ALTIUM
DESIGNER

ALTIUM
365

Altium Designer

Advanced Training with Altium 365

Design Rule Queries

Altium®
TRAINING

Altium®
TRAINING

Software, documentation and related materials:

1

Table of Contents

Design Rule Queries

1 Purpose

In this exercise you will learn techniques on how to generate queries from the *Find Similar Objects* dialog and apply them to design rules.

Specifically, you will target specific GND vias and set their polygon connection properties to a Direct Connect style, while all other all other vias will be connected as thermal reliefs.

2 Shortcuts

Shortcuts used when working with Design Rule Queries
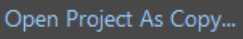
F1	Help
Select → right click → FSO	FSO (Find Similar Objects Dialog)
Shift+F → select object	FSO (Find Similar Objects Dialog)
D » R	PCB Rules and Constraint Editor
T » G » A	Repour All Polygon Pour
CTRL+S	Save Document

3 Preparation

1. Close all existing projects and documents.

2. Next, create a copy of the Training Project: `Design Rule Queries.`

3. Select **File » Open Project…** to open the *Open Project* dialog.

4. Enable the folder view button ⧉.

5. Navigate to the predefined Training Project `Design Rule Queries` `(Top\Projects\Altium Designer Advanced Training Course\...).`

6. Select **Open Project as Copy…** `Open Project As Copy…` .

7. In the new dialog *Create Project Copy:*

    a) Add your name to the project name: `Design Rule Queries – [Your Name].`

    b) Add a description: `Altium Advanced 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.

8. Wait until Altium Designer creates the copy of the project and opened the Project for you in the *Projects* panel, this may take up to 1 minute.

Hint: For details how to copy the predefined training project, see module *03 Getting started - Opening a Project.*

4 Creating the Rule

4.1 Find Similar Objects

9. From the *Projects* panel, open the `SL1 Xilinx Spartan-Queries.PcbDoc` document.

10. Zoom into the via stitching area, as shown in the bottom left area of Figure 1.
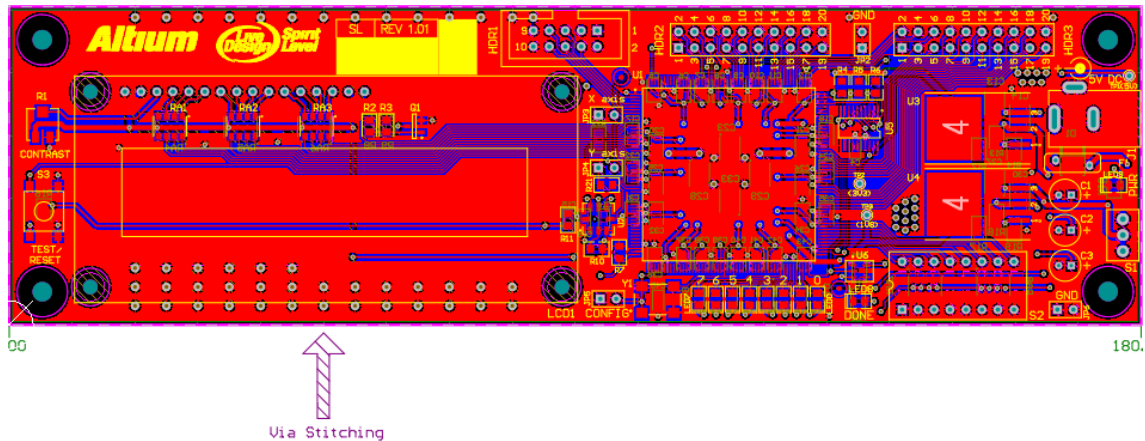

Figure 1. SL1 Xilinx Spartan Design with Via Stitching

11. Right-click on one of the GND vias and select **Find Similar Objects...**, as shown in Figure 2.
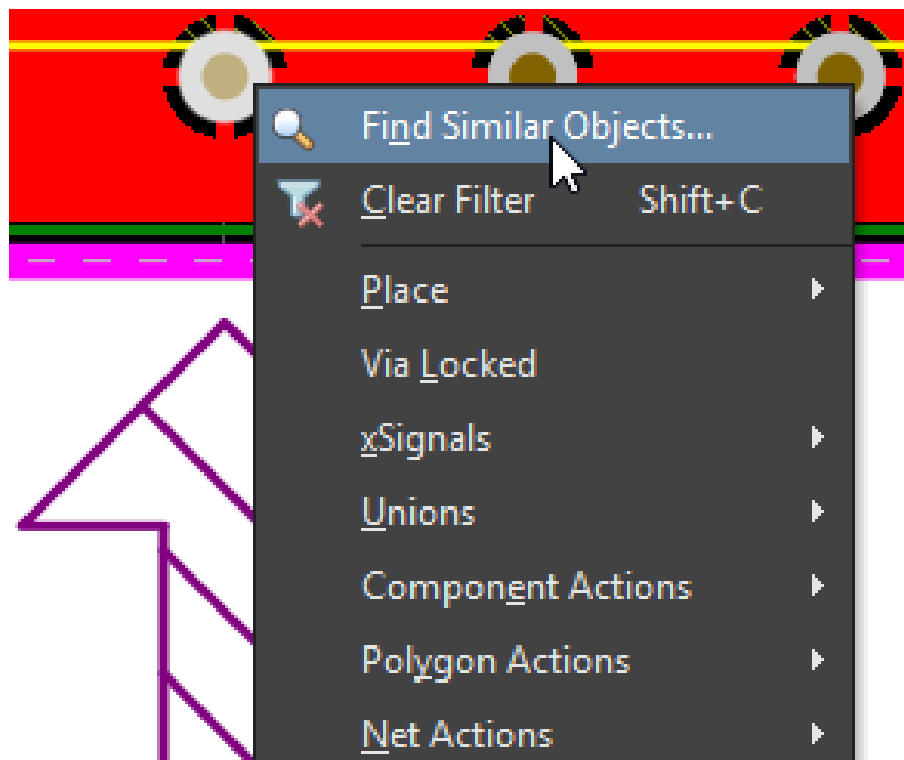

Figure 2. Using Find Similar Objects with a selected via

12. Ensure that the **Create Expression** and **Select Matched** options are enabled, as shown in Figure 3 below.

13. Set the viewing mode from **Normal** to **Mask**, using the drop-down menu.

14. From the *Object Specific* section of the panel, set the criteria for **Net**, **Via Diameter** and **Hole Size** to **Same,** as shown in Figure 3.

15. Select **Apply** to check your settings.
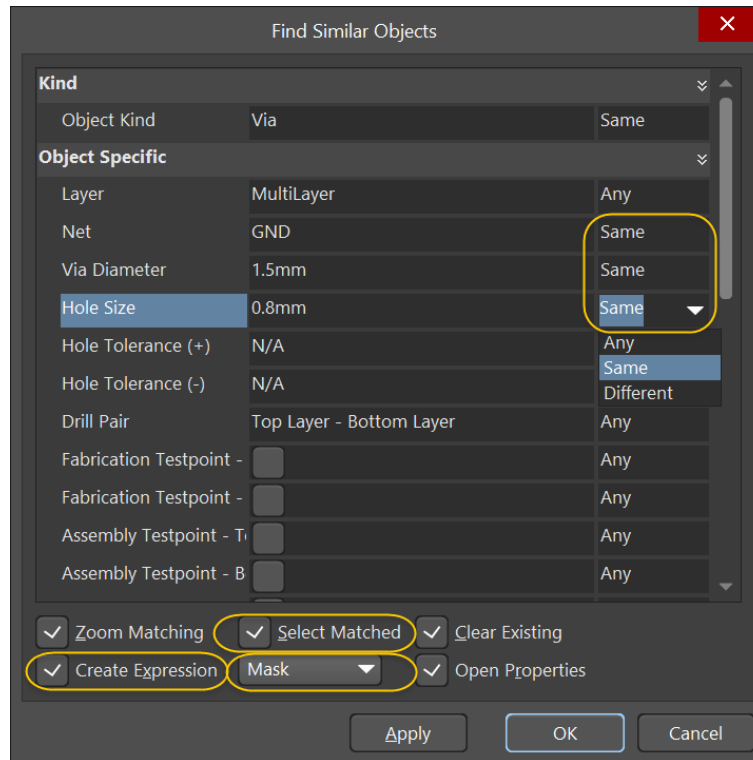
16. Select **OK** to continue.

Figure 3. Search Criteria for Find Similar Objects

17. You'll now see that all of the vias that match these criteria will be selected and masked, as shown in Figure 4. Your view may vary depending on your zoom level.
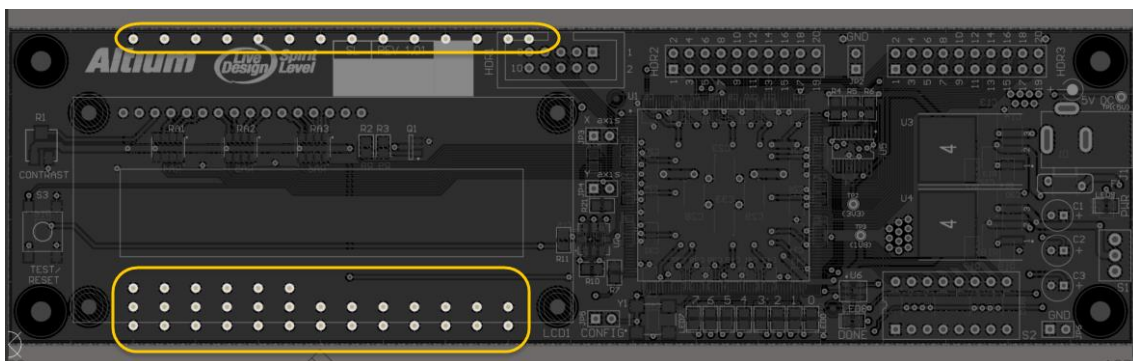

Figure 4. Masked selection from Matched Criteria

18. The *PCB Filter* panel will open. If not, it can be opened from the **Panels** button.

19. In the *Filter* pane, you will see the expression, visible in Figure 5 below, that was created based on your selection from the *Find Similar Objects* dialog.
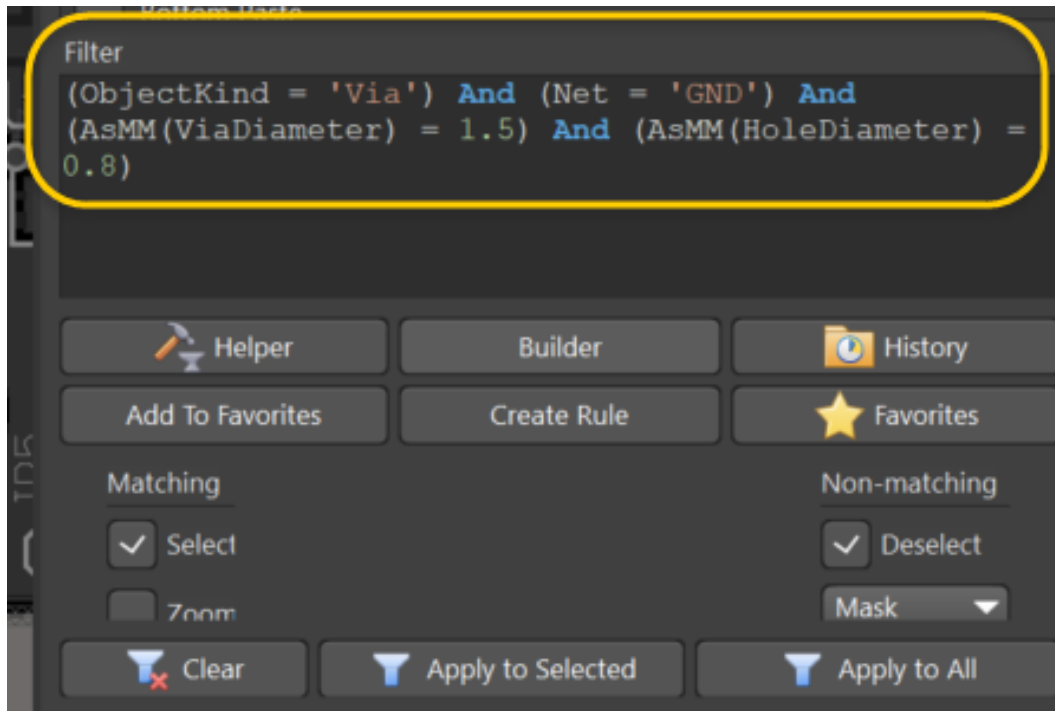

Figure 5. PCB Filter Expression

4.2 Applying the Created Expression

20. Below the *Filter* expression, select the **Create Rule** button, as shown in Figure 6.
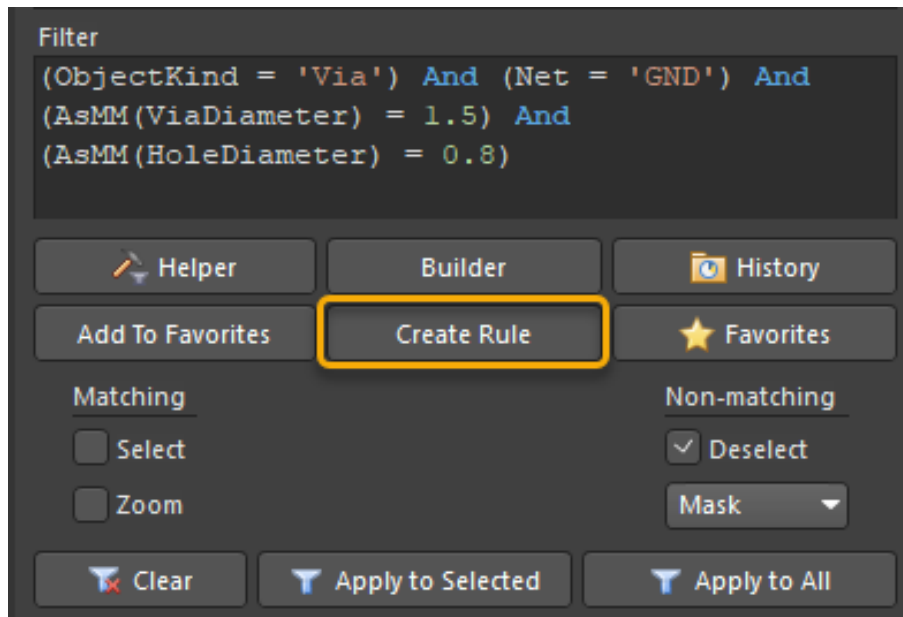

Figure 6. Create Rule from PCB Filter panel

21. In the *Choose Design Rule Type* window that opens, select the **Polygon Connect Style** under the *Plane* section, as shown in Figure 7.
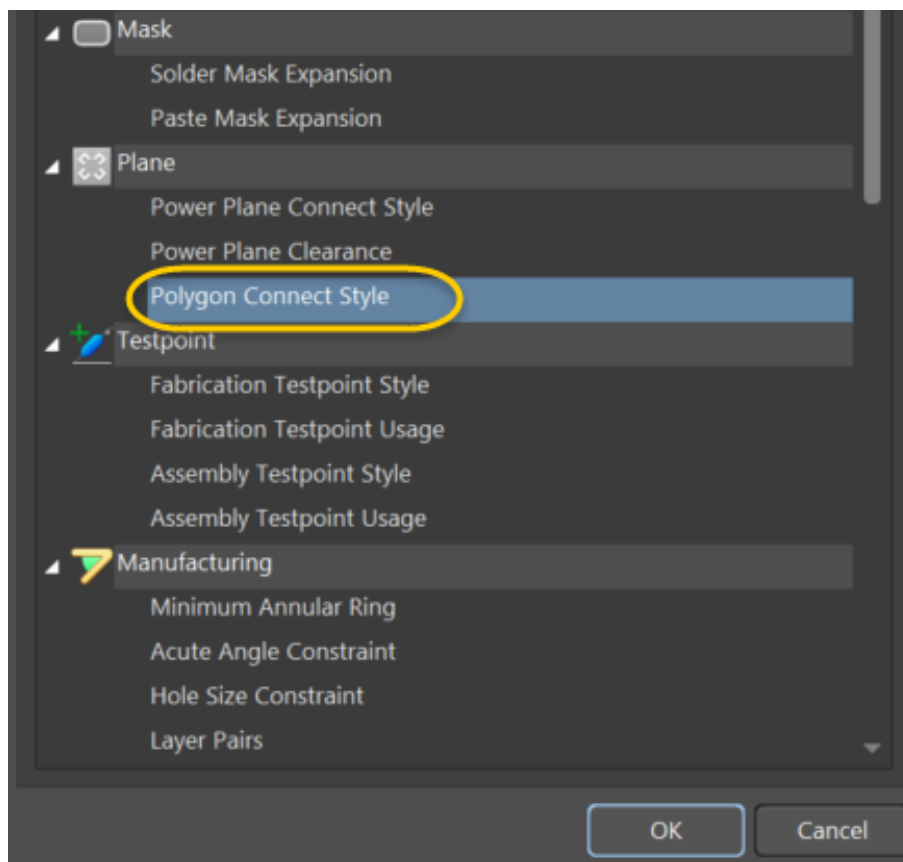

Figure 7. Selecting Polygon Connect Style

22. Select **OK** to continue.

8

23. The *PCB Rules and Constraints* Editor will open with a new *Polygon Connect Style* rule named `PolygonConnect_1`.

    a) Change the Name for the new rule to `Direct Connect Stitching Vias`.

    b) In the *Constraints* pane, change the *Connect Style* to **Direct Connect**, Figure 8.

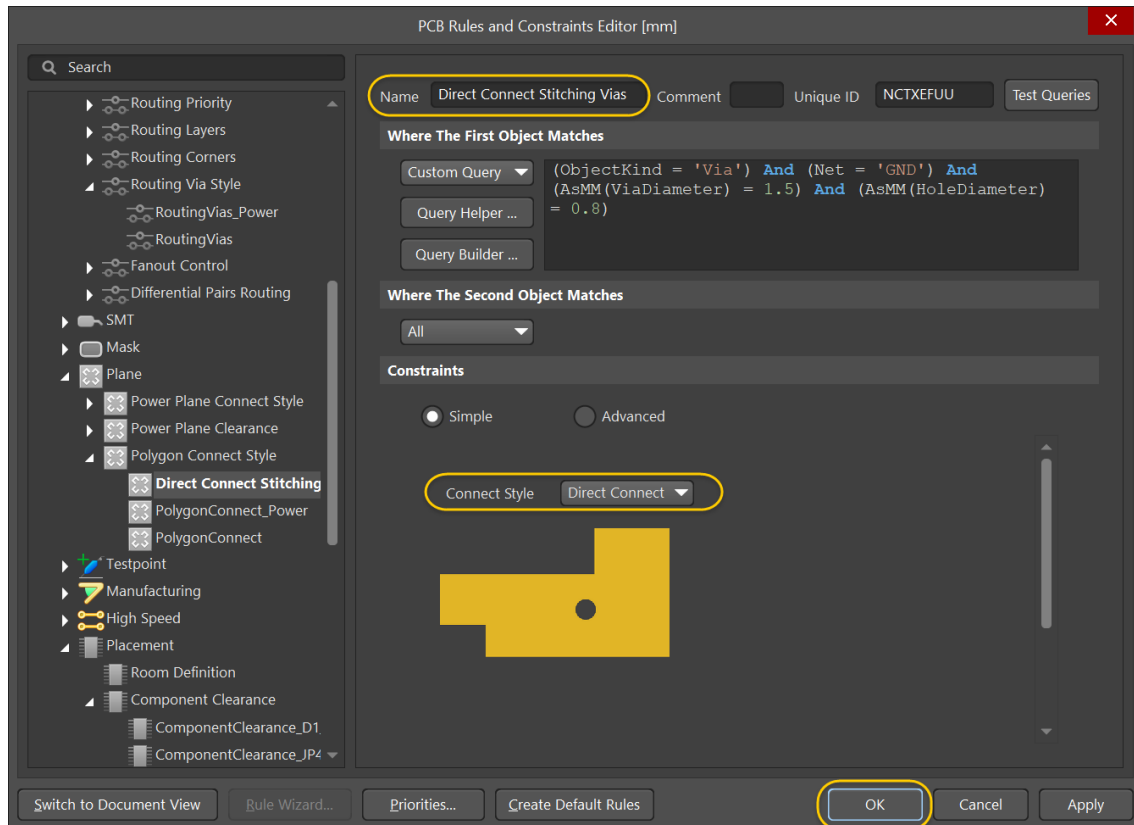   c) Select **OK** to close this window once finished.

Figure 8. Direct Connect Via Design Rule

5 Checking the Rule

5.1 Repouring Polygon

24. Clear any selection in the PCB by using **Shift+C**.

25. From the **Tools** menu, select **Polygon Pours » Repour All** to repour all of the polygons.

26. You'll now notice that all GND net vias, with a pad of diameter 1.5mm and hole size of 0.8mm, will be directly connected to the GND copper pour, as shown in Figure 9.
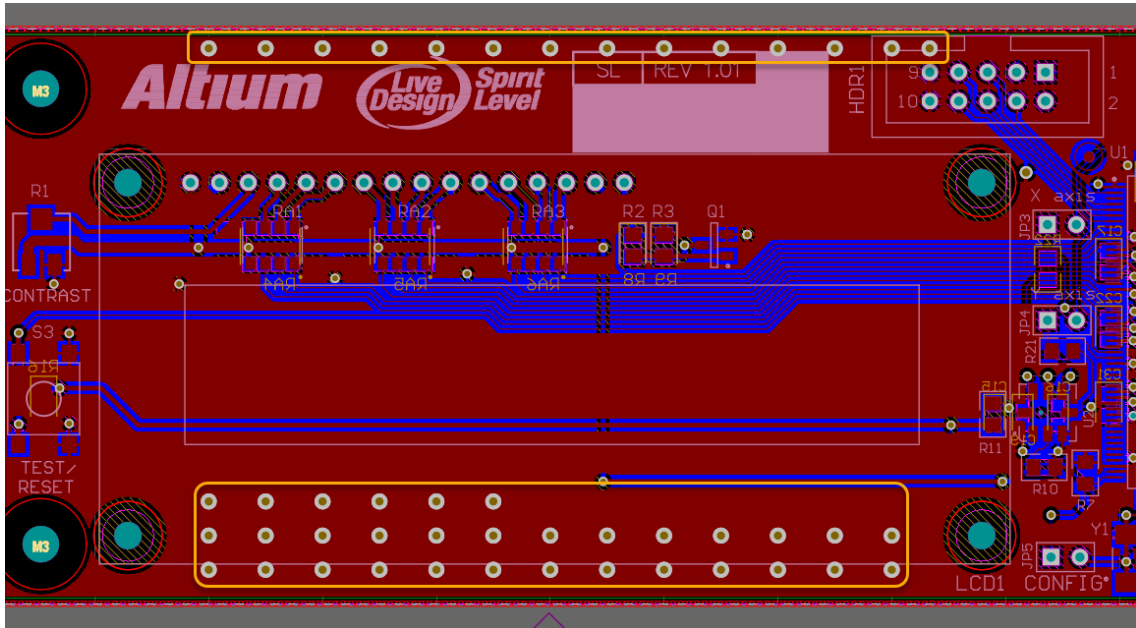

Figure 9. Direct Connected Via

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

28. Save the modifications to the server:

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

    b) Select **Save to Server**.

    c) In the dialog *Save [Project Name]:*

        i) Add the comment `Design Rule Queries – [Add Your Name] – Finished`.

        ii) Select **OK**.

29. When ready, close the project and any open documents, **Window » Close All**.

Congratulations on completing the Module!

Design Rule Queries

from

Altium Designer Advanced Training
with Altium 365

Thank you for choosing **Altium Designer**

ALTIUM®
TRAINING