Polygons Pours & Copper Regions

Summary

This application note looks at using polygon pours to create regions of solid copper on a PCB. It covers such topics as placing and modifying polygon pours, setting properties, controlling the clearance using design rules and repouring.

Application Note
AP0101 (v2.0) December 13, 2004

A copper pour on a signal layer is a common part of a PCB design. This may be a hatched ground pour on an analog design, a solid power supply pour for carrying heavy currents, or a solid ground pour for EMC shielding. Designers requiring large areas of solid copper in their PCB designs can use fills or copper regions, or polygon pours.

Fills and Copper Regions

Fills (**Place » Fill**) are limited to a rectangular shape and will not pour around other objects such as pads, vias, tracks, fills or text.

A copper region (**Place » Copper Region**) is a multi-sided solid copper area, i.e. a fill with more sides, and will not pour around other objects. You can set the layer and net associated with the copper region through the *Region* dialog. If you select the Cutout option, a negative copper region is created which will create a cutout when placed over a solid polygon pour.

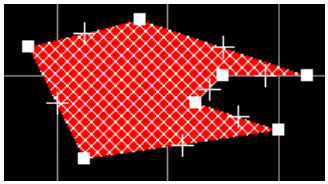


Figure 1. A selected copper region.

Polygon Pours

A Polygon Pour (**Place » Polygon Pour**) creates large areas of solid or hatch-filled copper and, since its outline is made up of a series of placed tracks and arcs, it can fill irregularly shaped areas. As they are poured, polygon pours allow for clearances around electrical objects of a different net or can connect to objects of the same net. Clearances can be set using the PCB Clearance design rules.

When you place a polygon pour, you are defining the outline of a polygon object. There are two ways that the polygon object can filled – solid or hatched.

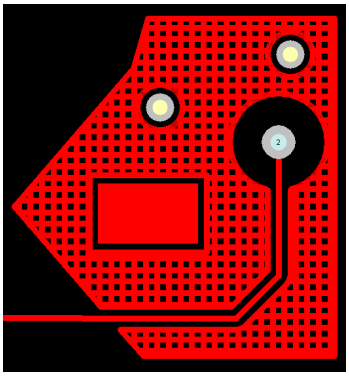


Figure 2. A hatched polygon pour showing varying clearances around electrical objects.

Solid pours

If the polygon pour's Fill Mode is set to **Solid (Copper Regions)** in the *Polygon Pours* dialog, the area inside the polygon boundary is filled with copper regions (complying with applicable design rules such as copper clearance). It places a copper region in each individual area that it finds within the boundary. These areas are created by existing objects such as tracks and pads.

Hatched pours

The copper pour for a polygon can be crosshatched at 90 or 45 degrees, or filled with horizontal or vertical lines. Thermal considerations may determine the style of hatching used in a design. If you have the polygon pour's Fill Mode set to one of the hatched or line options in the *Polygon Pours* dialog, it first outlines all the objects that are within the boundary using tracks and arcs (routing tracks, pads, vias, etc) and then fills in each outlined area with tracks.

Polygon Pour Cutouts

Polygon Pour Cutouts are negative copper regions, i.e. they are used for the removal of copper definitions. They can only be used to cutout the copper from a solid fill polygon pour.

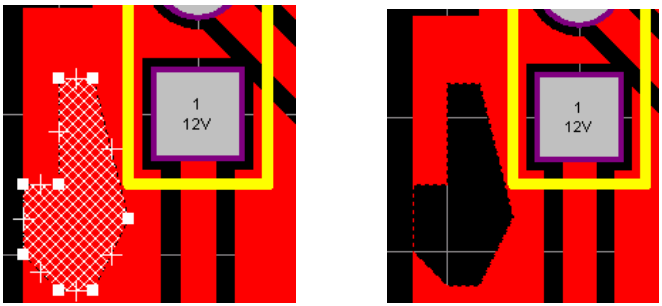


Figure 3. Polygon Pour Cutout placed on a Solid fill polygon pour (left) and the polygon repoured (right).

Placing polygons

Polygons can be placed on any selected layer of a PCB. Select the **Place » Polygon Pour** command and the *Polygon Pour* dialog (Figure) displays to allow you to set the fill and net connection options. See *Setting Polygon Pour properties* for more information. Click **OK** and the cursor changes to a crosshair, ready to draw the polygon outline.

Drawing the outline of a polygon pour is similar to placing tracks during routing, i.e. click to place the polygon vertices and right-click (or press **ESC)** to fill the polygon and exit polygon placement mode. While defining the shape of the polygon pour, press the **SPACEBAR** to cycle through the four polygon corner styles of any angle line, 90-degree arc, 45-degree or 90-degree line, as shown below in Figure .

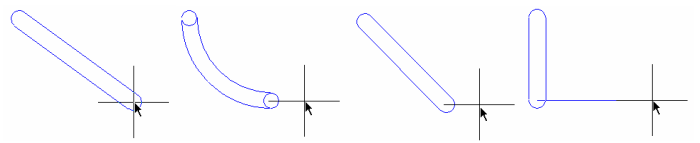


Figure 4. Polygon Pour corner styles

Use **BACKSPACE** to delete the last placed track. There is no need to close the polygon, as DXP will automatically complete the shape by adding a track from the start point to the last point placed. This 'auto close' track displays as a white, unfilled track as you place the polygon.

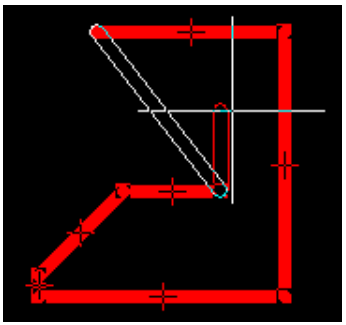


Figure 5. Placing a Polygon Pour showing the auto close track

Setting Polygon Pour properties

The *Polygon Pour* dialog displays when you select the **Place » Polygon Pour** command from the menus or you can press **TAB** while in placement mode. Alternatively, double-click on a polygon pour, or right-click on a selected polygon pour and select **Properties**.

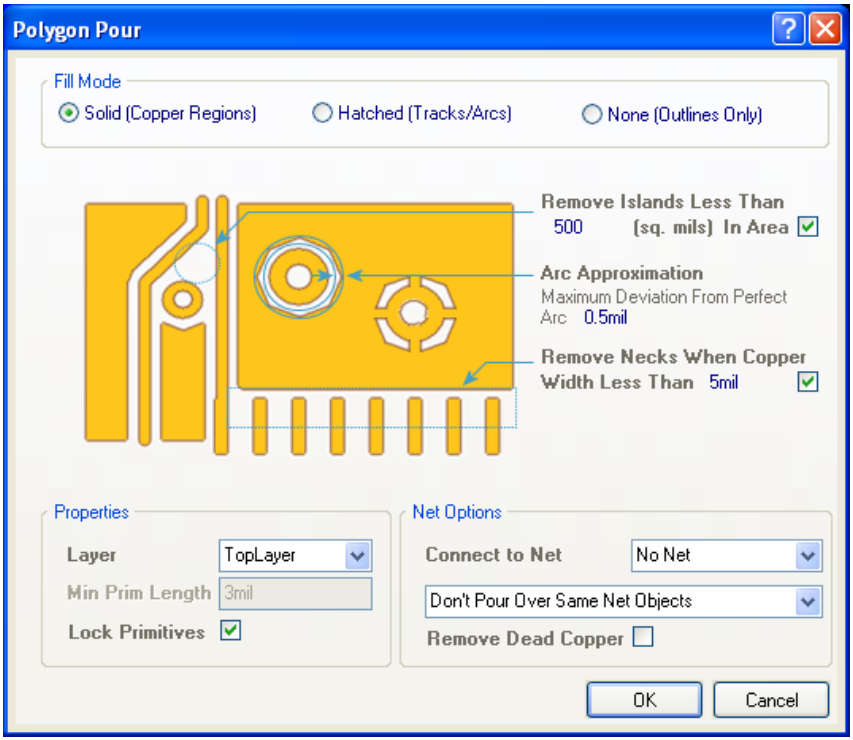


Figure 6. The Polygon Pour dialog

Setting the Fill Mode

To determine the appearance of the hatching fill used when copper is poured into the polygon pour outline, the **Hatched (Tracks/Arcs)** option is used with a combination of the **Grid Size**, **Track Width** and **Hatch Mode** options that become available when you select the Hatched option. The fill will be created out of tracks (and arcs) automatically placed according to these settings. The Grid Size determines the spacing between the centers of the tracks used for hatching. This grid ideally should be a fraction of the component pin pitch to allow efficient placement of the tracks. Next, set the width of tracks used to create the hatching when the polygon is poured.

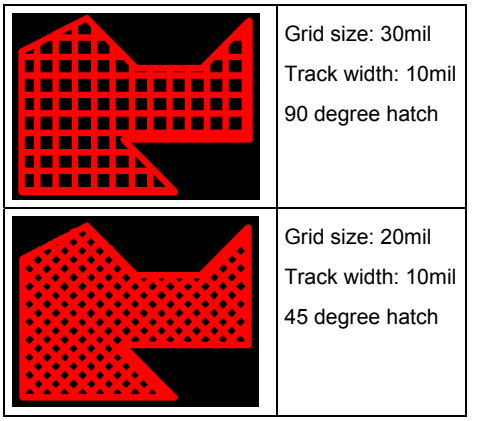


Figure 7. Hatching style examples

To create a solid copper pour, click on the **Solid (Copper Regions)** option and a region of copper will be created. You can use Polygon Pour Cutouts with a Solid filled polygon. In this mode, you can also **Remove Islands** less than an nominated area and **Remove Necks** when the copper width is less than the nominated width. The **Arc Approximation** option allows for a maximum deviation from the perfect arc used when the copper is poured around pads or vias (the lower the value, the smoother the arc).

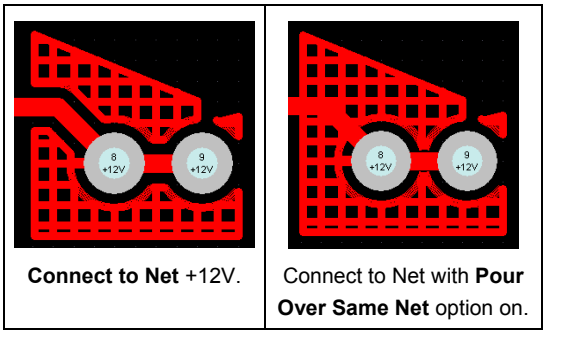
The Fill Mode option **None (Outlines Only),** i.e. no fill, can be useful during the design phase as it will not slow down system performance waiting for repours. Then, select the desired hatching style before generating output.

When polygons are poured, they can contain many short pieces of tracks and arcs, placed to create smooth edges around the existing objects on the board. Set the **Minimum primitive length** as appropriate, considering that a larger value gives faster pour times, screen redraws and output generation but downgrades the smoothness of the polygon edges. Setting the **Surround Pads With** option to Octagons instead of Arcs has a similar effect on the repour times and smoothing.

If you are changing a Hatched polygon pour to a Solid pour, use the **Tools » Polygon Pours » Convert Hatched Polygons to Solid** command to set the extents of the repoured polygon as well as remove islands and necks and set the arc approximation.

Defining the net connection styles

The polygon pour can be attached to a net. When a netlist exists in the PCB document, enable the **Connect to Net** option and select the required net from the drop-down list. The polygon will now connect to each pad on this net found within the outline of the polygon, in accordance with the Polygon Connect Style design rule (see *Setting the Polygon Connect design rule* for more information).



Connect to Net +12V.

Connect to Net with **Pour Over Same Net** option on.

Figure 8. Net Connection examples

Polygon pours will automatically pour around any existing electrical objects on a layer (pads, vias, tracks, etc). If you want the polygon to automatically connect to objects on the same net as the pour, check the **Pour Over Same Net** option in the *Polygon Pour* dialog. Any existing tracks within the polygon, which are part of the net being connected to, will be covered by the polygon. If this option is selected and the polygon is assigned to **No Net**, it will pour around all objects regardless of their assigned nets.

Removing dead copper

If the **Remove Dead Copper** option is enabled, any regions of 'dead' copper within the polygon will be deleted. Dead copper is created when an area of the polygon cannot be connected to the selected net and appears as unconnected 'islands' of copper within the polygon created when existing tracks, pads and vias prevent the polygon pouring as one continuous area.

The entire polygon is removed if it does not enclose any pads on the selected net, as it is all viewed as dead copper.

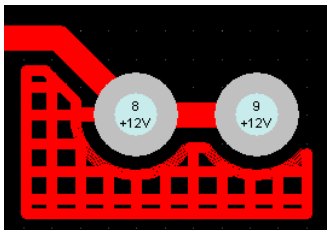


Figure 9. Connect to Net +12V with Remove Dead Copper option enabled.

Controlling the clearance using design rules

If a violation appears between the polygon and an object, e.g. a pad, you can repour the polygon by double-clicking on it and confirming the setup. This results in the polygon pouring around the violating object, taking into account clearance and polygon clearance design rules. These design rules may be set by using the **Design » Rules** command.

Polygon Clearance rule

Create a Polygon Clearance rule (under **Electrical/Clearance** in the *PCB Rules & Constraints Editor*) to control the distance between the polygon pour and any other objects, e.g. pads, tracks or fills. When creating Clearance rules for polygons, you must use the `InPolygon` (or `InPoly`) keyword, rather than `IsPolygon` (or `IsPoly`). This is because the clearance rules operate on the primitives (tracks and arcs) that make up (or are 'in') the polygon, rather than the polygon as a whole object. A valid clearance rule for polygons would be between "InPolygon" objects and all other objects.

The Polygon Clearance rule must have a higher priority than any general clearance rule if it is to have any effect (1 is the highest priority).

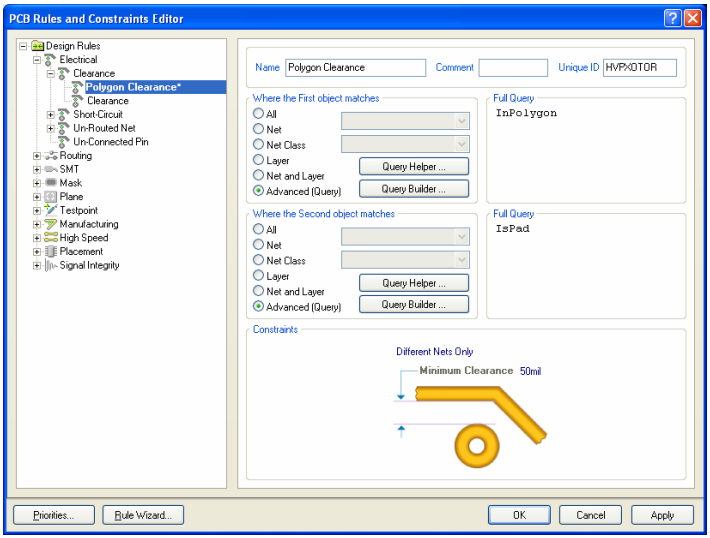


Figure 10. Polygon Clearance rule in PCB Rules & Constraints Editor

Online DRC (design rule check) will show violations but if there are any track width violations, for example, they are displayed only once. If you move a polygon or change the clearance, the online DRC will not display the violations again. This saves refresh time when large solid polygons are present. Run the DRC manually to check for any violations (**Tools » Design Rule Check**).

Setting the Polygon Connect design rule

To control how a polygon connects to pads when the **Connect to Net** option is used, set the **Plane/ PolygonConnect** design rule **(Design » Rules**). This rule specifies the style of the connection from a component pin to a polygon pour.

Three connection options are available: direct (solid copper to the pin); thermal relief (the width, number and angle of connections can be set) or no connection.

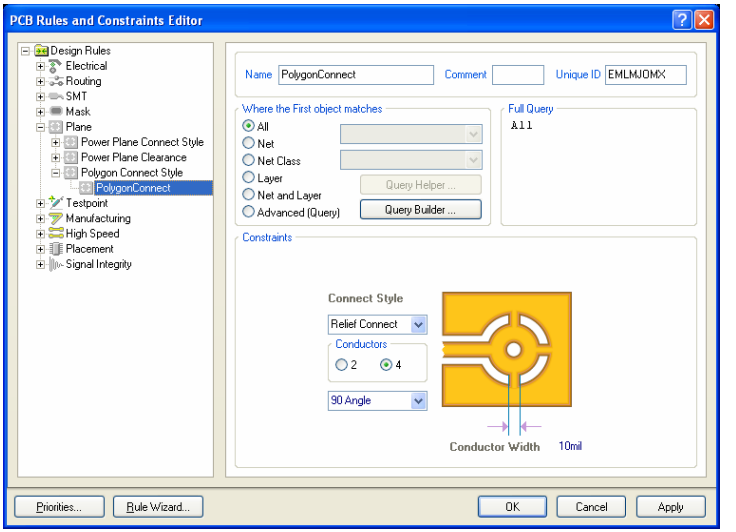


Figure 11. Polygon Connect rule in PCB Rules & Constraints Editor

Removing copper islands and narrow necks

If you have set the Fill Mode to Solid, you can enable options to remove islands of copper by determining the area threshold (**Remove Islands by Area**), or remove any copper that is under the Copper Width as defined by the Width Threshold (**Remove Under Width Copper**), such as narrow necks of copper. You can also set the **Arc Resolution** to determine how accurately the fill is drawn around other objects, e.g. pads, when the polygon is poured.

Repouring polygons

To repour a polygon, click to select and then move the polygon, or press **TAB** (or double-click) and change the settings. Alternatively, right-click on a selected polygon and choose from the **Polygon Actions » Repour** options. Confirm the repour by clicking **Yes** in the rebuild *Confirm* dialog.

Creating a polygon outline from a set of tracks

You can create a polygon outline from a set of placed tracks using a process parameter attached to the process `PCB:RunScissors`. Select **Customize** from the right-click menu when over the menu toolbar, copy the **Slice Polygon Pour** command from the **Place** menu, rename the caption and change the process parameter to `(mode=POLY_FROM_SEL_PRIMS)`. See the *Customizing DXP Resources* tutorial for more information. Then, select the joining tracks required to make the polygon and choose this new command. A new unfilled polygon is created on the same layer and with the same shape as the selected tracks. These tracks can be deleted if they are no longer required.

Working with polygons

Once you have defined the polygon pour, you can move, merge, copy and paste, slice or reshape it. Delete selected polygons by pressing the **Delete** key.

Moving polygons

Click and drag to move a polygon. Hold down **Shift** key to select multiple polygons to move. The rebuild *Confirm* dialog appears when you release the mouse button. Click **Yes** to repour the polygon(s) and refresh the screen.

Figure 12. The Rebuild confirmation dialog

Merging polygons

If you move a polygon toward another polygon and both polygons are attached to the same net, they will automatically merge. The new outline is visible while merging and you are prompted to pour only one. If two merged polygons leave a hole in the resultant polygon, a repour will create one solid polygon, as cutouts are not possible. Merged polygons cannot be unmerged back into their original outlines.

Slicing polygons

The **Place » Slice Polygon Pour** command is used to slice a single polygon pour into two or more separate polygon pours. You are now in slice mode (similar to line placement mode), so click (or press **ENTER**) to anchor a series of vertex points that define the shape of the slice. When defining the slice, press **SHIFT+SPACEBAR** to cycle through the placement modes. Press **SPACEBAR** to toggle between the Start and End modes. Use the **BACKSPACE** key to remove the last placed slice segment. When you have finished defining the slice, right-click (or press **ESC)**. Continue defining more slices, or right-click (or press **ESC)** to exit polygon slice mode.

A confirmation dialog appears stating how many new polygons will be created. Click **Yes** and confirm that you wish to rebuild the polygons.

Reshaping polygons

Selected polygons can be reshaped by clicking on the vertices or midpoints and dragging them to the required new location.

You can also edit the shape by moving and inserting vertices. Right-click and select **Polygon Actions » Move Vertices**. Click and drag on a vertex handle to move the vertex.

To insert a vertex, click-and-hold on the small midpoint cross in the line segment and drag it to where the new vertex is required. Right-click and confirm to rebuild the new polygon shape.

If you are having difficulties 'picking up' a vertex due to the hatching, click on the vertex, zoom out (**Page Down**) until you see the lines attached to the vertex move and then zoom back in (**Page Up**) to place the vertex.

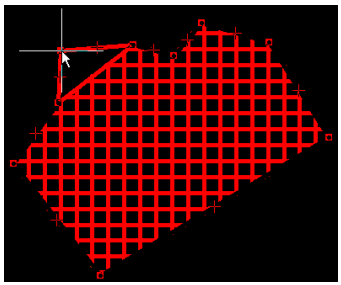


Figure 13. Moving polygon vertices

Shelving polygons

You may find that some large hatched polygons can take a while to repour, so use the **Tools » Polygon Pours » Shelve Polygons** command to temporarily hide polygons in the design. Use the **Tools » Polygon Pours » Restore Shelved Polygons** to reinstate them.

Using polygons on non-copper layers

Polygons can be used on non-copper layers for purely display reasons, e.g. to define separate areas of the board. If a polygon is placed on a non-signal layer, it will not pour around existing objects because they are not assigned to a net.

Managing polygons

For more information about the polygons in your PCB, you can use the Board Information Report or list the properties of a polygon and its children.

Using Reports – Board information

The number of polygons detected on your PCB can be shown by selecting **Reports » Board Information**. For a detailed listing of the polygon properties, use the **List** panel.

Using the List panel to view polygon properties

Select the polygons that you wish to view in the List panel by using the filter `IsPoly` in the Filter panel (click on the **Filter** tab or select **View » Workspace Panels » PCB » Filter**) and click on **Apply**.

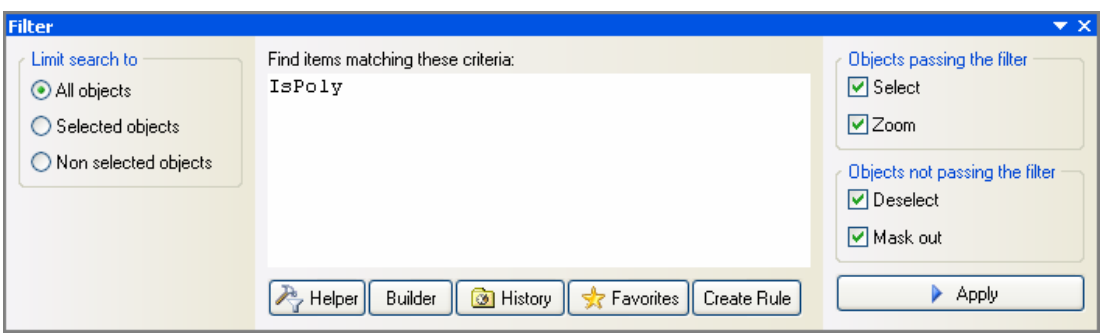


Figure 14. The Filter panel showing the IsPoly query.

Any polygons in the design will be selected. Open the List panel (select **View » Workspace Panels » PCB » List)** and a list of all polygons in the PCB will appear – both manually placed polygons and

internal planes (which are system-created polygons). Details such as their location, layer, net and copper pour options are also displayed.

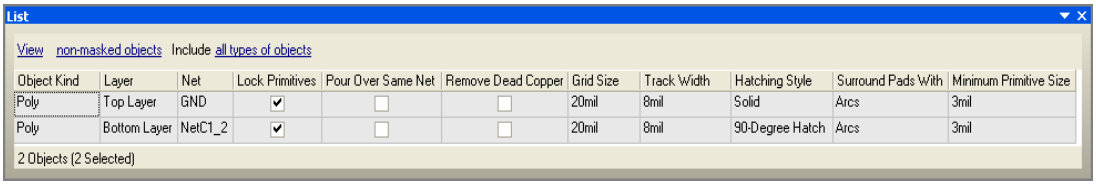


Figure 15. The List panel showing the results of an IsPoly query.

Parent-children attributes, such as the region(s) associated with a solid pour polygon, or the tracks and arcs that make up hatched polygons, can be listed by right-clicking on a polygon (Poly) in the Object Kind list of the List panel and selecting **Show Polygon Children**.

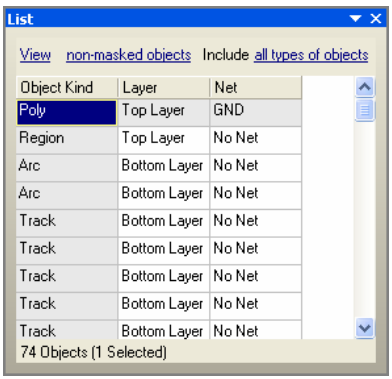


Figure 16. The List panel showing the results of an IsPoly query with Show Polygon Children enabled.

Other uses of polygons

Polygons are also used by DXP to outline each region of split internal power planes. A split plane is therefore just an empty polygon. Split planes can be viewed in the Split Planes Editor in the **PCB** panel. Refer to the *Internal Power & Split Planes* article for more information.

Revision History

Date	Version No.	Revision
9-Dec-2003	1.0	New product release
13-Dec-2004	2.0	Updates for SP2. Formerly *Polygon Planes and Copper Pours*.