

ALTium **365**

Altium Designer

Essentials Course - Altium 365

Module 23: PCB Polygon Pour and Planes

Software, documentation and related materials:

Copyright © 2024 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.

ACTIVEBOM®, ActiveRoute®, A365™, Altium 365®, Altium Concord™, Altium Concord Pro™, Altium Designer®, AD™, Altium NEXUS®, Altium OnTrack™, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Common Parts Library™, Concord™, Concord Pro®, Draftsman®, Dream, Design, Deliver®, DXP™, Easytrax®, EE Concierge®, Fearless HDI™, Geppetto®, Gumstix®, Learn, Connect, Get Inspired™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, OnTrack™, Overo®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter®, X2®, XSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its affiliated companies. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

Table of Contents

Module 23: PCB Polygon Pour and Planes	3
1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	4
1.4 Preferences	5
1.5 Configure Snap and Grid	6
1.6 Placing a Polygon Pour	7
1.6.1 Manually Creating the First Polygon Pour	7
1.6.2 Polygon Properties.....	9
1.6.3 Create a Polygon Pour Cutout (Optional).....	10
1.7 Polygon Manager	11
1.7.1 Flooding a Layer	13
1.8 Editing Polygons	14
1.8.1 Editing an Edge	14
1.8.2 Adding a Vertex (Optional)	15
1.8.3 Deleting a Vertex	15
1.9 Controlling the Connection Style	16
1.9.1 Polygon	16

Module 23: PCB Polygon Pour and Planes

1.1 Purpose

In Altium Designer, polygons and planes are essential features used in PCB design to create copper areas, including ground planes and power planes. These features play a crucial role in determining the layout and connectivity of your printed circuit board (PCB).



In this exercise, students will learn to place and edit polygons pours and power planes.

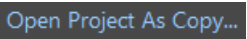

1.2 Shortcuts



Shortcuts when working with Module 23: PCB Polygon Pour and Planes

P » G:	Place polygon pour
Shift+Space:	Change Corner Mode (Drawing Polygon Shape)
Space:	Elbow side (Drawing Polygon Shape)
T » G » M:	Polygon Manager
T » G » H:	Shelve All Polygons
T » G » E:	Restore All Shelved Polygons
T » G » A:	Repour All Polygon
Shift+S:	Single layer Mode
Shift+H:	Toggle Heads Up visibility
F5:	Activate / Deactivate Color Overlay (SCH, PCB)
V » F:	Fit Board



1.3 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project Module 23 PCB Polygon Pour and Planes.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project Module 23 PCB Polygon Pour and Planes (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 name: Module 23 PCB Polygon Pour and Planes - [Your Name].
 - b) Add a description: Altium Essential Training - Module 23 - [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.

1.4 Preferences

8. Open the Module 23 PCB Polygon Pour and Planes.PcbDoc.
9. The PCB is preconfigured.
 - a) The Selection Filter is set to Polygons .
10. Open the Preferences  and navigate to *PCB Editor* branch and open the *General* page.
11. Disable both **Repour Polygon** options under the *Polygon Rebuild* section as shown in Figure 1. Click **OK** to accept these changes.

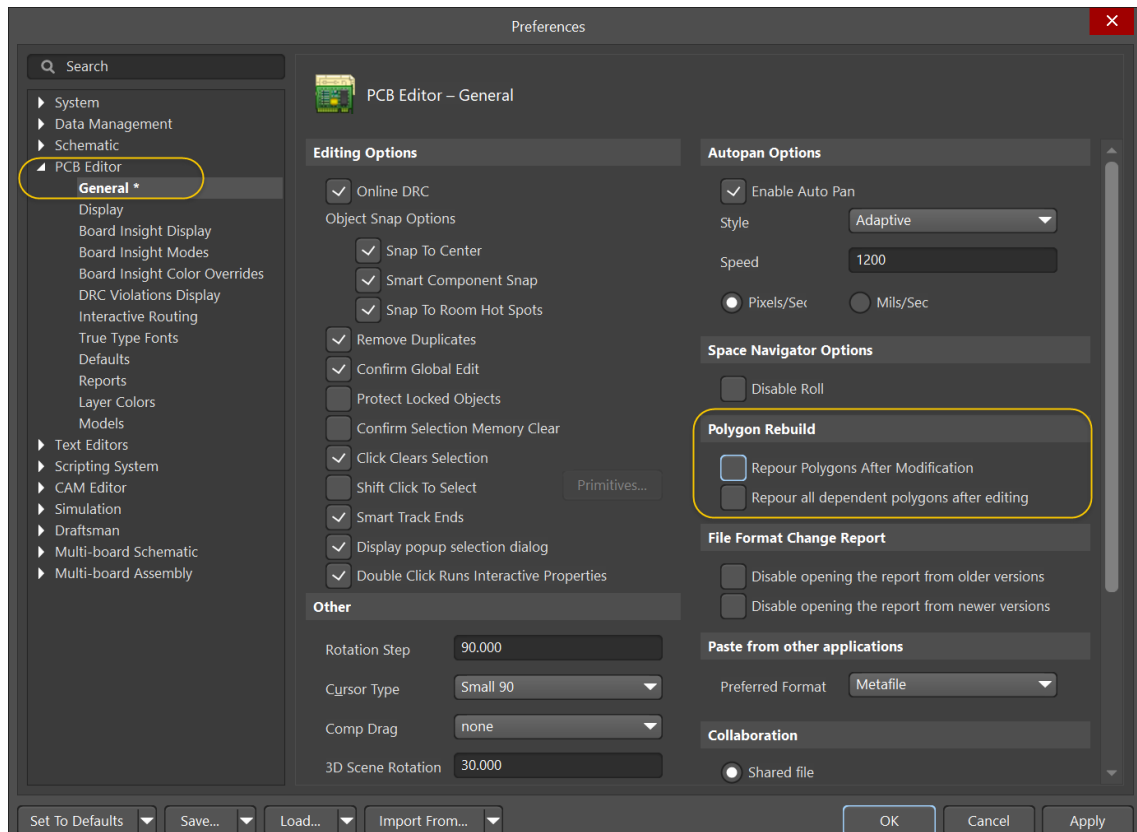



Figure 1. Disable Polygon Rebuild options



If you would like the polygons to be repoured automatically in the future, you would enable these two options. For this exercise, we would like to showcase the repour options directly from the Properties panel / menu.

1.5 Configure Snap and Grid

12. We will be placing a polygon on the bottom layer. If not already the active layer, click on the **Bottom Layer** in the layer bar at the bottom of the PCB editor to make it the active layer.
13. To view the entire board area, use the **V » F** shortcut keys.
14. With nothing selected, open the *Properties* panel and expand the *Snap Options* section or press **Ctrl+E**.
 - a) In the *Snap Options* section, activate **Axes** .
 - b) In the *Objects for Snapping* section just below, enable the **Board Shape** option, both of these options can be found in Figure 2.

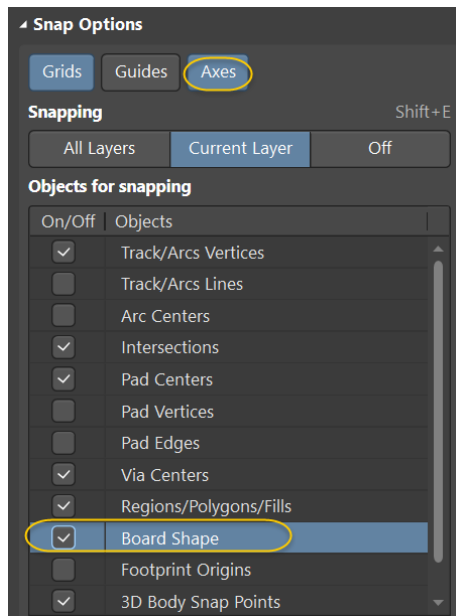



Figure 2. Snapping options in the Properties panel

15. Set the grid spacing to 10mil by pressing the **G** key and selecting 10mil. Ensure to be on imperial units as indicated on the status bar. The **Q** key will allow you to toggle between metric and imperial units on the fly.
16. If it is not visible, enable the *Heads-Up Display (HUD)* by pressing **Shift + H** as shown in Figure 3 below. You can change HUD window opacity and options in  *Preferences*, under the *PCB Editor* branch, on the *Board Insight Modes* page.

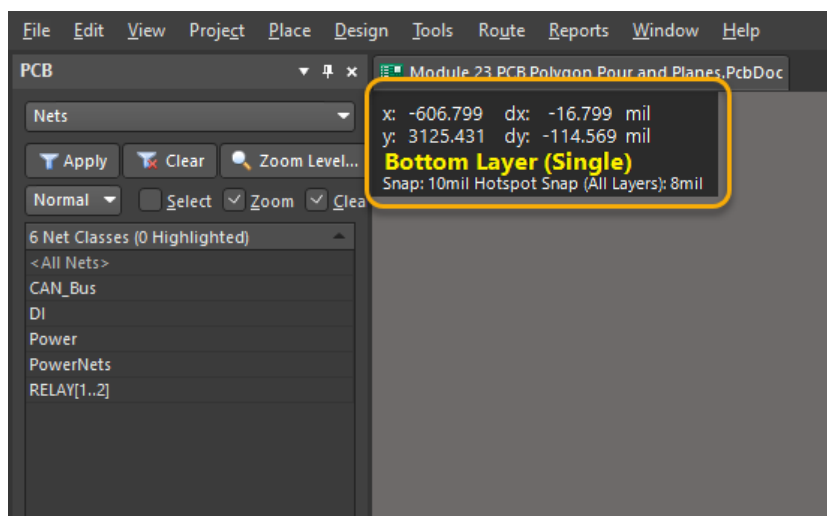


Figure 3. Heads Up Display toggled with the Shift+H keys

1.6 Placing a Polygon Pour

1.6.1 Manually Creating the First Polygon Pour

17. From the **Place** menu, select **Polygon Pour**. You can also use the **P » G** shortcut keys.
18. Press the **Tab** key to pause the command and open the *Properties* panel to access the *Polygon Pour* dialog as shown in Figure 4.
19. Be sure to assign it to the **5V** net and make sure it's on the **Bottom Layer**.
20. Near the bottom of the panel, below the preview of the polygon, select **Pour Over All Same Net Objects** from the drop-down menu as shown in Figure 4.

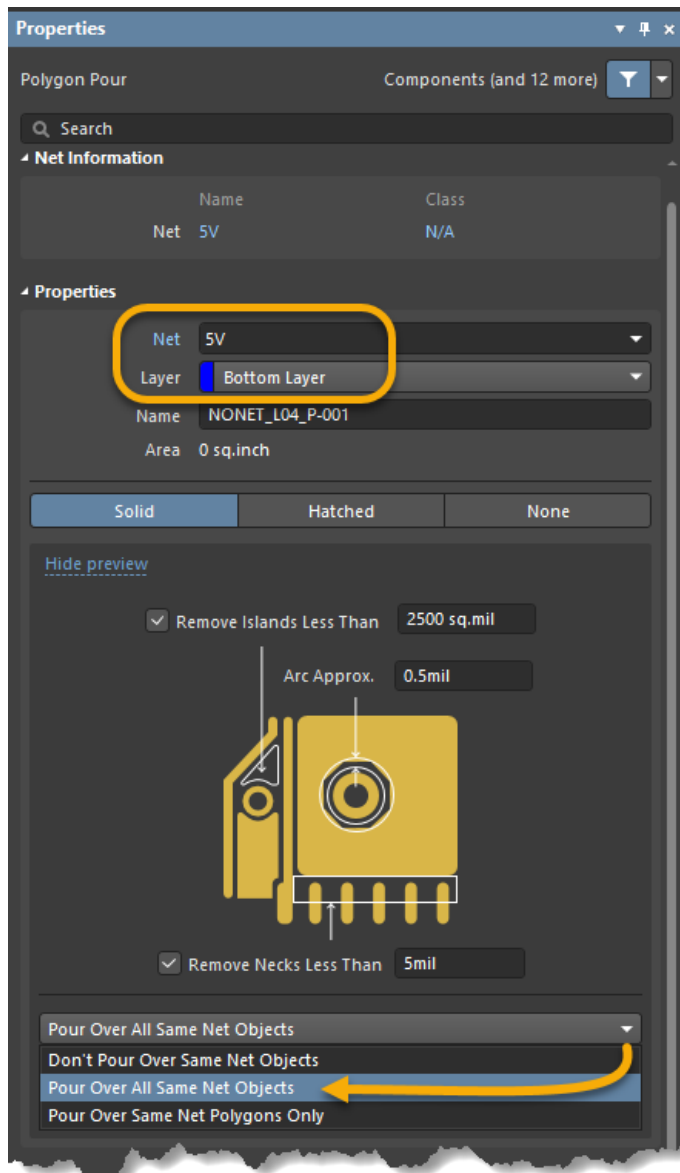



Figure 4. Polygon Pour dialog

21. Click on the pause symbol  to continue placing the polygon. Your cursor will turn into a crosshair indicating that you may begin drawing the polygon. You may want to work in **Single Layer Mode (Shift+S)** to only show the bottom layer while drawing the polygon.
22. Click on the board origin in the bottom left corner and the *HUD* should indicate the co-ordinates (0,0) as shown in Figure 5.

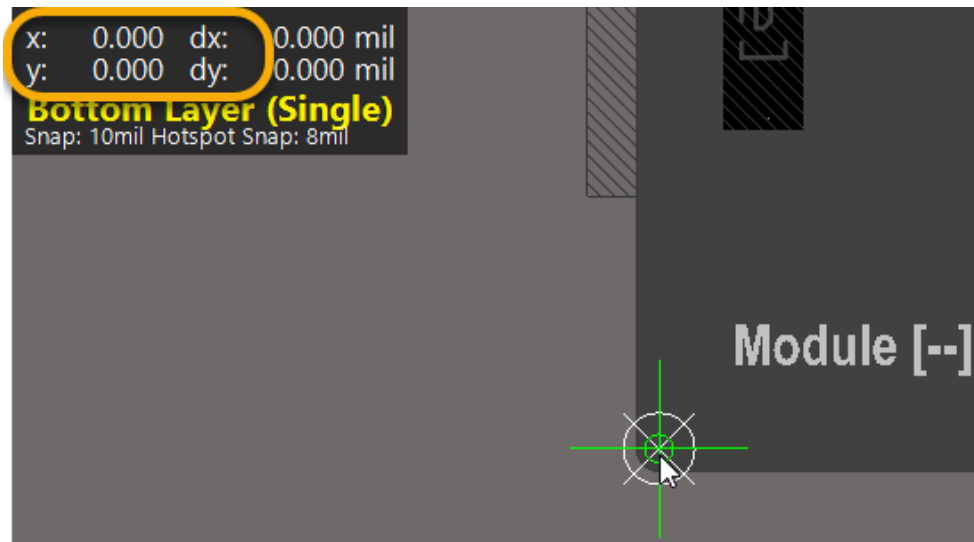


Figure 5. Heads-Up Display (HUD) showing (0,0) coordinates

23. Hit **Shift + Spacebar** until the *HUD* indicates *Line 90/90 Vertical* or *Horizontal Start* to provide a right angle for the polygon. Hitting the **Spacebar** will toggle between *Vertical* and *Horizontal* angles.
24. Draw a rectangular polygon similar to what is shown in Figure 6. Ensure that the polygon ends at the origin where you initially started the polygon.
The high of the polygon is 1550mil.
25. Right-click twice to end the command once you've completed drawing the polygon.

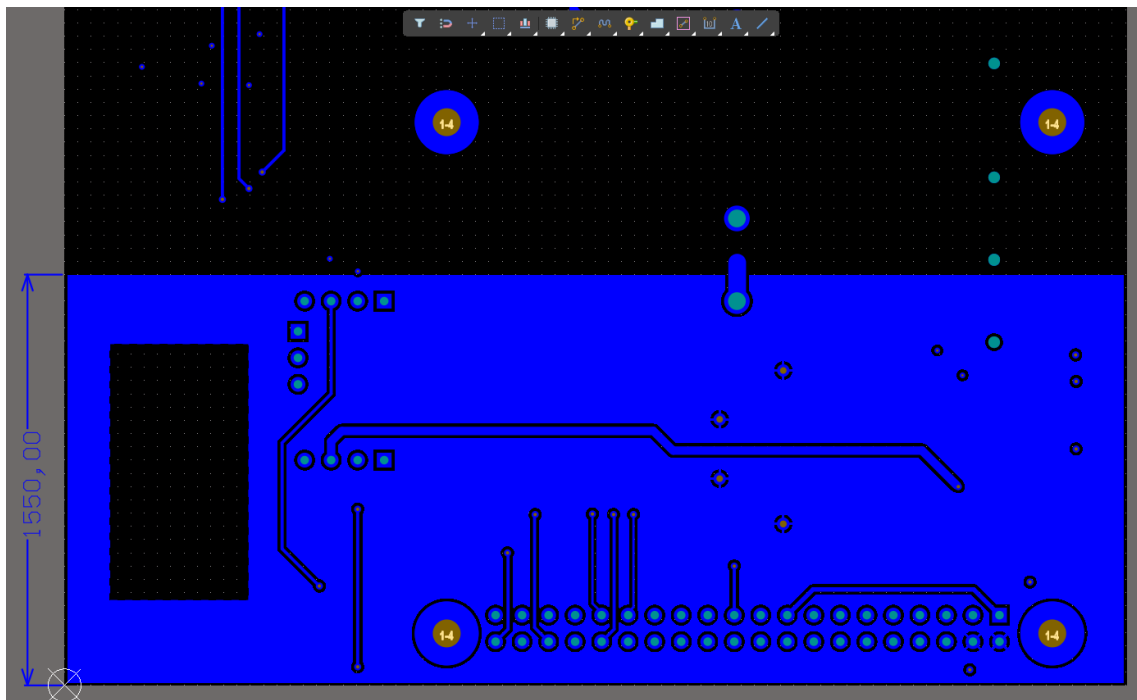


Figure 6. Polygon Pour added to the PCB, with Polygon Cutout on the left side.

1.6.2 Polygon Properties

26. Double-click on the newly created polygon on the Bottom Layer to open the *Properties* panel.



Depending on the Preference setting **Double Click Runs Interactive Properties** (PCB Editor – General) instead of the Properties panel the Component dialog could be opened.

27. Change the *Fill Mode* to **Hatched** as shown in Figure 7. After changing it to **Hatched**, the polygon will need to be repoured.
28. The repour dialog will be shown at the top of the *Properties* panel, as shown in Figure 7. Click the **Repour** option as shown. The polygon can also be repoured by going to **Tools » Polygon Pours » Repour All**.

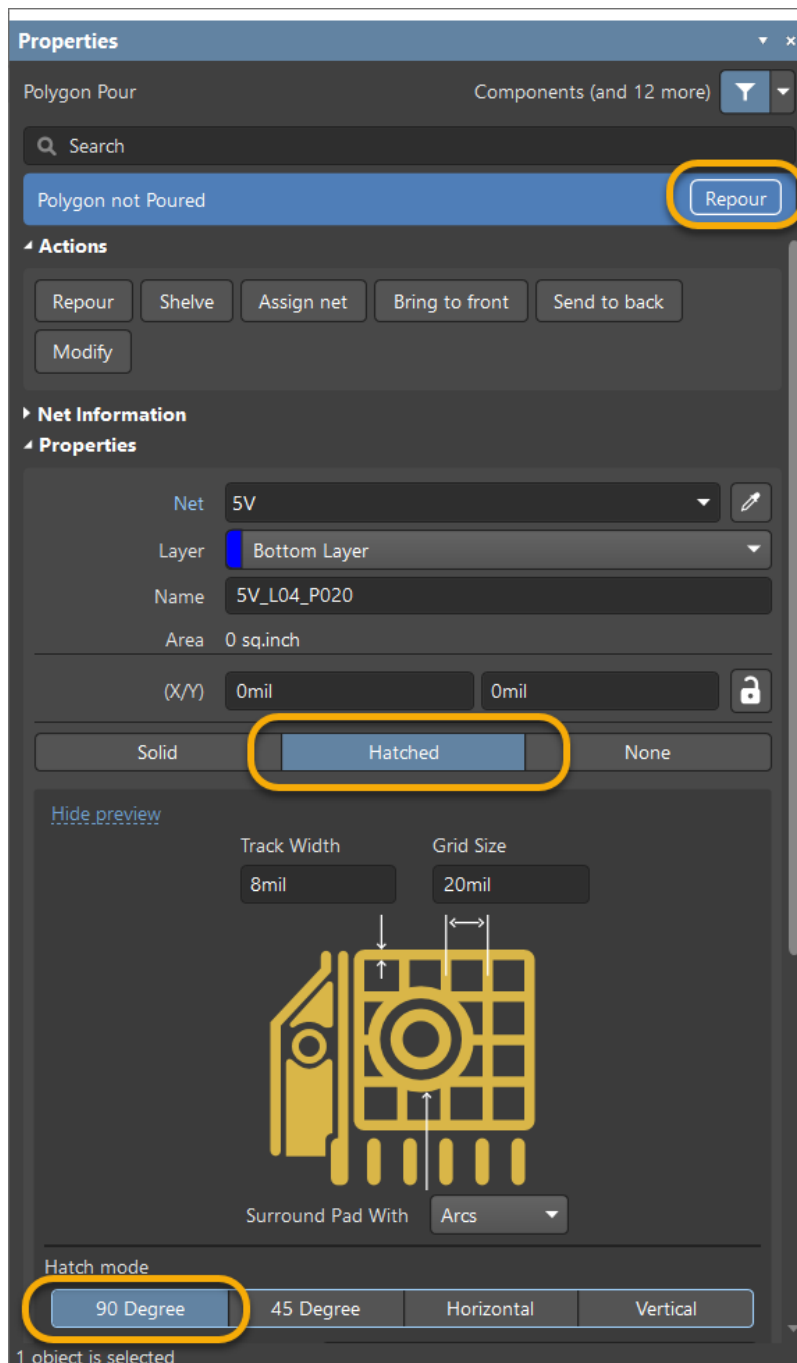


Figure 7. Changing the Polygon fill mode

1.6.3 Create a Polygon Pour Cutout (Optional)

29. From the **Place** menu, select **Polygon Pour Cutout**.
 - a) Using Figure 8 as a reference, draw a rectangular **Polygon Pour Cutout** on the left side of the polygon that we just created.
 - b) Once defined, right click to terminate the polygon outline and right-click once more to exit polygon pour cutout placement mode.
 - c) Repour the polygon to see the cutout by going to **Tools » Polygon Pours » Repour All**.

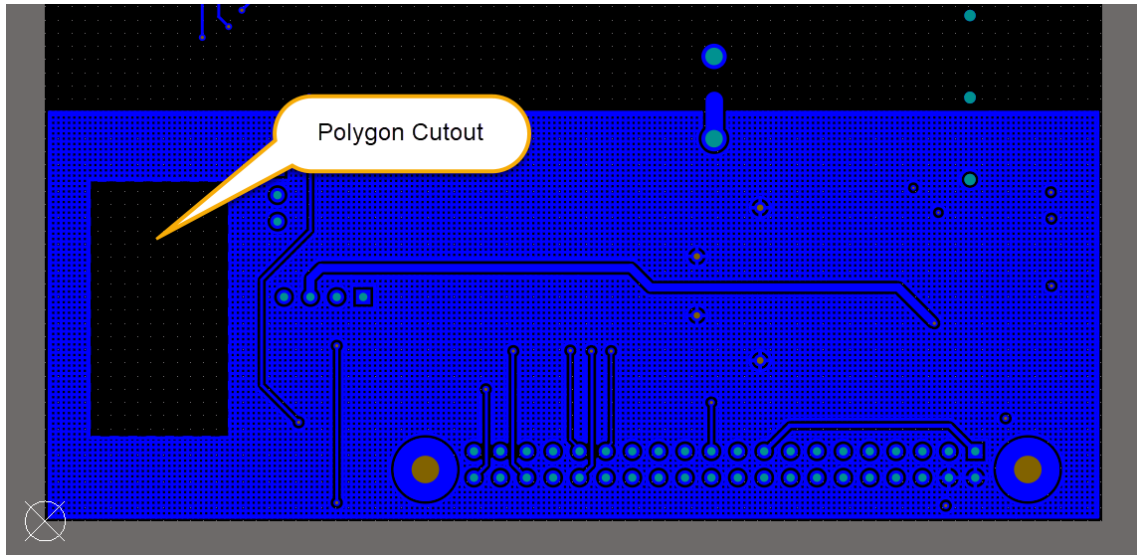



Figure 8. Polygon Pour Cutout

30. Open the  Preferences and navigate to the *PCB Editor* branch, and select the *General* page.
31. Enable both **Repour Polygon** options under the *Polygon Rebuild* section as shown in Figure 9.
32. Click **OK** to accept these changes.

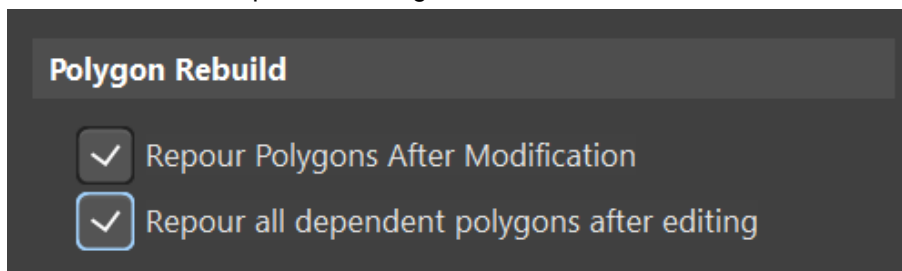


Figure 9. Enable Polygon Rebuild optionsPolygon Pour Manager

1.7 Polygon Manager

33. Click on the Top Layer from the layer bar to set it as the active layer. Notice the multiple polygon pours on this layer and the specific clearances as shown in Figure 10. With multiple polygons overlapping, pour order or priority must be defined.



Use **F5** to activate / deactivate the color overlay for the nets.

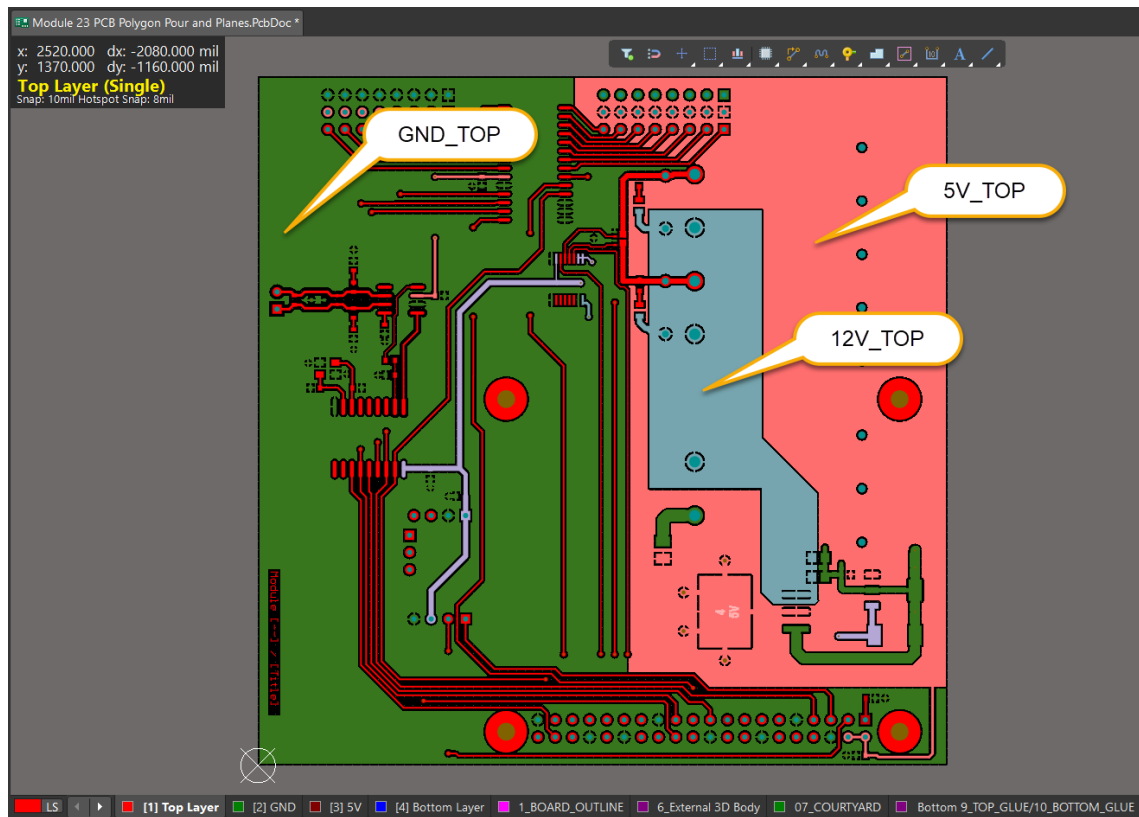


Figure 10. Polygon Pours on Top Layer

34. Open the *Polygon Pour Manager* by going to **Tools » Polygon Pours » Polygon Manager**. This is where you can manage all of the polygons in your design.
35. Within the *Pour Order* pane, we can change the pour order of the polygons. Select the 12V_Top polygon and click **Move Down** until it is moved to the bottom of the *Pour Order* list as shown in Figure 11.
36. Select **Apply** to observe the changes.



The polygon that is higher in the pour order will pour first, and the polygons lower in the list will then pour around existing polygons with necessary clearance.

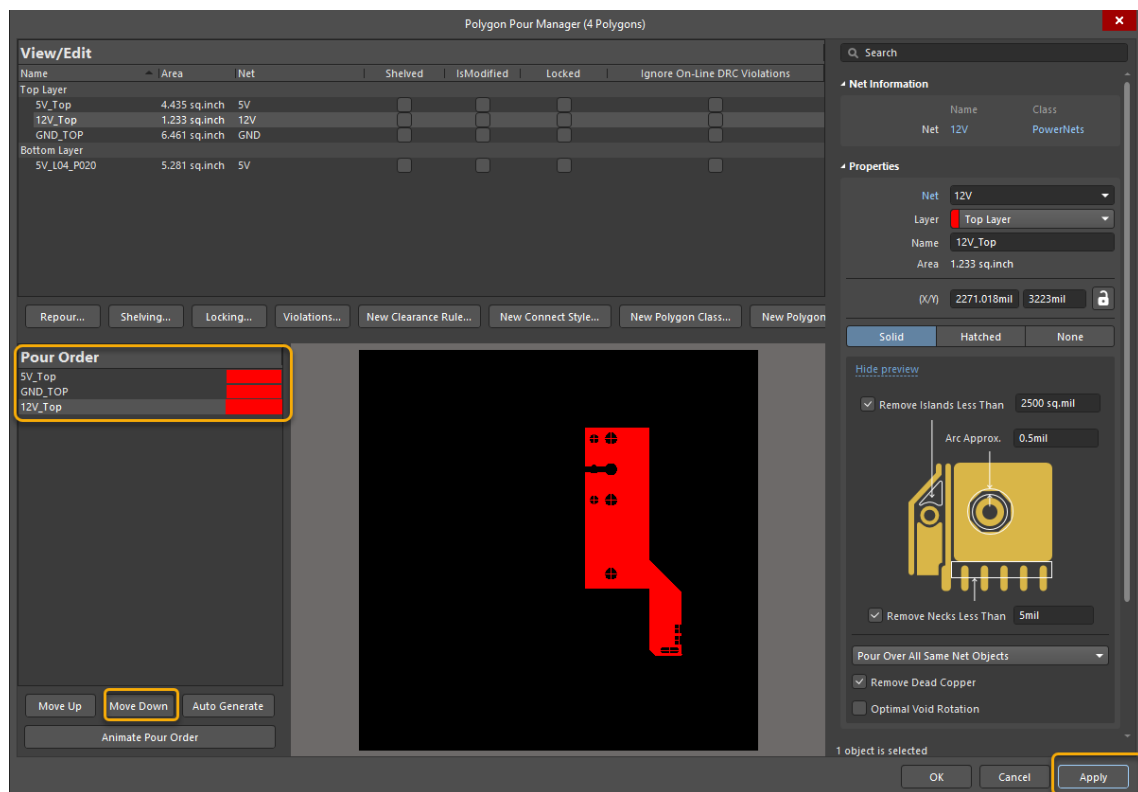


Figure 11. *Polygon Pour Manager*

1.7.1 Flooding a Layer

38. We will now create a polygon which floods an entire layer of the board. Go back to **Tools » Polygon Pours » Polygon Manager**.
39. Click on **New Polygon from...** then select **Board Outline** as seen in Figure 12. A new Polygon will be added in the Polygon Pour Manager.

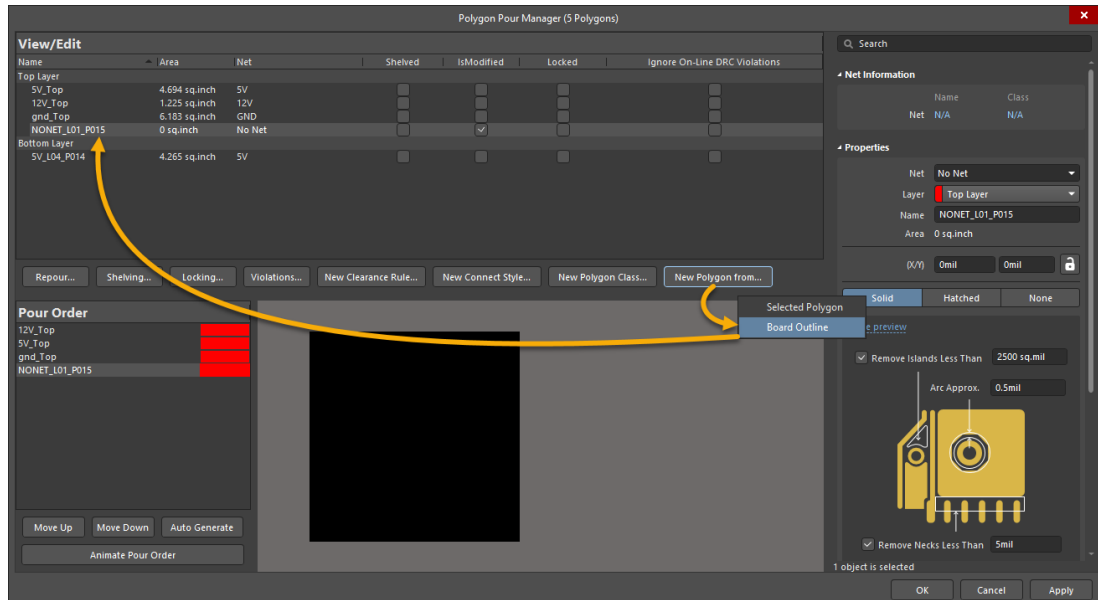


Figure 12. Creating a polygon from the board outline in the Polygon Manager

40. Match the settings of the newly created Polygon to what is shown in Figure 13. The shape of this polygon will be dependent on the first, Bottom Layer polygon that we created in a previous step. Your polygon may slightly vary from what is shown below.

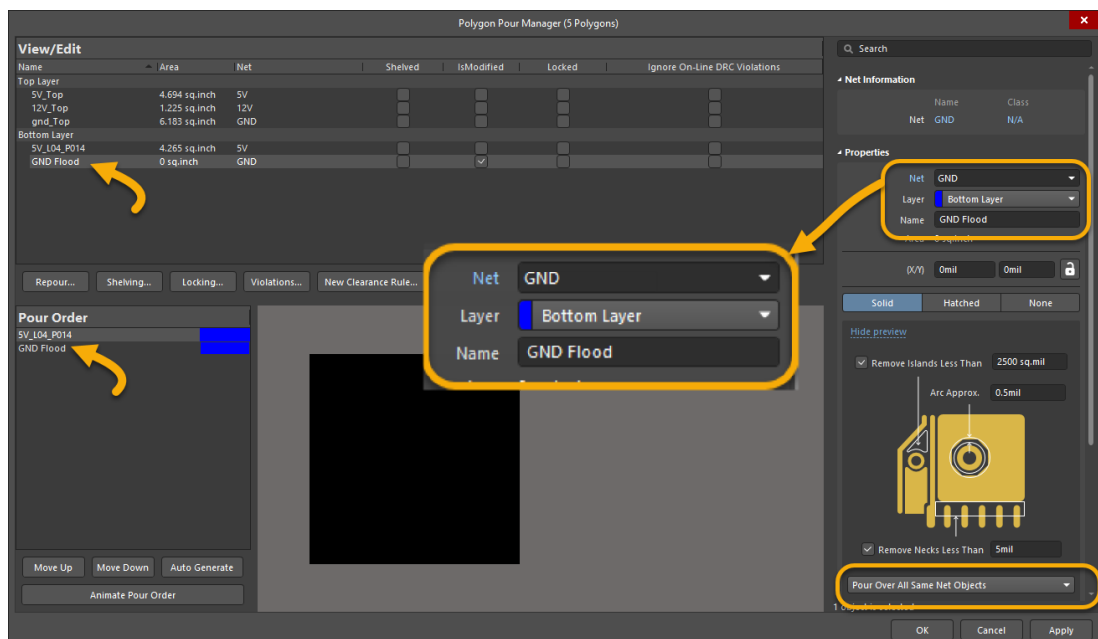


Figure 13. Polygon Pour dialog

41. Ensure that this newly created polygon is last in the pour order and click **OK** to close the *Polygon Manager*.
42. Your new polygon will now be flooded across the entire PCB. If you any receive green highlights showing violations, rebuild the polygon by going to **Tools » Polygon Pours » Repour All**.

1.8 Editing Polygons

1.8.1 Editing an Edge

43. In the PCB editor, make sure the **Bottom Layer** is still the active layer.
44. Navigate to the bottom of the board to the hatched polygon we created earlier in this exercise.
45. Left-click on the polygon to select it. If the first click selects the GND Polygon for the entire board, click again to select the smaller hatched polygon.
46. Hover over the right vertical edge of the polygon. The edge will be highlighted by a thick white line and the cursor will turn into a 2-way arrow as shown in Figure 14.

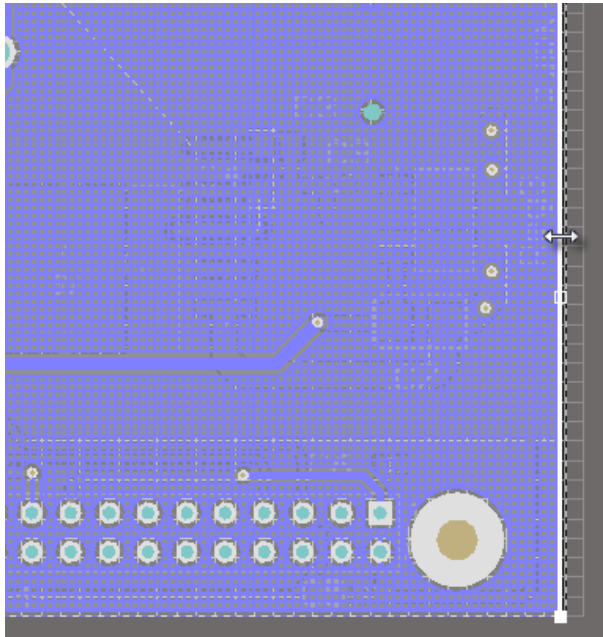


Figure 14. Hovering over the right edge of the polygon

47. Click and hold the left mouse button; the cursor will become a crosshair and the *HUD* will show that the *Sliding Edge* mode is active as shown in Figure 15.

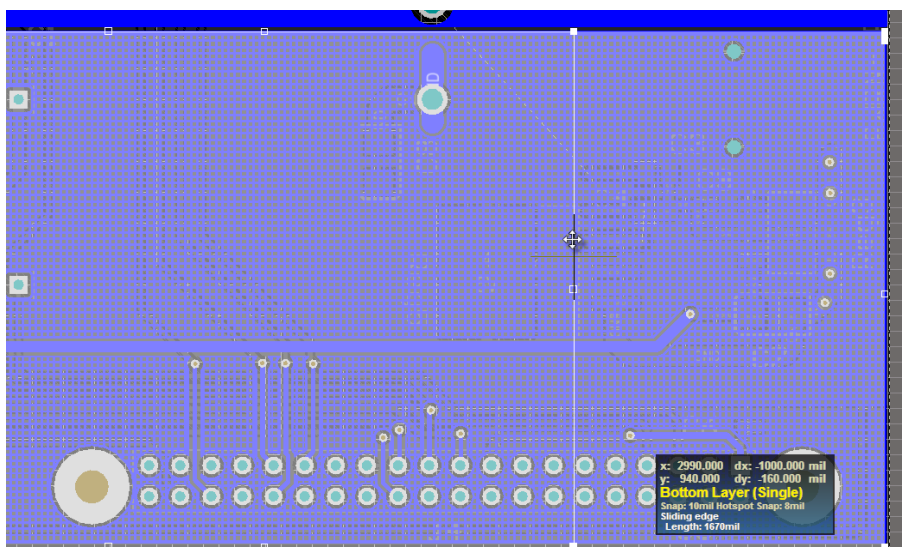


Figure 15. Moving the right edge of a polygon using the Sliding Edge mode

48. Slowly move the cursor to the left to make the polygon narrower, then release the mouse button. Notice that you are constrained to only the horizontal direction ensuring that the height of the polygon is unchanged.

1.8.2 Adding a Vertex (Optional)

You can add a vertex at any point on any edge of the polygon. Center vertices are also automatically added or removed as changes are made.

49. With the polygon still selected, hover over the top edge of the polygon so that it highlights with a thick white line.
50. While the edge is highlighted, hold the **Ctrl** key, and you'll notice a vertex box will appear. You may need to move your cursor a bit to see it.
51. While holding the **Ctrl** key, hold the left-click and move the mouse up. The new vertex will appear on your cursor as shown in Figure 16.
52. Let go of the left-click to place the new vertex and polygon outline at the desired location. You can also let go of the **Ctrl** key.

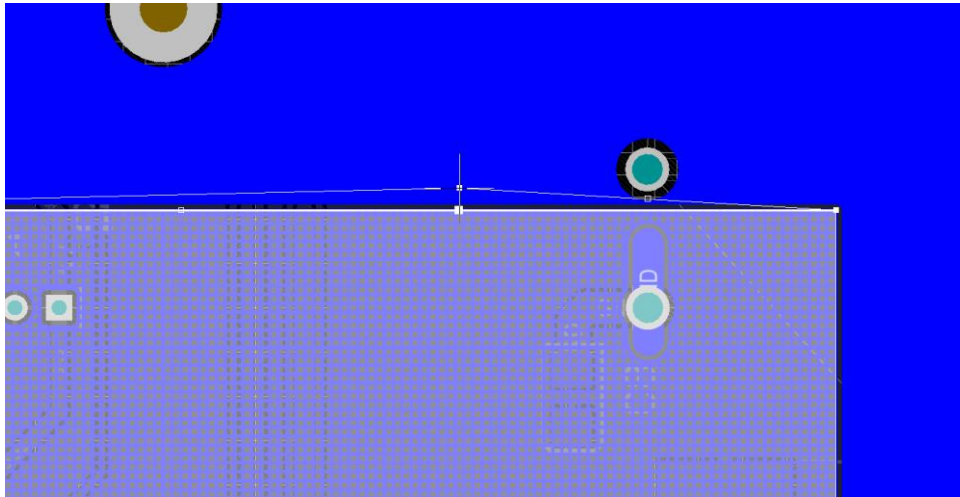


Figure 16. Adding a vertex to a polygon

1.8.3 Deleting a Vertex

53. Using the same polygon, click and hold a corner vertex. The cursor will turn into a crosshair, and the vertex can be removed with the **Delete** key, reshaping the polygon. An example of this is shown in Figure 17.

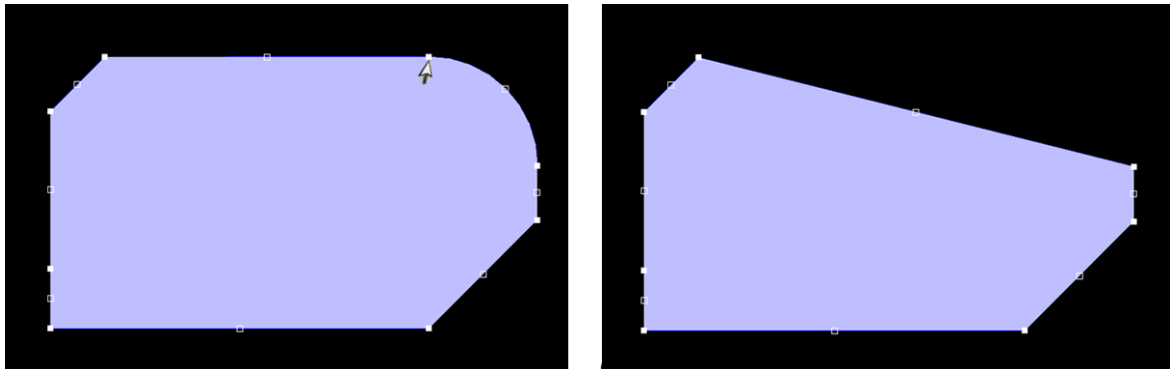


Figure 17. Deleting a vertex

1.9 Controlling the Connection Style

1.9.1 Polygon

54. Vias and through-hole pads will be connected to a polygon with either a **Relief Connect** or **Direct Connect** connection style. For example, the through-hole GND pads of component IOG_2 are connected with a relief connect style as shown in Figure 18.

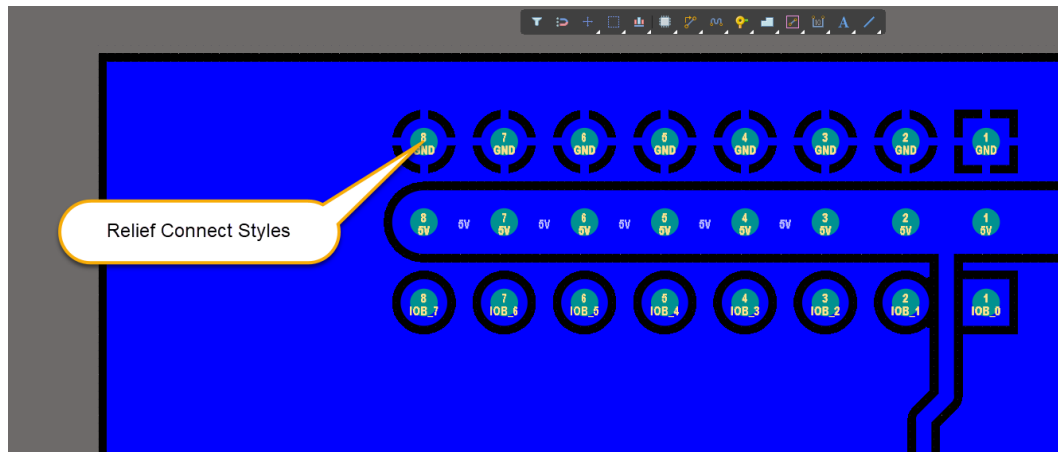


Figure 18. Relief connect style

55. We would like to change the connections from relief connect 90° to 45°. In the PCB, go to **Design » Rules...**
56. Under the *Plane* rule category, select *Polygon Connect Style* and then select the *PolygonConnect* rule to open it as shown in Figure 19.
57. Select the **Advanced** mode.
58. Set the *Rotation* for THT to **45 Angle** and click **OK** to return to the board.

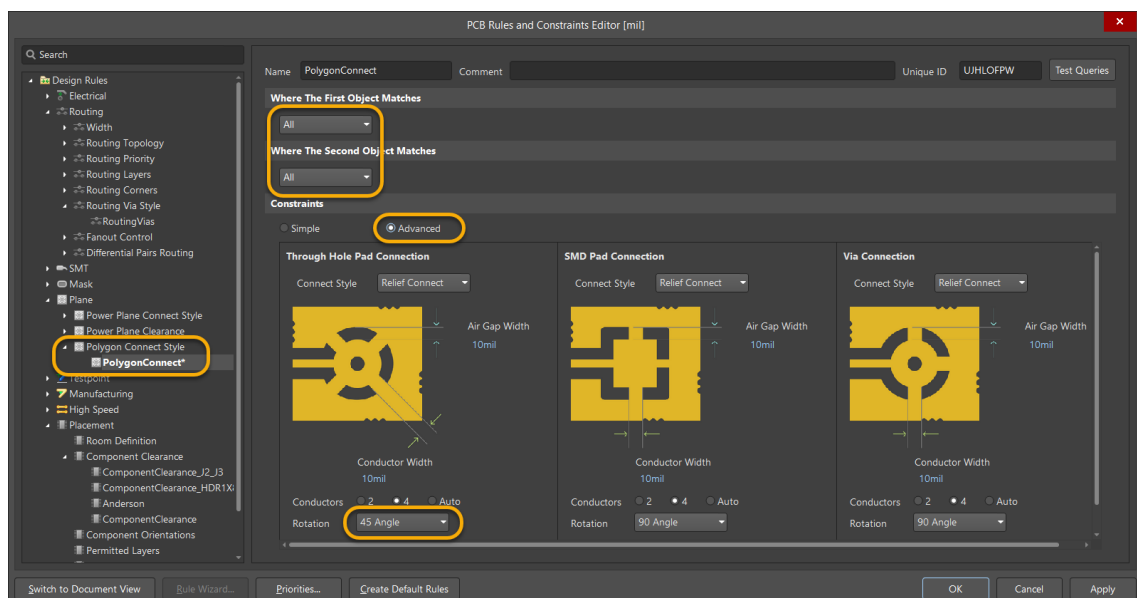


Figure 19. Polygon Connect Style rule

59. Go to **Tools » Polygon Pours » Repour All**. Note that the pads are now connected to the polygon as specified in the Design Rules. Figure 20.

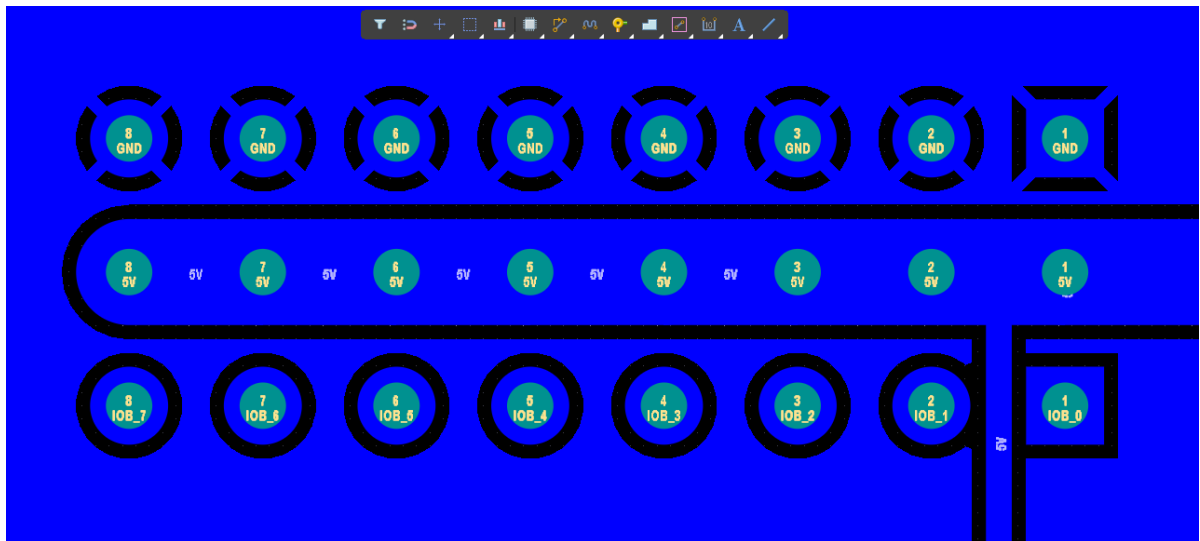


Figure 20. Repoured polygon



The design rule can be configured as both a simple and advanced mode. In Advanced mode, you can set the connection styles for through hole, surface mount and via connection.

60. To avoid design rule violations or automatic repour of polygons when routing, polygons can be shelved. Polygons can be shelved through a couple of different ways, e.g., by right-clicking on a polygon » **Polygon Actions » Shelf All**, or **Tools » Polygon Pours » Shelf Polygons** or in the *Polygon Manager* window.
61. Go to **Tools » Polygon Pours » Shelf 5 Polygon(s)**. Note the number will vary depending on the number of polygons on the board. Observe that all the polygons have disappeared.
62. Restore using **Tools » Polygon Pours » Restore 5 Shelved Polygon(s)**.

1.9.2 Power Plane

63. Activate the Plane Layer 5V.
64. Similar to polygons, we can also control the connection style of pads and vias for a Plane. Go to **Design » Rules...**
65. Navigate to the *Plane* section and open the rule for *Power Plane Connect Style*.
66. Open the default *PlaneConnect* rule. Ensure the fields in this dialog are set as the following:
 - Constraint: **Simple**
 - Connect Style: **Relief Connect**
 - Conductors: **4**
 - Conductor Width: 10 mil
 - Air-Gap: 10 mil
 - Expansion: 10 mil
 - Clearance: 20 mil

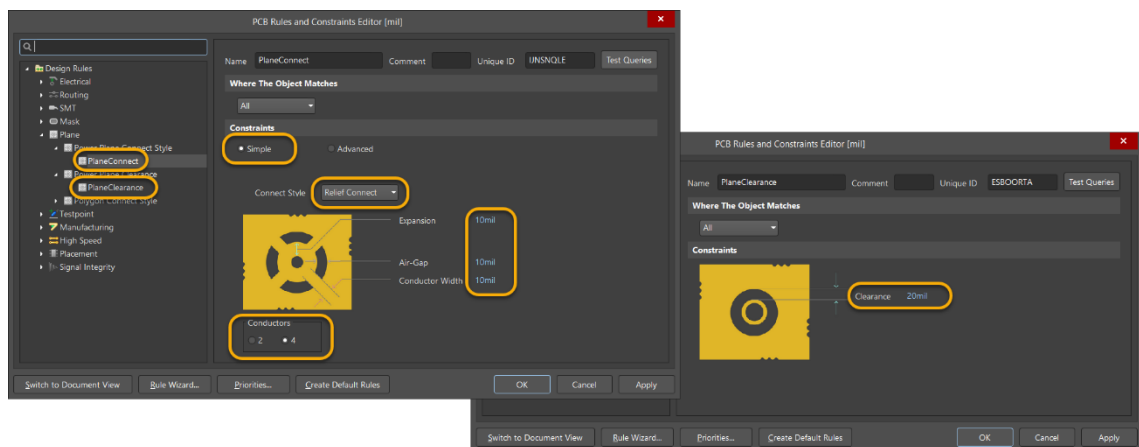


Figure 21. Connect and Clearance for Power Planes

67. Click **OK** to close the window.
68. Note that the pads connected to **5V** on the top left of the board should be connected with a relief connect to the power plane as shown in Figure 22.



Figure 22. Relief Connect style for the 5V plane

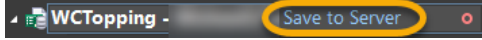


If any additional rules are added for this category, ensure that the new rule has a higher priority than the ALL scoped rule. Otherwise, the new rule will be ignored. Once the **ALL** scoping query is encountered, it overrides lower priority rules for that category.

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

70. 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 23: PCB Polygon Pour and Planes - [Add Your Name] - Finished`.

iii) Select **OK**.

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

Congratulations on completing the Module!

Module 23: PCB Polygon Pour and Planes

from the

**Altium Designer Essential Course
with Altium 365**

Thank you for choosing Altium Designer