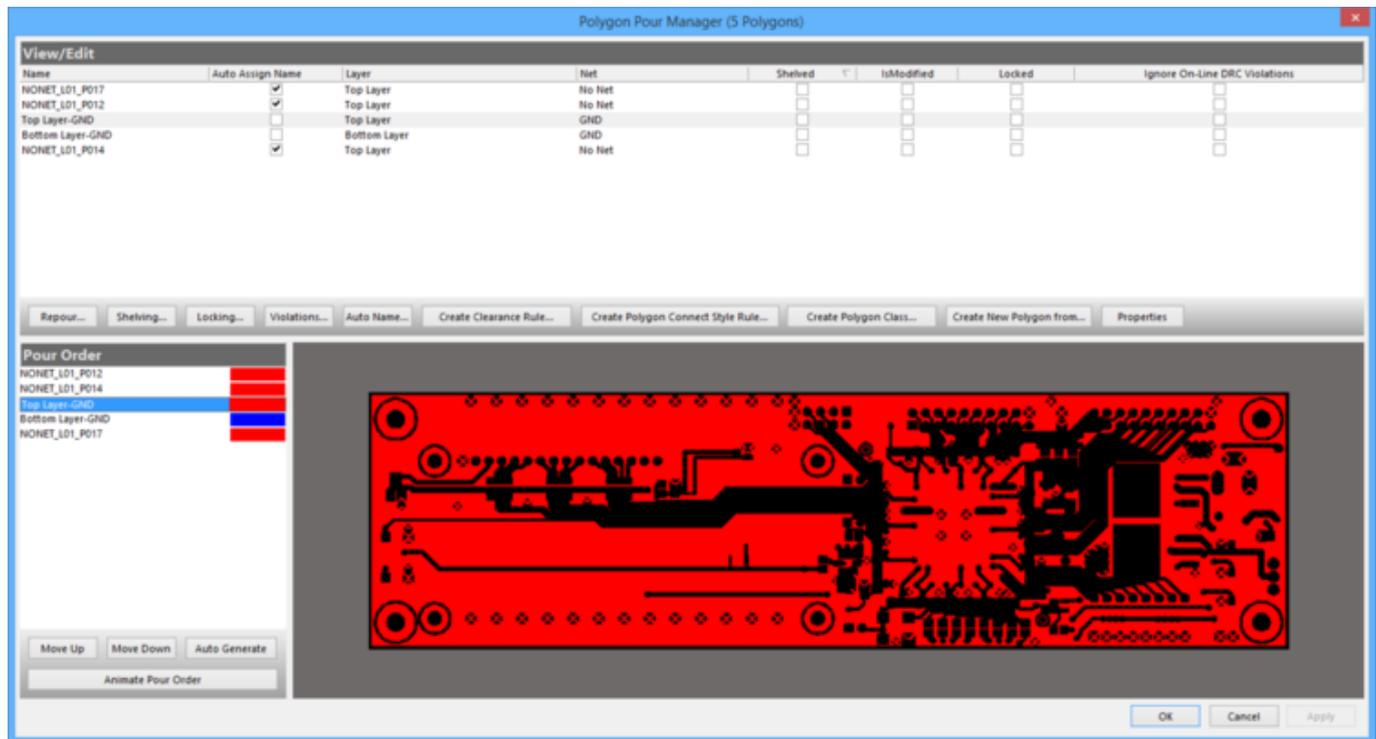


Using Altium Documentation

Modified by Phil Loughhead on Jun 19, 2017



The *Polygon Pour Manager* dialog

SUMMARY

The *Polygon Pour Manager* provides a high-level view of all polygons on the PCB design. The *Polygon Pour Manager* can also be used to rename polygons, set their pour order, perform repouring or disable pouring on selected polygons, add/scope the polygon connection style and clearance design rules, and add polygon classes for selected polygons.

ACCESS

Click **Tools » Polygon Pours » Polygon Manager** to access the dialog.

OPTIONS/CONTROLS

View/Edit

This is a list of all existing polygons in the PCB document; the columns show the current state of each polygon. Click a column heading to sort by that column. Polygon names can be edited. Once named they can be used to scope polygon rules or create queries. You can select one or more polygons (**Ctrl+click**) and execute repours, unpouring, locking and/or ignore Violations.

- **Repour** – use the sub-menus to select which polygon(s) to repour: **Modified Polygons**,

Selected Polygons, Violating Polygons, or Force Repour All Polygons.

- **Shelving** – use the sub-menus to select which polygon(s) to shelve or unshelve. To commit the action, click the **Apply** or **OK** button.
- **Locking** – use the sub-menus to select which polygon(s) to lock or unlock. To commit the action, click the **Apply** or **OK** button.

If you try to graphically move or edit a locked polygon, you will be prompted with a warning message before proceeding.

- **Violations** – use the sub-menus to selectively ignore or keep online DRC violations for all or selected polygons in the list. To commit the action, click the **Apply** or **OK** button.

Do not forget to check and resolve the violations of all polygons before submitting the PCB for manufacturing!

- **Auto Name** – use the sub-menus to set autonaming on or off for all or selected polygons.
- **Create Clearance Rule** – click to open the [Edit PCB Rule](#) dialog to create a clearance rule with a new query for the selected polygons. This rule specifies the minimum clearance between any two primitives on a copper layer.
- **Create Polygon Connect Style Rule** – click to open the [Edit PCB Rule](#) dialog to create a polygon connection style rule with a new query for the selected polygons. This rule specifies the style of the connection from a component pin to a polygon plane.
- **Create Polygon Class** – click to create a polygon class for the selected polygons. You will be required to provide a name for the new polygon class in the *Object Class Name* dialog. An object class is a set of objects treated as a group used by the design rules for example.
- **Create New Polygon from** – Click to create a new polygon then choose:
 - **Selected Polygon** – the [Polygon Pour](#) dialog will open in which you can change the properties of the new pour, such as name, net, layer, and other attributes. (The settings are cloned from the original polygon by default.) When you are satisfied these are set correctly, click on **OK** to close the *Polygon Pour* dialog. The polygon is added to the list at the bottom of the repour order list.

To effect the changes made in the *Polygon Pour Manager* dialog and pass them into the actual design, you must click the **Apply** button. In the viewing region of the dialog, you will see that the new polygon is indeed a clone of the original.

- **Board Outline** – the *Polygon Pour* dialog will open in which you can define the properties for the pour as required, including its name and on which layer it is to be associated. Upon exiting the dialog, the new pour will be added to the list of existing pours in the *Polygon Pour Manager* dialog.

To see the preview of the new board outline-based polygon pour, you will need to first click the **Apply** button, which commits and adds the polygon to the board.

Upon exiting the *Polygon Pour* dialog, the new pour will be added to the list of existing pours in the View/Edit region of the *Polygon Pour Manager* dialog.

The new polygon is inserted into the repour order according to the following logic:

Source Polygon	Other Polygon	New Polygon
Same layer	Same layer	Below Both
Same layer	Different layer	Below Source
Different layer	Same layer	Above Source
Different layer	Different layer	Above Source

- **Properties** – Click to open *Polygon Pour* dialog in which you can edit the properties of the selected Polygon Pour.

The above commands also are accessible on the right-click menu from anywhere in the region although the name and order of the commands is different.



A Polygon Pour can be deleted from the right-click menu **Delete** command.

Pour Order

Lists the current order in which polygons will be poured. The **Auto Generate** button will list the pour order from smallest to largest, which is typically the best order in which to pour polygons, as it ensures that a small polygon is not prevented from being poured by a larger, surrounding polygon. The **Move Up** and **Move Down** buttons allow you to tailor the order in any order you want.



You also can change the Pour Order using your mouse drag-and-drop functionality. This is much more expedient in designs that have many polygon pours.

- **Move Up** – click to move the selected polygon up in the repour order list. The upper the polygon is in the list the earlier it gets re-poured relative to other polygons lower in the list.
- **Move Down** – click to move the selected polygon down in the repour order list. The lower the polygon is in the list the later it gets re-poured relative to other polygons higher in the list.
- **Auto Generate** – click to have the system determine the pour order of polygons from smallest to largest. Use the **Move Up** and **Move Down** buttons to fine tune the pour order if required.
- **Animate Pour Order** – click to preview the order of polygon pours in the graphical representation of the PCB in this dialog. Use the **Auto Generate**, **Move Up**, and **Move Down** buttons to change the order of the pours.

Source URL:

[https://www.altium.com/documentation/cn/display/ADES/PCB_Dlg-PolygonManagerForm\(\(Polygon+Pour+Manager\)\)_AD](https://www.altium.com/documentation/cn/display/ADES/PCB_Dlg-PolygonManagerForm((Polygon+Pour+Manager))_AD)