

The G-Code Cam app lets you convert dxf files to a G code file that can be used on a CNC machine to cut the parts. This app only generates G codes for 2 D diagrams and allows the individual selection of entities to be cut.

About:

Creates Gcode for drawings in AutoCAD file. The Gcode output can be used directly for CNC machining.

This app creates a G-code file for geometric entities in an AutoCAD drawing file. The Gcode output can be used directly in various CNC machines like milling, Lathe, Router, Plasma, Laser etc.

Features:

- *Multi-pass G-code generation.
- *Tool path optimization
- *User Selectable origin for G-code
- *Support for most commonly used geometric entities like Line, Circle, Arc, point, polyline, Ellipse & Spline.
- *Geometries can be offset automatically before generating G-code.
- *Climb & conventional milling directions can be chosen.
- * No need for an external CAM software. Powerful features integrated into the World's most powerful CAD software.

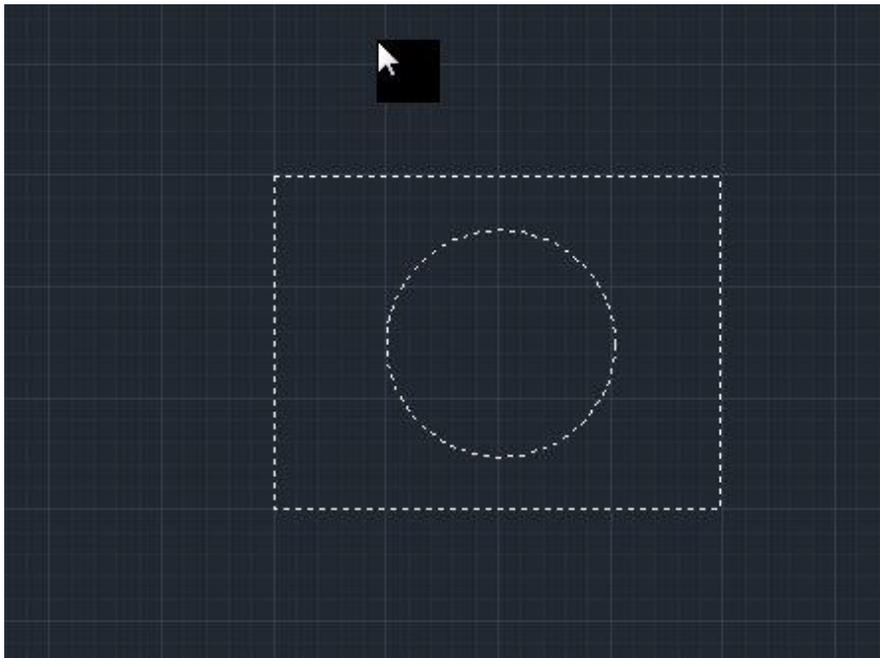
To use this app type “gcodecam” in the command line of Auto-Cad and hit Enter button. A window appears as shown:



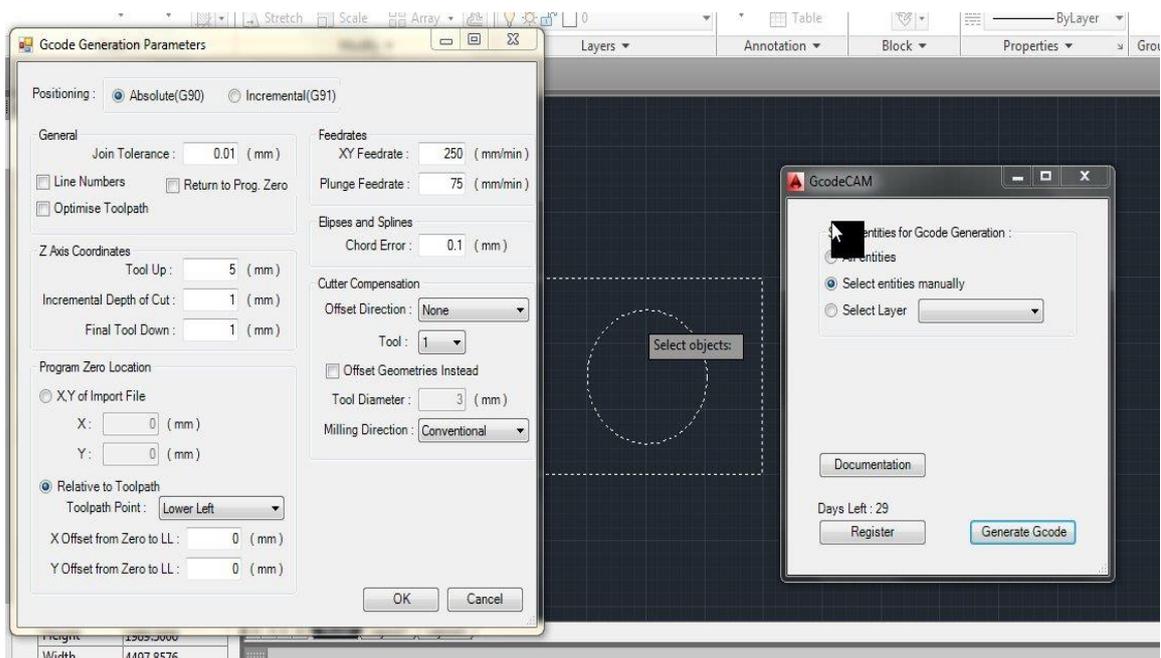
There are three options for selecting the entities:

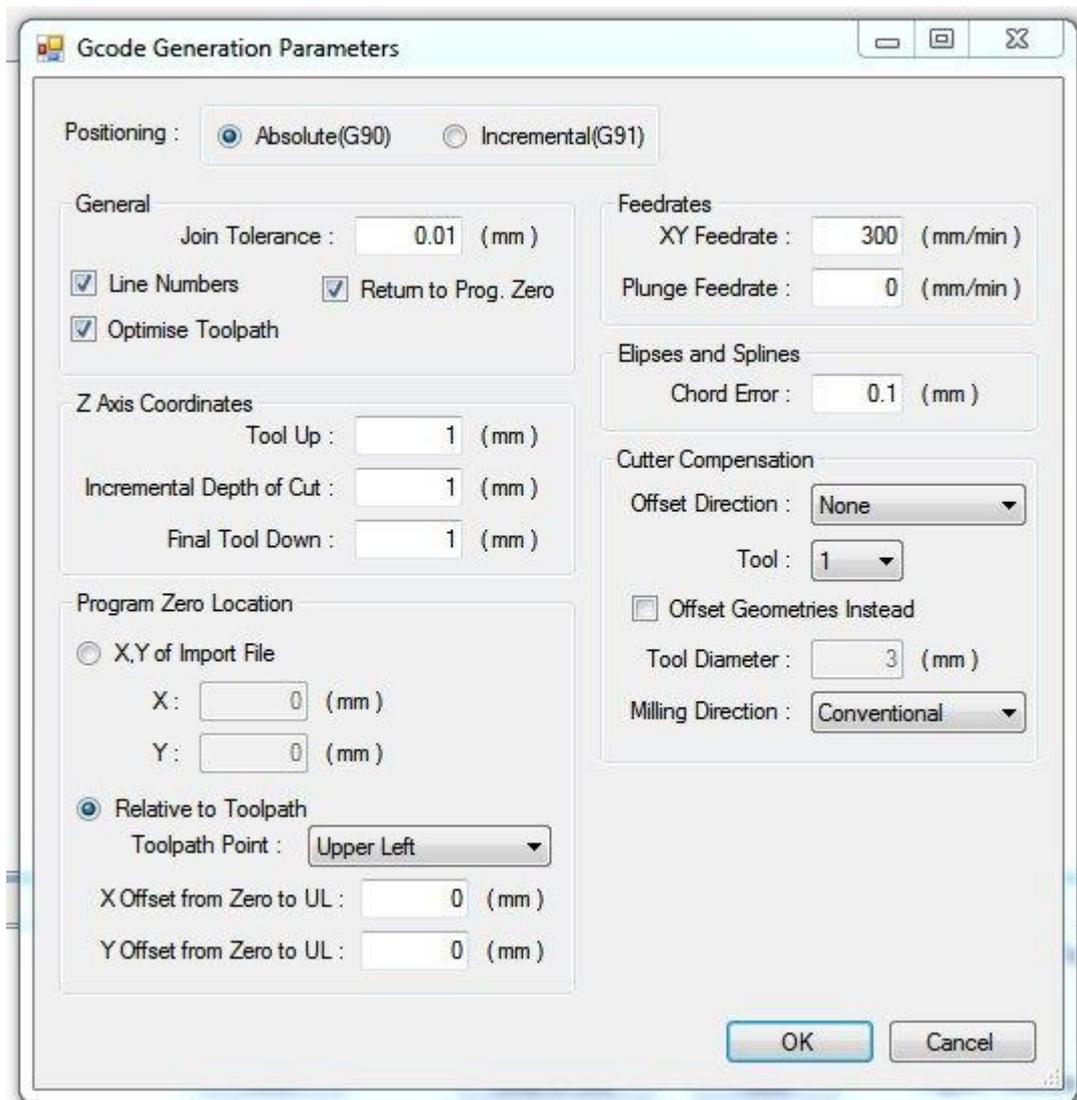
1. You can select the “all entities” option for generating the g code for all the entities in the screen you are looking/working.
2. You can use the “selecting entities manually” option to selecting individual geometries to cut which is useful when you don’t want to cut all the entities.
3. You can use the “select layer” option for selecting a particular layer in Auto – Cad to machine all the geometries in that layer.

After selecting the entities you can click the “Generate Gcode” button to generate the g code for the selected entities. The selected entities will appear in dotted lines:



When the “Generate Gcode” button is clicked the following window opens:



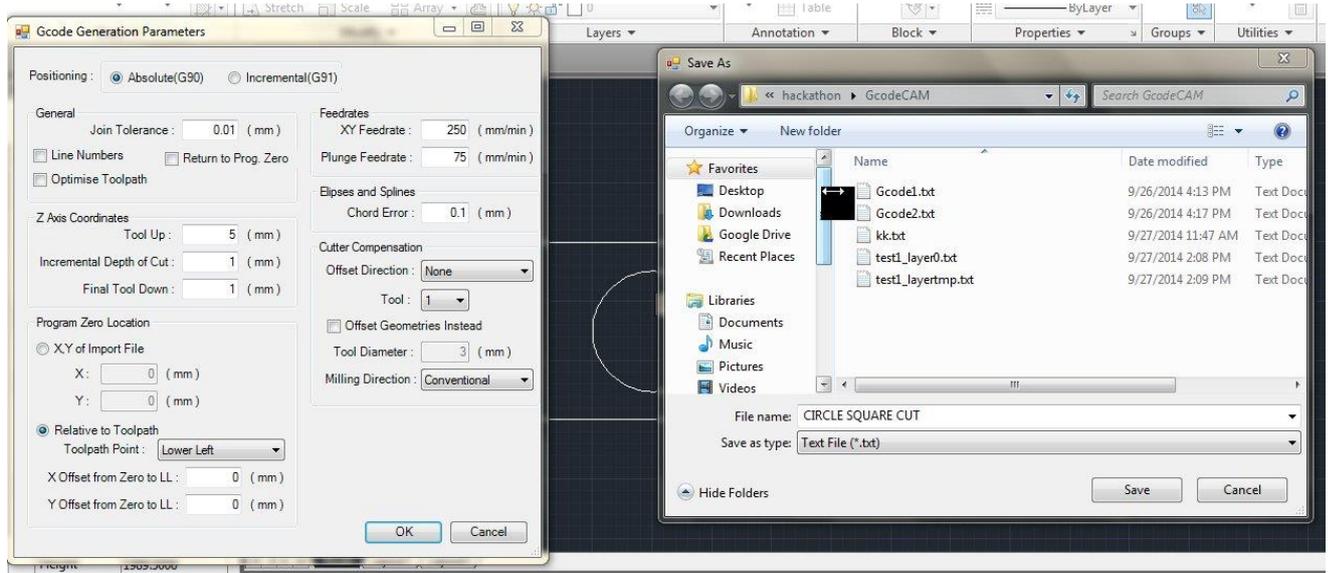


The above shown window has various parameters:

1. "Positioning" of the axes at the program Zero. This can be selected as either absolute or incremental. The default is always absolute.
2. Under the "general" box we have the following options:-
 - a. Join tolerance: it is the minimum gap between two broken lines that must be joined. If the gap is more than the specified tolerance then they will not be joined.
 - b. Next is the "line numbers". This adds line numbers in blocks of G code lines like N001, N002 etc. If it is checked then it will add the line numbers.
 - c. "Return to program Zero" will make the axes move to the program zero location after the cutting has finished if it is checked.
 - d. "Optimize tool path" will make router move in the shortest path possible between different cutting locations such that the total tool travel distance and time is reduced.
3. The next parameter is the "Z axis coordinates". We have three subdivisions under this:
 - a. Tool up is the distance the z axis must move from program zero in the positive Z axis direction between different cutting locations. This is useful to avoid any obstructions like clamps to hit the tool and break it. If it is set very high then the program run time will increase.

- b. “Incremental depth of cut “is the depth per cutting pass. If you have a thicker material like 6 mm thick then you can divide it into 3+3; so the incremental depth of cut should be 3mm.
 - c. The last option is “final tool down” which is how far the tool should go down in the negative Z axis from the program zero position. If you want a cut-out of your part then specify the “final tool down” little more than the thickness of the material.
4. The next parameter is the “program zero location” in which there are two options :
- a. X, Y of import file: this will take the zero as x and y origin in Auto-Cad which is present normally at the bottom left corner. Sometimes the user makes the geometry very far away from the x, y origin of auto-cad and if this option is selected then your part may be out of the machine limits; so it’s hardly used. However you can specify the x and y offsets from the x, y origin of auto-cad to bring it close to the origin. Even then the distance has to be measured from auto-cad and then entered.
 - b. The other option is “relative to tool path”. In this option the tool takes one of the corners of the part to be cut with the part being in any one of the 4 quadrants as in a typical graph and the corner being always the origin. The quadrant can be selected from the “tool path point” as: - upper left (2nd quadrant), upper right (1st quadrant), lower left (3rd quadrant), lower right (4th quadrant). All the quadrants are specified when the machine is viewed from the back side.
5. The next parameter is the “X, Y feedrate” which is generally how fast you want to cut the part in X,Y direction. It is always in mm/min.
6. The next parameter is the “plunge feedrate” which is how fast the tool should penetrate into the work piece. It is always in mm/min.
7. Chord error is the accuracy of the curve in a spline, ellipse etc. The smaller the value the better the curve but the G-code lines will increase and the program will take more time.
8. The next parameter is the “cutter compensation”. You need to specify the following:
- a. Direction of offset: for circles it is mostly inside and for squares to be cut out, the direction is mostly outside. For engraving the direction must not be given hence “none”.
 - b. Next specify the tool number that you are going to use for cutting. Not that this tool number must be the same as specified in your tool library.
 - c. There is a check box showing “offset geometries instead”. This is very useful for machines without cutter compensation like G41 and G42. Use this if your machine does not support cutter compensation g codes.
 - d. Specify the tool diameter to take the diameter offset of the tool.
 - e. Finally specify the milling direction either – climb or conventional milling.

When all the parameters are defined click ok button and a dialog box appears asking you where to save the generated G-Code as shown below:



Type the name of your file and click save to save it. Now go to the location where you have saved the G-Code and open it with notepad. You will see the generated G-Code.

