Altium Designer

Advanced Training with Altium 365

Via Stitching and Via Shielding

Software, documentation and related materials:

Table of Contents

Via Stitching and Via Shielding 3

1 Purpose 3

2 Shortcuts 3

3 Preparation 4

4 Via Shielding to a Net 5

4.1 Adding Via Shielding 5

4.2 Modifying the PCB for Optimized Shielding Result 8

5 Via Stitching 9

5.1 Adding Stitching Vias to Net and Area 9

5.2 Removing Stitching Vias 12

5.3 Adding Stitching Vias to a Net for the PCB 13

Via Stitching and Via Shielding

1 Purpose

Via stitching is a technique used to tie together larger copper areas on different layers, resulting in a strong vertical connection through the board structure. Via stitching helps maintain low impedance and short return loops.

Via shielding is used in combination with guard rings to create a via wall, helping create an electromagnetically 'quiet' PCB. Also, you can use via shielding to tie areas of copper that might otherwise be isolated from their net, to that net.

2 Shortcuts

Shortcuts used when working with Via Stitching and Via Shielding

F1	Help
T » H » A	Stitching Dialog
T » H » R	Remove Via Stitching Group
CTRL+S	Save Document

3 Preparation

1. Close all existing projects and documents.
2. Next, create a Copy / Clone of the Training Project: `Via Stitching and Via Shielding`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button  .
5. Navigate to the predefined Training Project `Via Stitching and Via Shielding (Top\Projects\Altium Designer Advanced Training Course\...)`.
6. Select **Open Project as Copy...**  .
7. In the new dialog *Create Project Copy:*
   a) Add your name to the project name: `Via Stitching and Via Shielding – [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 Via Shielding to a Net

4.1 Adding Via Shielding

9. Open the `Via Shielding - Via Stitching.PcbDoc` document. The PCB has two Polygon Pours that are currently shelved.
   a) Open the *PCB Rules and Constraint Editor* (**D»R**) to check the two special rules added to the PCB. You can find the two rules in the branch *Routing – Routing Via Style* with the two rules Shielding Vias and Stitching Vias.
   b) Close the *PCB Rules and Constraint Editor*.
10. Next, you will add a shielding to two groups of nets, with two nets each.
11. For the first group (**NetR5_2 and NetR6_2**), use the following commands:
    a) Jump to Pad U1-34 with the Jump - Pad command (**J»P**). Add `U1-34` in the *Jump* dialog and start the jump with **OK**.
    b) From the **Edit** menu, click on **Select**, then select **Connected Copper** or use the key combination **S»P.**
    c) Select the Pad U1-34. The Pad U1-34, the tracks and the pads with the assigned net NetR6_2 are now selected.
    d) Press and hold **Shift** key. Select the Pad U1-33. Pad U1-33, tracks, and pads with the assigned net NetR5_2 are now selected. Use Figure 1 as reference.

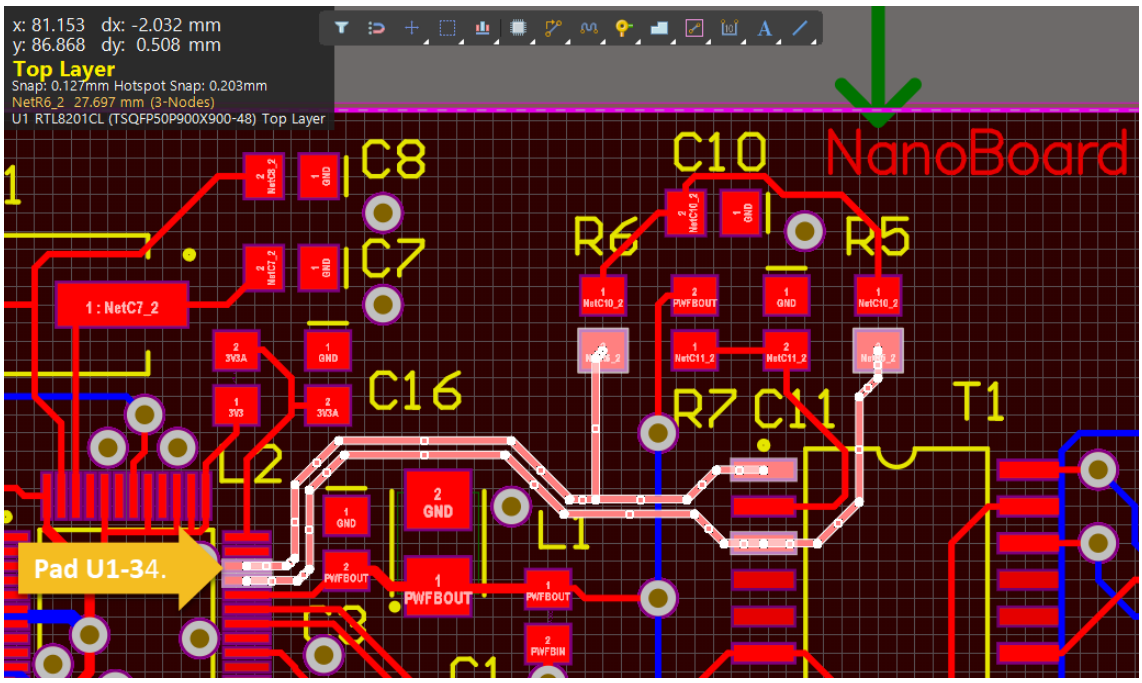


Figure 1. Select net for shielding

12. From the **Tools** menu, select **Via Stitching/Shielding » Add Shielding to Net...**
13. Use Figure 2 below as a reference to change the following values:
    a) Check that the checkbox for Selected **Objects** is ticked.
    b) Change the *Row spacing* to `0.2mm` and *Distance* to `0.3mm`.
    c) Change the *Grid* to `0.3mm`.
    d) Ensure *Rows* is set to `1`.
    e) At the moment, the **Add Shielding Copper** option is disabled. This could create polygon pours where vias exist in the via shielding trail.
    f) Instead of adding user values, you will load the Via size by clicking **Load values from Routing Via Style Rule**. The Values for the Via Hole Size and Via Diameter are updated to 0.2mm and 0.5mm.
    g) Set the Net used for shielding to GND.
    h) Review all of your changes and select **OK** when done.

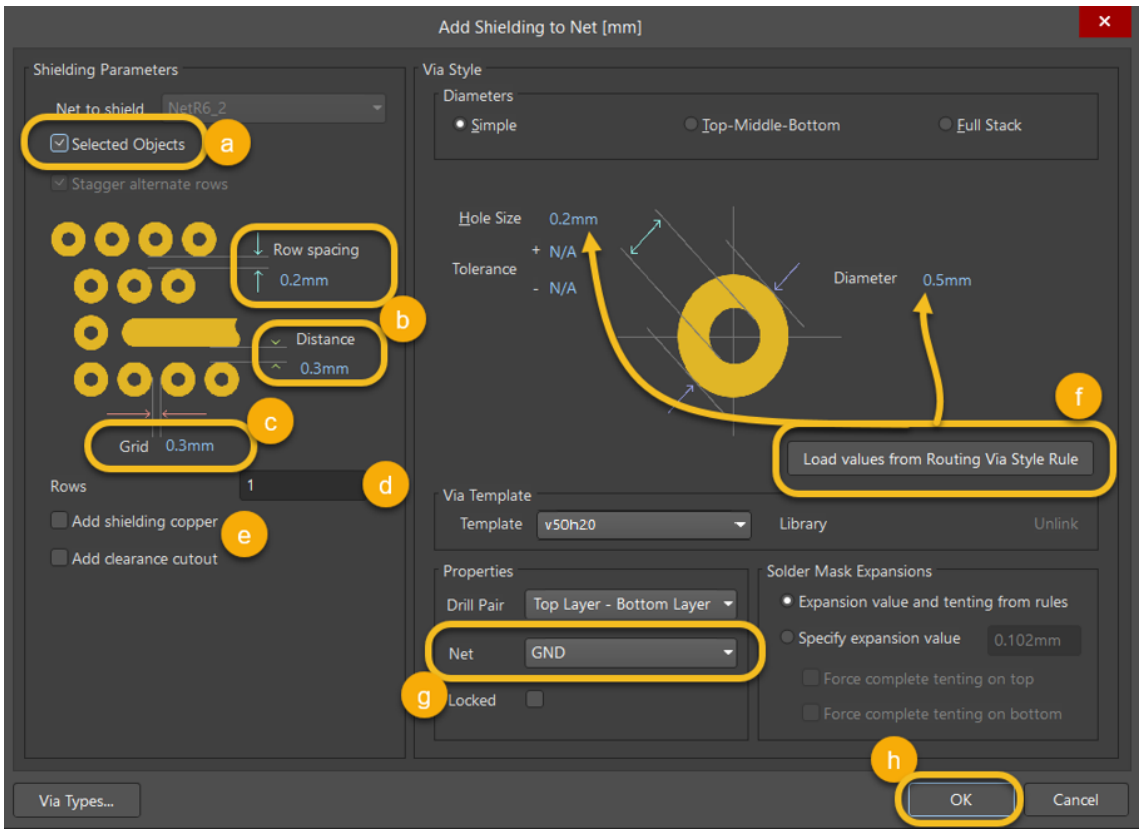


Figure 2. Via Shielding properties

14. The via shielding is added to that net, as shown in Figure 3.

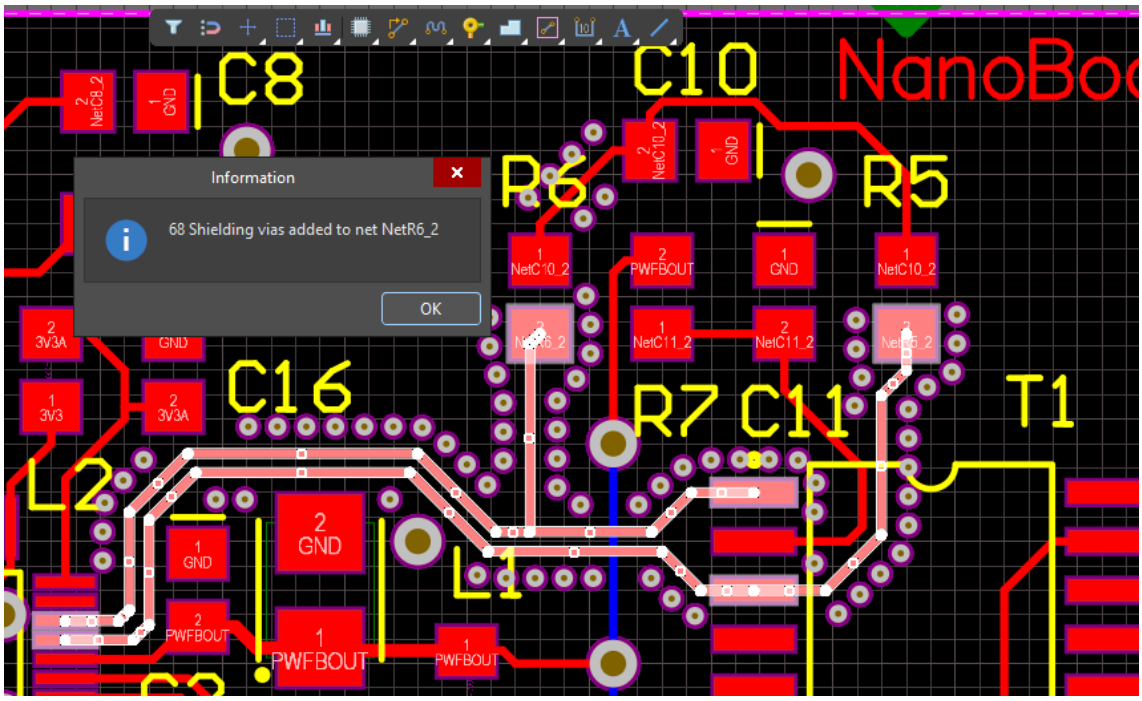


Figure 3. Via Stitching added to NetR5_2 and NetR6_2

15. Select **OK** in the *Information* dialog window.
16. For the second group (**NetR2_1 and NetR3_1**), use the following commands:
    a) Open the PCB panel.
    b) Select the view *Nets* and activate the option *Select*.
    c) Select the Net class Shielding 2. All Pad, tracks from the net **NetR2_1 and NetR3_1** are selected**.**

Note: To make the selection easier, a Net Class has been created.

17. From the **Tools** menu, select **Via Stitching/Shielding » Add Shielding to Net…**
18. Using the same settings as for the first group, create the shielding for net NetR2_1 and NetR3_1, but now add the option to **Add Shielding Copper**. Use Figure 4 as reference.

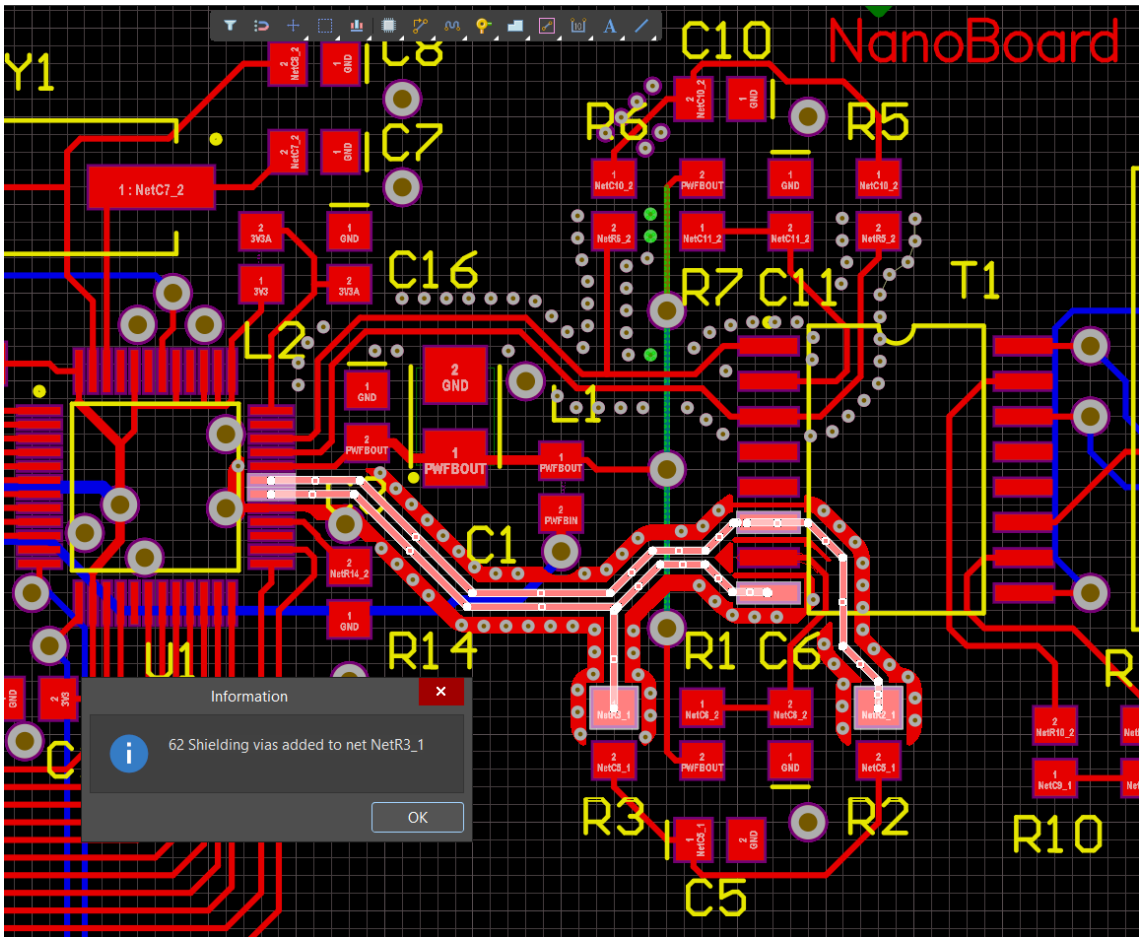


Figure 4. Shielding for NetR2_1 and NetR3_1

19. Select **OK** in the *Information* dialog window to continue.

4.2 Modifying the PCB for Optimized Shielding Result

Based on the result of the generated shielding, you can decide to optimize the existing routing for a better shielding result, or modify the shielding options, for example, Change Distance, Change Grid, and so on.

20. To modify the existing shielding parameters, select the shielding in the PCB and change the parameters in the *Properties* panel.
21. To delete the existing shielding, from the **Tools** menu, select **Via Stitching/Shielding » Remove Via Shielding Group …** and click in a shielding via. Vias that are part of a shielding group will have a `[VSH]` identifier on the via itself.
22. To optimize the shielding, you can move individual vias to new positions or delete vias from the shielding group, for example., the vias that create DRC violations in your design.

5 Via Stitching

5.1 Adding Stitching Vias to Net and Area

Now, you will add a Stitching to your design. First, restore the shelved polygons.

23. Select **Tools » Polygon Pours » Restore 2 Shelved Polygon(s)**. The Polygons are recreated, and you will see the two Polygons: one on the Top Layer, one on the Bottom Layer.
24. From the **Tools** menu, select **Via Stitching/Shielding » Add Stitching to Net….**
25. In the *Add Stitching to Net* dialog, select the *Net* dropdown and select **GND**, Figure 5.

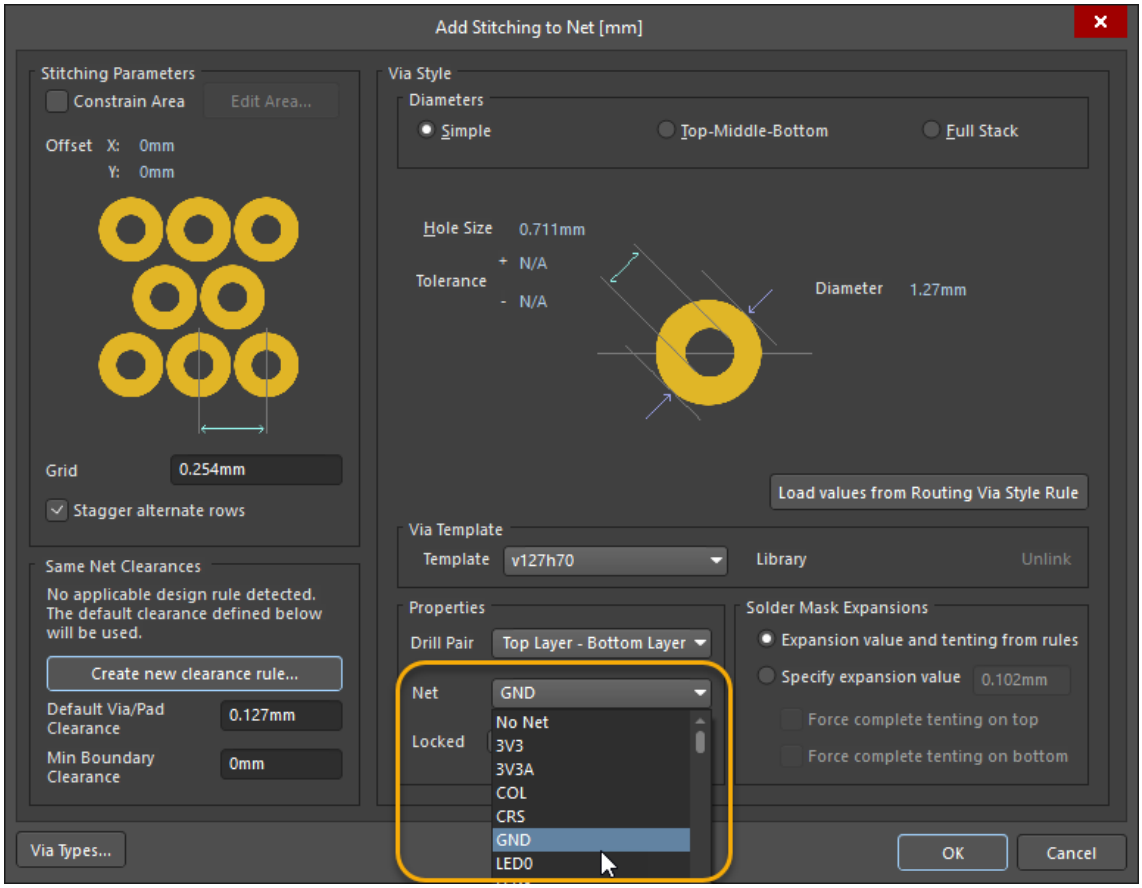


Figure 5. Adding Stitching to Net Menu

26. Next, you will use the *Constrain Area* to add stitching to a portion of the polygon. Otherwise, the entire polygon would be used.
    a) Select the **Constrain Area** checkbox in the top left corner of the Stitching window, Figure 7. This will change focus to the PCB with a crosshair on your cursor.
    b) Click on each of the four arrows, as shown in Figure 6 below. Feel free to change the Grid, key **G**, if needed.

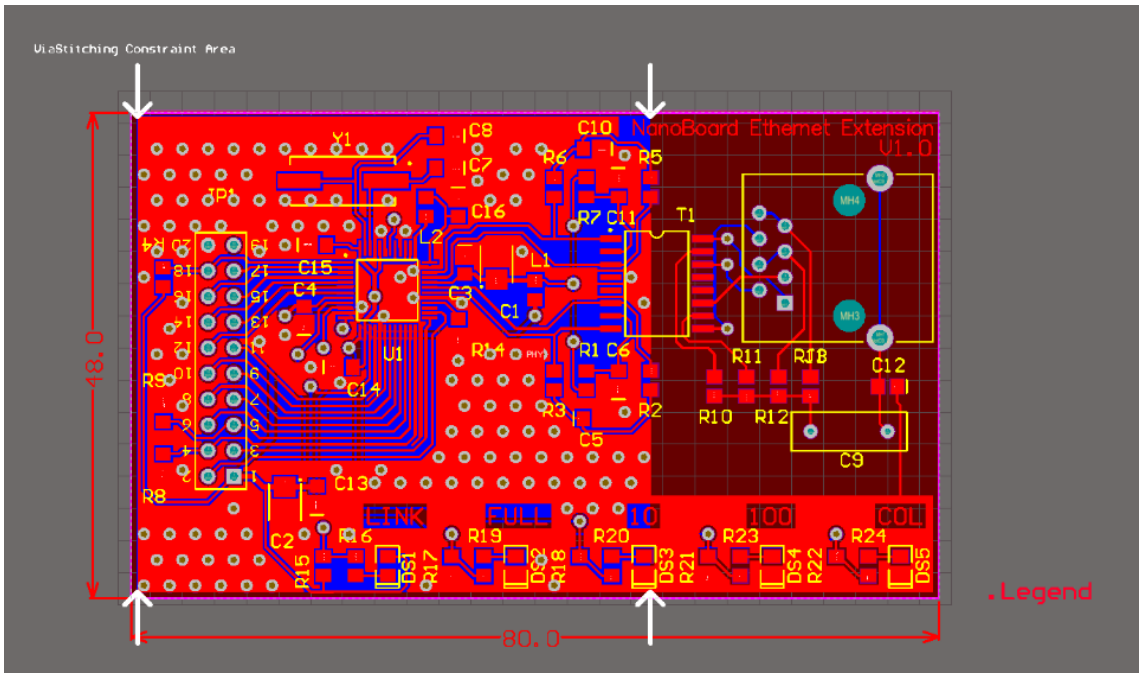


Figure 6. Via Stitching area

Hint: You can also change the color of the four arrows for better visibility. To do that, open the *View Configuration* panel, navigate to **Mechanical Layers (M)** section, select **Via Stitching Constraint Area,** and change the color to the one you like.

27. Once the area has been defined, hit the **ESC** or **right click** key to return to the *Add Stitching to Net* dialog. Use Figure 7 as a reference for the next steps:
    a) In the *Stitching* dialog window, change the **Grid** size from `0.254mm` to `2.54mm`.
    b) Instead of adding user values, load the Via size by clicking **Load values from Routing Via Style Rule**. The values for the Via Hole Size and Via Diameter are updated to 0.3mm and 0.7mm.
    c) Enable the **Specify expansion value** radio button to force tenting on the `Top` and `Bottom` of each via.
    d) Check the **Force complete tenting on top** and **bottom** checkboxes under the *Solder Mask Expansions* area to have the added stitching vias covered over with Solder Mask, as shown in Figure 7.

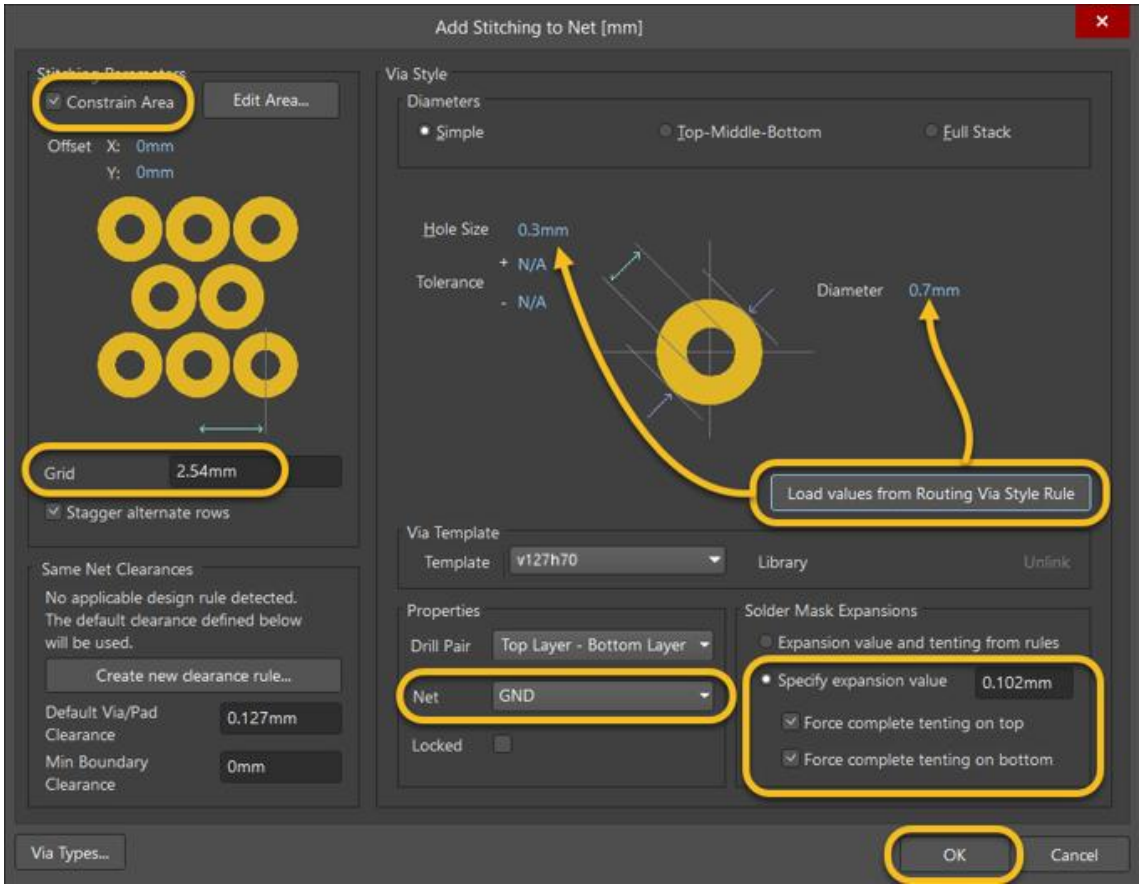


Figure 7. Via hole and tenting values

e) Select **OK** to generate the Via Stitching. An *Information* dialog box will appear, stating the number of vias that have been added, as presented in Figure 8 for illustration purposes.

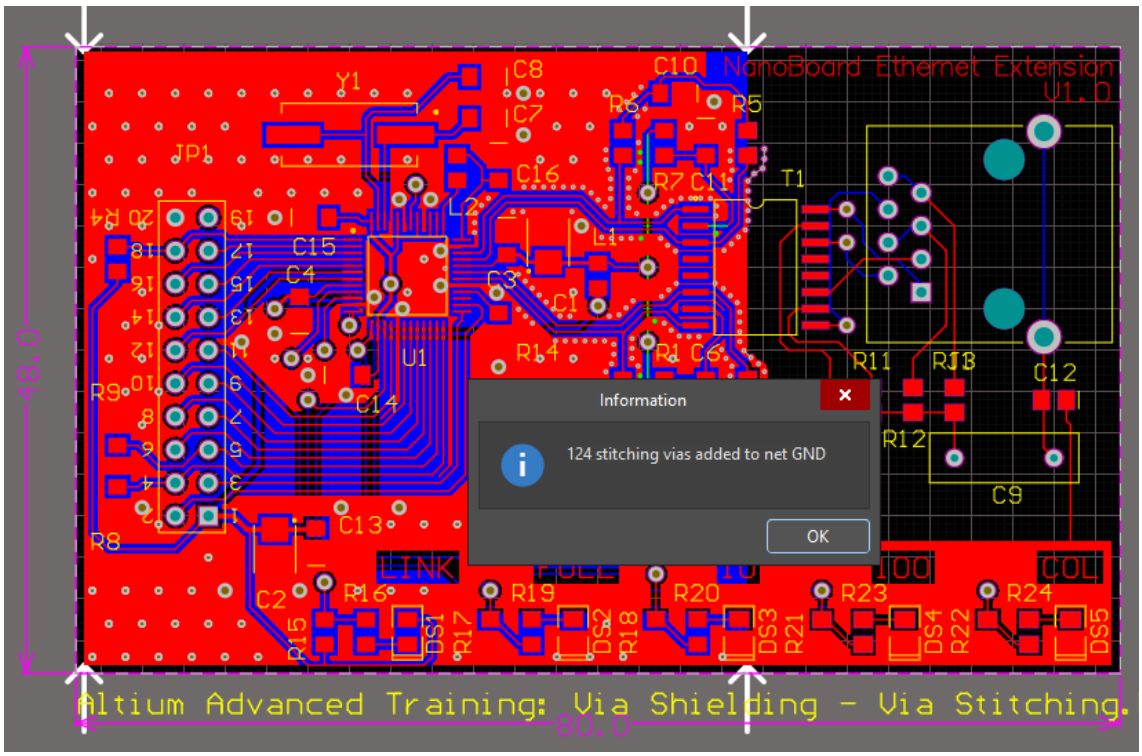


Figure 8. Stitching Vias added

28. Select **OK** on the *Information* dialog box.

5.2 Removing Stitching Vias

Next, you will remove the Stitching Vias to recreate them with different parameters.

29. Remove the via stitching using **Tools » Via Stitching/Shielding » Remove Via Stitching Group**.
30. Once the cursor turns into a crosshair, select one of the vias in the stitching group to remove the entire group. Vias that are a part of a stitching group, will have a `[VS]` identifier on the via.
31. After clicking on one of the stitching vias, the stitching group is removed.

5.3 Adding Stitching Vias to a Net for the PCB

32. From the **Tools** menu, select **Via Stitching/Shielding » Add Stitching to Net…**
33. Using the same information as before (Net, Via Size, Grid, and so on), create a stitching for the Board by checking that the **Constraint Area** isn't ticked.
34. The tool reports 124 Vias.
35. Select **OK** on the *Information* dialog box.

Hint: There aren't any Stitching Vias added at the lower right side of the Top Layer Polygon. This is because in this board area, there's copper from a Polygon Pour on Top Layer but no copper from a Polygon Pour on Bottom Layer.

36. In the next step, you will modify the via stitching to add an offset to the via location.
    a) Click on a Stitching Via within any part of the constraint area, where the Via Stitching exists. This will populate the *Properties* panel with the *Via Stitching* information.

Hint: If you have problems to select the Stitching Area, try the following solutions:
- Double-click on a Stitching Via to open a selection box of possible objects.
- Change the selection filter. Deactivate all options and just keep the **Other** option active.

    b) In the *Offset (X/Y)* area of the *Properties* panel, change the X value to `-0.4mm`.
    c) Once you press **Enter**, you'll be prompted to apply the new changes, Figure 9.

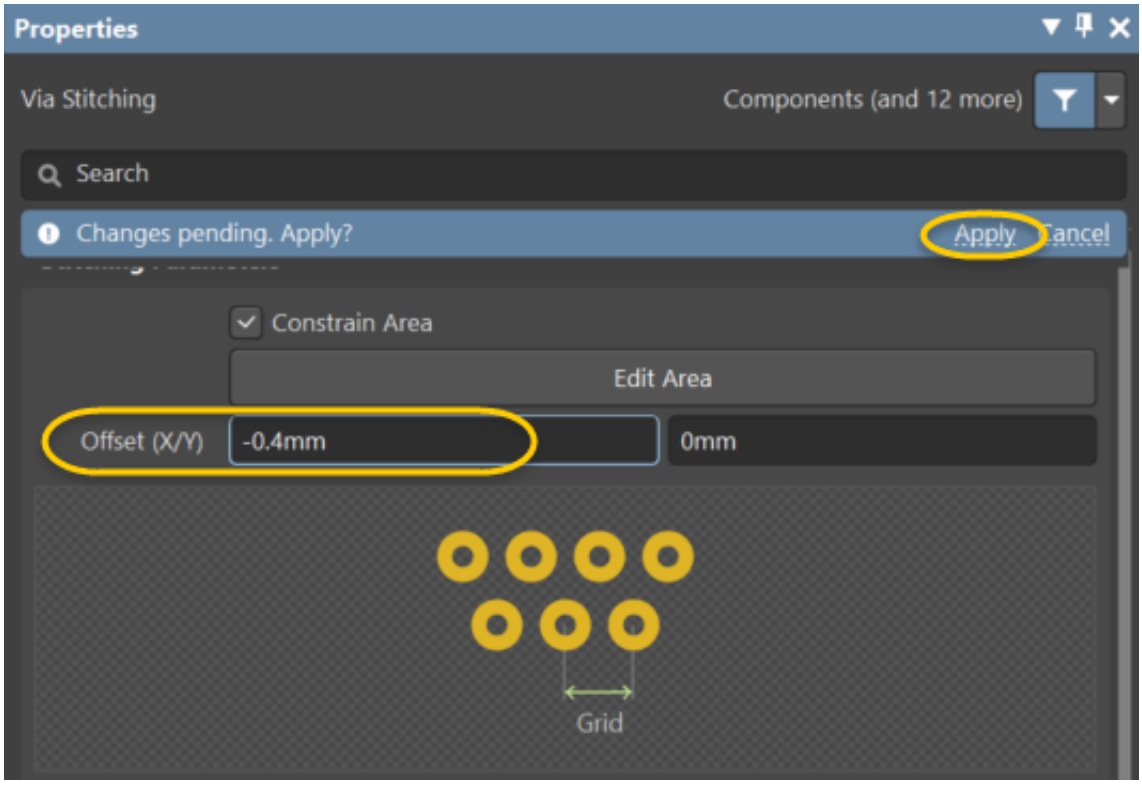


Figure 9. Changing the Via Stitching properties

    d) Select the **Apply** button that appears at the top of the *Properties* panel.

e) A new *Information* dialog appears, displaying the new number of vias added to the stitching area, as shown in Figure 10.

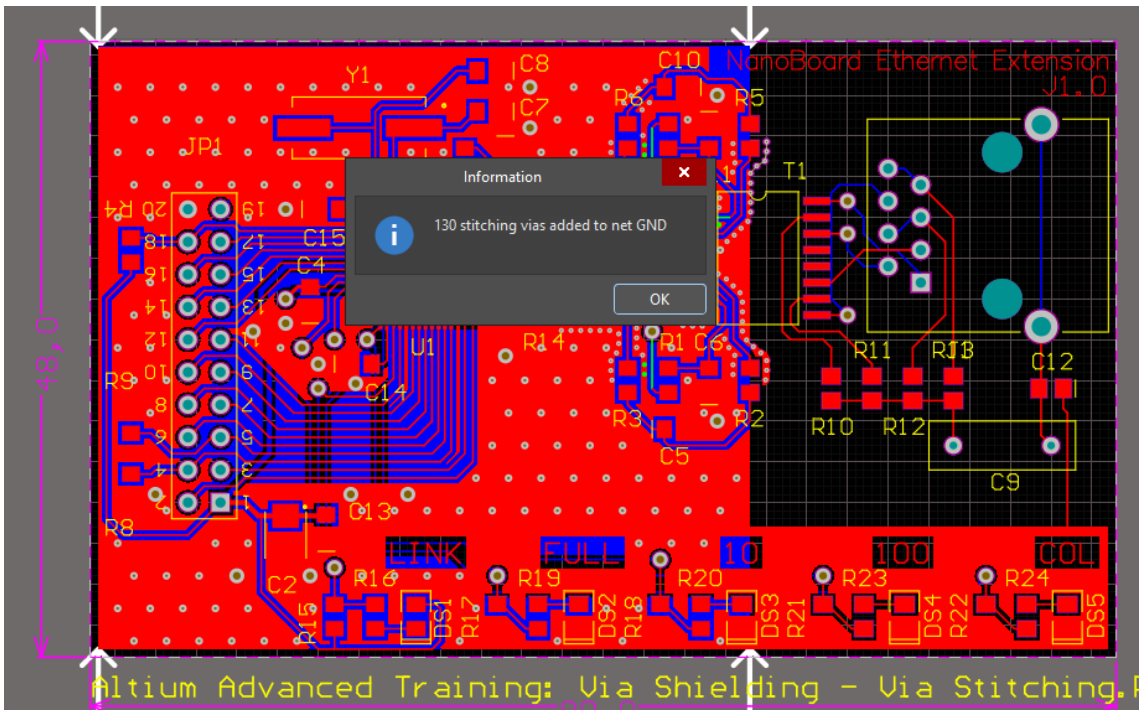
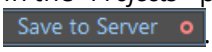


Figure 10. New Information dialog with added vias

37. Select **OK** in the *Information* dialog box to continue.
38. Save all documents using **File » Save All**.
39. Save the modifications to the server:
    a) In the *Projects* panel, next to the Project name you find the command **Save to Server**  .
    b) Select **Save to Server**.
    c) In the dialog *Save [Project Name]:*
        i) Add the comment `Via Stitching and Via Shielding - [Add Your Name] - Finished`.
        ii) Select **OK**.
40. When ready, close the project and any open documents, **Window » Close All**.

Congratulations on completing the Module!

Via Stitching and Via Shielding

from

Altium Designer Advanced Training with Altium 365

Thank you for choosing **Altium Designer**