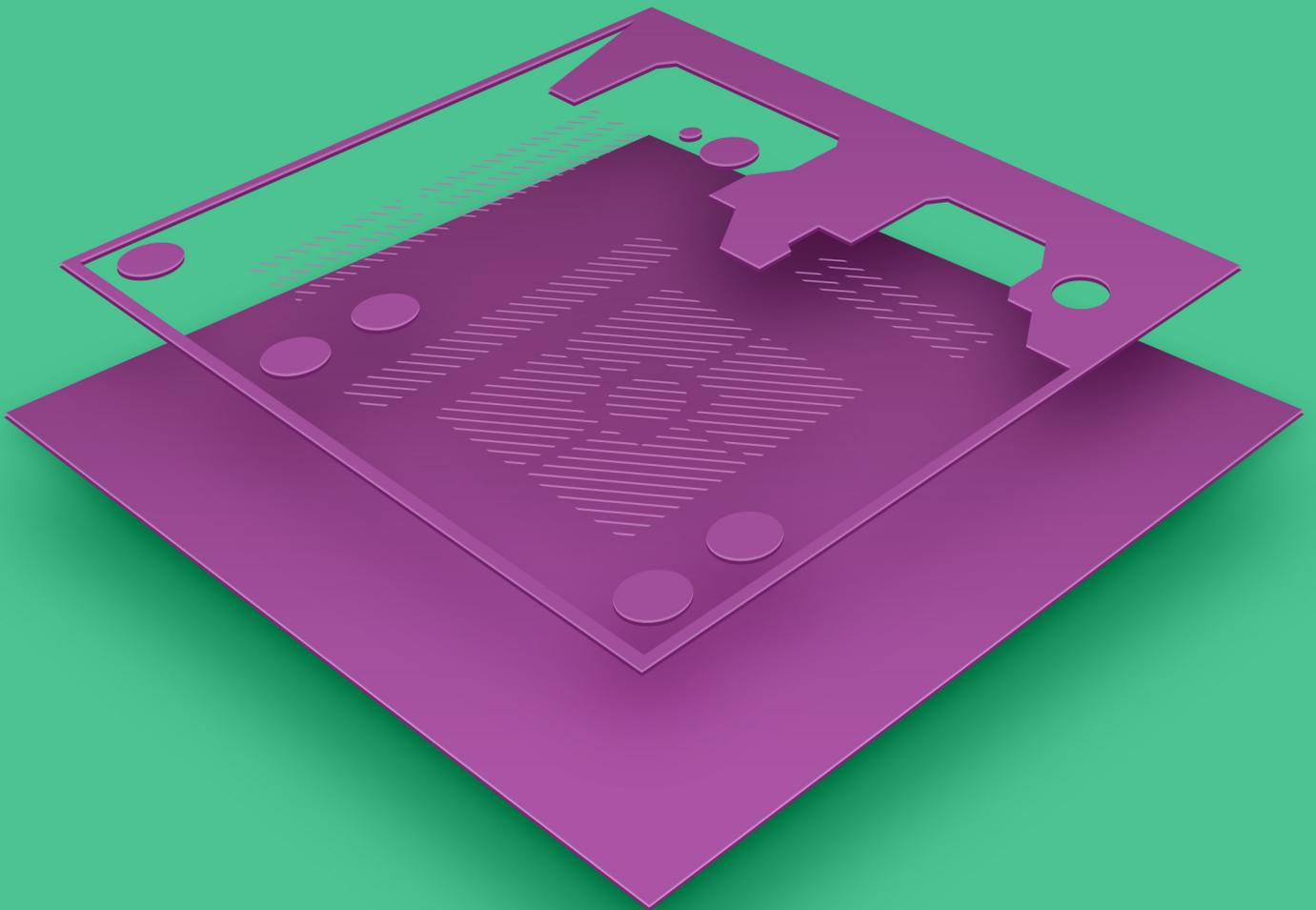


***Altium***<sup>®</sup>

# Polygon or Plane, Which is Better?



**John Magyar**

Senior Applications Engineer

# POLYGON OR PLANE, WHICH IS BETTER?

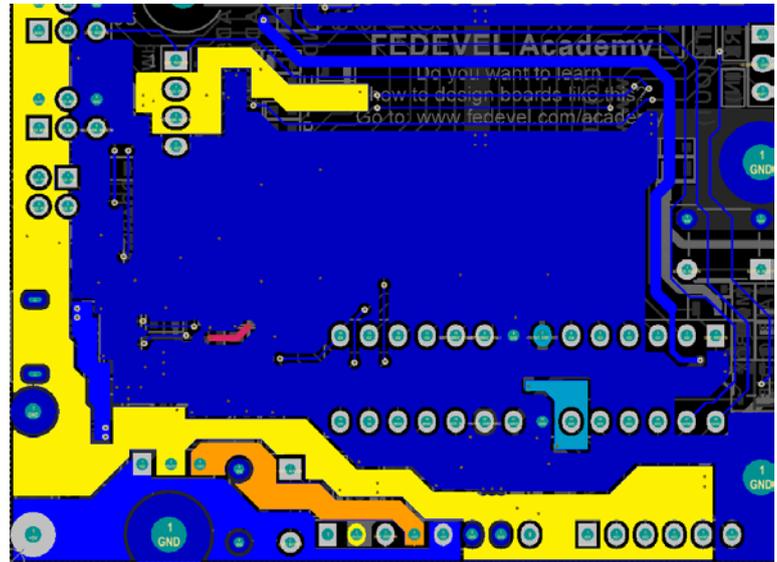
Users have two choices when implementing large copper areas for power and ground nets: polygons or planes. Users often ask which one is better for implementing power nets, but there is no one correct answer as they both have similar end results. Both constructs enable users to create appropriate power and ground nets. This paper explores the similarities and differences between polygon and plane constructs to help users decide which is better for their requirements.

## POLYGONS

Polygons, also known as copper pours or polygon pours, are areas of the PCB that are filled or flooded with copper poured around existing components and traces. Polygons can only be defined on signal layers, which are rendered in the positive, and the shapes placed on it represent the addition of copper.

Places polygons are commonly used include:

- surface layers containing placed components and routed traces
- internal signal layers containing routed traces and signal layers exclusively dedicated for power
- ground without any placed components or routed traces.



*Figure 1. Polygons on a signal layer  
(layout courtesy of FEDEVEL Academy)*

Polygons can be solid copper, hatched copper or simply a contiguous outline. Attributes of polygon pours are managed through the **Properties** panel or **Global Editor**, and can be modified directly within the **PCB Editor** or imported from the MCAD tool in DXF/DWG format.

## PLANE LAYERS

Plane layers, also known as internal planes, power planes or split planes, are areas of the PCB where the entire internal layer is initially all copper. Planes can only be defined on internal plane layers, which are rendered in the negative; therefore, the shapes placed on it represent the *removal* of copper.

Internal plane layers are used exclusively for power and ground nets. Components and routed traces cannot be placed on internal plane layers. A plane layer can be divided/split into multiple sections to represent multiple power and ground nets. Split planes can be nested within an existing plane.

## WHICH IS BETTER?

Users commonly ask which method is better for implementing power and ground nets: polygons or planes? There is no right or wrong answer as both methods enable the creation of the required copper shapes. However, certain aspects of each construct can make one method preferable over another.

**Polygon pours** must be placed on signal layers and cannot be used on internal plane layers. One advantage of polygons is that the entire layer need not be dedicated to net power distribution. Any signal layer (including top and bottom layers) may contain multiple polygons to distribute power and ground nets, yet still contain areas for non-power related signal trace routing. The fact that interactive trace routing is not at all possible on internal plane layers can be a deciding factor on which construct to use.

## POLYGON OR PLANE, WHICH IS BETTER?

Another advantage of polygons is that they render in the positive—so the shape you see on a signal layer in the **PCB Editor** is exactly the shape that will be rendered in the actual PCB copper. Plane layers render in the negative, so objects placed on an internal plane layer define what is NOT copper, which means that working in the negative can make it difficult or uncomfortable to visualize the true definition of the plane shape.

Polygons also offer the option for hatched copper and polygon outline instead of just solid copper, while internal planes do not. Hatched copper enables better thermal properties to manage copper expansion and board warping in high temperatures. Hatching can also provide the ability to better control impedance characteristics in RF layouts.

**Power or ground planes** can be quickly and easily defined on an internal plane layer. The internal plane layer begins as all copper, and as lines, arcs, fills or region objects are added to an internal plane layer, copper is removed within the resulting shape. Therefore, it is very efficient to describe multiple power nets on a plane layer by simply placing division lines between each plane. Even nested planes can be defined by simply placing a closed outline within an existing plane. While these same copper shapes could be achieved using polygons on a signal layer, their creation, management and any subsequent modifications can be much more challenging and time-consuming.

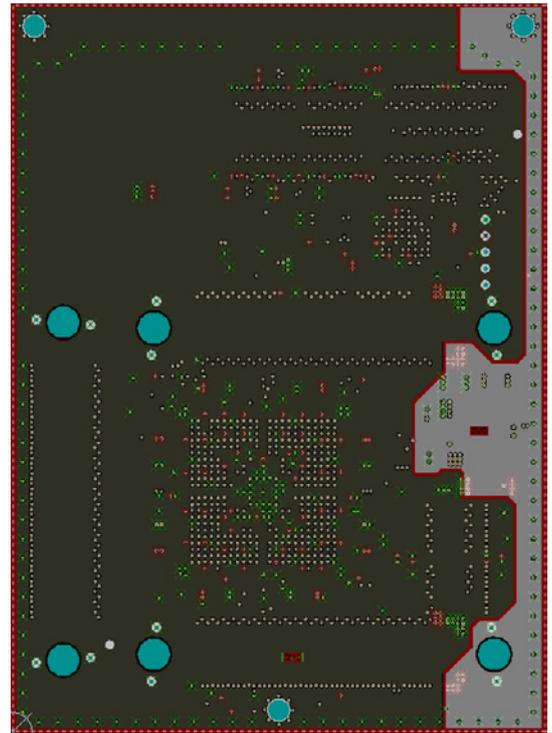
As mentioned earlier, internal plane layers are rendered in the negative. The historical reason for this is because negative plane shapes can be represented by much less data than positive plane shapes. Consequently, one benefit of negative layers is a significantly smaller Gerber file size. While the file size may no longer be a concern these days, it is important to note that smaller sized Gerber files will process much faster during photo plotting.

### CONCLUSION

Both polygons or planes can be used to implement power and ground net distribution. If restricted to a 2-layer stackup only, polygons are the only solution. Working with 4-layer or greater stackups, you have a choice to use polygons on signal layers, planes on internal plane layers, or both.

Polygons on signal layers offer the best flexibility regarding space use. Because polygons are placed on signals layers, you may dedicate the majority of the mid-layer for power and ground net distribution. However, you always have the option to route non-power related traces on the same layer. If you find working in the negative on internal plane layers difficult to visualize or verify, then you will be happy to find using polygons on signal layers as an alternative.

If dedicating numerous power and ground nets to specific layers, internal plane layers can be easier to work with and manage. The ability to divide or split an internal plane layer makes the creation and subsequent management of numerous and complex shapes easy and efficient. Because internal plane layers are rendered in the negative, they result in much smaller sized Gerber files, which can be processed faster during fabrication.



*Figure 2. Power net planes on a split internal plane layer*