

OFFSET CAM is an APP that is useful for machines that don't have cutter compensation, namely - G41 and G42. This app automatically offsets 2D geometries by tool radius. The offsetted geometry can be directly imported to a CNC software for machining operation.

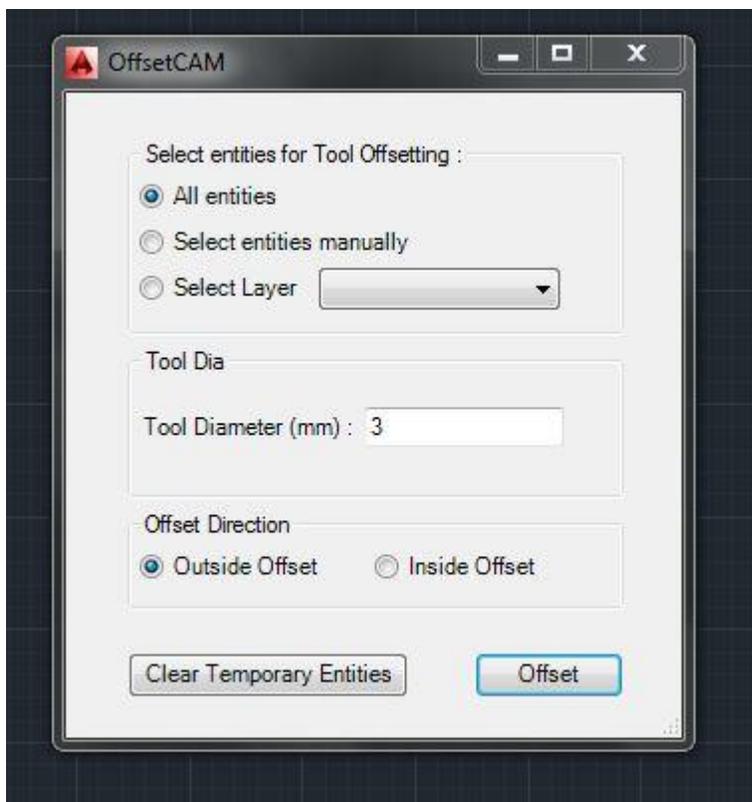
When machining a geometry in CNC machine, the geometry needs to be offsetted by the tool radius and the tool has to travel on this offset path, so that the part-size produced is accurate. Some of the Advanced and costly CNC controllers can do this automatically (using G41, G42 commands). In some of the entry level and low-cost CNC controllers, G41 command is not available. Even if G41 is available, it may not work correctly for complex geometries. Offsetting is a highly computational algorithm, it is better to do it in a CAD/CAM software rather than in a CNC software. Also machines like Laser cutting machines don't run on G-codes and they don't support the offset command (G41 etc.).

Offset CAM automatically identifies Inner and Outer geometries. After identifying: It offsets the outer geometries in Outside direction and Inner geometries in Inside direction. (Or vice-versa: if Inside offset option is chosen).

The original geometries are moved to temporary later called Acam_tmp and can be deleted with a button click.

The offsetted geometries can be imported into a basic CAM software to get a G-code output that can be fed directly to any CNC machine.

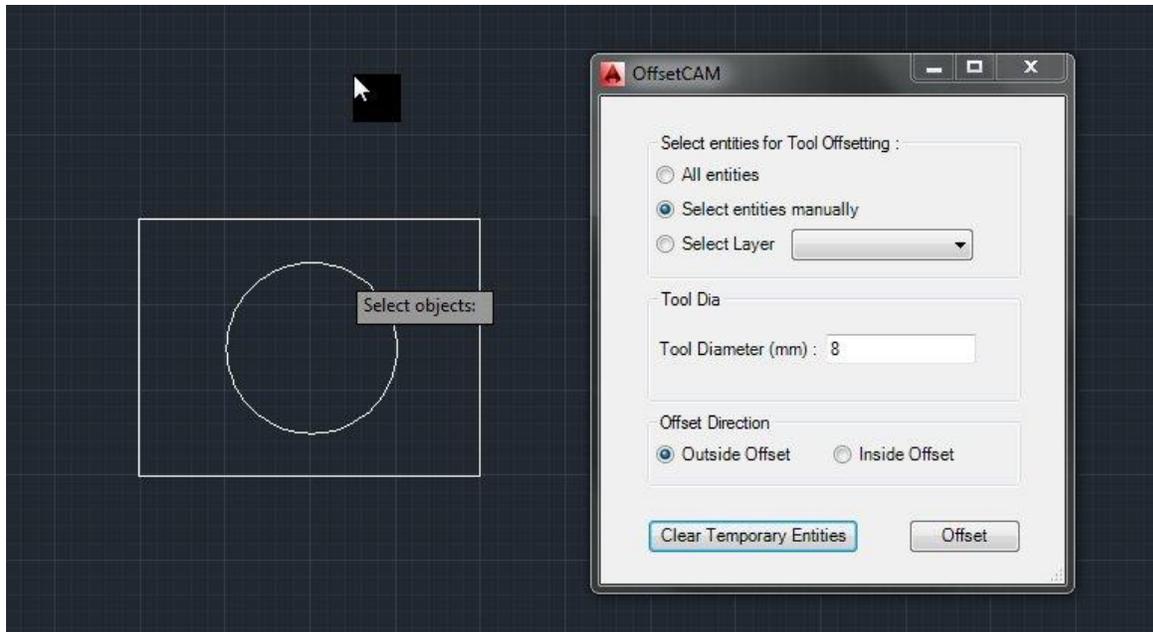
To use the APP, type "offset cam" in the command line and hit Enter button. The following window will open:



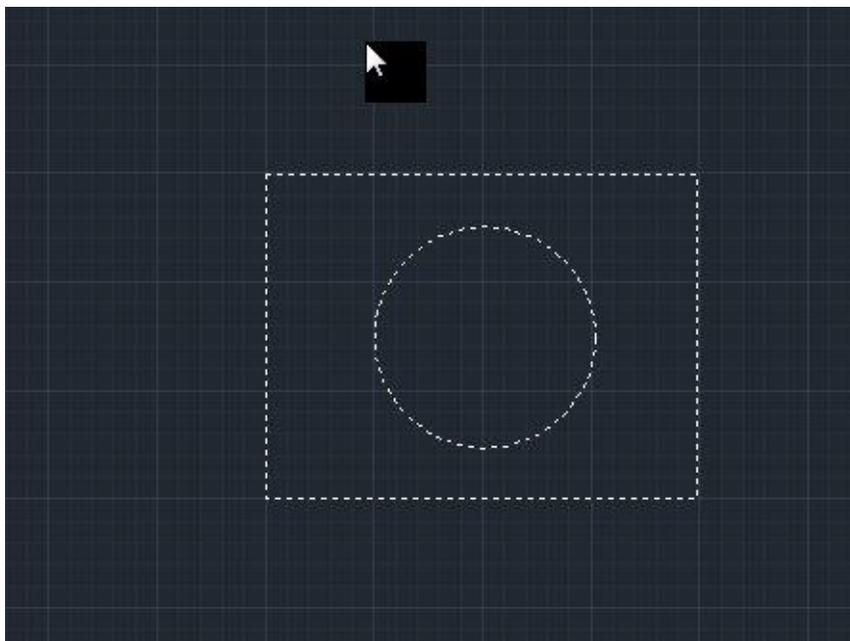
You have three options for offsetting geometries:

1. You can offset all the geometries in your screen with one click by clicking at one corner and dragging the mouse pointer to the diagonally opposite corner to select all the geometries in the box.
2. You can select individual geometries which is useful when you want to offset some geometries inside and some outside.
3. You can select the layer in which the geometries are present.

For this example we will use the “select entities manually” option and will select all the entities in the screen.



The geometries that are selected are highlighted with dotted lines which show your selection.

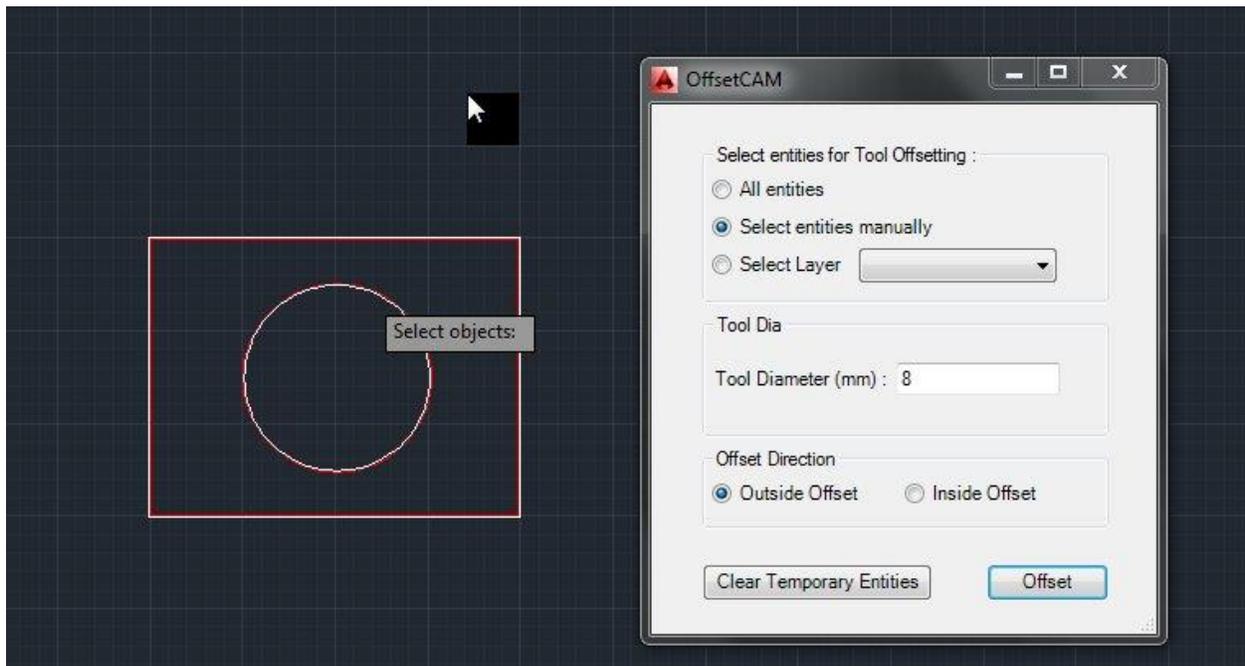


The next entry is the tool diameter of the tool which you want to cut with. The tool diameter must be entered and the geometries will be offsetted by a value equal to the tool diameter specified.

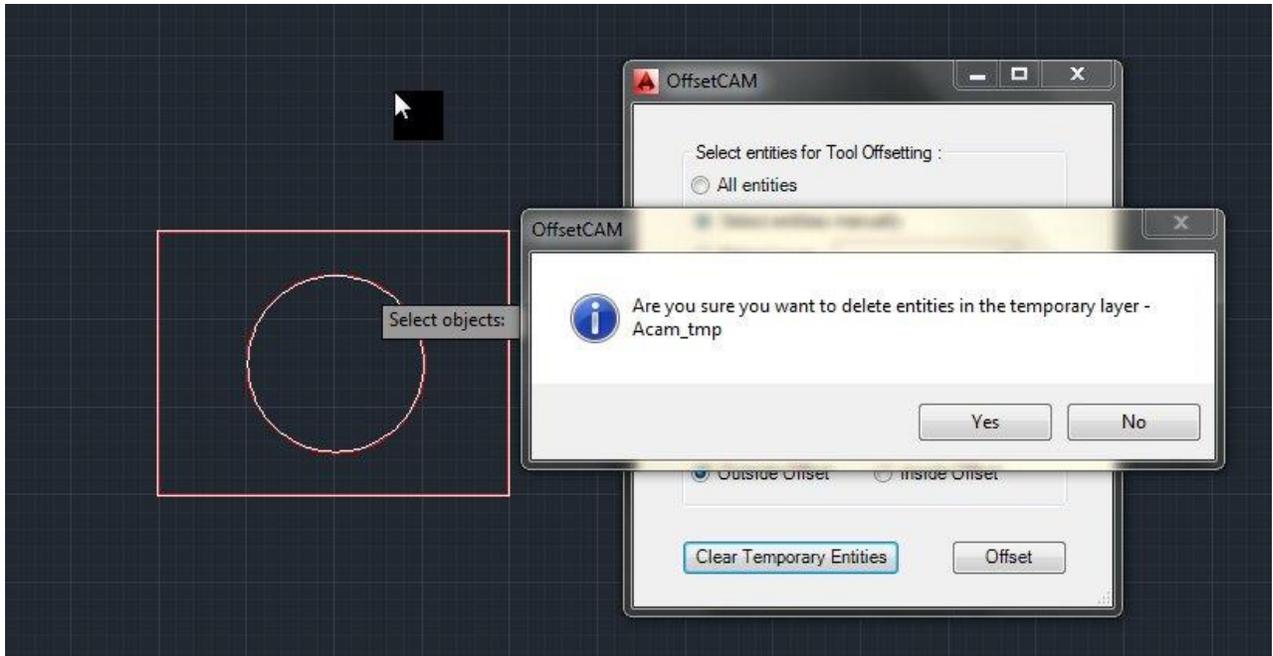
Next option is the offset direction:

1. Inside offset will cause an offset on the inner side of the geometries for eg. Cutting a circle.
2. Outside offset will cause an offset on the outside of the geometry for eg. Cutting a rectangle or square.

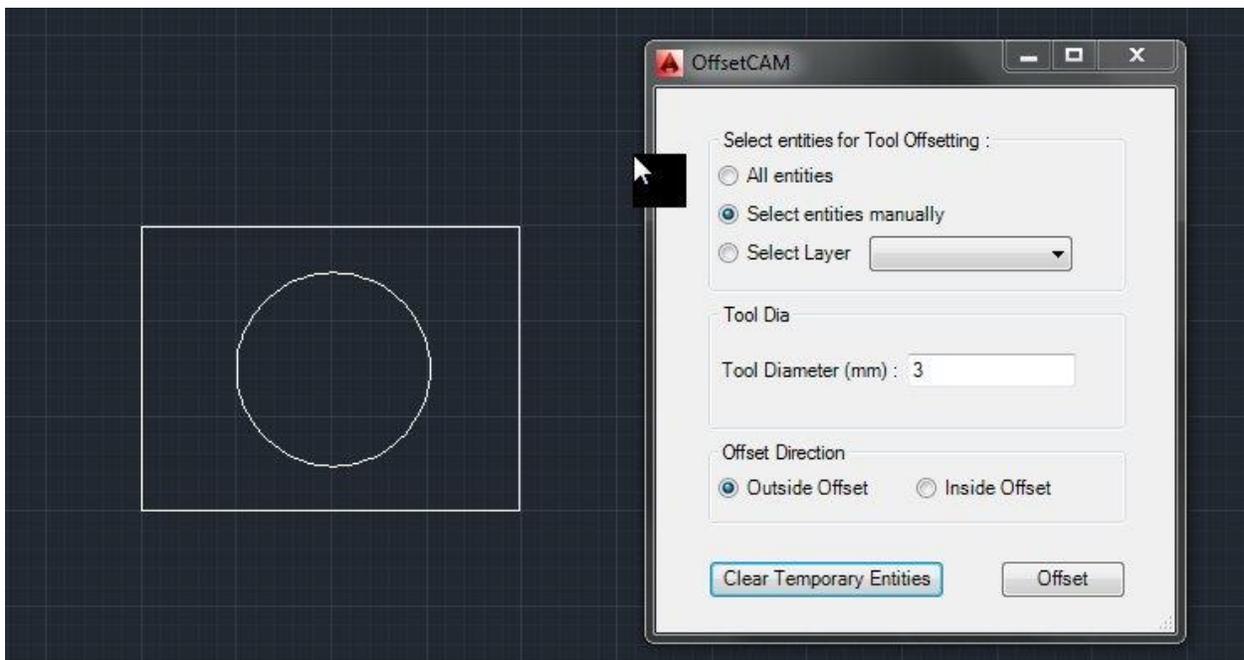
These options can be used individually when selecting the entities manually as discussed above. So we will use a tool diameter of 8mm and offset direction as “outside offset” and click “Offset”. The offsetted geometries will appear in red color as shown.



Now if you want to clear the existing geometries without any offset then click “clear temporary entities”, a dialog box will appear as shown:



Click yes to delete the temporary entities.



Now the existing geometries without any offset will be deleted and the offsetted geometries which are in red color will now appear in white color. You can use this to import in a CNC controller and convert it to a G-code file.