

POCKETCAM is an APP that lets you to create a pocket in a CNC machine by modifying the drawing in Auto-Cad by continuously offsetting the geometry on the inside with the specified parameters. The final drawing is a continuous polyline.

About:

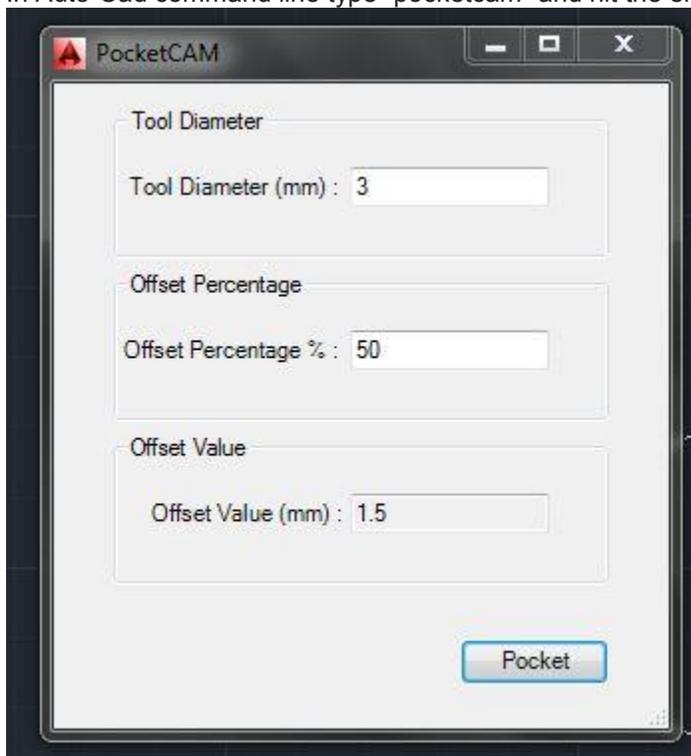
Creates pocketing tool path for closed geometries. The created tool path can be used in CNC machines for pocketing operation.

Pocketing operation needs tool paths which are offset from original geometry several times, till the whole area inside the geometry is covered. This app automates the process of creation of these pocketing tool paths.

The output of the software can be imported into any CAM software for converting to G-codes.

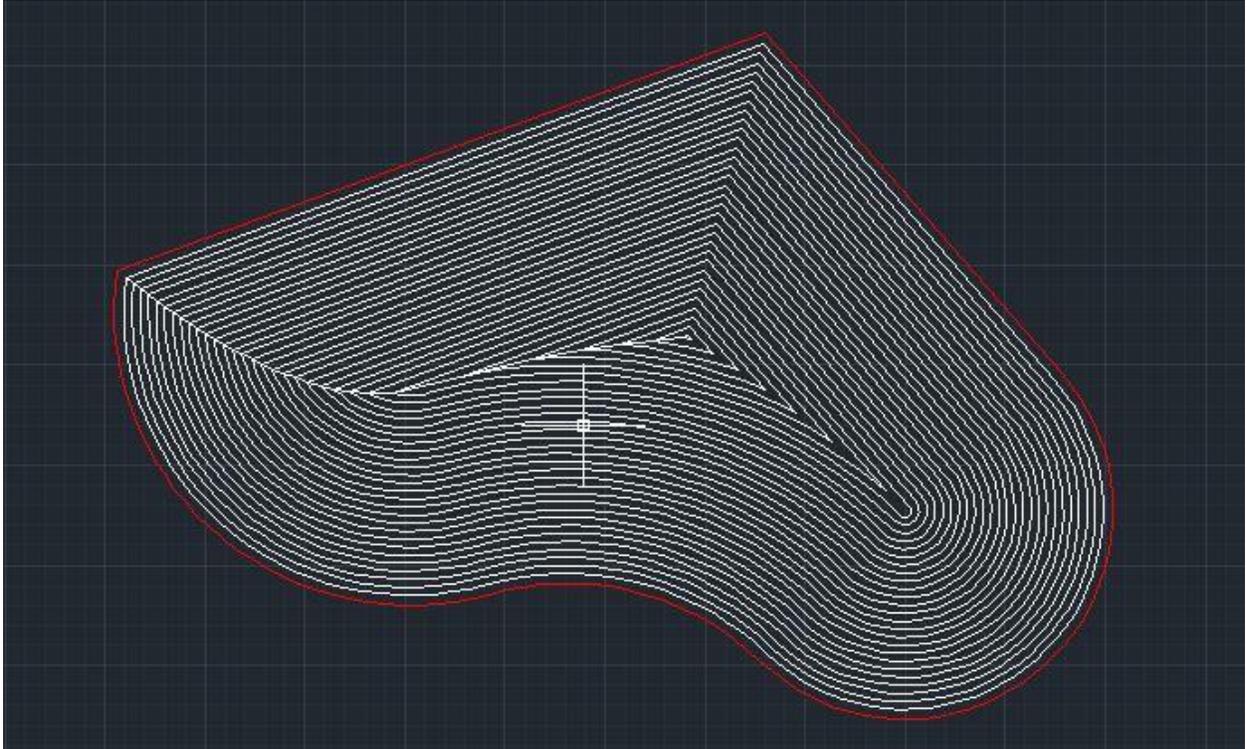
Using POCKETCAM:

In Auto-Cad command line type “pocketcam” and hit the enter button. The following dialog box opens:



It is very simple to use this app. Just enter the tool diameter that you are using to cut and then the offset percentage. The offset value will be automatically calculated from the offset percentage and displayed in the box. The offset percentage is the step over after every pass. It is the tool overlap between cutting paths.

Enter all the parameters and click pocket. Now with the mouse pointer click the geometry for which you want to create the pocket. When clicked you will see the following:



You can clearly see that the geometry has been continuously offsetted on the inside direction in the form of polylines and the distance between the offsetted geometries on the inside is the “offset percentage” and all the offsetted geometries are joined together to form a single polyline.

Now you can save this as a dxf file and use “G-Code cam” to generate the G-Code for this to be cut out on a CNC by specifying the depth of the pocket in the G-Code cam dialog box.