Dmap2gcode Manual

This page is a work in progress...
I will be updating as time permits. In the mean time if you have a question
you can e-mail (make sure to add a subject containing dmap2gcode)

Manual Contents

Tips from Scorch

This is the advice I would give to a friend that started using Dmap2gcode.

Open old .ngc files	Dmap2gcode can open the g-code (.ngc) files that were saved by Dmap2gcode. If you like the results you had last time or just want to make small modifications open the file you saved and work from those settings. There is no need to start over from scratch.

Command Line Options

Usage	python dmap2gcode-XXX.py [-g file] or dmap2gcode-XXX.exe [-g file]
-g (or --gcode_file)	dmap2gcode g-code output file to read
-h (or --help)	print syntax help to console window.
Example (Using python script with preinstalled python distribution)	python dmap2gcode-XXX.py -g my_file.ngc
Example (Using precompiled Windows executable)	dmap2gcode-XXX.exe -g my_file.ngc

Keyboard Shortcuts

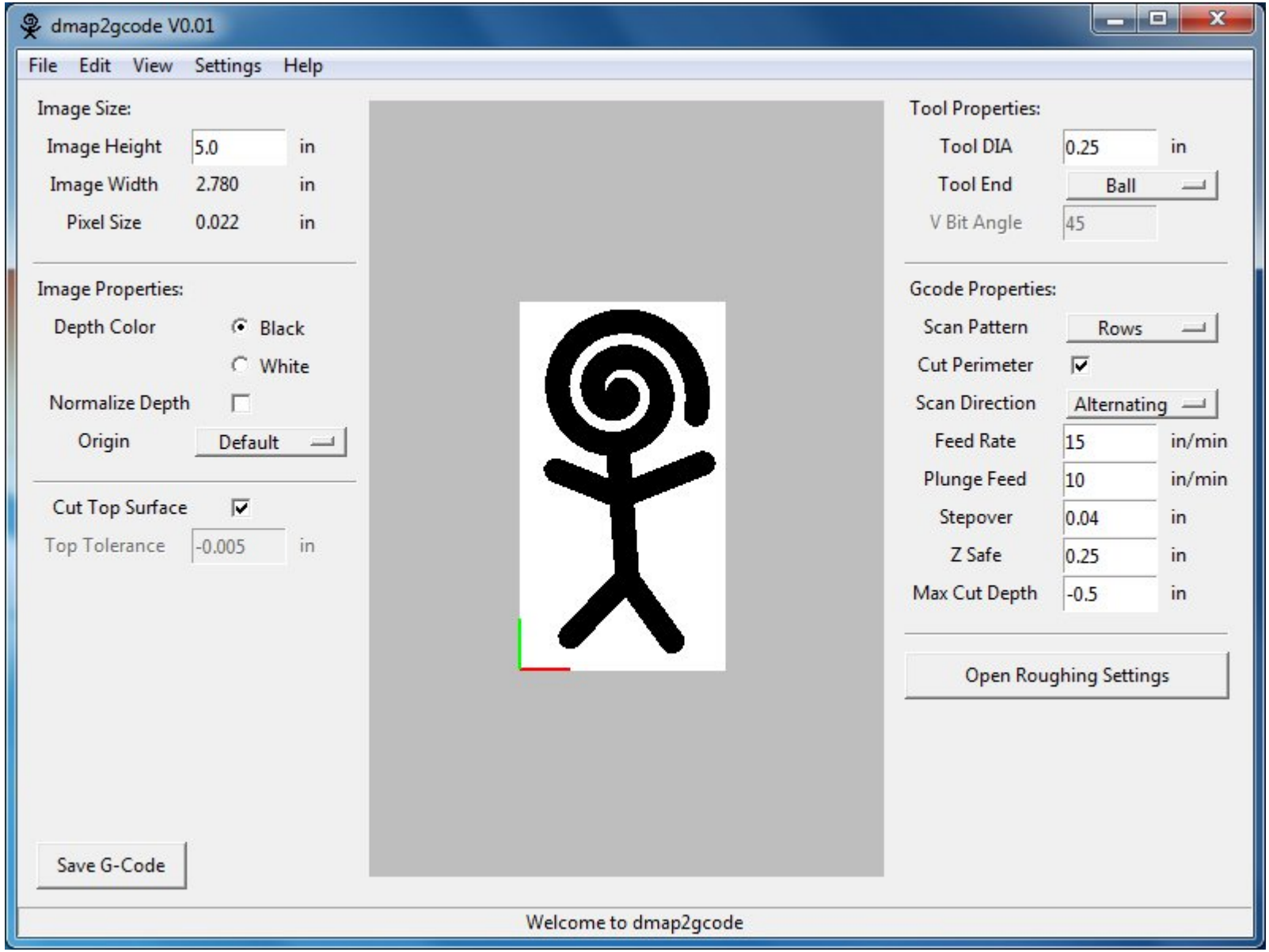
F1	Open the Help dialog box. (There is nothing in the help except a reference back to the web page and my

	e-mail address)
F2	Opens General Settings Window
F3	Opens Roughing Settings Window
F5	Refresh display

Configuration File

dmap2gcode.ngc or **.dmap2gcoderc**	When Dmap2gcode starts up the the directory in which Dmap2gcode starts in is searched for a file named "dmap2gcode.ngc". If the file is found it is read and all of the Dmap2gcode options are set according to the settings saved in the config file. After searching for the config file in the current directory the users home directory is searched for "dmap2gcode.ngc" or ".dmap2gcoderc" if either of these are found it will be used as the default settings. Only the first config file found is read. To create a config file with your preferred options for startup just open Dmap2gcode and change the settings to your liking and save a file called "dmap2gcode.ngc" in the users home directory or the directory Dmap2gcode is called from. (If you are not using the precompiled windows executable you could also just edit the settings in the .py file)

Main Window

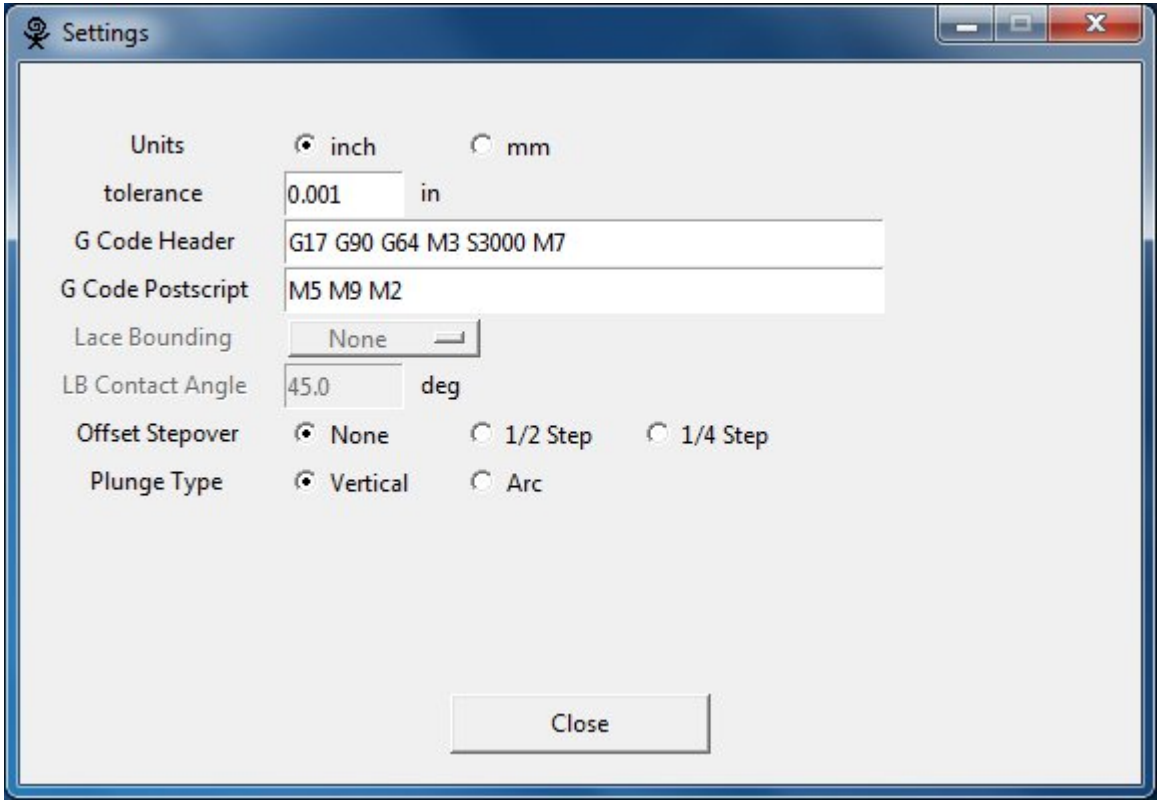

dmap2gcode V0.01
File
Edit
View
Settings
Help
Image Size:
Image Height
5.0
in
Image Width
2.780
in
Pixel Size
0.022
in
Image Properties:
Depth Color
Black
White
Normalize Depth
Origin
Default
Cut Top Surface
Top Tolerance
-0.005
in
Save G-Code
Tool Properties:
Tool DIA
0.25
in
Tool End
Ball
V Bit Angle
45
Gcode Properties:
Scan Pattern
Rows
Cut Perimeter
Scan Direction
Alternating
Feed Rate
15
in/min
Plunge Feed
10
in/min
Stepover
0.04
in
Z Safe
0.25
in
Max Cut Depth
-0.5
in
Open Roughing Settings
Welcome to dmap2gcode

Input Field	**Description**
Image Height	The size of the G-Code output for the image in the Y direction (displayed as vertical on the screen) If cutting the full image the tool will move exactly the Image height in the Y direction. This means that if you are cutting with a flat cutter the actual cut size will be the Image height plus the diameter of the cutting tool.
Image Width	This dimension is automatically calculated and displayed to indicate the width or X axis size of the image G-Code output. This dimension is a function of the image aspect ratio and the "Image Height".
Pixel Size	This dimension is automatically calculated and displayed to indicate the resulting size of a single pixel based on the selected image and the "Image height" input.
Depth Color	The depth color setting determines whether the depth of cut is determined by the darkness (black) or lightness (white) of the image. If "black" is selected the portions of the image that are black will be the deepest cuts and white portions of the image will be at Z equals zero. Conversely if "white" is selected the white areas of the image will be cut the deepest.
Normalize Depth	Selecting this option tells the program to treat the darkest pixel as if it were black and the lightest pixel as if it were white. The remaining levels of grey between are stretched fill the range.
Origin	The origin determines the relative location of the g-code x and y zero location. The current location of the origin is displayed in the display window as a red and green lines. The red and green lines follow the RGB convention Red is the x-axis, Green in the y-axis.

Cut Top Surface	If "Cut Top Surface" is selected the whole image is cut even if the cut depth is zero. If this option is not selected the top surface is not cut. The tool path will move to Z-safe and rapid past the areas not cut.
Top Tolerance	The "Top Tolerance" determines how far a calculated cut has to be below the zero position before it is not considered a top surface cut.
Tool DIA	Specify the tool diameter.
Tool End	Specifies the tool shape (Ball End, Flat End, or V-Bit)
V Bit Angle	If the V-bit tool end option is selected this option determines the included angle of the V-bit.
Scan Pattern	This option determines if g-code is generated for the "Rows" (X direction), "Columns" (Y direction) or both "R then C" (Rows then Columns) or "C then R" (Columns then Rows).
Cut Perimeter	This option adds a perimeter cut to the g-code output when only "Rows" or only "Columns" is selected for the "Scan pattern". This can eliminate a scalloped look at the edge of an image.
Scan Direction	"Scan Direction" determines the cutting direction methodology. **Positive**: Start milling at a low X or Y axis value, and move towards a high X or Y axis value **Negative**: Start milling at a high X or Y axis value, and move towards a low X or Y axis value **Alternating**: Start on the same end of the X or Y axis travel that the last move ended on. This reduces the amount of traverse movements **Up Milling**: Start milling at low points, moving towards high points **Down Milling**: Start milling at high points, moving towards low points

Feed Rate	Specify the tool feed rate that is output in the g-code output file.
Plunge Feed	Specify the tool feed rate that is output in the g-code output file when moving vertically into the material.
Stepover	The distance between adjacent rows or columns.
Z Safe	This is the z location that the tool will be directed to prior to any rapid moved.
Max Cut Depth	The depth of the deepest possible cut into the material.

General Settings

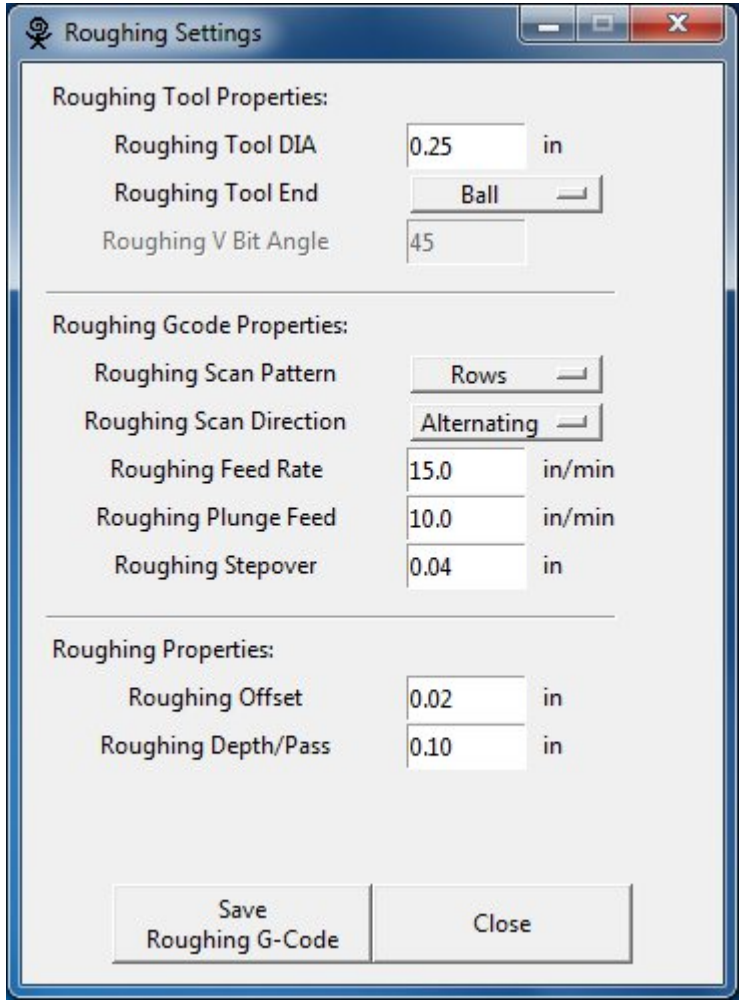


Input Field	Description
Units (inch/mm)	Set the units used by Dmap2gcode to inches or mm. This option affects the displayed units and the units output in the g-code.
Tolerance	When a series of points are within tolerance of being a straight line, they are output as a straight line. Increasing tolerance can lead to better contouring performance in LinuxCNC, but can also

	remove or blur small details in the image.
G Code Header	The text that is entered in the G-Code header entry box is added to the beginning of the g-code commands in the Dmap2gcode output file. The commands are interpreted by the CNC machine as entered. The default values (G17 G90 G64 M3 S3000 M7) are explained below: G17 Sets XY plane G90 Fixed cycle, simple cycle, for roughing (Z-axis emphasis) G64 G64 without P option keeps the best speed possible, no matter how far away from the programmed point you end up. M3 S3000 Spindle start at 3000 M7 Turn mist coolant on (if it is available)
G Code Postscript	The text that is entered in the G-Code Postscript entry box is added to the end of the g-code commands in the Dmap2gcode output file. The commands are interpreted by the CNC machine as entered . The default values (M5 M9 M2) are explained below: M5 Stop Spindle M9 Turn all coolant off M2 End Program
Lace	"Lace Bounding" controls whether areas that are relatively flat along a row or column are skipped.

Bounding	This option only makes sense when both rows and columns are being milled. Possible bounding options are: **None**: Rows and columns are both fully milled. **Secondary**: When milling in the second direction, areas that do not strongly slope in that direction are skipped. **Full**: When milling in the first direction, areas that strongly slope in the second direction are skipped. When milling in the second direction, areas that do not strongly slope in that direction are skipped.
LB Contact angle	When Lace bounding is not None, slopes greater than Contact angle are considered to be "strong" slopes, and slopes less than that angle are considered to be weak slopes.
Offset Stepover	This option allows offsetting the stepover by 1/2 or 1/4 step. If you cut an image and find that the scan lines were not close enough together switch this option to 1/2 and generate a new g-code file. The new file will have the same stepover, however, the steps will be offset by 1/2 step so that the scan lines will be between the previous scan lines. Repeating the process with 1/4 step will automatically half the step size and cut between all of the previously cut scan lines
Plunge Type	Specifies whether a straight vertical plunge or an arced plunge is to be used for entry into the cut. (NOTE: The arc entry may cause more material to be removed at the entry point. Creating a scallop outside of the image (height + tool diameter) or (width + tool diameter) boundary)

Roughing Settings



Input Field	Description
Roughing Tool DIA	Analogous to the "Tool DIA" specifically for the roughing operation.

Roughing Tool End	Analogous to the "Tool DIA" specifically for the roughing operation.
Roughing V Bit Angle	Analogous to the "V Bit Angle" specifically for the roughing operation.
Roughing Scan Pattern	Analogous to the "Scan Pattern" specifically for the roughing operation.
Roughing Scan Direction	Analogous to the "Scan Direction" specifically for the roughing operation.
Roughing Feed Rate	Analogous to the "Feed Rate" specifically for the roughing operation.
Roughing Plunge Feed	Analogous to the "Plunge Feed" specifically for the roughing operation.
Roughing Stepover	Analogous to the "Stepover" specifically for the roughing operation.
Roughing Offset	The "Roughing Offset" is the thickness of material to be left behind after the roughing operation os complete.
Roughing Depth/Pass	The maximum depth of cut four each roughing pass.

Scorch's Home