



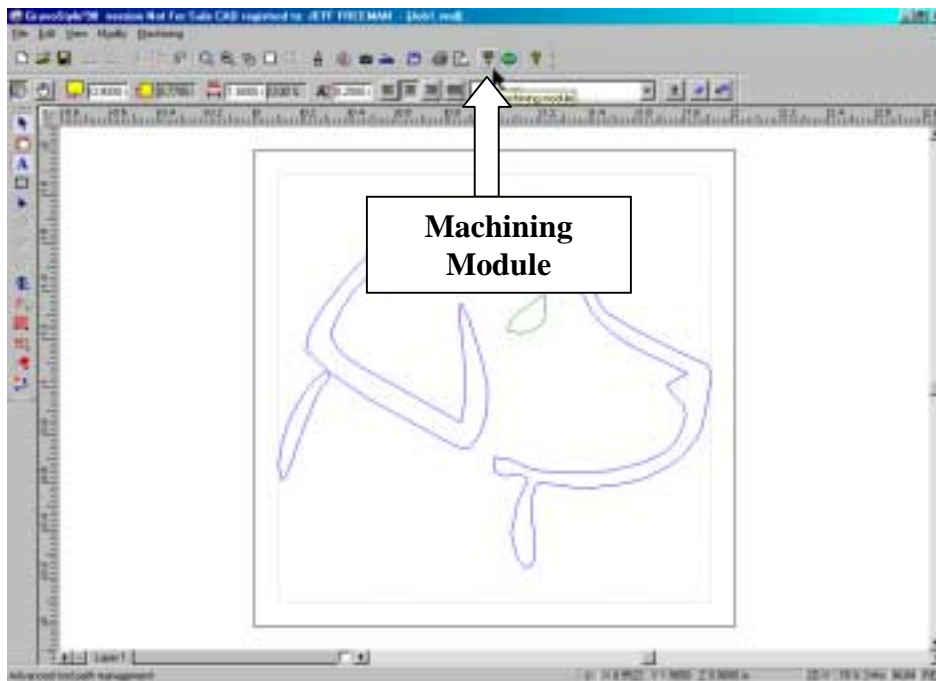
GRAVOGRAPH  NEW HERMES

ENGRAVER'S

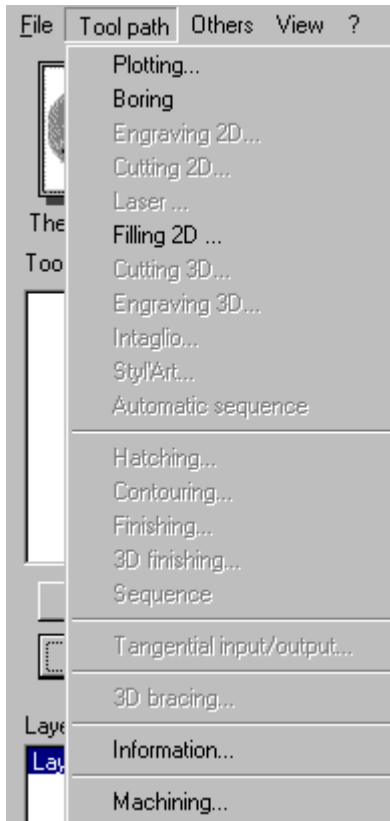
A C A D E M Y


Machining Module

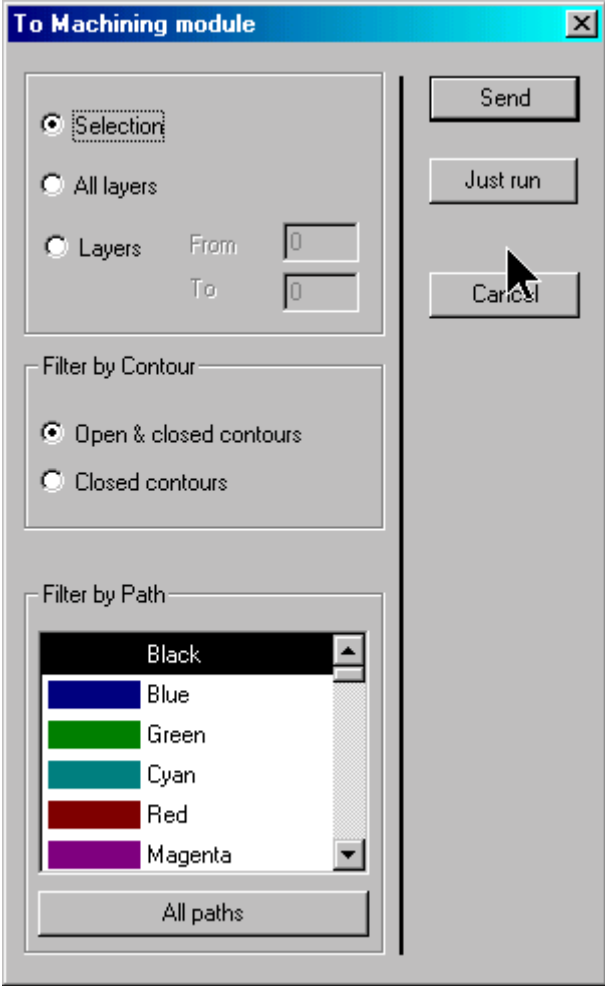
1. In the Graphic Level of Gravostyle you have access to the Machining Module, which enables you to define and combine different tool paths to start and finish a sophisticated or industrial engraving. Each tool path is calculated in relation to the contour of an object, depending on the tool parameters and the type of path chosen. It enables you to have far greater control of the engraving tool path.
2. Save the job before you take it to the Machining Module. When Tool paths are created in the Machining Module a Geometric Tool path will be generated in the same folder that the job is save in. It will have the same name as the job but will be followed by .P00. Select the part of your job that you want to bring into the Machining Module. If you create a Tool path in the Machining Module, that Tool path will generate to everything that is brought in the Module. Click on the Machining Module icon .



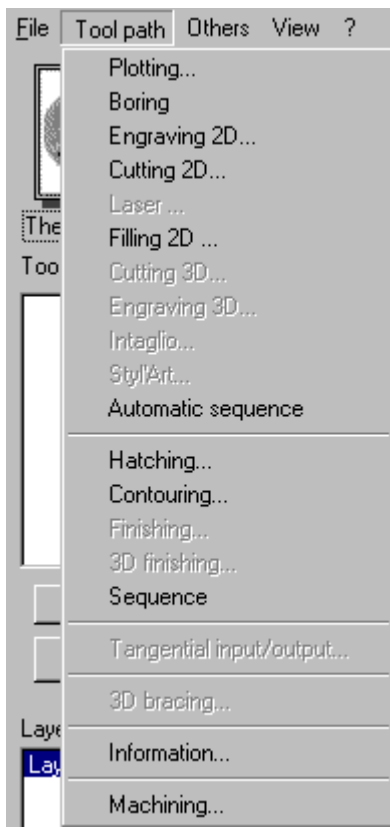
3. If you click on the Machining Module icon you will be restricted to what tool paths you will have access to. Once the object is brought into the Machining Module, click on Tool path and you will see that you are very limited to the tool path that you can create. I.e. Boring, Plotting and Filling 2D.



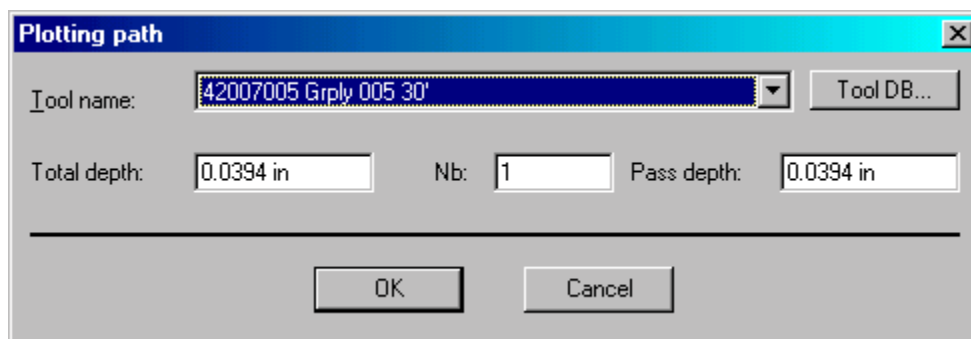
- 4. If you click on the Machining Module icon  while pressing on the Control key you will have a far greater selection of tool paths. The Next page that you will see is the To Machining Module page. This is where you will choose what you selected object is i.e. open and Closed contours or just Closed contours. Then click on the Send button.



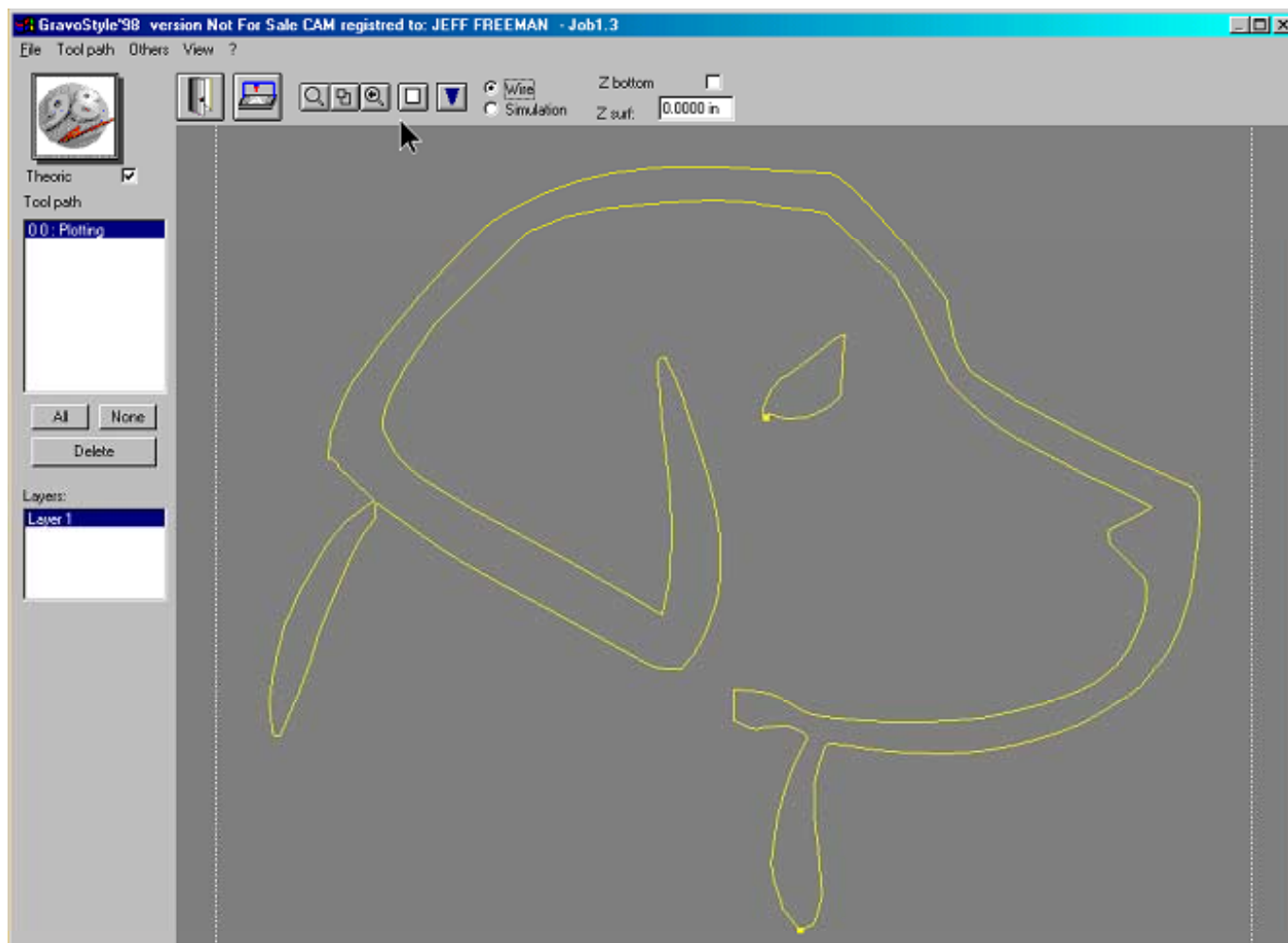
5. Now when you click on the Tool path drop down menu you will see the other tool paths that are available.



- The Plotting Tool path will follow the contours that you have in the Machining Module. Enter the Depth that you want to engrave and hit the Tab key to take effect. If you want create multiple passes to you tool path change the Pass Depth to a lower number.

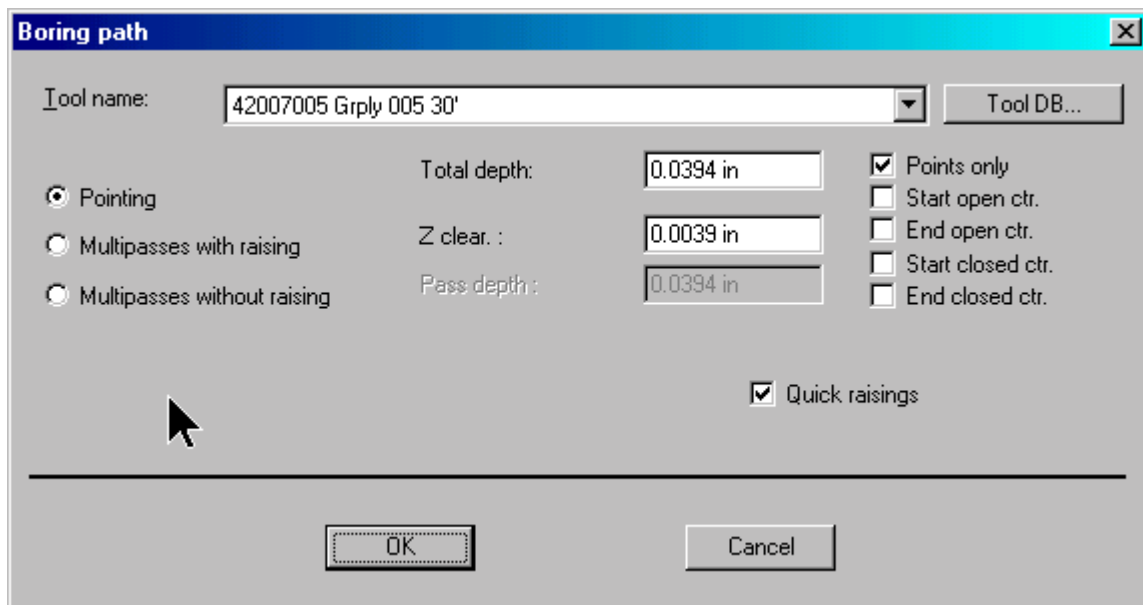


Click OK and the path will be generated.



If you wish to see what the width of the tool path will look like then click on the Simulation circle. This will depict the rotation of the tool.

7. The Boring Tool path is used for drilling holes into your piece. If you have placed Drilling Points into the Machining Module this is the page that you can control the drilling stroke.



Choose the tool that you are going to use and set the Total Depth that you are going to drill.

Pointing- The tool will drill into the Material and keep a steady pace in the down stroke, not Raising up or stopping until it gets to the Total Depth set by you.

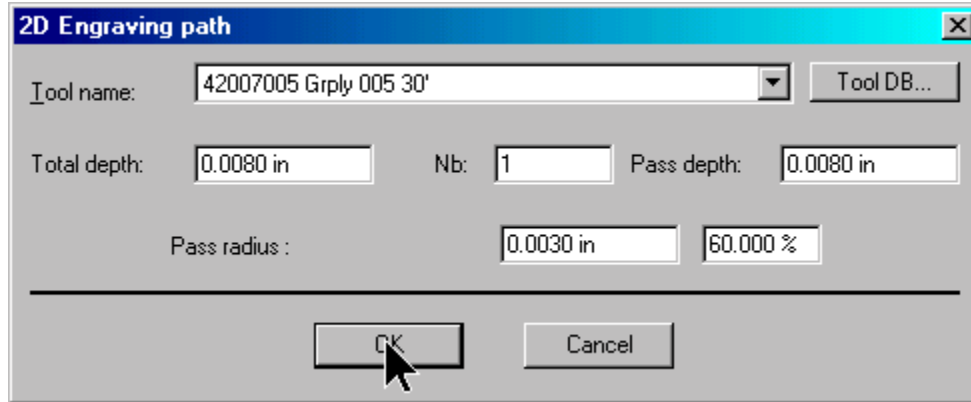
Multi-passes with Raising- The tool will drill a portion of the of the Total Depth and then Raise to the height that you set in the Z Clear. Creating a pecking motion.

Multi-passes without Raising- The tool will drill a portion of the Total Depth and then pause in place and then drill again until it reaches the Total Depth.

Quick Raisings- with the Multi-passes with Raising selected, the raises will be fast but the drill Speed will remain the same.

Points Only- Boring is for the points only. If you are using contours it is not a true boring stroke.

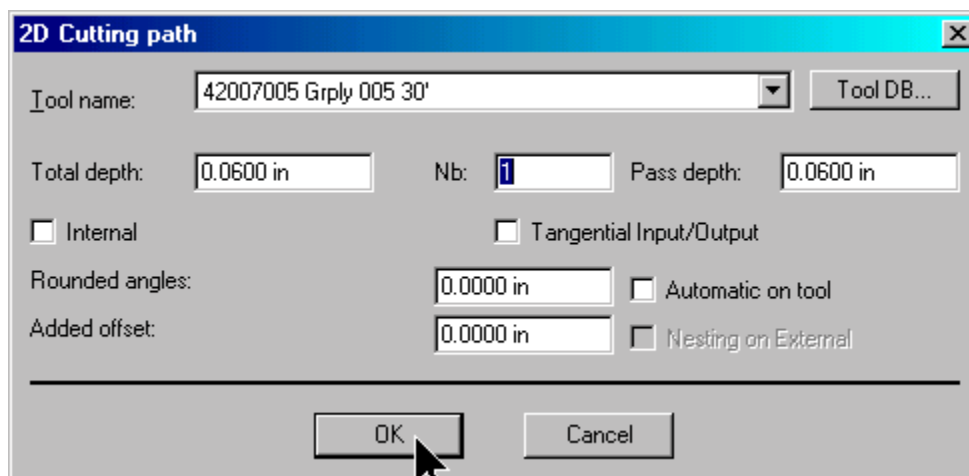
8. The Engraving 2D will give you a 2D fill in the middle of your contours, using the contours as boundaries. Select the tool and the depth that you want to engrave. Press the Tab key and your changes will take effect. If you want to make multiple passes then enter either Nb. Of passes or the Pass Depth and press the Tab key. Enter the Pass Radius or the Percentage (how far you want the tool to step over for the next path). Either one will change the other by pressing the Tab key.



Press the OK button to generate the tool path.



9. The Cutting 2D is for cutting your piece out of the material. It will either cut to the outside of the contour or on the inside of the contour. It will also allow you to round the corners and adjust an offset from the contour. Tangential In/Out will allow you to enter the material away from the contour. Cutting 2D is most often used when manufacturing ADA signage, when using the inlay method. Rounding the corners allows the pieces to fit together properly with no spaces.



Select the tool and enter the Total Depth and Nb of passes.

If no mark is in the Internal box then the cut will be on the outside of the contour. If there is a check in the box the cut will be on the inside of the contour.

Tangential Input/Output means that the tool will drill down to depth outside of the intended tool path and go into the path at an angle. Automatic on tool means that the rounded corners will be calculated on the geometry of the tool.

Rounded Angles will round off your corners to make piece fit together with no spaces.

Added Offset gives you the ability to add more space between your contour and the cut path.

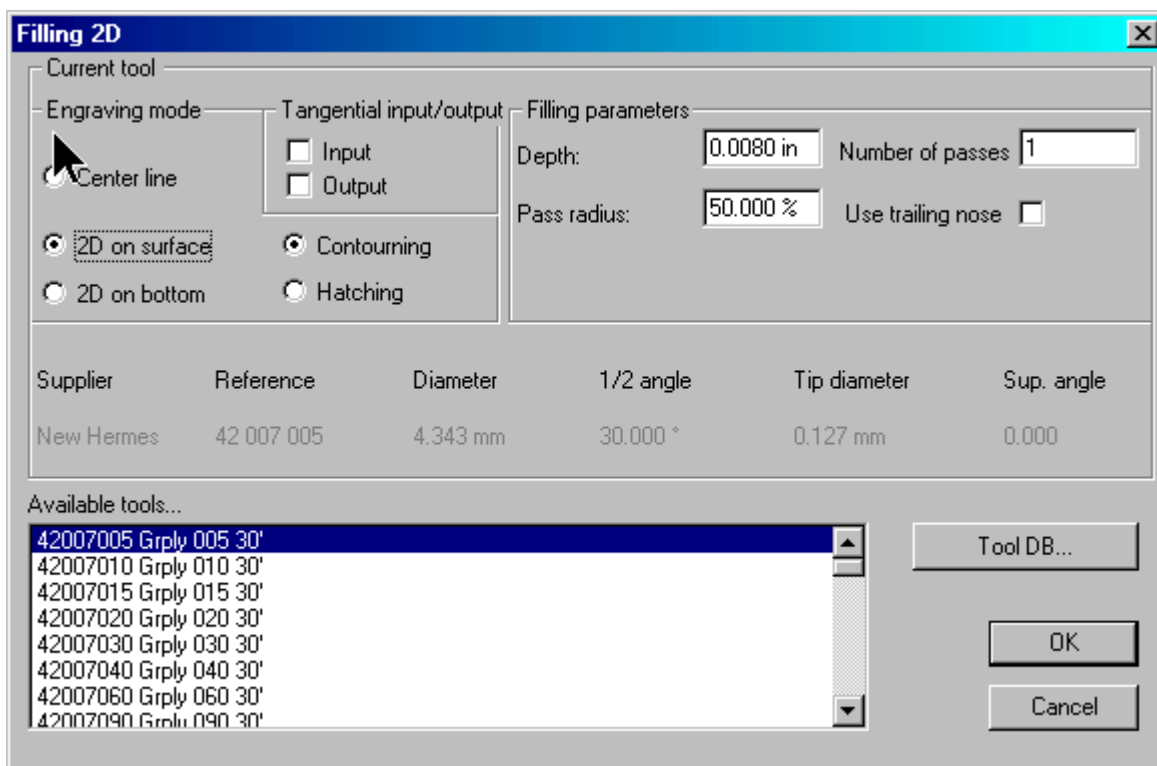
Outside Cut



Outside Cut with Tangential



10. With the Filling 2D tool you can accomplish all the above functions with this one tool. You can plot, cut, 2D Contouring and 2D Hatching on surface and on the bottom.



Select tool from the Available Tools window and the Depth and Pass Radius.

Center Line will give you a plotting path.

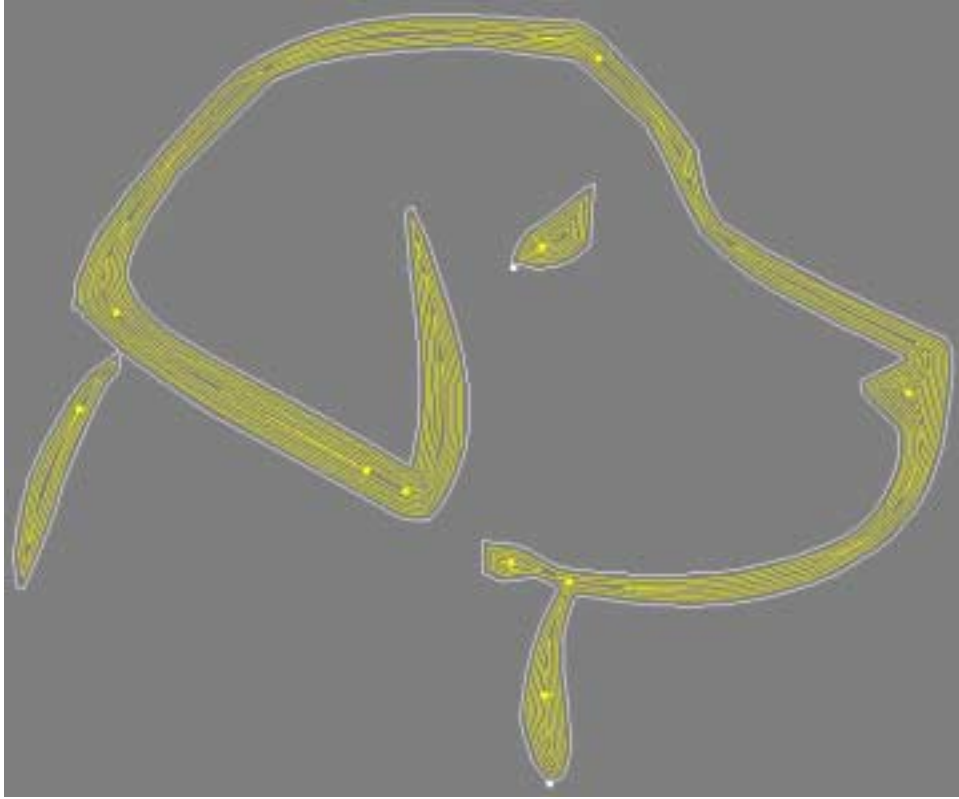
2D on Surface will give you a fill that is projected at the surface of the material.

2D on Bottom will give you a fill that is projected at the bottom of your material at the depth that you assigned.

Tangential Input and Output selects whether you want the tool to drill down to depth out away from your tool path. Then move into the path.

Selecting Use Trailing Nose will disable the Hatching Mode and the Tangential In/Out.

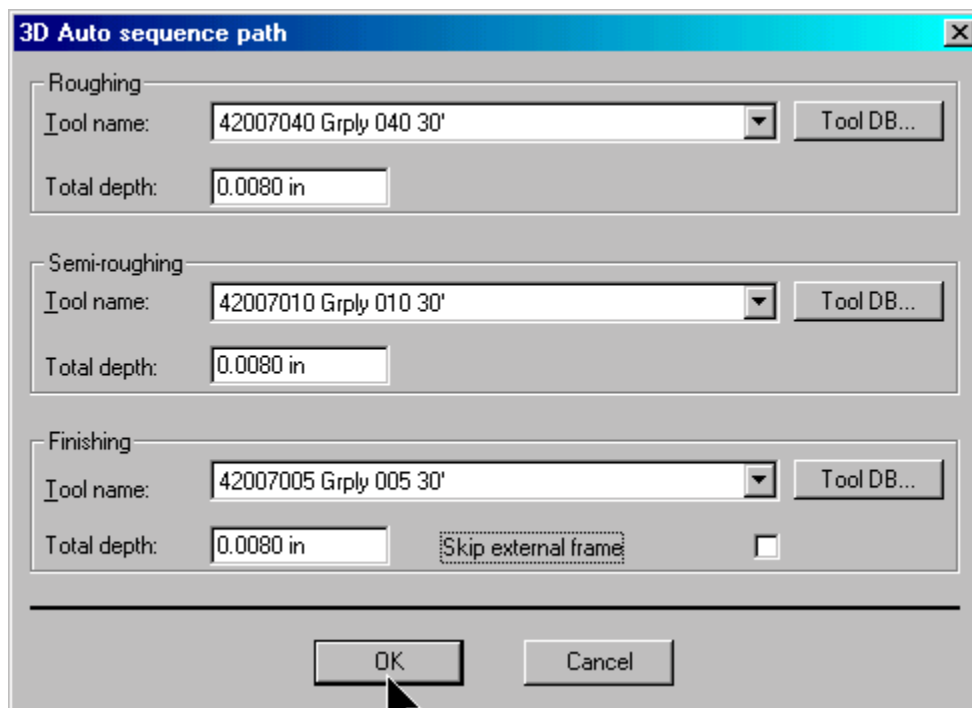
Contouring



Hatching



11. As you can see for the contouring to get into the corners you need to use a smaller tool. This will generate a lot of tool paths to fill the entire contour. The Automatic Sequence tool gives you the ability to use three tools to remove the same amount of material but in less time because fewer tool paths will be generated.



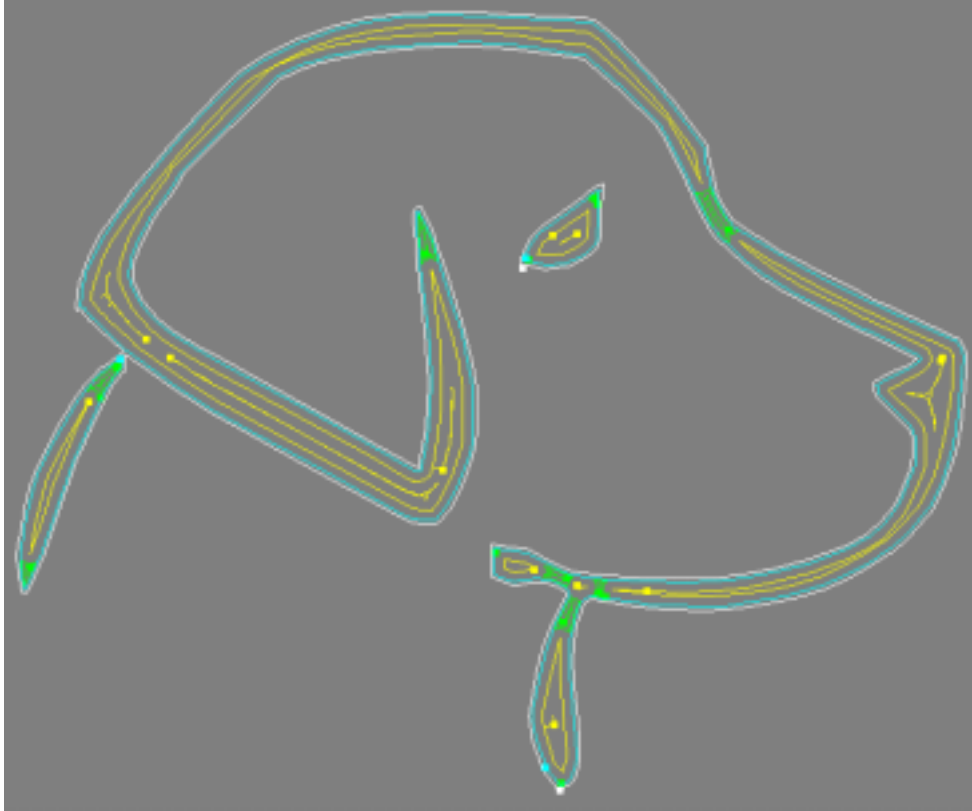
Select the first tool that will take the most material and set the depth. This will be the Roughing pass.

Select the Semi-roughing tool that will take what material is left over from the Roughing pass. Enter the depth.

For the Finish pass we will choose a small tool to give us the nice sharp edges and get into the corners. Enter the depth.

Click the OK button.

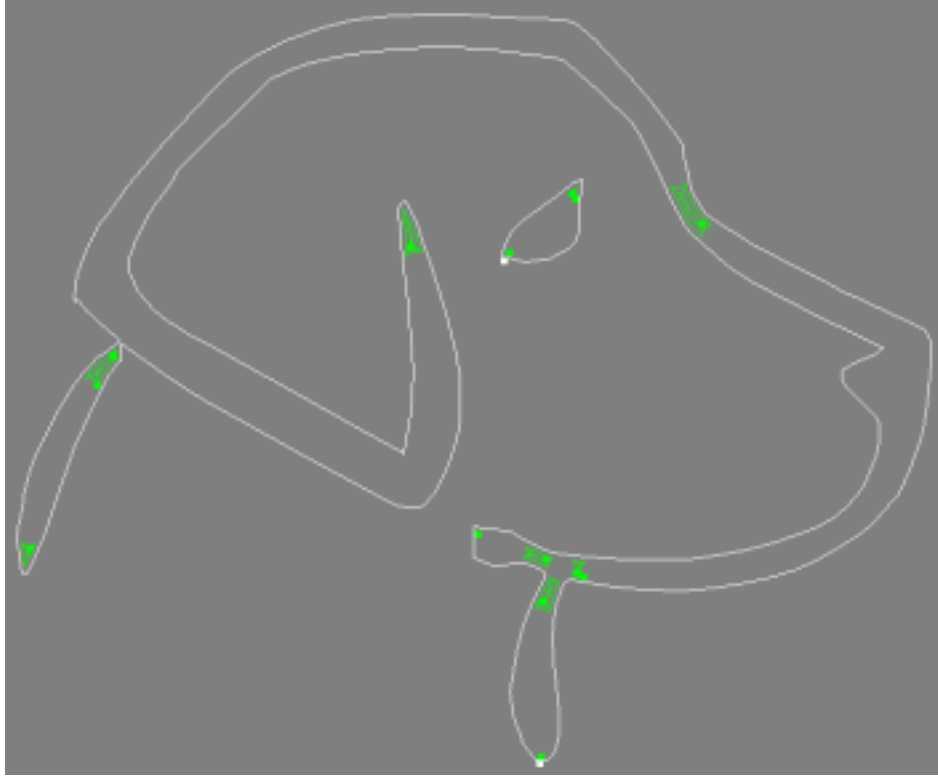
All Three Paths



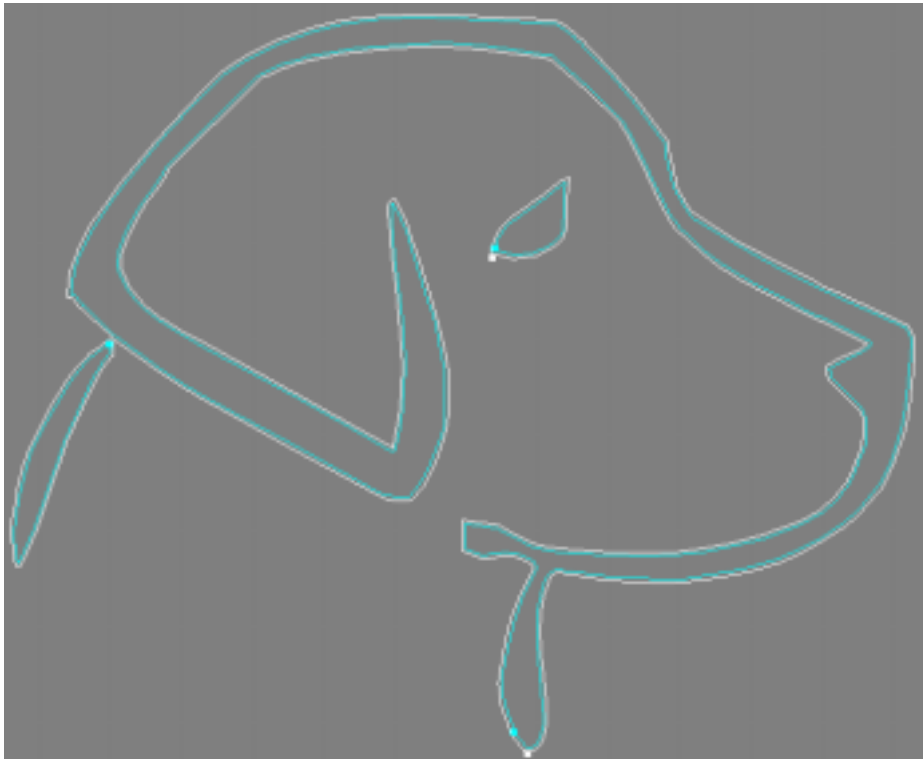
Roughing Path



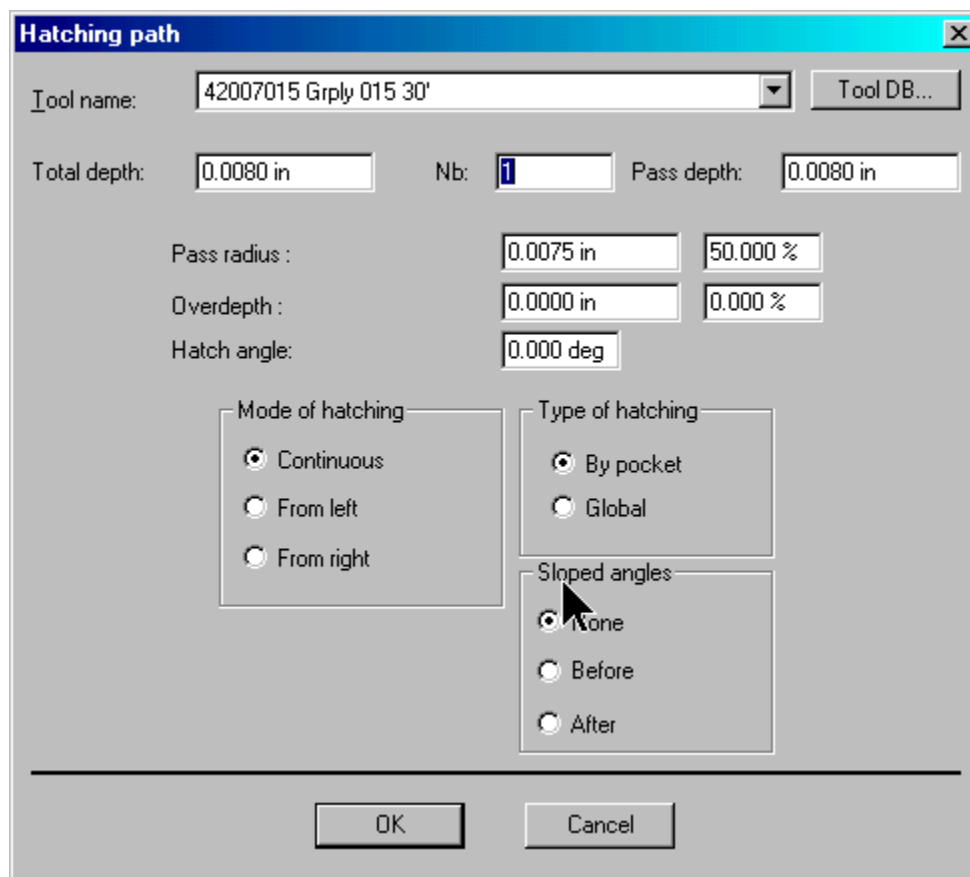
Semi-roughing Path



Finish Path



12. The Hatching Tool gives you even more control than even the Filling 2D tool.



Select the Tool and the Depth and Pass Radius as before.

Over-depth is the leaving of material for a finish pass if you want. It gives the choice to leave a little bit of material for a smaller tool to do a finish pass.

Enter the Hatch angle in degrees.

Mode of Hatching- The machine will engrave back and forth on the angle that you entered and Only pick up if you when it moves to a new pocket.

Type of Hatching- by Pocket means that it will engrave in pockets. This is the most efficient Way to hatch.

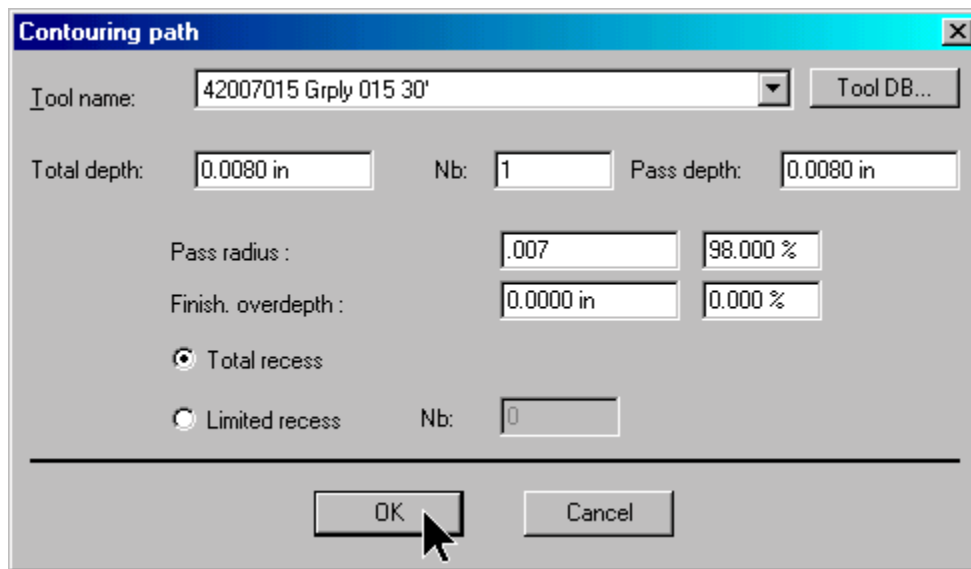
Global means that it will start at one end of your contour and not change Throughout the hatching process. Usually it starts at the top and hatches down the contour, which means it pick up from one side to the other.

Sloped Angles- None means there be no finish pass going around your hatching to give It nice smooth edges.

Before means the finish pass will be done before the hatching is Accomplished.

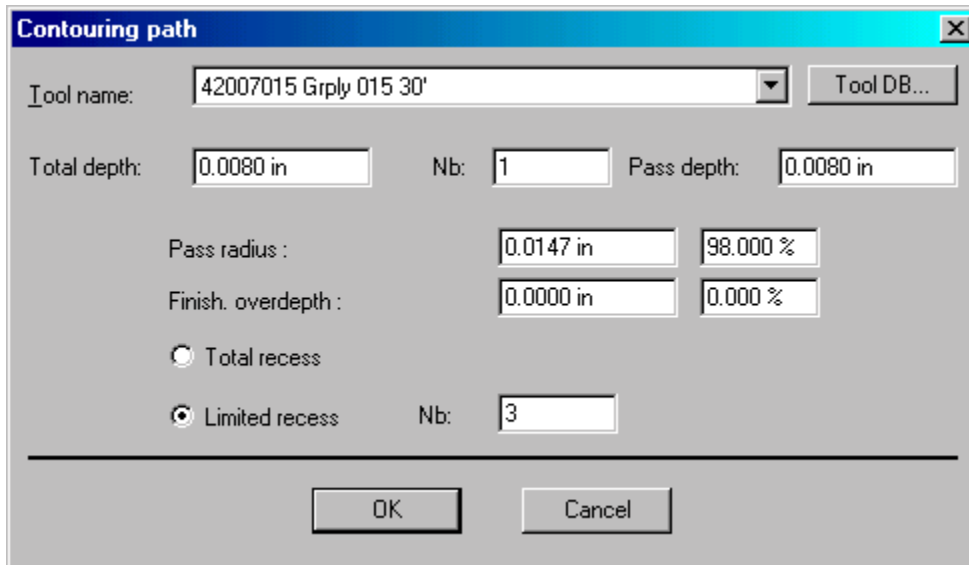
After means the finish pass is done last.

13. The Contouring Tool gives you more control of the contouring tool path. It gives you the ability to restrict the number of tool paths that you generate and to have overdepth.



Total Recess means that it will engrave all the material out from between your contour lines.





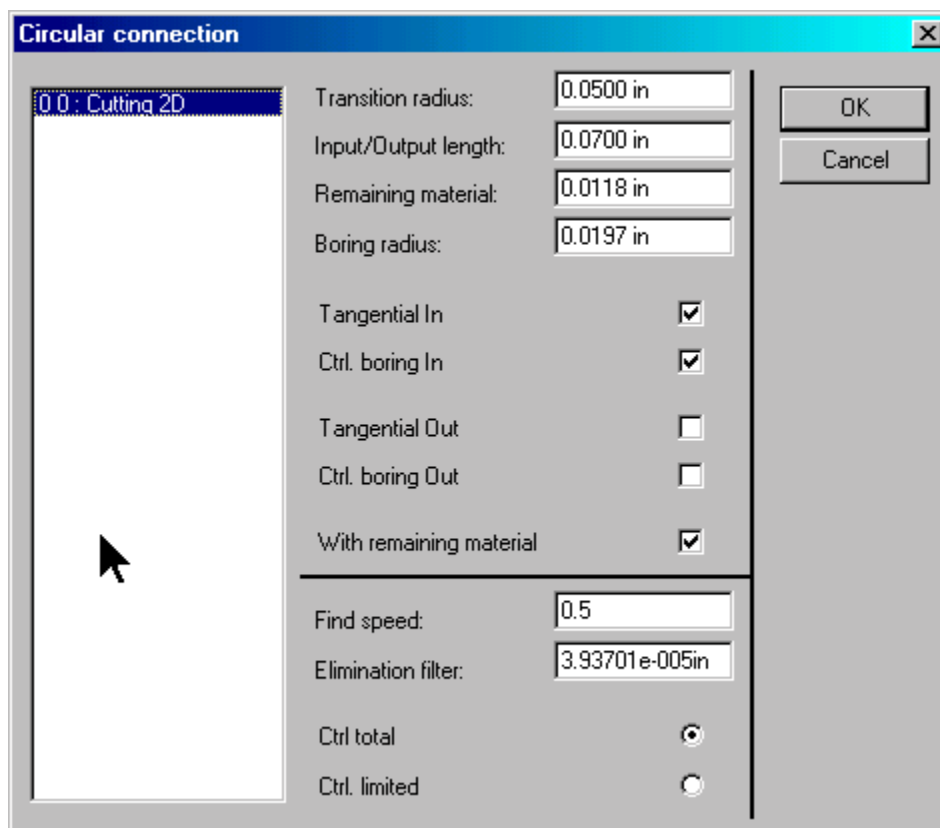
Limited Recess will restrict the amount of tool paths. Meaning that we will only have three tool paths on each side of the contour where it has space to do so.



14. The Tangential Input/Output gives the ability to control the Tangential tool path once an engraving tool path has been generated. Create a 2D Cutting path and you will see that the Tangential Input/Output Tool will highlight.



Click on it and the Tangential Input/Output page will pop up.



Transition Radius- Controls the radius of the Tangent.

Input/Output Length- Controls how far out from the contour it will drill down to depth.

Remaining Material- Determines how far it will overlap the stop point of the contour.

This screen also controls if you want control in or out and even both. It also determines if you want the Tangents to overlap by selecting remaining material.

The Find Speed is how fast it moves form the Tangent start point to the contour.



15. The Information Tool gives you information about the tool path that you generated, such as:

Path name Directory and name under which the "tool path" file is saved. This file has the Name of the GravoStyle'98 document, followed by the extension .Pxx (xx = number Corresponding to the order in which the path was created).

Type of path- (Laser, Plotting...).

Tool name-Selected in the tool database.

Machining length-Total distance covered by the tool in the material.

Fast machining length-Length of the quick movements carried out by the tool (offsets, drop...).

Time of machining-Estimated milling time depending on the characteristics entered in the tool database. So that this value is as exact as possible, you should set the different speed parameters for the tool correctly.

XMin YMin Zmin-Minimum co-ordinates of the Tool path.

XMax YMax Zmax-Maximum co-ordinates of the Tool path.

Depth-Engraving depth defined during the tool path calculation.

