HEXAGON
MANUFACTURING INTELLIGENCE

# ALPHACAM 2020.1
# HOUSING PLATE AUTO Z

# Copyright



.

# Contents

# Table of Images

# Conventions used in this manual

To enable you to use the information in this guide effectively, you need to understand the conventions used in the guide to represent differing types of information.

- Buttons on the screen are represented as the button text in square brackets.
  For example: Click on **[OK]**.
- Keys on the keyboard are represented as bold lettering in between **< >** characters.
  For example: Press **<Enter>**.
- Ribbon Tab options are represented as a path with the Ribbon Tab in **UPPER** case with sub menus Capitalised and separated with an arrow
  For example: Select **FILE > Open.**
- Field names are represented as bold text. And the value to be entered will be represented by Bold Text.
  For example: Enter the value **50** in the **Offset** field. Or
  When prompted for the X & Y values type **100,50 <Enter>**

 Denotes a **<LClick>** or Primary Mouse Button Click.
 Denotes a **<RClick>** or Secondary Mouse Button Click.

 This is a note. It contains useful or additional information.

 This is a reference. It directs you to another part of the user guide.

 This is a thought box. It is generally used in exercises and contains a question for you to consider.

 This is a highlighted note to emphasise information

 This is a warning; it contains information that you must not ignore.

 This is a tip. It is generally used in exercises and offers further advice.

1. This is the first line of a number list item
2. This is the second item of the numbered instructions, which you must
3. Follow in sequence.

- This is a list
- of items, in which
- The order is not important.

# Recommended Operating Systems and Hardware for ALPHACAM

## Supported Operating Systems

- **Operating System**
- **64bit** operating systems of the following list are supported,
  - Windows 7 (Professional, Enterprise or Ultimate) SP1 required,
  - Windows 8.1 Professional and Enterprise,
  - Windows 10 Professional and Enterprise.
- ALPHACAM will install and run on the 'Home' editions of the above operating systems. However, this is not recommended, and we cannot guarantee to fix any ALPHACAM issues specifically related to these operating systems.
- Nvidia or ATI Open GL Graphics Card with 1Gb dedicated memory

We recommend you keep up to date with the with the latest Software Updates for the supported operating systems and drivers for your hardware base.

Any Windows Operating system (OS) prior to and including Vista, is not a supported operating system.

## ALPHACAM Minimum Specifications

The latest minimum specification can be found at http://www.alphacam.com/systemrequirements

This minimum specification is to run any **Alphacam Essential** module, you will need to considerably increase the specification if you are working with solid models and producing the NC code for 3D machining and 3, 4, or 5 axis simultaneous machining strategies.

Your minimum specifications should be the fastest processor with the most memory and the highest specification video card that your budget will allow.

If using Autodesk Inventor Files, please check the current Inventor View requirements at autodesk inventor view

# Hexagon Customer Portal

At Hexagon, we strive to provide you and your business with first class technical support and services. The Customer Support Portal allows you the tools you need to receive the best from your software. In addition to generating new and updating existing support cases, the portal allows you:

- Unlimited user logins for your company.
- Access to all your licenses for easy reference.
- Get the latest releases and software update at the touch of a button.
- View what is available on your support and maintenance schedule.
- Reference to the Customer Support Charter at any time.
- View the status of your Technical Support cases.
- View all purchased Professional Services like Training and Consultancy.

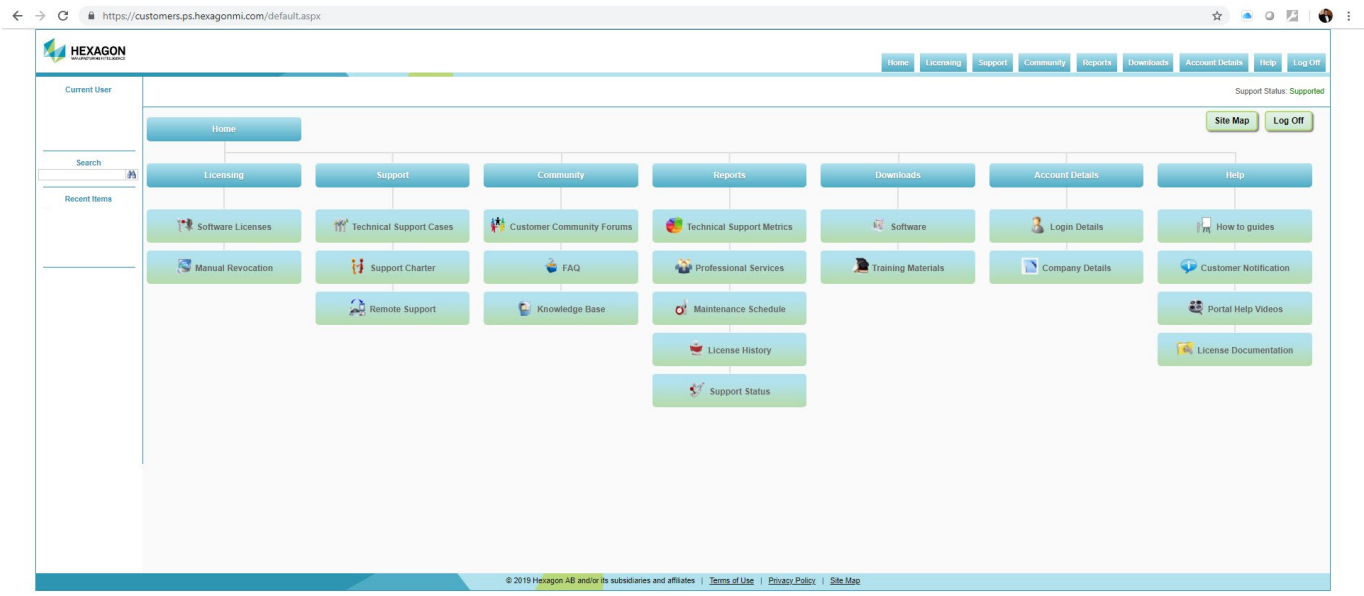



**Figure 1 - Customer Support Portal**

For the Hexagon Customer Portal visit customers.ps.hexagonmi.com

# ALPHACAM esupport

Another location to gain valuable information about using the software or asking other experienced users for assistance are the esupport forums.

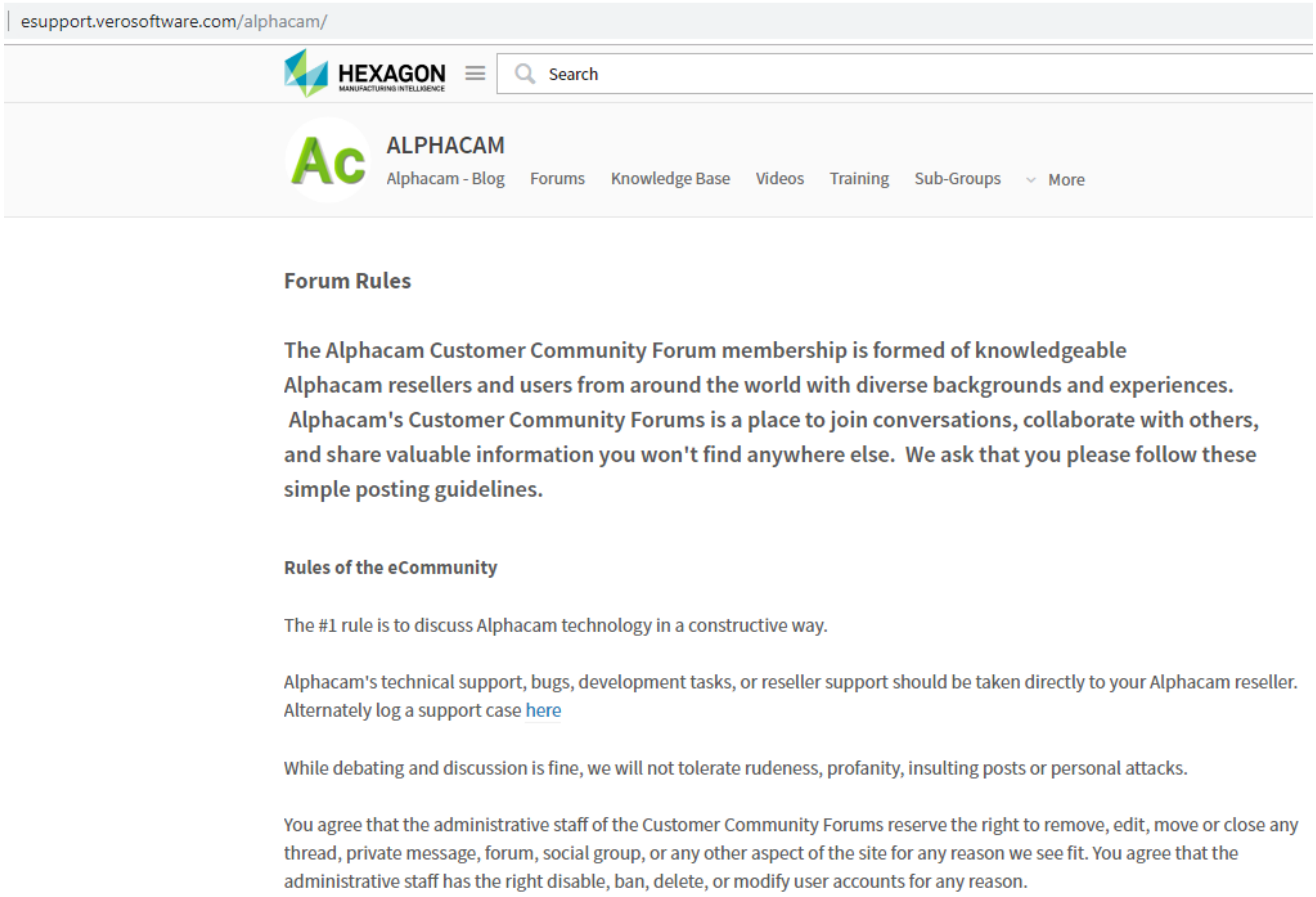



**Figure 2 - esupport page**

Asking a question of the community, using the knowledge base or other available information links could save you time if you have a problem that someone else may already have supplied a solution for.

# Housing Plate Machining 2D Flat Geometry with Z Level Attributes

This document is to be used in conjunction with the Standard 2D Machining document and the tutored example from the classroom training.

## Geometry Creation

Before creating the geometry, it is necessary to produce a process plan as this will determine what geometry is required to be created to control the machining.

### Process Plan

| OP. NO. | DESCRIPTION | TOOLNO. | TOOL DESCRIPTION |
|---|---|---|---|
| 1 | Pocket Centre Recess Profile 6 and central flanges Profiles 7 & 8 leave 0.5mm stock. | 1 | 16mm Flat |
| | Spiral Pocket 0.5mm stock Ø20 & Ø30 Holes (Profiles 9,10) | 1 | 16mm Flat |
| | Rough & Finish Profile 2, 3mm deep 4 XY cuts 34mm stock. | 1 | 16mm Flat |
| 2 | Residual Pocket Profiles 6,7,8 leave 0.5mm stock and Finish Profile to size. | 2 | 8mm Flat |
| | Finish Profiles 9 & 10 using Helical option. | 2 | 8mm Flat |
| 3 | Spot Chamfer Ø8 Holes 1, at Z 0 and produce the 1mm x 30° chamfer. | 3 | 15mm x 120 Spot Drill |
| | Spot Chamfer Ø6 Holes 3, at Z -3 and produce the 1mm x 30° chamfer. | 3 | 15mm x 120 Spot Drill |
| | Spot Chamfer Ø10 Circles 5, at Z-3. | 3 | 15mm x 120 Spot Drill |
| 4 | Drill Ø8 mm Holes 1, Through. | 4 | 8mm Twist Drill |
| 5 | Drill Ø6 mm Holes 4, 10mm Deep. | 5 | 6mm Twist Drill |
| | Drill Ø6 mm Holes 3, 12mm Deep. | 5 | 6mm Twist Drill |
| 6 | Machine 2mm Radii on top edges. (Profiles 2, 7, 8, 9, 10) | 6 | 2mm corner rounding |
| | Machine 2mm Radii on Profile 6. | 6 | 2mm corner rounding |
| 7 | Chamfer 1x45° on Outer Edge Profile 1. | 7 | Ø20mm x 45° |
| 8 | Engrave Part No.27854. | 7 | Engrave – 30° Point end |
| 9 | Rough Profile 1 Outer Edge 16mm deep leave 0.5mm stock. | 1 | 16 mm Flat |
| 10 | Residual Rough Profile 1 leave 0.5mm stock. | 2 | 8mm Flat |
| | Finish Profile 1 the outer edge Through. | 2 | 8mm Flat |

The Geometry is drawn with square corners and the Radius and Chamfer are generated by the tool.

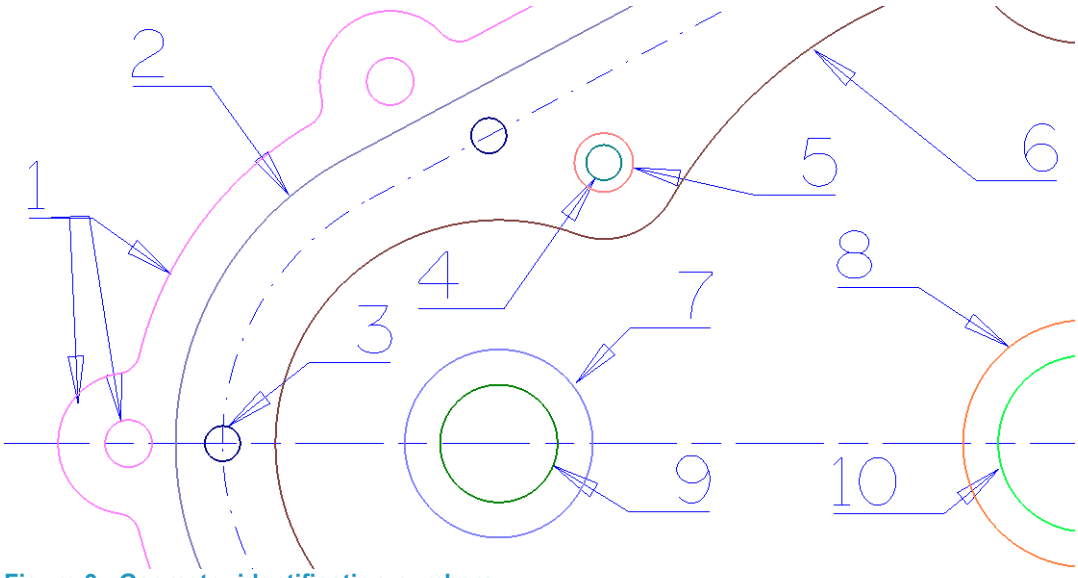



Figure 3 - Geometry identification numbers

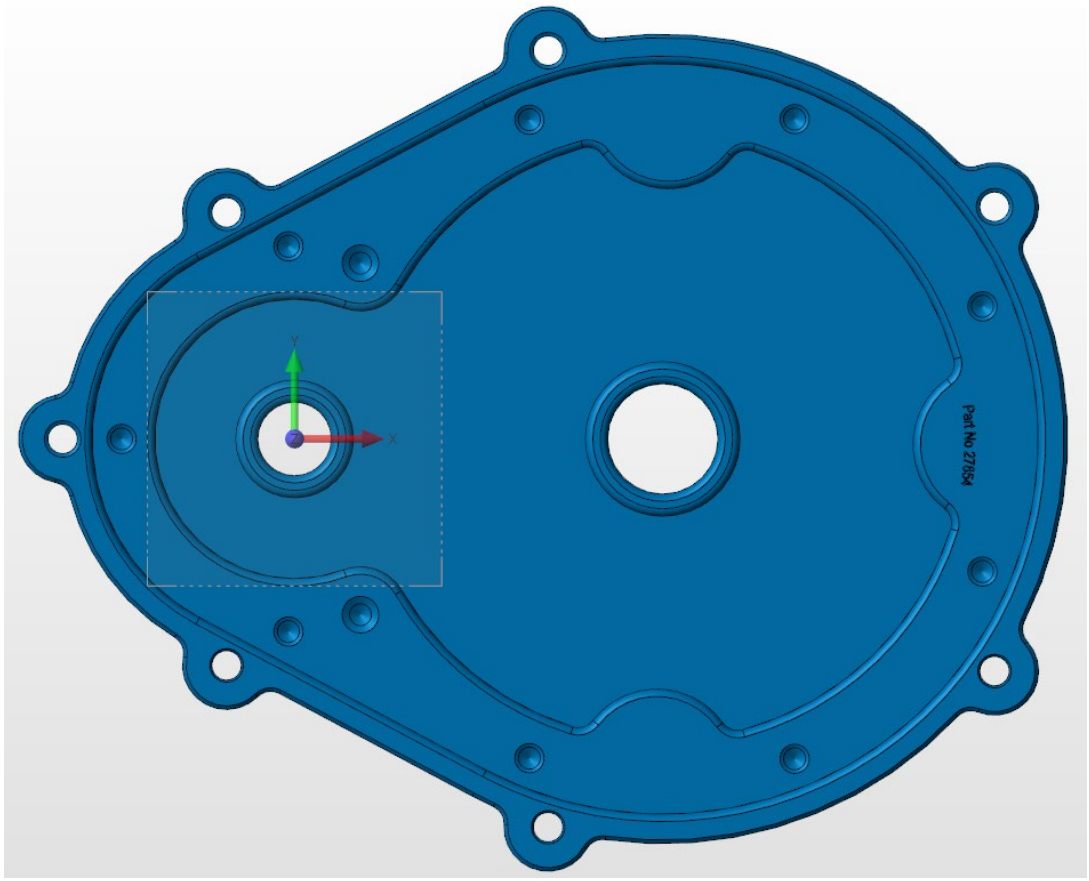



Figure 4 - Plan view of 3D Housing Plate model

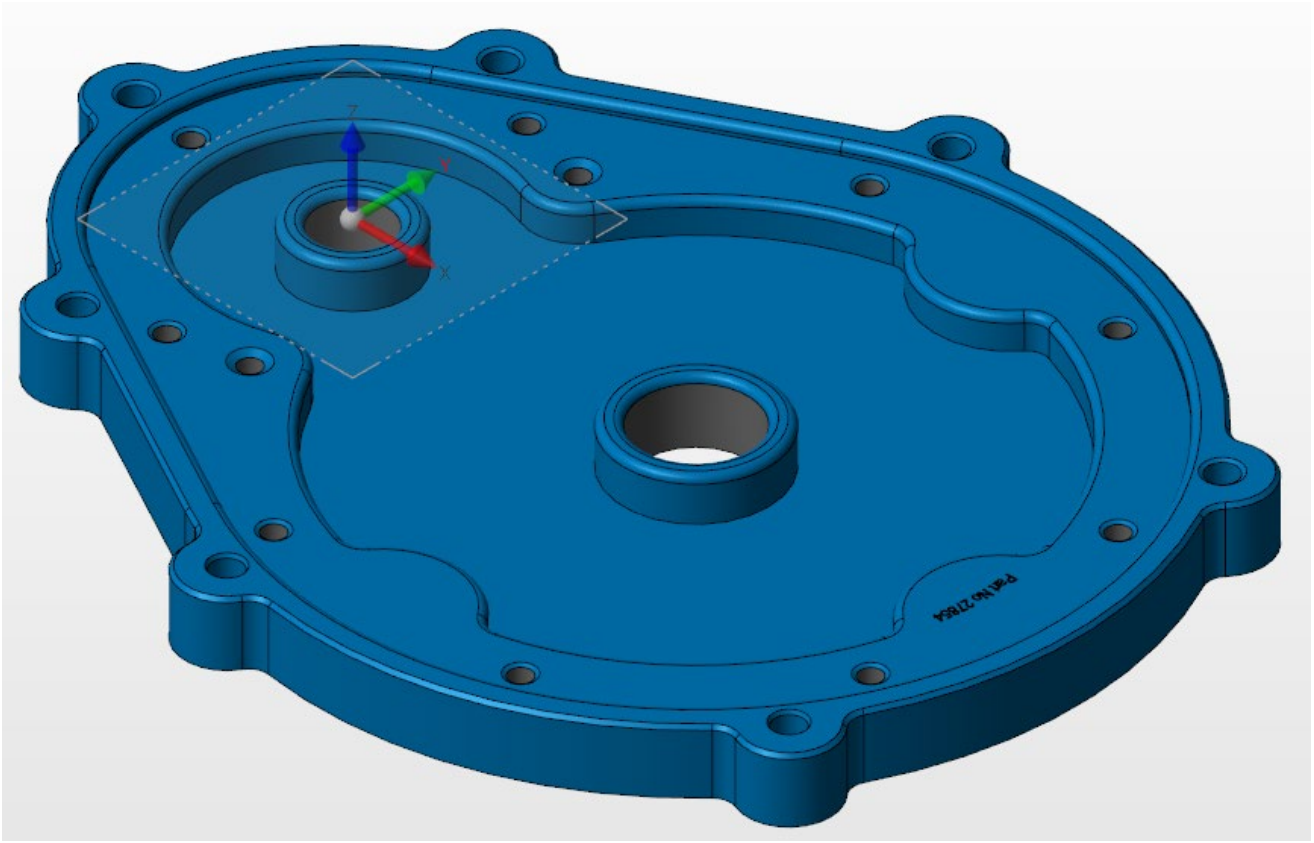


**Figure 5 - Isometric view of Housing Plate 3D model**

# Material Creation

To gain a better idea of your machined part within ALPHACAM, you can employ the inbuilt simulator to run your toolpaths in and, using a material form, see if the part has been machined correctly.

## AutoSet Material

Create a material around the part using **3D > Auto Set Material.** 

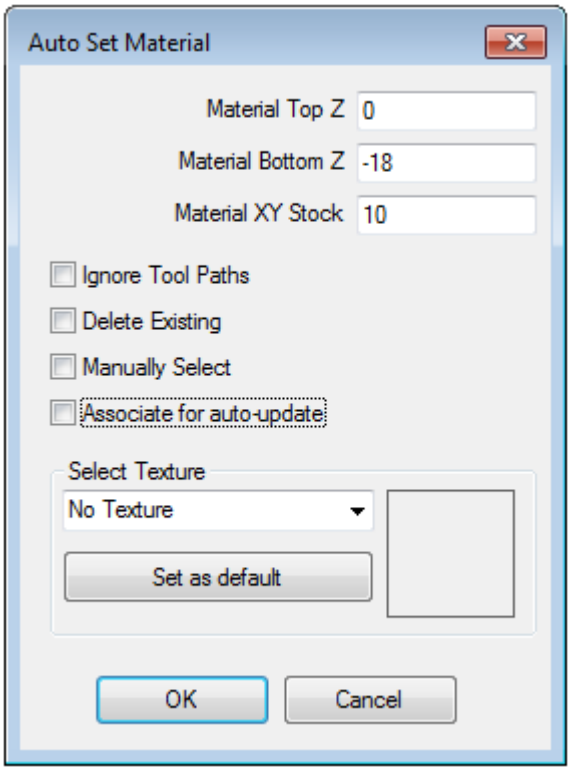



**Figure 6 - Auto Set Material dialogue**

Make the values in the dialogue as follows,

Material Top = 0,
Material Bottom = -18
Material XY Stock = 10

If the option Auto Set Material is not available, use **ADD-INS/Macros > Add ins** 
then look for **Create Default Material** on the left hand window, tick this option to add it to the 3D menu.

> Auto Set Material uses a fixed allowance around the entire part to be machined which is a suitable method if your part is machined from a plate or panel.
> If you have material that has different allowances on the sides compared to the ends or is an irregular shape then you would need to draw the corresponding shape first and use 3D > Set Materials instead.

## Set Material

Another method is to use **3D > Set Materials** 
With this method, a shape must be drawn around the item to be machined that represents the physical shape of the material to use.

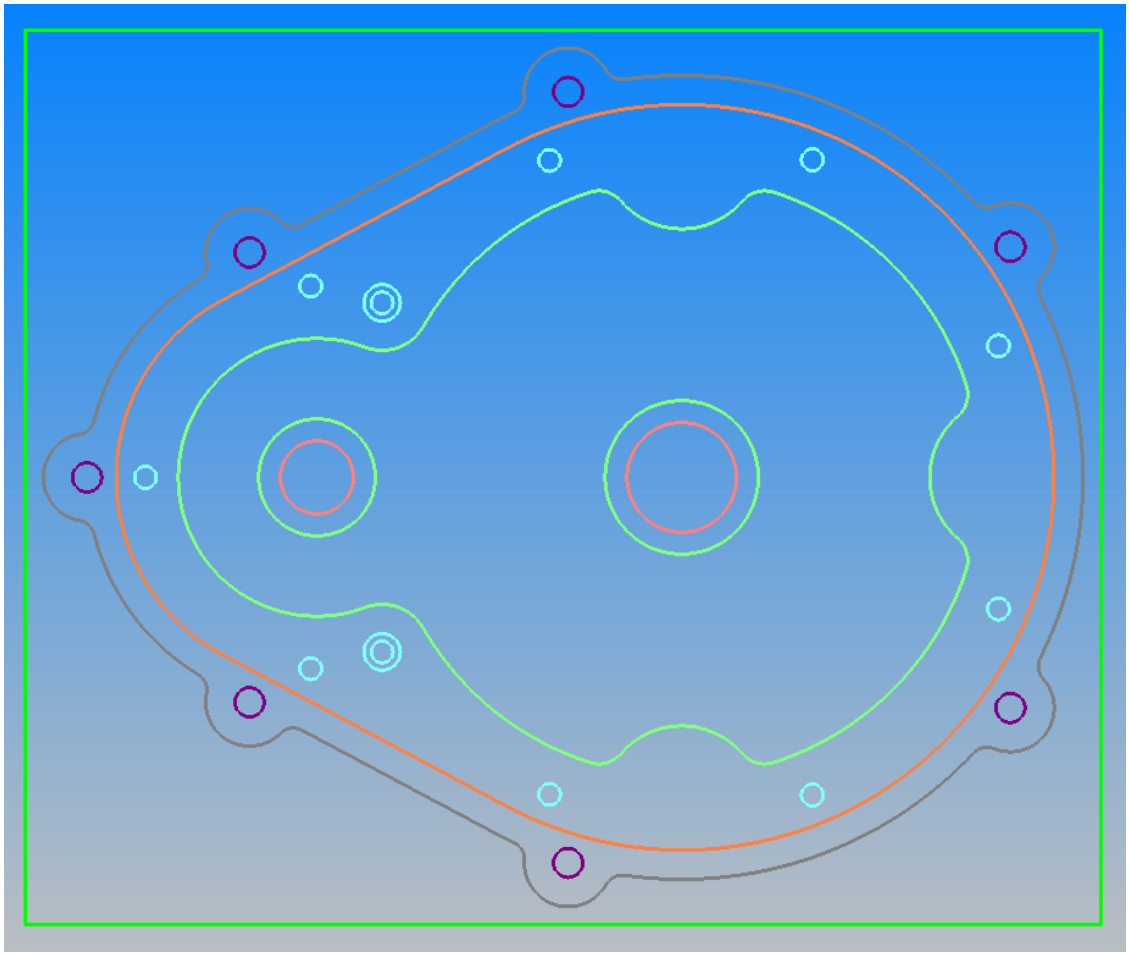


**Figure 7 - Material outline in place prior to selection**

Once the command is selected, you are requested to **<LClick>** on the required shape representing the material profile.

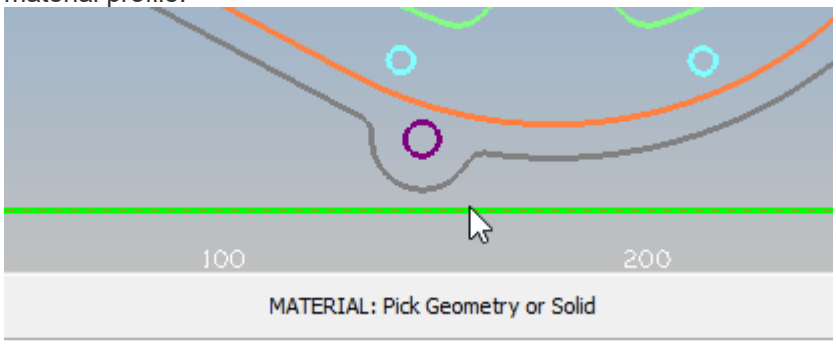



**Figure 8 - Select the required profile**

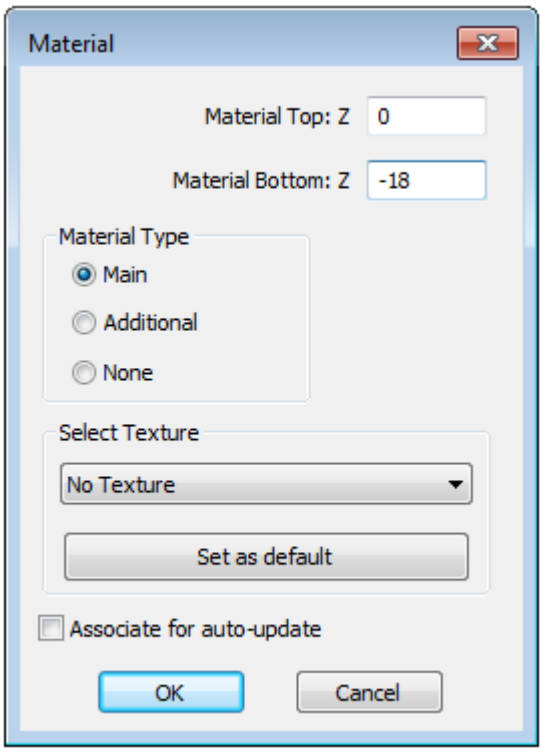



**Figure 9 - Set Material dialogue**

Auto Set Material and Set Material both perform the same task.

The Auto option creates a uniform allowance around the part, Set Material can be any form. It is the only method you can use to create circular material.

## Move to machining position

Before applying any machining to your part, it is advisable that you have the part located correct for machining, this saves problems later should you decided to reposition the part after machining has been applied where, in some unique situations, toolpaths could become un-editable.

The datum for the Housing Plate was chosen to make the drawing process easy; the datum position may not be suitable for some styles of machine. We need to move the part from its current location.
Make sure that all items to move with the part are visible in the **Layers** Project Manager page, if the items are hidden, they will not be moved.

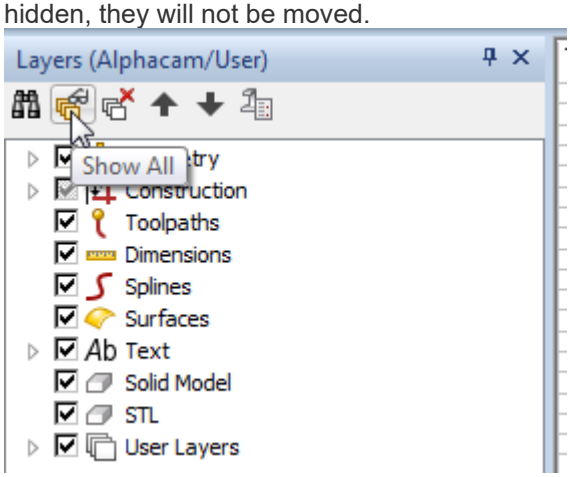



**Figure 10 - Show All option**

Using the **Show All** option is the easiest method to view any hidden items within a drawing.

Using **EDIT > Move,Copy etc > Move**
Select **[All]** from the bottom of the drawing window.
**<RClick>** to continue or **<LClick>** the **[Finish]** button.
For the base point select the lower left hand corner of the material you have just created.
**<LClick> [OK]**.

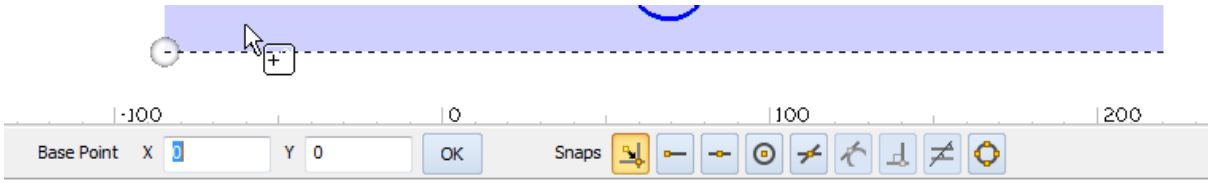



**Figure 11 - Base point location prior to <LClick>**

For the new position make the X and Y values 0.
**<LClick> [OK]**.
**<RClick>** to complete the command.

This will make the new datum position the corner of the material.

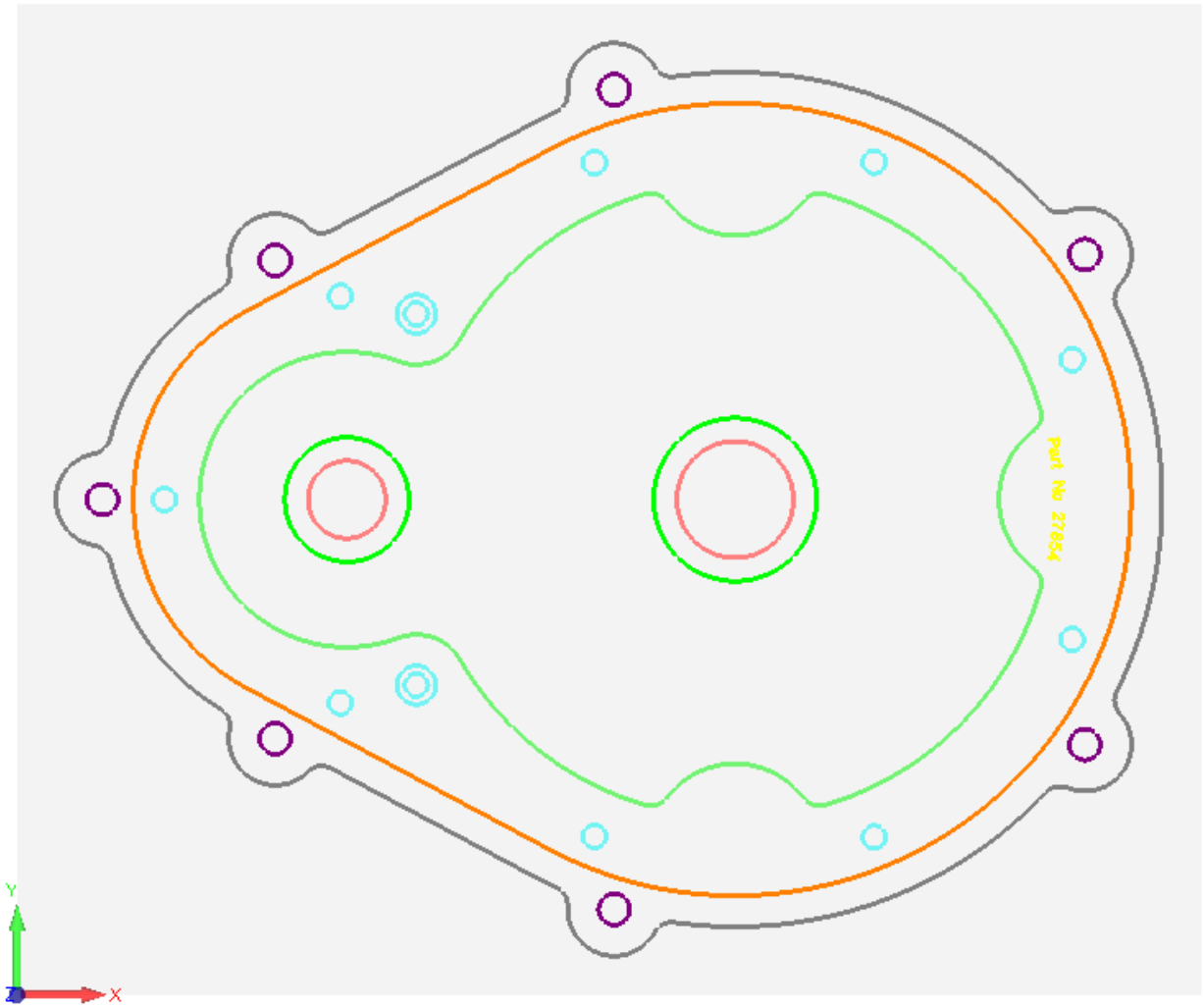



**Figure 12 - Housing Plate correctly located for machining**

# Prepare the Geometry for Machining

Prepare each of the geometric contours for the machining operation which is to be performed on it. This is done by

- Setting the cutting direction of the tool path.
- The side on which the cutter is to be positioned.
- The position at which the cutting will start on the contour.

You can see from the process plan the indicated tool side and direction.

## Tool Directions

Select **MACHINE > Tool Directions**  . This will display a non-modal Dialogue box.

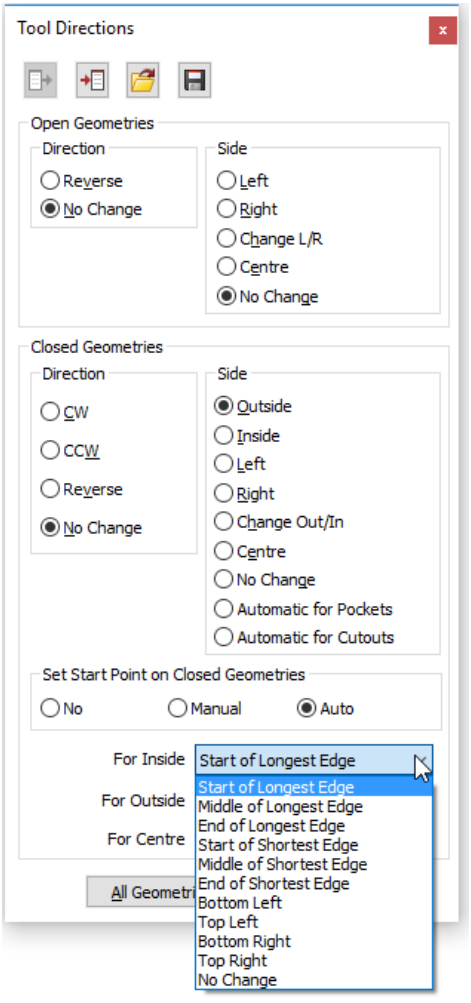



**Figure 13 - Tool Directions Dialogue**

A non-modal Dialogue is where the Dialogue remains open whist you select the geometries that are to have the settings applied to.

You can change the settings and select geometries until you have completed the task when you can select [Close] on the Dialogue.

The Dialogue box allows you to define the relationship between the geometry and the cutting tool.
There are three different sections.

- The top section affects **Open Geometries**.
- An Open Geometry is one whose start and end points are in different locations, for example, a straight line.
- The centre section affects **Closed Geometries**.
- A Closed Geometry is one whose start and end points are in the same location, for example, a circle.
- The bottom section allows for the alteration of the Start Point from where the machining will commence.

The settings for either of these sections are mutually exclusive, setting options on Open Geometries will have no effect on Closed Geometries and vice versa.
You can set the start point automatically for geometries set to the inside outside and centre, the start point's position can be set from the dropdown list.

Select **VIEW > Display Options > Ghost Tool** or press **<Ctrl> +<G>** to display the current ghost tool positions.

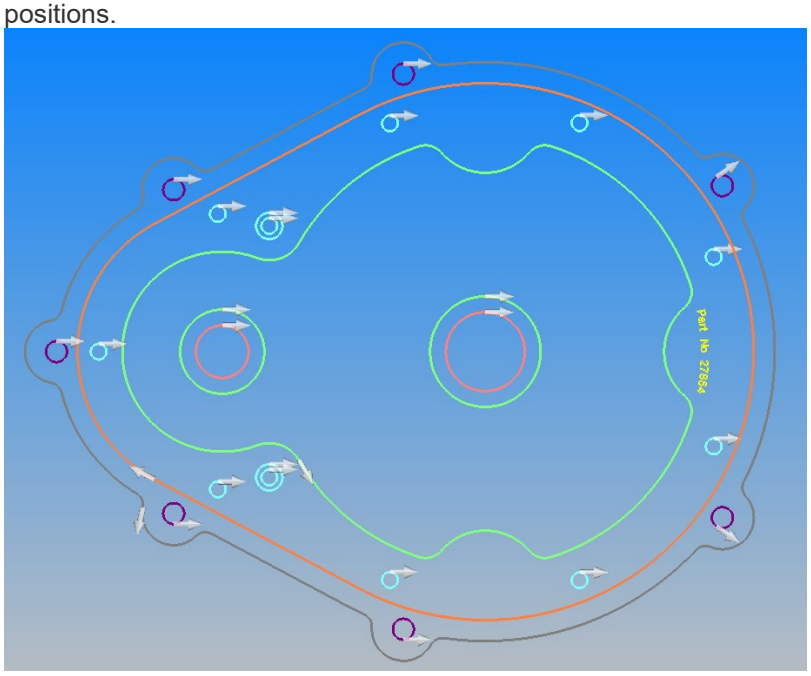



**Figure 14 - Ghost Tools as drawn**

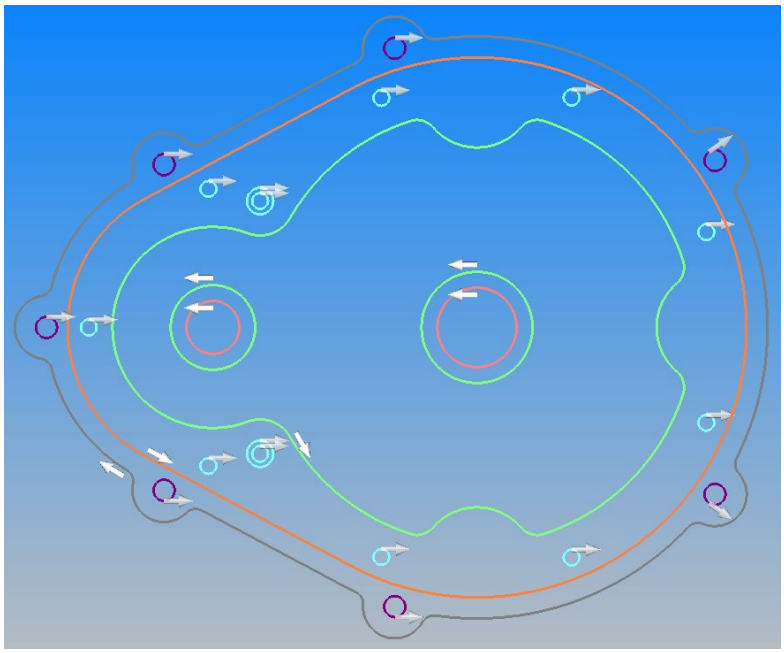


**Figure 15 - Ghost Tools set correctly for machining**

Set the tool directions as shown in the picture above.
Make the options in the Dialogue ⊙ **CW** and ⊙ **Automatic for Pockets**, and then select All Geometries.
Alter the option to ⊙ **Change Out/In** and select the Outside Profile and the offset egg shape, profiles 1 & 2.
Select Close

Select **EDIT > Start Order > Start Point** 

This command allows you to select a new starting position on geometries. Using the Auto Snap function **<F2>** select the starting positions as shown in the picture above.
The reasoning behind the changes is to not have a Start point in an internal corner and to ensure that the internal pocket starts are clear of any potential fouling which may occur.

It can prove beneficial to set the Start Points prior to machining; however, they can be adjusted during or after any toolpaths have been applied. The benefit of this is that you may not be aware of potential collisions until after applying the toolpaths to the actual part.

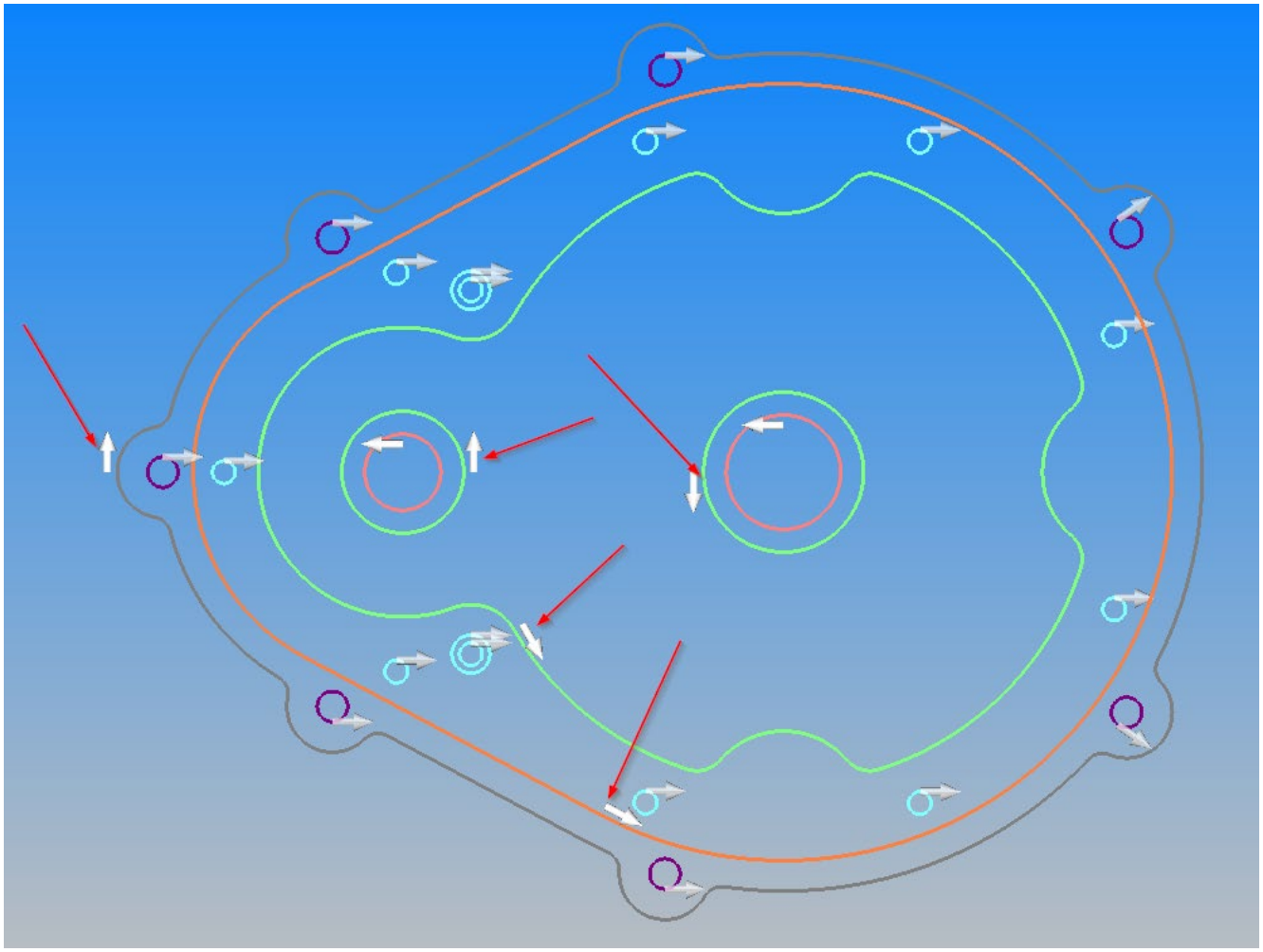


**Figure 16 - Ghost Tools and Start Points set correctly for machining.**

## Hole Geometry Sorting

One thing to take into consideration when using drawings in ALPHACAM is that the order that the geometries are created is the order in which they will also be machined. As with the Tool Directions, setting the geometry order before machining will create more desirable toolpaths, but like the Tool Directions options, you may not be aware of the order until after the toolpaths have been added, so you can alter the order during or after the toolpaths have been created.
If a hole pattern is to be machined more than once (e.g. Centre Drill, Drill, and Chamfer) it is preferable to set the geometry order. If they are only machined once you can order the tool paths.
To see the default order that the geometries were drawn the geometry connection lines need to be switched on.

Select **VIEW > Display Options > Draw Geometry Rapids**  ,

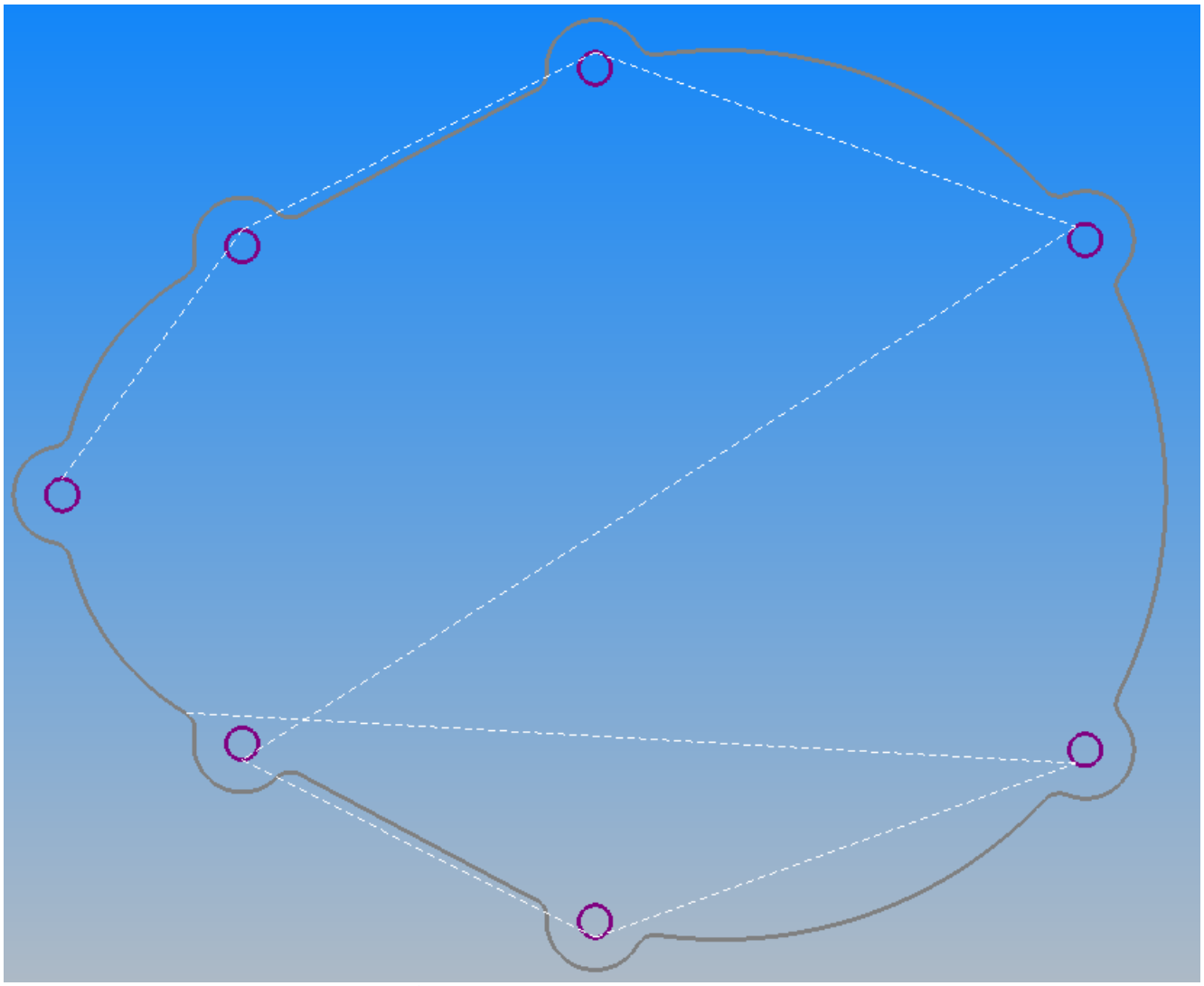


**Figure 17 - Original drawn order of the Ø8mm circles**

Select **EDIT > Start Order > Order**

This command allows you to alter the order in which the geometries and tool paths are actioned.

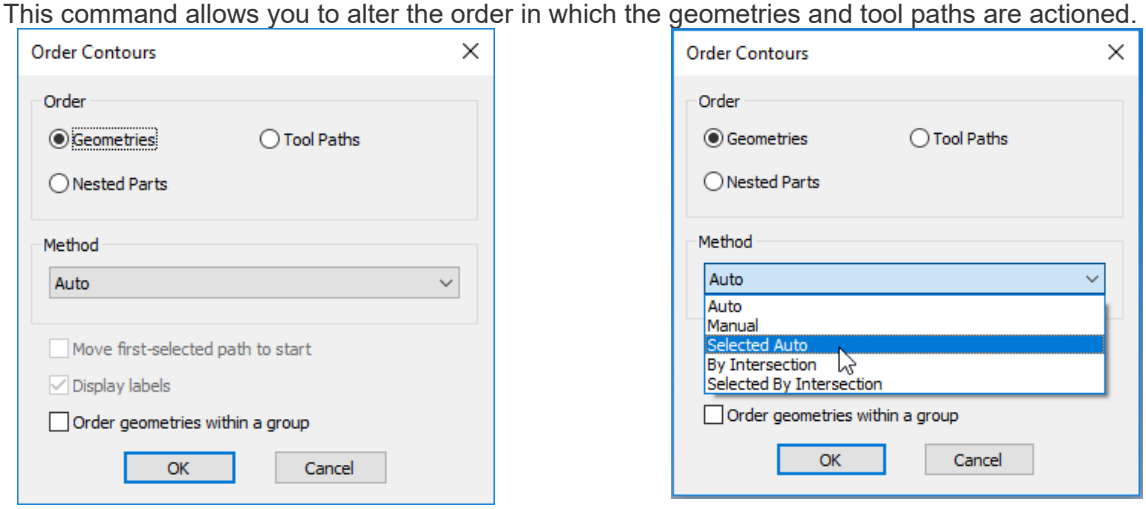



**Figure 18 - Order dialogue**

1. Set the Dialogue options; **Order = ⊙ Geometries, Method= ⊙ Selected Auto**
2. Activate the **☑ Move** first-selected path to start setting.

The system then prompts you to select the geometries that are to be put in order so that there is the shortest distance between the consecutive geometries.
**<LClick>** all the 8mm diameter circles ensuring that you choose the furthest left one first to set this as the first geometry in the ordering.
**<RClick>** or **<Esc>** to finish.

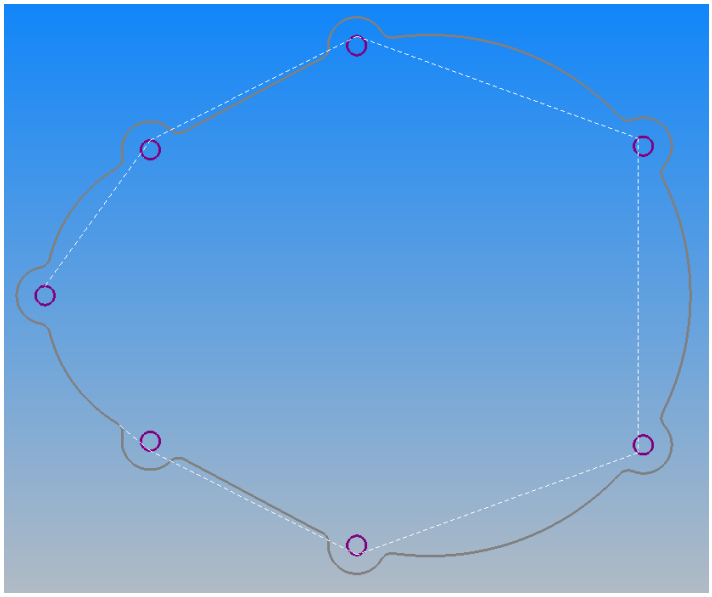


**Figure 19 - Ø8mm circles re-ordered**

It is good practice to order the geometries prior to adding tool paths. Should you edit the tool paths later having ordered them, they may revert to the original geometry order instead.

# Set Geometry Z Levels

**Select 3D > Set Geometry Z Levels**

The system prompt asks you to Select Geometries.

**<LClick>** on profiles 1,9,10 and the holes in profile 1 then **<RClick>**.

In the Dialogue Set the **Top Z = 0** and the Bottom **Z = -18** then **[Ok].**

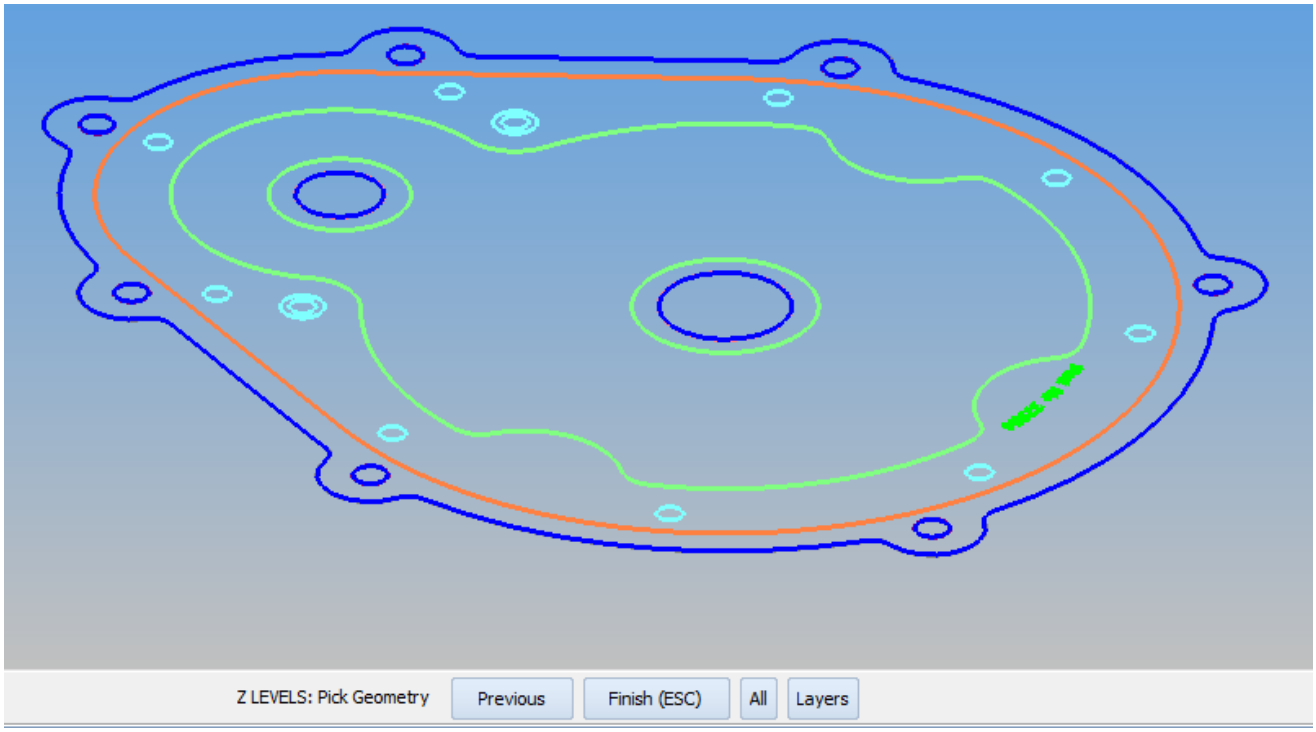



**Figure 20 - Items to select for first Z Level application**

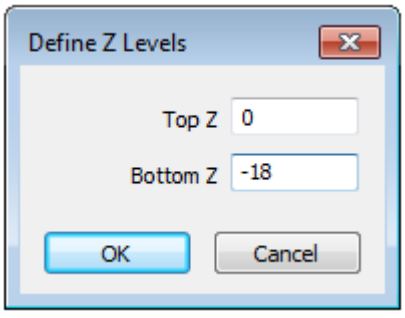



**Figure 21 - Geometry Z Level dialogue**

The system prompt asks you to Select Geometries,
**<LClick>** on profiles 7 & 8 then **<RClick>**.
In the Dialogue Set the Top **Z = 0** and the Bottom **Z = -12** then **[Ok].**

The system prompt asks you to Select Geometries,
**<LClick>** on Profile 2 then **<RClick>**.
In the Dialogue Set the Top **Z = 0** and the Bottom **Z = -3** then **[Ok].**

The system prompt asks you to Select Geometries,
**<LClick>** on Profile 6 then **<RClick>**.
In the Dialogue Set the Top **Z = -3** and the Bottom **Z = -12** then **[Ok].**

The system prompt asks you to Select Geometries,
**<LClick>** on hole detail 3 then **<RClick>**.
In the Dialogue Set the Top **Z = -3** and the Bottom **Z = -10** then **[Ok].**

The system prompt asks you to Select Geometries,
**<LClick>** on hole detail 4 then **<RClick>**.
In the Dialogue Set the Top **Z = -3** and the Bottom **Z = -12** then **[Ok].**

The system prompt asks you to Select Geometries,
**<LClick>** on hole detail 5 then **<RClick>**.
In the Dialogue Set the Top **Z = -3** and the Bottom **Z = -3** then **[Ok].**
With the top and bottom Z the same value it is allows machining using Auto Z.

**<RClick>** to finish the command.

Engraving is not an Auto Z machining operation; Z levels are not set on profile 11.

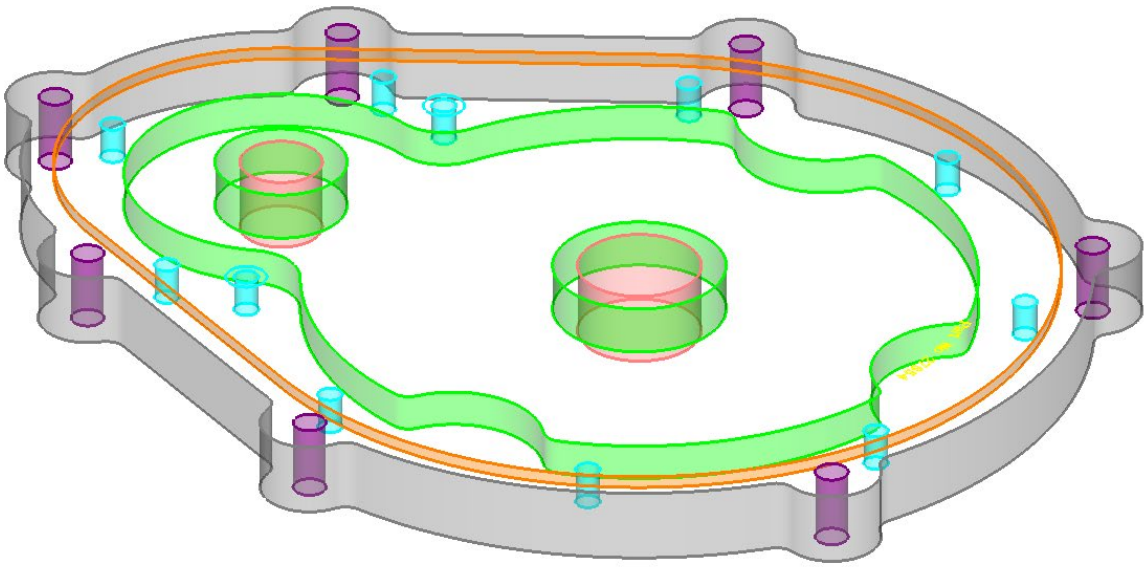


**Figure 22 - Applied Geometry Z Levels in ISO projection**

# Machining

## Select correct Post Processor

Select **HOME > Select Post** Select a relevant post processor. This is only required if the current default post processor is not the one required.

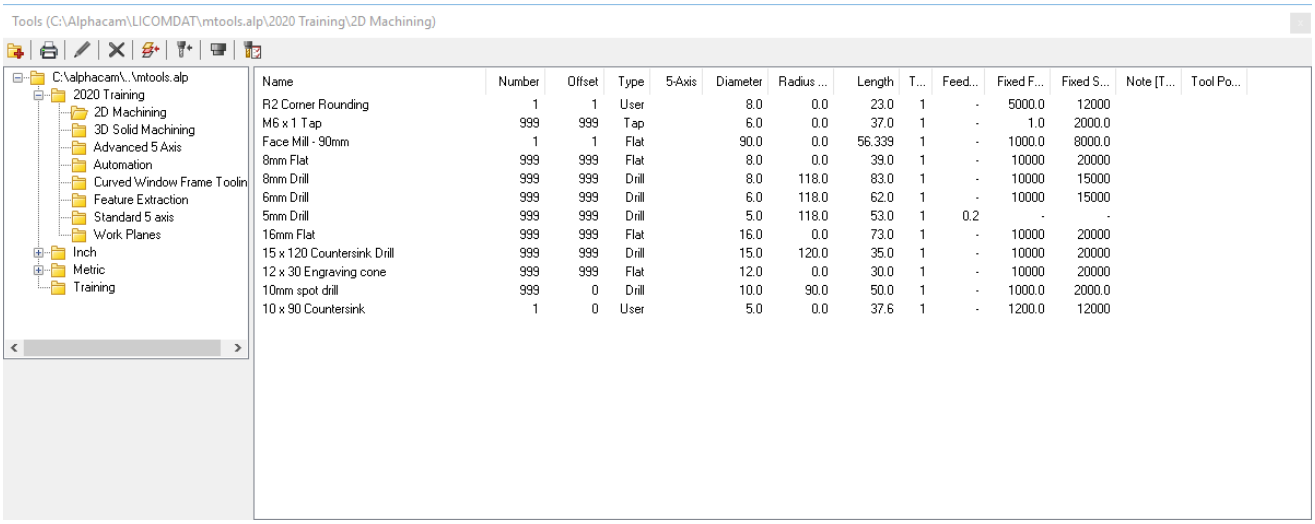
## Choose Material

Select **MACHINE > Select Material** This is only required when using tools that require feed and speed calculations.

## Select tool

Select **MACHINE > Select Tool**

All the tools used in the tutorial can be found in the Training folder on your USB stick or the specific 2D Machining folder.




| Name | Number | Offset | Type | 5-Axis | Diameter | Radius ... | Length | T... | Feed... | Fixed F... | Fixed S... | Note [T... | Tool Po... |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| R2 Corner Rounding | 1 | 1 | User | | 8.0 | 0.0 | 23.0 | 1 | - | 5000.0 | 12000 | | |
| M6 x 1 Tap | 999 | 999 | Tap | | 6.0 | 0.0 | 37.0 | 1 | - | 1.0 | 2000.0 | | |
| Face Mill - 90mm | 1 | 1 | Flat | | 90.0 | 0.0 | 56.339 | 1 | - | 1000.0 | 8000.0 | | |
| 8mm Flat | 999 | 999 | Flat | | 8.0 | 0.0 | 39.0 | 1 | - | 10000 | 20000 | | |
| 8mm Drill | 999 | 999 | Drill | | 8.0 | 118.0 | 83.0 | 1 | - | 10000 | 15000 | | |
| 6mm Drill | 999 | 999 | Drill | | 6.0 | 118.0 | 62.0 | 1 | - | 10000 | 15000 | | |
| 5mm Drill | 999 | 999 | Drill | | 5.0 | 118.0 | 53.0 | 1 | 0.2 | - | - | | |
| 16mm Flat | 999 | 999 | Flat | | 16.0 | 0.0 | 73.0 | 1 | - | 10000 | 20000 | | |
| 15 x 120 Countersink Drill | 999 | 999 | Drill | | 15.0 | 120.0 | 35.0 | 1 | - | 10000 | 20000 | | |
| 12 x 30 Engraving cone | 999 | 999 | Flat | | 12.0 | 0.0 | 30.0 | 1 | - | 10000 | 20000 | | |
| 10mm spot drill | 999 | 0 | Drill | | 10.0 | 90.0 | 50.0 | 1 | - | 1000.0 | 2000.0 | | |
| 10 x 90 Countersink | 1 | 0 | User | | 5.0 | 0.0 | 37.6 | 1 | - | 1200.0 | 12000 | | |

**Figure 23 - Tool Selection dialogue**

Select **16mm Flat** from the **Training folder**.
The tool will be displayed on the screen **<LClick>** or press **<Enter ↵>** to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

# Operation 1

## Pocket Centre Recess Profile 6 and central flanges Profiles 7 & 8 leave 0.5 Stock

Select **MACHINE > Pocketing**

### Types

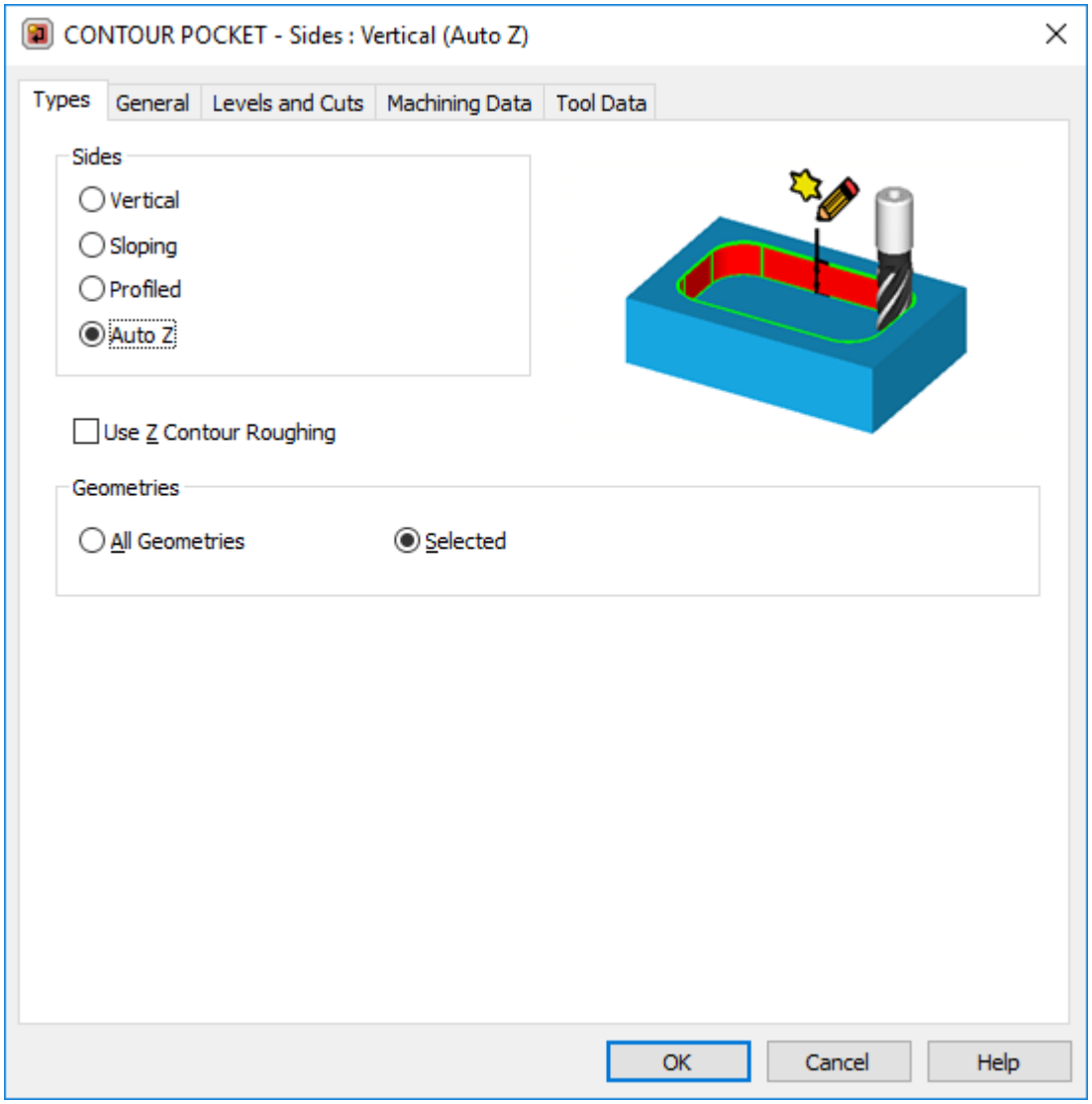



**Figure 24 - Pocketing Types tab**

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** then ⊙ **Selected.**

## General

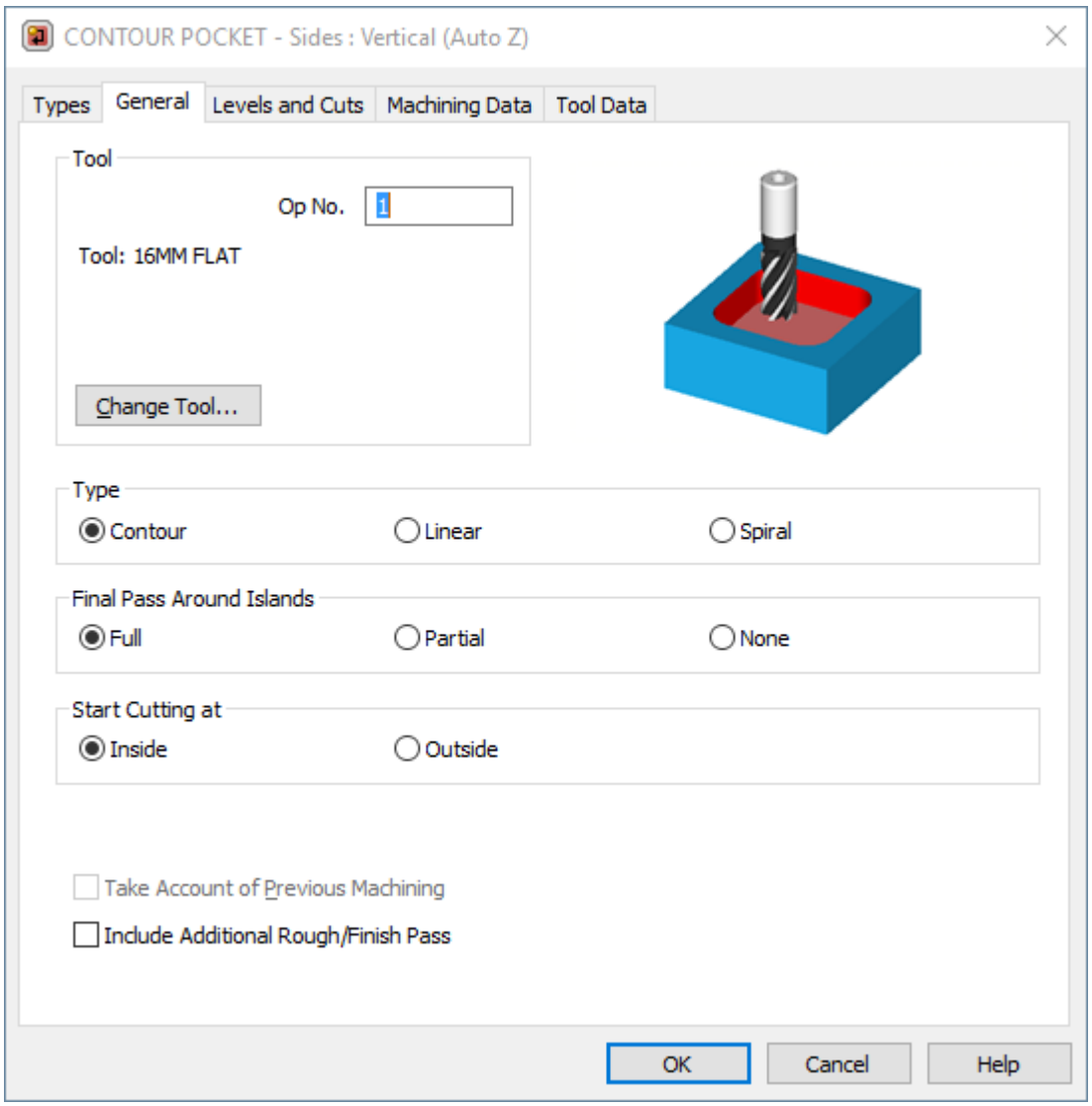



**Figure 25 - Pocketing General tab**

General sets the pocketing style.
Set the options ⊙ **Contour**.

## Levels and Cuts

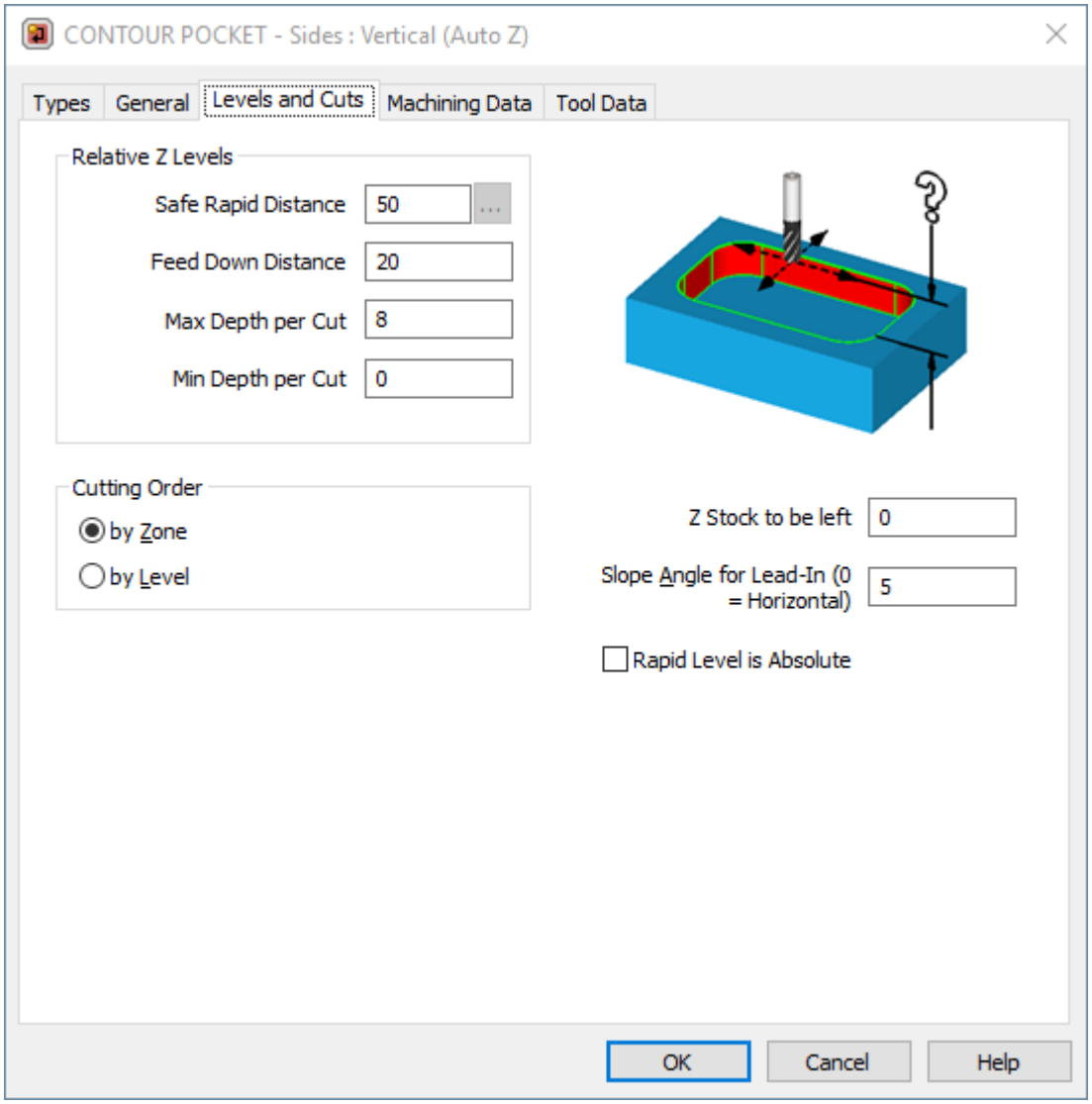



**Figure 26 - Pocketing Levels and Cuts tab**

Levels and Cuts sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid**, **Rapid Down To** as appropriate,

**Max Depth per Cut = 8,**

**Slope Angle for Lead In = 5.**

## Machining Data

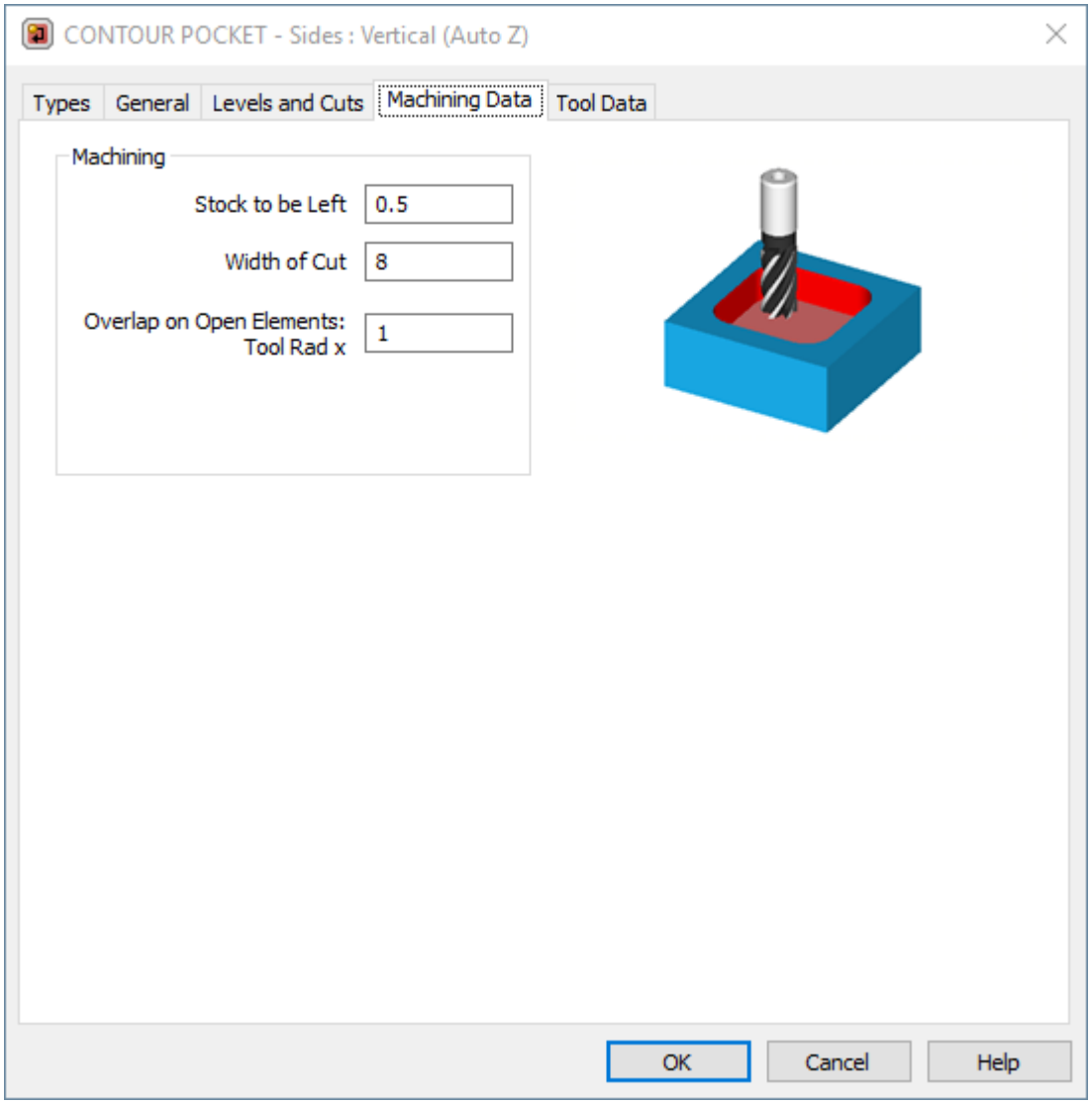



**Figure 27 - Pocketing Machining Data tab**

Machining Data sets the stock allowance and width of cut.
**XY Stock to be Left** = **0.5**.

## Tool Data

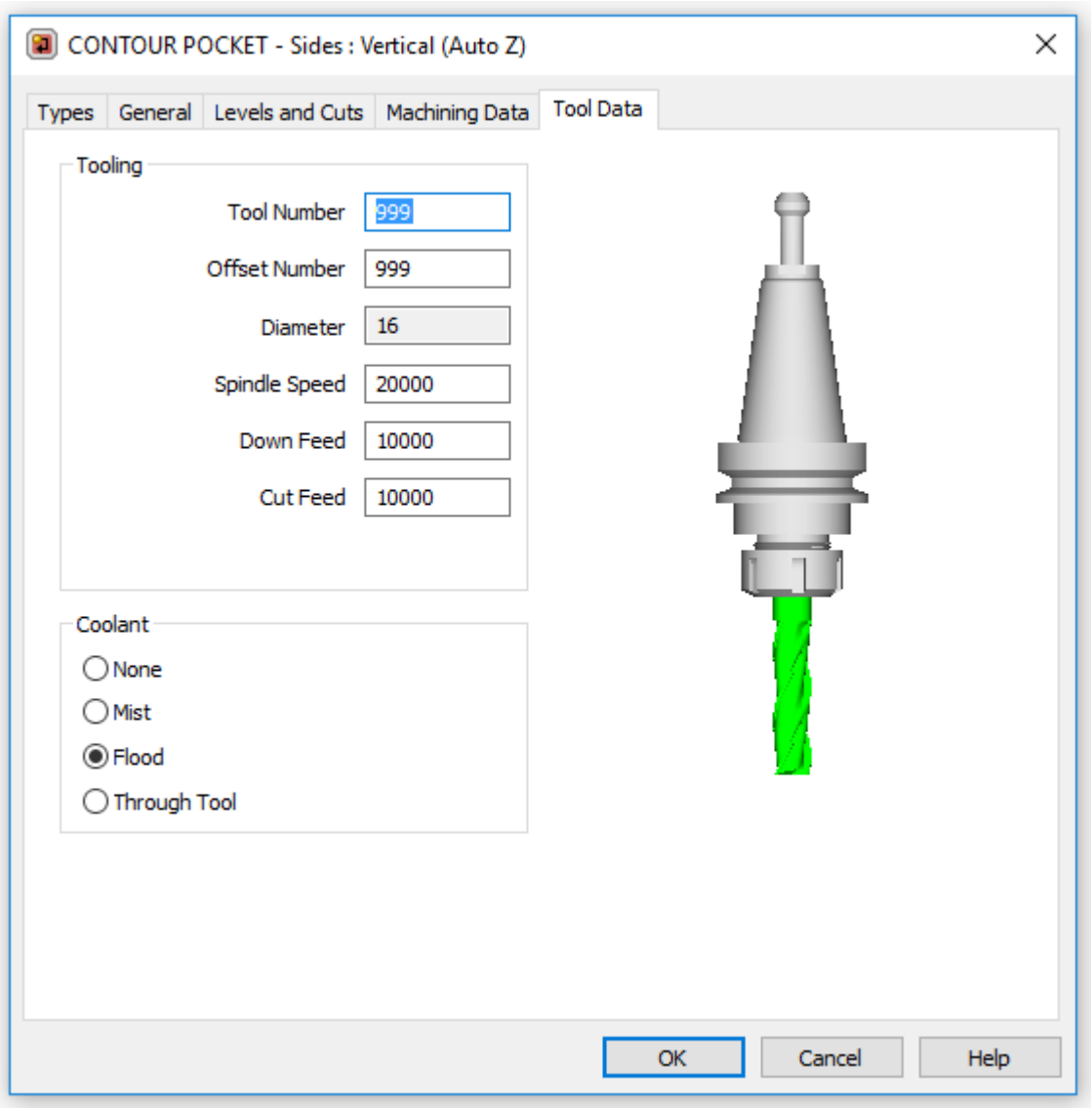



**Figure 28 - Pocketing Tool Data tab**

Tool Data sets the tooling speed and feed rate options.

Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select Profiles shown below then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

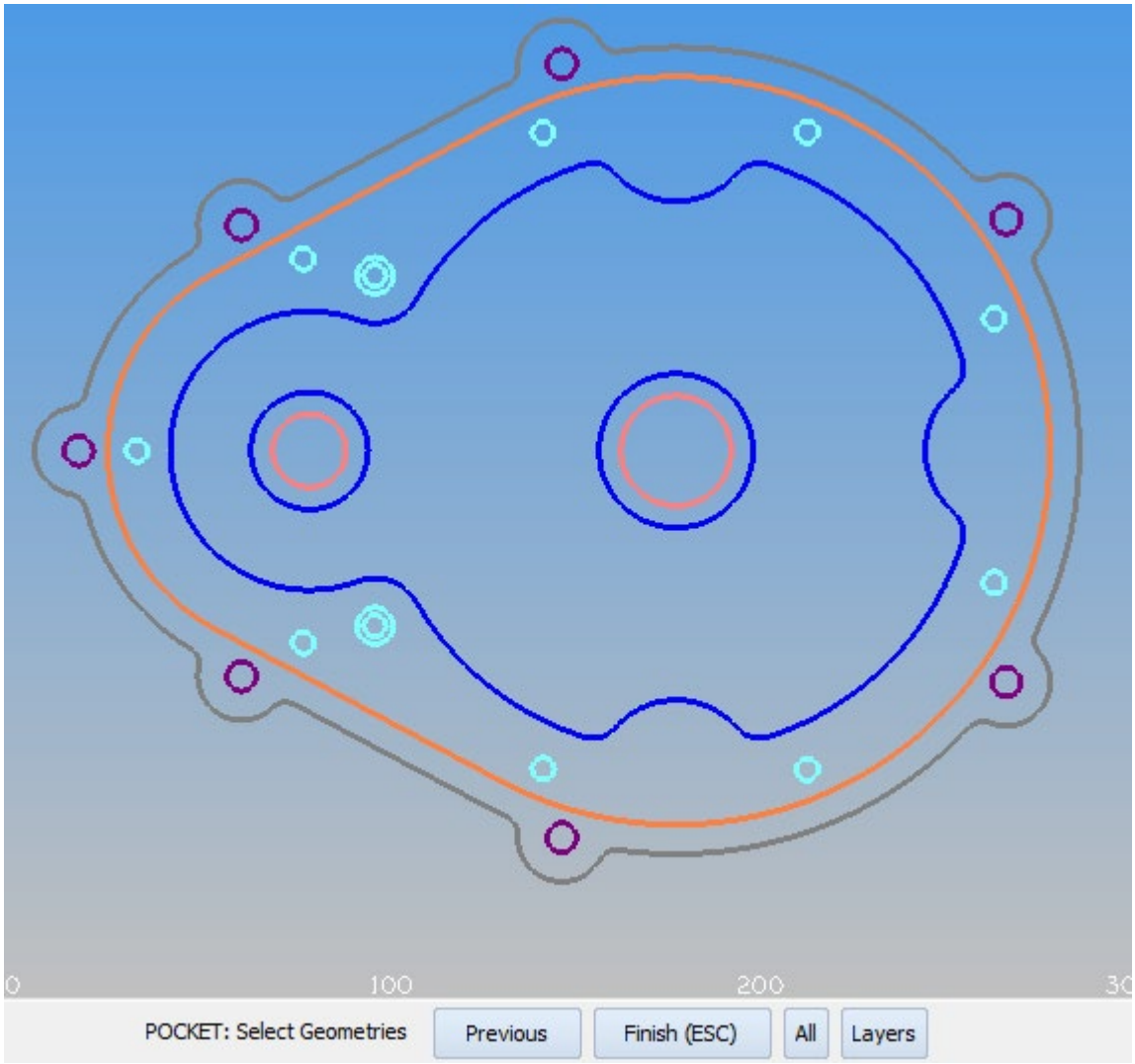



**Figure 29 - Geometry Selection for Waveform Pocketing**

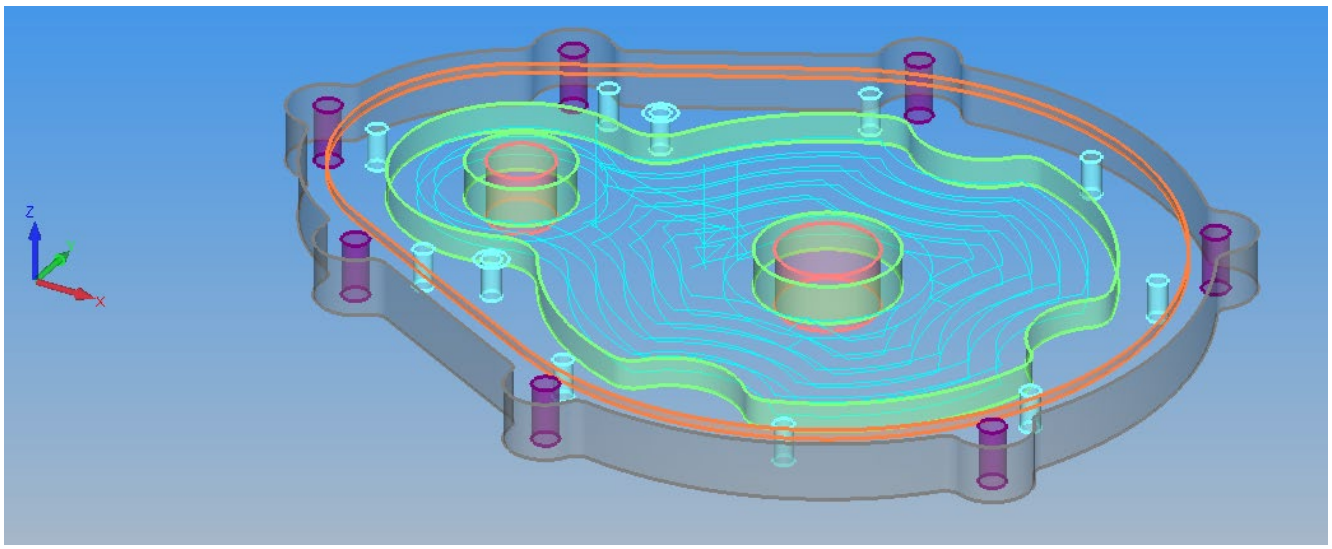


Figure 30 - Waveform pocket visualised in Isometric view

Because the tool is 16mm diameter and there are internal fillet radii of 5mm this means there is excess material over the 0.5mm stock allowance that needs to be removed prior to the finishing cut.

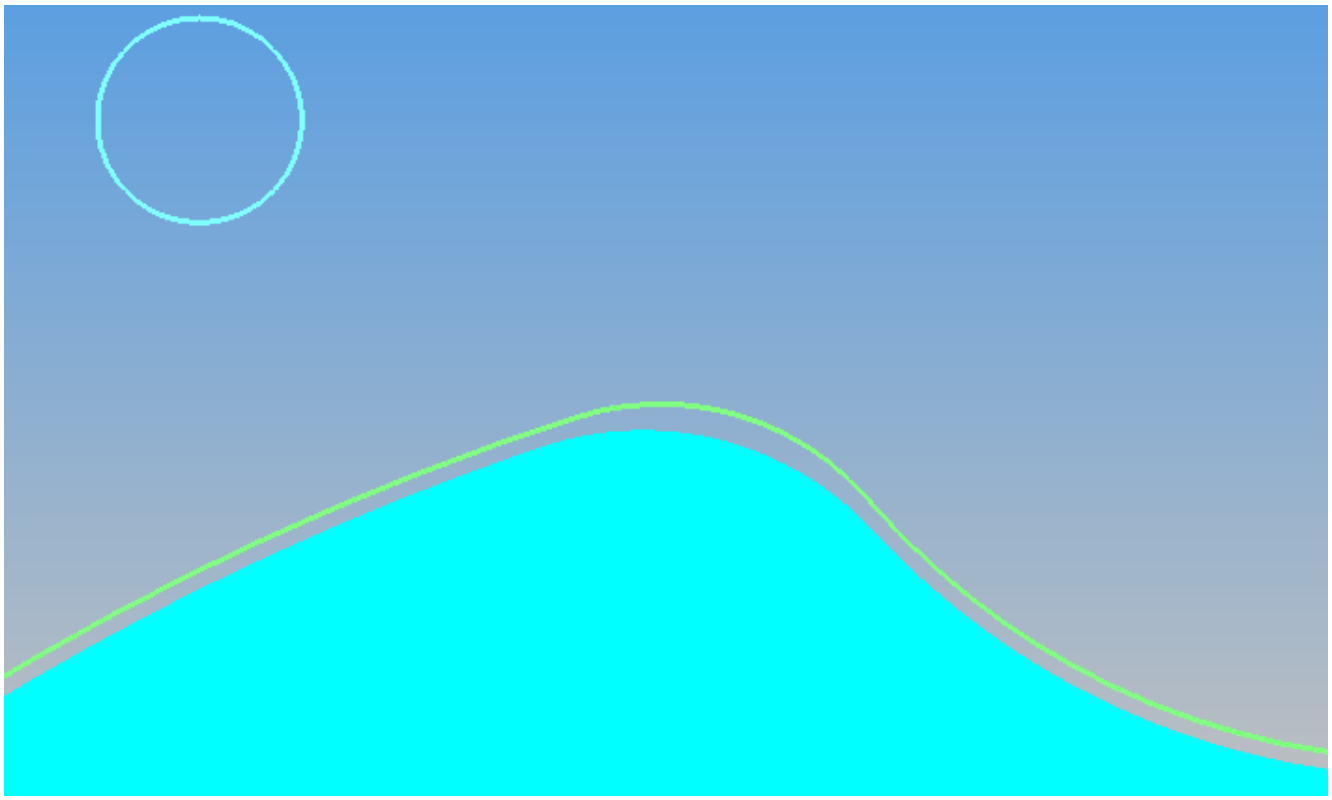


Figure 31 - Incomplete fillet radii areas

# Pocket Ø20 & Ø30 Holes (Profiles 9,10,) leave 0.5 Stock

Select **MACHINE > Pocketing**

## Types

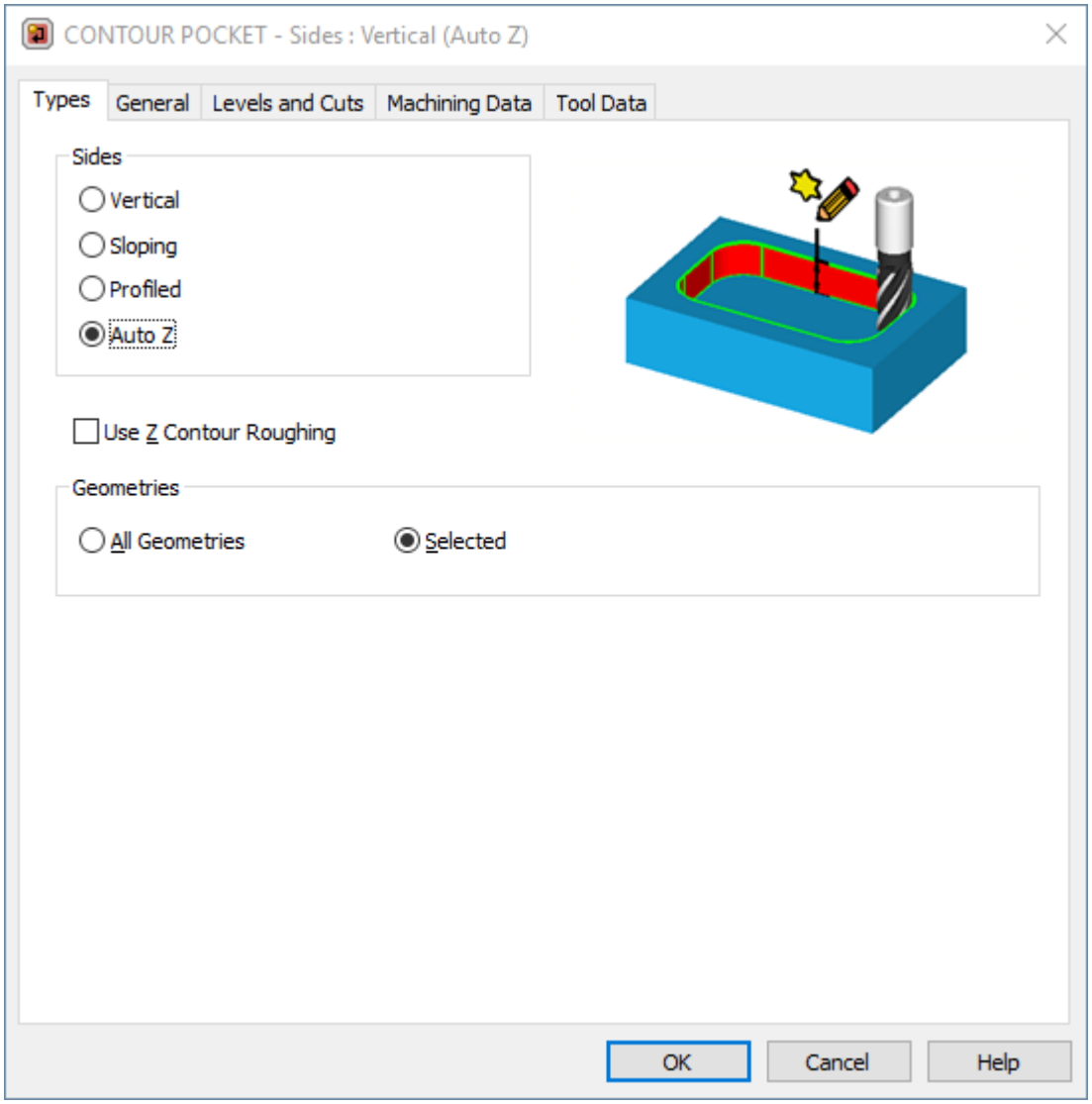



**Figure 32 - Pocketing Types tab**

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** then ⊙ **Selected.**

## General

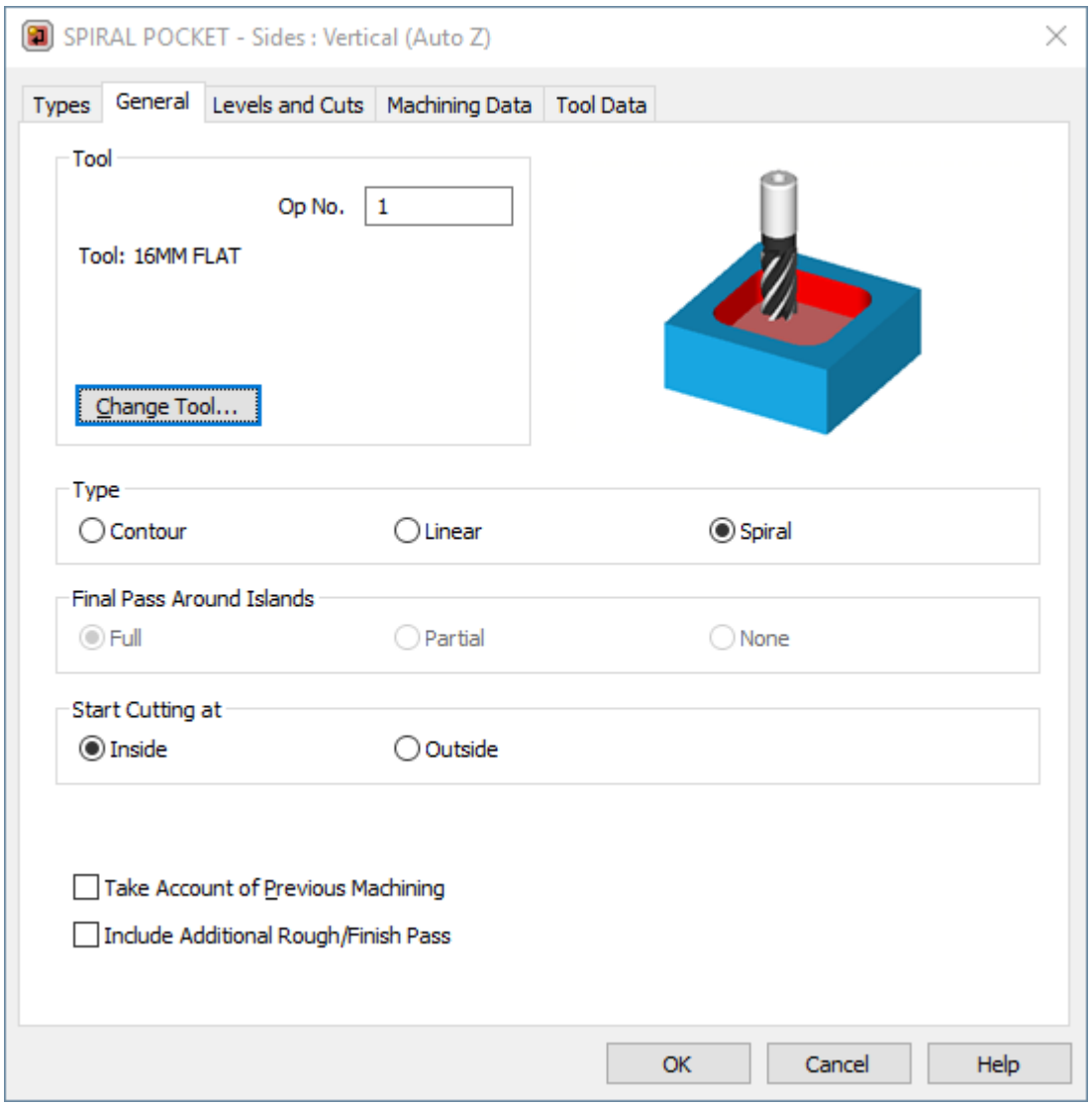



**Figure 33 - Pocketing General tab**

General sets the pocketing style.
Set the options ⊙ **Spiral.**

> It is worth noting that using the ☑ Use Z Contour Roughing in the previous dialogue option will allow the selection of any form to work with the Spiral style of machining, however, if you do not use the Z Contour option, ⊙ Spiral will only work using circles as the chosen profile geometry.

## Levels and Cuts

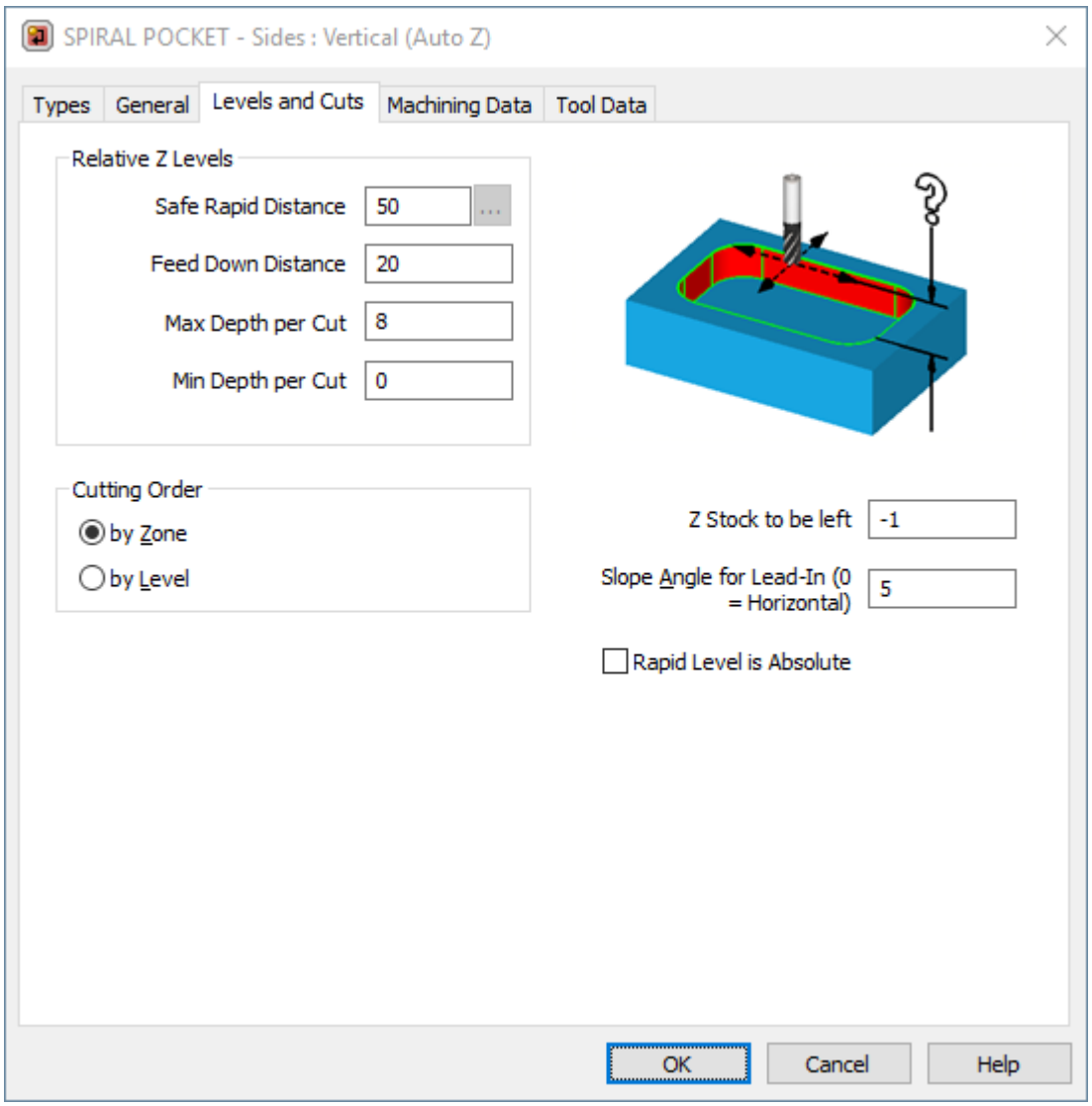



**Figure 34 - Pocketing Levels and Cuts tab**

Levels and Cuts sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid, Rapid Down To** as appropriate, **Max Depth per Cut =8 and the Z Stock to be left to -1** to allow the tool to cut deeper.

## Machining Data

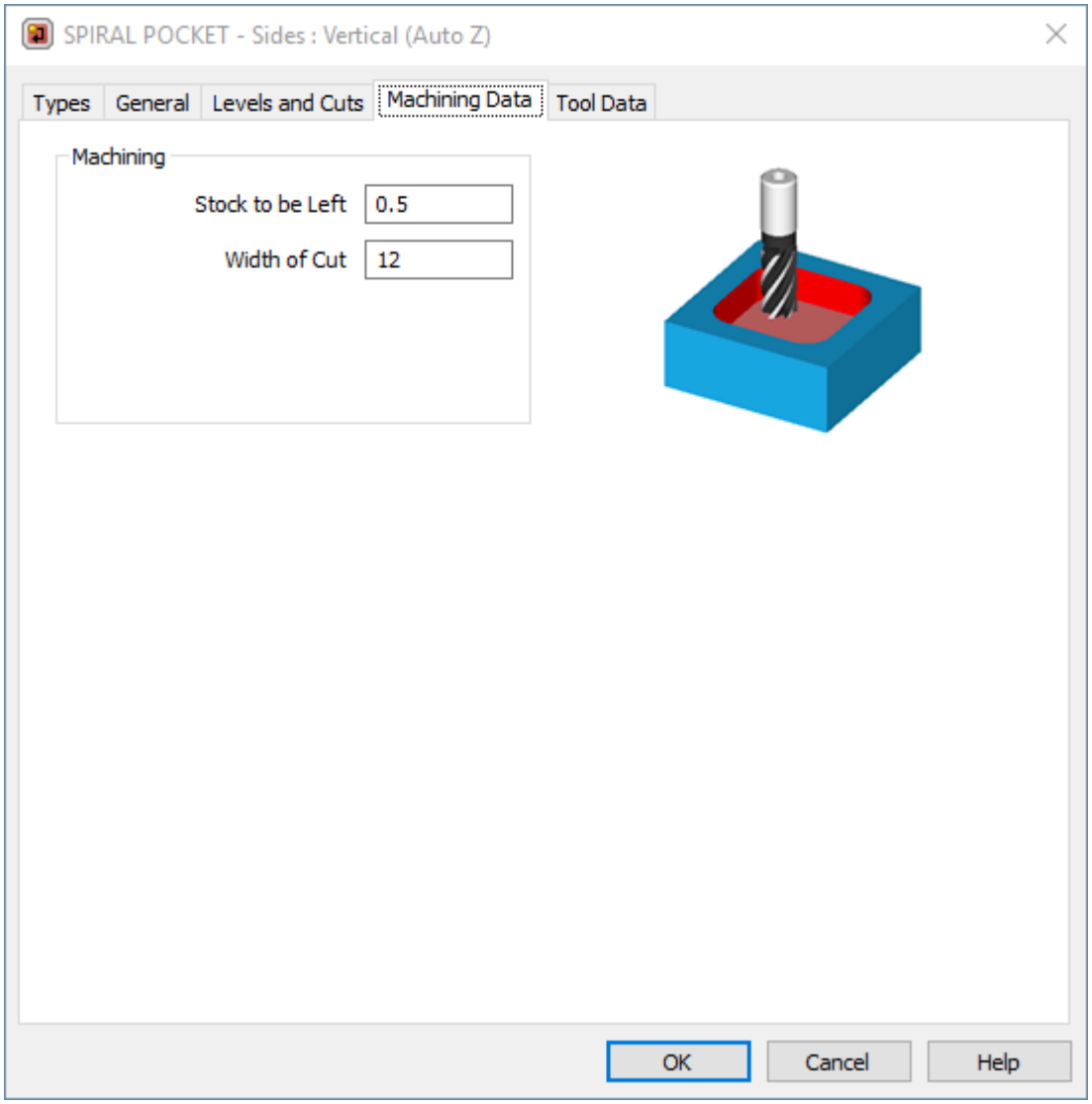



**Figure 35 - Pocketing Machining Data tab**

Machining Data sets the stock allowance and width of cut.
**Stock to be Left = 0.5**, **Width of Cut = 12**.

## Tool Data

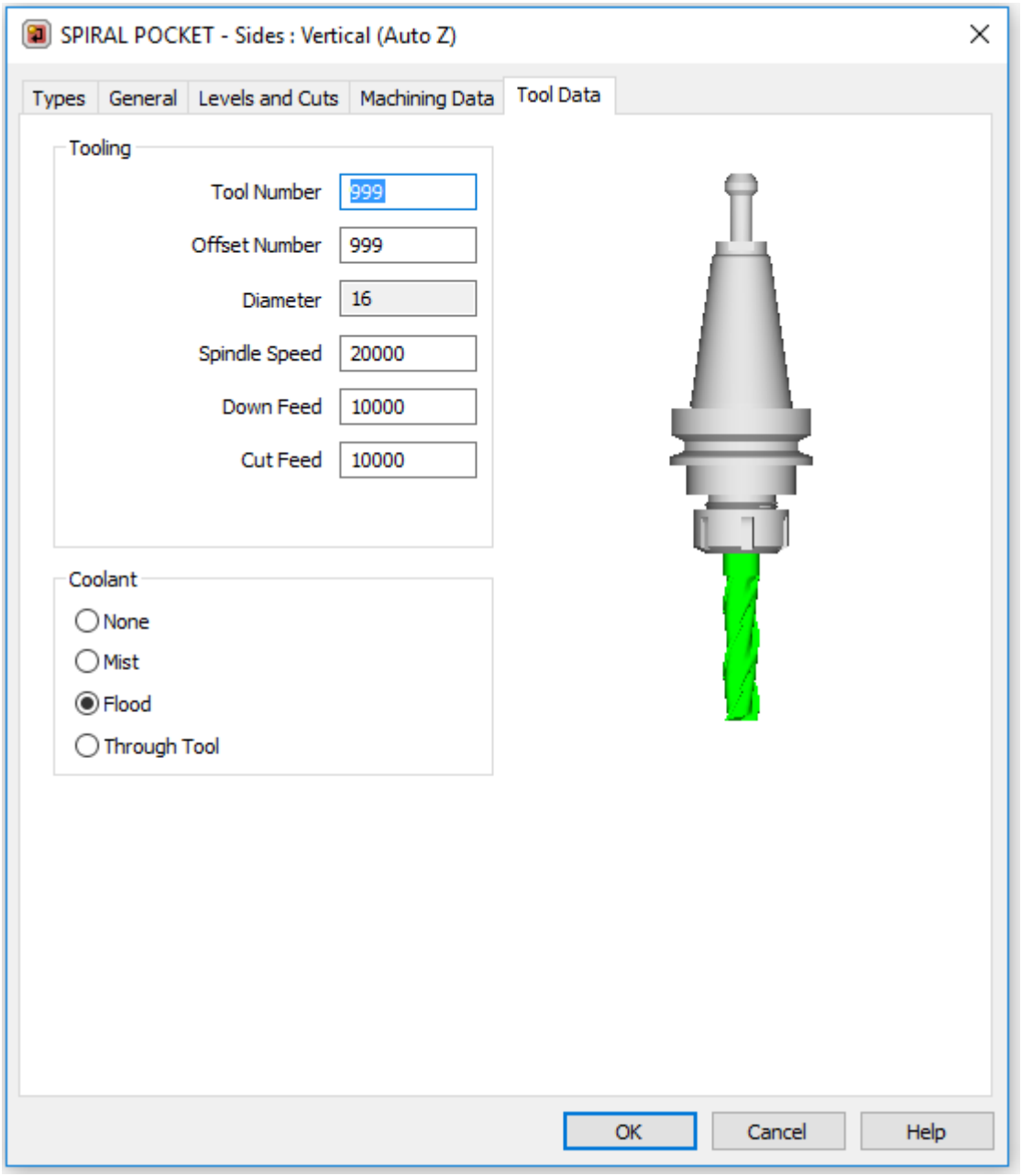



**Figure 36 - Pocketing Tool Data tab**

Tool Data sets the tooling speed and feed rate options.

Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select the geometries shown below.

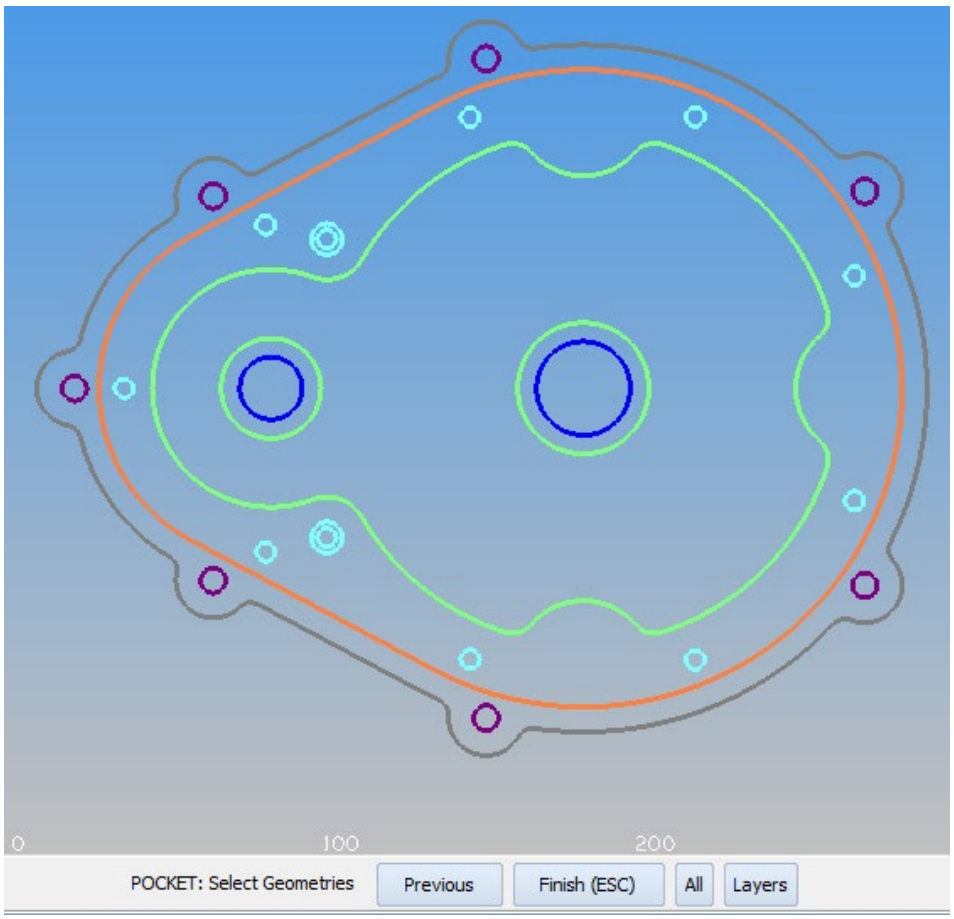



**Figure 37 - Geometries to Spiral Pocket**

**<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

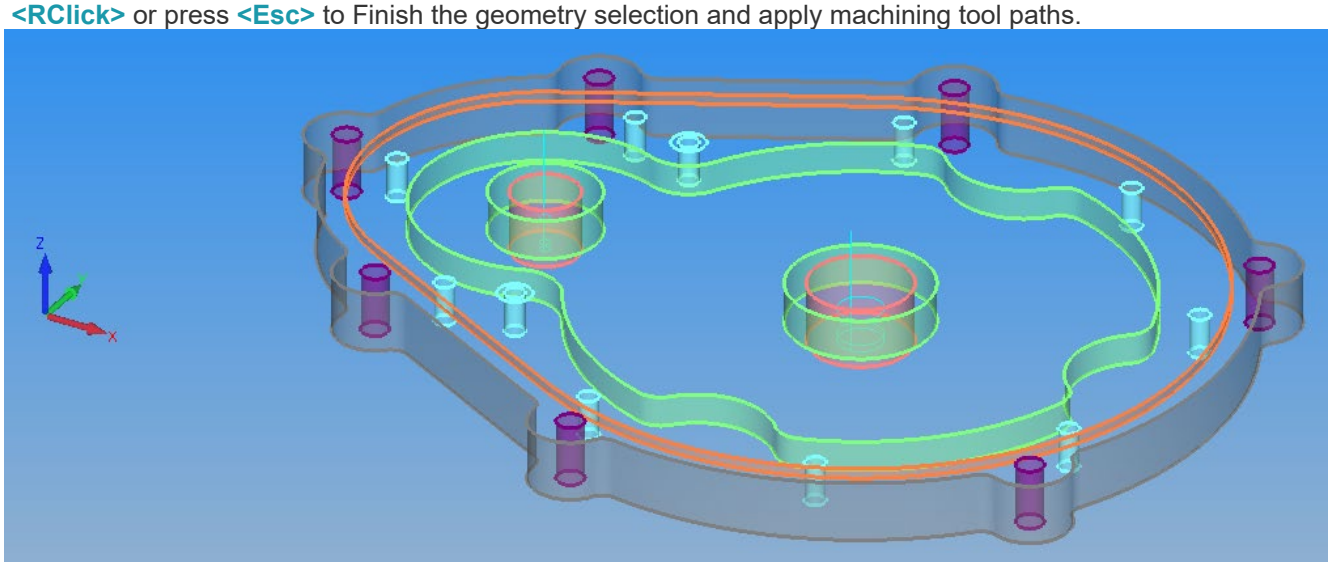



**Figure 38 - Two through holes seen in Isometric projection**

# Adding or removing geometries from a Machining strategy.

If you are presented with a scenario, whereby you have either,

a. selected more geometries than required,

or, more likely

b. not selected enough geometries, or the design is modified and extra features have been added.

It is possible to add or remove geometries from an existing machining strategy.
Firstly, open the tree structure for the machining strategy in the **Operations** Project Manager Page by **<LClick>** on the **>** next to the name.

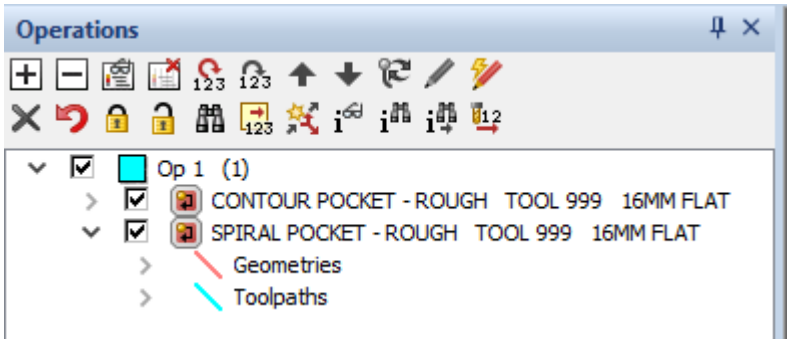



**Figure 39 - - Expanded Pocket cycle in the Operations page of the Project Manager**

To Add more geometries into the operation **<RClick>** on the **Geometries** name to display the local popup menu and select **Add Geometries** then select the geometries to be added to the operation.

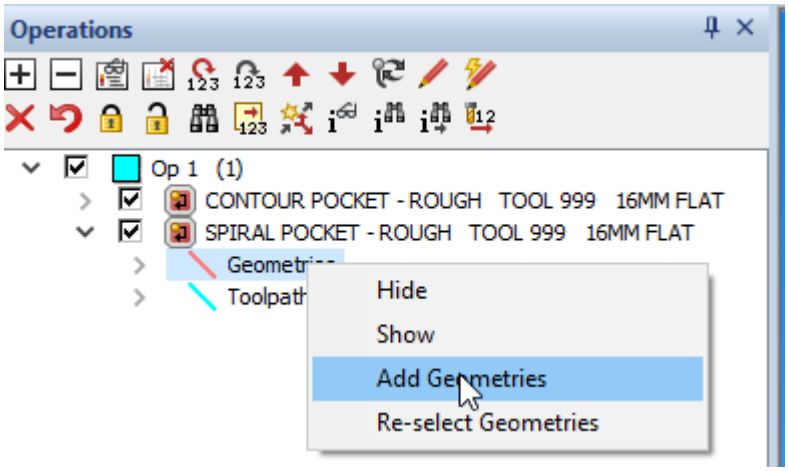



**Figure 40 - <RClick> options to add extra geometries to existing operations**

To Remove a geometry or Move a geometry to its own operation **<RClick>** the Contour (Geo2) name in the tree structure.

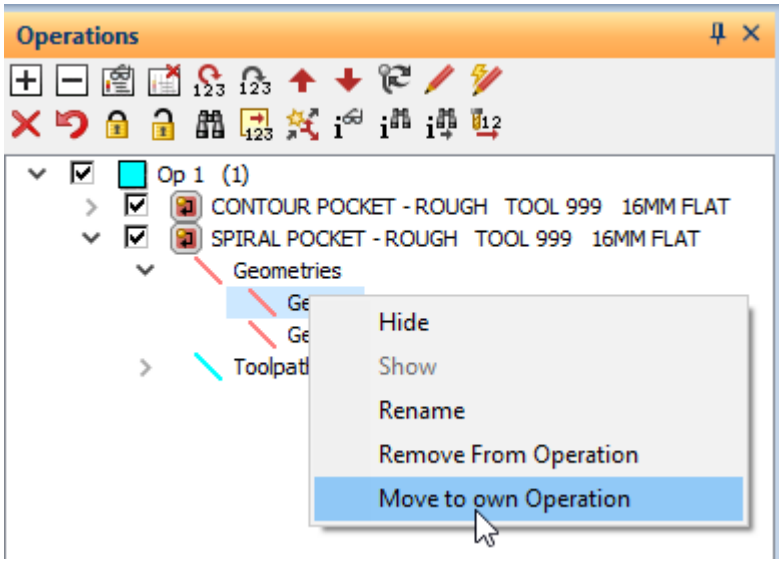



**Figure 41 - <RClick> options to remove geometries from existing operations**

## Edit the Strategy Parameters

To edit the strategy parameters **<RClick>** on the strategy name and select **Edit** from the popup menu. This will then allow access the tool path creation Dialogues as describe previously.

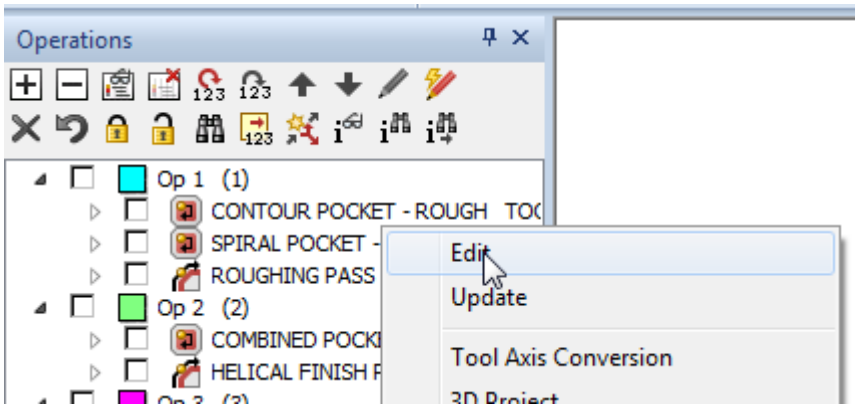



**Figure 42 - <RClick> options on an operation**

# Rough & Finish Profile 2, 3mm deep and 4 XY cuts from 34mm stock

Select **MACHINE > Rough or Finish**

## Types

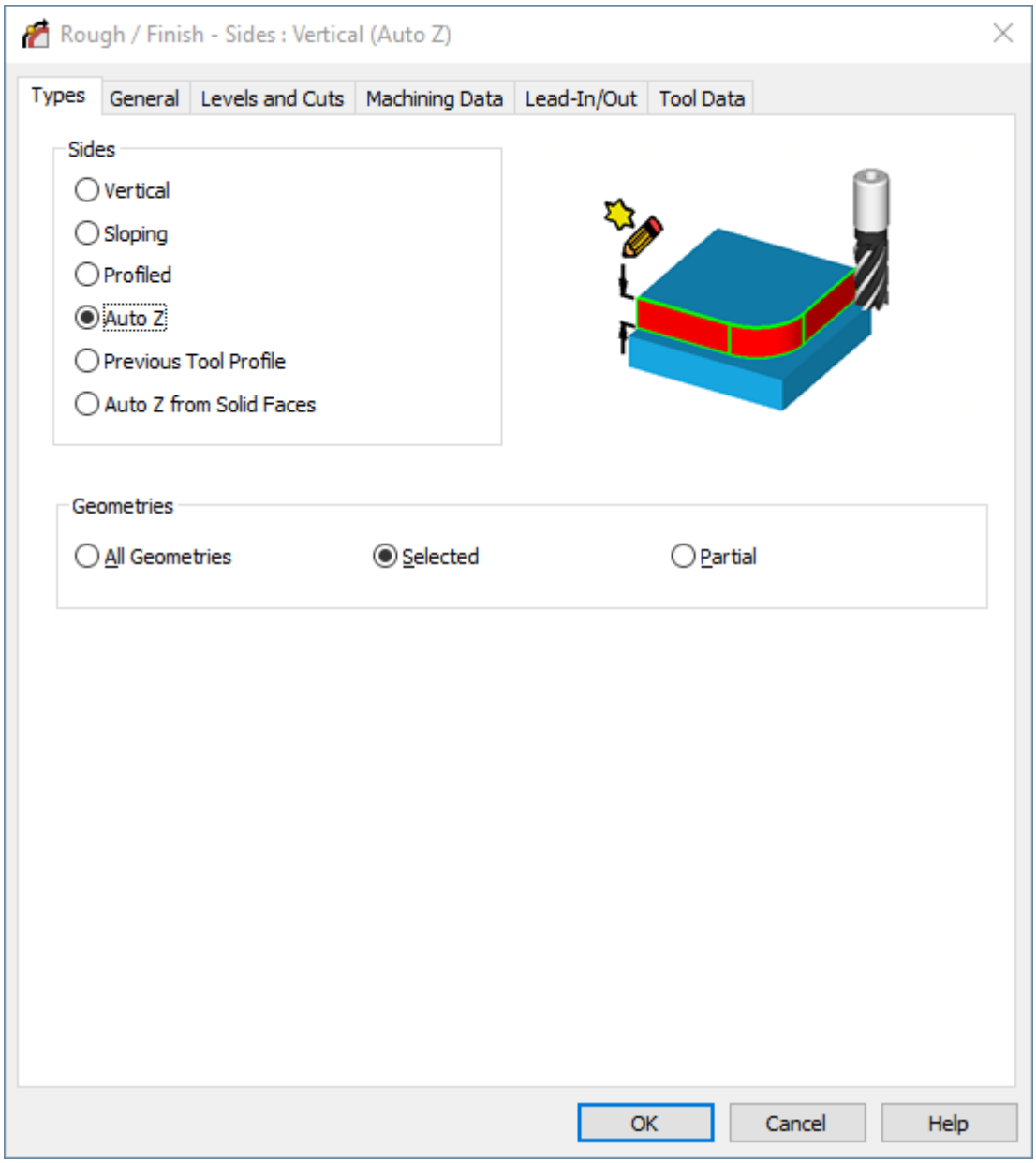



**Figure 43 - Rough and Finish Types tab**

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** ⊙ **Selected.**

## General

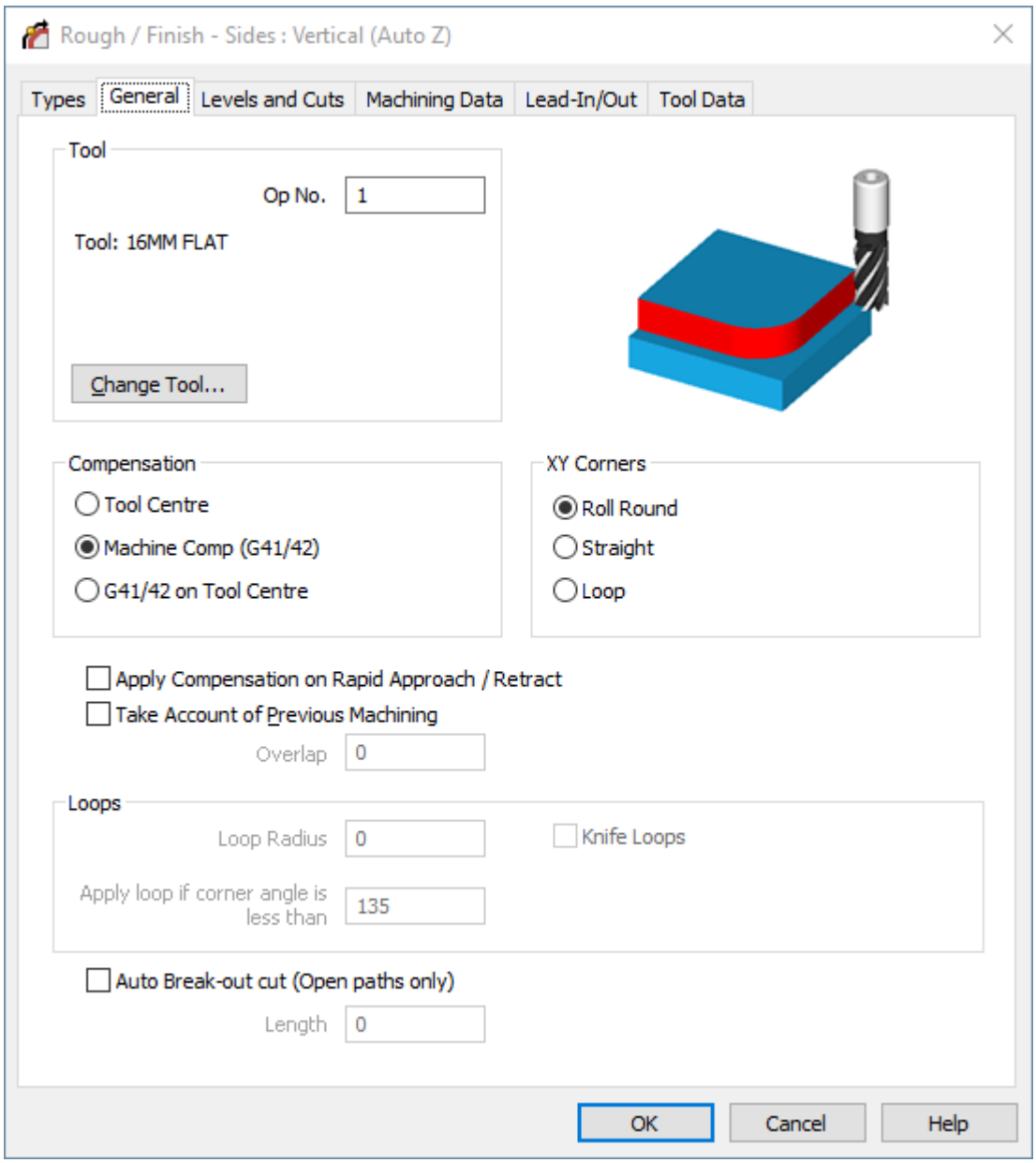



**Figure 44 - Rough and Finish general tab**

General sets the output type. Set the options **⊙ Machine Comp (G41/42)**.

> Some routers require the option ☑ Apply Compensation on Rapid Approach / Retract to be activated, select this option if it is applicable to your post processor and Router. If you select this option and it is not applicable you will be presented with an error Dialogue when listing or outputting the NC file.

## Levels and Cuts

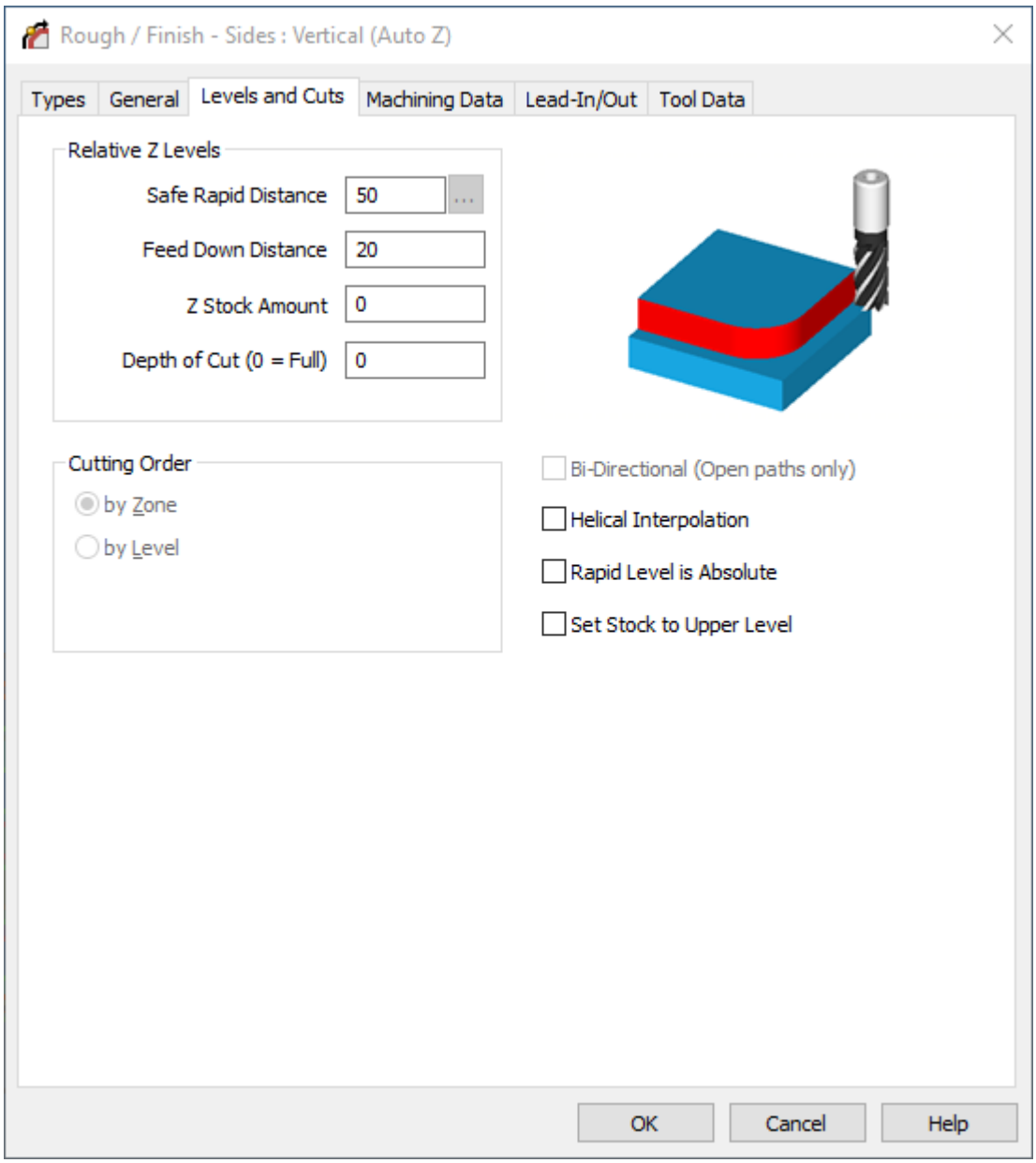



**Figure 45 - Rough and Finish Levels and Cuts tab**

Levels and Cuts sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid, Rapid Down To** as appropriate, **Z Stock Amount = 0 (to finish the floor) Depth of Cut = 0,** ☐ Helical Interpolation.

## Machining Data

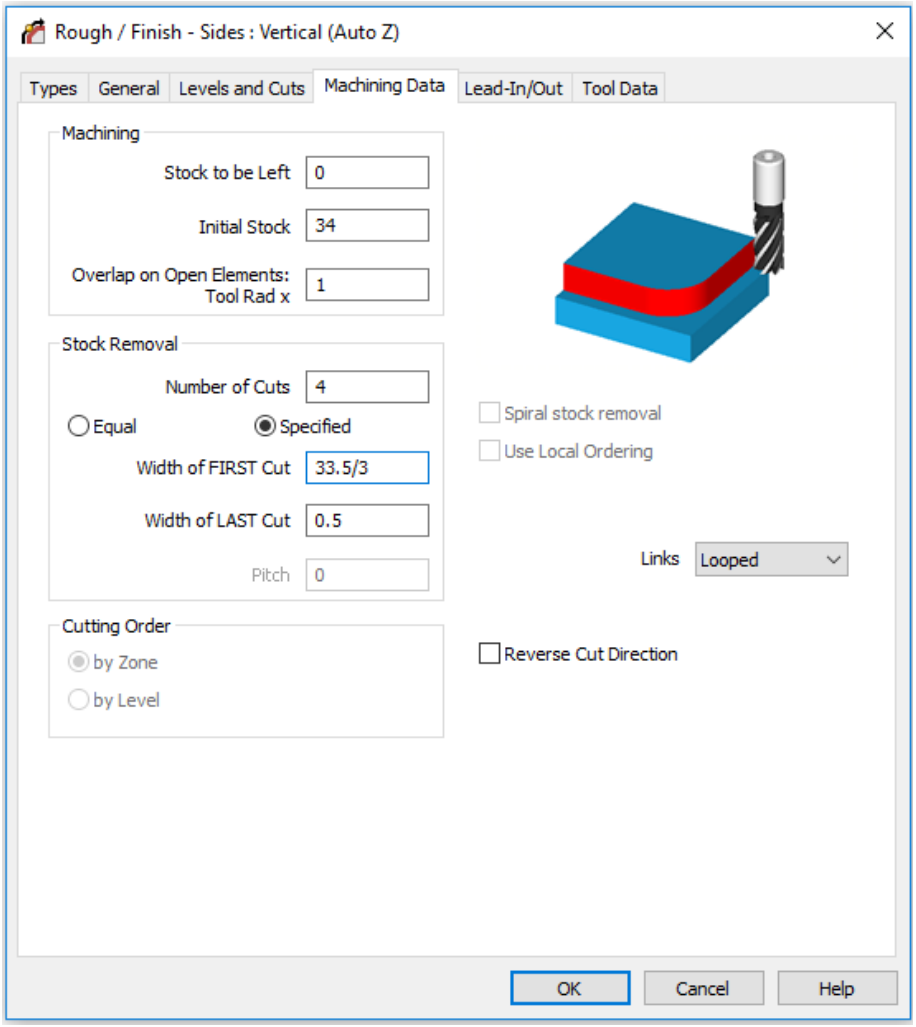



**Figure 46 - Rough and Finish Machining Data tab**

Machining Data sets the information regarding cutting options for the X&Y action.
Set the **Stock to be Left** = **0**,
**Initial Stock = 34,**
**Number of Cuts = 4**,
**Widths of Cut** ⊙**Specified**,
**Width of FIRST Cut =33.5/3**
**Width of LAST Cut = 0.5,**
**Set the links = Looped.**

This set of options allows for three equal roughing passes in the XY direction to remove most the stock on the 3mm deep level, and then apply a finish pass removing the final 0.5mm.
Setting the Links option to **Looped** allows the cutter to remain at the final Z depth and not lift up and down between passes, which can waste time in some environments.

> This type of cycle definition can save the need of repeating the same cycle again for multiple rough and finish passes, it must be noted however, that the cutter must be of a type that you allow doing both rough passes and finishing passes.

## Lead-In/out

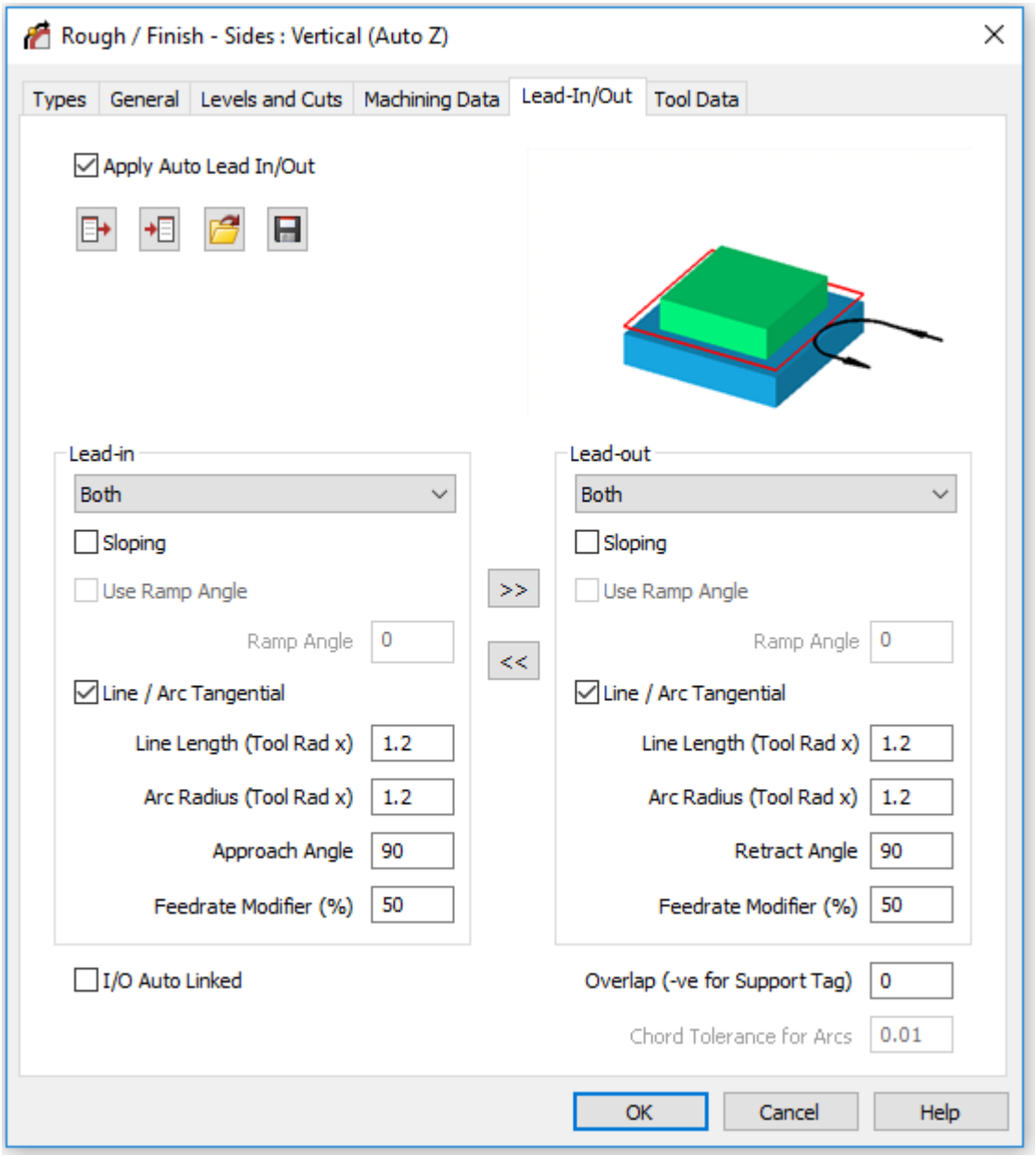



**Figure 47 - Rough and Finish Lead-In/Out tab**

Ensure that the option **☑ Apply Auto Lead In/out** is active.

Make the options choices using the drop-down menus and the various dialogue options to suit your own requirements for manufacturing.

Using the **Save Defaults**  option will allow ease of applying the same values during later machining operations.

## Tool Data

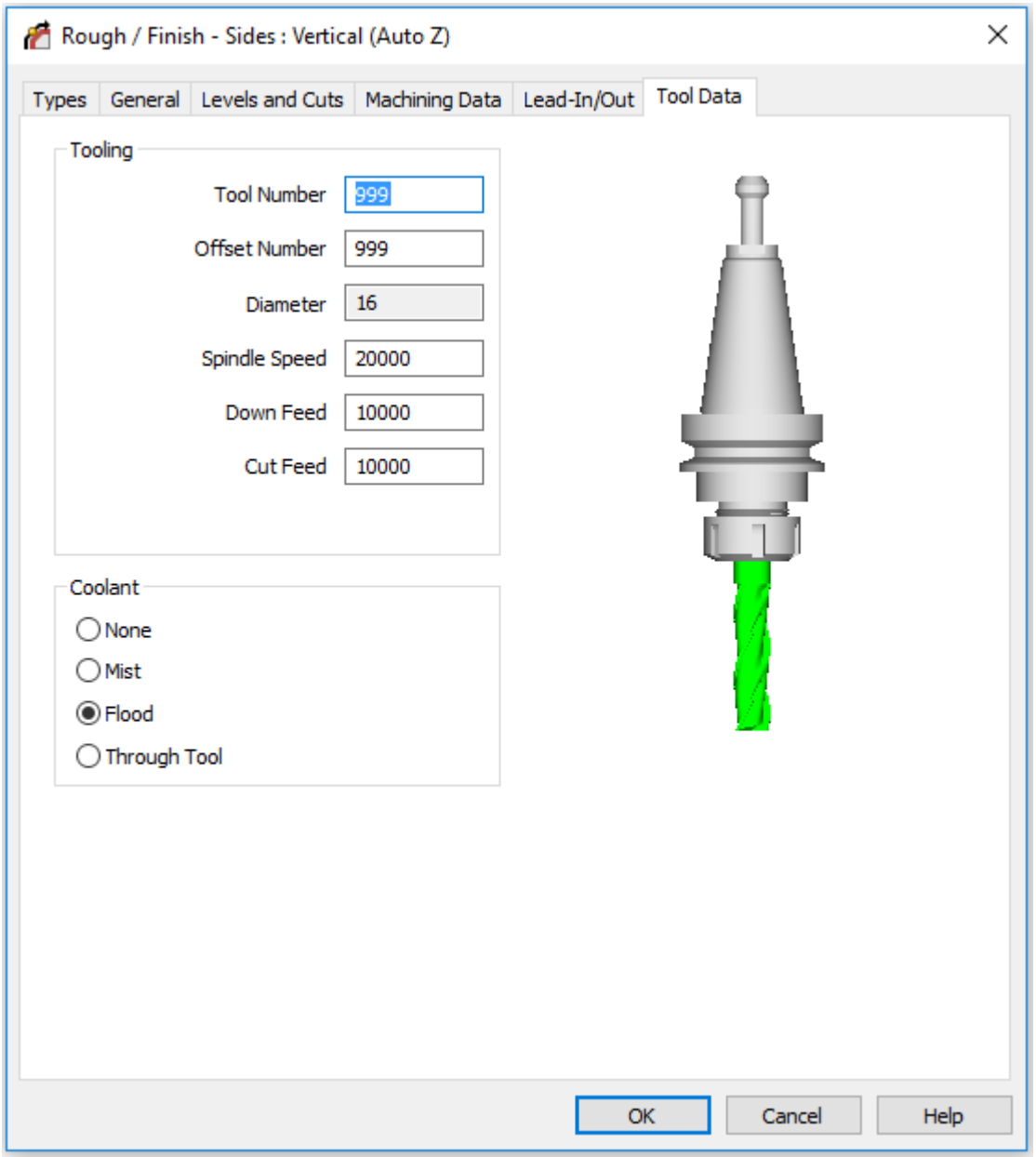



**Figure 48 - Rough and Finish Tool Data tab**

Tool Data sets the tooling speed and feed rate options.

Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select the oval geometry shown below.

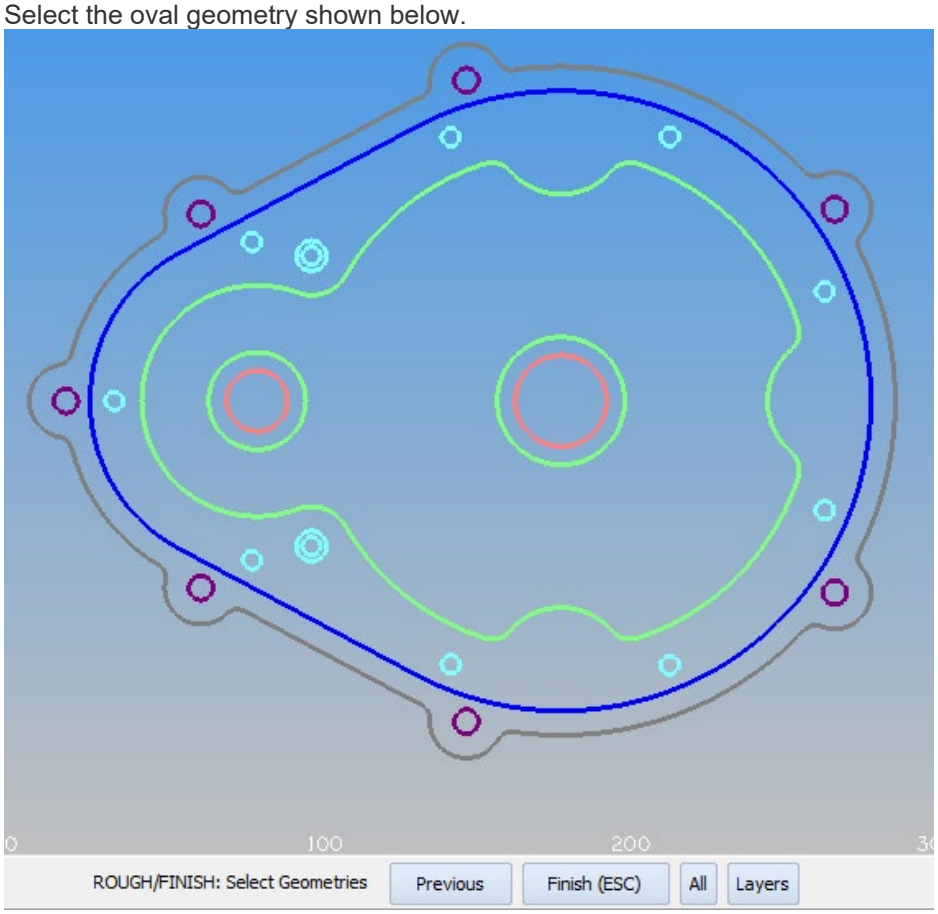



**Figure 49 - Geometry selection for the oval profile**

**<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

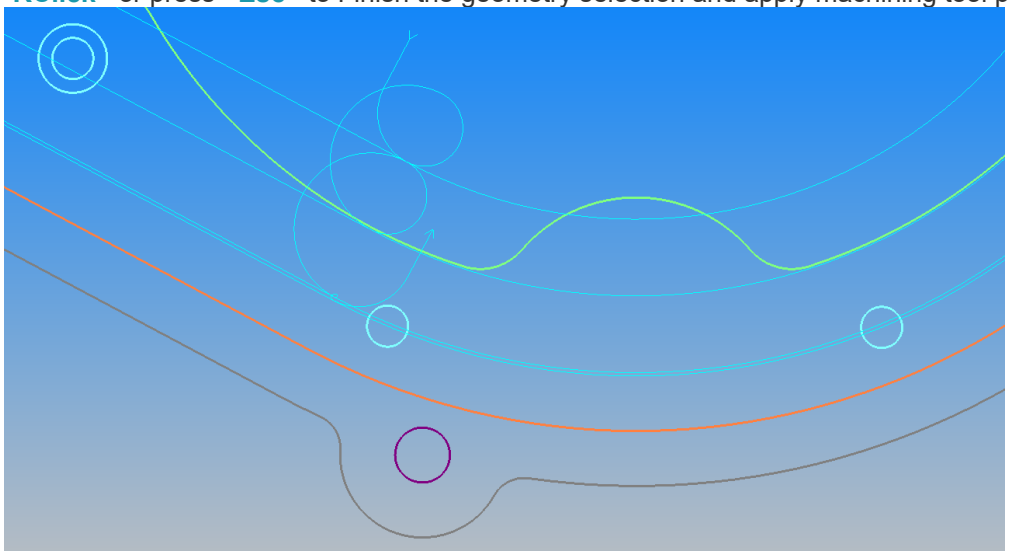


**Figure 50 - Applied machining for the oval profile**

# Operation 2

## Residual Pocket Profiles 6, 7, 8 leave 0.5 Stock and Finish Profile to size

Select **MACHINE > Select Material**

> This is only required for tools that require feed and speed calculations

Select **MACHINE > Select Tool**
Select the **8mm Flat** from the **Training** folder.
The tool will be displayed on the screen **<LClick>** or press ↵ to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

Select **MACHINE > Pocketing**

### Types

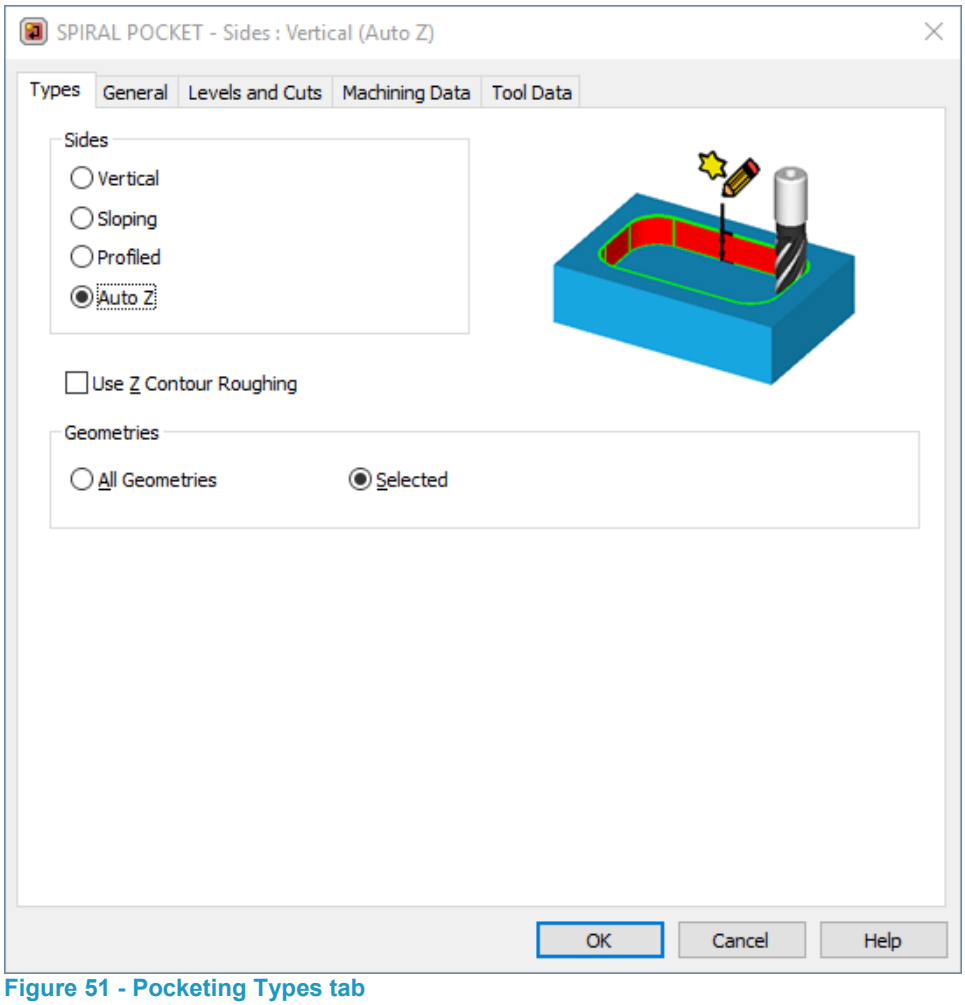



**Figure 51 - Pocketing Types tab**

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** ☑ **Use Z Contour Roughing**.

## General

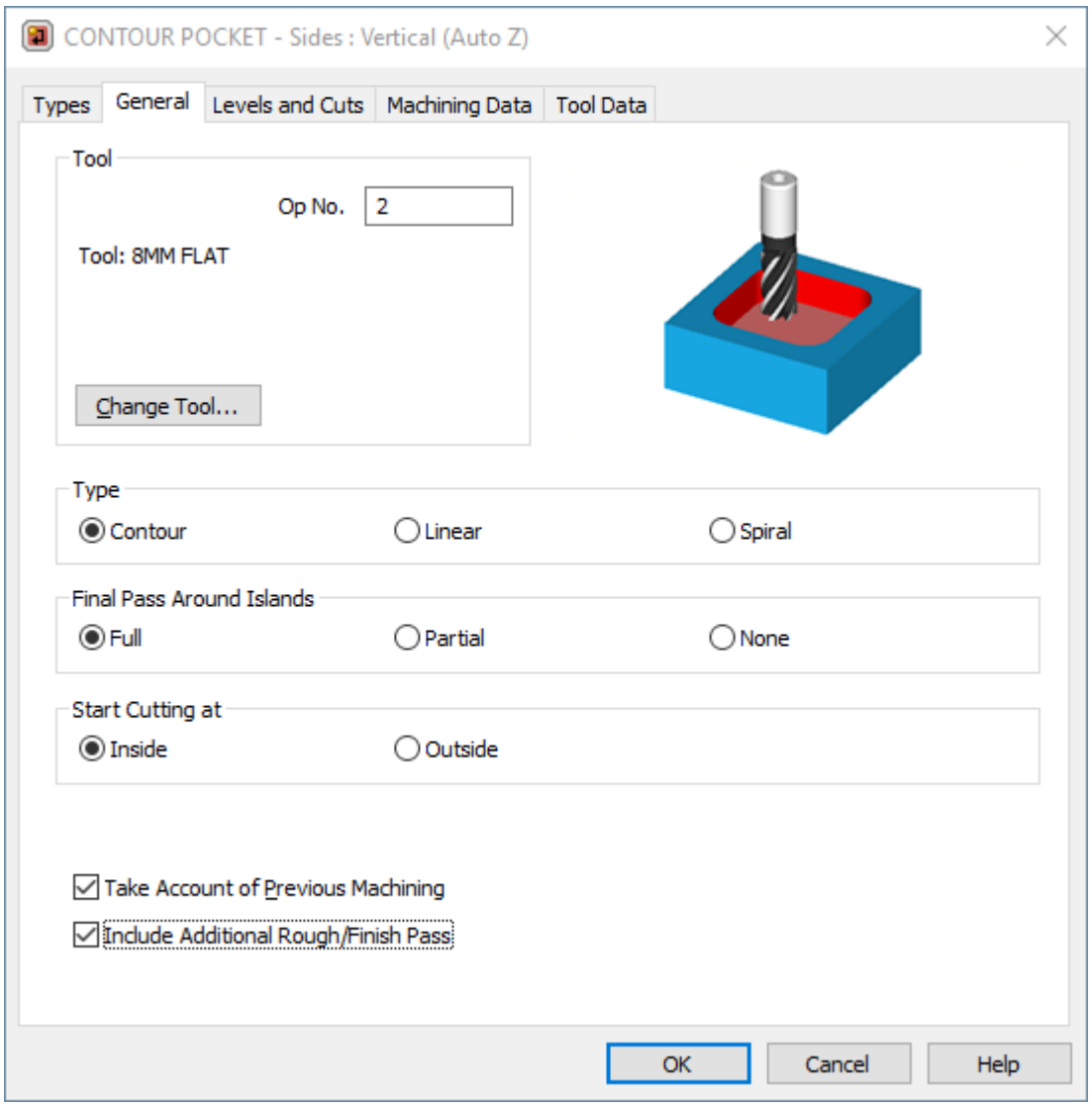



**Figure 52 - Pocketing General tab**

General sets the pocketing style.
Set the options ⊙ **Contour.**

Select the options,
☑ **Take Account of Previous Machining,**
☑ **Include Additional Rough/Finish Pass**.

## Levels and Cuts

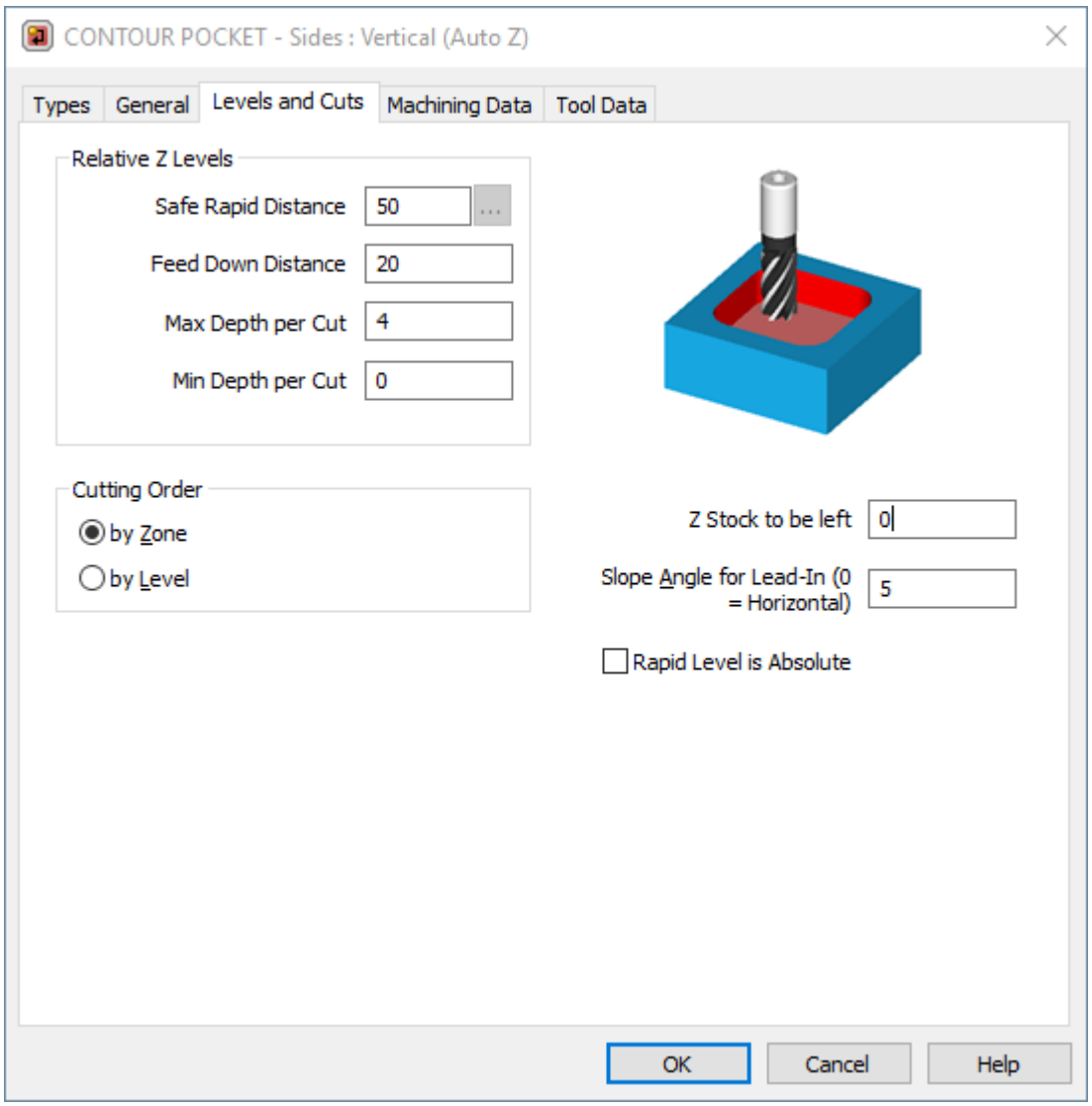



**Figure 53 - Pocketing Levels and Cuts tab**

Levels and Cuts sets Z levels and number and nature of the cuts in Z.

Set the **Safe Rapid**, **Rapid Down To** as appropriate, **Max Depth per Cut = 4, Slope Angle for Lead In = 5**.

## Machining Data

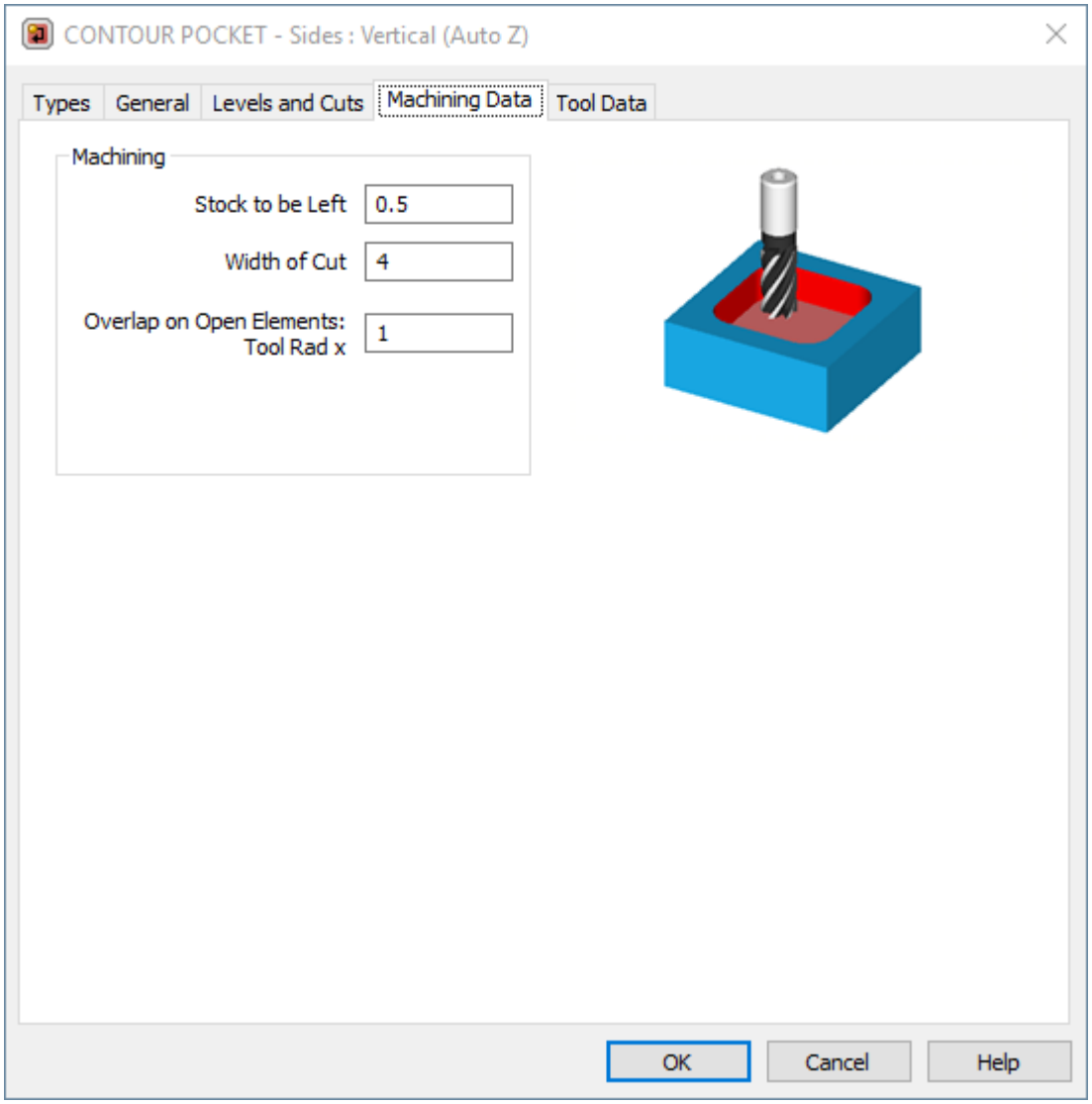



**Figure 54 - Pocketing Machining Data tab**

Machining Data sets the information for X&Y cuts.
Set the **Width of Cut for Final Pass = 0.2, Width of Cut = 4**.

Note that on this dialogue page, the Stock options MUST match the cycle being compared.

When using the Waveform strategy, these are automatically greyed out so that you cannot alter them by accident.

## Tool Data

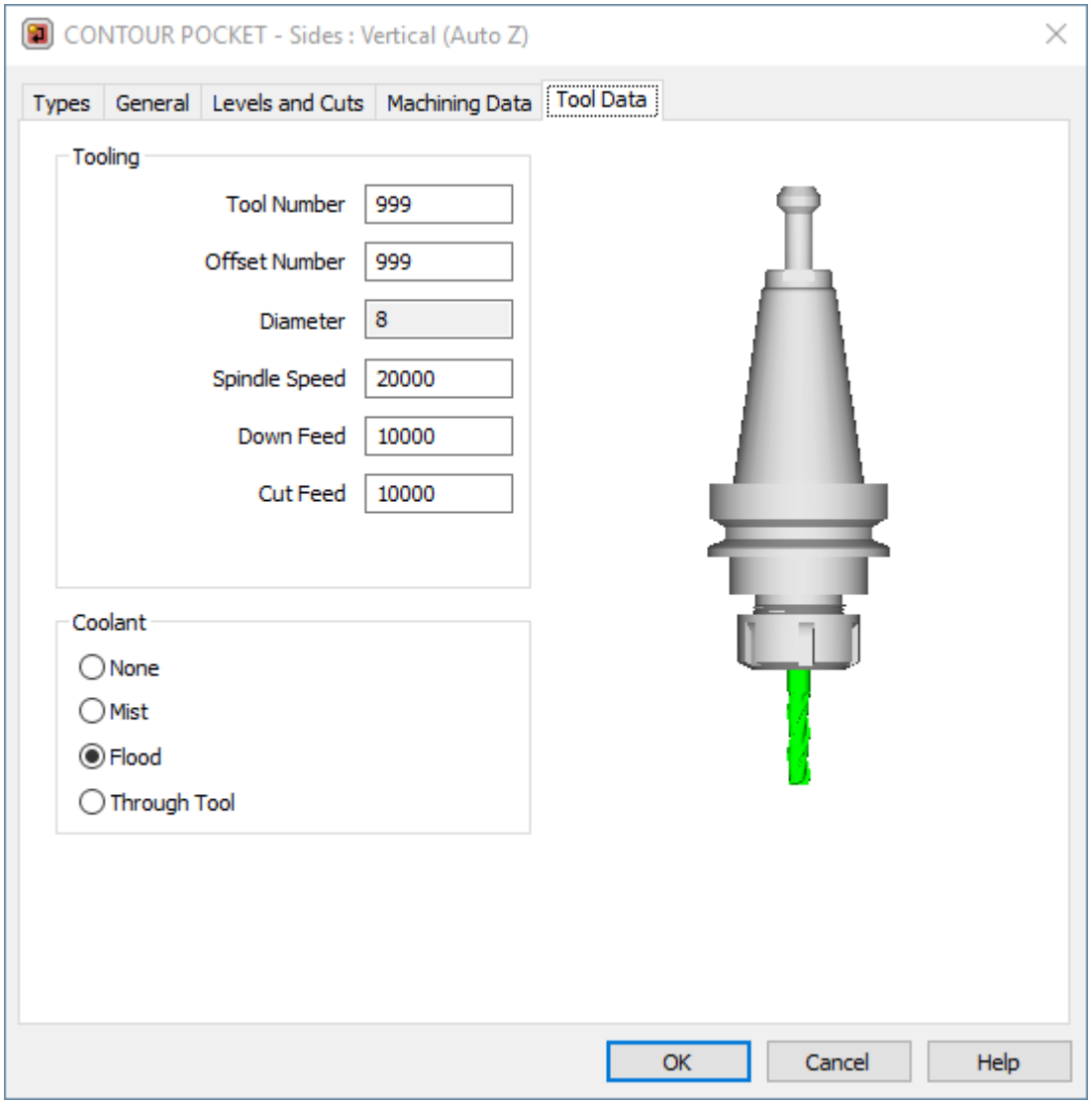



**Figure 55 - Pocketing Tool Data tab**

Tool Data sets the tooling speed and feed rate options.

Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK]**.

# The Finishing Dialogues will be displayed

## General

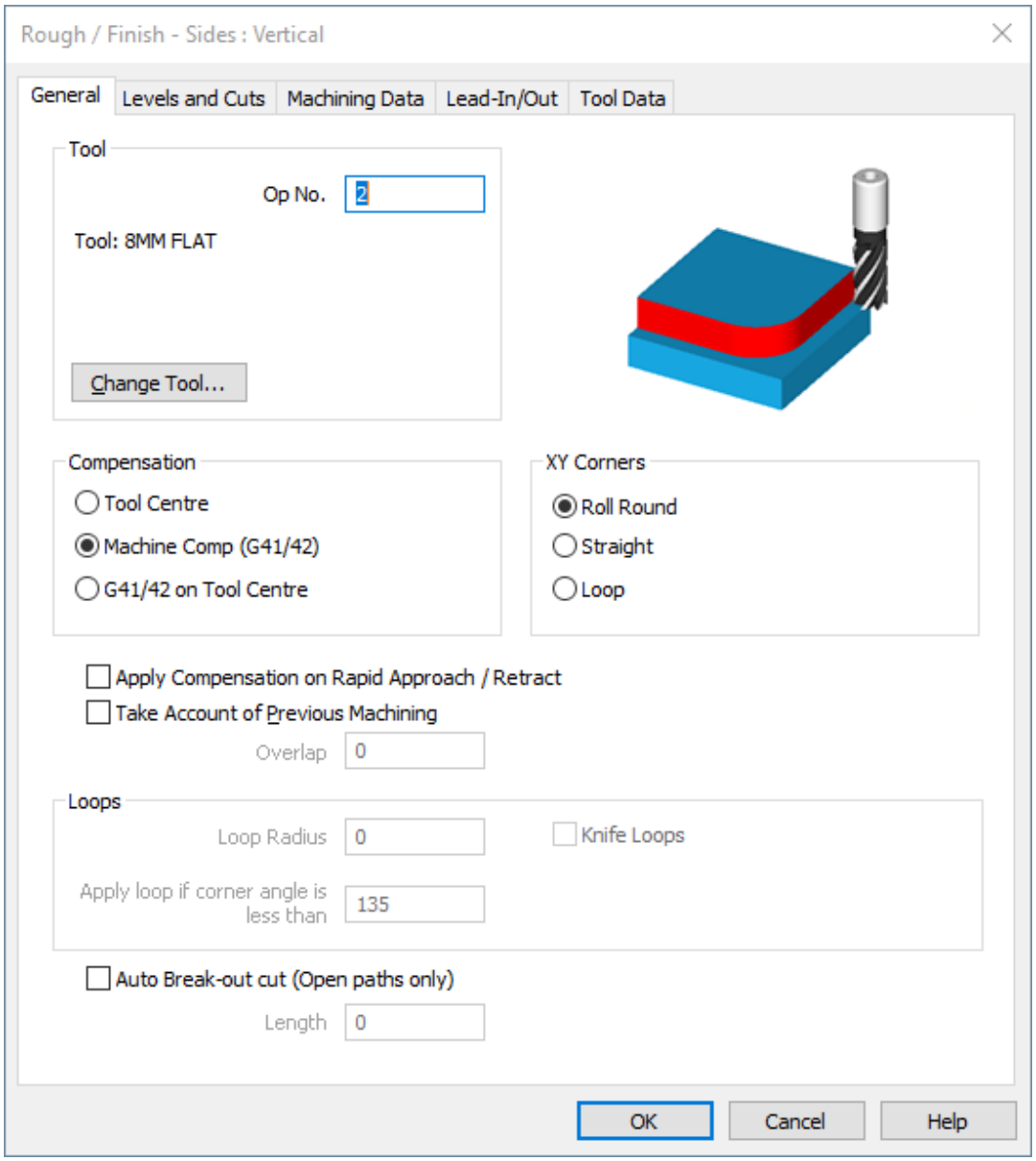



**Figure 56 - Rough and Finish general tab**

General sets the output type.
Set the options ⊙ **Machine Comp (G41/42),** Corners N/A on this job.

Some machines require the option ☑ Apply Compensation on Rapid Approach / Retract to be activated, select this option if it is applicable to your post processor and machine. If you select this option and it is not applicable you will be presented with an error Dialogue when listing or outputting the NC file.

## Levels and Cuts

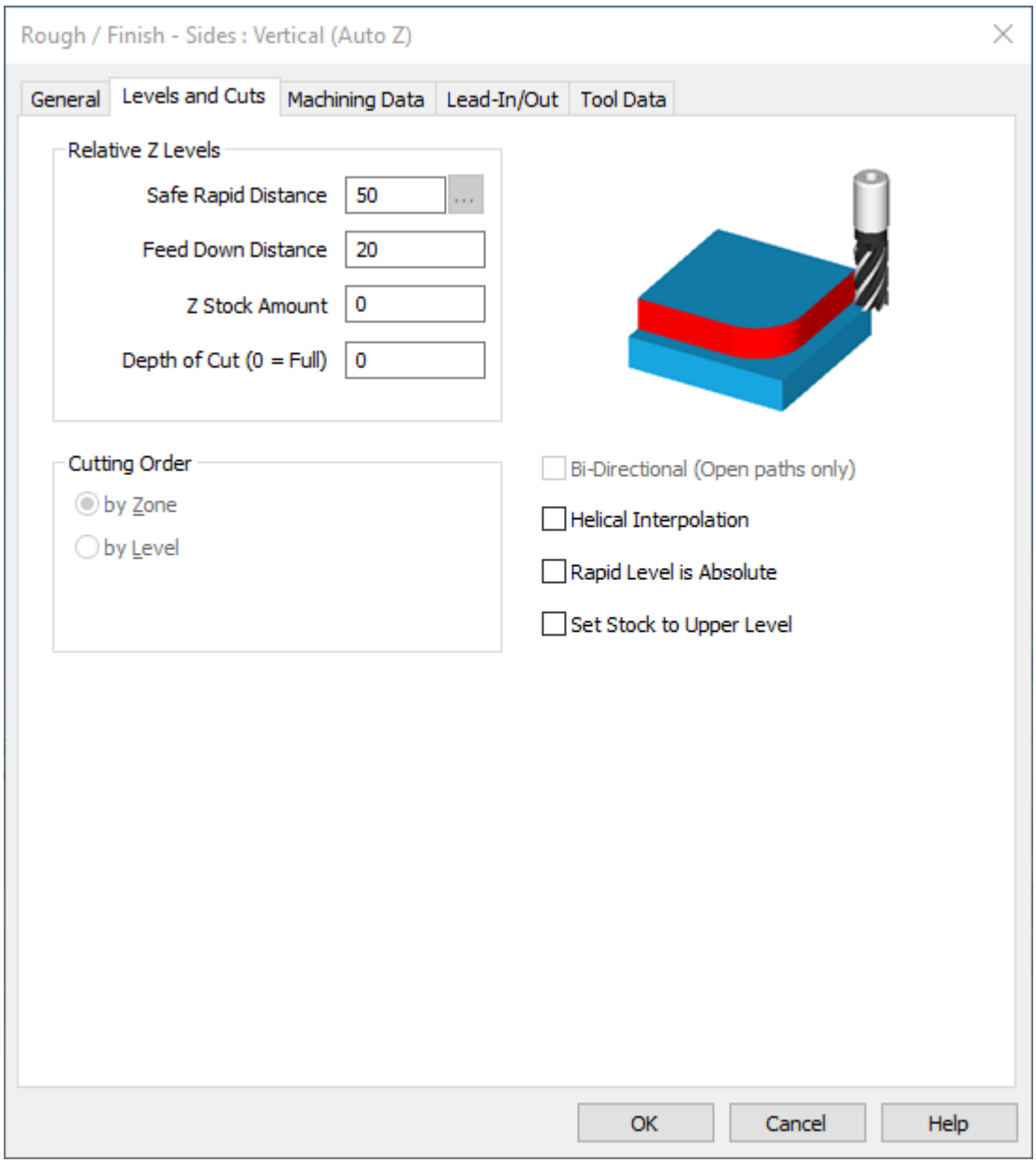



**Figure 57 - Rough and Finish Levels and Cuts tab**

Levels and Cuts sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid, Feed Down Distance** as appropriate, **Z Stock Amount = 0, Depth of cut (0 = Full) = 0**.

## Machining Data

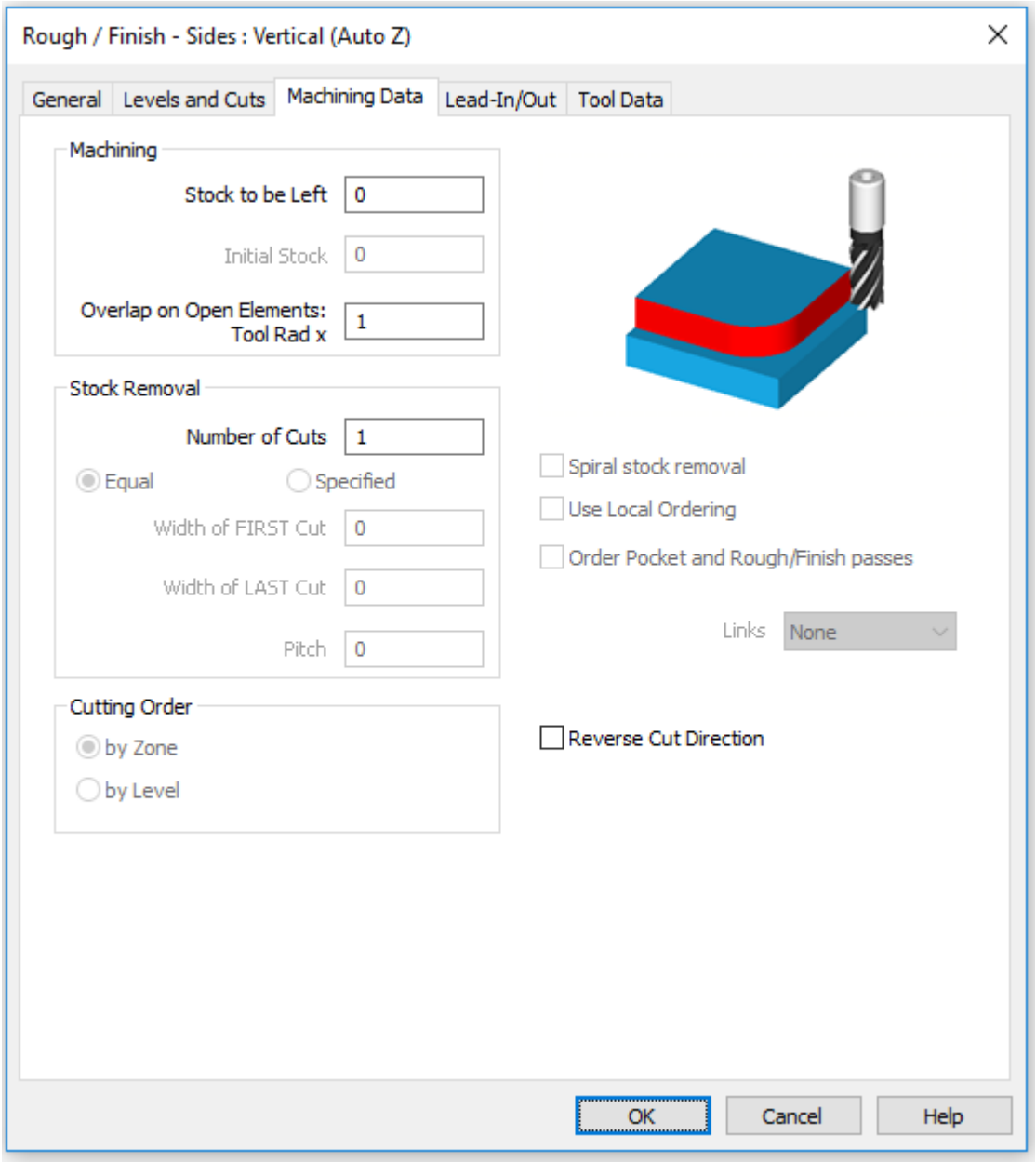



**Figure 58 - Rough and Finish Machining Data tab**

Machining Data sets the X&Y cutting options, **Stock to be Left = 0, Number of Cuts =1**.

## Lead-In/out

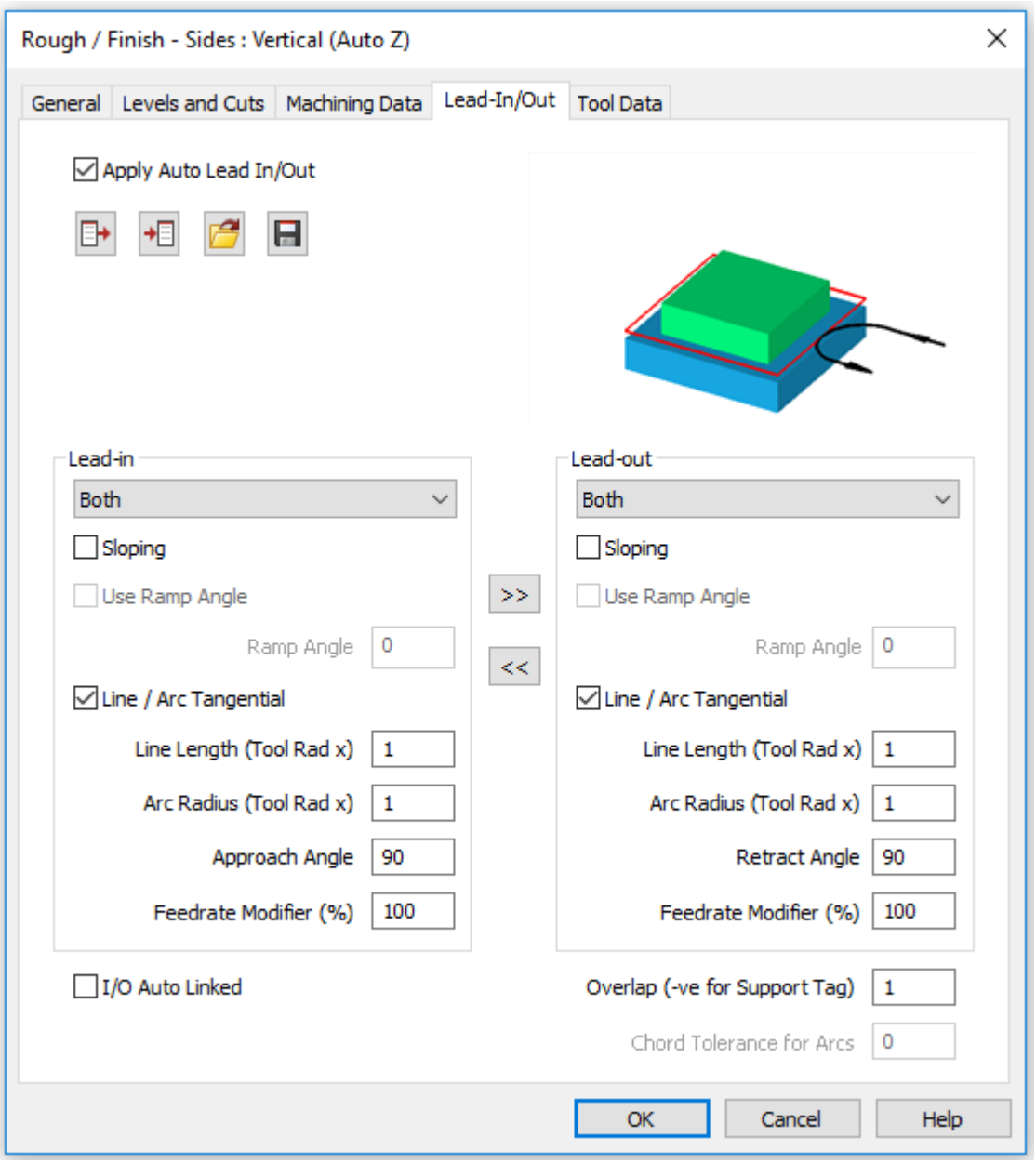



**Figure 59 - Rough and Finish Lead-In/Out tab**

Ensure that the option **☑ Apply Auto Lead In/out** is active.

Make the options choices using the drop-down menus and the various dialogue options to suit your own requirements for manufacturing.

Using the **Save Defaults**  option will allow ease of applying the same values during later machining operations.

## Tool Data

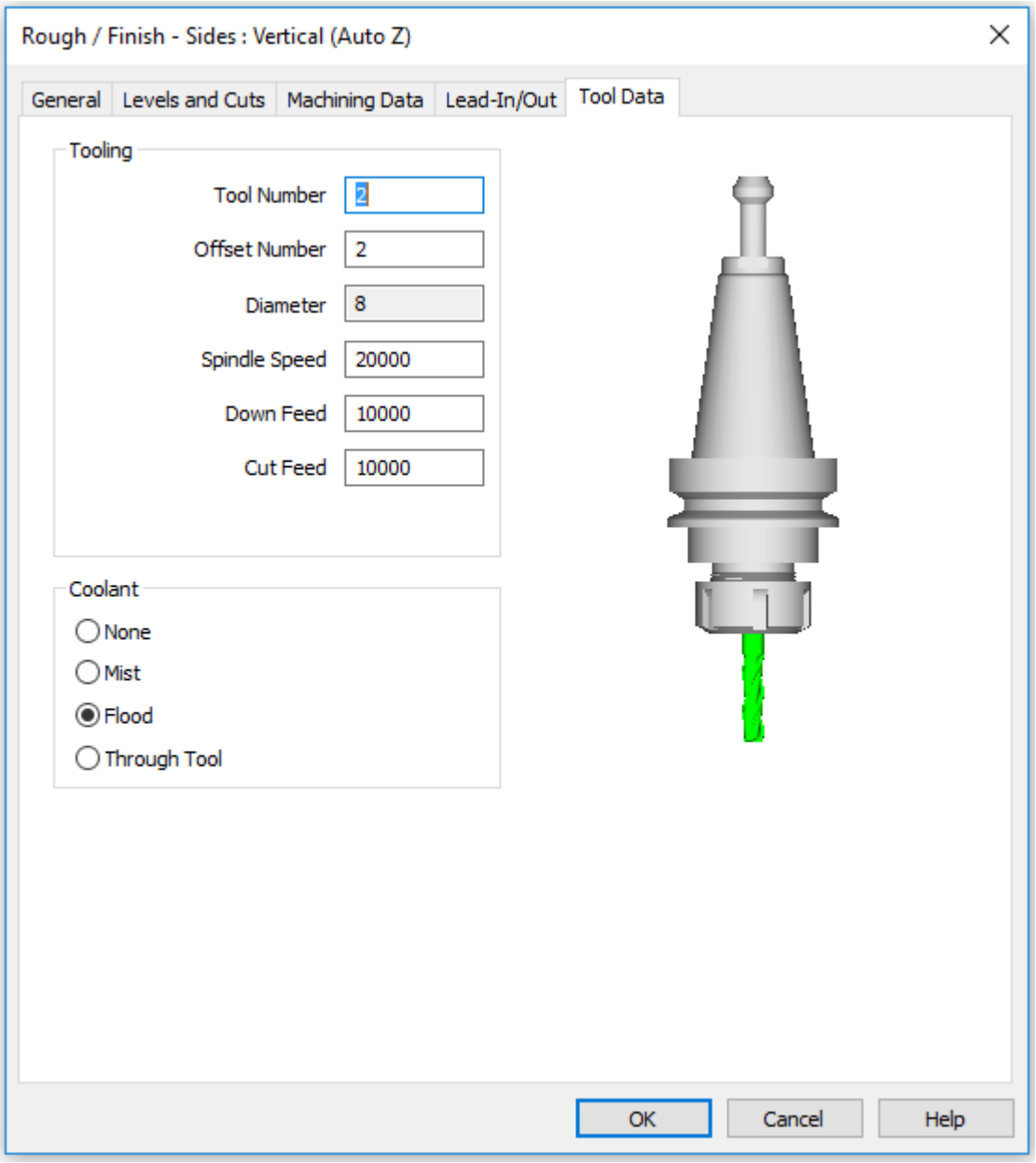



**Figure 60 - Rough and Finish Tool Data tab**

Tool Data sets the tooling speed and feed rate options.

Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select Profiles as shown below.

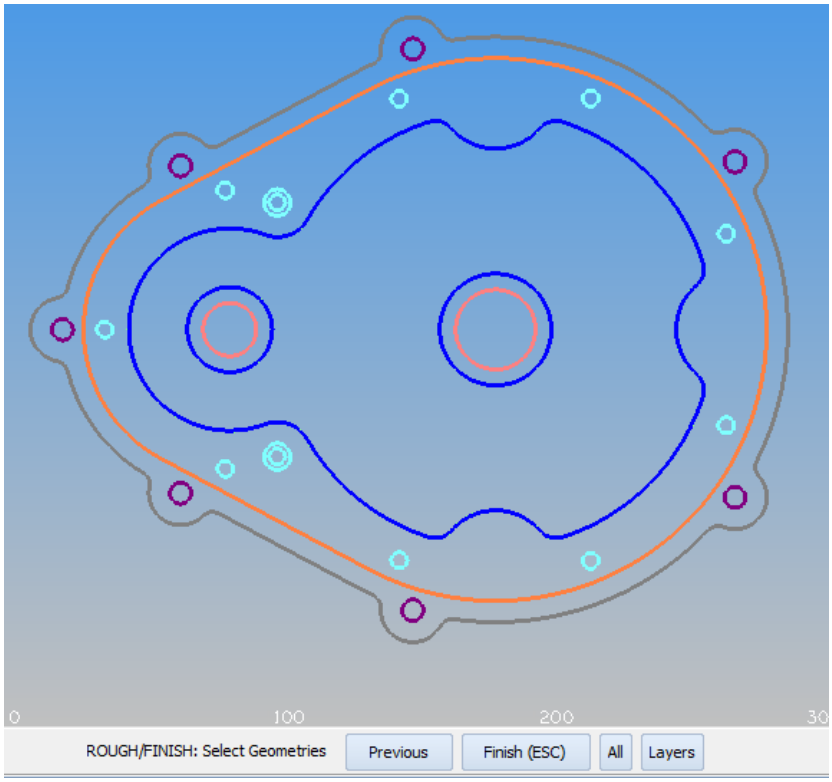



**Figure 61 - Geometries selected for machining**

**<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

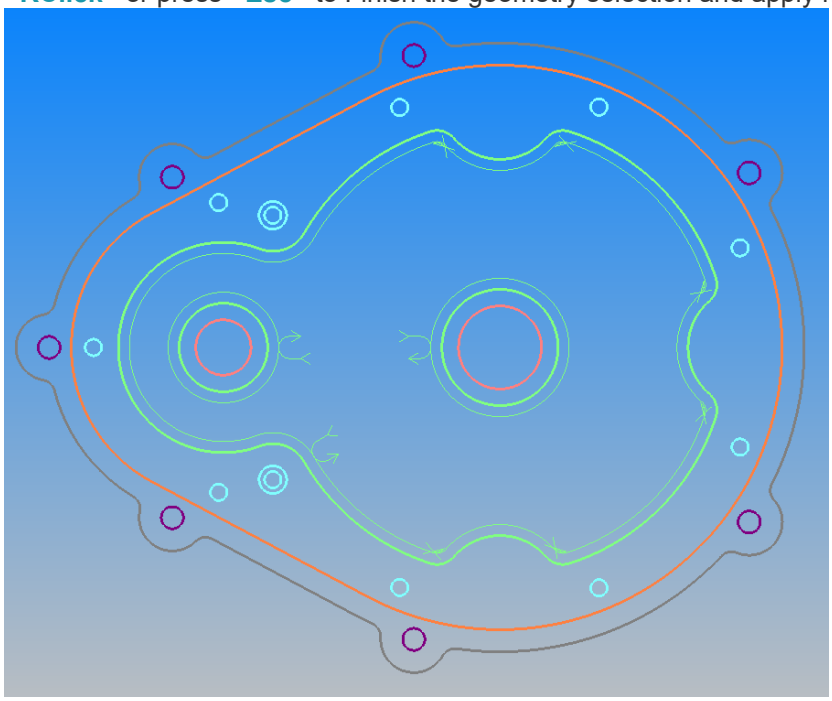


**Figure 62 - Applied toolpath**

# Finish Profile items 9 & 10

Select **MACHINE > Rough or Finish**

## Types

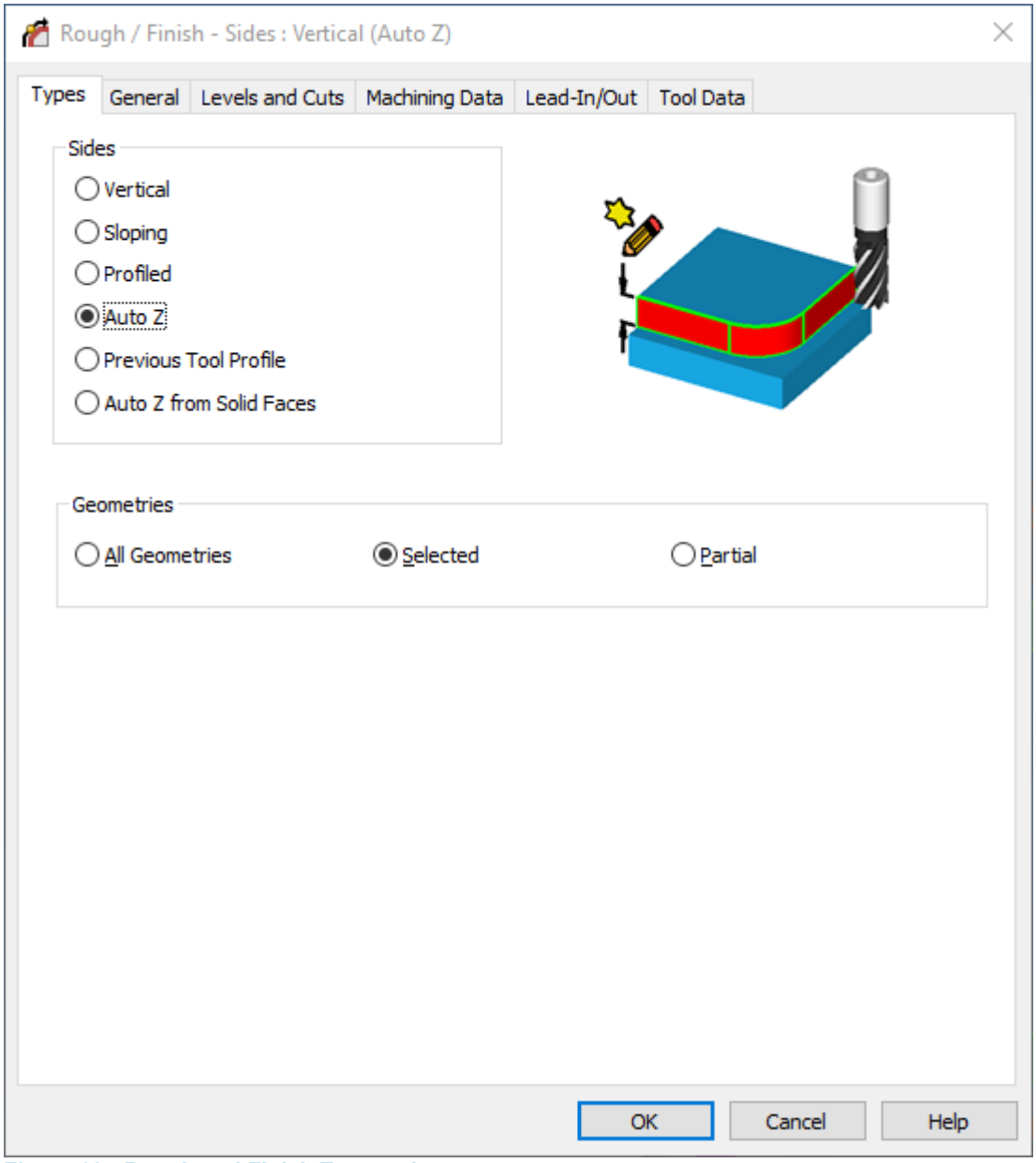



**Figure 63 - Rough and Finish Types tab**

Types sets the sides and geometries.
Set the options **⊙ Auto Z, ⊙ Selected.**

## General

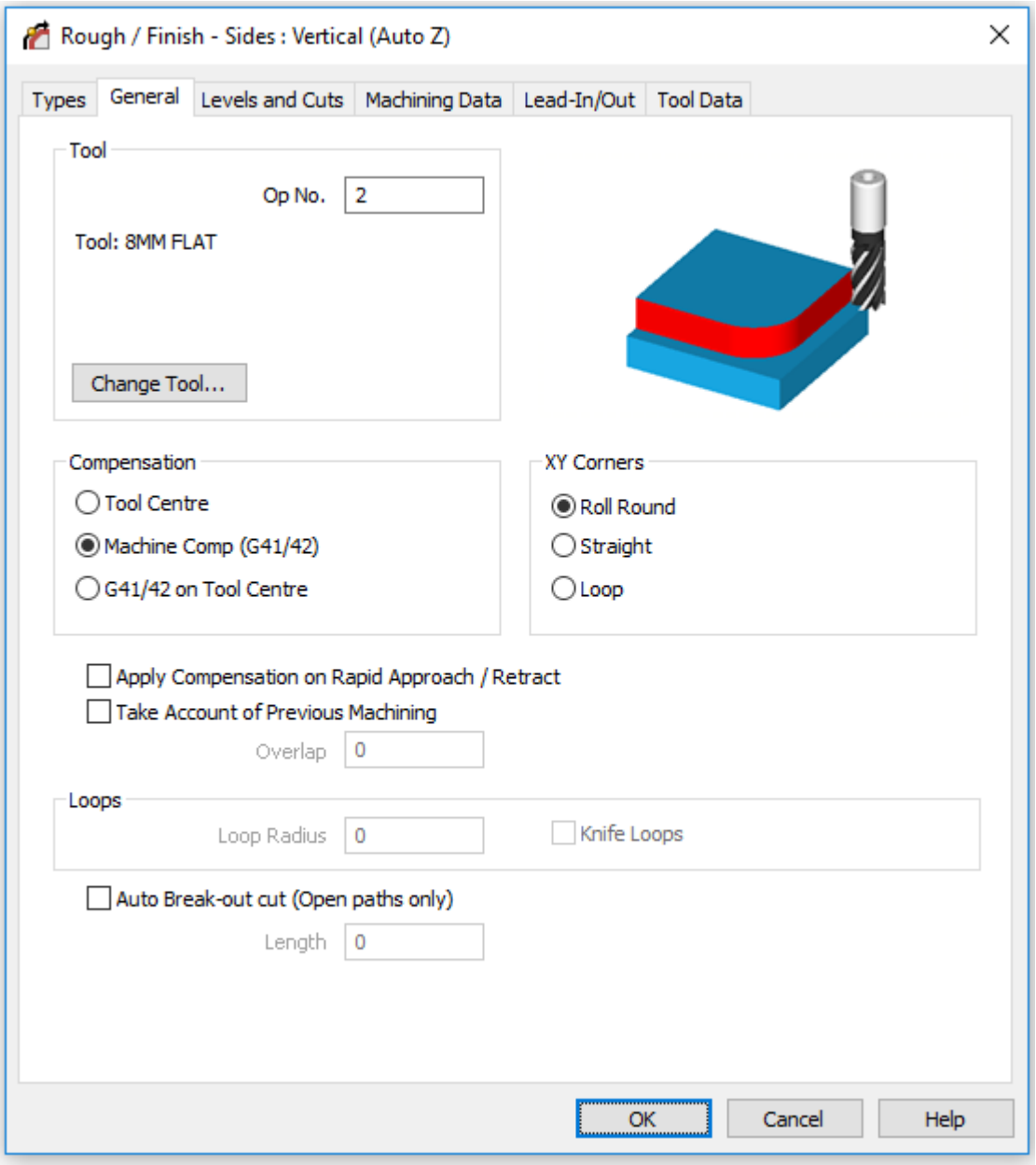



**Figure 64 - Rough and Finish General tab**

General sets the output type.
Set the options ⊙ **Machine Comp (G41/42),** Corners N/A on this job.

Some machines require the option ☑ Apply Compensation on Rapid Approach / Retract to be activated, select this option if it is applicable to your post processor and router. If you select this option and it is not applicable you will be presented with an error Dialogue when listing or outputting the NC file.

## Levels and Cuts

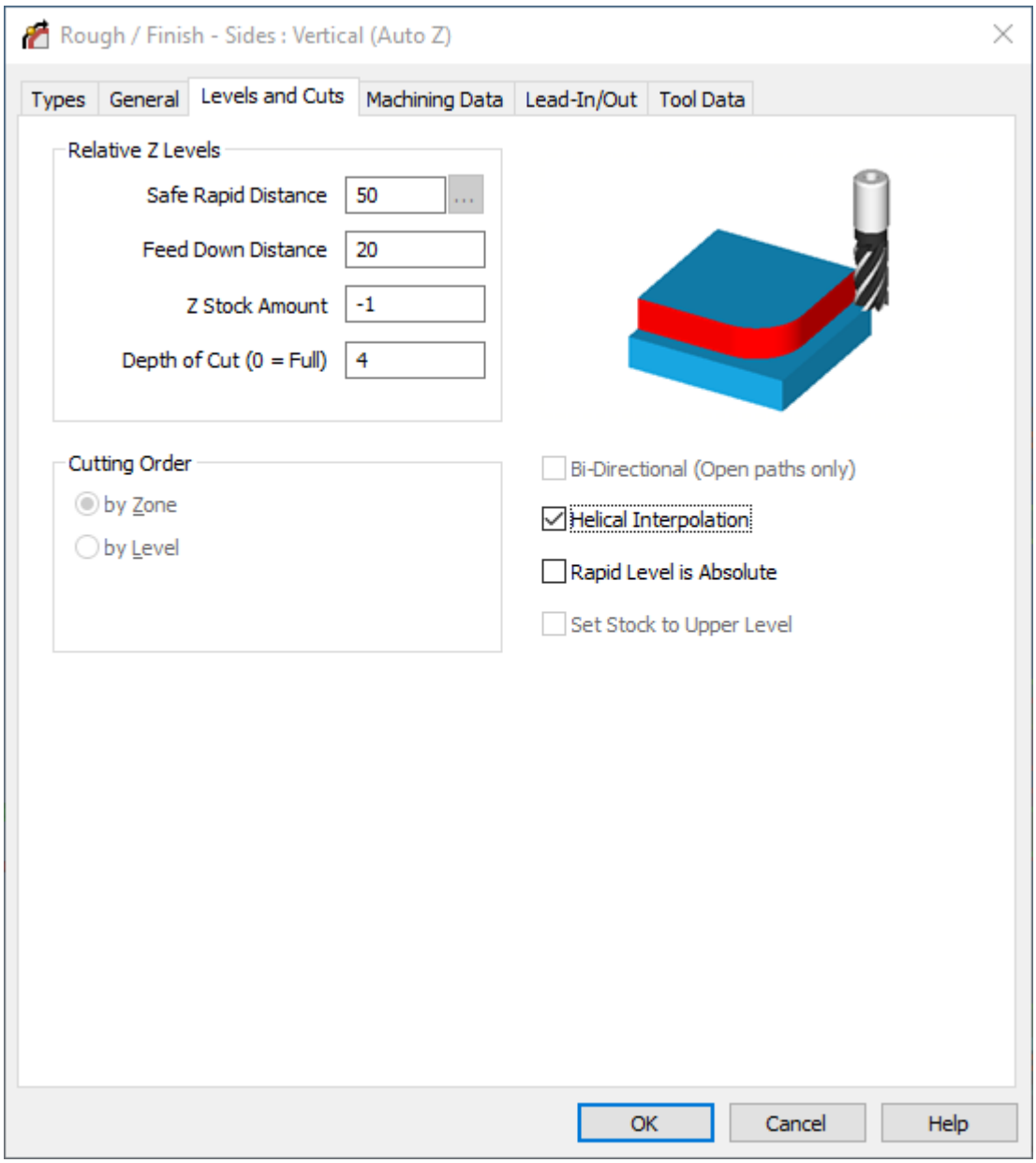



**Figure 65 - Rough and Finish Levels and Cuts tab**

Levels and Cuts sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate,
**Z Stock Amount = -1**,
**☑ Helical Interpolation**,
**Depth of Cut = 4.**

Using the Helical option allows the tool to spiral down in one continuous motion giving a better finish for delicate or precise features. The helical path drops in the Z axis by the amount placed in the Depth of Cut option, for every complete pass around the chosen geometric profile.
At the final depth, the cycle performs on full pass around the profile to remove the slope generated from the helical pass previous.

## Machining Data

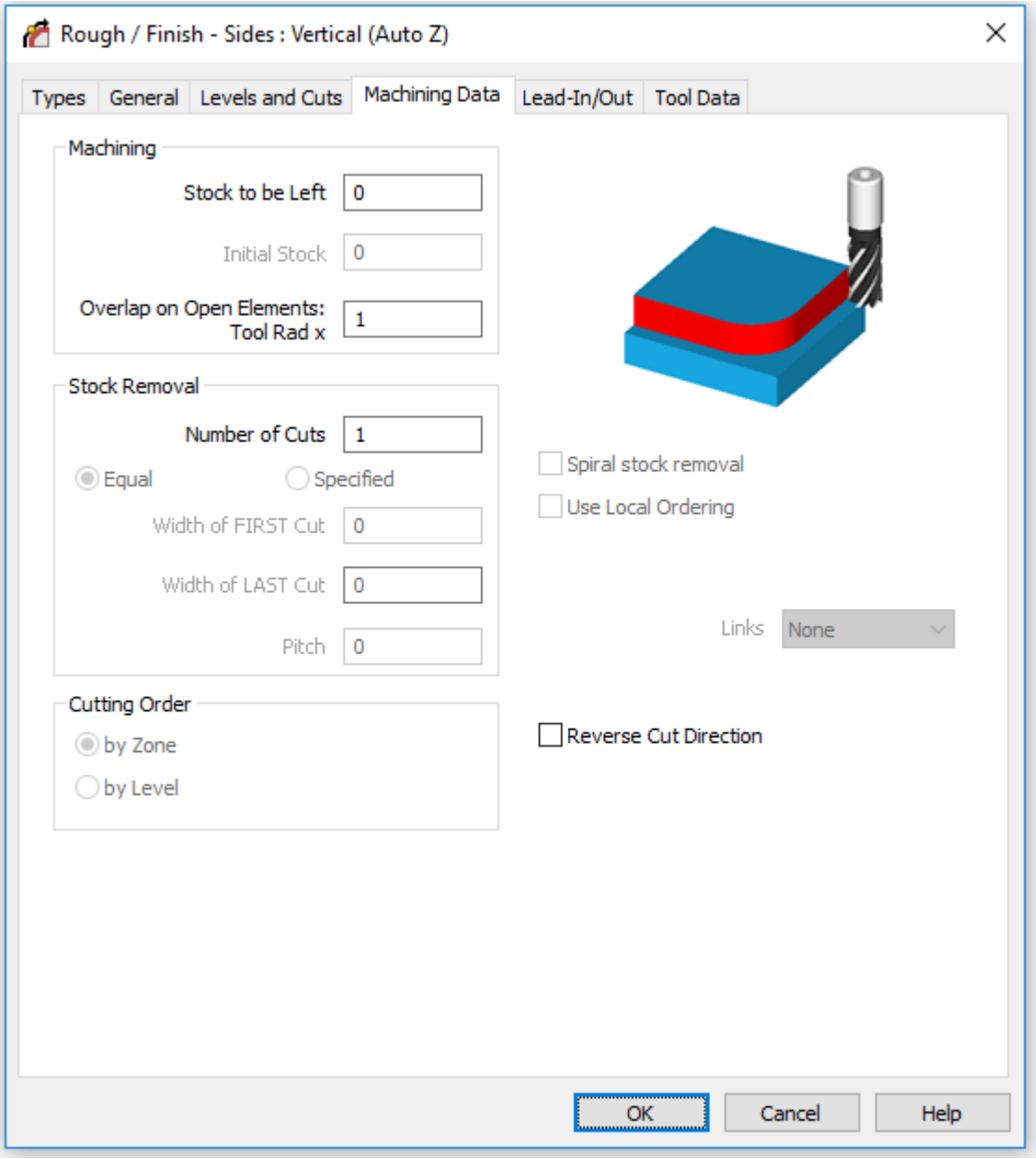



**Figure 66 - Rough and Finish Machining Data tab**

Machining Data sets the X&Y cutting options, **Stock to be Left = 0, Number of Cuts =1.**

## Lead-In/out

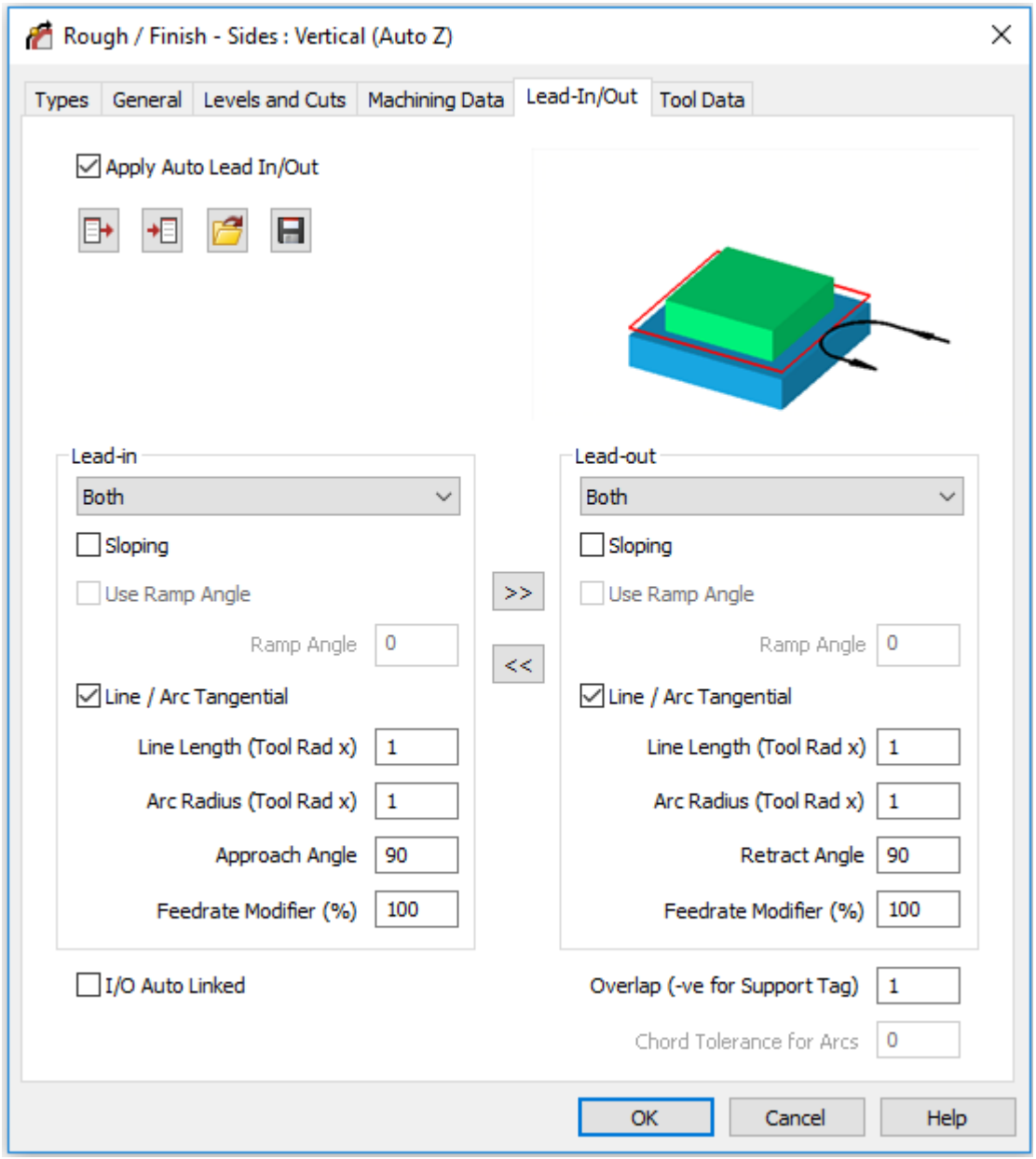



**Figure 67 - Rough and Finish Lead-In/Out tab**

Ensure that the option **☑ Apply Auto Lead In/out** is active.

Make the options choices using the drop-down menus and the various dialogue options to suit your own requirements for manufacturing.

Using the **Save Defaults**  option will allow ease of applying the same values during later machining operations.

## Tool Data

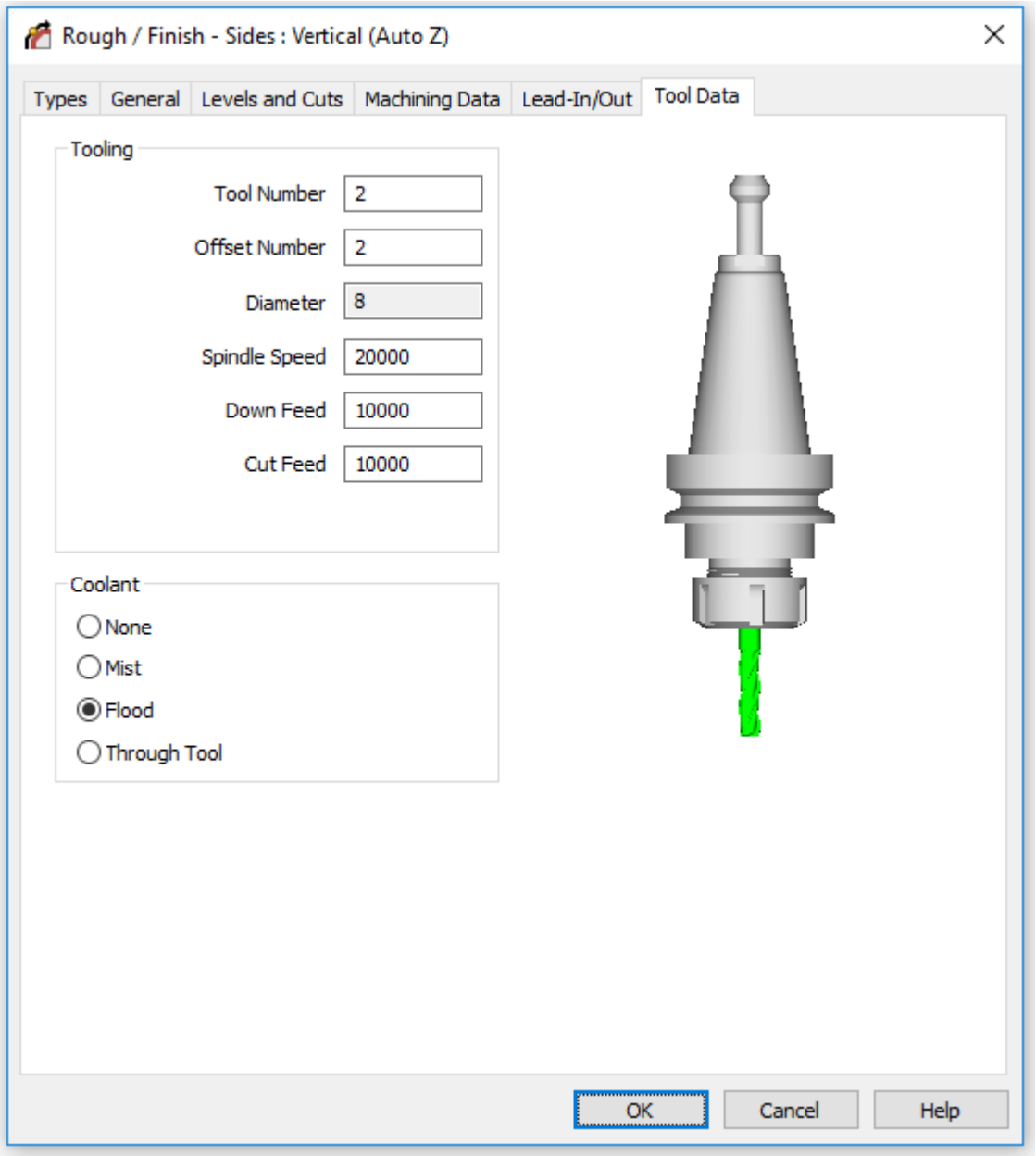



**Figure 68 - Rough and Finish Tool Data tab**

Tool Data sets the tooling speed and feed rate options.

Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select the geometries shown below.

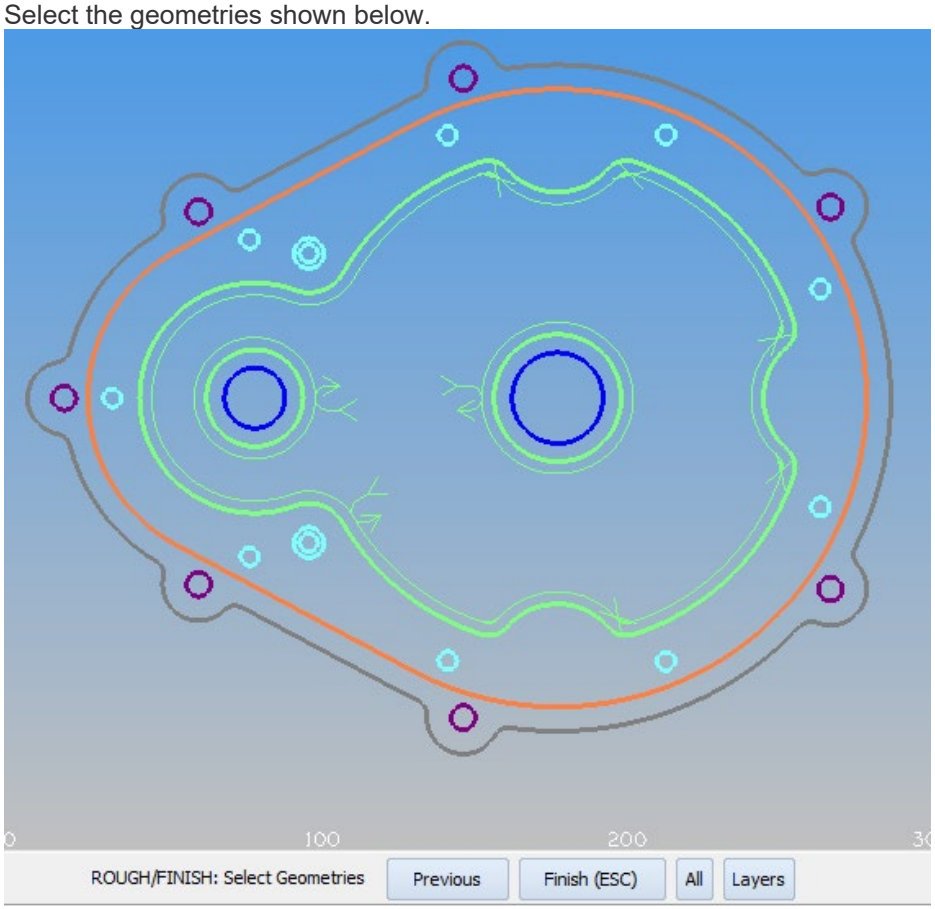



**Figure 69 - Geometry selection**

**<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

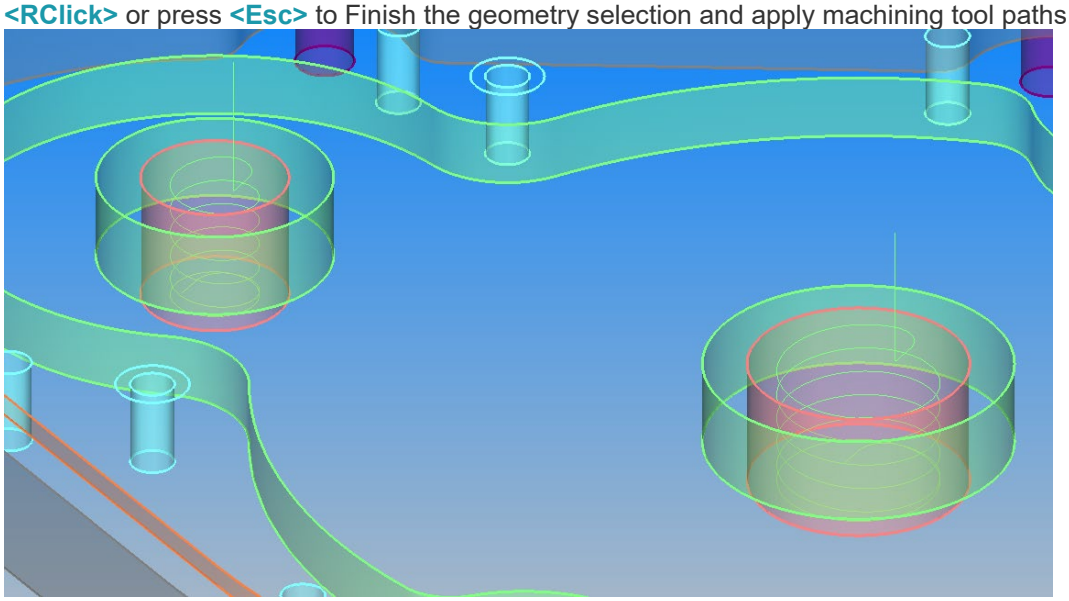


**Figure 70 - ISO view of through holes Helical machined**

# Operation 3

## Spot Chamfer Ø8 Holes 1, at Z 0 producing the 1mm x 30° chamfer

Select **MACHINE > Select Material**

> This is only required for tools that require feed and speed calculations

Select **MACHINE > Select Tool**
Select the **15x120deg Spot Drill** from the **Training** folder.
The tool will be displayed on the screen **<LClick>** or press ↵ to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

Select **MACHINE > Drill/Machine Holes > Drill/Tap Holes Auto Z**

### General

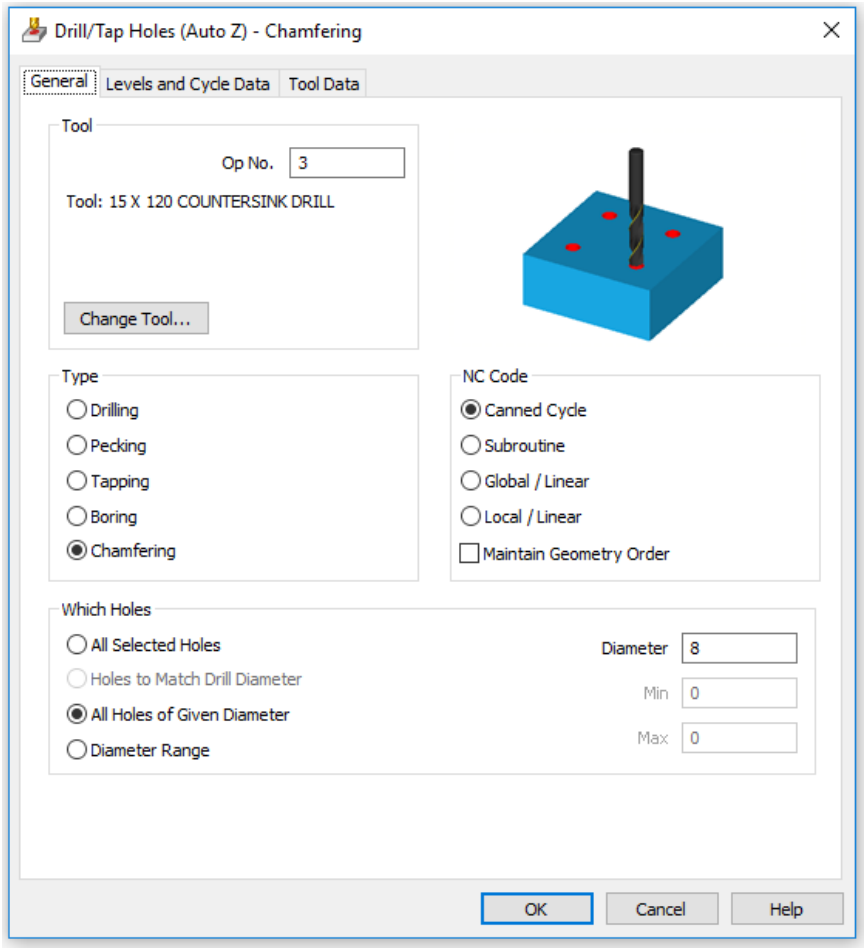



**Figure 71 - Drill/Tap Holes General tab**

General sets the type of cycle and hole selection.
Set the options ⊙ **Chamfer,** ⊙ **All Holes of a Given Diameter Diameter=8**.

## Levels and Cuts

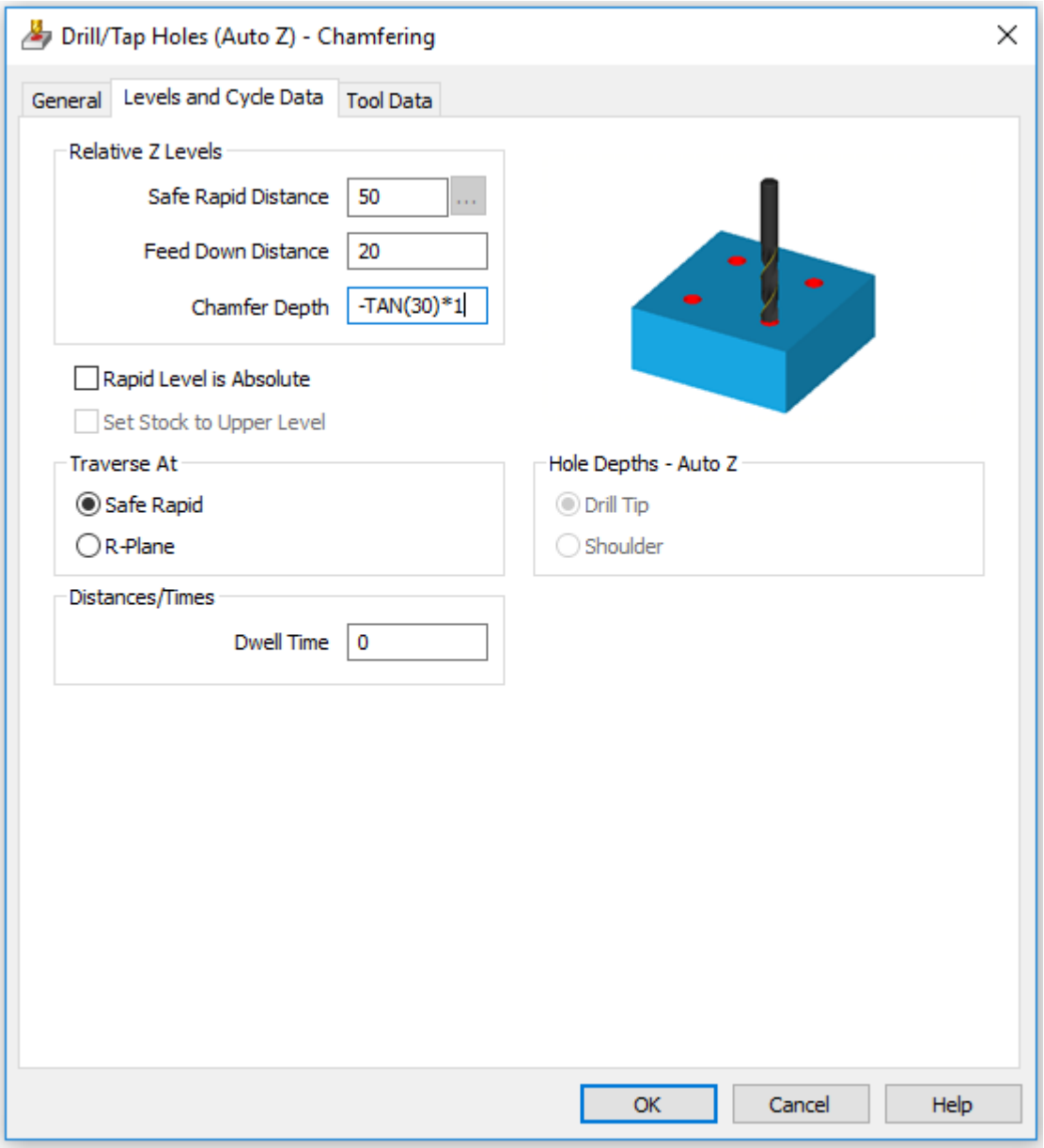



**Figure 72 - Drill/Tap Holes Levels and Cycle Data tab**

Levels and Cycle Data sets Z levels.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate,
**Chamfer Depth = -TAN (30)*1**,
**( Final depth should calculate to -0.5773502 )**
⊙ **Safe Rapid**.

ALPHACAM will calculate to 10 decimal places but will round up to 4 decimal places for the actual required value in the dialogue box.

## Tool Data

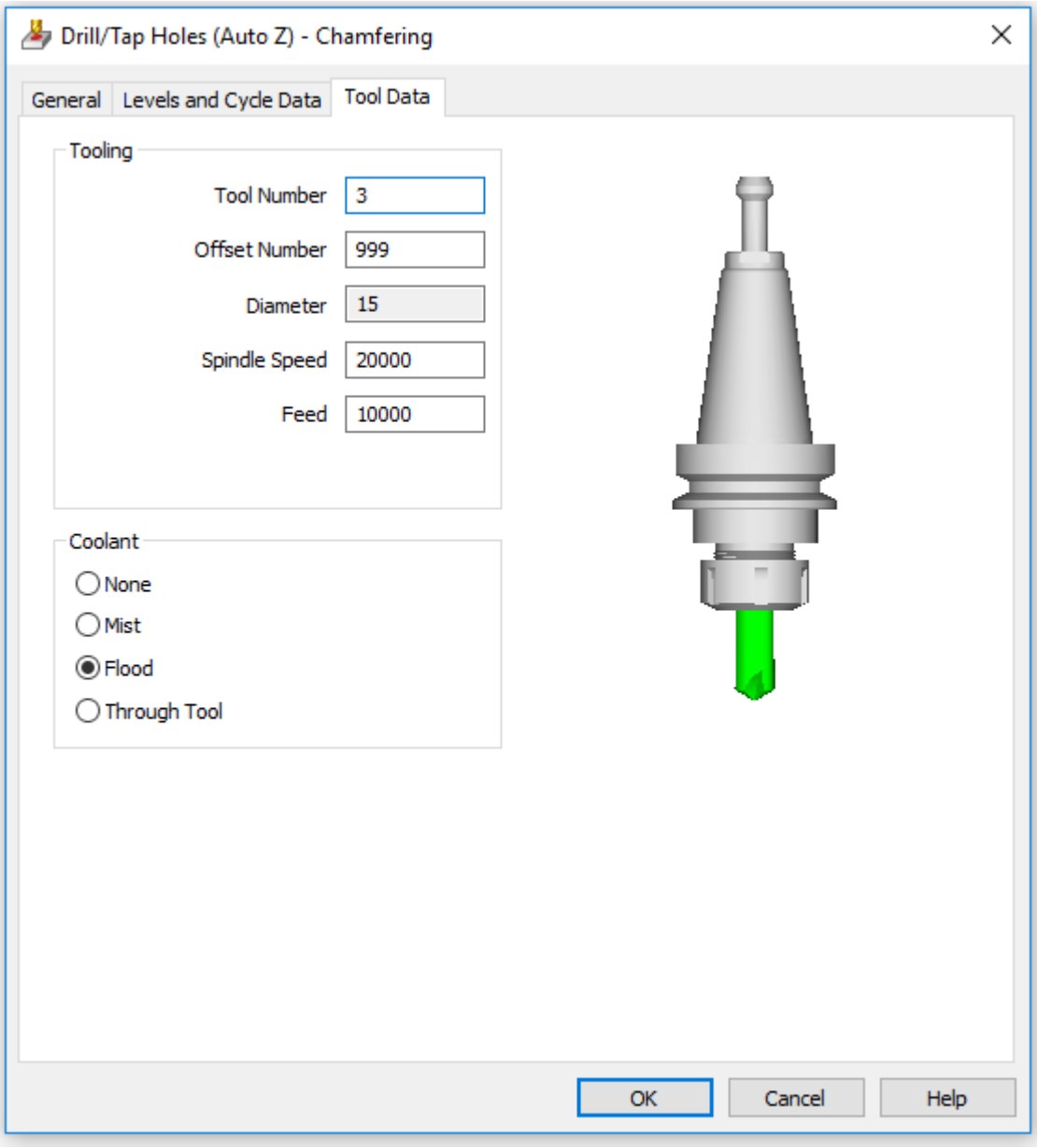



**Figure 73 - Drill/tap Holes Tool Data tab**

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed, Feeds,** and **Coolant** as appropriate, then **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select **[All]** on the prompt line then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

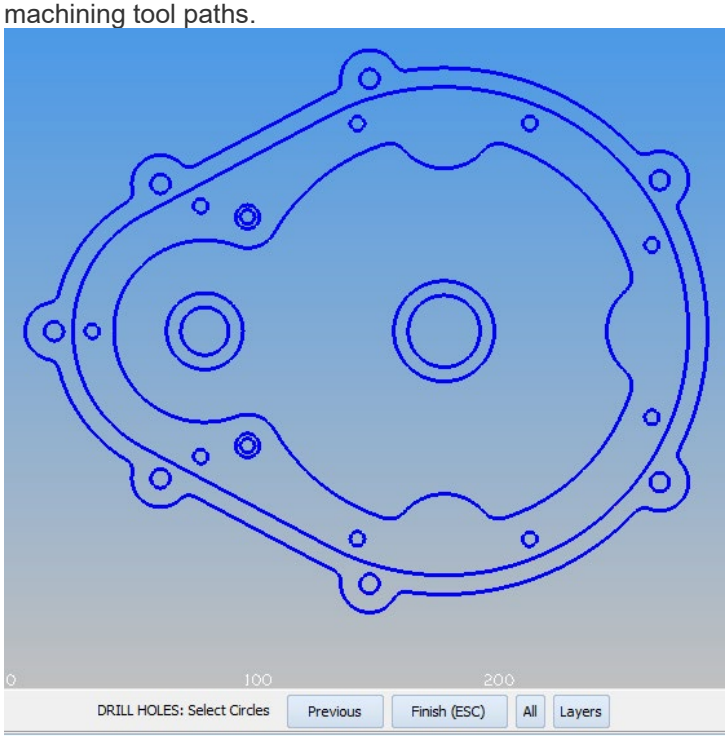



**Figure 74 - Geometry selection for Ø8mm holes**

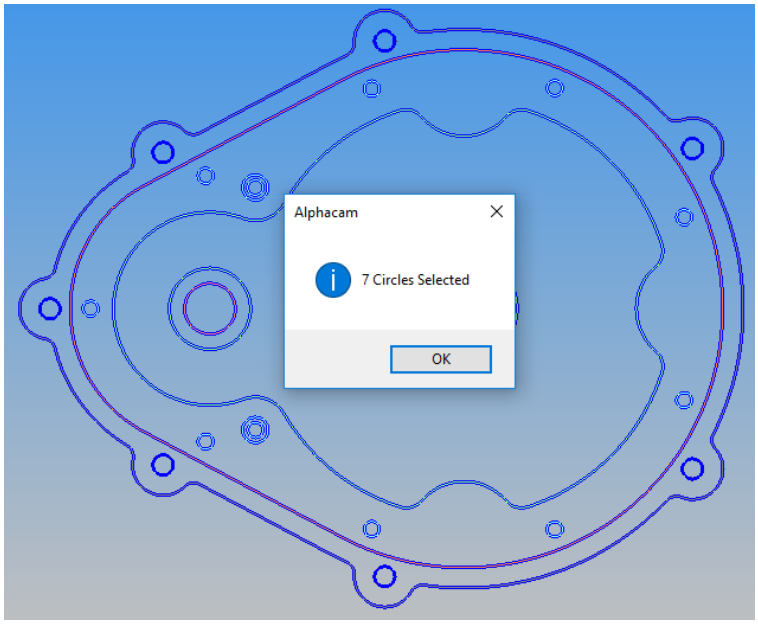



**Figure 75 - Confirmation of the number of holes selected**

A confirmation dialogue will indicate that 7 circles have been selected.

# Spot Chamfer 9off Ø6 Holes 3, at Z -3 producing the 1mm x 30° chamfer

As there are 2off Ø6mm holes that are to have a Ø10mm chamfer, it is necessary to hide these so they will not be selected as part of this command. If the holes were drawn on different layers i.e. through holes and blind holes the selection would be a little easier.

Select **CAD > Hide Parts**

Select the two Ø6mm holes on the R13 fillet centres,

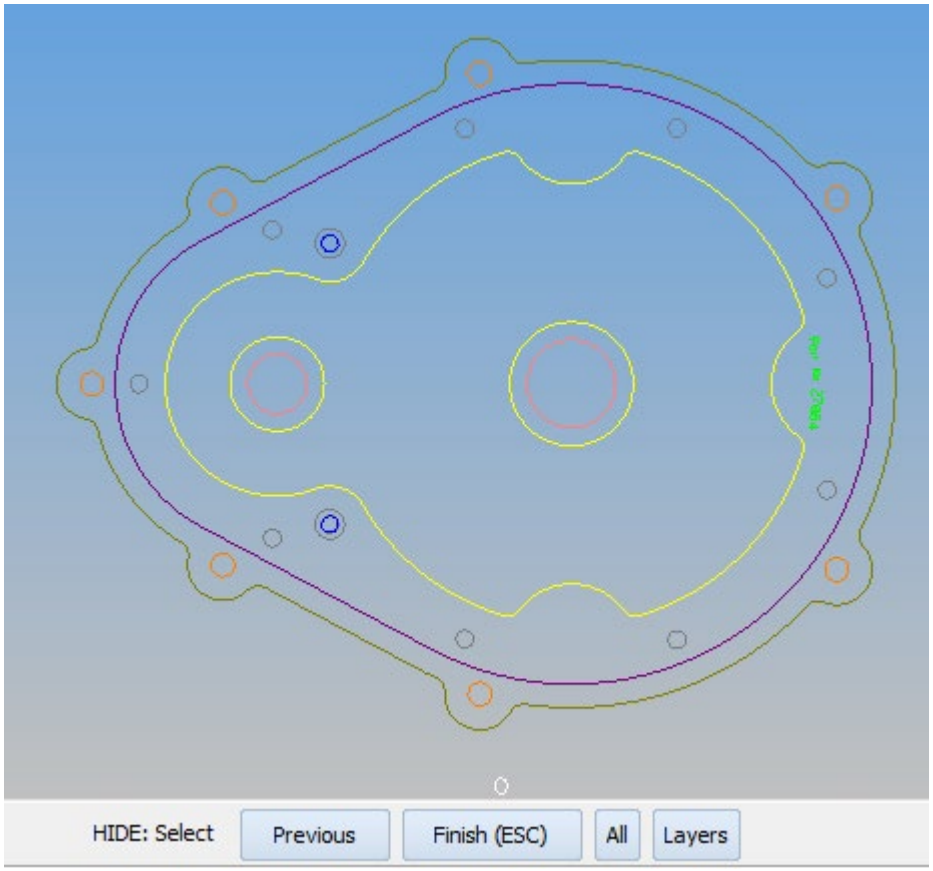



**Figure 76 - Using Hide Parts to turn off these holes**

then **<RClick>** to hide them. The geometries will be unchecked in the **Layers** project manager.

Select **MACHINE > Drill/Machine Holes > Drill/Tap Holes Auto Z** 

## General

General sets the type of cycle and hole selection.
Set the options **⊙ Chamfer, ⊙ All Holes of a Given Diameter Diameter=6**.

## Levels and Cycle Data

Levels and Cycle Data sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate,
**Chamfer Depth = -TAN (30)*1**,
**( Final depth should calculate to -0.5773502 ) ⊙ Safe Rapid.**

## Tool Data

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed, Feeds,** and **Coolant** as appropriate, then **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select **[All]** on the prompt line then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

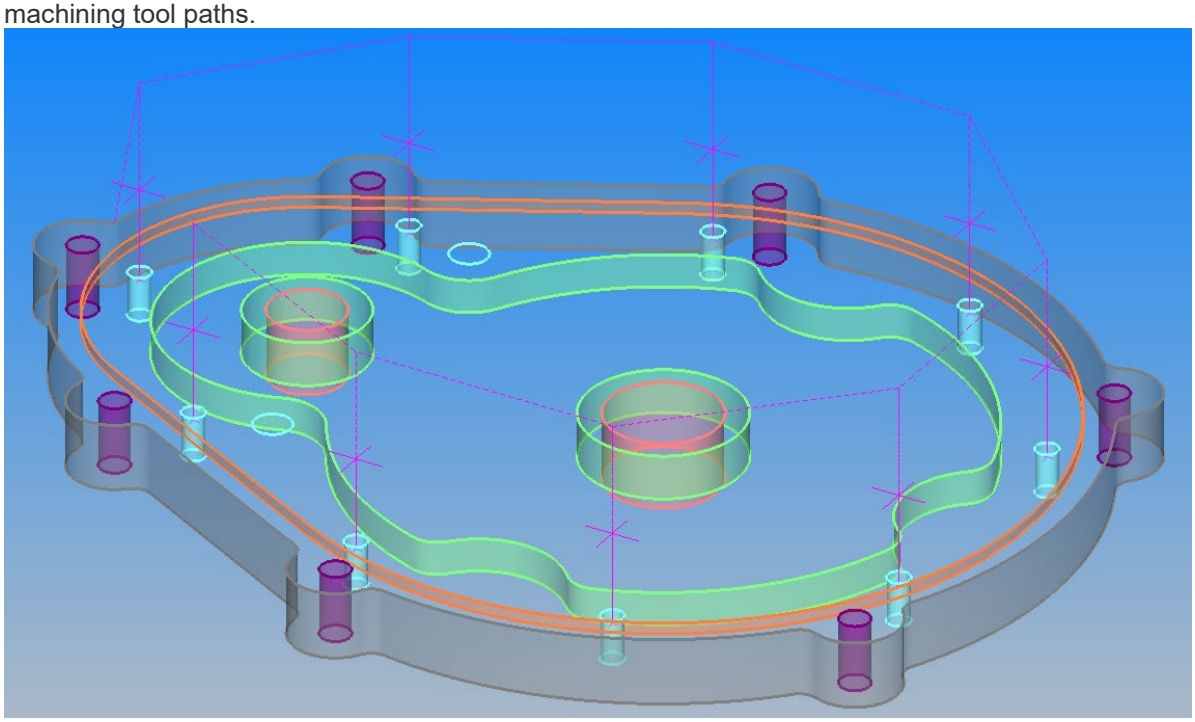


**Figure 77 - 9off Ø6mm holes selected**

A confirmation dialogue box will pop up indicating that 9 holes were selected.

# Spot Chamfer 2off Ø6 Holes 3, at Z -3 producing the 2mm x 30° chamfer

Use **CAD > Show All**  to display the previously hidden circles.

Select **MACHINE > Drill/Machine Holes > Drill/Tap Holes Auto Z** 

## General

General sets the type of cycle and hole selection.
Set the options **⊙ Chamfer, ⊙ All Holes of a Given Diameter Diameter=10**.

## Levels and Cycle Data

Levels and Cycle Data sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate, **Chamfer Depth = 0**, **⊙Safe Rapid**.

## Tool Data

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed, Feeds,** and **Coolant** as appropriate, then **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select **[All]** on the prompt line then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.
A popup info Dialogue is displayed informing you how many geometries were selected.

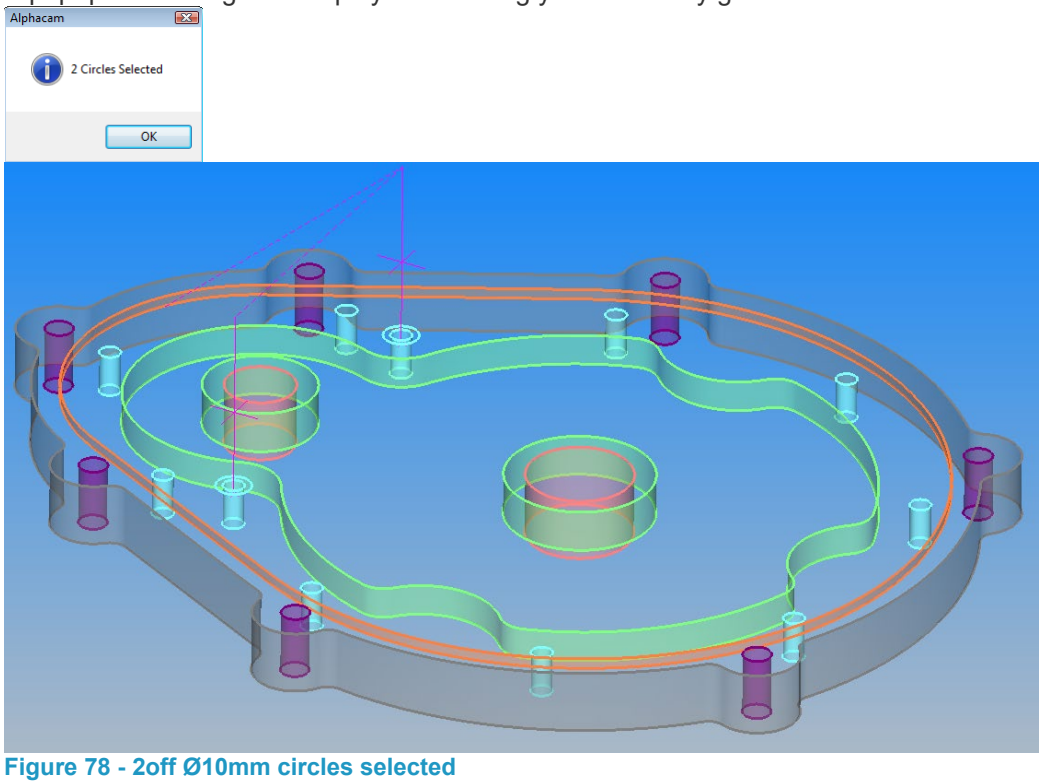



**Figure 78 - 2off Ø10mm circles selected**

# Operation 4

## Drill Ø8mm Holes 1, Through

To make the screen clearer as above you can hide the existing operations select the **Hide All** button at the top of the operations project manager page.
You can also turn on and off individual operations by **<LClick>** on the operation check box.
Select **MACHINE > Select Material**

> This is only required for tools that require feed and speed calculations

Select **MACHINE > Select Tool**
Select the **8mm Twist** from the **Training** folder
The tool will be displayed on the screen **<LClick>** or press ↵ to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

Select **MACHINE > Drill/Machine Holes > Drill/Tap Holes Auto Z**

### General

General sets the type of cycle and hole selection.

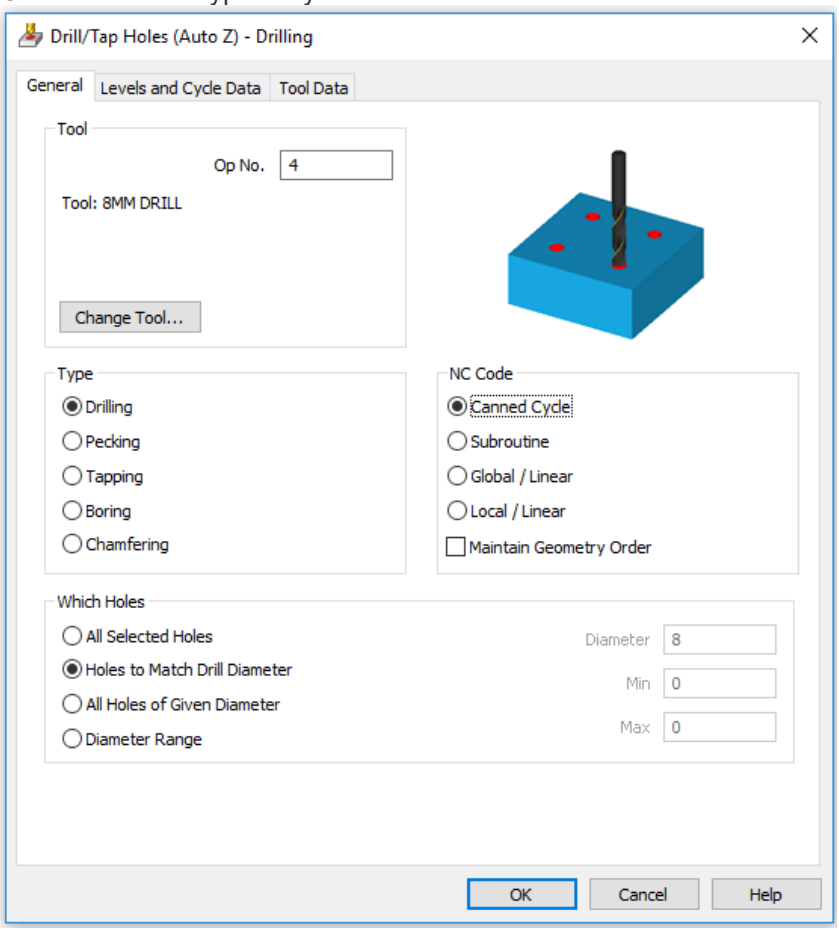



**Figure 79 - Drilling General tab**

Set the options **⊙ Drilling, ⊙ Holes to Match Drill Diameter**.

## Levels and Cycle Data

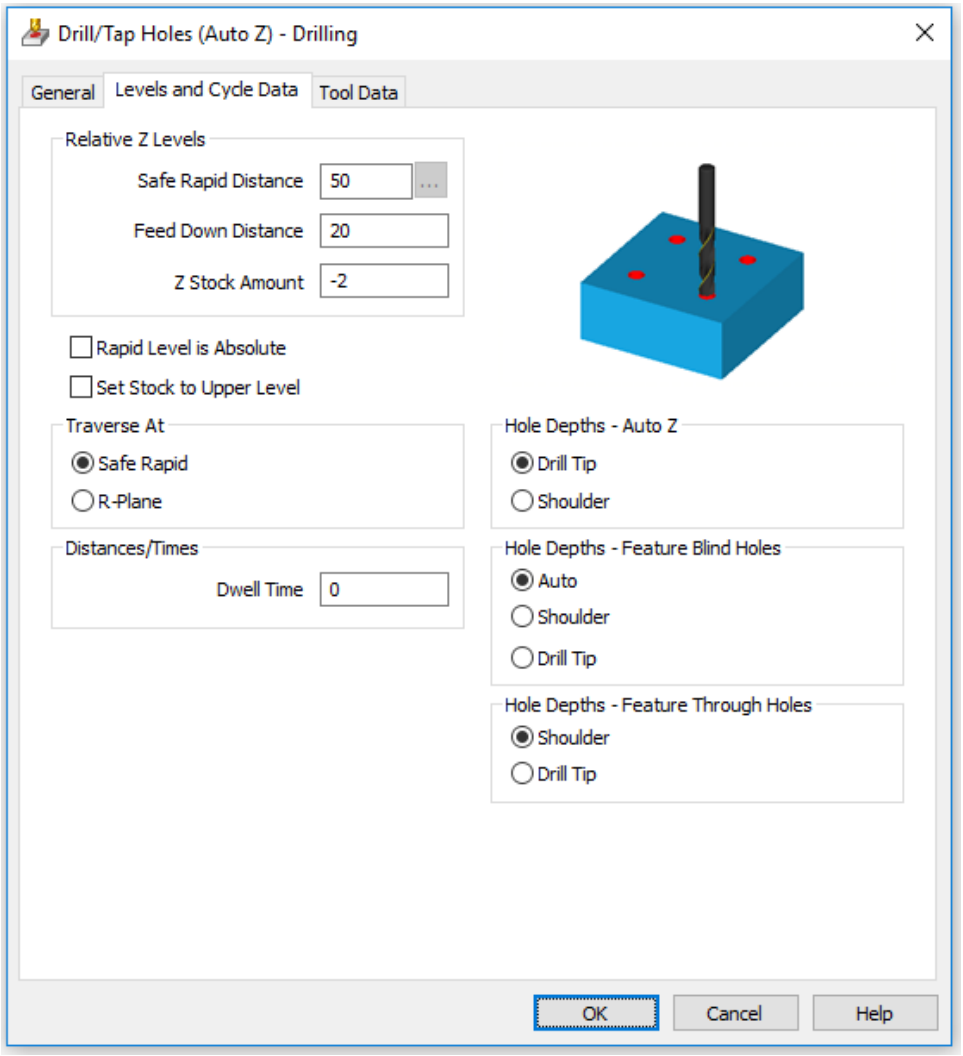



**Figure 80 - Drill/Tap Holes Auto Z Levels and Cycle data tab**

Levels and Cycle Data sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate, **Z Stock Amount = -2**, **⊙ Safe Rapid.**

**Hole Depths – Auto Z ⊙ Drill Tip.** This controls all the drilling levels.

**Hole Depths – Feature Blind Holes ⊙ Auto.** This automatically compares the actual defined drill angle against a Feature Extracted geometry and applies the best option.

**Hole Depths – Feature Through Holes ⊙ Shoulder.** Overrides the main Auto Z option if applicable on Feature Extracted geometry.

## Tool Data

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed, Feeds,** and **Coolant** as appropriate, then **<LClick> [OK]**.

## Select Geometries

The system will then prompt you to select the circles. Because you set the option to only drill holes that match the drill you can select **[All]** (on the prompt line) all the geometries on screen will be highlighted **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.
A popup info Dialogue is displayed informing you how many geometries have been selected.

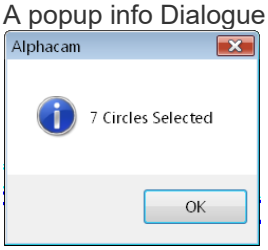



**Figure 81 - Circle confirmation dialogue**

**<LClick> [Ok]**

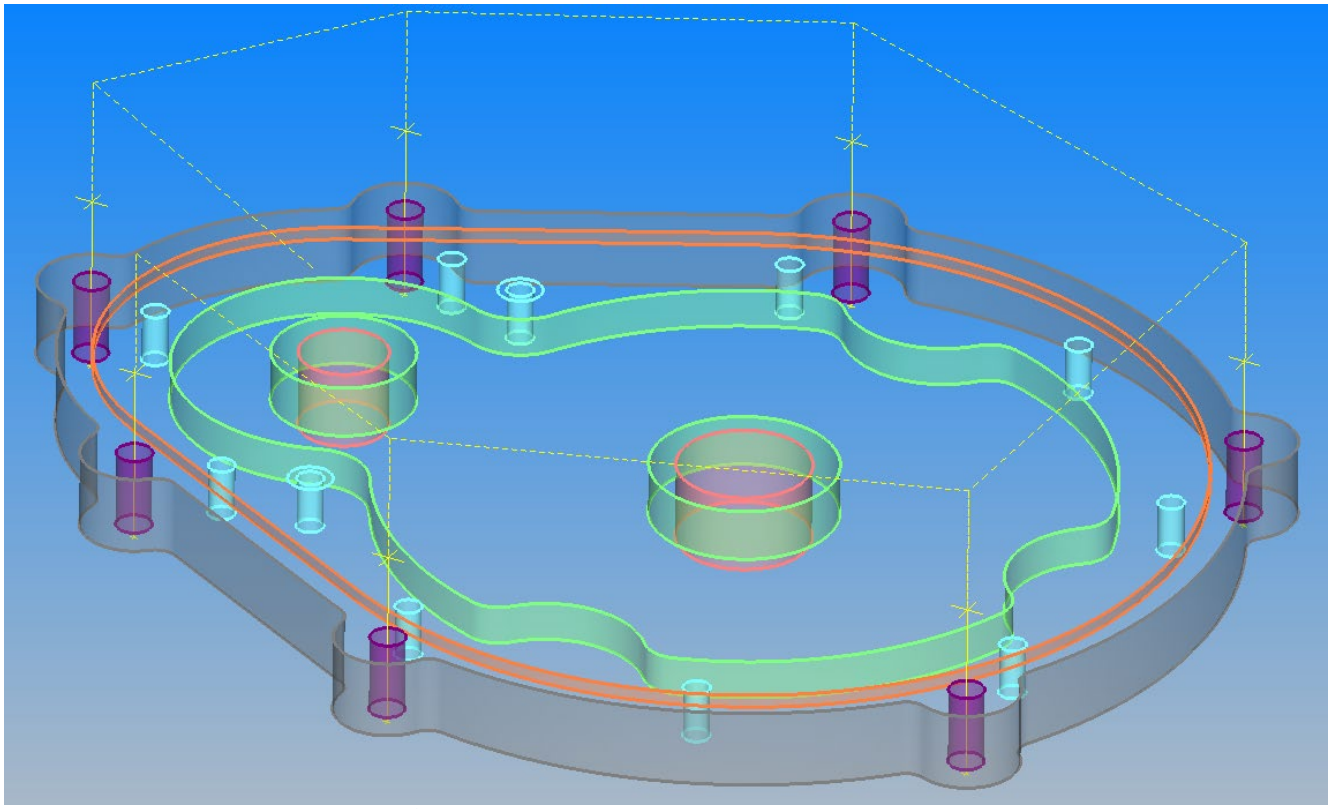


**Figure 82 - Geometry selection for Ø8mm holes**

# Operation 5

## Drill Ø6mm Holes 3, 10mm Deep

Select **MACHINE > Select Material**

> This is only required for tools that require feed and speed calculations

Select **MACHINE > Select Tool**
Select the 6mm Twist from the Training folder
The tool will be displayed on the screen **<LClick>** or press ↵ to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

Select **MACHINE > Drill/Machine Holes > Drill/Tap Holes Auto Z**

### General

General sets the type of cycle and hole selection.
Set the options ⊙ **Drilling,** ⊙ **Holes to Match Drill Diameter**.

### Levels and Cycle Data

Levels and Cycle Data sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate, **Z Stock = 0**,
⊙ **Safe Rapid.**

**Hole Depths – Auto Z** ⊙ **Tip.** This controls all the drilling levels.

**Hole Depths – Feature Blind Holes** ⊙ **Auto.** This automatically compares the actual defined drill angle against a Feature Extracted geometry and applies the best option.

**Hole Depths – Feature Through Holes** ⊙ **Shoulder.** Overrides the main Auto Z option if applicable on Feature Extracted geometry,

### Tool Data

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed, Feeds,** and **Coolant** as appropriate, then **<LClick> [OK]**.

## Select Geometries

The system will then prompt you to select the circles. Because you set the option to only drill holes that match the drill you can select **[All]** (on the prompt line) all the geometries on screen will be highlighted
But the **<LClick>** the pair of Ø6 circles inside the Ø10 circles to de-select them as they are a different depth of hole and do not want to be drilled on this operation. **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.
A popup info Dialogue is displayed informing you how many geometries have been selected.

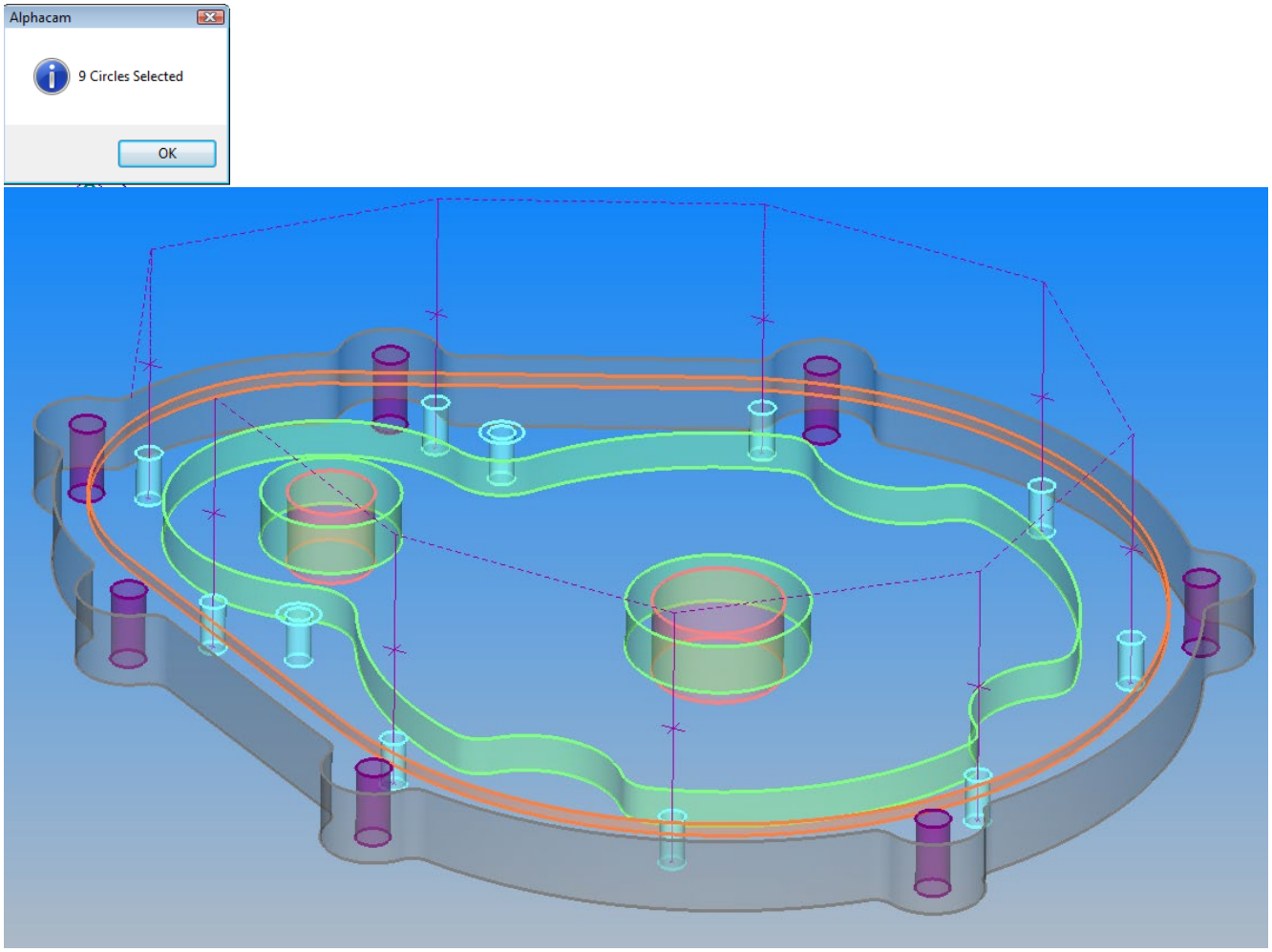



**Figure 83 - Circle confirmation and applied toolpaths**

# Drill Ø6mm Holes 4, 12mm Deep

Before you can action this command, it is necessary to turn back on the two circle geometries previously turned off. This can be done in the Layers project manager page and select the two unchecked geometries under the Geometry layer.

Select **MACHINE > Select Material**

> This is only required for tools that require feed and speed calculations

Select **MACHINE > Select Tool**
Select the **6mm twist** from the **Training** folder.
The tool will be displayed on the screen **<LClick>** or press ↵ to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

## General

General sets the type of cycle and hole selection.
Set the options ⊙ **Drilling,** ⊙ **Holes to Match Drill Diameter**.

## Levels and Cycle Data

Levels and Cycle Data sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate, **Z Stock = 0**, ⊙ **Safe Rapid.**

**Hole Depths – Auto Z** ⊙ **Tip.** This controls all the drilling levels.

**Hole Depths – Feature Blind Holes** ⊙ **Auto.** This automatically compares the actual defined drill angle against a Feature Extracted geometry and applies the best option.

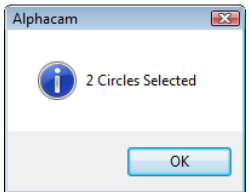
**Hole Depths – Feature Through Holes** ⊙ **Shoulder.** Overrides the main Auto Z option if applicable on Feature Extracted geometry.

## Tool Data

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed, Feeds,** and **Coolant** as appropriate, then **<LClick> [OK]**.

### Select Geometries

The system will then prompt you to select the circles. Select the pair of Ø6mm circles at the centre of the R13 fillet radii then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.
A popup info Dialogue is display informing you how many geometries have been selected.



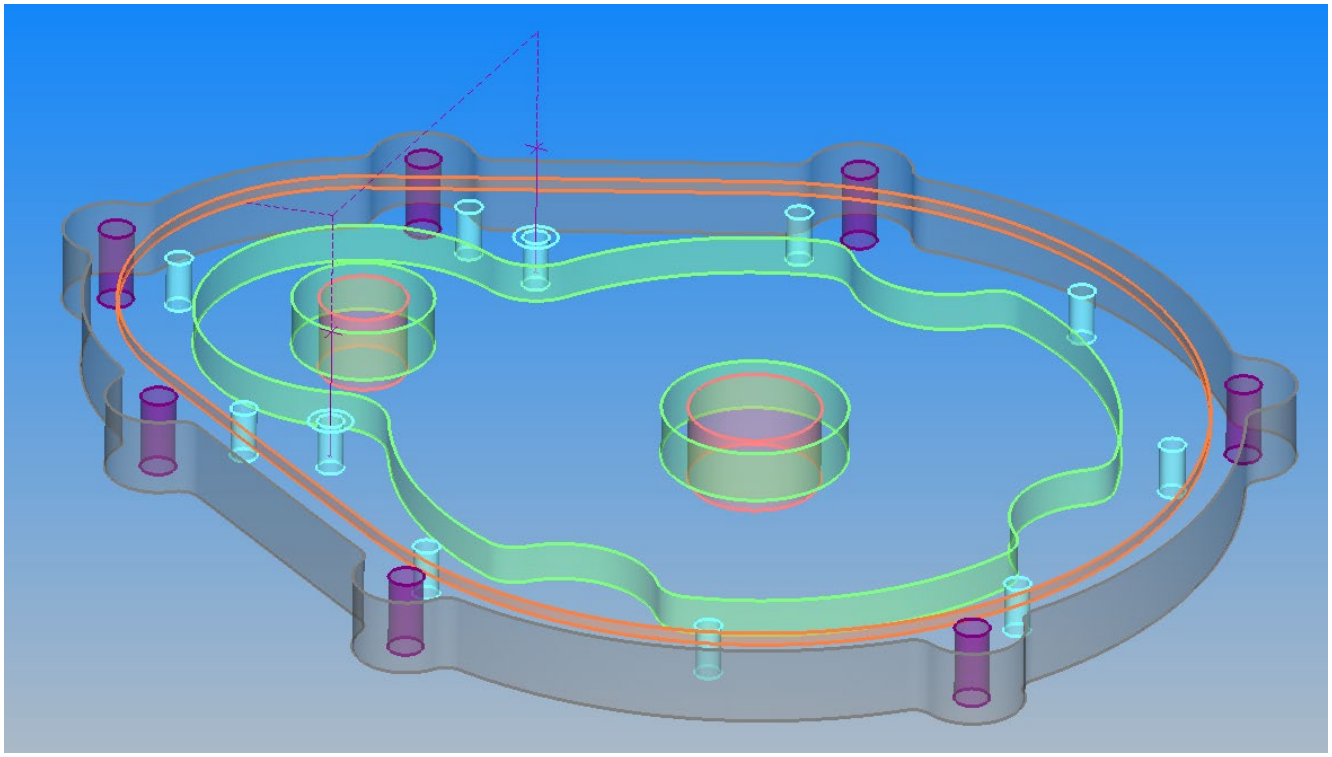



**Figure 84 - Circle confirmation and applied toolpaths**

## Auto Z Drilling tip

It should be noted at this stage that we created two separate drilling operations only for clarity and practice for the different depth holes.

It is perfectly acceptable to include all 11 geometries, in this example, in one cycle. The process will group like depths together and produce NC code to suit, drilling each hole depth in turn.

# Operation 6

## Machine 2mm Radii on top edges, (Profiles 2, 7, 8, 9, 10).

Select **MACHINE > Select Material**

> This is only required for tools that require feed and speed calculations.

Select **MACHINE > Select Tool**
Select the **R2 Corner Rounding cutter from the Training folder.**
The tool will be displayed on the screen **<LClick>** or press ↵ to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

Select **MACHINE > Rough or Finish**.

### Types

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** and ⊙ **Selected**.

### General

General sets the output type. Set the options ⊙ **Machine Comp (G41/42)**.

> Some machines require the option ☑ Apply Compensation on Rapid Approach / Retract to be activated, select this option if it is applicable to your post processor and machine. If you select this option and it is not applicable you will be presented with an error Dialogue when listing or outputting the NC file.

### Levels and Cuts

Levels and Cuts sets Z levels and depth of the cuts in Z.
Set the **Safe Rapid, Rapid Down To** as appropriate, **Z Stock Amount = 0, Depth of Cut (0=Full) = 0**

☑ **Set Stock to Upper Level** is set to active so that the Z Depth control point of this tool, the upper edge of the R2 on the actual tool, will be guided by the top level of the geometry and not the bottom.

> This allows us to machine correctly with User Defined tooling when the guide geometry may not be the lower level option.

## Machining Data

Machining Data sets the cutting information.
**Stock to be Left** = **0**, **Number of Cuts =1**.

## Lead-In/Out

Ensure that the option ☑ **Apply Auto Lead In/out** is active.
Make the options choices using the drop-down menus and the various dialogue options to suit your own requirements for manufacturing.

## Tool Data

Tool Data sets the tooling speed and feed rate options.
Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select the geometries as shown below.

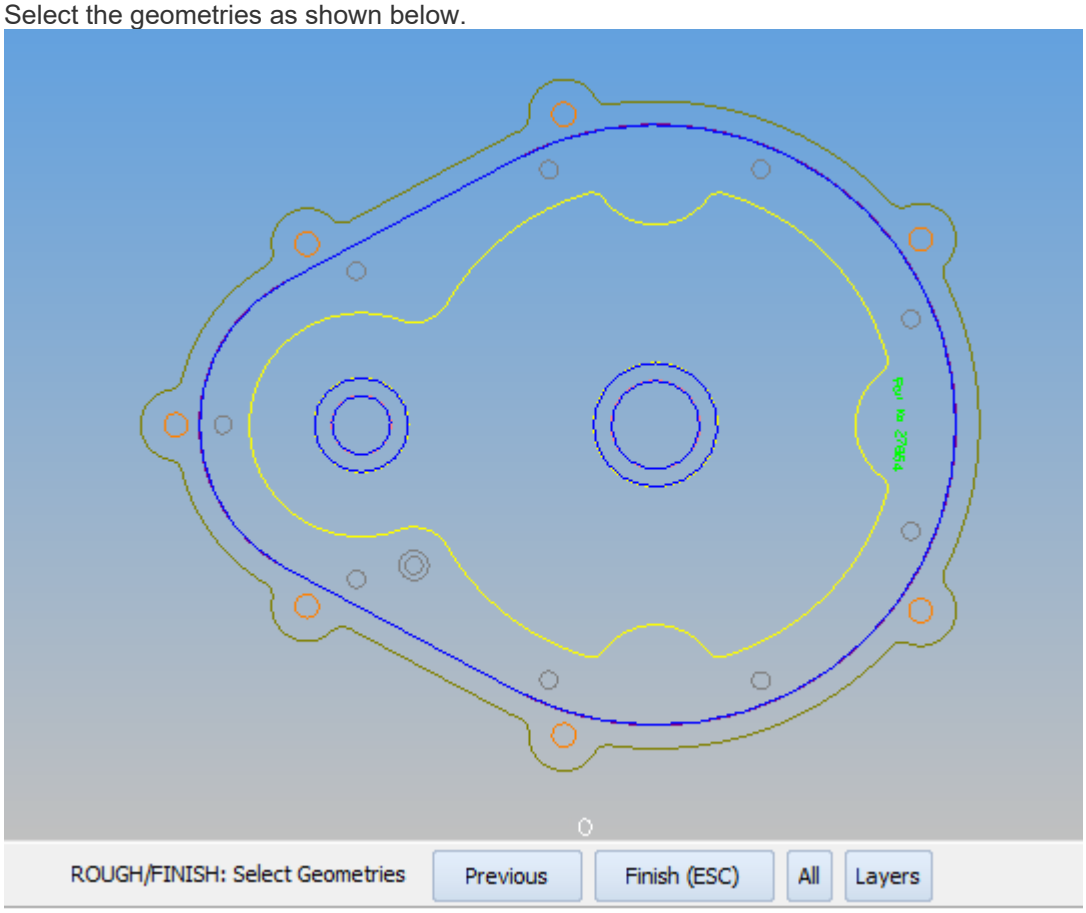



**Figure 85 - Geometry selection**

**<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

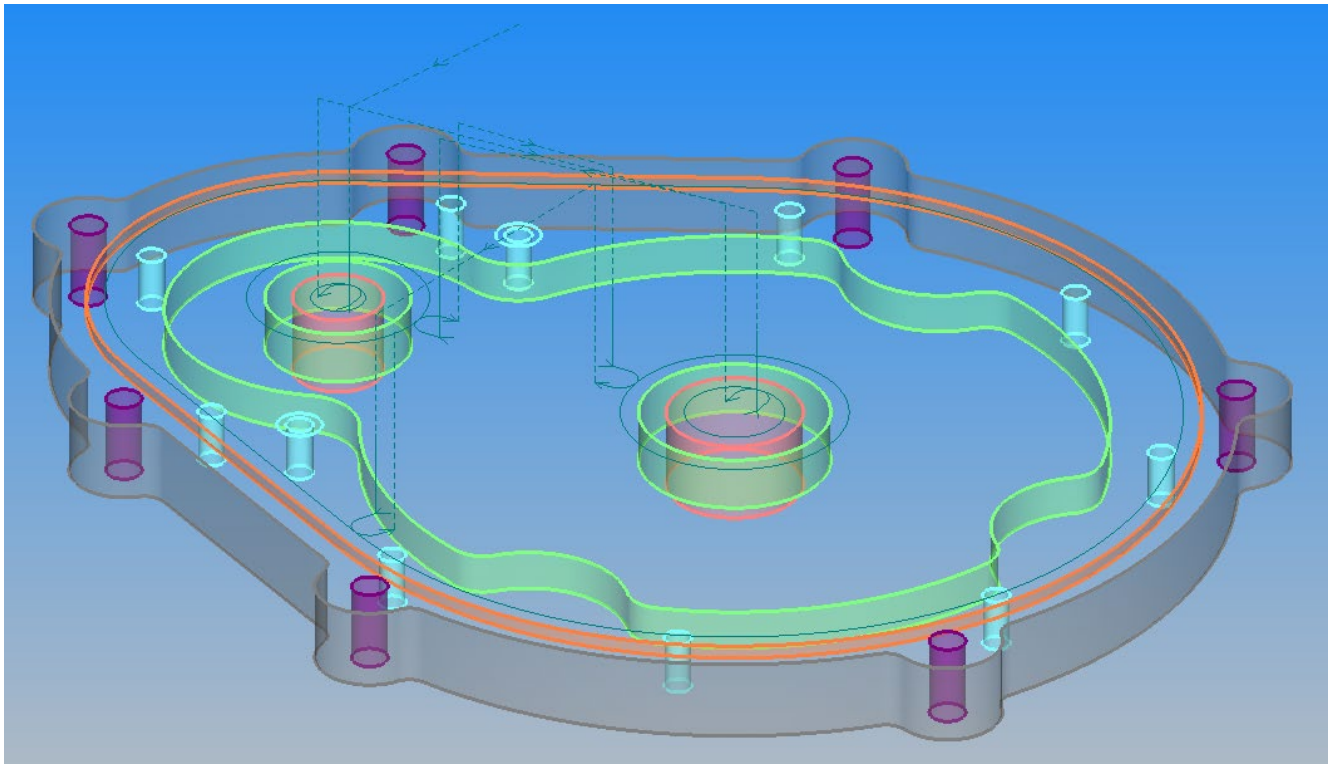


**Figure 86 - Toolpaths applied**

# Machine 2mm Radii on Profile 6 at -3

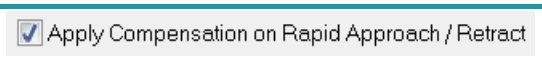
Select **MACHINE > Rough or Finish** .

## Types

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** and ⊙ **Selected**.

## General

General sets the output type. Set the options ⊙ **Machine Comp (G41/42)**.

Some machines require the option 

to be activated, select this option if it is applicable to your post processor and machine. If you select this option and it is not applicable you will be presented with an error Dialogue when listing or outputting the NC file.

## Levels and Cuts

Levels and Cuts sets Z levels and depth of the cuts in Z.
Set the **Safe Rapid, Rapid Down To** as appropriate, **Z Stock Amount = 0, Depth of Cut (0=Full) = 0**

☑ **Set Stock to Upper Level** is set to active so that the Z Depth control point of this tool, the upper edge of the R2 on the actual tool, will be guided by the top level of the geometry and not the bottom.

## Machining Data

Machining Data sets the cutting information.
**Stock to be Left** = **0**, **Number of Cuts =1**.

## Lead-In/Out

Ensure that the option ☑ **Apply Auto Lead In/out** is active.
Make the options choices using the drop-down menus and the various dialogue options to suit your own requirements for manufacturing.

## Tool Data

Tool Data sets the tooling speed and feed rate options.

Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK].**

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select the geometries shown below.

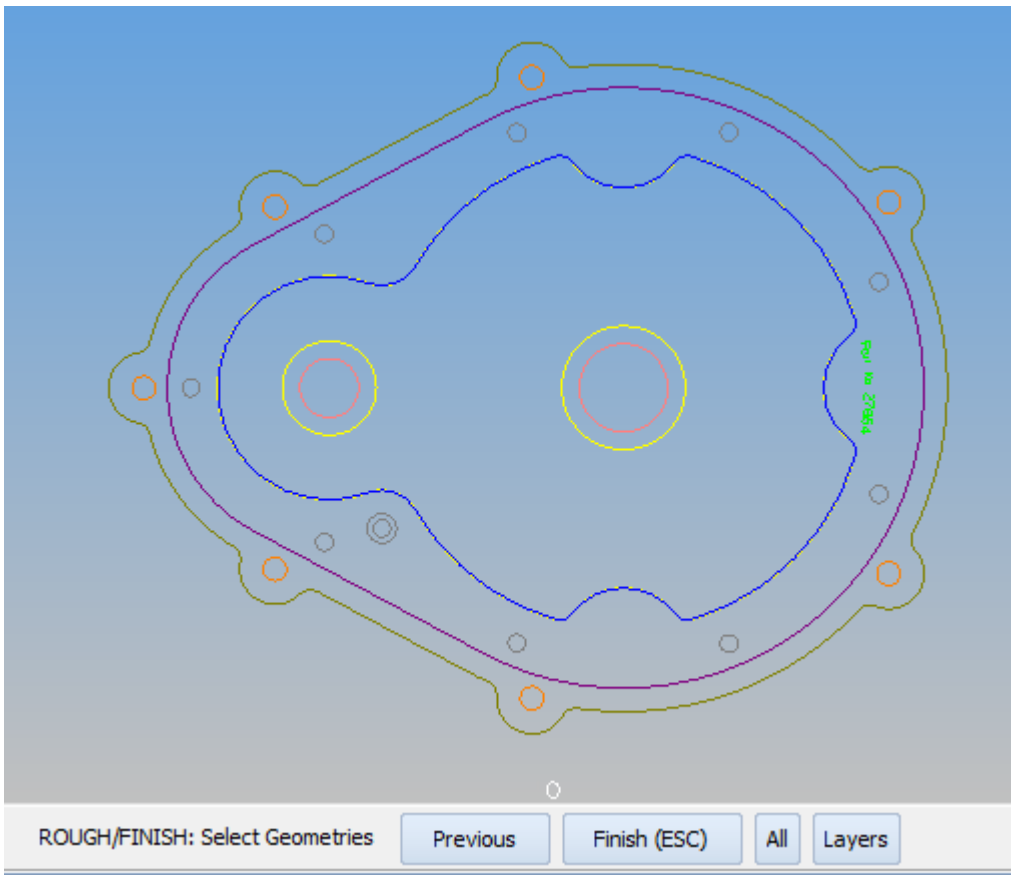



**Figure 87 - Geometry Selected**

**<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

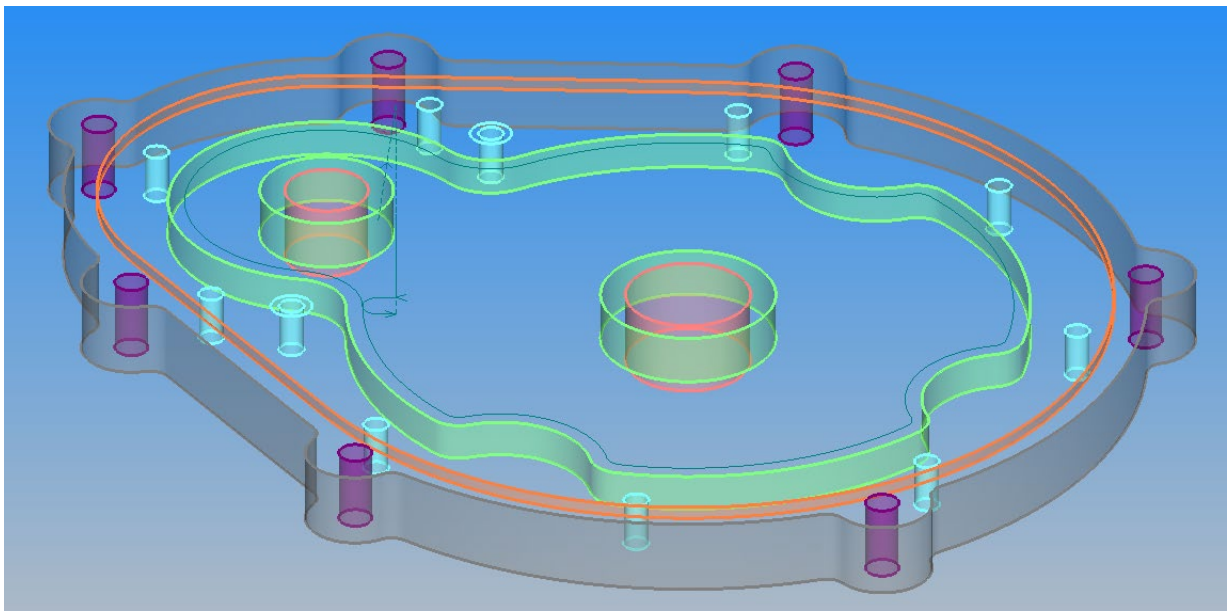


Figure 88 - Applied toolpaths

Note that during these two cycle applications, we used a separate pair of cycle for the differing depths of the profiles, it is entirely conceivable to have all six profiles in a single cycle to make the Operations listing shorted, the NC code will still be created correctly via the levels assigned to the geometries.

# Operation 7

## Chamfer Outer edge.

Select **MACHINE > Select Material**

> This is only required for tools that require feed and speed calculations

Select **MACHINE > Select Tool**
Select the **10mm Chamfer Tool** from the **Training** folder.
The tool will be displayed on the screen **<LClick>** or press ↵ to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

Select **MACHINE > Rough or Finish** .

### Types

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** ⊙ [**Selected**].

### General

General sets the output type. Set the options ⊙ **Machine Comp (G41/42),** Corners N/A on this job.

### Levels and Cuts

Levels and Cuts sets Z levels and depth of the cuts in Z.
Set the **Safe Rapid, Rapid Down To** as appropriate, **Z Stock Amount = -1,**
**Depth of Cut (0=Full) = 0**

☑ **Set Stock to Upper Level** is set to active so that the Z Depth control point of this tool, a location on the angle of this tool is the Depth control point and placing a **negative 1** value in the Z Stock Amount sets the correct size of chamfer for this tool.

### Machining Data

Machining Data sets the cutting information.
**Stock to be Left** = **0**, **Number of Cuts =1**.

### Lead-In/Out

Ensure that the option ☑ **Apply Auto Lead In/out** is active.
Make the options choices using the drop-down menus and the various dialogue options to suit your own requirements for manufacturing.

## Tool Data

Tool Data sets the cutting information.
Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate, **<LClick> [OK]**.

> Using this type of User Defined tool to create the chamfer detail allows you to see from the actual size of the chamfer to be created in the Z Stock Amount.

> Altering the **Z Stock Amount** or the **Stock** to be Left option will give you adjustments for the chamfer size.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select the outer profile geometry, then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.
**<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

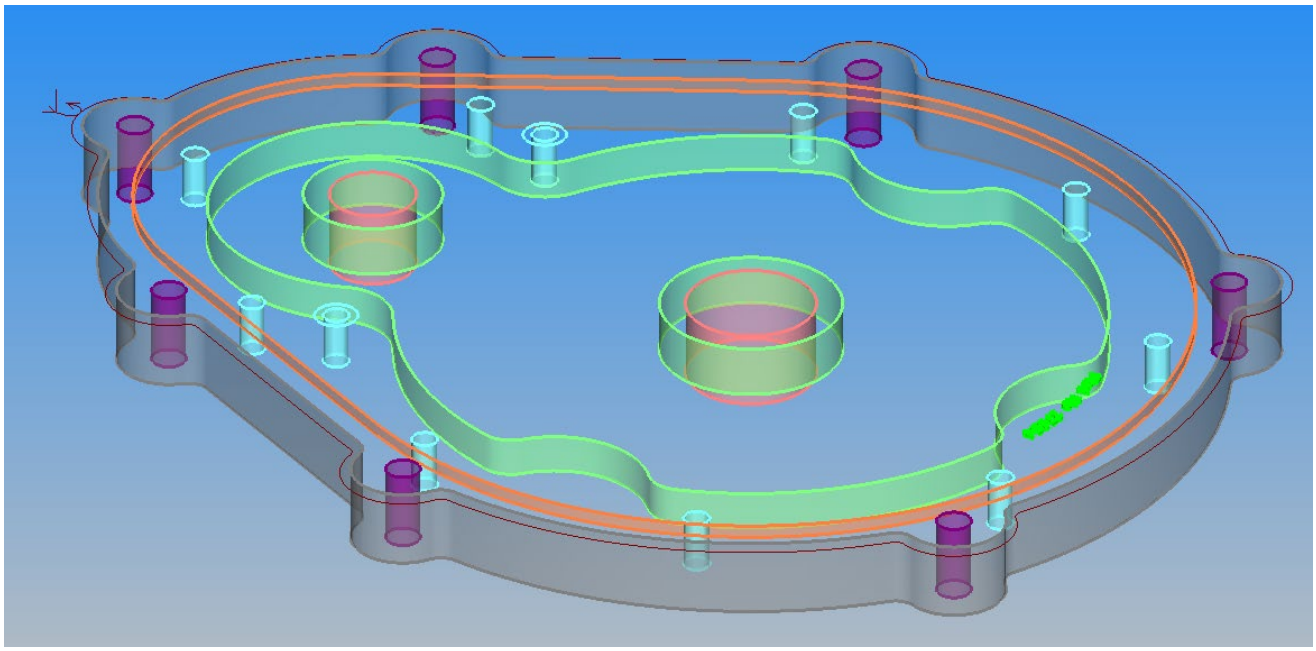


**Figure 89 - Outer chamfer toolpath applied**

# Operation 8

## Engrave Part No.27854

Before the text can be engraved it is necessary to convert the text to geometry and set the tool directions.
Select **GEOMETRY > Edit Text**

**<LClick>** the text on the drawing to select it.

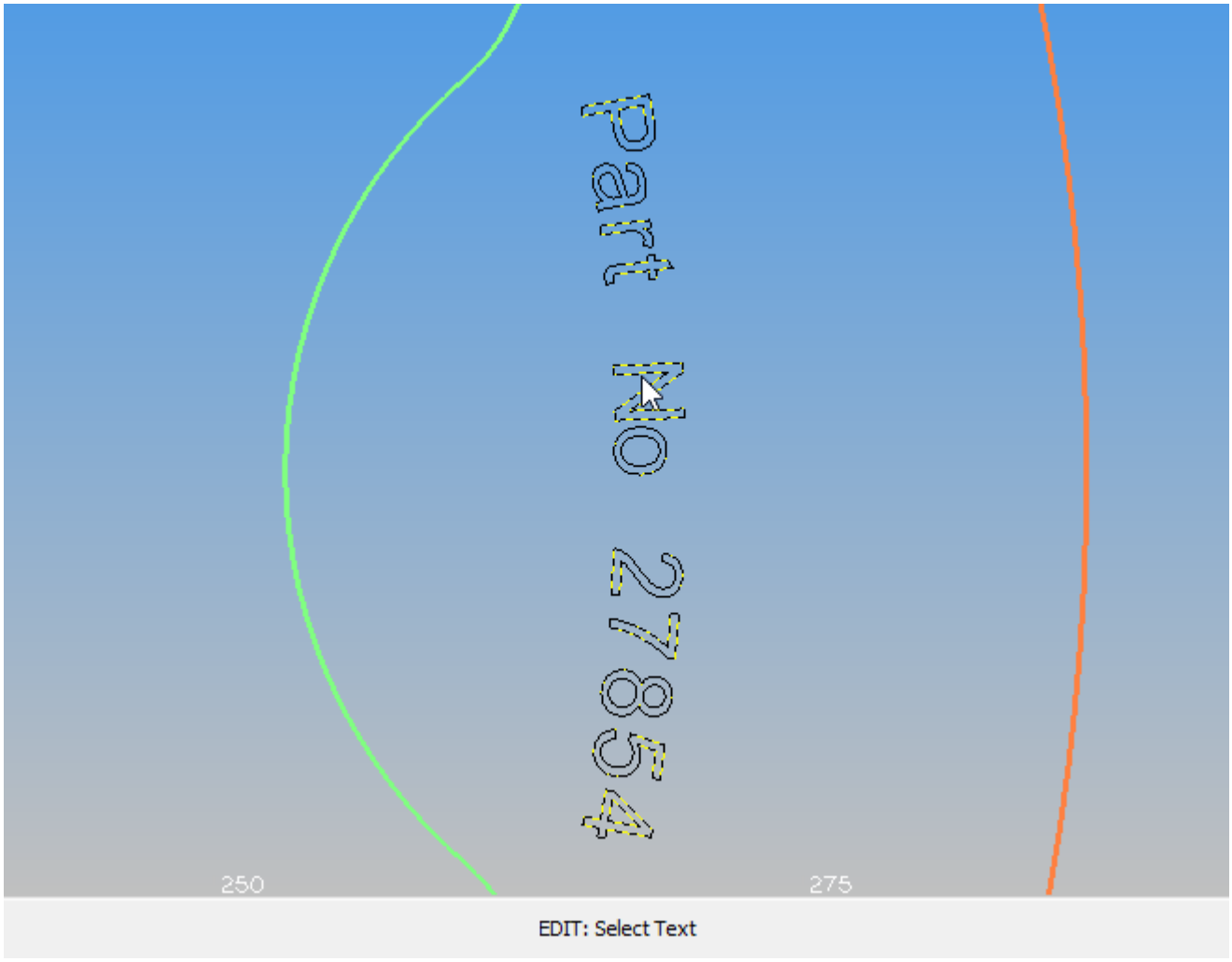



**Figure 90 - Selecting the text to edit**

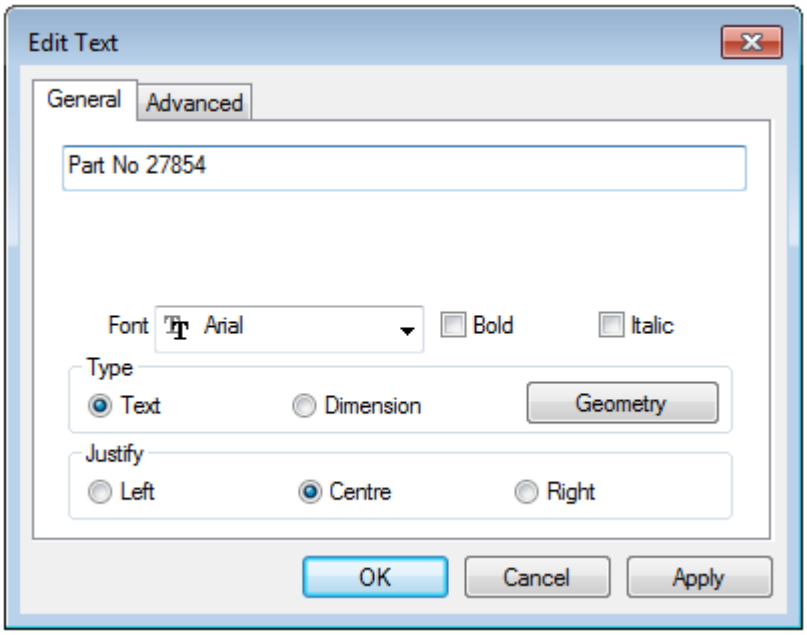



**Figure 91 - Edit Text dialogue**

Make any alterations if required and from the Dialogue select the **[Geometry]** button, then **[Ok]**.
This will automatically switch the Text to Geometry.

If there are no problems with the spelling or text layout, you can also use the **EDIT > Change**  to switch the test to geometry.

Select **MACHINE > Tool Directions** 

Set the options to ⊙ **CW** and ⊙ **Automatic for Pockets**.
Select all the geometries to be engraved using a window pick, **<LClick>** at one extreme then **<LClick>** at the diagonal opposite extreme to select all the required items.
**<RClick>** to action, then select **[Close]**.

> It cannot be stressed enough when applying Tool Directions at this stage that the **All Geometries** option is **NOT** used. This would affect every visible geometry on screen and the associated toolpaths.

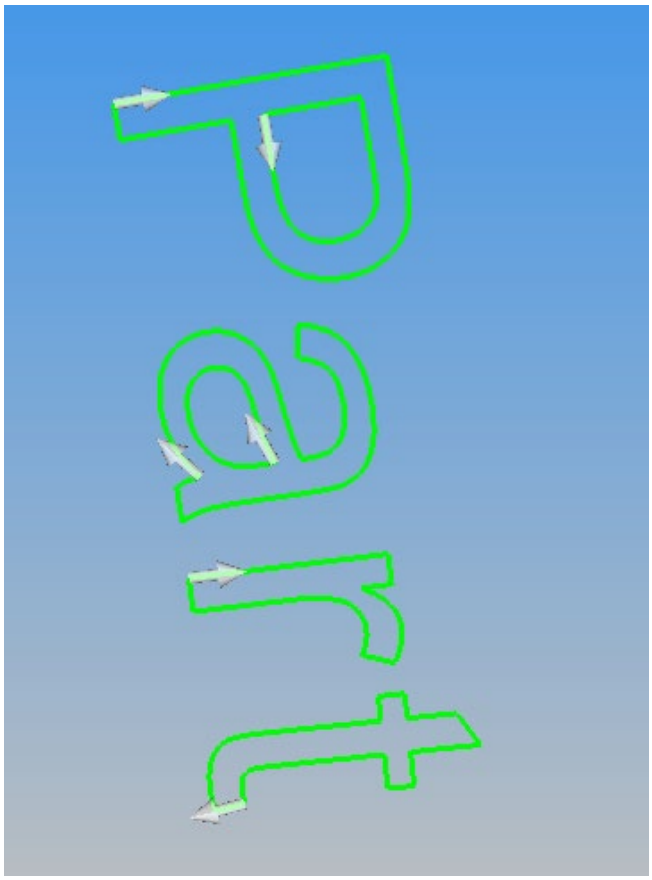


**Figure 92 - Part view of text before setting tool side**

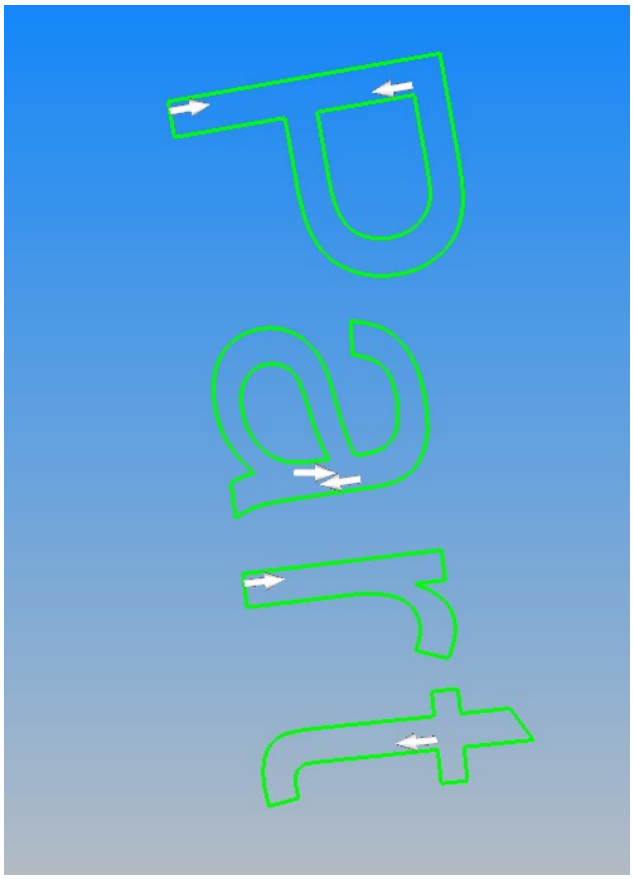


**Figure 93 - Part view of text after setting tool side**

Select **MACHINE > Select Material**

> This is only required for tools that require feed and speed calculations.

Select **MACHINE > Select Tool**
Select the **12 x 30° Engraver** from the **Training** folder.
The tool will be displayed on the screen **<LClick>** or press ↵ to accept and load this tool, **<RClick>** or pressing **<Esc>** will return to the tool selection screen.

Select **MACHINE > 3D Engrave**

## General

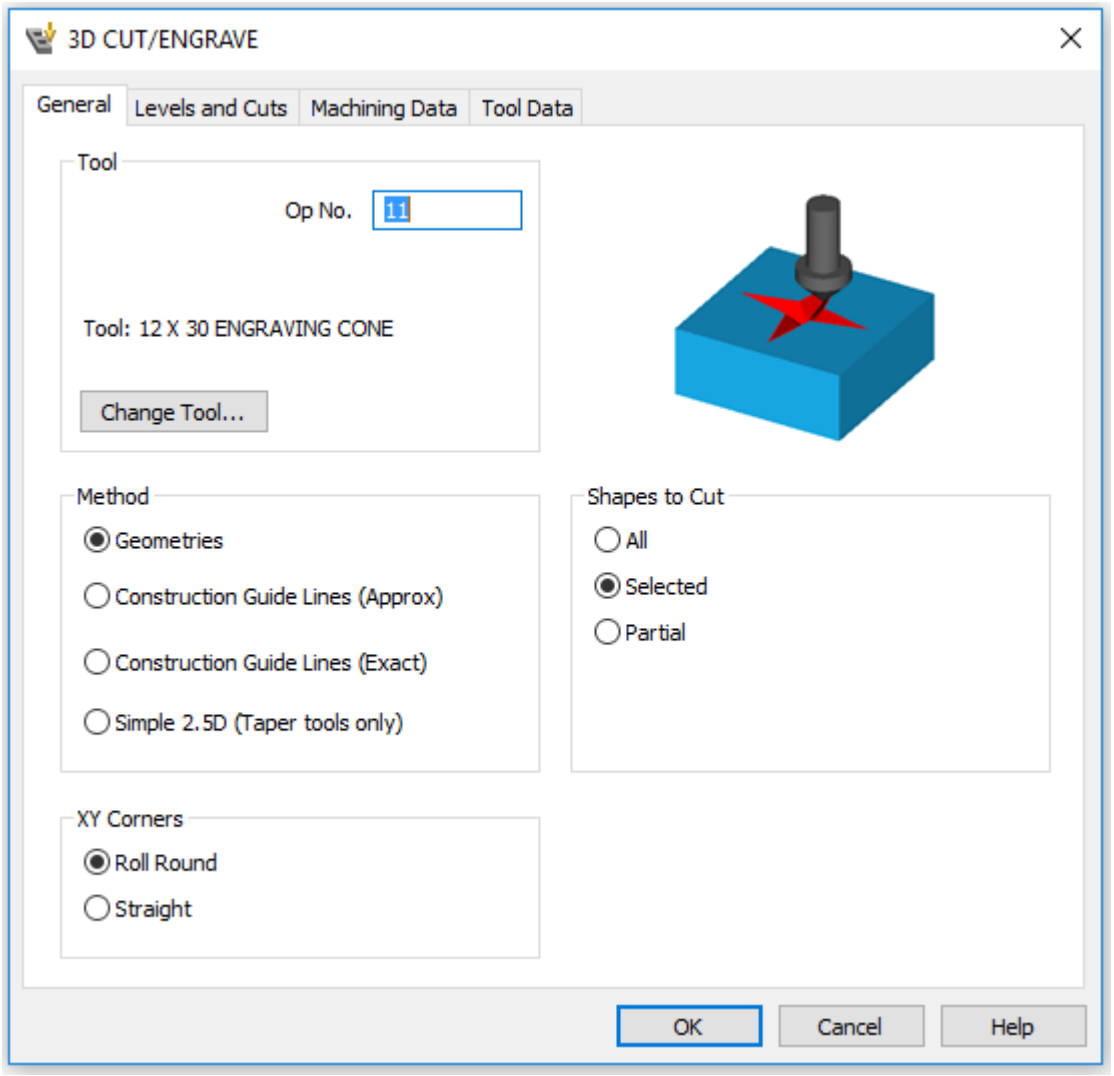



**Figure 94 - 3D Engraving General tab**

General sets the element that require engraving and how.
Set the options ⊙ **Geometries,** ⊙ **Selected,** ⊙ **Roll Round**.

## Levels and Cuts

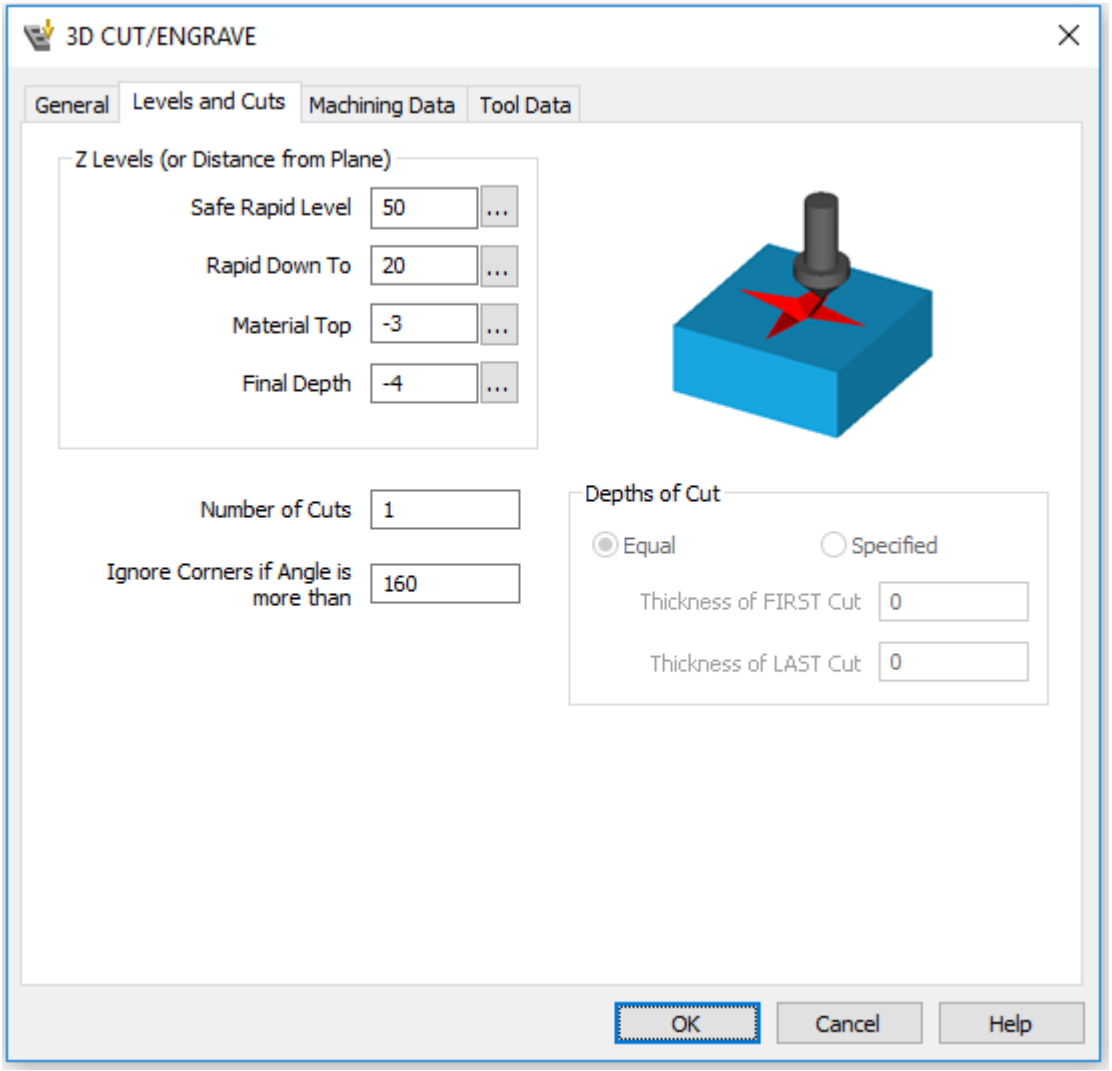



**Figure 95 - 3D Engraving Levels and Cuts tab**

Levels and Cuts sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid, Rapid Down To** as appropriate, **Material Top = -3, Final Depth = -4 Number of Cuts =1**.

> In this case the depth is set greater than required as the actual cut depth is determined by the tool size and the text size.

## Machining Data

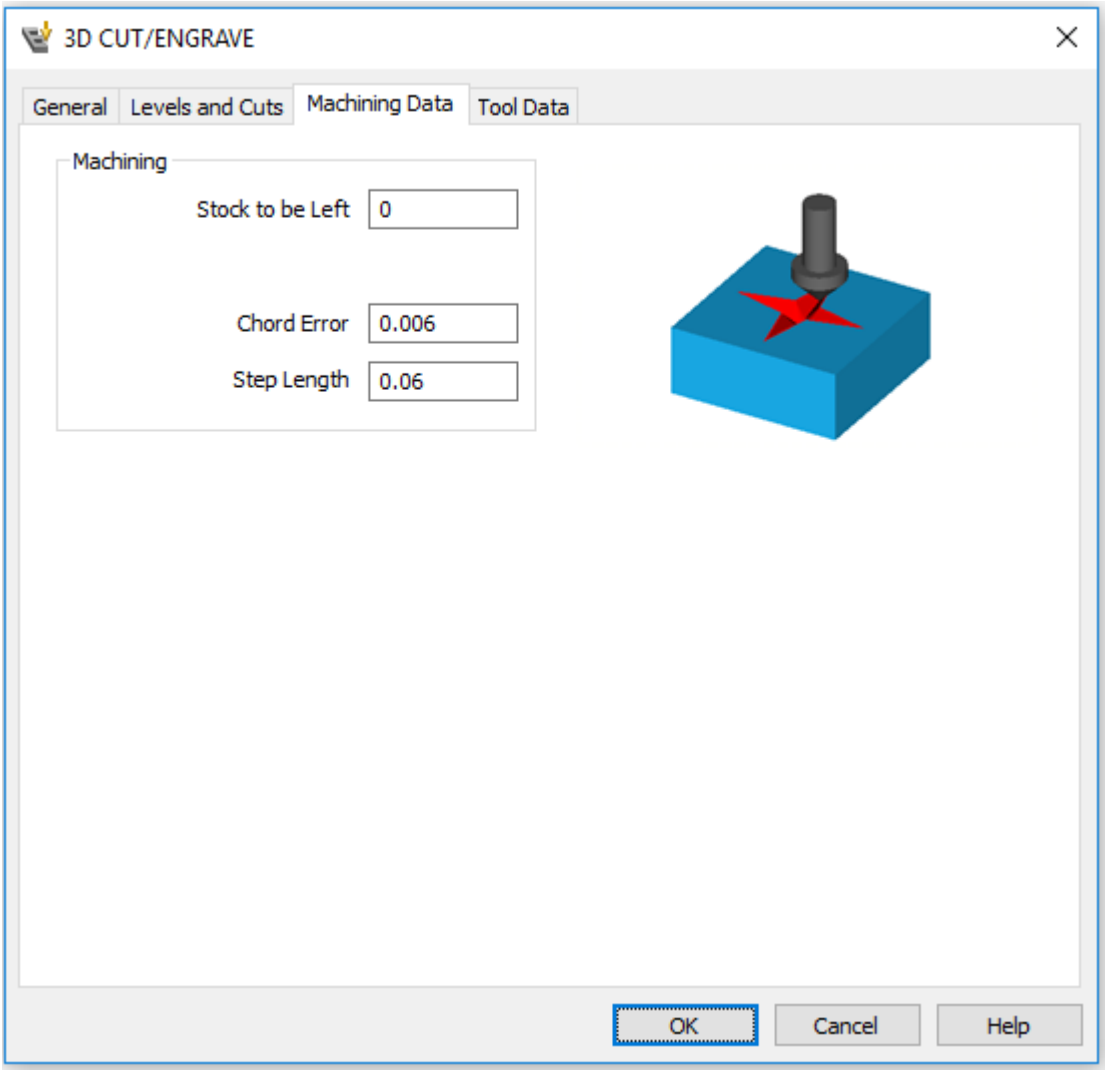



Figure 96 - 3D Engraving Machining Data tab

Machining Data sets the stock and precision options.
This can allow the cycle to be a roughing setup or a finishing set up.

The **Chord Error** dictates the accuracy that the toolpath will follow along curved profiles. The smaller the number, the finer or more accurate the toolpath will be.

The **Step Length** is the distance the tool travels in a straight line to generate the feature, prior to altering direction to continue along the profile. The smaller this figure, the finer or more accurate the changes of direction within the toolpath.

## Tool Data

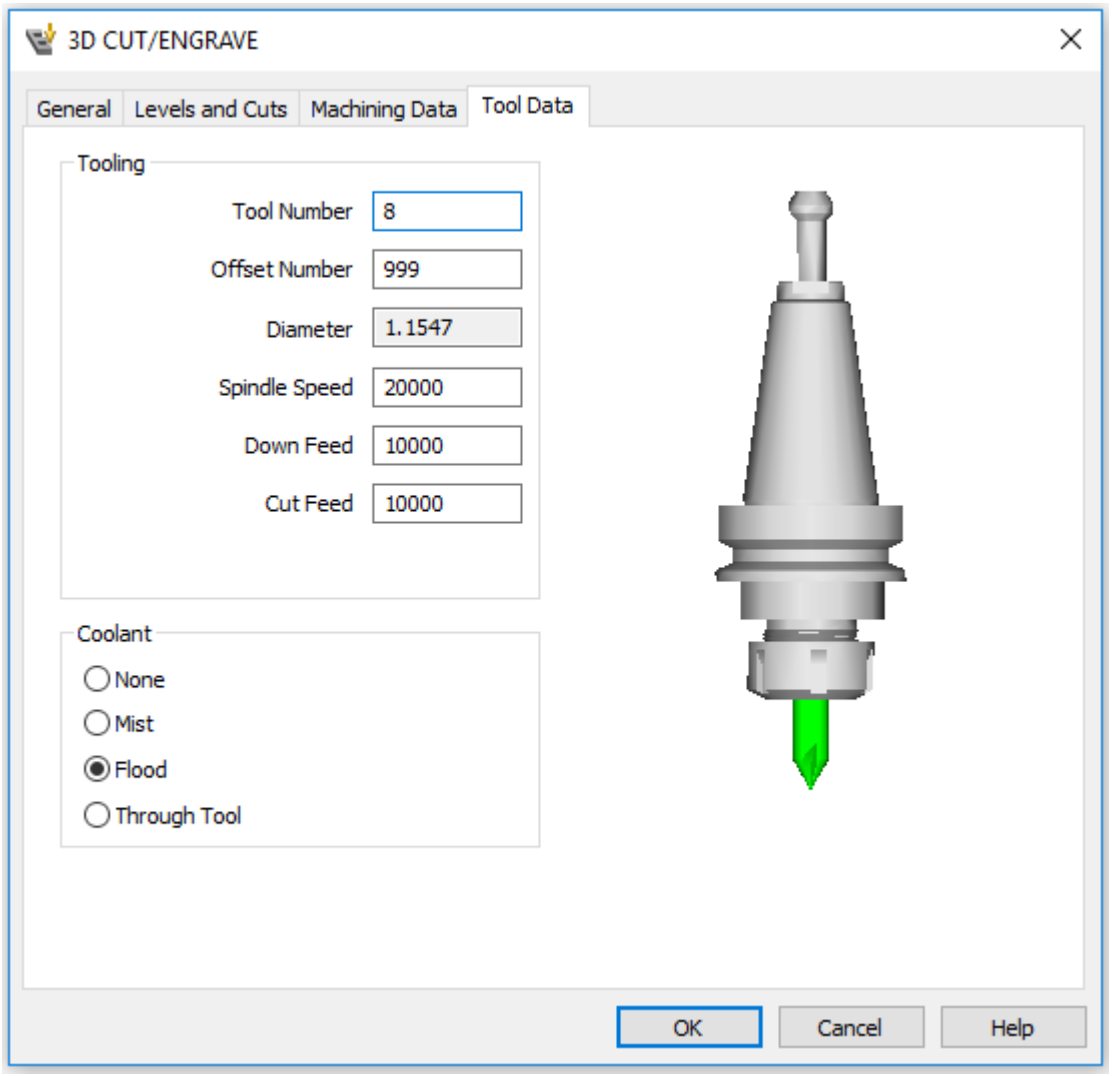



**Figure 97 - 3D Engraving Tool Data tab**

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate,
**Coolant** set as appropriate, then **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select the text Part No. 27854 using a window then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

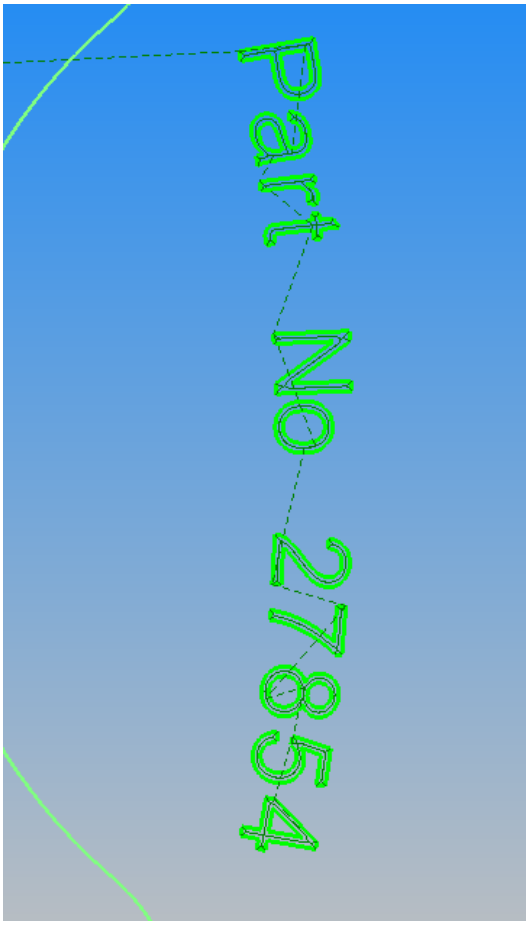



**Figure 98 - Geometries selected and engraved**

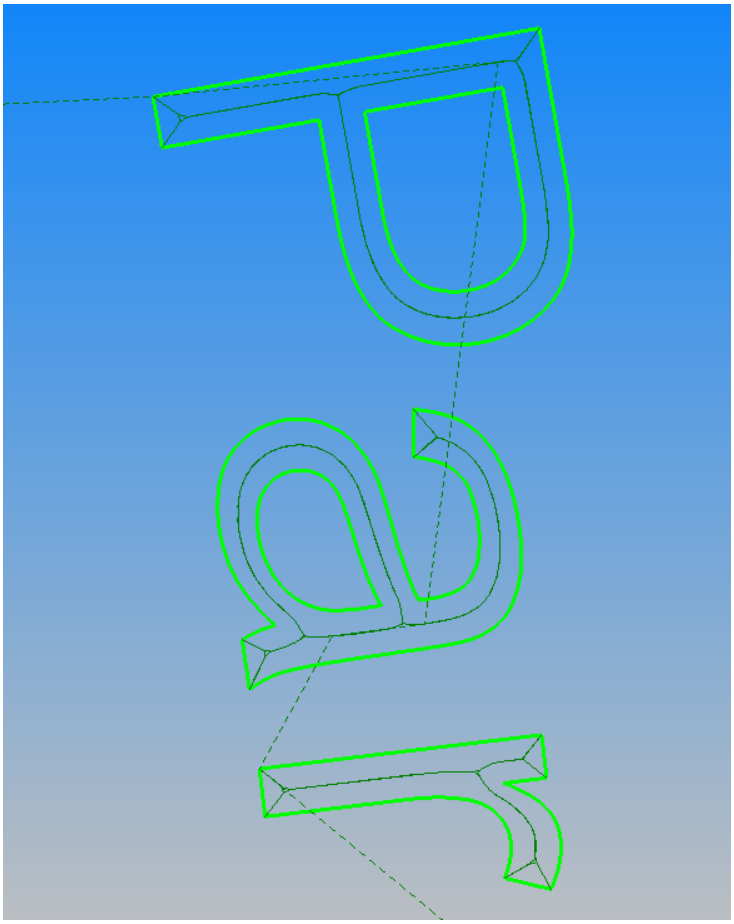


**Figure 99 - 3D Engraving close up**

# Operation 9

## Rough Profile 1 Outer Edge 16 deep leave +0.5 Stock

To make use of the original tooling used earlier in this tutorial, on the Operations page of the Project Manager, locate one of the cycles created using the 16mm Flat cutter.
**<RClick>** on the cycle to display the pop up menu.

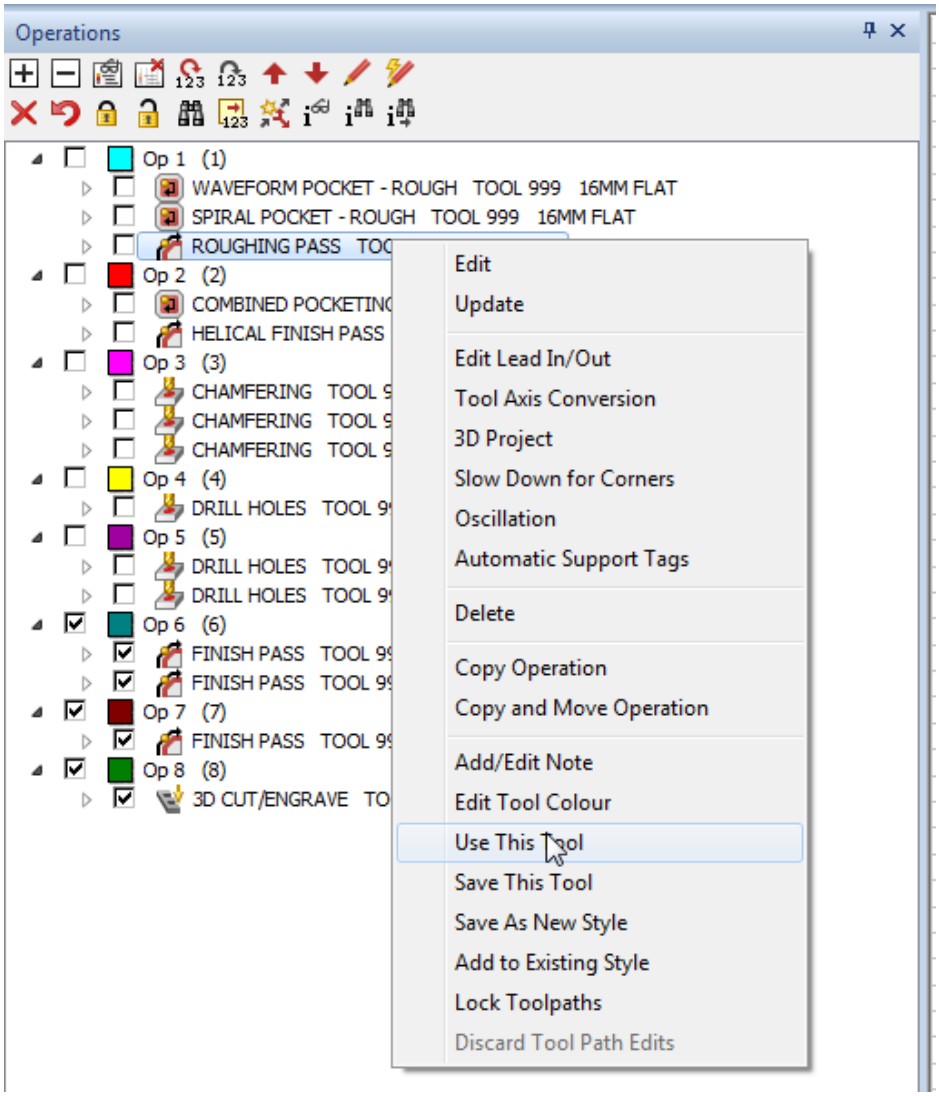



**Figure 100 - Making use of the Use This Tool option**

From the menu, **<LClick>** the option **Use This Tool,** this will maintain any alterations to tool feeds and speeds and not revert to the generic settings if the tool was select by the tooling selection dialogue option.

You must still **<LClick>** in the drawing area to accept the tool in the same manner you would if using the **Select Tool** menu option.

Select **MACHINE > Rough or Finish**

## Types

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** then ⊙ **Selected**.

## General

General sets the output type. Set the options ⊙ **Machine Comp (G41/42)**.

> Some machines require the option ☑ Apply Compensation on Rapid Approach / Retract to be activated, select this option if it is applicable to your post processor and machine. If you select this option and it is not applicable you will be presented with an error Dialogue when listing or outputting the NC file.

## Levels and Cuts

Levels and Cuts sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate, **Z Stock Amount = 1 Depth of Cuts =9**. *(This set number of cuts in Z)*

## Machining Data

Machining Data sets the cutting information.
Set the **Stock to be Left** = **0.5**, **Number of Cuts =1**

## Lead-In/Out

Ensure that the option ☑ **Apply Auto Lead In/out** is active.
Make the options choices using the drop-down menus and the various dialogue options to suit your own requirements for manufacturing.

## Tool Data

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate,
**Coolant** set as appropriate, then **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select outer profile then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

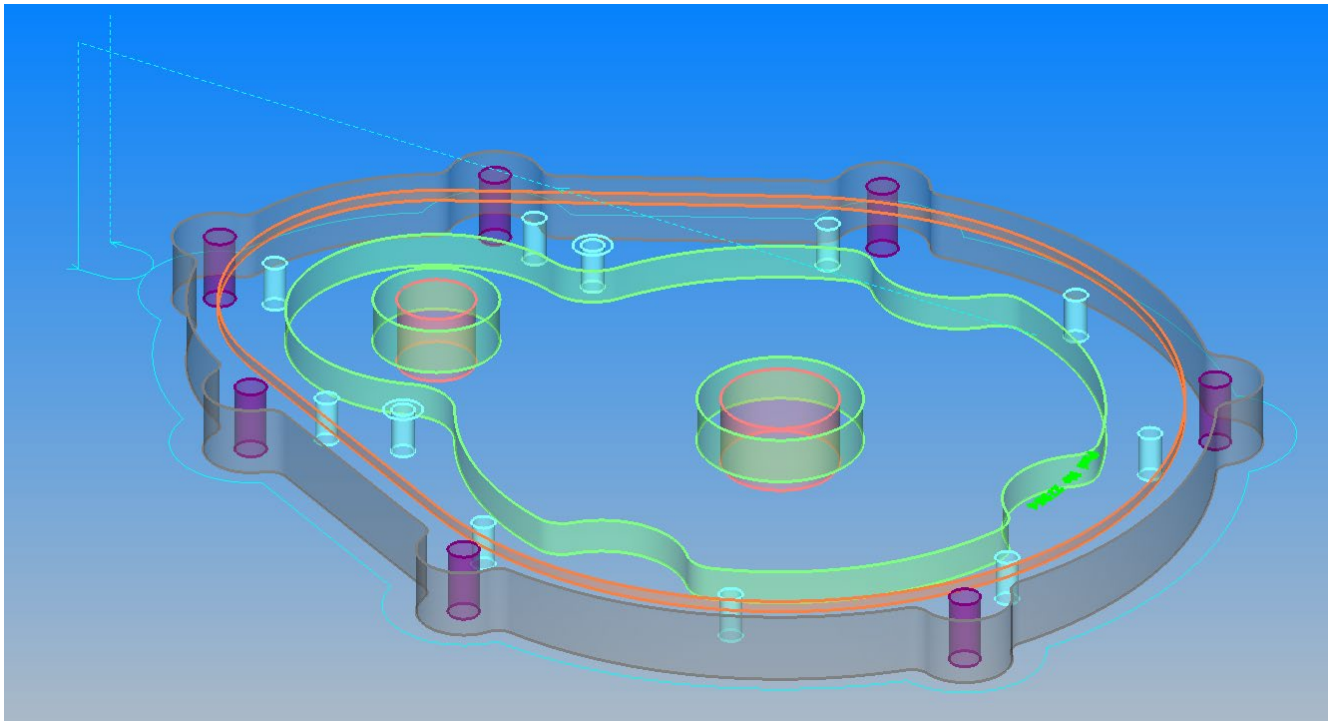


**Figure 101 - Outer profile toolpath applied**

# Operation 10

## Residual Rough Profile 1 leave +0.5 stock

To make use of the original tooling used earlier in this tutorial, on the **Operations** page of the Project Manager, locate one of the cycles created using the **8mm Flat** cutter.
**<RClick>** on the cycle to display the pop up menu.

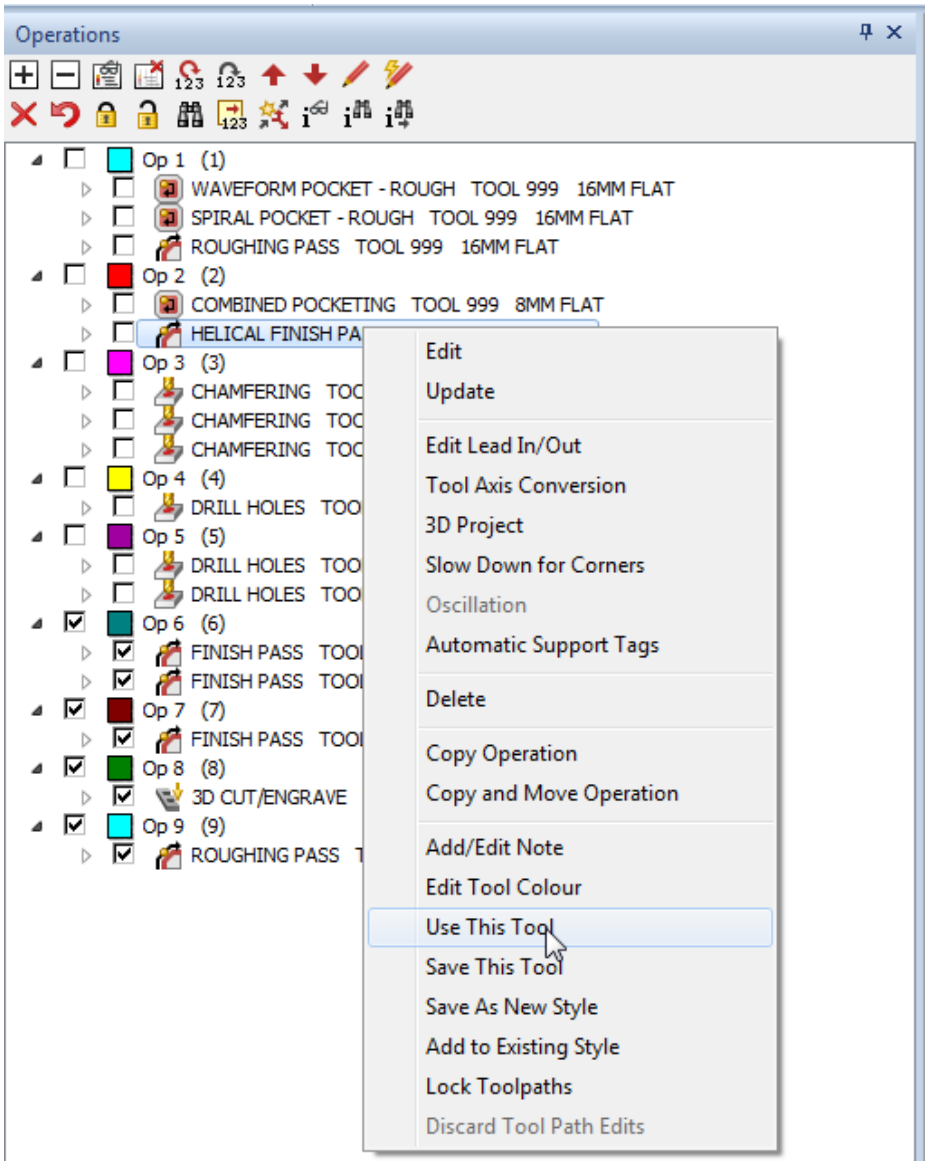



**Figure 102 - Making use of the Use This Tool option**

From the menu, **<LClick>** the option **Use This Tool,** this will maintain any alterations to tool feeds and speeds and not revert to the generic settings if the tool was select by the tooling selection dialogue option.

You must still **<LClick>** in the drawing area to accept the tool in the same manner you would if using the **Select Tool** menu option.

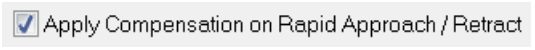
Select **MACHINE > Rough or Finish**  .

## Types

Types sets the sides and geometries.
Set the options ⊙ **Auto Z,** then ⊙ **Selected**.

## General

General sets the pocketing style.
Set the options ⊙ **Machine Comp (G41/42),** ☑ **Take Account of Previous Machining, Overlap = 1**
Corners N/A on this job.

Some machines require the option 

to be activated, select this option if it is applicable to your post processor and machine. If you select this option and it is not applicable you will be presented with an error Dialogue when listing or outputting the Nc file.

## Levels and Cuts

Levels and Cuts sets Z levels and number and nature of the cuts in Z.
Set the **Safe Rapid Distance, Feed Down Distance** as appropriate, **Z Stock Amount = 1**
**Depth of Cut = 6**.

## Machining Data

Machining Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate,
**Stock to be Left = 0.5, Number of Cuts =1**.

## Lead-In/Out

Ensure that the option ☑ **Apply Auto Lead In/out** is active.

Make the options choices using the drop-down menus and the various dialogue options to suit your own requirements for manufacturing.

## Tool Data

Tool Data sets the tooling and coolant information.
Set the **Tool Number, Offset Number, Speed** and **Feeds** as appropriate,
**Coolant** set as appropriate, then **<LClick> [OK]**.

## Select Geometries

The system will prompt you to select the geometries that are to be machined.
Select outer profile then then **<RClick>** or press **<Esc>** to Finish the geometry selection and apply machining tool paths.

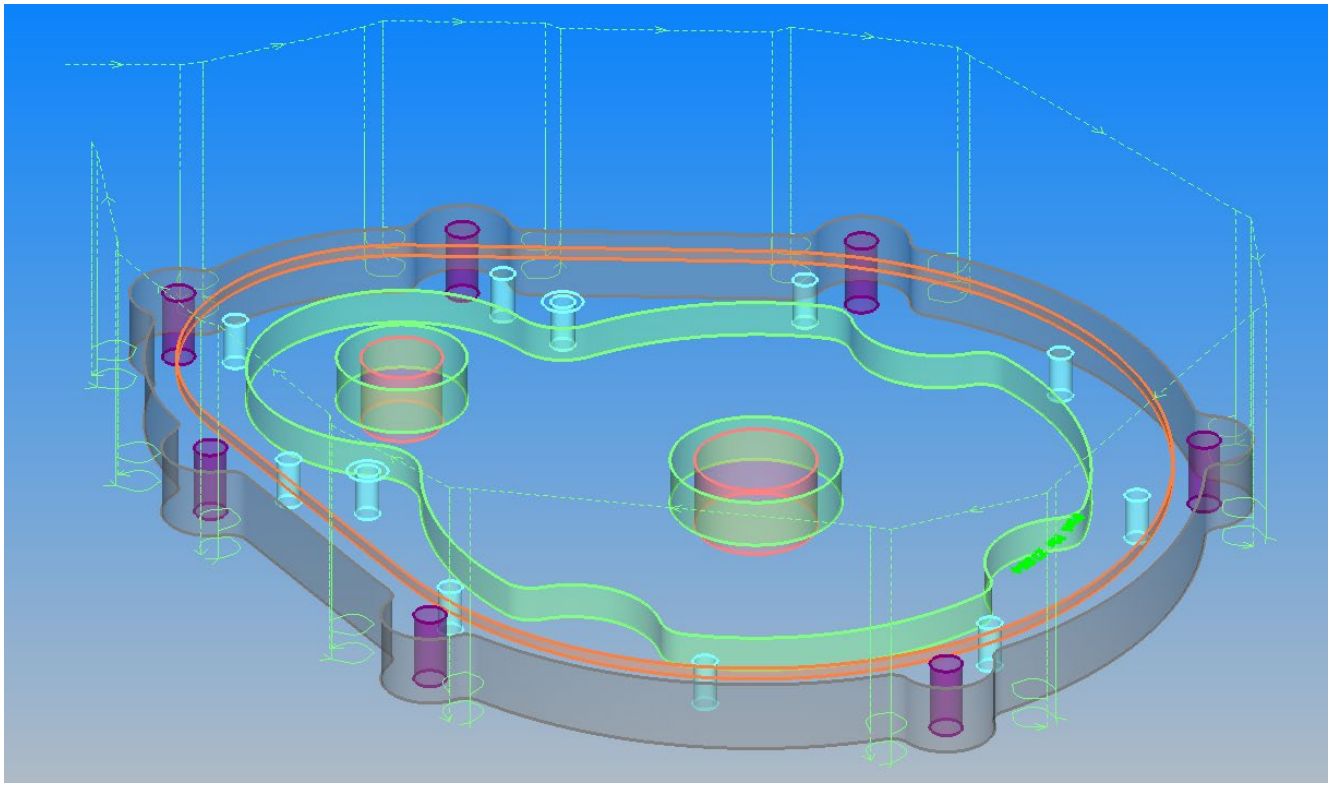


**Figure 103 - Take Account of Previous Machining applied**

## Finish Profile 1

This operation is essentially the same as the previous, so you can copy the operation and edit the parameters. Because this is a single cut operation you can add support tags if required.
**<RClick>** on the previously created cycle and select **Copy Operation**. The operation will then be duplicated.

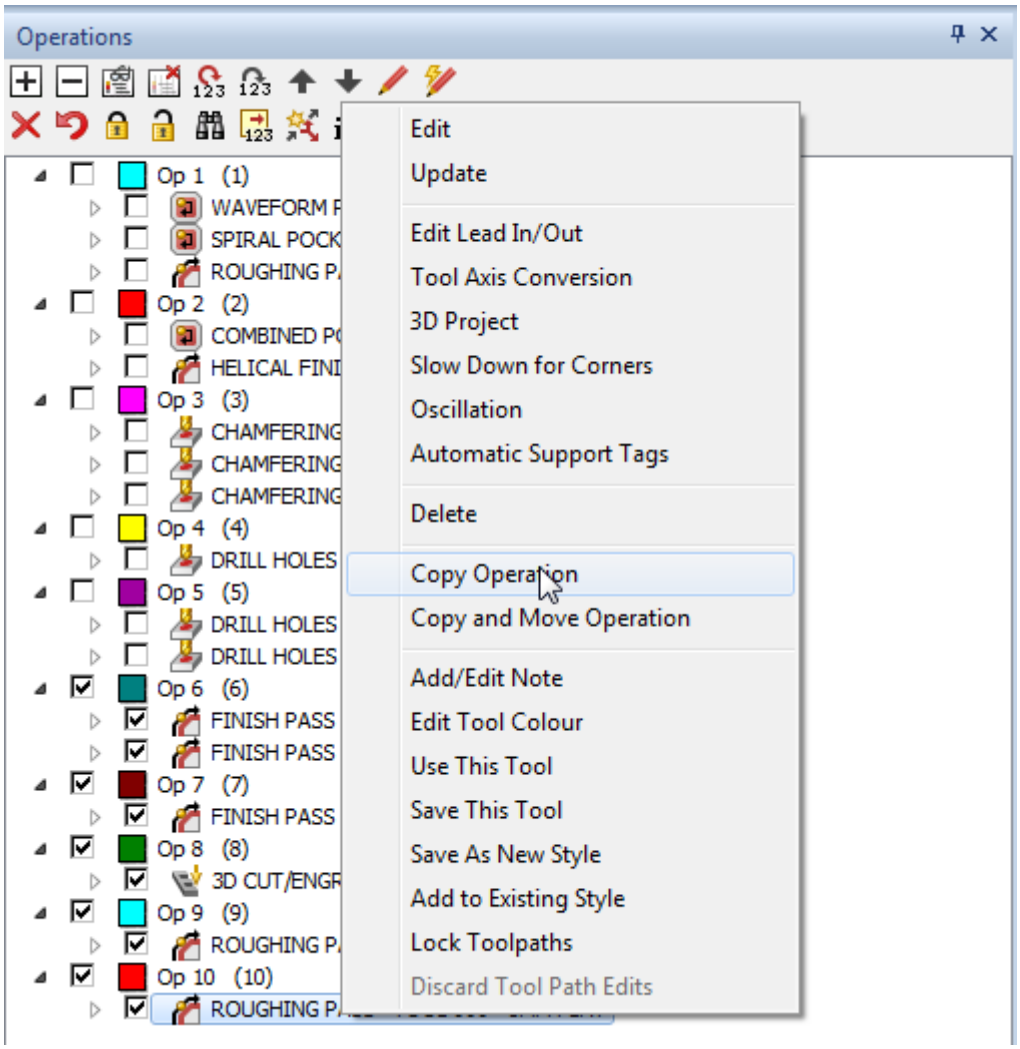



**Figure 104 - <RClick> menu to Copy Operation for time saving**

**<RClick>** on the second Roughing Pass in Op 10 then **<LClick>** on **Edit.**

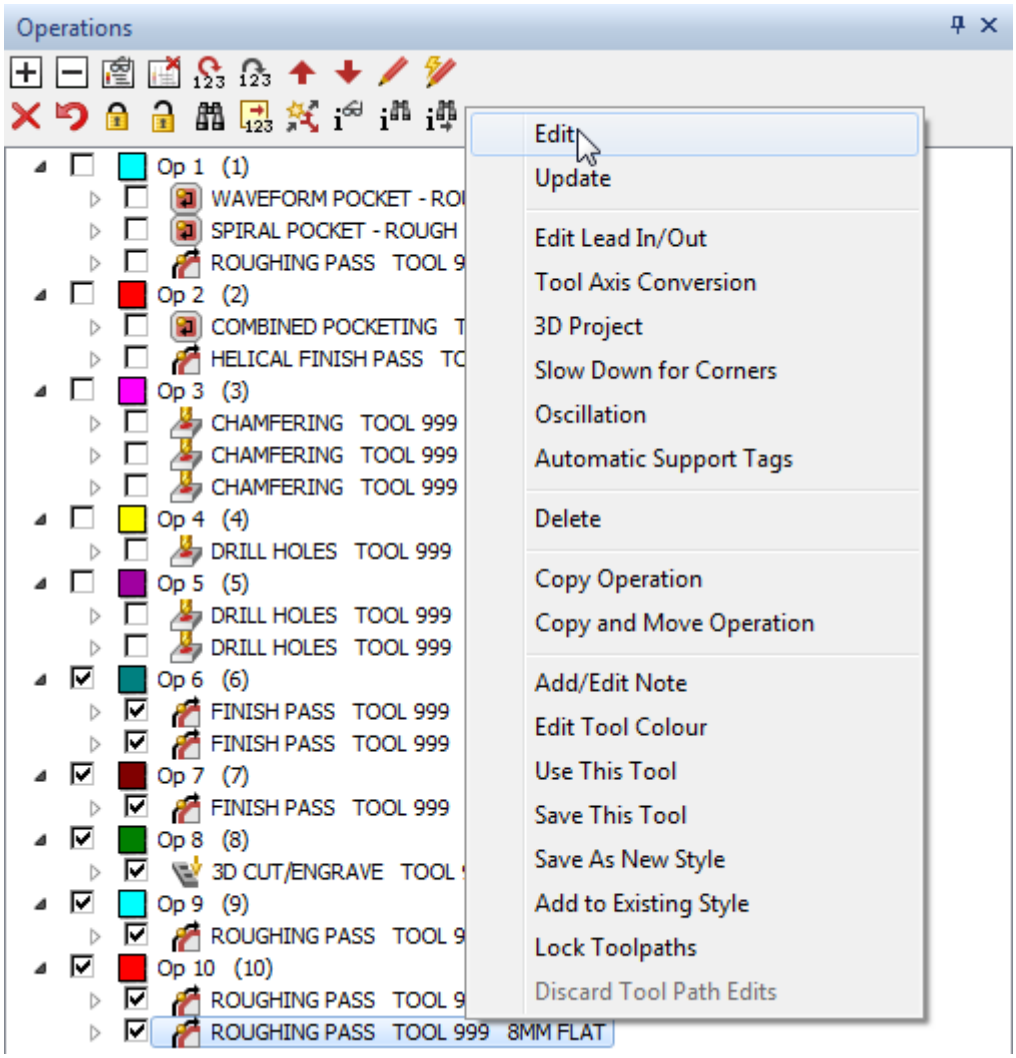



**Figure 105 - <RClick> to Edit the copy to create a Finish pass**

## General

On the General tab, untick the ☐ **Take Account of Previous Machining** option.

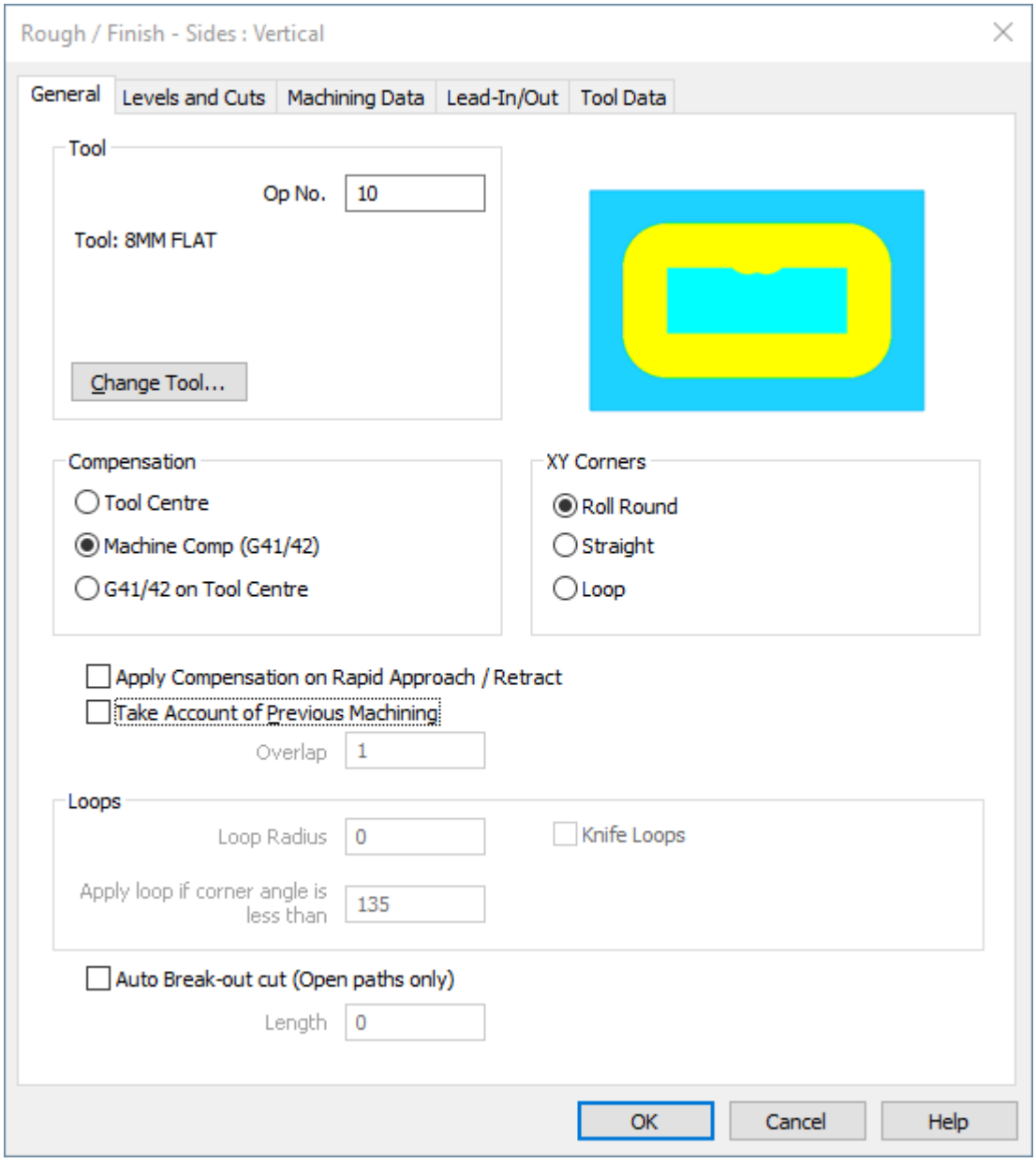



**Figure 106 - Take Account of Previous Machining not ticked**

On the Levels and Cuts tab, set the **Safe Rapid Distance, Feed Down Distance** as appropriate, **Z Stock amount = -0.5 Depth of Cut =0**

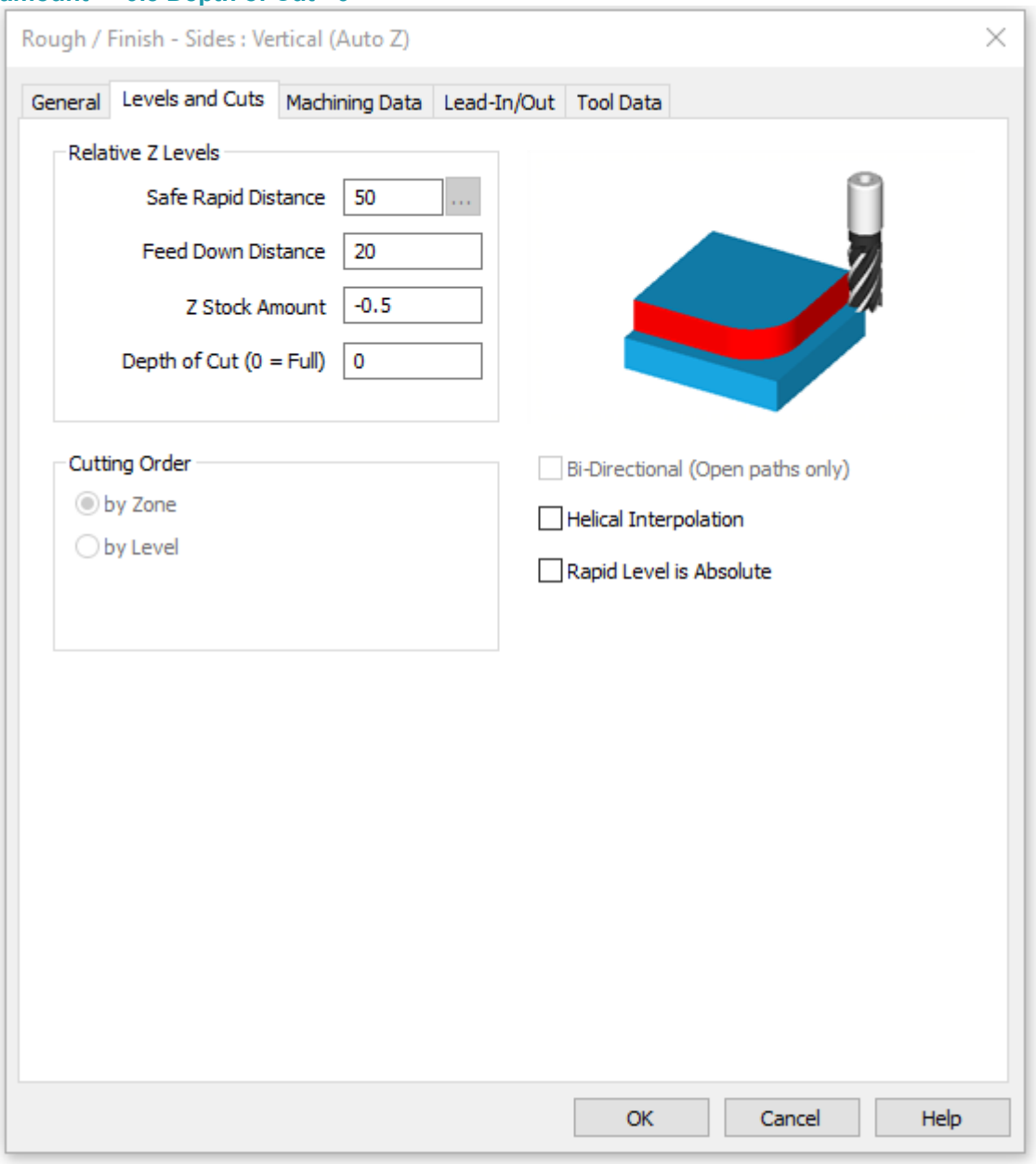



**Figure 107 - Levels and Cuts tab altered to suit the required practice**

On the Machining Data tab, set the **Stock to be Left = 0**, **Number of Cuts =1**

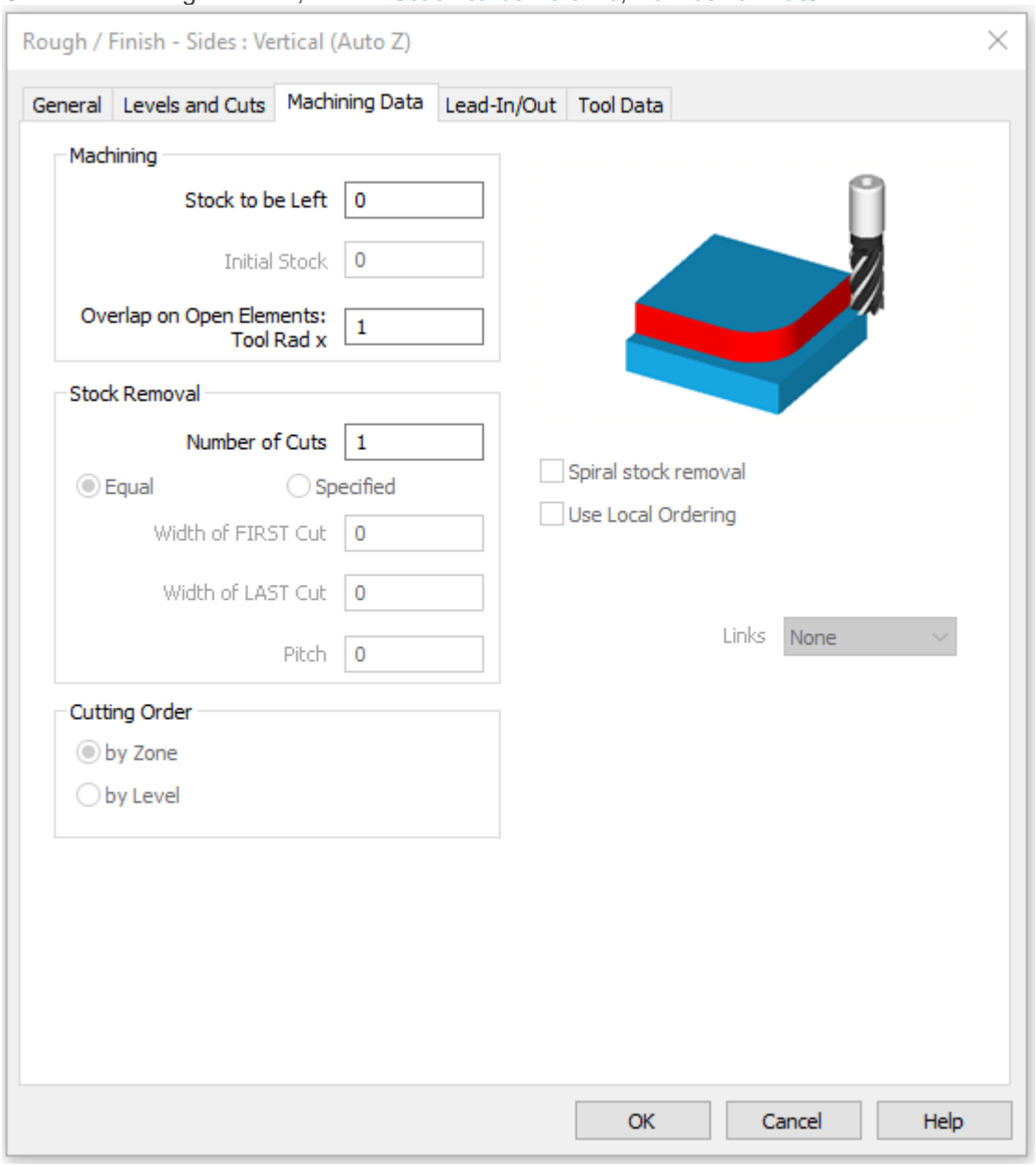



**Figure 108 - Machining Data tab altered to zero Stock to be Left**

Apply the appropriate Lead In/Out options.

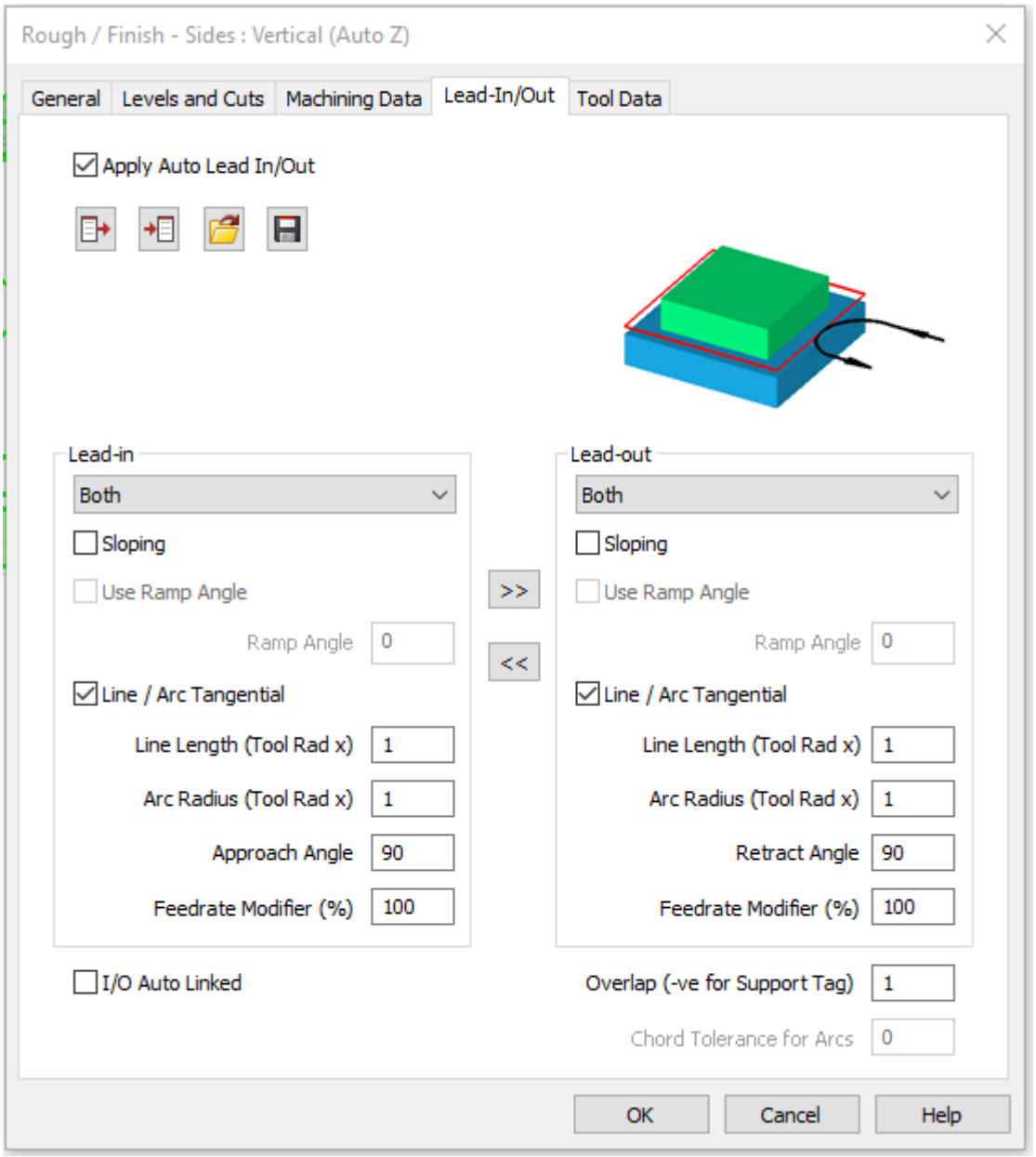



**Figure 109 - Rough and Finish Lead-In/Out tab**

Then **<LClick> [Ok]** to apply the alterations to the copied cycle.

The Roughing Pass will now be listed as a Finish Pass.

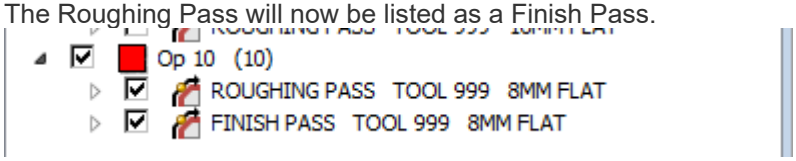



**Figure 110 - New Finish Pass**

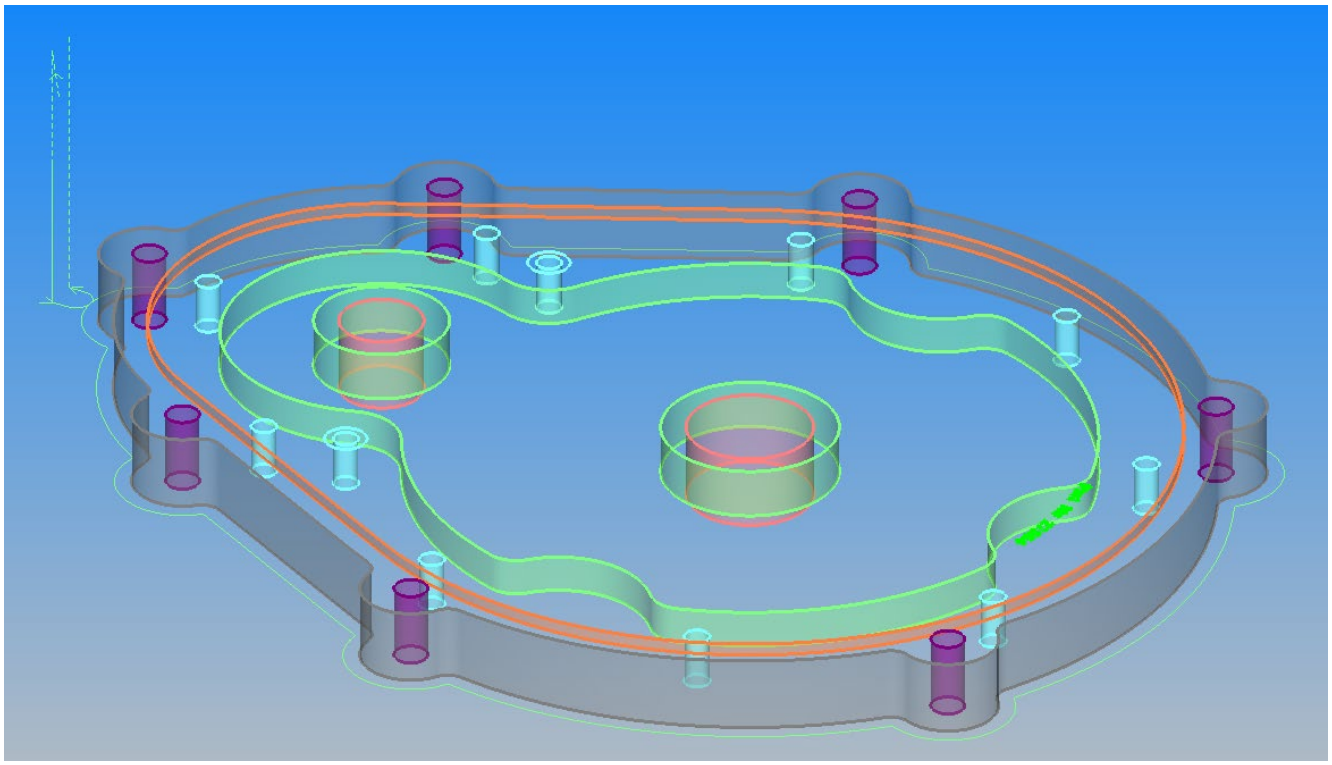


**Figure 111 - Applied Finish pass after editing the rough pass**

To add support tags, select **MACHINE > Edit Machining > Add Support Tags** and set accordingly.

To apply **Slow Down for Corners <RClick>** on the operation and select the option from the popup menu.

Prior to seeing the display options, it will be necessary to switch on all the operations.
Select the **Show All** button at the top of the operations project manager page.

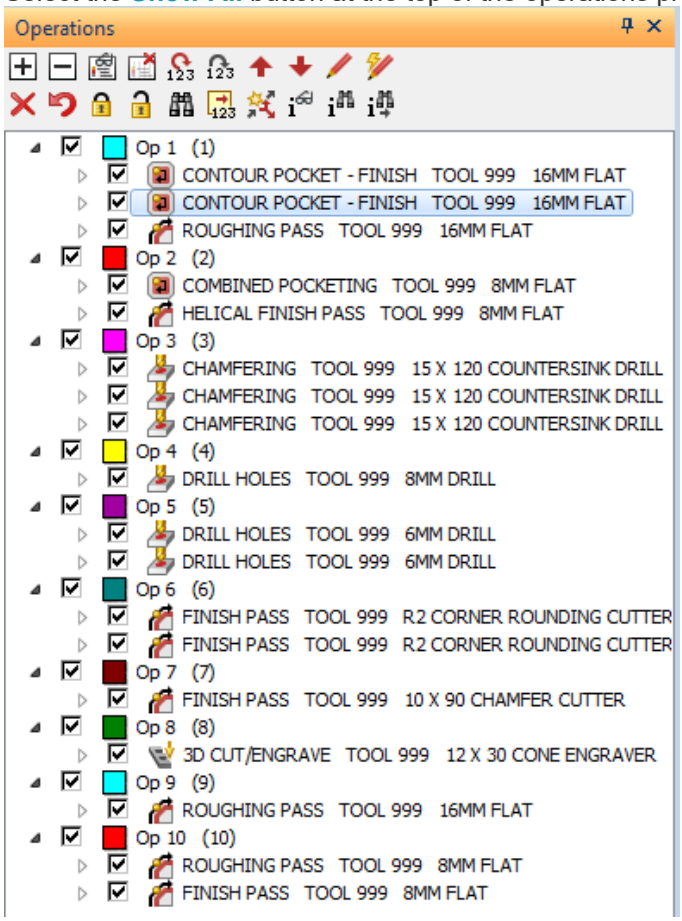



**Figure 112 - Show All option from the Operations page**

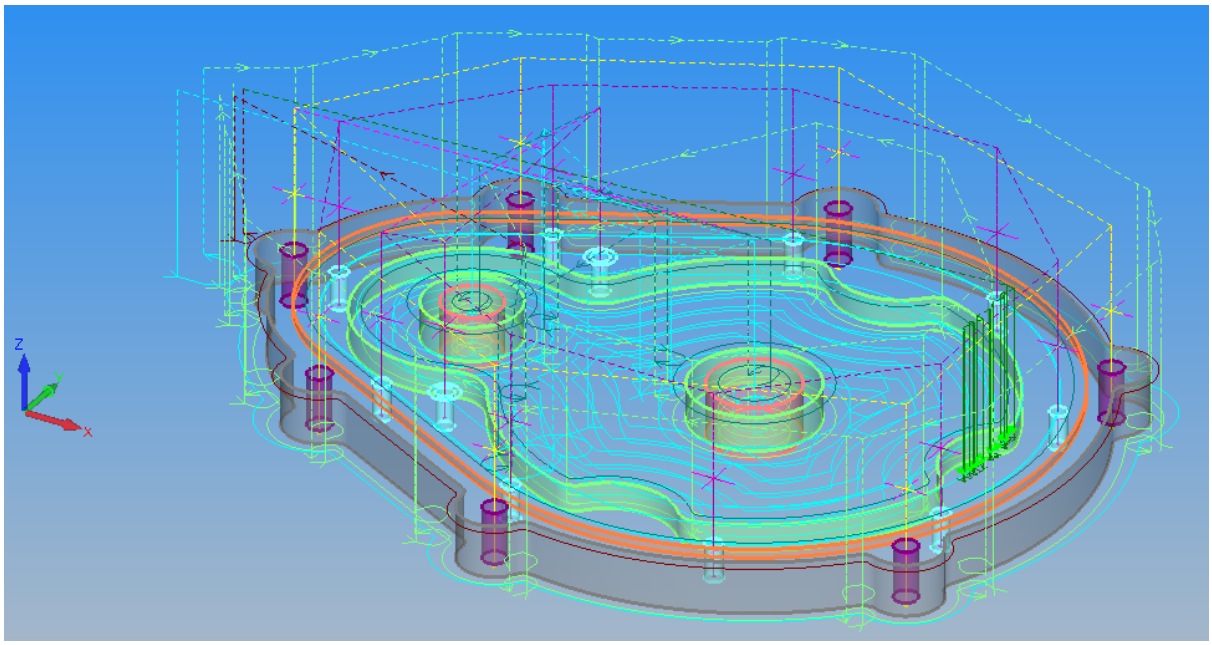



**Figure 113 - ISO view with all toolpaths visible**

# Run simulation

Select **VIEW > Solid Simulation** 

The view will show the material shaded. Select ISO view  and run the simulation.


**Figure 114 - Simulation option to view tool colours**

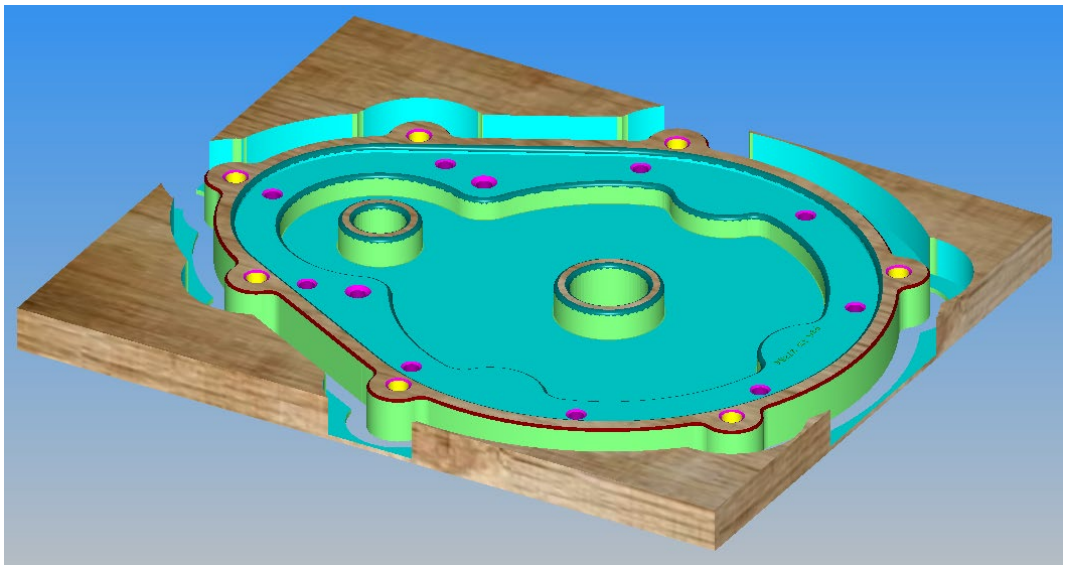


**Figure 115 - Housing Plate simulated in the Internal Simulator**


**Figure 116 - Close up of simulated 3D Engraving**

Using the Show Tool Colours  option will display the toolpaths in the colours assigned to the tools used. This also gives visual confirmation of any areas that have been forgotten.

Using the Send to Simulator option the redundant Corners would be removed from the display automatically.

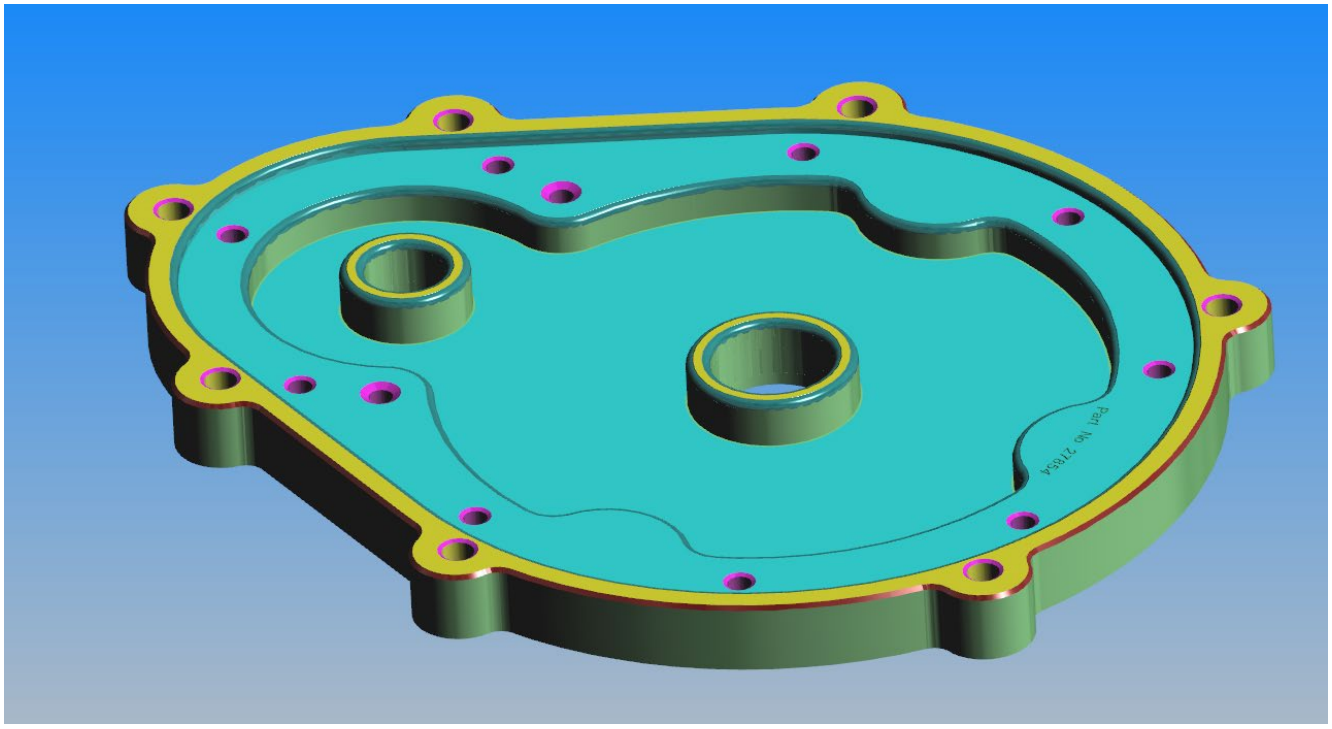


**Figure 117 - Housing Plate simulated in the External Simulator**

# Save & Output NC

Using the **Quick Edit**  option on the **Operations Project Manager** page, it may be necessary to alter tool numbers to suit the actual machine on which the part will be created.

When the job is complete save it to a suitable folder.
To send the program to the machine tool it first must be output as NC code.
The NC code output can be sent, directly to the machine using the serial cable, to a text file, or to both the machine and a text file.
NC code text files can be edited and communicated to the machine tool using Alphaedit.

Select **HOME > Output NC** 

.Ensure you are using the correct post processor and select the required output option then select **[Ok].**
During the output, you may be asked questions relating to the machine the code is being output for.

> If the component is going to be nested remember to delete any materials and work volumes before saving.

## Version amendments

| V | Amendment Description | A | Software Version | Amended Date |
|---|---|---|---|---|
| 12 | Minor text formatting alterations. | 1 | 2020.1 | 02/10/2019 |
| 12 | Template altered to Hexagon branding | 0 | 2020.0 | 15/03/2019 |

HEXAGON
MANUFACTURING INTELLIGENCE

# ALPHACAM 2020.1
# Housing Plate Auto Z


**ALPHACAM**
Floor Two
No. One, Waterside Drive
Arlington Business Park
Theale
Reading
Berkshire
RG7 4SA
United Kingdom

**Tel.**+44 (0) 1189 226677
**Email**. alphacaminfo@hexagon.com
**Web**. www.alphacam.com

**Support**. support.alphacam.uk@hexagon.com
**Portal.** customers.ps.hexagonmi.com