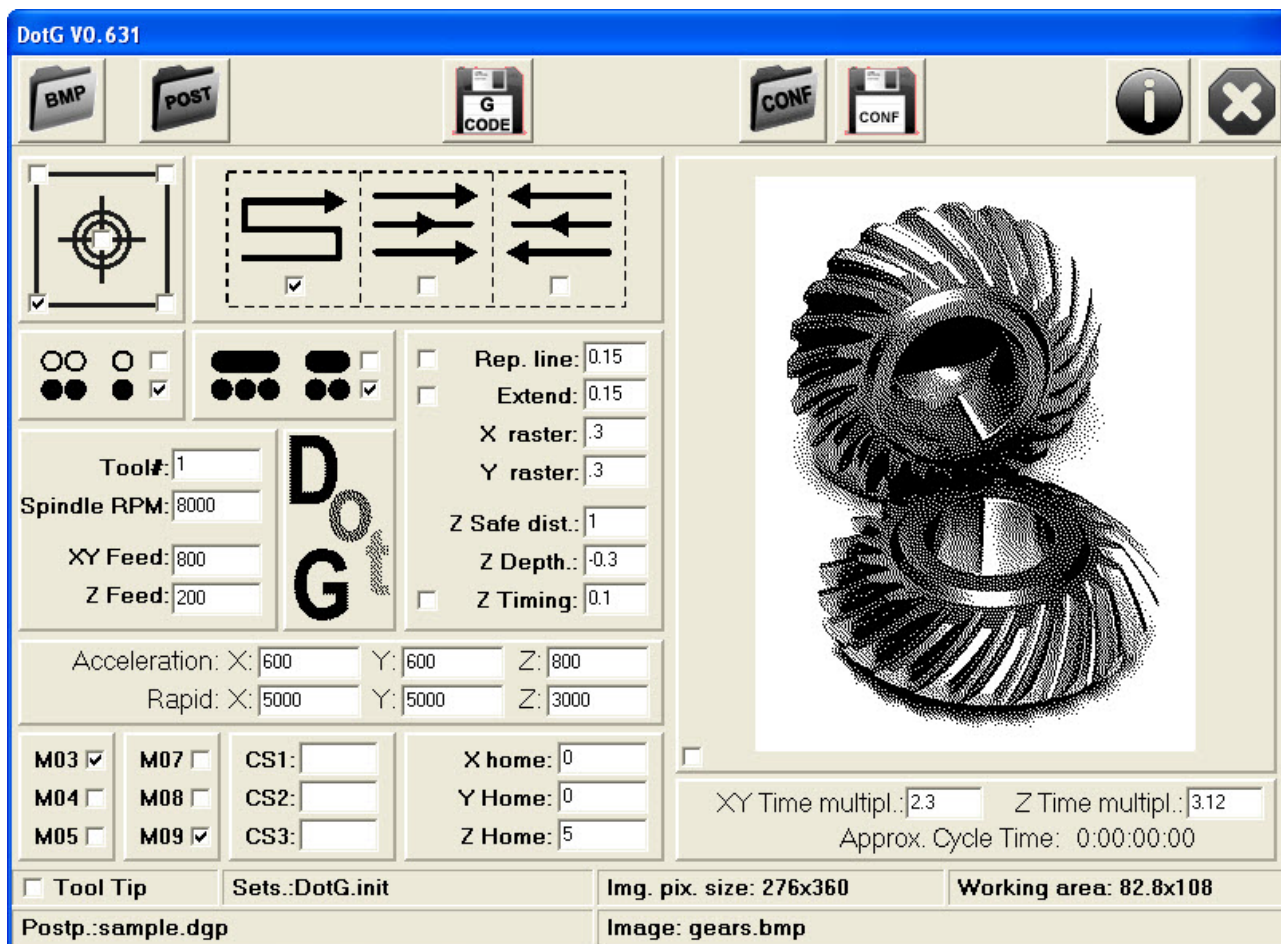# DotG
V0.631

Raster mode photo etching.

Introduction:

DotG is an extremely easy to use software which has been designed to create a machine-readable GCode which will produce a half-tone image using either a Dot or a Line technique.

The actual method chosen for producing the image will depend, to a large extent, on the material being used but these can include Dot Drill, Dot Impact, Diamond Drag, Spindle engrave, Laser and more.

To use DotG the initial image must first be prepared by converting it to a half-tone then saved to file as a 1 Bit bitmap. There are various methods and independent programs which can be used to produce the required half-tone and these will be discussed in a separate document.

Some of the best known uses for DotG include mirror etching, marble (tombstone), copper, wood, plastics and acrylic in fact almost any material can be used to carry an image or design.

The external, freely configurable, post-processor can generate the appropriate GCode to suit almost any machine controller and some ready made examples are already included in the DotG Post folder.

The legend / labels shown in the user interface are themselves fully user configurable and this is discussed later in this document.

The Graphic User Interface:

The user interface has been designed for simplicity and ease of use and the following is a summary of the various functions;

Toolbar:

BMP – Loads a bitmap image from the location of your choice. This must be a 1 bit BMP file, greater colour depths will be invalid.

POST – This will load your selected post-processor from the location of your choice - typically from the DotG Post folder.

G CODE – This generates the GCode file and saves it to the location of your choice – typically the DotG Gcode folder. Wait for the 'Code Saved' message before closing the program.

CONF – This will load a specific configuration file from the location of your choice - typically from the DotG Conf folder.

CONF – Here you can save your current configuration to the location of your choice – typically the DotG Conf folder.
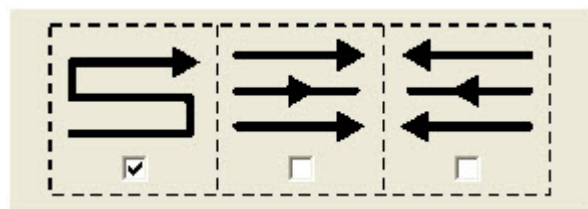
I – Information about DotG.

X – Close program.

Workpiece Zero point: Here you can choose the starting (origin) point for the work, either the centre or any of the four corners.

Machining direction: There are three different choices, zigzag, from left to right or from right to left.

Machining colour choice: This determines if you will be machining the white areas of the image or the black areas.

Machining method: This gives the choice of producing individual dots or a line joining adjacent dots. As an example, line may be used for diamond drag and dot for impact engraving.

Machine data: Here you can set the machine data which will be used when the GCode is created. Tool number, Spindle RPM, X and Y axis Feedrate and the Z axis Feedrate.

Machine parameters:

Rep.line - When selected this repeats each horizontal line a second time and the points are shifted vertically in the Y+ direction by the amount entered.
This is useful when the tool size is small compared to the image so the pixels produced can effectively be increased in size.

Extend – When selected this shifts the repeated line of points in the X direction by the amount entered. This function is especially useful for diamond drag engraving.
As an example if the Raster is set to 0.3 then the repeated line shift (Rep.line) should preferably be set to 0.15 and the Extend line shift set to the same amount.

X raster and Y raster – This is the distance between adjacent points. Normally it is set to the diameter of the tool but it can be increased or decreased as required.
Z Safe distance: This is the height the Z axis will be raised above the datum or surface of the work in-between creating points or lines.
Z Depth: The 'negative' value entered here is the tool penetration depth into the work as would be set for spindle engraving.
Z Timing: When selected this includes a G04 dwell (pause) with the specified P value entered into the generated GCode immediately following any –Z movement. This gives time for the tool to 'bite', prior to any subsequent X axis movement and would be particularly useful for plasma or low power laser work.

Acceleration and Rapid movements: The values entered here are used to calculate the Approximate Cycle time and they are expressed as mm/sec² or inch/sec²

In practice only the duration of the X and Z axis movements are used in the calculation as, in comparison, the Y axis movement times are negligible.

The exact calculation is made in the following way;

Downward movement of the Z axis will be calculated at the G01 feedrate and doubled to account for the corresponding upward movement (because the Z axis movement is extremely small, it will be within it's acceleration / deceleration zone so no allowance for rapid movement needs to be made here). Every point and line will then be added to the calculation with any movements at the Z safety level calculated as rapids and any movement at Z Depth being calculated at the G01 feedrate.

As different values can be entered into the post-processor than those considered here empirical coefficients are available as 'XY Time multiply' and 'Z Time multiply'. By suitable choice here reasonably accurate machining times can be forecast with the Approximate Cycle Time expressed as <day> : <hour> : <minute> : <second>

Output functions: Here the spindle and coolant output functions can be selected for addition to the generated GCode. As shown in this example, the spindle clockwise rotation has been selected (M03) and the coolant is turned off (M09).

String functions: There are three freely definable string functions, each of which can contain up to 30 characters. Any strings entered here will be added to the generated GCode and are particularly useful for the inclusion of start up strings, M Codes, Macro's or special control codes, for example laser power settings etc.

The strings entered here will be placed into the GCode by the post-processor in place of the variables CS1 – CS3

There are some other options / functions available within the GUI which I have not described here but I leave you to experiment and discover these for yourself.

Home position: The values entered here will be the final home position of the tool on completion of the work.

Customizing the graphic user interface:

It is a feature of DotG that the user interface text / legends can be easily customised or, for example, converted to any other language, the only restriction being that the chosen text must fit into the available space. Locate the 'LangFile.txt' within the DotG folder (if this file does not exist then copy the 'Labels.txt' file, rename it to 'LangFile.txt' and save it to the same location).
LangFile.txt contains all the labels / text / legends and tooltips - you must preserve the layout of this file (the total number of rows and their sequence) but by using a text editor such as Notepad, any text alterations made and saved here will be applied to the user interface next time DotG is started.

Notes:

The DotG generated gcode should be tested before any real work. There are a lot of good GCode simulators and most of them have built in test functions and warn of any conflicts etc.

Free CNC simulator: http://www.cncsimulator.com/

DotG latest versions, contact, user guides: http://dotg.weebly.com/

Any feedback is welcome. If you have any suggestions for improvements or corrections please do not hesitate to contact me from my website http://dotg.weebly.com/

Terms of use:

I have made DotG freely available to all, it can be used and distributed without restriction. All I ask is that you respect my intellectual property rights to this software and do not reverse engineer or copy parts of my code for use with competitive programs.

Enjoy !

Beni.

04th February 2012.