Profile To Helix (Profile2Helix) Plugin
[Vers 4.0.0]

Purpose
To take a user drawn polyline that represent the profile of a rotated shape, then produce helical toolpaths in order to create g-code. Toolpaths consist of; Finishing, Final Roughing, Waste Clearance Roughing.
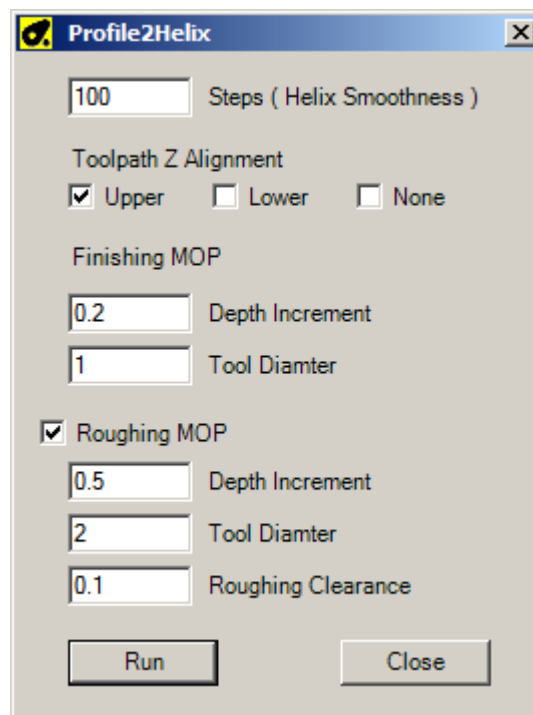
Roughing and Finishing toolpaths are created to compensate for tool diameter, however that works for Ball Nose tools only.

Installation
The Profile2Helix.dll file should be placed into the CamBam Plugins folder. On re-start the plugin will be found in the CamBam Draw menu as 'Profile To Helix'

User Interface

This is the user interface:

The fields are:

- *Step* : the number of short polylines that make up one turn of the helix, the higher the number the more smooth is the helical toolpath. It's best to keep this as low as possible while still producing acceptable results.

- *Toolpath Z Alignment* : this works in the same way as the CamBam 'Align' function
  Upper – aligns top of the toolpaths to Z0;
  Lower – aligns bottom of the toolpaths to Z0;
  None – aligns the toolpaths to wherever the profile polyline is positioned.

- *Finishing MOP :* creates and partially populates a Finishing Machining Operation (MOP).

- *Depth Increment :* how far the tool is moved down in one complete helix

- *Tool Diameter :* diameter of the finishing tool
- *Roughing MOP :* creates and partially populates a Roughing Machining Operation (MOP).

- *Depth Increment :* how far the tool is moved down in one complete helix

- *Tool Diameter :* diameter of the roughing tool

- *Roughing Clearance :* the amount of stock to leave after the final roughing cut, remaining stock is typically removed later in a finishing pass.

- *Run :* Once all fields of the Form have been completed, click this button to run the plugin and produce the toolpaths.

- *Close :* Click this button to close the Form without running the plugin

The Form parameters are saved in Windows' Registry.

Profile Polyline Requirements

A profile polyline, shown in red, produced toolpaths, shown in green. Toolpaths are created so that cutting takes place from the top down along the Z axis. Note, the bottom left point of this profile line is at X0,Y0,Z0, this produces a 'solid' object rotated around the Y axis.
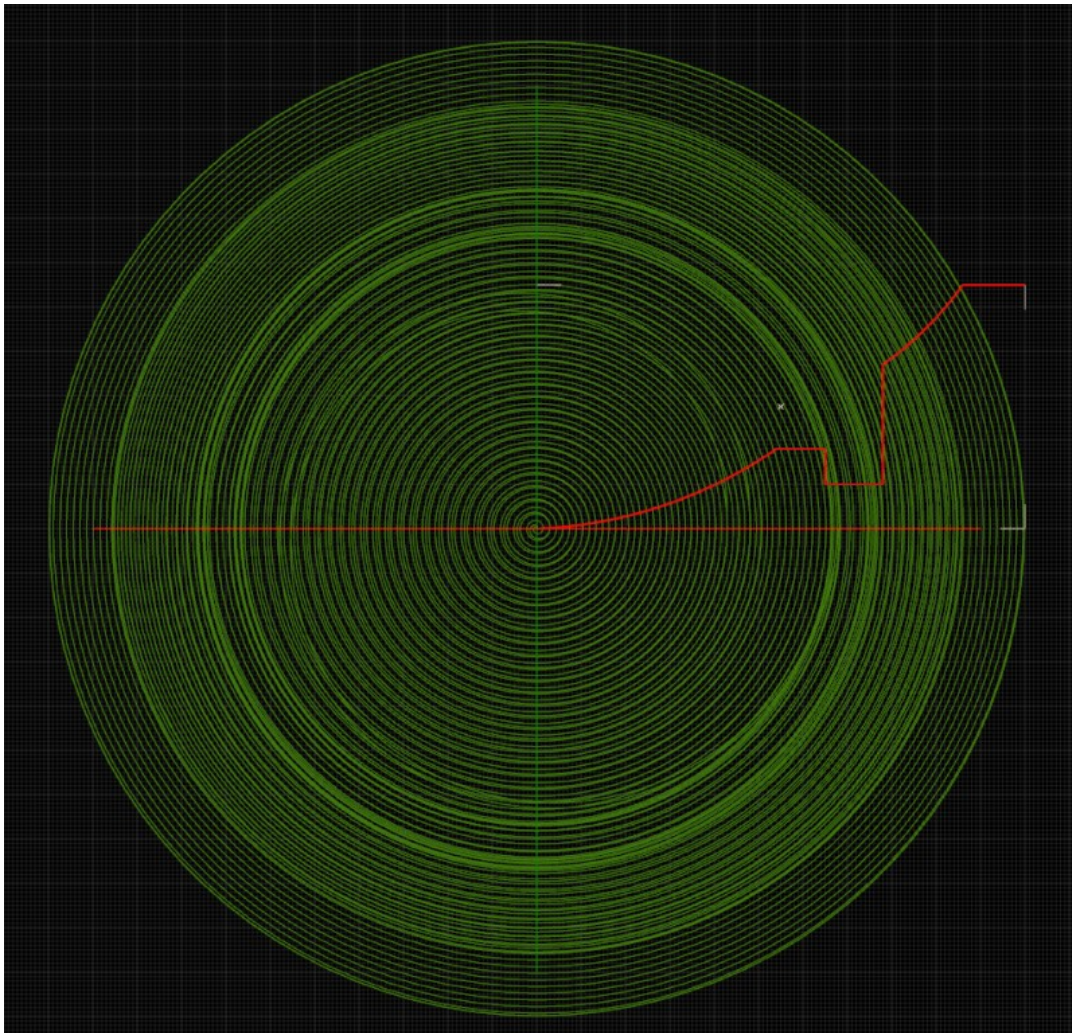
And rotated, it looks like this, red shows roughing toolpaths, green shows the finishing toolpath.
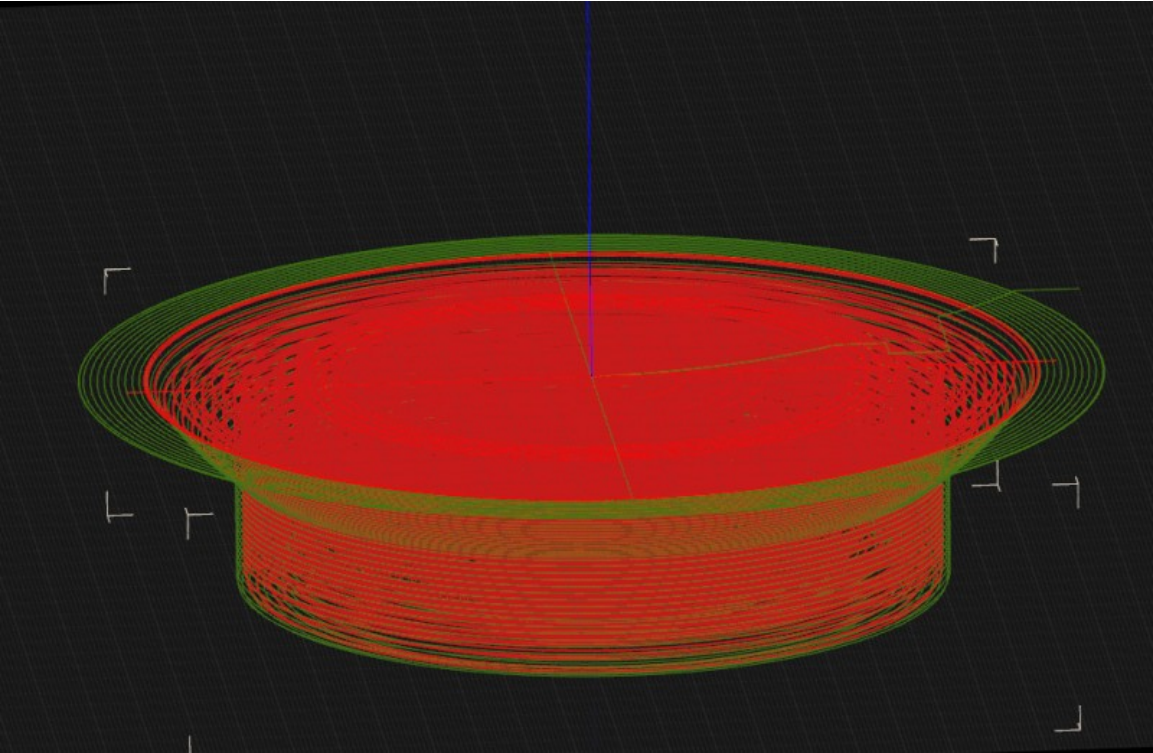
And from the simulator.
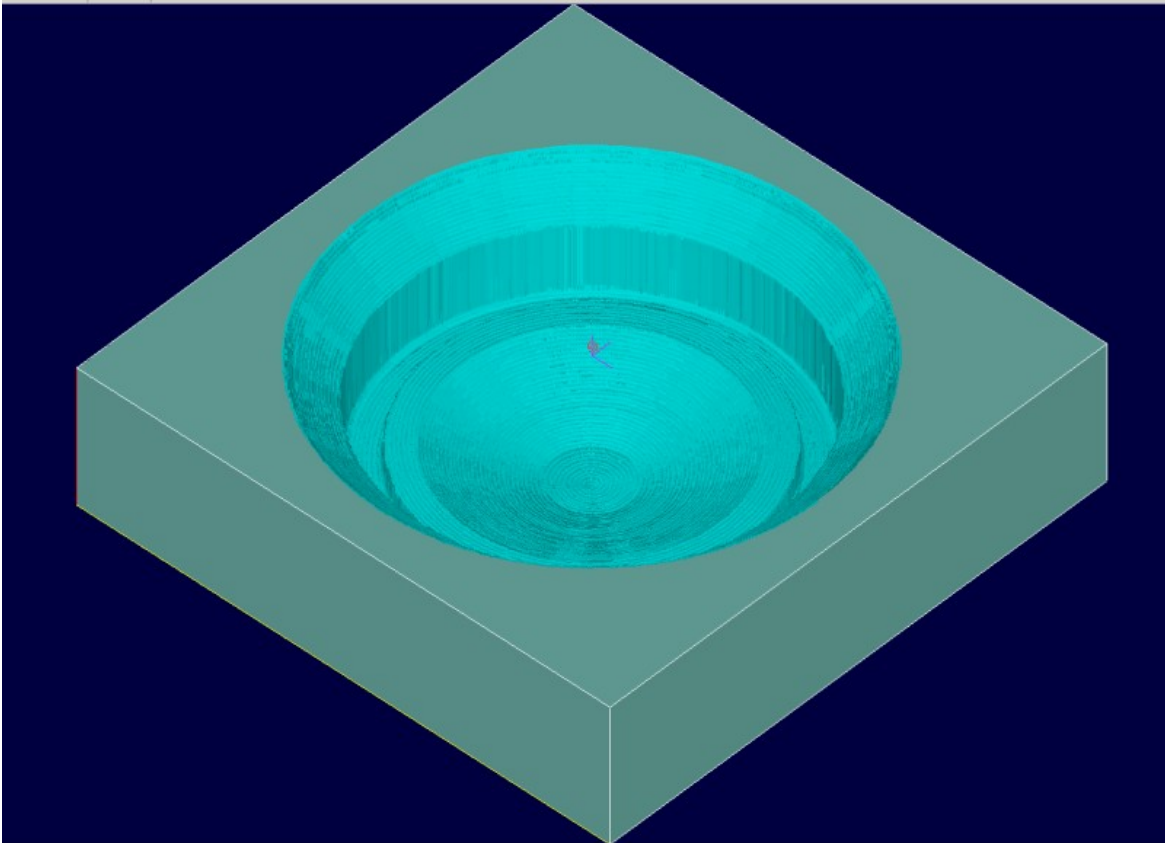
By offsetting the profile polyline, an object is created with a central hole having a diameter of twice the X distance from the profile polyline bottom left point to the Y0 axis.
This may be useful but the plugin was created to use the central zero point as the profile polyline starting point.
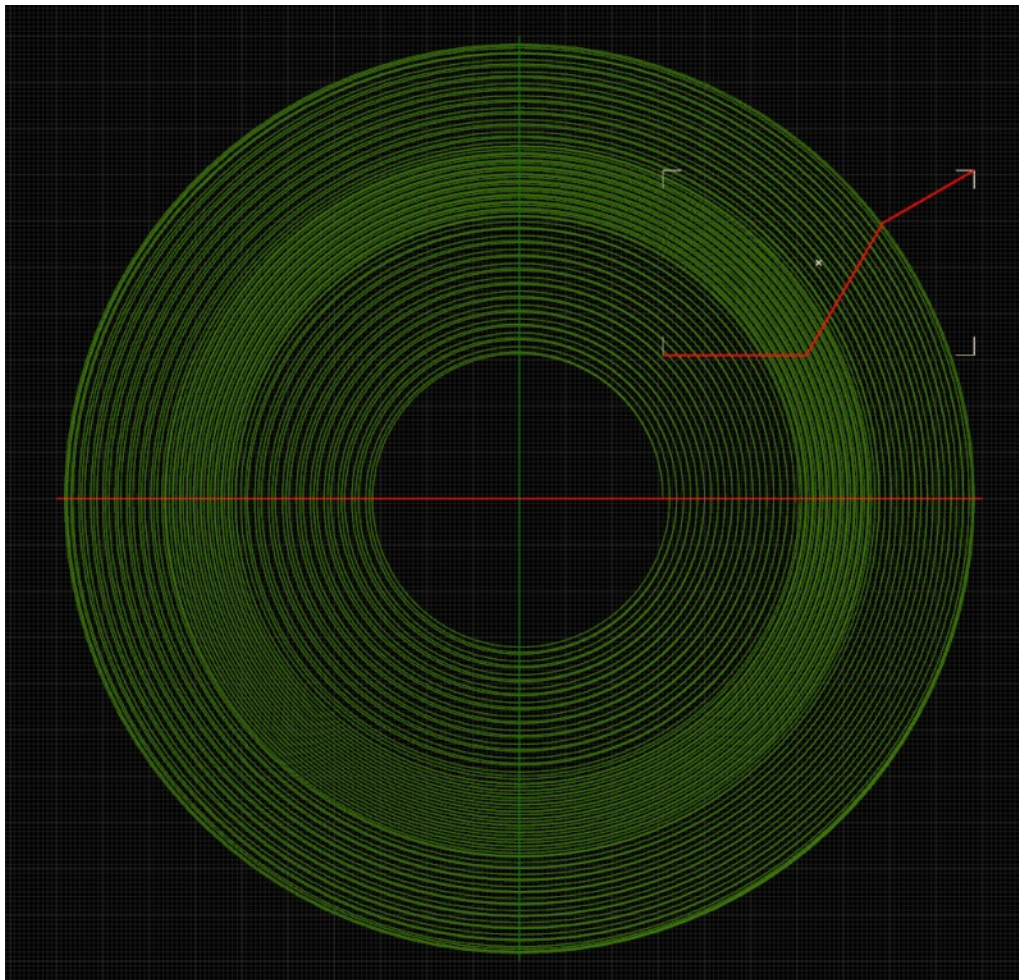
Profiles such as those shown below are not allowed.
1. Where the top end turns left, it should continue in a direction to the right or vertical.
2. Where the end of the polyline dips back below the highest point of the polyline.

Layers, Parts, MOPs

The plugin creates a new Part, new Layers and new MOPs each time it runs.

Created MOPs should be checked manually to ensure all parameters are consistent with the target CNC machine.
Please note that the plugin populates Tool Diameters without regard to Tool Libraries that may be in place, you may therefore have to manually select a Tool Number from your Library that points to a ball nose tool of the correct diameter.