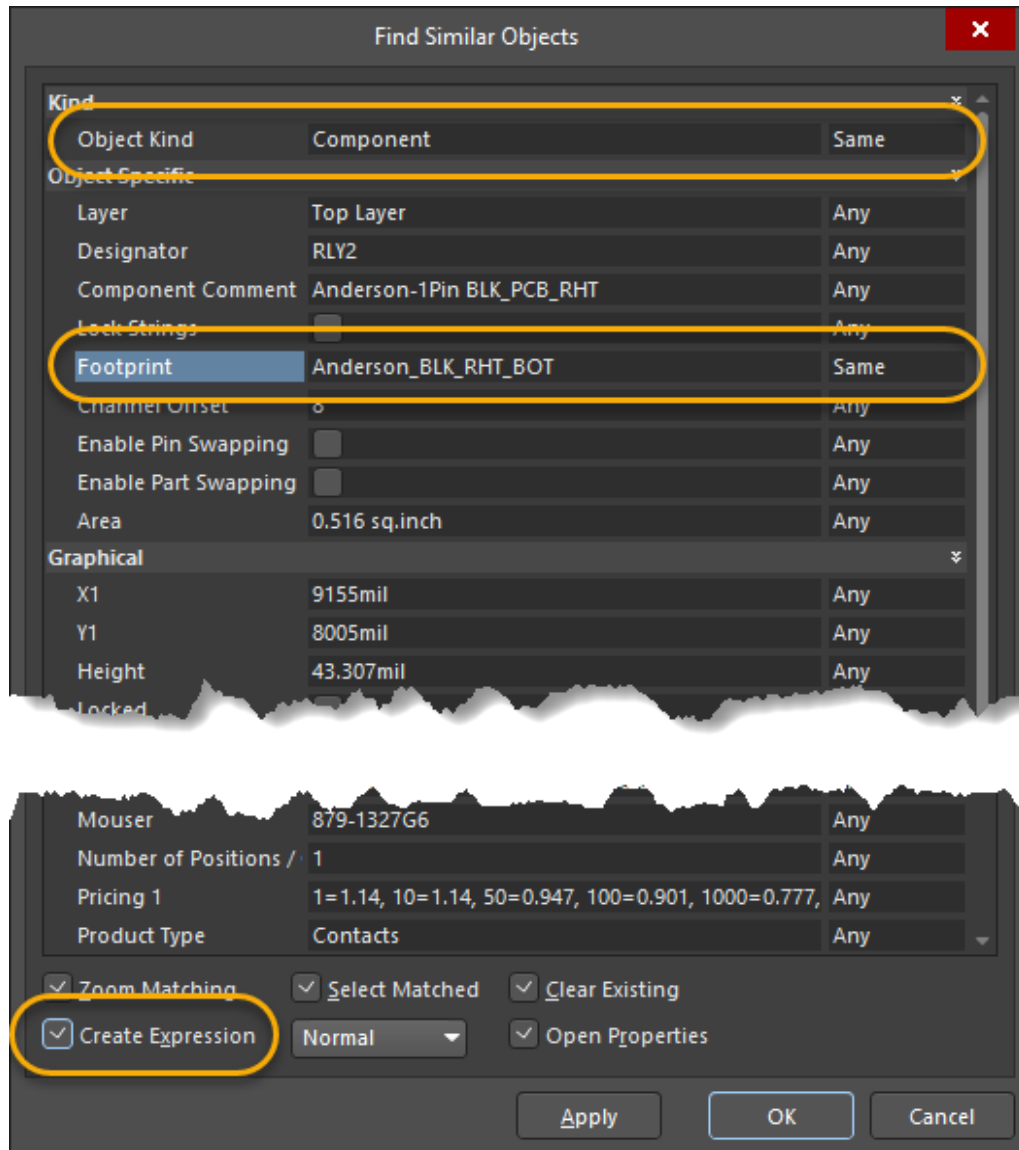
28. If not already open, open the *PCB Filter* panel from the **Panels** button and select **PCB Filter** or go to **View » Panels » PCB Filter**. Feel free to dock this panel wherever you wish.

29. Within this panel, the query created by running *Find Similar Objects* will be shown in the *Filter* pane. We'll now create a *Placement* rule that will apply to the selected components.

30. Within the *PCB Filter* panel, click on **Create Rule** near the bottom of the panel as shown in Figure 10.
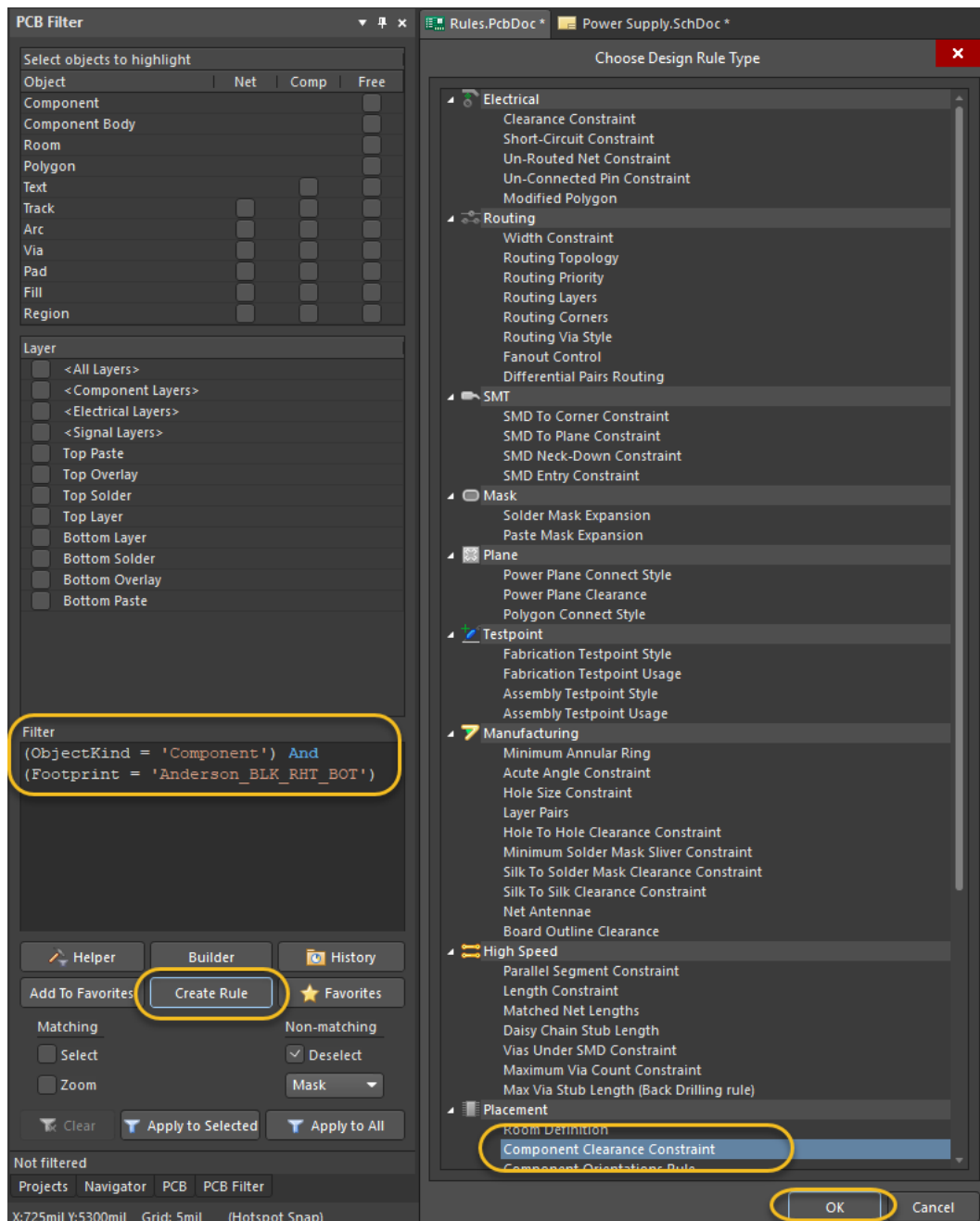

Figure 10. Creating a Clearance Rule from the PCB Filter panel

31. In the *Choose Design Rule Type* window, under **Placement**, click on **Component Clearance Constraint**.
32. Click **OK**.

33. We will define a component clearance constraint between components of the `Anderson_BLK_RHT_BOT` footprint and components of the `Anderson_Red_RHT_BOT` footprint using Figure 11 below.

    a) Select and Copy the query from *Where The First Object Matches* field with **Ctrl+C**.

    b) Change the *Where The Second Object* Matches from **All** to **Custom query** and paste the query into the field using **Ctrl+V**.

    c) From the 2ⁿᵈ object query, change `Anderson_`**`BLK`**`_RHT_BOT` to `Anderson_`**`Red`**`_RHT_BOT`.
       Please keep in mind that the query is case sensitive.

    d) Rename the rule to `Anderson`.

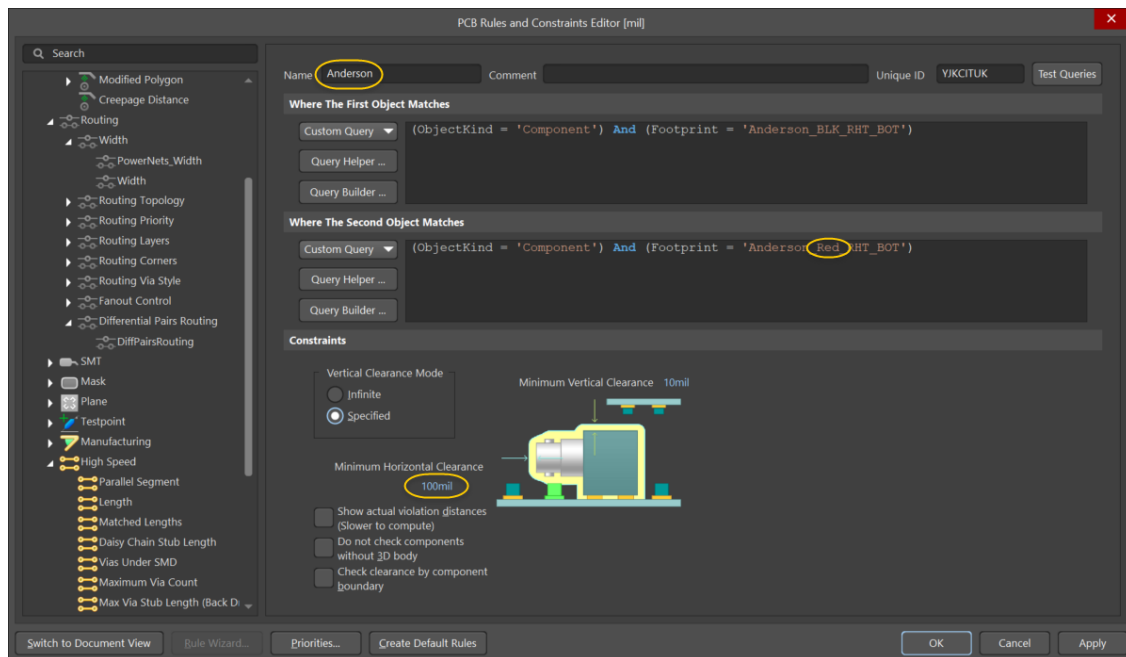   e) Set the minimum horizontal clearance to `100mil`.

Figure 11. Anderson Component Clearance rule definition

34. Select **OK** to close the *PCB Rules and Constraint Editor*.

35. Open the *Constraint Manager* with **D » R.**

    a) Select the constraint type *All Rules* [All Rules] .

    b) Select from the section *Rule Class - Placement* the rule category *Component Clearance*.

    c) *A new advanced rule with the name* `Anderson` *will be available.*

    d) *The new rule* `Anderson` *has priority 1, highest priority. The default rule* `ComponentClearance` *has the priority 2, lower priority.*
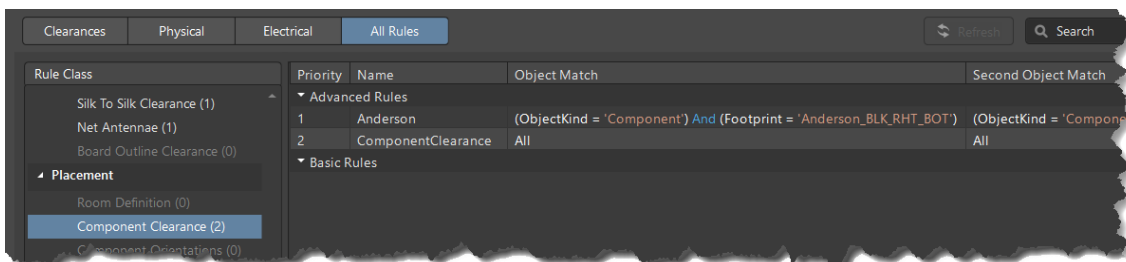

Figure 12. New Component Rule

36. Save the modifications and close the Constraint Manager.

5.2 Verifying Applied Design Rules in the PCB

37. 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.

38. Moving the components (RLY4, RLY3, RLY2, 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.

39. Move the four components (RLY4, RLY3, RLY2, RLY1) inside the PCB area.

40. Drag RLY3 towards RLY4. While dragging, hit **R** to change the component pushing mode to *Push:Avoid*, as indicated in the Heads-up Display. Drop it to the right of RLY4.

41. Drag RLY4 away from the other components to remove existing violations between RLY4 and RLY3 if you did not used the component pushing mode *Push:Avoid.*

42. Lastly, drag RLY1 and drop it to the left of RLY3, change the component pushing mode to *Push:Avoid* . Notice the clearance allowed between RLY1 and RLY3 is smaller than between RLY4 and RLY3, because of the *Anderson Component Clearance* rule defined earlier. This can be seen in Figure 13 below.
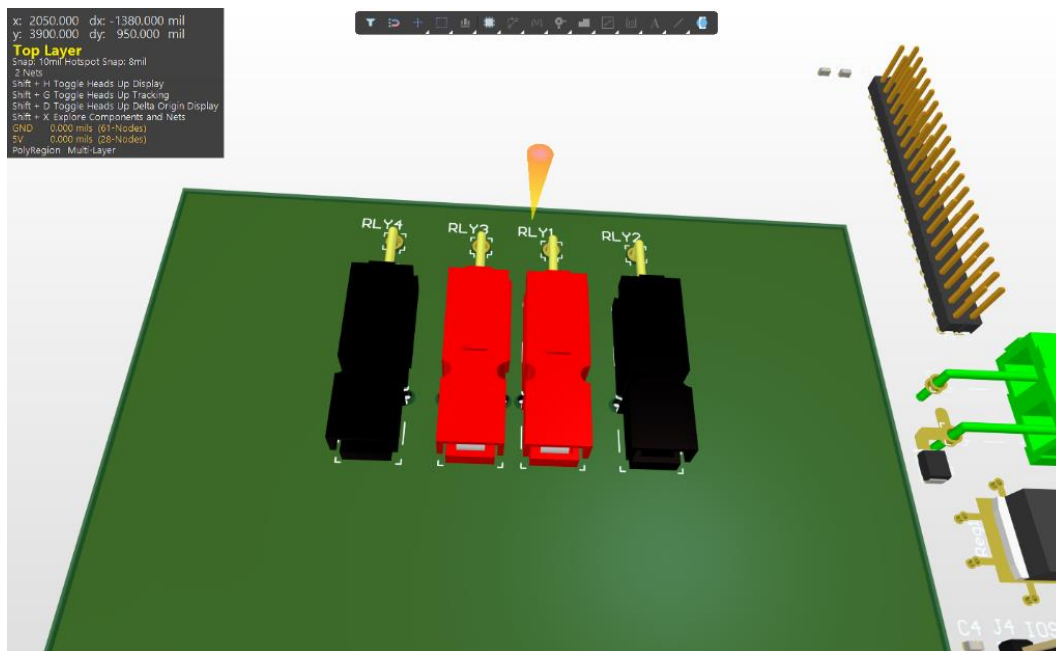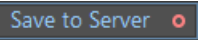

Figure 13. Applied Anderson Component Clearance constraint

6 Engineering Change Order

43. Now that we have updated the Constraint manager from the PCB, we will execute the ECO to sync the PCB information with the Schematic.

44. From within the PCB, select **Design » Update Schematics in Module [Project Name].**

45. At the new dialog *Engineering Change Order* (ECO) select the **Execute Changes** button in the lower left of the dialog.

46. After the ECO is executed, enable the **Only Show Errors** checkbox to check for any errors, the training example should be without errors.

47. When ready, close the *Engineering Change Order* dialog with **Close**

48. Save all documents using **File » Save All**.

49. Save the modifications to the server:

    a) At the *Project* panel, next to the Project name you find the command **Save to Server** `Save to Server  ◎`.

    b) Select **Save to Server.**

    c) At the dialog *Save [Project Name],*

        i) Add the comment `Module 20B: PCB Design with Constraint Manager – [Add Your Name]– Finished.`

        ii) Select **OK.**

50. When ready, close the project and any open documents**.**

Altium
TRAINING

Congratulations on completing the Module!

Module 20B: PCB Design with Constraint Manager

from

**Altium Designer Essentials Training
with Altium 365**

Thank you for choosing **Altium Designer**

Altium.
TRAINING