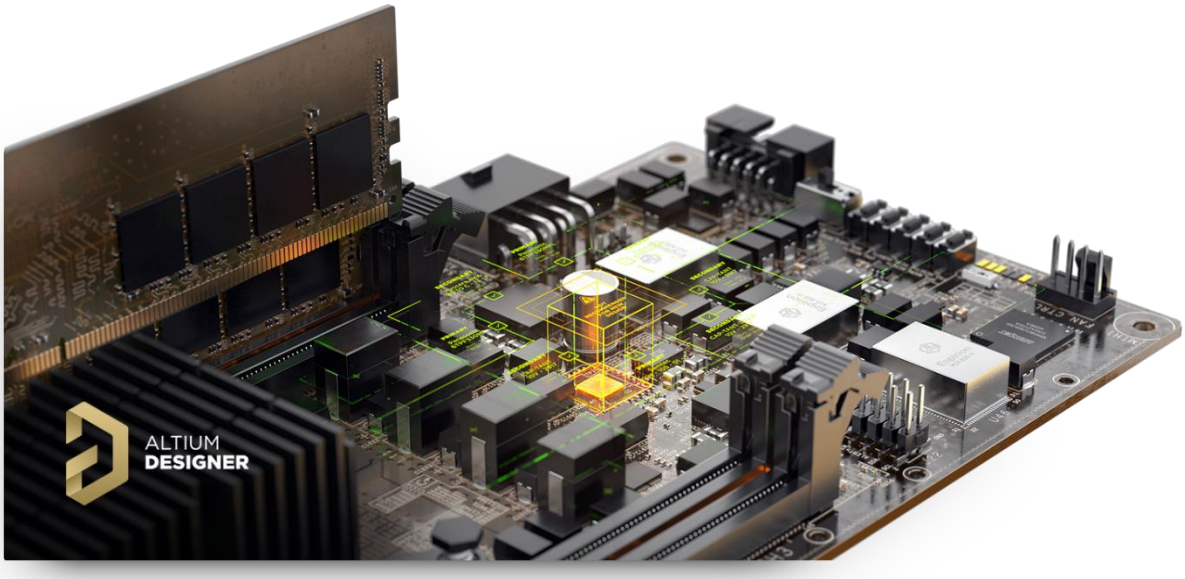

Altium Designer

Advanced Course

Module: Design Rule Queries

Table of Contents

Design Rule Queries

1.1 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.

1.2 Shortcuts

Shortcuts when working with Design Rule Queries

F1:	**Help**
Shift+F:	FSO
D-R:	**PCB Rules and Constraint Editor**
CTRL+S:	Save Document

1.3 Preparation

1. **Close all existing projects and documents.**
2. Open the `Design Rule Queries.PrjPCB` project found in its respective folder of the Advanced Training.

1.4 Creating the Rule

1.4.1 Find Similar Objects

3. From the *Projects* panel, open the `SL1 Xilinx Spartan-Queries.PcbDoc` document.
4. Zoom into the via stitching area as shown in the bottom left area of Figure 1.

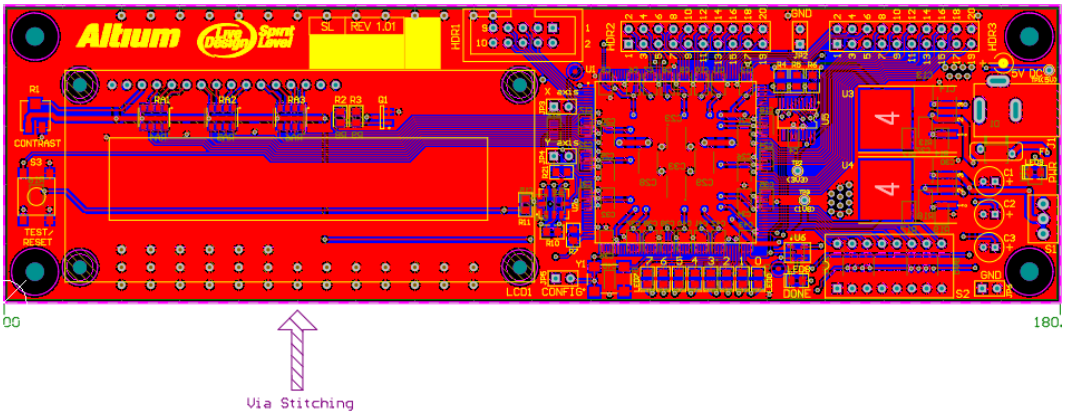

Figure 1. SL1 Xilinx Spartan Design with Via Stitching

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

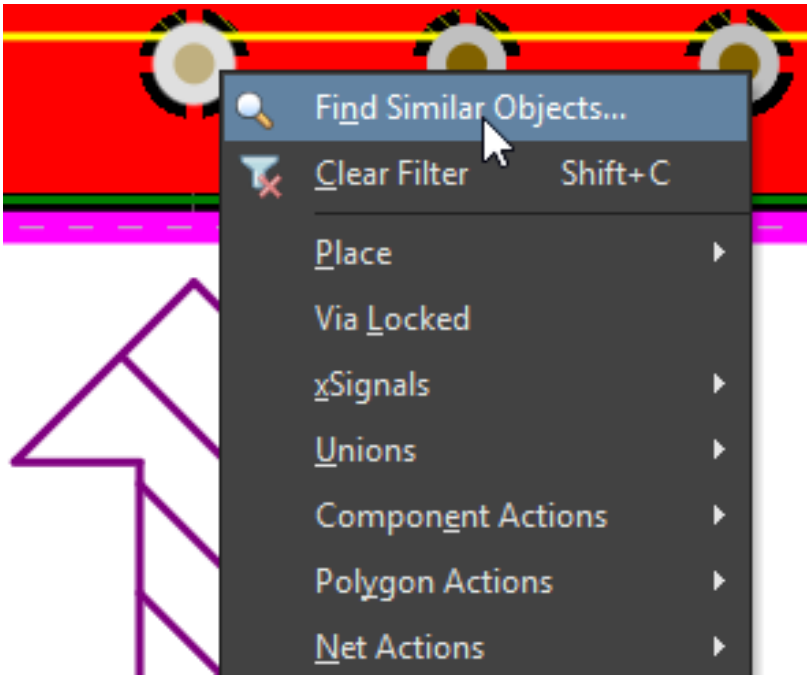

Figure 2. Using Find Similar Objects with a selected via

6. Ensure that the **Create Expression** and **Select Matched** options are enabled as shown in Figure 3 below.
7. Set the viewing mode from **Normal** to **Mask** using the drop-down menu.
8. 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.
9. Click **Apply** to check your settings, click **OK** to continue.

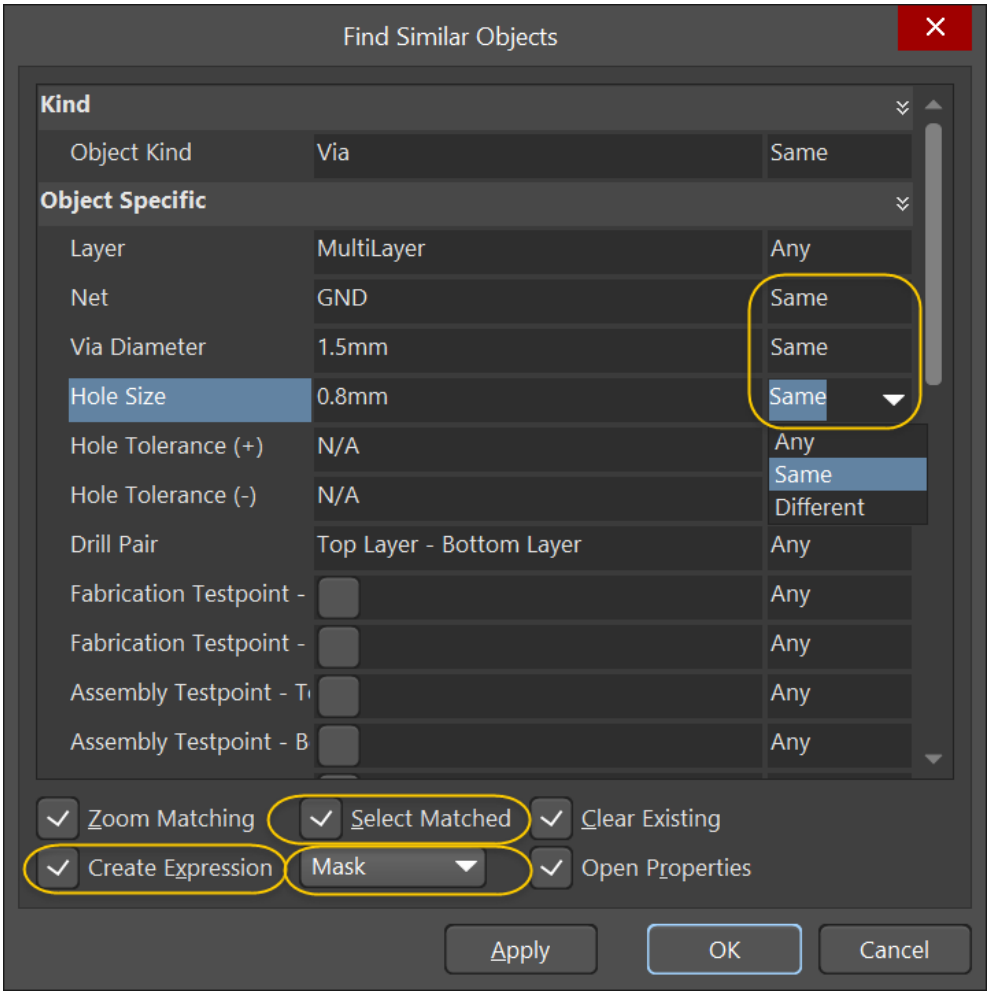

Figure 3. Search Criteria for Find Similar Objects

10. 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.

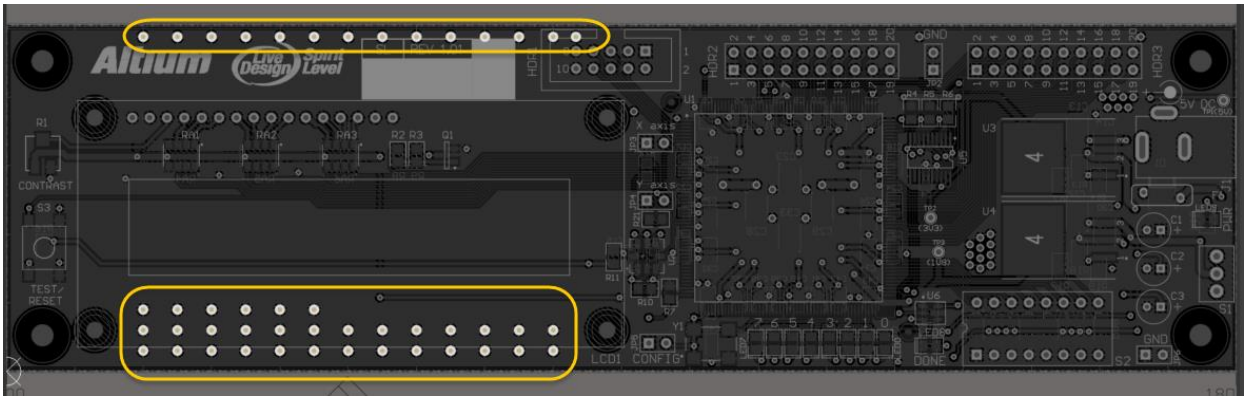

Figure 4. Masked selection from Matched Criteria

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

12. In the *Filter* pane, you will see the expression in Figure 5 below which was created based on our selection from the *Find Similar Objects* dialog.

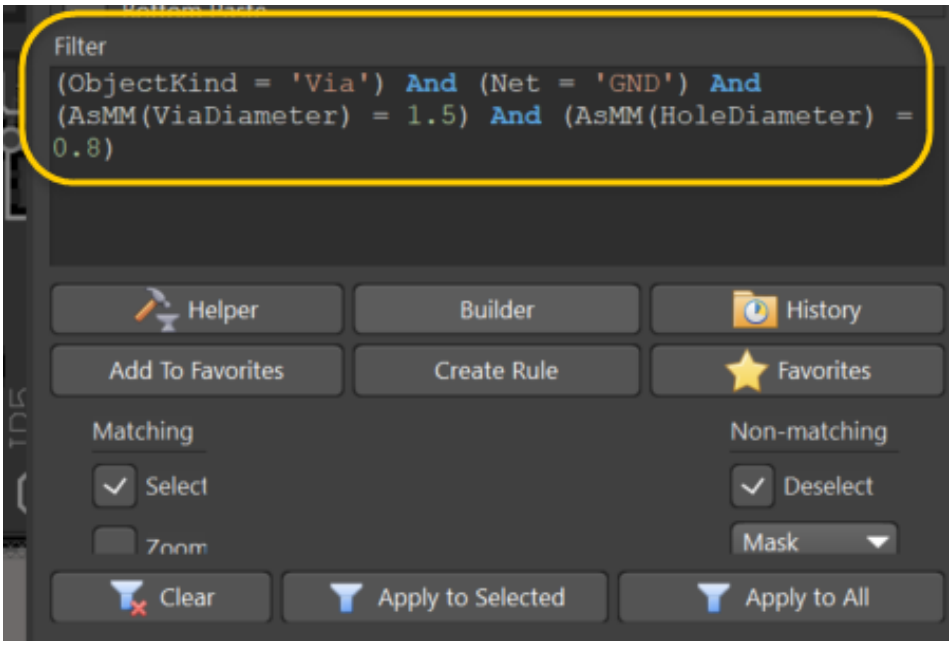

Figure 5. PCB Filter Expression

1.4.2 **Applying the Created Expression**

13. Just underneath the *Filter* expression, click the **Create Rule** button as shown in Figure 6.

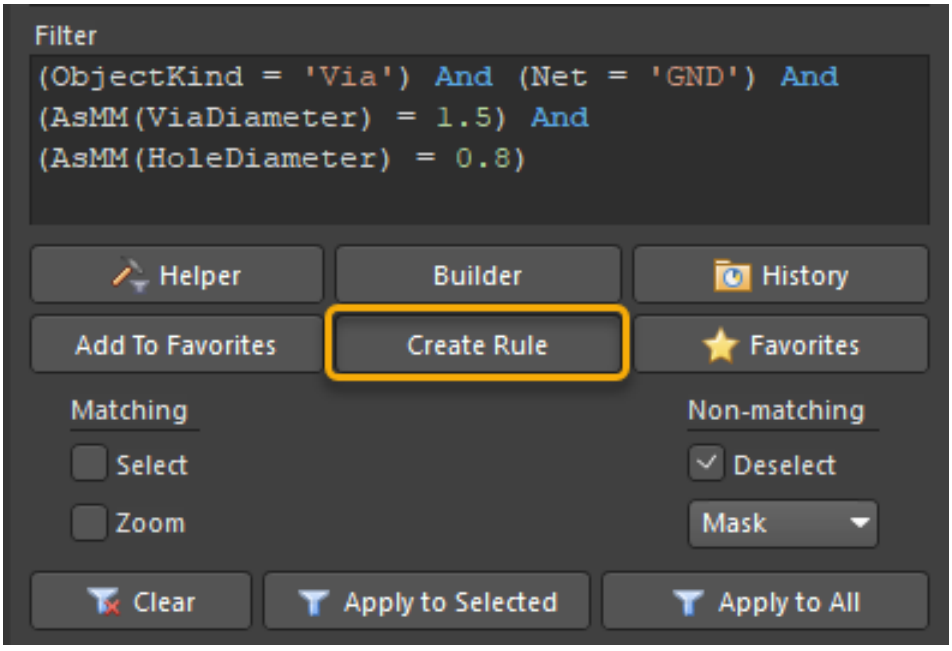

Figure 6. Create Rule from PCB Filter panel

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

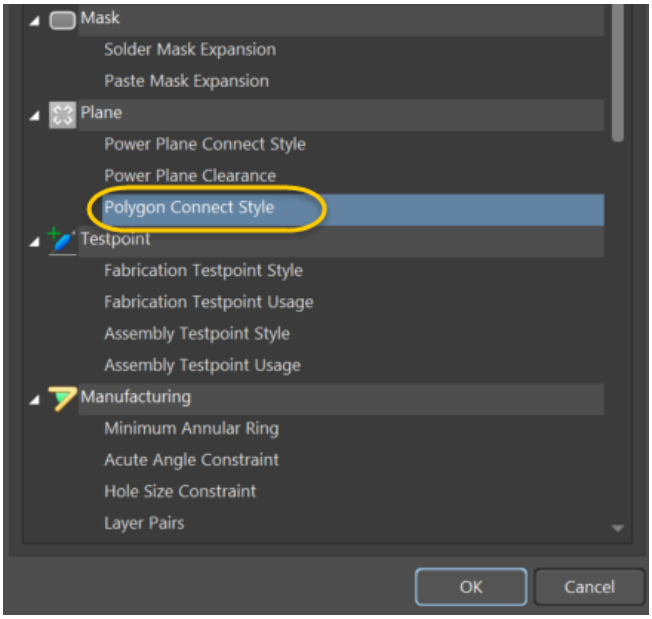

Figure 7. Selecting Polygon Connect Style

15. Click **OK** to continue.
16. 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** as shown in Figure 8.
    c) Click **OK** to close this window once finished.

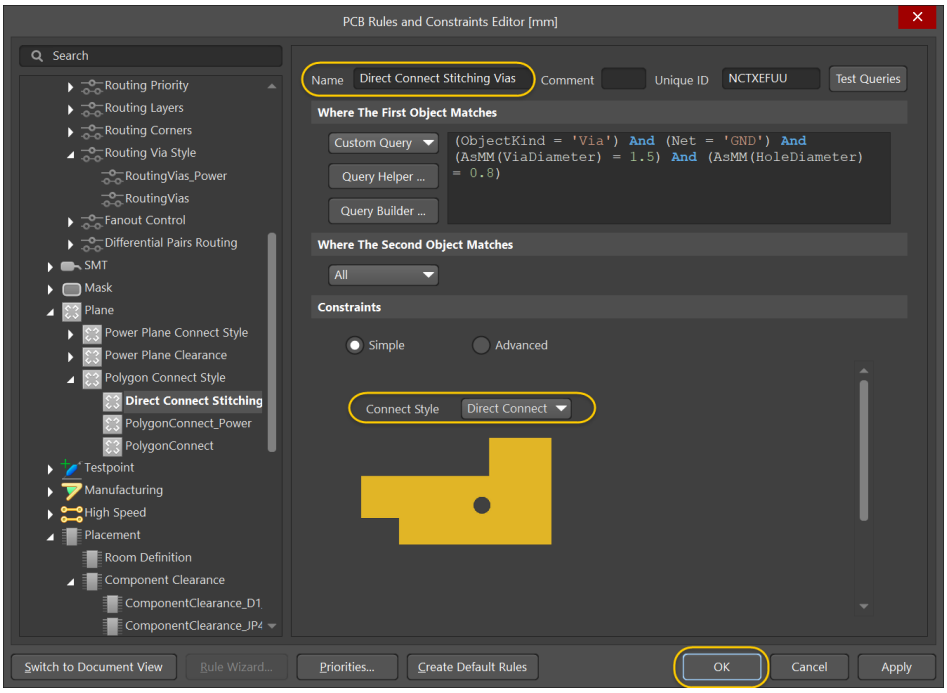

Figure 8. Direct Connect Via Design Rule

1.5 Checking the Rule

1.5.1 **Repour Polygon**

17. Clear any selection in the PCB by using **Shift+C**.
18. From the **Tools** menu, select **Polygon Pours » Repour All** to repour all of the polygons.
19. 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.

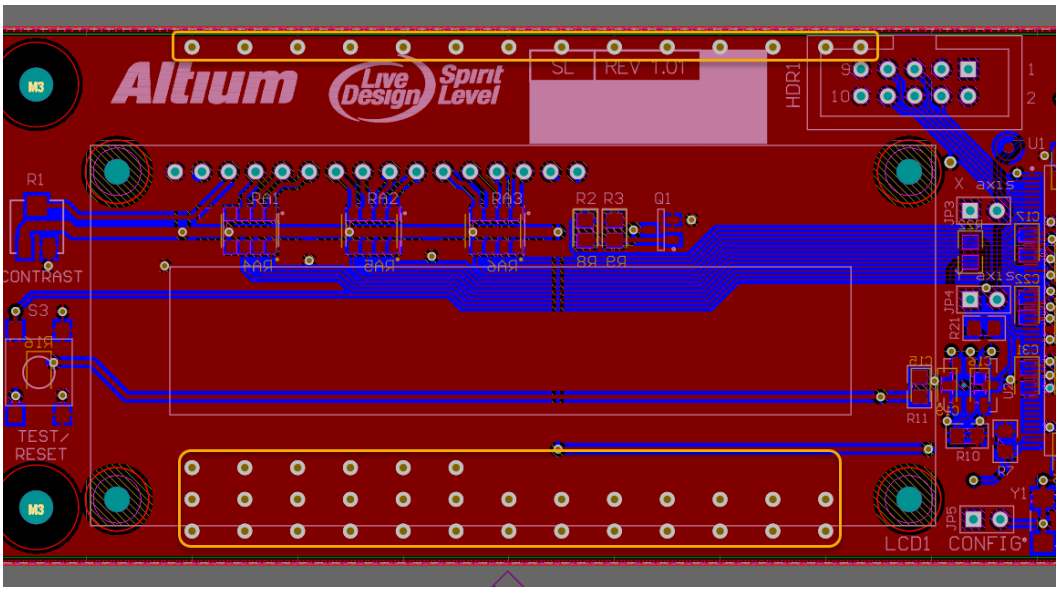

Figure 9. Isolated Direct Connected Via

20. Feel free to save the changes you made.
21. **Close the project and any open documents.**

Congratulations on completing module

Design Rule Queries

from the

Altium Designer Advanced Course

Thank you for choosing Altium Designer