Altium PCB Polygon Pour Instructions.

We will now cover the use of PolyGon Pours.

After your PCB has been routed and you are ready to pour copper on the top and bottom layers

Click the Design toolbar dropdown and select Rules....
Right-click on the clearance rule and select “New Rule”.
Rename the new rule to “PolyGon Pour Clearance” by clicking on the name of the new rule.

Click on the new rule on the left column. This will display the properties on the PolyGon Pour Clearance rule.
Change the first Object match to Advanced and change the Query from All to InPolyGon.

Next change the Minimum Clearance from 10mil to 20mil. Click Apply and then click OK.
Click the Place tool bar button and select “PolyGon Pour...”.

Set the layer to Top Layer and also set the “Connect to Net” to your ground net.

Click OK.

Your mouse pointer will change to a cross hair. Use this pointer to draw a box around your PCB.

After drawing a box around your PCB right click your mouse to exit the draw mode.
Altium will fill the empty areas of your top layer on your PCB with copper connected to ground.
We need to repeat this procedure for the bottom layer.

Select the Bottom Layer tab at the bottom of your screen.
Click the Place tool bar button and select “PolyGon Pour...”.

The bottom layer should be selecter as well as the connection to the ground net.

Click OK.

Your mouse pointer will change to a cross hair. Use this pointer to draw a box around your PCB.

After drawing a box around your PCB right click your mouse to exit the draw mode.
Altium will fill the empty areas of your bottom layer on your PCB with copper connected to ground.

This concludes the PolyGon Pour tutorial.