**This Gcode combining software was written specifically to
be used with PicEngrave Pro® 4, 5 & 6 + Laser programs.**

Instructions for PicLaser 3D® Gcode combining software.

This program will combine different axis movements from a 2.5D or 3D spindle engraving Gcode files to be shaded afterwards using an Analog Modulated or PWM Laser Diodes. The key for this combining Gcode software to work properly, these instructions need to be followed very closely.

You will need to generate two Gcode files with two image files the exact same size and aspect ratio. The first image file needs to be a grayscale Depth Map image to generate a Gcode for the 2.5D or 3D spindle relief engraving with the Z axis used for the depth in the file. The second image file needs to have shades for the ability of the Laser Diode to use analog modulation or PWM with the A,B or C axis letters. With GRBL and some other CNC controllers, the S laser power command can be used for PWM.

There are two ways to get these two separate images for this process. Either the Depth Map image needs to be created from an original photo, or by using a computer generated Depth Map image and editing it for the Laser Diode's shading. Here is one example.

Original Image

Original Image edited/manipulated into a Depth Map

Three total Gcode files need to be generated for this Relief Shading process. First thing is using the Depth Map image to generate a Gcode for the 2.5D or 3D spindle relief engraving. This file can include edge cutting, engrave Ellipse, Crop, safe retract height to miss clamps, feed rate for cutting tools used, feed rate change and setting comments in the file. Engraving angles that should be used is horizontal or vertical. 45 degree is not recommended. All three of these files should use the same engraving angle. It must be set with the same Pixel Resolution (step over), cutting angle direction that the next two files will be generated with. The max Z axis cutting depth must be the same as the second Gcode file.

The second Gcode file will be generated from the same Depth Map as the first file. The Z axis letter must be used in this file to have the laser diode's focal point follow the spindle engraving relief's contours. The settings in this file must be set at the same Z axis depth, pixel resolution, safe retract height and cutting angle as the first file was generated. No edge cutting or feed rate change options should be used to generated this Gcode file. Set the feed rate as you would for the Analog/PWM laser shade engraving file. Engraving angles that should be used is horizontal or vertical. 45 degree is not recommended. All three of these files should use the same engraving angle.

The third gcode file for the laser diode will be created from either the original photo the spindle engraving Depth Map image was generated from, or the edited computer generated Depth Map image file to get the shading . The settings will be the same as the second file, except the max depth which will need to be set based on your Analog/PWM laser settings. The axis letters A,B or C should be used to control the laser diode's varied power in this file. Or with grbl, the S PWM laser power command. The feed rate change option can be used in this file also and PicLaser 3D® will automatically combine this if it's present. Engraving angles that should be used is horizontal or vertical. 45 degree is not recommended. All three of these files should use the same engraving angle.

If Crop, Ellipse and or Extended edge was used in the first & second spindle relief files created, then the slider settings need to be identical in the third Analog/PWM laser Gcode file.

After starting PicLaser 3D®, select File then Open and select either the second or third Gcode file that was generated. It will come up in both file selection windows. Then open the other of the second or third file and it too will also come up in both selection windows.

In the top selection window for the laser file (file 1), single click the Analog/PWM laser power Gcode file so it is highlighted and patiently wait for the Gcode file to load in the File1 window below. After the Gcode appears in that window and says "Done", in the relief selection window (file 2) single click and select the relief file so it's highlighted. Patiently wait for that Gcode file to load and appear in file 2 window below.

Scroll down through to the bottom of both Gcode files to confirm the line count are exactly the same. If they are not, one file may have more comment characters then the other file so you must confirm this. After confirming the difference is in the Comment Character count, then you can proceed in combining these two files. Below the "Save File",type in the Comment Character used.

If both line counts are not the same after confirming the difference, then go back and check your settings in PicEngrave Pro® and generate the Gcodes again. Once you have confirmed the line count are equal, then select Start. When the message window appears, select OK and the combining of these two files will begin. When processing the combined file, PicLaser 3D® may appear to be not responding, but just be patient. This program requires allot of your PC's resources to combine these two files. When it says "Finished", a new file will be created in the same folder where the Analog/PWM laser file was selected from, but "combined" will added to the laser's file name.

Final results on Poplar

The demo version will only Create 500 lines of GCode. A purchased Registration License key will need to be input into the Options/Register before the restriction is disabled.